FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

A LOS ASISTENTES A LOS CURSOS

Las autoridades de la Facultad de Ingeniería, por conducto del jefe de la División de Educación Continua, otorgan una constancia de asistencia a quienes cumplan con los requisitos establecidos para cada curso.

El control de asistencia se llevará a cabo a través de la persona que le entregó las notas. Las inasistencias serán computadas por las autoridades de la División, con el fin de entregarle constancia solamente a los alumnos que tengan un mínimo de 80% de asistencias.

Pedimos a los asistentes recoger su constancia el día de la clausura. Estas se retendrán por el periodo de un año, pasado este tiempo la DECFI no se hará responsable de este documento.

Se recomienda a los asistentes participar activamente con sus ideas y experiencias, pues los cursos que ofrece la División están planeados para que los profesores expongan una tesis, pero sobre todo, para que coordinen las opiniones de todos los interesados, constituyendo verdaderos seminarios.

Es muy importante que todos los asistentes llenen y entreguen su hoja de inscripción al inicio del curso, información que servirá para integrar un directorio de asistentes, que se entregará oportunamente.

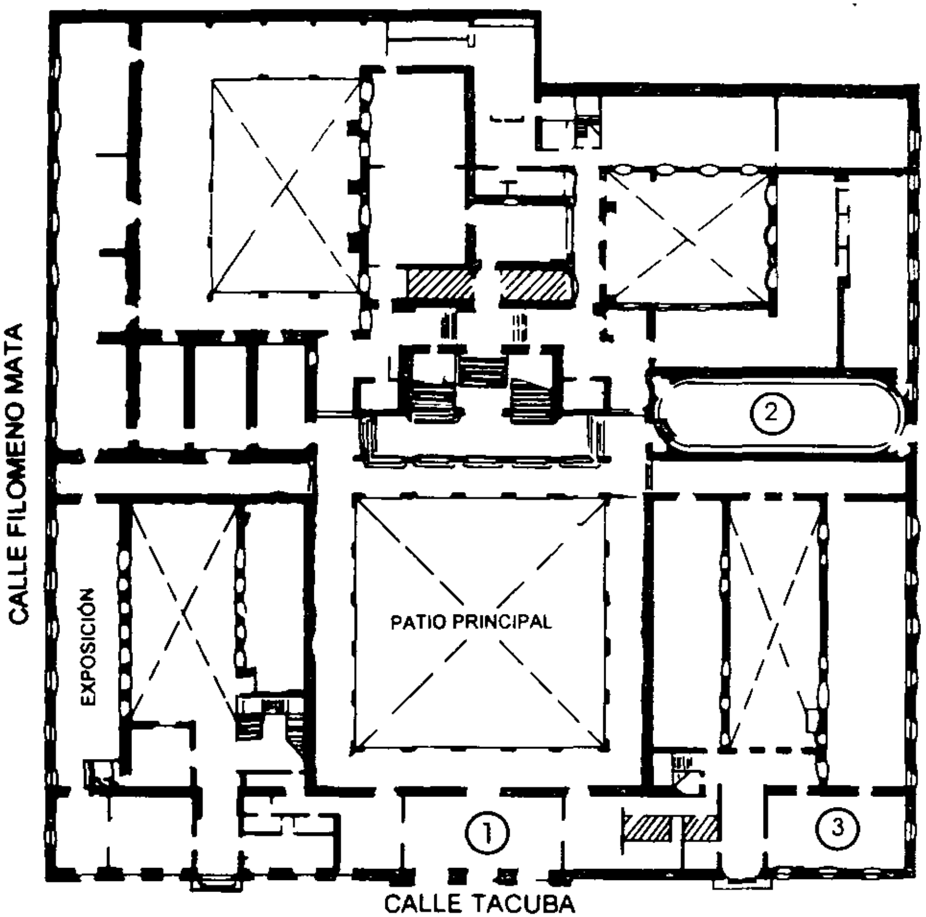
Con el objeto de mejorar los servicios que la División de Educación Continua ofrece, al final del curso deberán entregar la evaluación a través de un cuestionario diseñado para emitir juicios anónimos.

Se recomienda llenar dicha evaluación conforme los profesores impartan sus clases, a efecto de no llenar en la última sesión las evaluaciones y con esto sean más fehacientes sus apreciaciones.

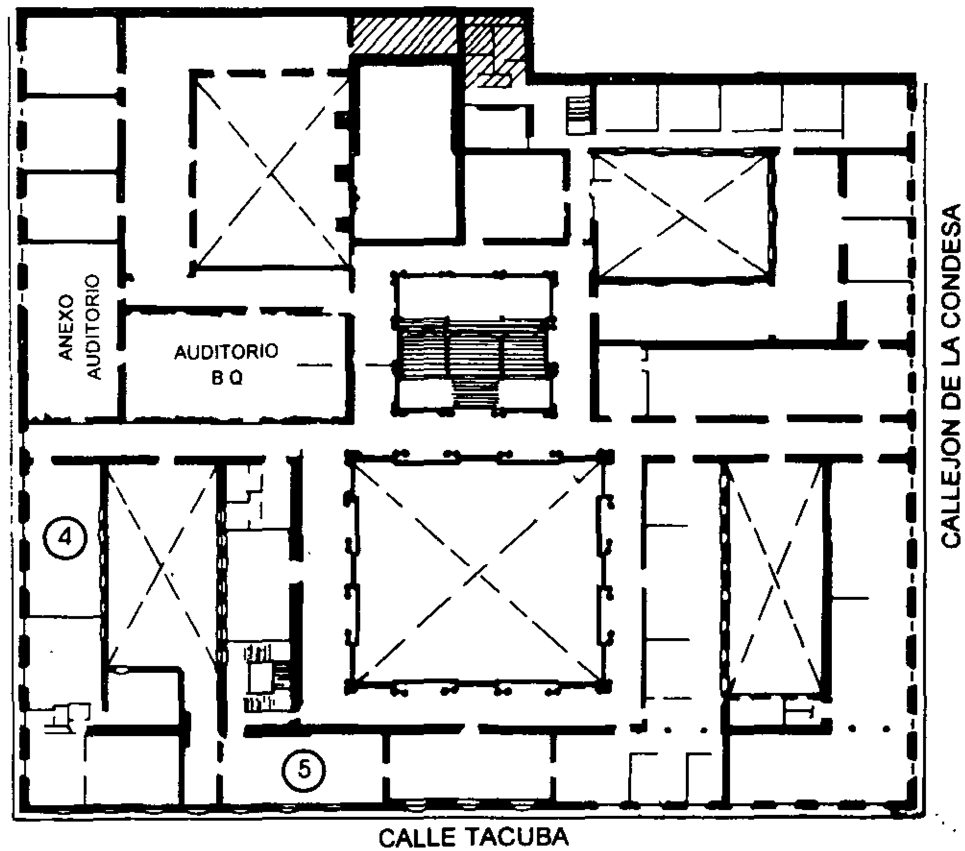
Atentamente
División de Educación Continua.

Palacio de Minería Calle de Tacuba 5 Primer piso Deleg. Cuauhtémoc 06000 México, D F APDO. Postal M-2285
Teléfonos: 5512-8955 5512-5121 5521-7335 5521-1987 Fax 5510-0573 5521-4021 AL 25

PALACIO DE MINERIA

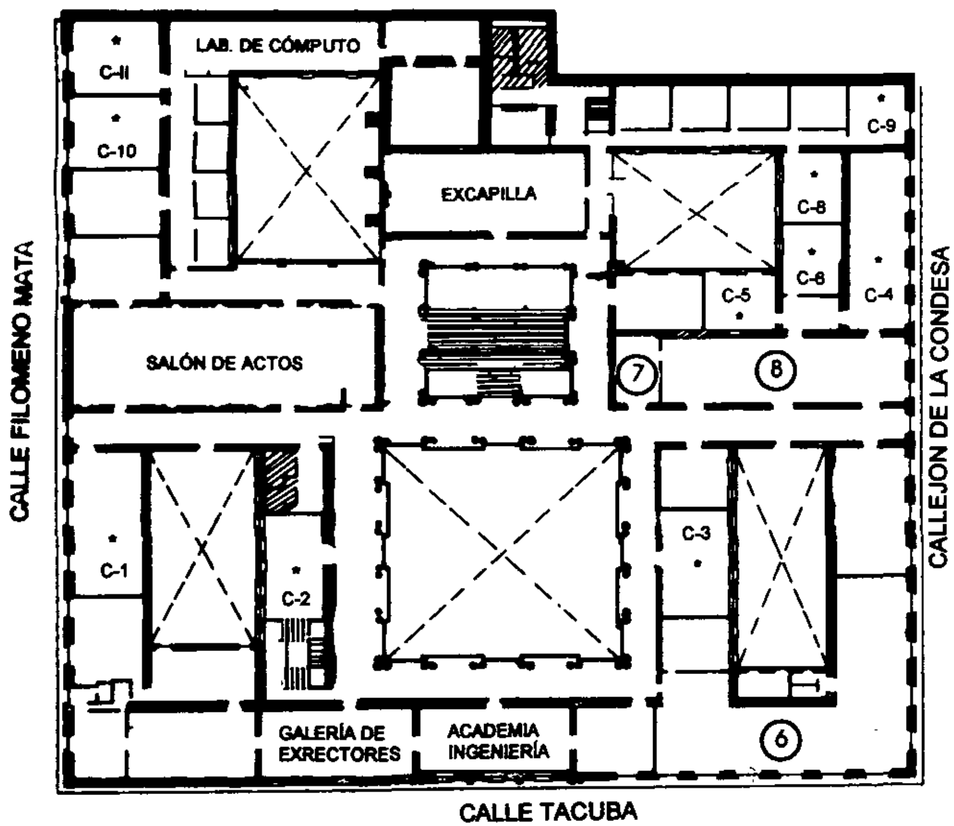


PLANTA BAJA



MEZZANINNE

PALACIO DE MINERIA



Ier. PISO

GUÍA DE LOCALIZACIÓN

1. ACCESO
2. BIBLIOTECA HISTÓRICA
3. LIBRERÍA UNAM
4. CENTRO DE INFORMACIÓN Y DOCUMENTACIÓN "ING. BRUNO MASCANZONI"
5. PROGRAMA DE APOYO A LA TITULACIÓN
6. OFICINAS GENERALES
7. ENTREGA DE MATERIAL Y CONTROL DE ASISTENCIA
8. SALA DE DESCANSO

SANITARIOS

* AULAS

DIVISIÓN DE EDUCACIÓN CONTINUA
FACULTAD DE INGENIERÍA U.N.A.M.
CURSOS ABIERTOS

DIVISIÓN DE EDUCACIÓN CONTINUA

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP – 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

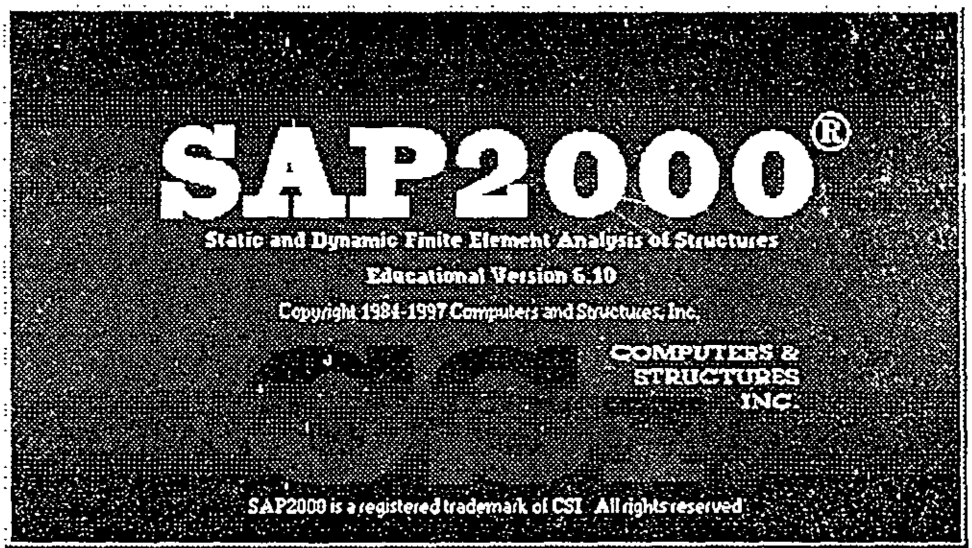
INSTRUCTIVO PARA LA UTILIZACIÓN DEL PROGRAMA DE CMPUTADORA SAP 2000

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
SEPTIEMBRE DEL 2001

UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO
FACULTAD DE INGENIERÍA

SAP-2000

(STRUCTURAL ANALYSIS PROGRAM
Análisis y Diseño Integrado de Estructuras por el Método de Elementos Finitos)



INSTRUCTIVO PARA LA UTILIZACION DEL PROGRAMA DE COMPUTADORA SAP 2000

FERNANDO MONROY MIRANDA
DIVISIÓN DE INGENIERÍA CIVIL, TOPOGRÁFICA Y GEODÉSICA
DEPARTAMENTO DE ESTRUCTURAS

UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO
FACULTAD DE INGENIERÍA

INSTRUCTIVO PARA LA UTILIZACION DEL PROGRAMA DE COMPUTADORA SAP 2000

Análisis y Diseño Integrado de Estructuras por el Método de Elementos Finitos

FERNANDO MONROY MIRANDA

DIVISIÓN DE INGENIERÍA CIVIL, TOPOGRÁFICA Y GEODÉSICA
DEPARTAMENTO DE ESTRUCTURAS
LABORATORIO DE CÓMPUTO ESTRUCTURAL.

PRÓLOGO

La serie de programas SAP son quizá los programas más conocidos, probados y utilizados en el campo de la Ingeniería Estructural, particularmente en el Análisis Estructural, desde las primeras versiones SOLIDSAP, SAP 3, SAP IV, etc., hasta la más reciente **SAP 2000**, han sido utilizadas por un gran número de ingenieros en nuestro país y en muchas partes del mundo, cuenta con respaldo y soporte técnico al que tiene derecho el usuario autorizado así como a los manuales respectivos.

Por lo anterior, desde hace algunos años el Departamento de Estructuras de la División de Ingeniería Civil Topográfica y Geodesia del la Facultad de Ingeniería de la UNAM consideró conveniente impartir una serie de cursos para enseñar a manejar el programa, para ello el contar con un instructivo que permita introducir al usuario de una manera fácil al programa facilitará el objetivo anterior, por lo que se sugiere que el lector asista a los cursos que organiza del Departamento de Estructuras o la División de Educación Continua de la FI de la UNAM.

En este instructivo se describen algunos de los principales elementos que intervienen en el uso del programa de computadora para Análisis y Diseño Estructural **SAP-2000**, cuya principal utilización será para los alumnos de la materia "Diseño Estructural" de la carrera de Ingeniero Civil que se imparte en la Facultad de Ingeniería de la UNAM.

Se ha procurado realizar este instructivo de una manera sencilla y resumida para que el usuario no emplee demasiado tiempo en leerlo y pueda resolver su problema en lo que respecta al Análisis y Diseño de Estructuras utilizando el programa **SAP-2000**.

Se recomienda que si algunos de los elementos no son descritos ampliamente se consulten los manuales respectivos o la ayuda en línea incluida en el programa y se observen los ejemplos que se desarrollan al final del instructivo. Se supone que el usuario esta familiarizado con la nomenclatura y terminología utilizada en el Análisis y Diseño Estructural y que cuenta con conocimientos básicos de computación en lo que respecta a manejo de información (archivos) y ejecución de programas en ambiente Windows 95, 98.

El autor agradece al Ing. Miguel Ángel Rodríguez Vega, Jefe del Departamento de Estructuras el apoyo para el desarrollo de este tipo de actividades, por las facilidades otorgadas para la realización de este trabajo así como la revisión del presente instructivo.

FERNANDO MONROY MIRANDA

Cd. Universitaria, Marzo del 2000

CONTENIDO

PRÓLOGO

CAPÍTULO 1 EL PROGRAMA SAP 2000

1.1 Introducción al programa SAP 2000

CAPÍTULO 2 RECOMENDACIONES PARA EL USO DEL PROGRAMA

2.1 Paso 1. Tipo de estructura

2.2 Paso 2. Definición de la geometría

2.3 Paso 3. Definición de las propiedades elásticas de los materiales

2.4 Paso 4. Definición de las propiedades geométricas de los elementos

2.5 Paso 5. Definición las características de las fuerzas y de las combinaciones

2.6 Paso 6. Elección del tipo de análisis y resultados

2.7 Paso 7. Diseño de elementos

CAPÍTULO 3 MÓDULOS DEL PROGRAMA DESCRIPCION GENERAL

3.1 Ejecución del programa, módulos que lo componen

3.2 Descripción general

CAPÍTULO 4 OPCIONES PARA LA GENERACIÓN DE LA ESTRUCTURA

4.1 Introducción

4.2 Descripción General

4.3 Generación de la Geometría

4.4 Definición y asignación de propiedades geométricas

4.5 Definición y Asignación materiales

4.6 Condiciones de Frontera, tipos de apoyo

4.7 Asignación de Fuerzas y combinaciones

4.8 Opciones de Análisis y Diseño, selección de resultados

CAPÍTULO 5 ANÁLISIS DE LA ESTRUCTURA

5.1 Verificando algunos elementos del proceso de análisis

CAPÍTULO 6 SELECCIÓN E INTERPRETACION DE RESULTADOS

6.1 Introducción

6.2 Ver la estructura deformada

6.3 Ver los diagramas de elementos mecánicos

6.4 Ver los resultados de diseño

6.5 Otras características

CAPÍTULO 7 OPCIONES ADICIONALES

7.1 Introducción

7.2 Ver el archivc de entrada

7.3 Ver el archivo de salida

7.4 Relación con AUTO CAD

CAPÍTULO 8 EJEMPLOS E INTERPRETACIÓN DE RESULTADOS

Ejemplo No. 1

Ejemplo No. 2

Ejemplo No. 3

Ejemplo No. 4

Ejemplo No. 5

CAPÍTULO 9 COMENTARIOS FINALES

EL PROGRAMA SAP 2000

CAPÍTULO I

1.1 INTRODUCCIÓN

En los últimos años, el desarrollo de los equipos y sistemas de computo ha permitido una comunicación mucho más rápida, directa y sencilla entre el usuario y la computadora logrando la posibilidad de desarrollar programas que, utilizando las características de las computadoras de hoy en día, nos permitan usarlas mas eficientemente y entre otras cosas facilitándonos la posibilidad de explorar varias alternativas de solución de problemas estructurales o bien considerar más variables en el comportamiento de las estructuras con el objeto de lograr un mejor modelo de la estructura.

Tomando en cuenta lo anterior, **SAP 2000** es el resultado de un trabajo desarrollado en los Estados Unidos de Norteamérica cuyo principal objetivo fue desarrollar un programa para Análisis y Diseño de Estructuras en donde el usuario tenga gran versatilidad en el manejo del mismo a través de una interacción directa en la mayor parte de la ejecución de los módulos que componen el programa y junto con la sencillez y facilidad de uso son algunas de sus principales características.

El Sistema **SAP 2000** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis y Diseño de Estructuras bajo uno o varios sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura.

SAP 2000 fue desarrollado bajo la hipótesis de que la estructura está formada por barras prismáticas (aunque también maneja cierto tipo de barras de sección variable) de eje recto, considerando también la posibilidad de modelar elementos placa y sólido (Elementos finitos)

Consta básicamente de una serie de menús (Véase Figura 1) que se despliegan en la pantalla al inicio del programa y por lo general después de terminada la ejecución de cada una de las opciones, con ellas, el usuario puede introducir y/o modificar datos, o bien almacenarlos para su procesamiento posterior, analizar la estructura, ver resultados en la pantalla o imprimirlos, ver resultados de diseño, etc

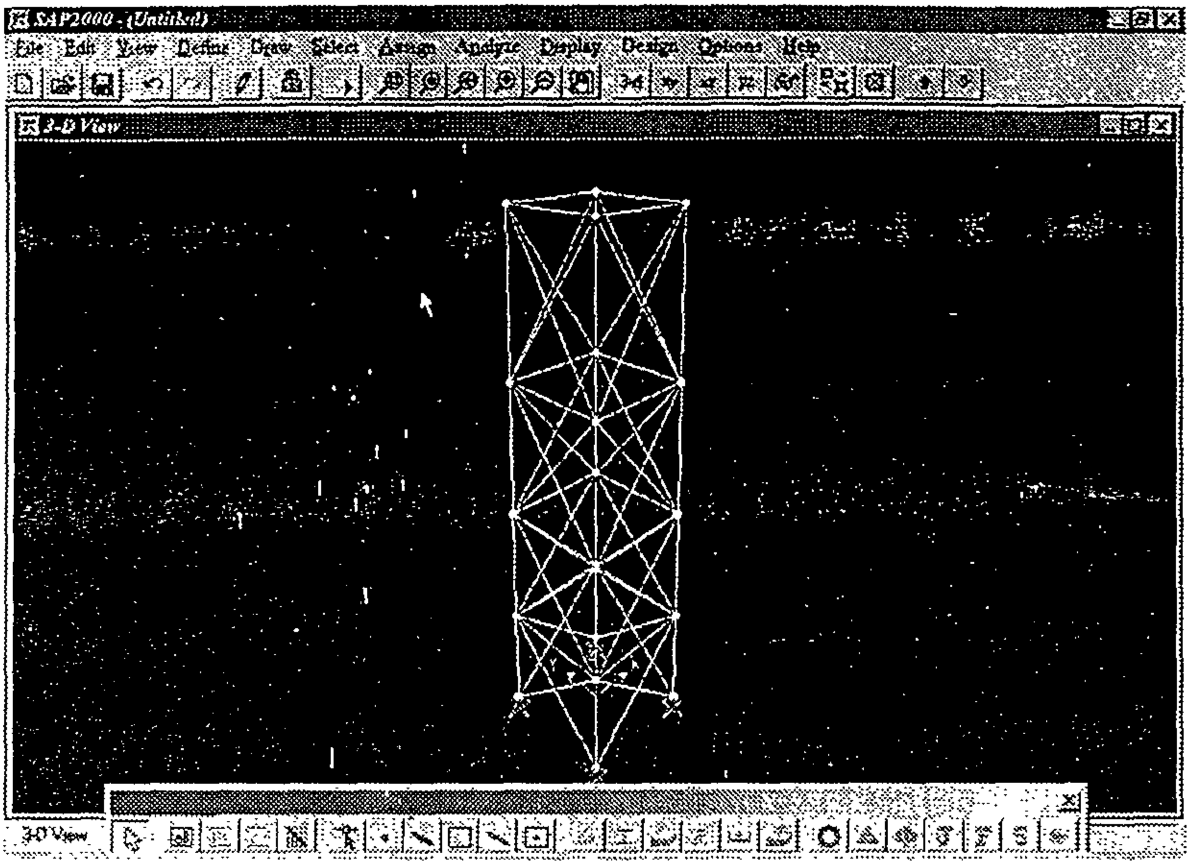


Figura 1.1 SAP 2000, menú principal.

Una de las principales características del programa es la interacción que se puede establecer entre éste y el usuario, y debido al número de opciones que el usuario puede activar, se requiere aprender su lenguaje específico para poder utilizarlo, ya que, el usuario puede seleccionar varias opciones y la ejecución de cada una de ellas genera otras más, **SAP 2000** es un programa orientado a eventos (seleccionar un elemento con el ratón, elegir una opción, activar/desactivar sucesos, etc.) y no siempre solicita textualmente los elementos (datos) que se vayan requiriendo para la ejecución completa de ese módulo, por otro lado además es necesario saber las convenciones de signos empleadas, los sistemas de referencia utilizados así como algunas recomendaciones para su uso, éstas y algunas caracteristicas más son descritas en los capítulos posteriores.

En el capítulo 2 se dan las recomendaciones necesarias para facilitar la preparación e introducción de datos, en el capitulo 3 se comentan los módulos que componen el programa, el capítulo 4 describe el módulo para crear la estructura, en el capítulo 5 se presenta el módulo de análisis, en el capítulo 6 se presentan las opciones para ver resultados del Análisis y Diseño, en el capítulo 7 se describen algunas opciones adicionales o complementarias, el capítulo 8 contiene algunos ejemplos con la correspondiente interpretacion de los resultados obtenidos por el programa **SAP 2000**, por último en el capítulo 9 se incluyen algunos comentarios y sugerencias finales.

RECOMENDACIONES PARA EL USO DEL PROGRAMA

CAPÍTULO 2

2.1 INTRODUCCION

El programa **SAP 2000** posee una interfase gráfica como una opción que le permite al usuario modelar, analizar, diseñar y desplegar tanto datos como resultados de una estructura, una vez que se cuenta con los datos de geometría, propiedades de los materiales de los cuales están hechos los elementos estructurales así como las cargas y desde luego un completo y correcto entendimiento del problema, se esta en condiciones de utilizar el programa, para ello habrá necesidad de modelar a los elementos anteriores, una vez definido el modelo que se utilizará para esos elementos se introducirá el modelo completo utilizando por ejemplo la interfase gráfica.

La estructura idealizada estará formada por:

- Elementos barra (FRAME) usados para representar a las vigas, columnas, diagonales, etc.
- Elementos placa (SHELL) usados para representar muros, losas, rampas, etc
- Elementos sólidos (SOLID) usados para modelar estructuras continuas tridimensionales.
- Nudos (JOINTS) que representan la conexión entre los elementos barra, placa y sólido.
- Propiedades fisicas y elásticas de los materiales
- Apoyos y resortes que representan las restricciones de desplazamiento del nudo.
- Cargas (concentradas, uniformes, etc) que representan a las acciones (peso propio, viento, sismo, ocupación, etc.).

2.1 PASO 1. TIPO DE ESTRUCTURA

SAP 2000 permite manejar a la estructura en un sistema coordenado tridimensional, sin embargo, antes de realizar el análisis se pueden seleccionar determinados grados de libertad (ver figura 2 1) y así aunque la estructura este referida a un sistema tridimensional se pueden analizar:

Marcos y vigas en un plano vertical
Retículas (en un plano horizontal)

Desde luego se permite modelar y analizar Armaduras y marcos tridimensionales.

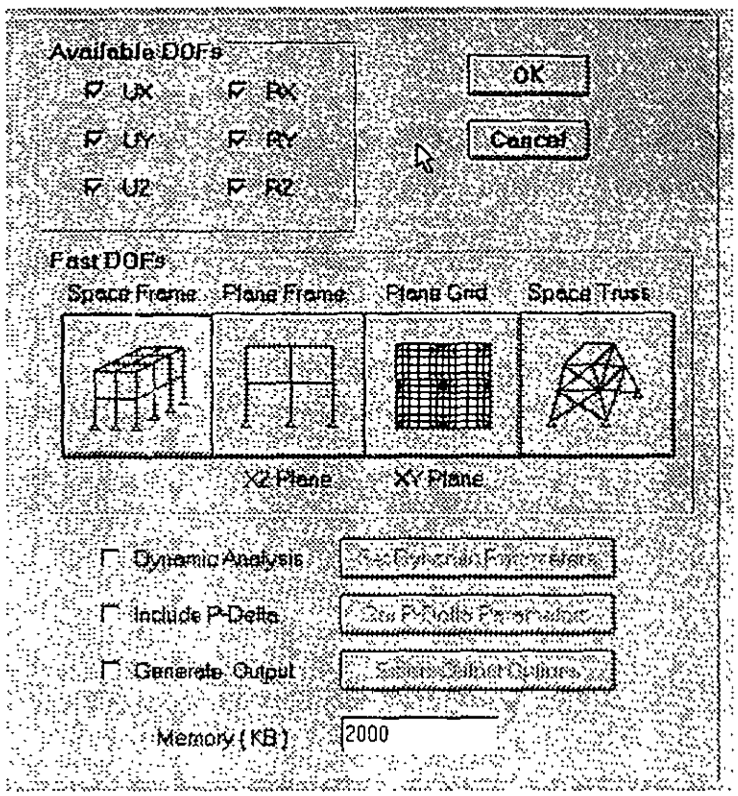


Figura 2.1. Selección de grados de libertad de acuerdo al tipo de estructura

Para el caso de las estructuras tipo armadura sólo se considerará el efecto axial en el análisis.

En las estructuras planas se consideran cortante y axial en el plano de la estructura y flexión perpendicular a ese plano.

El tipo retícula permite analizar estructuras con acciones perpendiculares a su plano considerando flexión en el plano, torsión y cortante

El caso general lo constituye el tipo marco tridimensional en donde sé consideran flexión y cortante en dos direcciones, torsión y axial con seis grados de libertad por nudo, desde luego que se pueden liberar extremos de las barras a algún elemento mecánico y suprimir o ligar grados de libertad (diafragma rígido por ejemplo).

2.2 PASO 2. DEFINICIÓN DE LA GEOMETRÍA

Antes de iniciar la ejecución del programa SAP 2000 es conveniente como segundo paso definir completamente la geometría del modelo. La estructura real se idealizará mediante una serie de elementos estructurales conectados entre sí, los cuales, de acuerdo a sus características se podrán modelar como elementos barra (trabes, columnas, diagonales), elementos placa (losas, muros) o elementos sólidos tridimensionales (elementos continuos), estos elementos estarán unidos en puntos comunes (nudos), algunos nudos estarán completamente o parcialmente restringidos (apoyos), en uno o varios grados de libertad.

La definición de los elementos (barra, placa, sólido, etc.) se logra localizando sus nudos extremos (incidencias) en un sistema coordenado cartesiano proporcionando las coordenadas de esos nudos

No es necesario numerar en ningún orden a los nudos que forman parte de la estructura ya que el programa los numera. Es conveniente localizar nudos en donde se tenga cambio de propiedades geométricas o elásticas, recordando que el elemento barra requiere de dos nudos para localizarlo, el elemento placa 3 ó 4 y el sólido comunmente 8 nudos.

Como se verá posteriormente el editor gráfico permite introducir la geometría de la estructura de una manera bastante sencilla y directa, ya que con la ayuda del "ratón" (dispositivo tipo puntero o *mouse*) simple y sencillamente por ejemplo haciendo clic en las coordenadas de los puntos extremos de la barra automáticamente se definen sus incidencias así como las coordenadas de esos nudos.

2.3 PASO 3. DEFINICIÓN DE LAS PROPIEDADES GEOMÉTRICAS DE LOS ELEMENTOS

SAP 2000 permite manejar una gran variedad de formas predefinidas para la sección transversal de las barras que componen la estructura (ver figura 2.2), como por ejemplo·

- Secciones I, canal, T, ángulos, ángulos dobles, cajón, tubos, etc.
- Secciones rectangulares, circulares.
- Secciones cualquiera (proporcionando sus propiedades)
- Sección no prismáticas (propiedades variables).

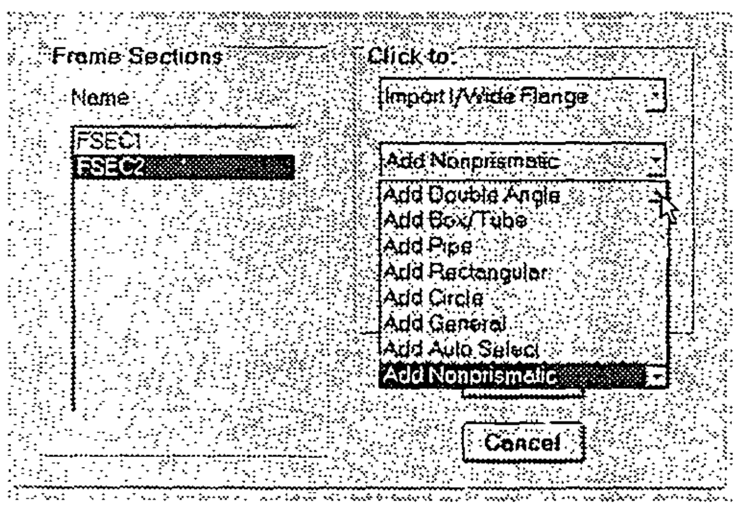


Figura 2.2. Algunas formas para la sección transversal de las barras

Una vez elegida la forma de la sección transversal será necesario introducir los datos relativos a las dimensiones (tamaño) de la forma seleccionada (ver figura 2.3).

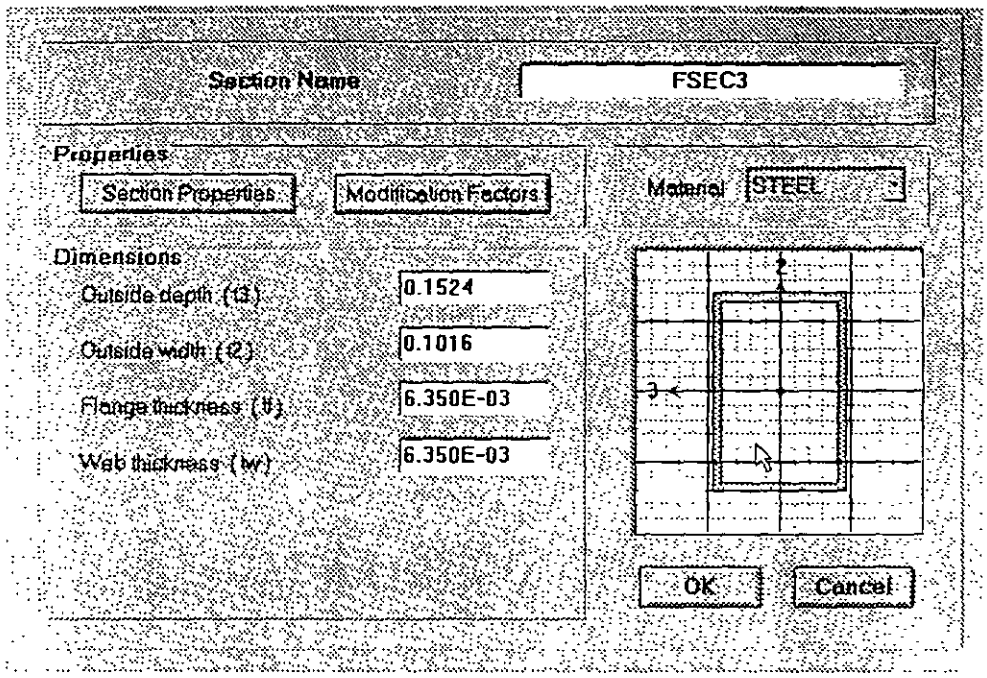


Figura 2.3. Dimensiones de una forma de sección transversal específica.

Para los elementos barra prismáticos (general) de una estructura tridimensional se requiere proporcionar las siguientes propiedades referidas a ejes locales, centroidales y principales de la barra (ver figura 2.4).

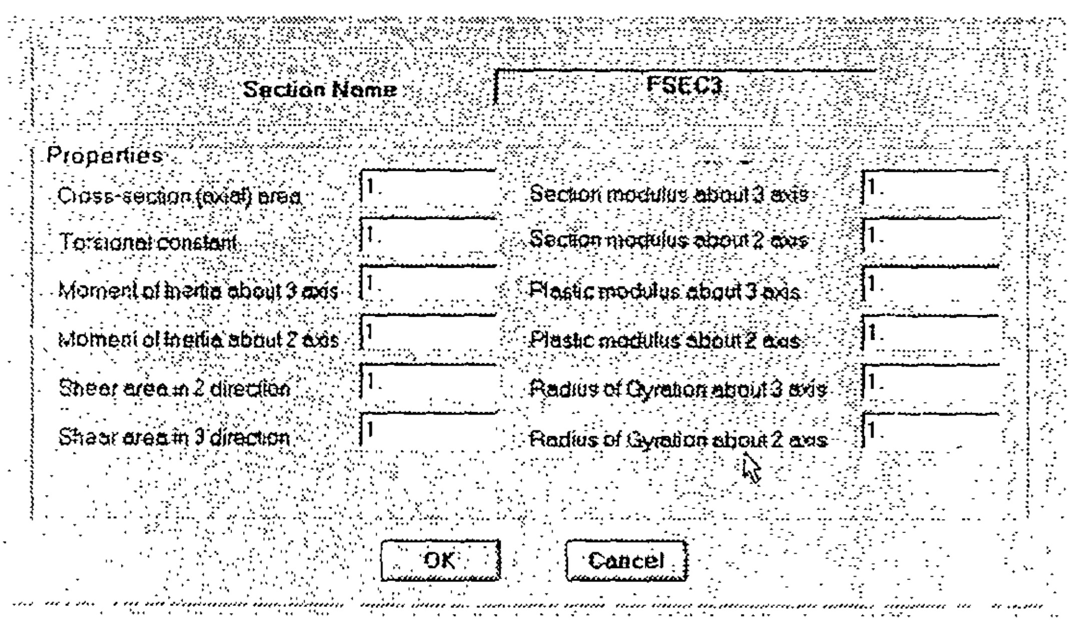


Figura 2.4 Características del tipo de sección transversal "general"

Dependiendo del tipo de estructura, en la tabla 2.1 se muestran las propiedades geométricas mínimas que es necesario proporcionar para que el análisis se pueda realizar.

Tipo de estructura	Propiedad requerida
TRUSS	AX
PLANE	AX, IZ ó IY
FLOOR	IX, IZ ó IY
SPACE	AX, IX, IY, IZ

Tabla 2.1 Propiedades geométricas mínimas requeridas.

El programa **SAP 2000** permite asignar las propiedades de los elementos barra de acuerdo a una tabla de perfiles de acero estándar (P. ej. tabla AISC, ver figura 2.5) o tomarlas de una tabla definida por el usuario.

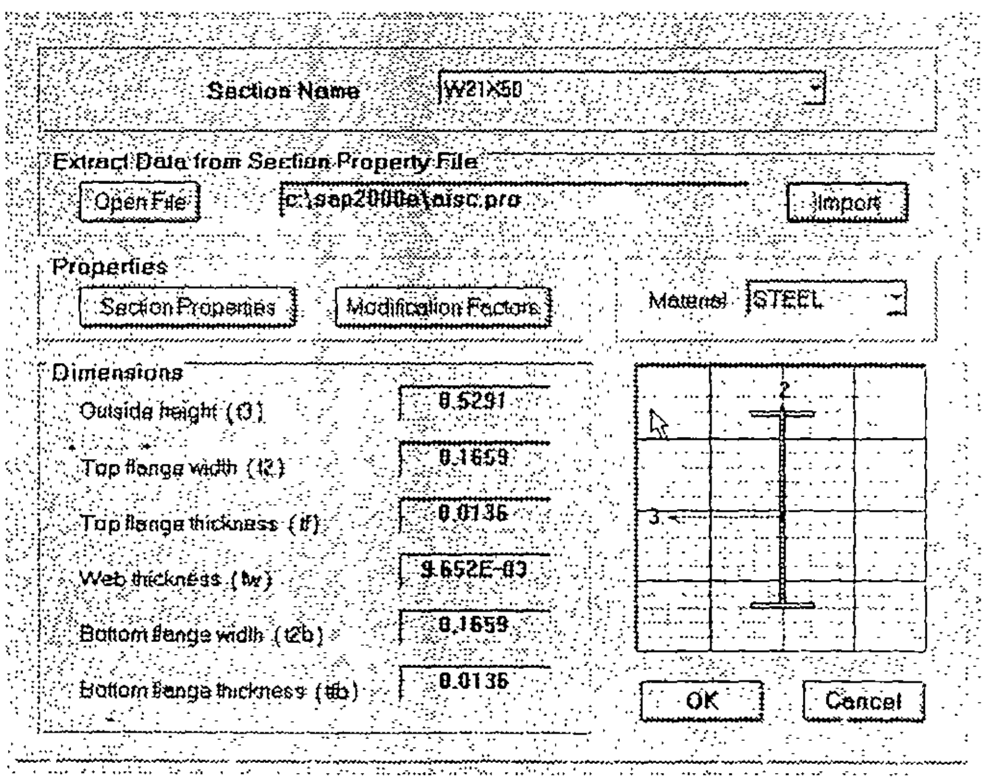


Figura 2.5. forma y propiedades geométricas tomadas de una tabla de perfiles.

Para el caso de los elementos placa será necesario proporcionar el espesor de la placa y seleccionar el tipo de trabajo de esta ("Shell", "Membrane" o "Plate", ver figura 2.6), para el sólido no es necesario proporcionar propiedades geométricas, sólo constantes elásticas.

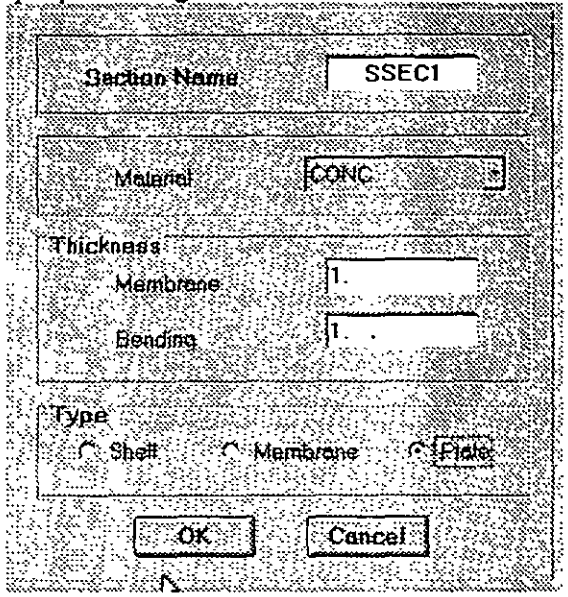


Figura 2.6. Datos para los elementos placa.

2.4 PASO 4. DEFINICIÓN DE LAS PROPIEDADES ELÁSTICAS DE LOS MATERIALES

Para realizar el análisis se requiere tener definidas las constantes del material del cual están o estarán hechos los elementos (barra, placa, sólido) como son E (Módulo elástico), y ν (relación de Poisson). Para incluir el peso propio es necesario proporcionar el peso volumétrico, si se desea emplear alguna opción de análisis dinámico entonces es necesario proporcionar la masa por unidad de longitud (en un modelo de masas distribuidas), masas en los nudos (modelo de masas concentradas), si quiere que se considere efectos de temperatura será necesario especificar el coeficiente lineal de dilatación térmica (ver figura 2.7).

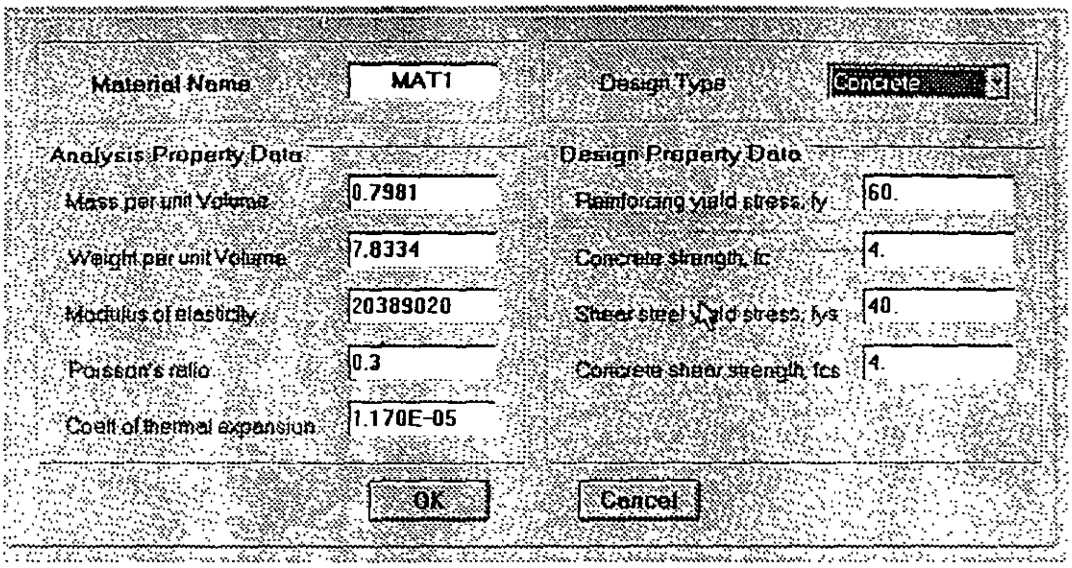


Figura 2.7. Datos para las propiedades de un material.

2.5 PASO 5. TIPOS DE FUERZAS Y COMBINACIONES

Es necesario tener completamente identificados comunmente los sistemas o conjuntos de fuerzas (condiciones de carga) bajo los que se realizará el análisis (P. ej. peso propio, carga viva, sismo, viento, etc.) y para cada condición de carga las características de las fuerzas (tipo, magnitud, dirección, etc.) que forman parte de ese sistema de fuerzas.

Por ejemplo una condición de carga puede ser la carga muerta que puede estar formada por ejemplo por: fuerzas uniformes en algunas barras simulando el peso de los muros divisorios, fuerzas concentradas simulando el peso de tanques, etc.

Otra condición de carga puede ser el sismo, que por ejemplo pudiera ser representado por una serie de fuerzas estáticas (sismo estático) aplicadas en determinados nudos.

Una condición más puede ser la carga viva, idealizada como una fuerza por unidad de área actuando en una determinada zona de la estructura (P. ej azotea, entrepiso, escaleras, etc.).

Los sistemas de carga independientes pueden ser utilizados para formar sistemas de carga dependientes es decir combinaciones, si lo anterior se desea, es necesario saber de antemano el número de combinaciones a incluir en el análisis y para cada combinación las condiciones de carga que se incluirán así como su participación respectiva (factor de carga), por ejemplo teniendo como marco al Reglamento de Construcciones para el D.F. pensando en una estructura del grupo A, localizada en el D F. una combinación es 1.5 de la carga muerta + 1.5 de la carga viva máxima, por lo que el factor de carga (o de participación) de las condiciones anteriores (1 y 2) es 1.5, siendo 1 y 2 las condiciones de carga respectivas.

2.6 PASO 6. ELECCIÓN DEL TIPO DE ANÁLISIS Y RESULTADOS

SAP 2000 permite realizar un análisis elástico lineal de 1er. orden, también se pueden incluir efectos P-Δ o bien un análisis dinámico, por lo anterior habrá que decidir sobre el tipo de análisis a efectuar por el programa.

En cuanto a los resultados que el programa puede proporcionar, será necesario saber cuales se requerirán, por ejemplo: desplazamientos, elementos mecánicos, gráficas y diseño, y de que elementos se requieren; por ejemplo: de algunos o de todos los nudos, de algunos o de todas las barras, gráficas de la deformada, de algún marco o de toda la estructura, etc., lo anterior se tendrá que definir para una, algunas o todas las condiciones de carga y/o combinaciones. Si el usuario no selecciona o define los elementos (nudos, barras, etc.), condiciones y/o combinaciones la impresión la realiza para todos los elementos y todos los sistemas de fuerzas existentes.

2.7 PASO 7. DISEÑO DE ELEMENTOS

SAP 2000 permite diseñar elementos de acero y concreto por lo que será necesario definir un código o especificaciones a utilizar (ACI, AISC, LRFD, ASSTHO, etc.) y proporcionar los valores de los parámetros a utilizar (f'c, fy, etc.), así como especificar los elementos que se diseñarán y el criterio a seguir para su diseño (viga, columna, etc.).

DESCRIPCION GENERAL

CAPÍTULO 3

3.1 INTRODUCCION

Una vez que se ha modelado la estructura (previo al uso del programa), es decir, seleccionada la forma de la sección transversal de las barras, definidas las características fisicas y mecánicas de los materiales estructurales, especificados los sistemas de fuerzas (definidas cada una de las fuerzas que componen a cada sistema o condición de carga y combinaciones) bajo las cuales se analizará el modelo estructural, seleccionado el tipo de análisis así como el tipo de resultados, entonces se esta en condiciones de introducir los datos antes mencionados utilizando la interface gráfica que ofrece el programa con la cual es posible:

Manejar (Definir, mover, copiar, borrar) elementos estructurales (barra, placa, etc.).

Definir Tipos de apoyo (fijo o con grados de libertad, resortes).

Definir y asignar propiedades geométricas a los elementos barra de acuerdo a una tabla de perfiles estándar (AISC por ejemplo) o usar secciones prismáticas (circular, rectangular, Te, etc.), también es posible la utilización de secciones no prismáticas o de sección variable.

Definir el espesor de los elementos placa.

Definir y asignar propiedades a uno o varios elementos o grupo de elementos (barra, placas), las propiedades pueden ser densidad, módulo elástico, relación de Poisson, coeficiente de dilatación térmica, etc. Así como definir la posición de la sección dentro de la estructura (posición de ejes locales con respecto a los globales). Algunas de las propiedades se tienen predefinidas para ciertos materiales (acero y concreto) o se pueden introducir valores particulares.

Es posible seleccionar barras para liberarlos de algunos elementos mecánicos en sus extremos, también se pueden definir diafragmas rígidos.

Desde luego se permite introducir fuerzas estáticas aplicadas a los nudos, desplazamientos prescritos en ellos, en el caso de barras se puede incluir el peso propio, fuerzas uniformes; concentradas, con variación lineal, de presfuerzo y debidas a incrementos de temperatura, a ajustes en la longitud inicial de los elementos y algunas otras.

Además de las fuerzas de tipo estático, se puede incluir cargas variables (móviles), de acuerdo a AASHTO (HS2O, HS15, H20, HI5, etc.), o al UBC, o bien especificadas por el usuario Una buena variedad de fuerzas dinámicas (fuerza-tiempo o aceleración-tiempo) pueden incluirse como sistemas de fuerzas, especificadas de acuerdo a sus características dinámicas (amplitud y frecuencia), definiendo el lapso de tiempo de actuación de la fuerza.

Una vez introducida la geometría, propiedades y fuerzas que actúan sobre la estructura, **SAP 2000** permite la realización del Análisis operando sobre el contenido del archivo que se ha seleccionado o definido previamente el cual desde luego debe contener los datos de la estructura en estudio, el módulo de análisis interpreta cada una de las ordenes o definiciones indicadas en el archivo de datos en el orden en que se encuentran, el contenido del archivo de datos e instrucciones puede introducirse manualmente vía algún editor previo a la ejecución de **SAP 2000** o bien mediante la instrucción **Save** al estar creando la estructura a través del editor gráfico característico del programa, ambas opciones se describirán posteriormente.

Después de ejecutada la opción de análisis, **SAP 2000** genera archivos conteniendo los resultados de la fase de análisis, si este concluye satisfactoriamente se desplegará la configuración deformada de la estructura. Enseguida se podrán seleccionar opciones y elementos para que de ellos se muestren en el monitor los resultados numéricos y gráficos obtenidos por el programa como resultado del análisis.

3.2 EJECUCIÓN DEL PROGRAMA, MENU DE OPCIONES

Para iniciar el programa se puede hacer doble clic en el icono del programa o bien desde el menú de inicio hacer clic en la carpeta programas **SAP 2000 educacional** (versión educativa) o **SAP 2000 NonLinear** (versión profesional), enseguida se ejecuta el programa presentándose la imagen mostrada en la figura 3.1, una vez haciendo clic en la caja OK de la ventana en la parte central ("*Tip of the day*") desaparece esta dejando lugar a la ventana principal del programa **SAP 2000**.

En el "renglón" superior de esta ventana se encuentra en su extremo izquierdo el nombre del programa (**SAP2000**) seguido del nombre de archivo en donde se almacenarán los datos o de donde han sido tomados, en el extremo derecho se encuentran los iconos de minimizar, restaurar la ventana y cerrarla (una forma de finalizar la ejecución del programa es haciendo clic en este icono), debajo de lo anterior se localiza la barra de menú conteniendo las opciones que el programa tiene disponibles (File, Edit, View, etc.) las cuales se describirán posteriormente, debajo de esas opciones se encuentran una serie de iconos que realizan acciones de uso frecuente (seleccionar elementos, cambiar alguna opción de presentación, elegir algún tipo de resultado, etc.), se recomiendo al lector consultar las tablas que se presentan al final de este trabajo en donde se describe cada uno de esos iconos (incluyendo los de la barra flotante que también forma parte de la ventana de **SAP 2000**).

Debajo de los iconos está el área de presentación (con fondo negro) en la que se muestra gráficamente el modelo de la estructura por analizar así como diversa información en forma de ventanas que seran desplegadas por el programa después de que el usuario seleccione alguna de las opciones disponibles de **SAP 2000**.

Por último, en la parte inferior debajo de la barra flotante de iconos se muestra información acerca de las características del área de dibujo (vista o plano de presentación, coordenadas de algún nudo, etc.) y un poco a la derecha esta el cuadro de selección de unidades en las que se introducirá la información, antes de este cuadro se muestra información acerca del estado que guarda alguna instrucción o del programa.

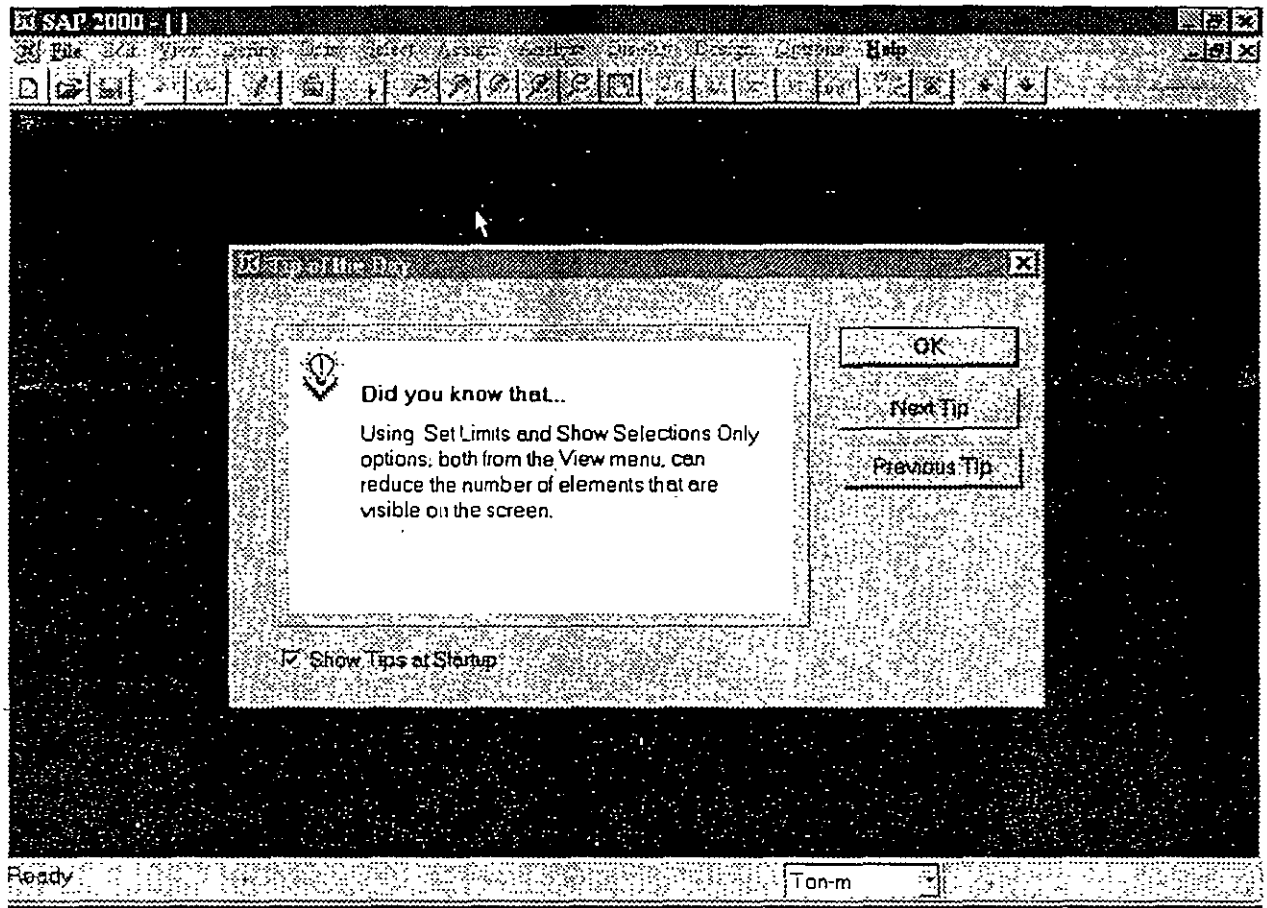


Figura 3.1 Iniciando el programa **SAP 2000**

En la versión 6.1 del programa SAP 2000 se pueden seleccionar varias opciones, las que se describen a continuación pueden ser las de uso más frecuente.

3.3 El menú File

EL menú File (ver figura 3.2) permite entre otras opciones manejar la información de alguna estructura contenida en un archivo, esa información pudo haberse generado previamente a la ejecución del programa o durante su uso, las opciones de este menú permiten:

New Model	Iniciar un problema nuevo.
New Model from template	Iniciar un problema nuevo, seleccionando una geometria tipica de algunas formas estructurales como las mostradas en la figura 3.3
Open...	Abrir un archivo existente con datos de alguna estructura.

Save	Guarda los datos de la estructura.
Save As	Guarda los datos de la estructura en otro archivo.
Import	Permite ingresar los datos de un archivo generado con AutoCad, o bien para SAP90.
Export	Proporciona la flexibilidad de poder enviar los datos de la estructura existente a una archivo para **SAP2000** con extensión .S2K el cual puede ser modificado por ciertos procesadores de texto (p.ej. WordPad) y poder ser utilizado nuevamente por **SAP2000,** o bien enviarlos a un archivo .DXF y poder ser interpretado por AutoCad por ejemplo.
Print...	Nos permite configurar características de impresión, imprimir el contenido del área de dibujo así como una lista de datos y resultados.
Exit	Cerrar el programa y regresar a Windows.

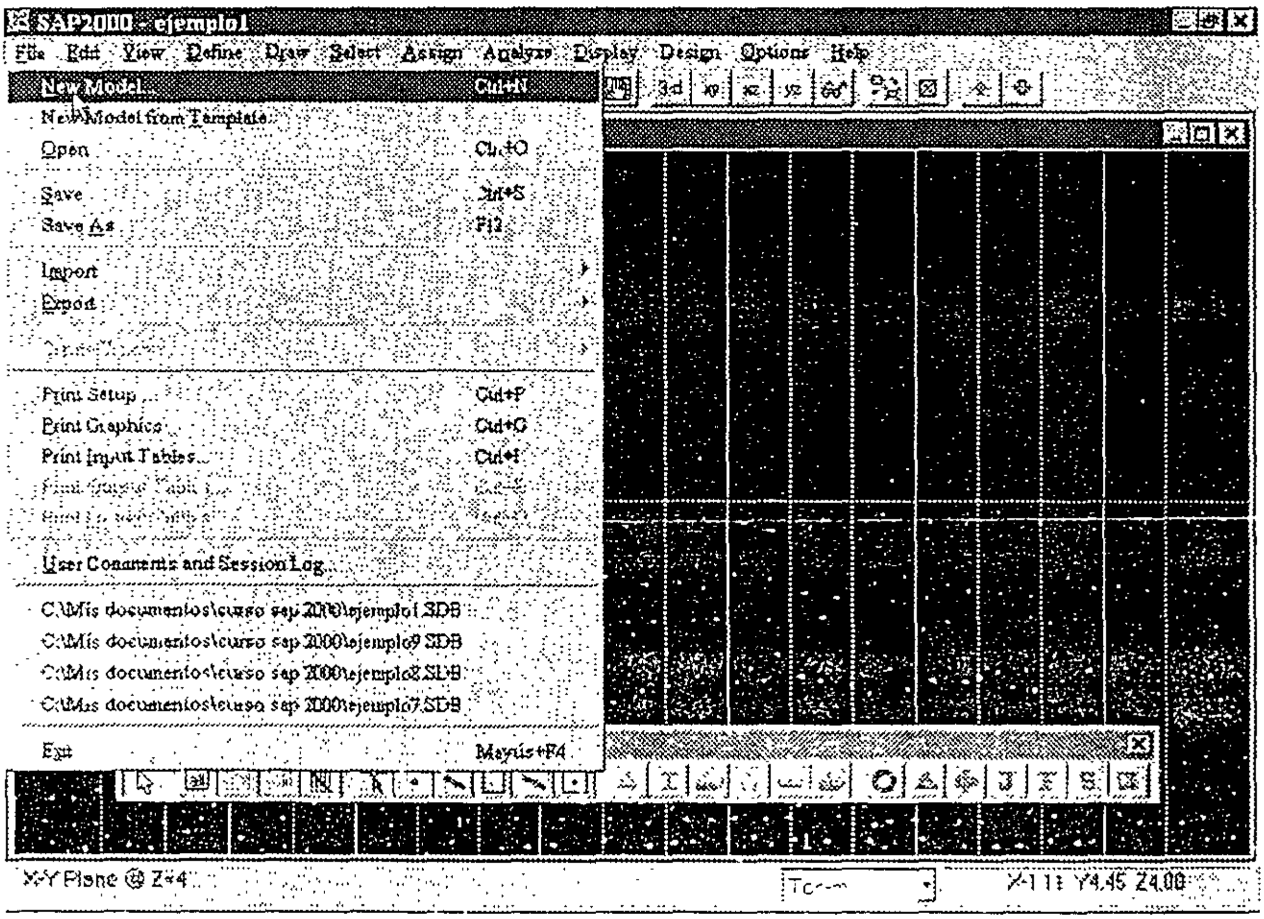


Figura 3.2 Módulos principales del menú **File**.

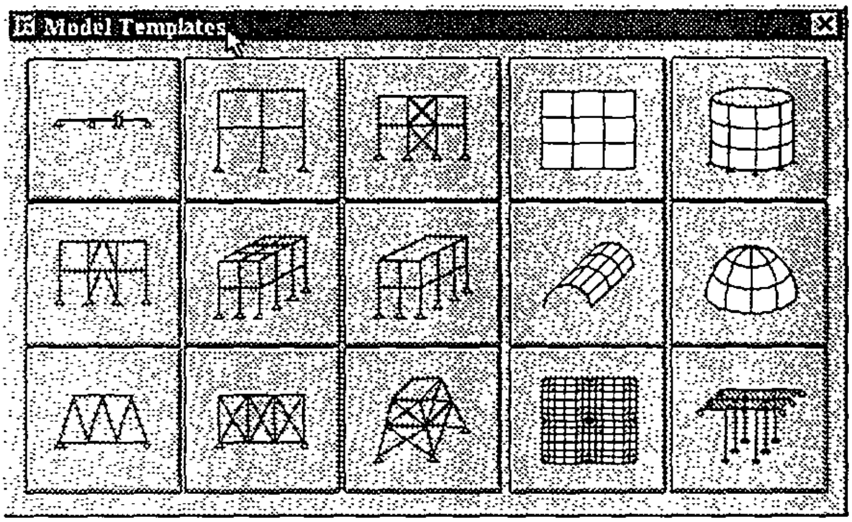


Figura 3.3. Geometrías predefinidas en la opción **New Model from Template**

Existen dentro de este menú otras opciones de uso no muy frecuente. Al iniciar **SAP 2000** se recomienda seleccionar las unidades en las que se van a introducir los datos de la estructura a analizar, por ejemplo si estas fueron ton-m (toneladas y metros) los valores de las fuerzas uniformes se deben de proporcionar en ton/m, de las inercias en m^4, para el módulo elástico en ton/m^2, etc., es decir los valores deben ser consistentes.

3.3 El menú Edit

EL menú Edit (ver figura 3.4) permite desde introducir y hacer cambios a la geometría del modelo hasta suprimir algunos de sus elementos muchas de las opciones contenidas en este menú operan en conjunto con las del menú Select (ver siguiente sección), las opciones de este menú permiten:

Cut Suprimir los elementos seleccionados, guardándolos en la memoria temporal.

Copy Copiar sin borrar los elementos seleccionados a la memoria temporal.

Paste Insertar los elementos contenidos en la memoria temporal especificando nuevas posiciones.

Delete Suprimir los elementos seleccionados.

Merge Joints Juntar los nudos que tengan una separación menor que un cierto valor (dejando uno solo y suprimiendo los demás es decir los nudos duplicados).

Move Mueve los nudos seleccionados especificando el incremento en sus coordenadas, moviendo también los elementos que estén conectados a esos nudos.

Replicate Realiza una copia (réplica) de los elementos seleccionados especificando el incremento en las coordenadas de sus nudos extremos.

Divide frames Divide a las barras seleccionadas en un número especificado por el usuario.

Join frames Junta varias barras seleccionadas en una sola (operación inversa de Divide frames).

Change Labels Cambia la numeración de los elementos seleccionados (renumera).

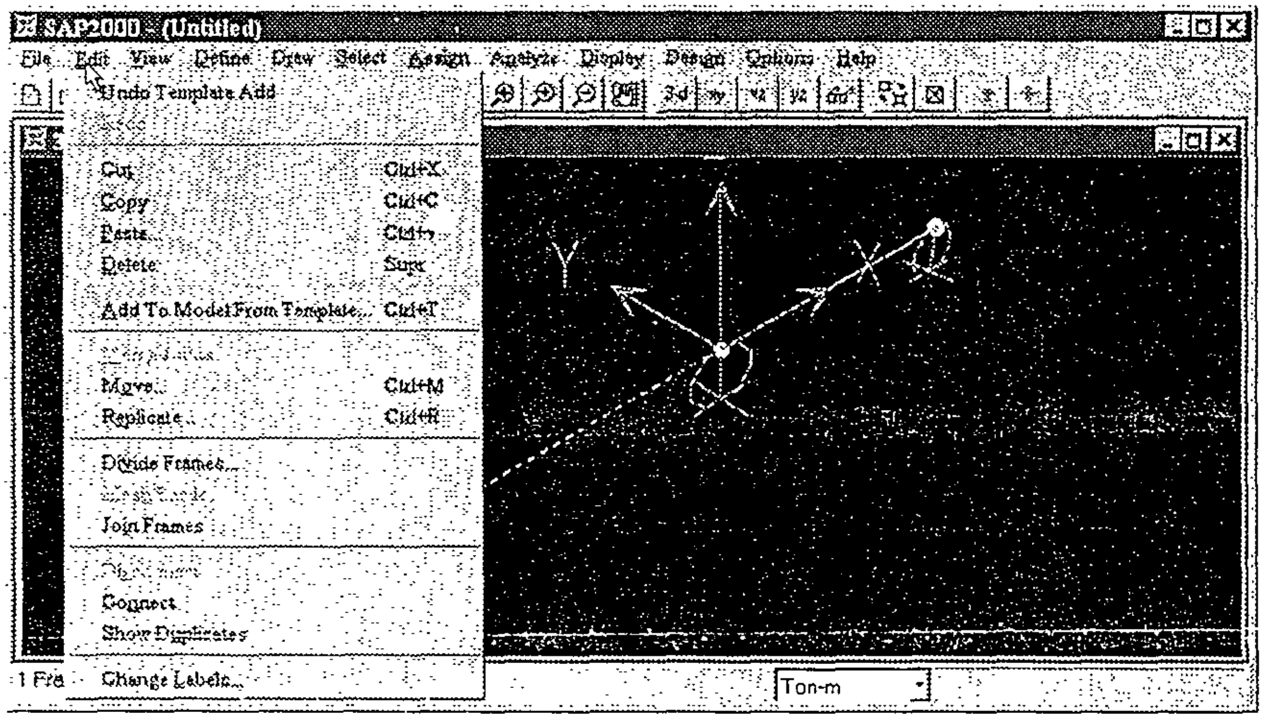


Figura 3.4. Opciones del menú **Edit**.

3.4 El menú View

EL menú View (ver figura 3.5) permite cambiar la presentación del área de dibujo de la estructura, algunas opciones que resultan de uso cotidiano son

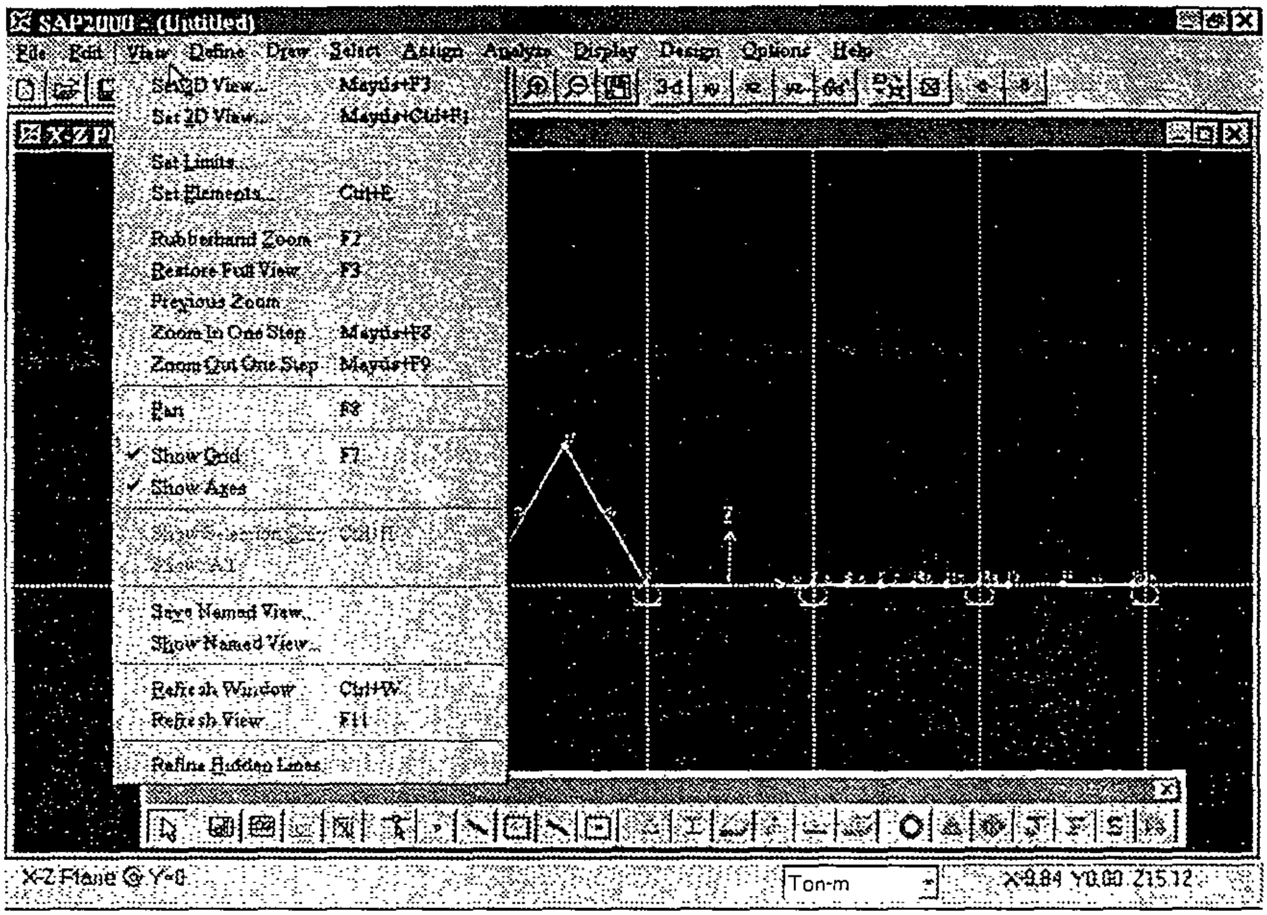


Figura 3.5. Opciones del menú View.

Set Elements Permite seleccionar la información a ser incluida dentro del área de dibujo (numeración de nudos, barras, ejes, etc.), ver figura 3.6

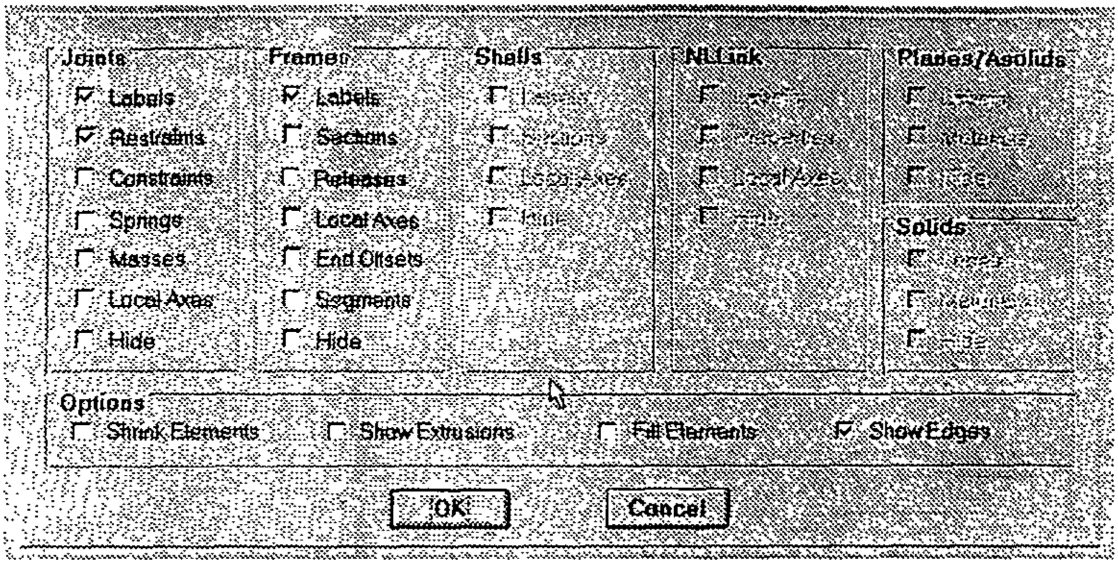


Figura 3.6. Opciones de **Set Elements.**

Show Grid Permite Activar (mostrar) o desactivar la malla auxiliar para dibujo de elementos.

Show Axes Dibuja o suprime los ejes globales de la estructura.

Se deja al lector que pruebe el efecto de las otras opciones, las características de algunas de ellas se verán posteriormente en el desarrollo paso a paso de algún ejemplo.

3.5 El menú Define

El menú Define (ver figura 3.7) permite especificar propiedades de los materiales (Materials ...), características geométricas como forma, dimensiones, material, etc. para las barras del modelo (Frame Sections..) y algunas características para los elementos placa (Shell Sections...). También permite definir características generales de las condiciones de carga estática como su titulo o identificación, el tipo de carga (de acuerdo a su origen) y si se incluirá el peso propio en la condición de carga.

En este menú se podrá seleccionar o introducir un espectro de respuesta así como funciones de excitación para análisis dinámico, también se podrán definir las combinaciones de carga (Load Combinations...) seleccionando las condiciones de carga que se incluirán en cada combinación con sus respectivos factores de carga

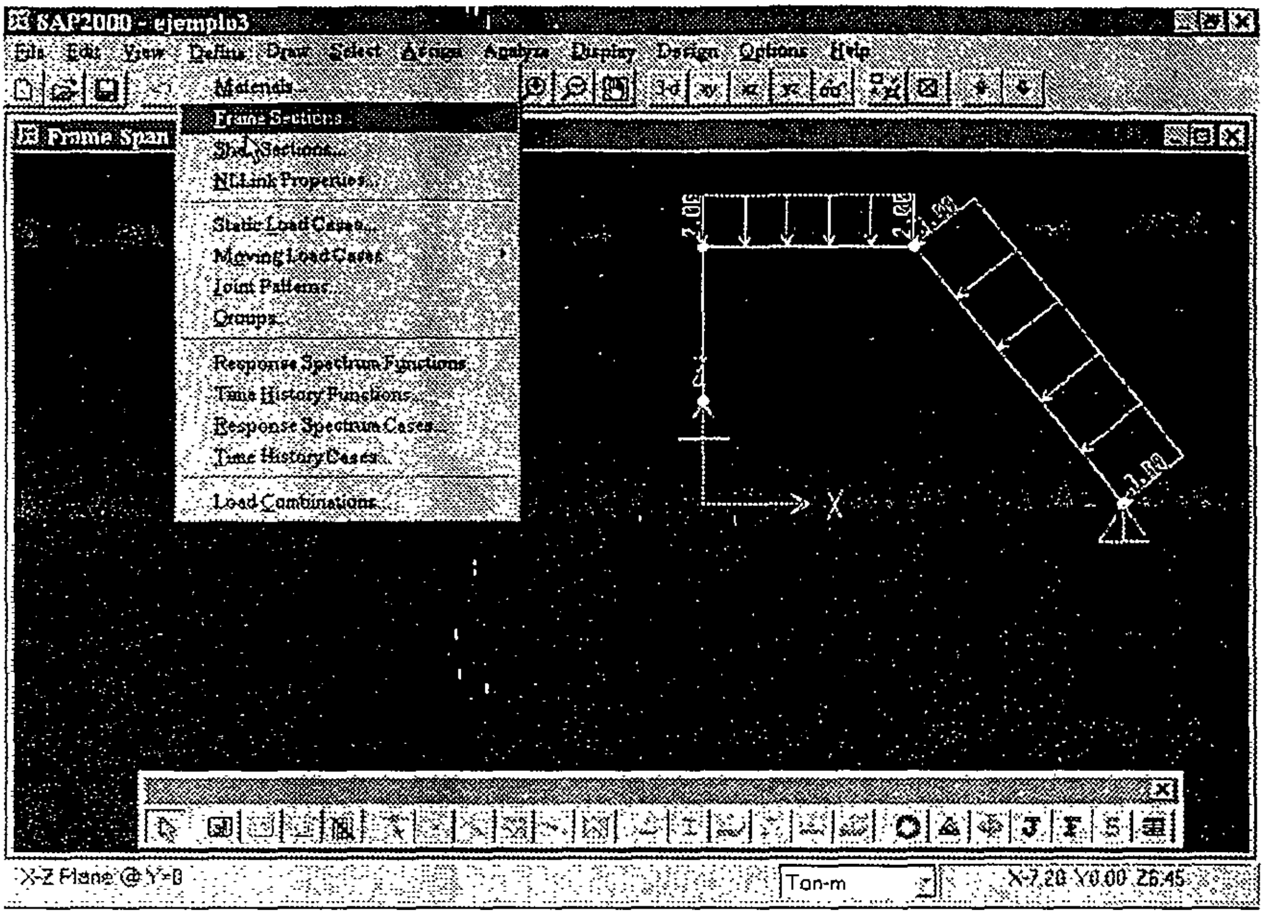


Figura 3.7. Opciones del menú <u>D</u>efine.

3.5 El menú Dr<u>a</u>w

Algunas de las opciones del menú Draw (ver figura 3.8) permiten dibujar esquemáticamente a los elementos barra, placa, etc. con los que se irá construyendo el modelo estructural por analizar, algunas opciones de uso frecuente son:

<u>E</u>dit Grid — Permite adicionar, modificar, suprimir, etc. las líneas que forman la malla auxiliar para dibujo de elementos.

Draw <u>F</u>rame Element — Permite iniciar el dibujo (con la ayuda del ratón) de elementos barra, después de seleccionar esta opción se hace clic izquierdo del ratón en el nudo donde inicia la barra (en caso de que este no halla sido creado se hace clic en sus coordenadas), luego se desplaza el puntero (sin arrastrar) hacia el nudo final de la barra haciendo clic izquierdo en el nudo con lo que queda especificada esa barra (se recomienda utilizar la malla auxiliar cambiando la separación de las lineas de la malla para que algunas de las intersecciones de esas lineas coincidan con la mayoria de los nudos de la estructura), la secuencia de dibujo de barras se puede interrumpir con un doble clic del botón derecho en cualquier parte del área de dibujo (con lo que es posible dibujar barras en otras posiciones), para terminar el dibujo de barras se hace clic en el icono de puntero de la barra flotante de iconos, posteriormente se puede dibujar más barras volviendo a seleccionar esta opción, lo anterior se puede hacer tantas veces como se requiera.

Draw Shell Element Permite iniciar el dibujo (con la ayuda del ratón) de elementos placa, funciona de manera muy similar a la opción anterior solo que en este caso se seleccionaran tres o cuatro nudos dependiendo del tipo de elemento finito que se quiera dibujar, la selección de nudos se hará en sentido horario o antihorario.

Quick Draw Frame Element y Quick Draw Shell Element permiten el dibujo de barras y placas respectivamente con un solo clic izquierdo cerca de alguna de las líneas de la malla auxiliar (para el caso de barras) y en algún punto dentro de un área delimitada por líneas de la malla auxiliar de dibujo (para el dibujo de placas), se deja al lector la práctica con estas opciones antes de abordar los ejemplos que se presentan en el capítulo correspondiente.

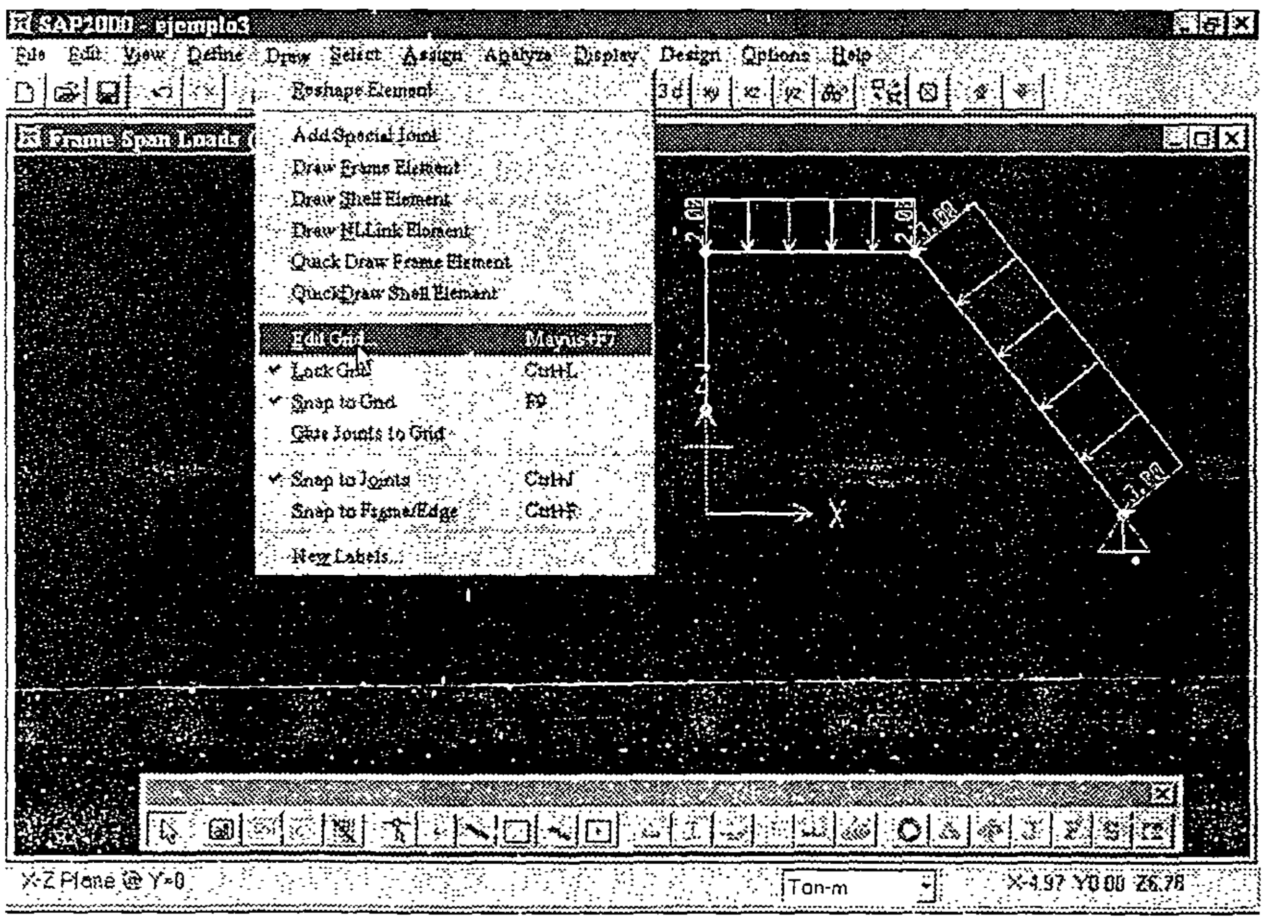


Figura 3.8. Opciones del menú Draw.

3.6 El menú Select

Algunas de las opciones del menú Select (ver figura 3.9) nos permitirán seleccionar elementos ya existentes dentro del modelo, la selección de elementos es necesaria para poder asignar (ver menú Assign) algunas características a los mismos, por ejemplo si se seleccionan barras se les podrá asignar secciones, cargas, etc. las siguientes son algunas opciones que resultan de uso frecuente:

Pointer/Window Permite seleccionar a los elementos que quedan contenidos dentro de un área rectangular que se define haciendo clic izquierdo en una de las esquinas del área y arrastrando el puntero del ratón hasta la esquina opuesta y soltando el botón del ratón en esa esquina, los elementos seleccionados cambian su aspecto de línea continua a línea interrumpida (punteada).

Intersecting Line Con esta opción se seleccionan a aquellos elementos que son intersectados por una línea que se define haciendo clic izquierdo en uno de los extremos de la misma y arrastrando el puntero del ratón hasta el otro extremo de la línea y soltándolo ahí mismo.

Las otras opciones de Select permiten seleccionar elementos que tienen alguna característica en común.

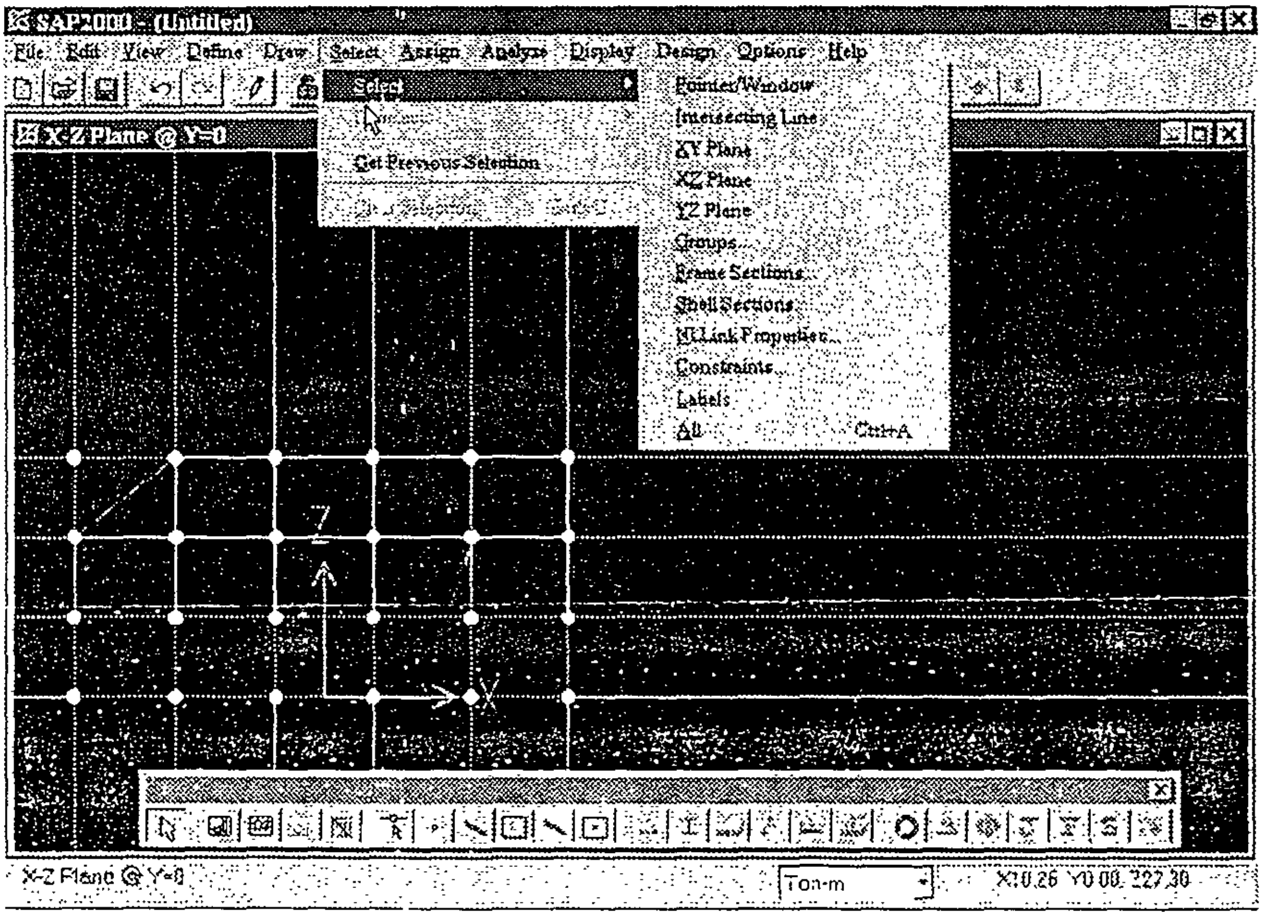


Figura 3.9. Algunas opciones del menú Select.

El menú Select dispone de las mismas opciones para excluir elementos ya seleccionados, lo anterior se realiza con la opción Unselect, otra manera de quitar elementos de la selección es haciendo clic en el icono de flecha de la barra flotante de iconos y luego hacer clic en cada uno de los elementos que han sido previamente seleccionados y que se quieren excluir, inclusive si se hace clic en un elemento no seleccionado este se selecciona y viceversa

3.7 El menú Assign

Una vez seleccionados algunos elementos (nudos, barras, etc.) podemos asignarles alguna característica propia del elemento (restricciones, fuerzas, secciones, etc.), el menú Assign (ver figura 3.10) junto con sus opciones nos permitirán realizar esa actividad, enseguida una breve descripción de algunas opciones del menú assign.

Joint — Permite asignar a los nudos seleccionados restricciones o apoyos (restraints), asignar el mismo desplazamiento (constraints), asignar resortes (springs), etc.

Joint Static Loads — Con esta opción se asignan a los nudos seleccionados fuerzas (Forces) o desplazamientos prescritos (Displacements).

Frame — Permite asignar a las barras seleccionadas propiedades (Sections), liberarlas de algún elemento mecánico (Releases), especificar sus ejes locales (Local Axes), etc.

Frame Static Loads — Con esta opción se asignan fuerzas estáticas de gravedad (Gravity), puntuales y/o uniformes (Point and Uniform), con variación lineal (Trapezoidal), efectos de temperatura (Temperature), y efectos de presfuerzo (Prestress).

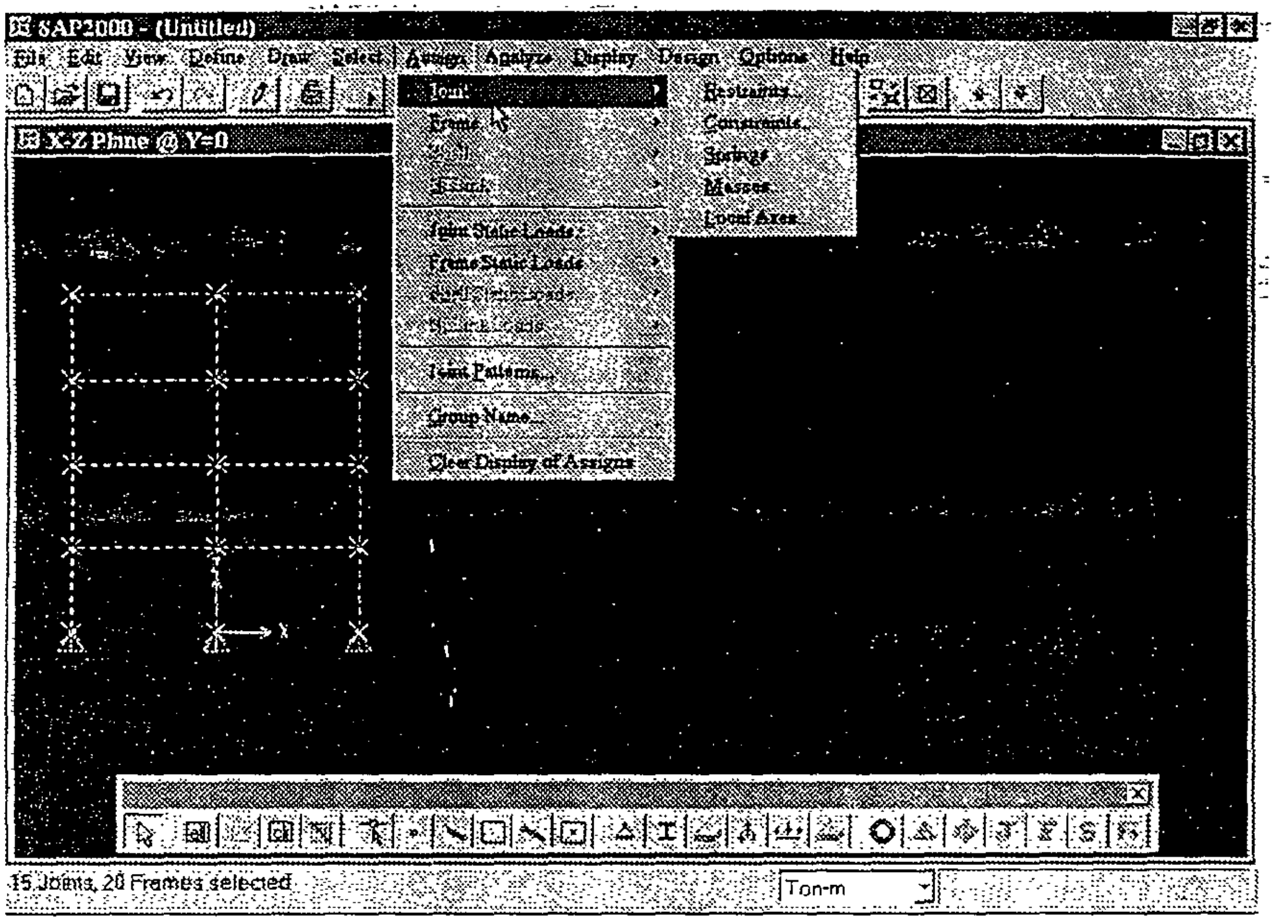


Figura 3.10. Algunas opciones del menú Assign.

3.8 El menú Analyze.

El menú **Analyze** (ver figura 3.11) permite seleccionar algunas opciones de análisis (**Set Options...**), o bien se puede solicitar que el programa **SAP 2000** realice el análisis (**Run**) con los resultados desplegados en una ventana normal o bien en una ventana minimizada (**Run Minimized**), se recomienda guardar el archivo de trabajo antes de solicitar el análisis (inclusive guardarlo en disco flexible y luego en el disco duro).

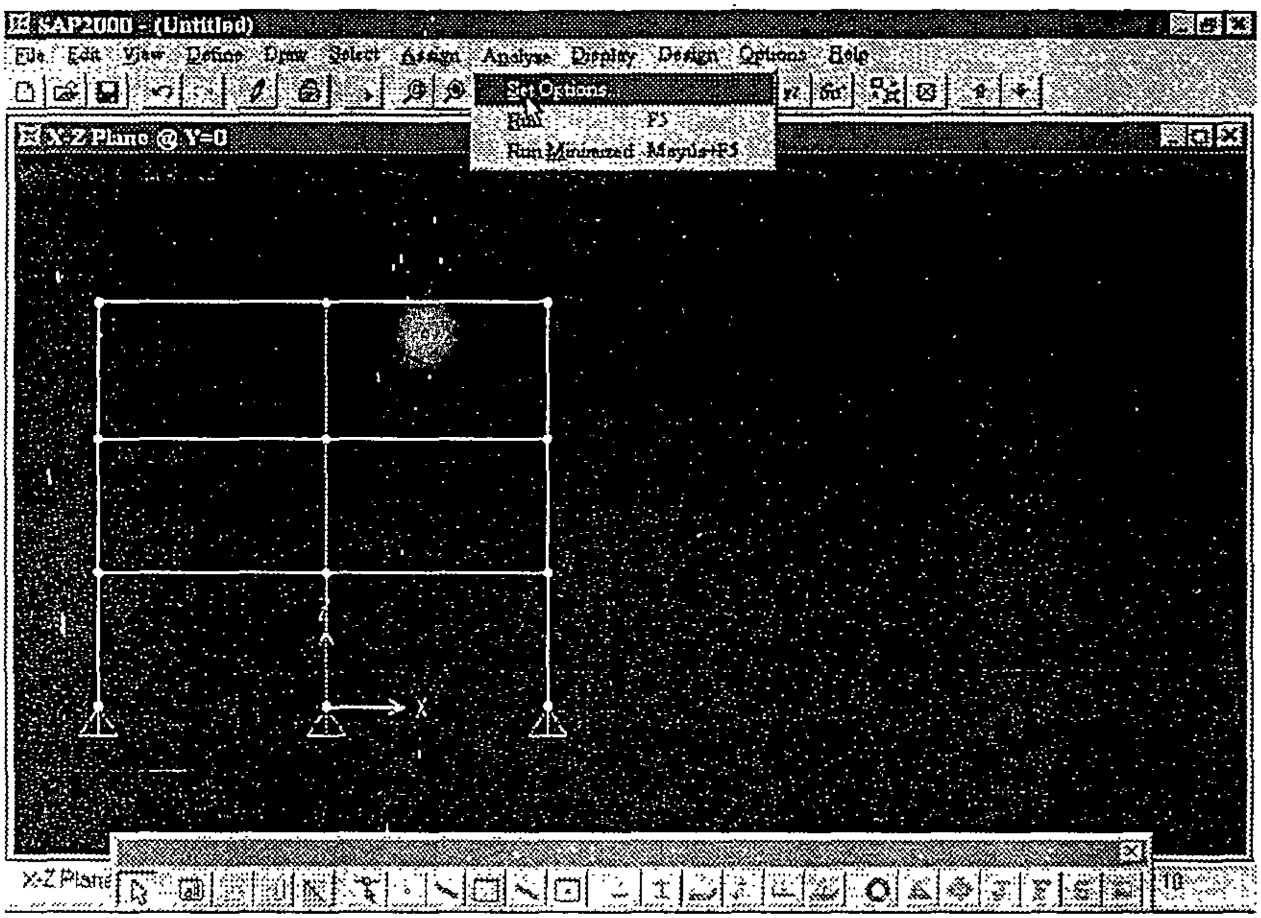


Figura 3.11 Opciones en el menú Analyze.

Las opciones de **Set Options...** (ver figura 3.12) permiten seleccionar los grados de libertad activos (**Available DOFs**) dependiendo del tipo de estructura que se analizará, será necesario identificar y seleccionar haciendo clic en los cuadros respectivos del área correspondiente (un cuadro en blanco significa que ese grado de libertad no esta activo), otra manera de seleccionar los grados de libertad es utilizando la opción de seleccionado rápido (**Fast DOFs**), lo anterior se realiza haciendo clic en alguna de las figuras que corresponda a nuestra estructura, la selección inadecuada de los grados de libertad puede generar resultados incorrectos o estructura inestable (división entre cero) durante la fase de análisis.

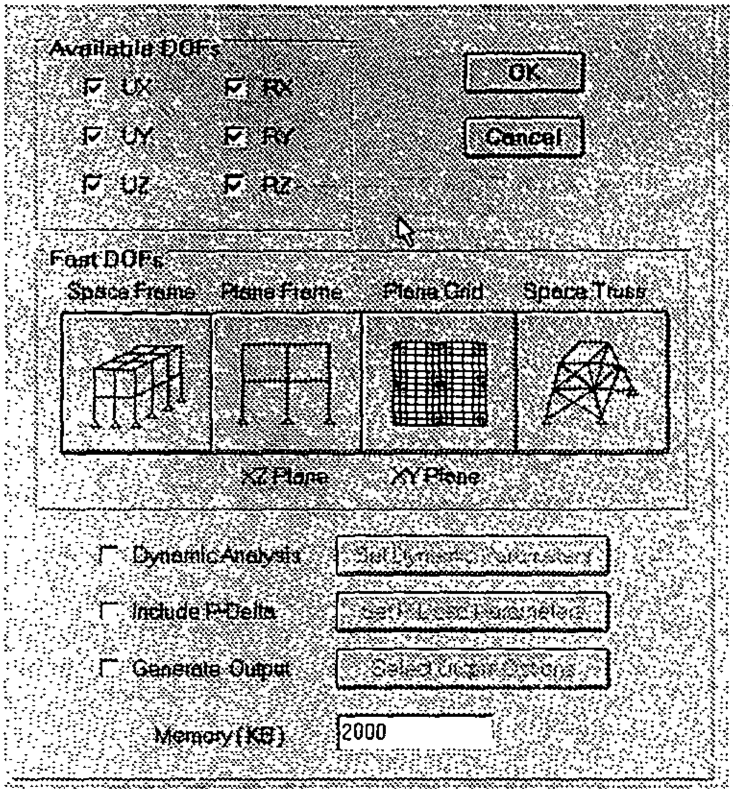


Figura 3.12. Opciones de **Set Options...** del menú **Analyze**.

Al final de la caja de selección se puede indicar que se realice un Análisis Dinámico (**Dynamic Analysis**), que se incluyan efectos P-Δ (**Include P-Delta**) y que se generen archivos de salida (**Generate Output**), para estas ultimas opciones es conveniente indicar algunos parámetros y seleccionar algunas opciones especificas.

Cuando se selecciona la opción de Análisis (**Run**), y algunos resultados del proceso se van desplegando en la pantalla (ventana) quedando al final algo similar a lo que se muestra en la figura 3.13.

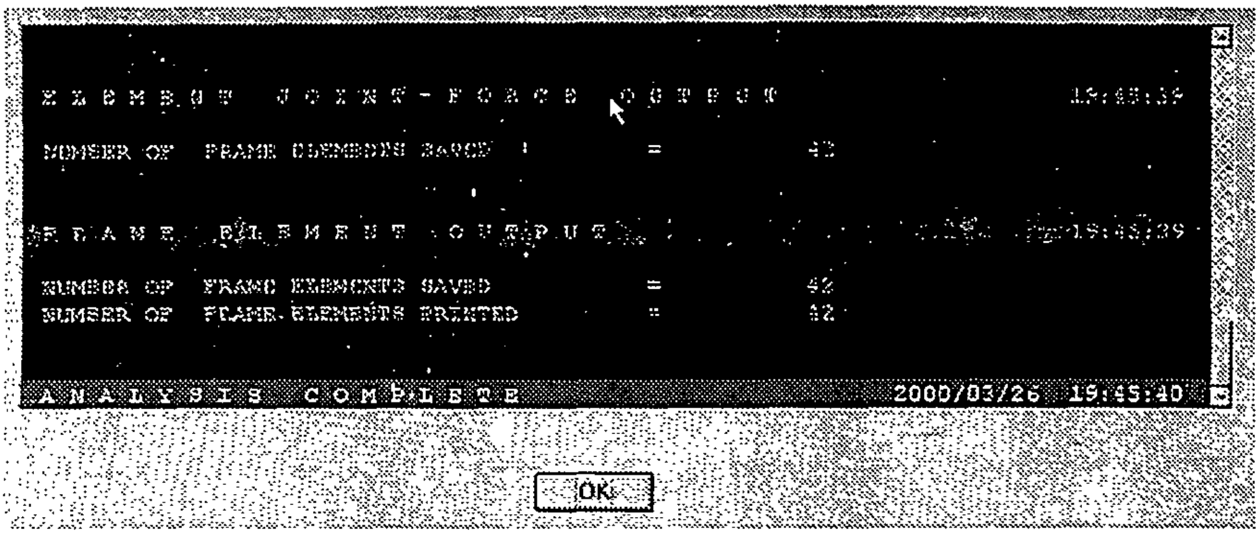


Figura 3.13. Ventana al finalizar el análisis.

Al hacer clic en el botón OK de la ventana que se muestra al final del análisis, se despliega en el área de dibujo la configuración deformada de la estructura para determinada condición de carga, en esta parte del programa se podrán seleccionar los resultados del análisis por ejemplo desplazamientos de los nudos, reacciones, elementos mecánicos, diagramas de elementos mecánicos, configuraciones deformadas, etc.

3.9 El menú Display

Este menú permite solicitarle al programa que muestre la geometría no deformada del modelo (**Show Undeformed Shape**), las cargas en los nudos (**Show Loads**), en las barras, en los elementos placa o no mostrarlas.

Mediante la opción **Show Input Tables** (ver figura 3.14) se solicita al programa que muestre en una ventana conteniendo una lista con los datos numéricos de la geometría en lo que respecta a nudos (coordenadas, restricciones, etc.), barras (incidencias, tipo de sección, etc.) y cargas (en los nudos, en las barras y en las placas), produciendo una salida parecida a la de la figura 3.15, la tabla mostrada puede imprimirse o grabarse en un archivo.

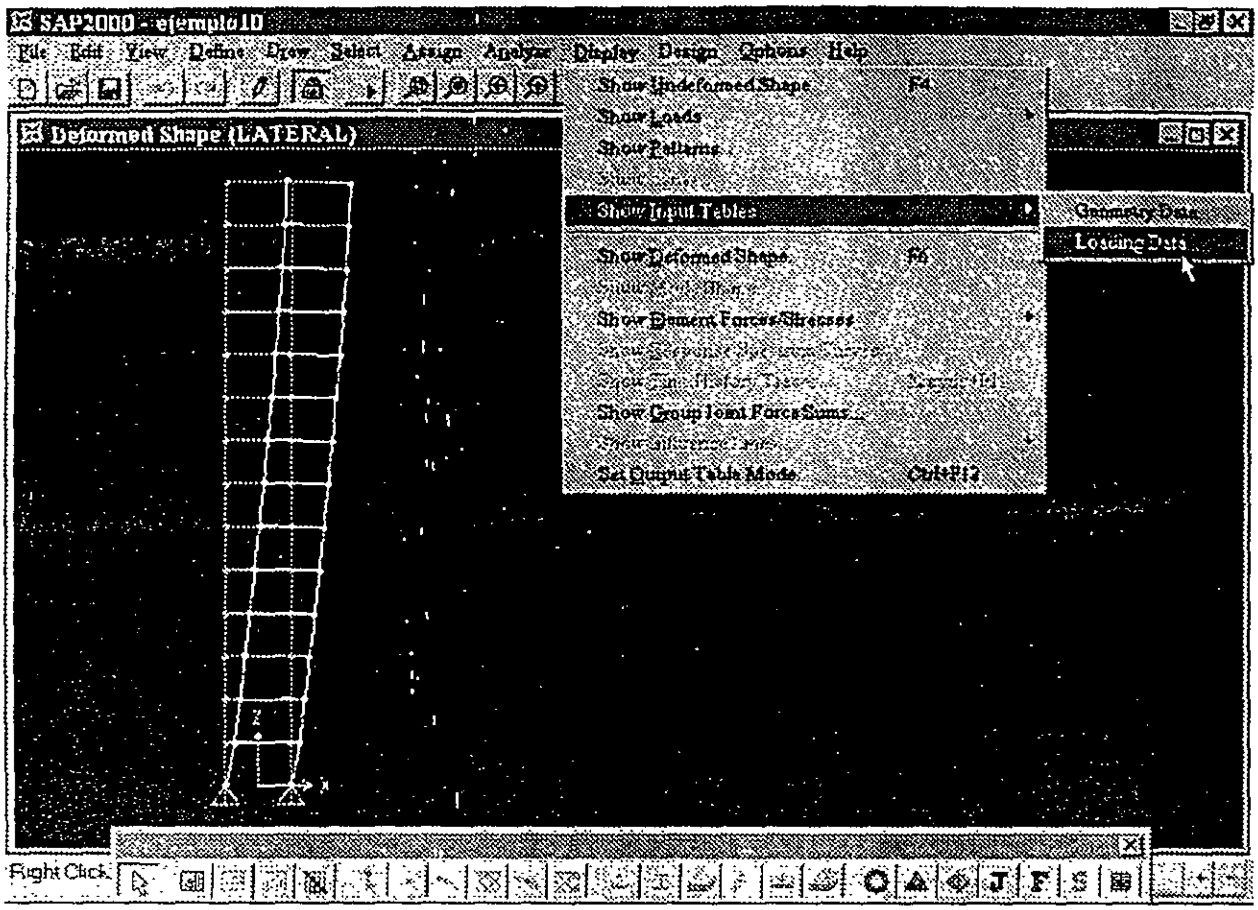


Figura 3.14. Opciones del menú **Display**.

JOINT	GLOBAL-X	GLOBAL-Y	GLOBAL-Z	RESTRAINTS	ANGLE-A
1	-3.00000	0.00000	0.00000	1 1 1 1 0 1	0.000
2	-3.00000	0.00000	4.00000	0 0 0 0 0 0	0.000
3	-3.00000	0.00000	8.00000	0 0 0 0 0 0	0.000
4	-3.00000	0.00000	12.00000	0 0 0 0 0 0	0.000
5	-3.00000	0.00000	16.00000	0 0 0 0 0 0	0.000
6	-3.00000	0.00000	20.00000	0 0 0 0 0 0	0.000
7	-3.00000	0.00000	24.00000	0 0 0 0 0 0	0.000
8	-3.00000	0.00000	28.00000	0 0 0 0 0 0	0.000
9	-3.00000	0.00000	32.00000	0 0 0 0 0 0	0.000
10	-3.00000	0.00000	36.00000	0 0 0 0 0 0	0.000

Figura 3.15. Salida típica a partir de **Show Input tables** del menú **Display**.

Mediante la opción **Show Deformed Shape** y después de seleccionar la condición de carga, SAP 2000 muestra la configuración deformada correspondiente (ver figura 3.16).

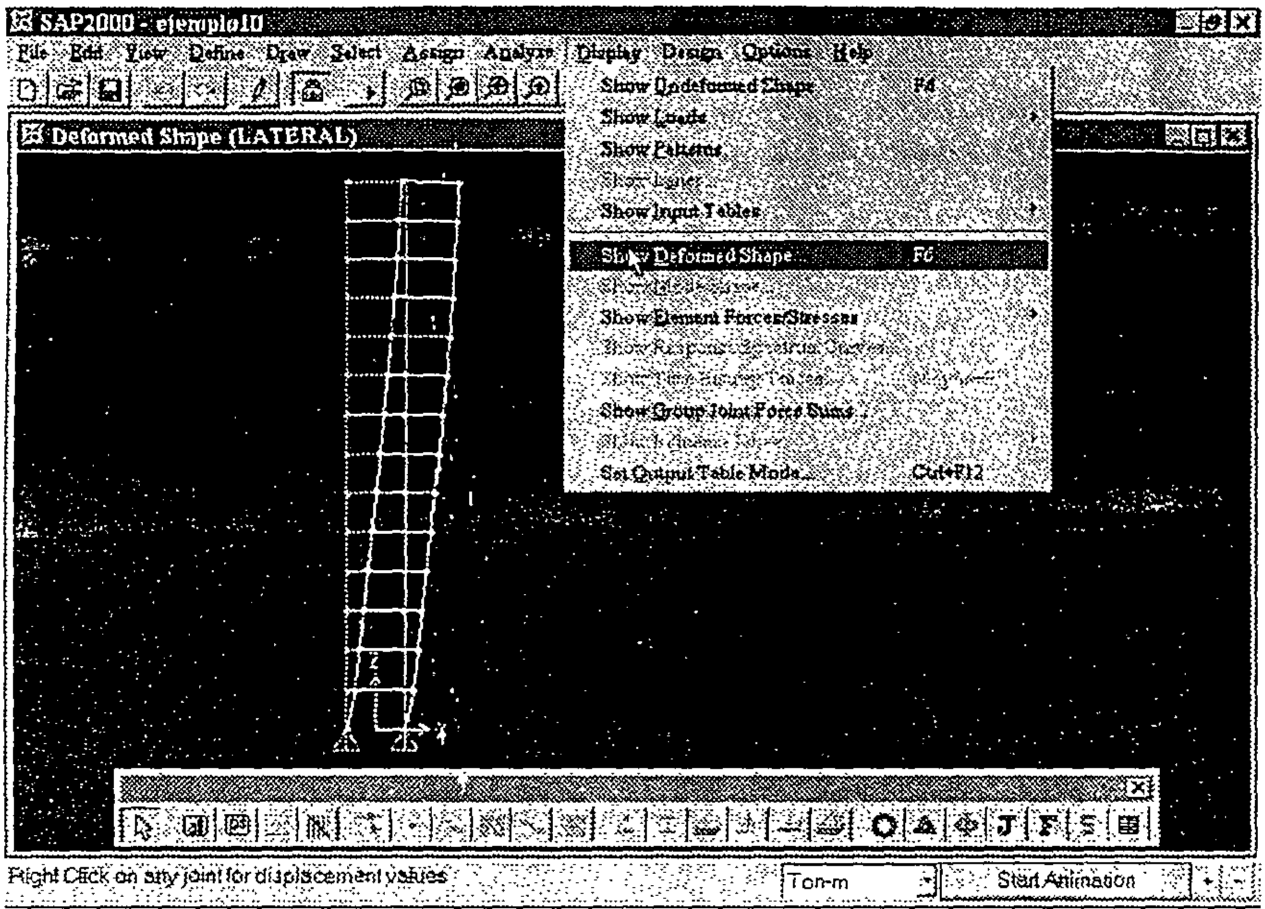


Figura 3.16. Salida típica a partir de **Show Deformed Shape** del menú **Display**

La opción **Show Element Forces/Stresses** y dependiendo de la selección que se haga **SAP 2000** puede mostrar elementos mecánicos, esfuerzos, reacciones, etc. produciendo una salida similar a la que se muestra en la figura 3 17.

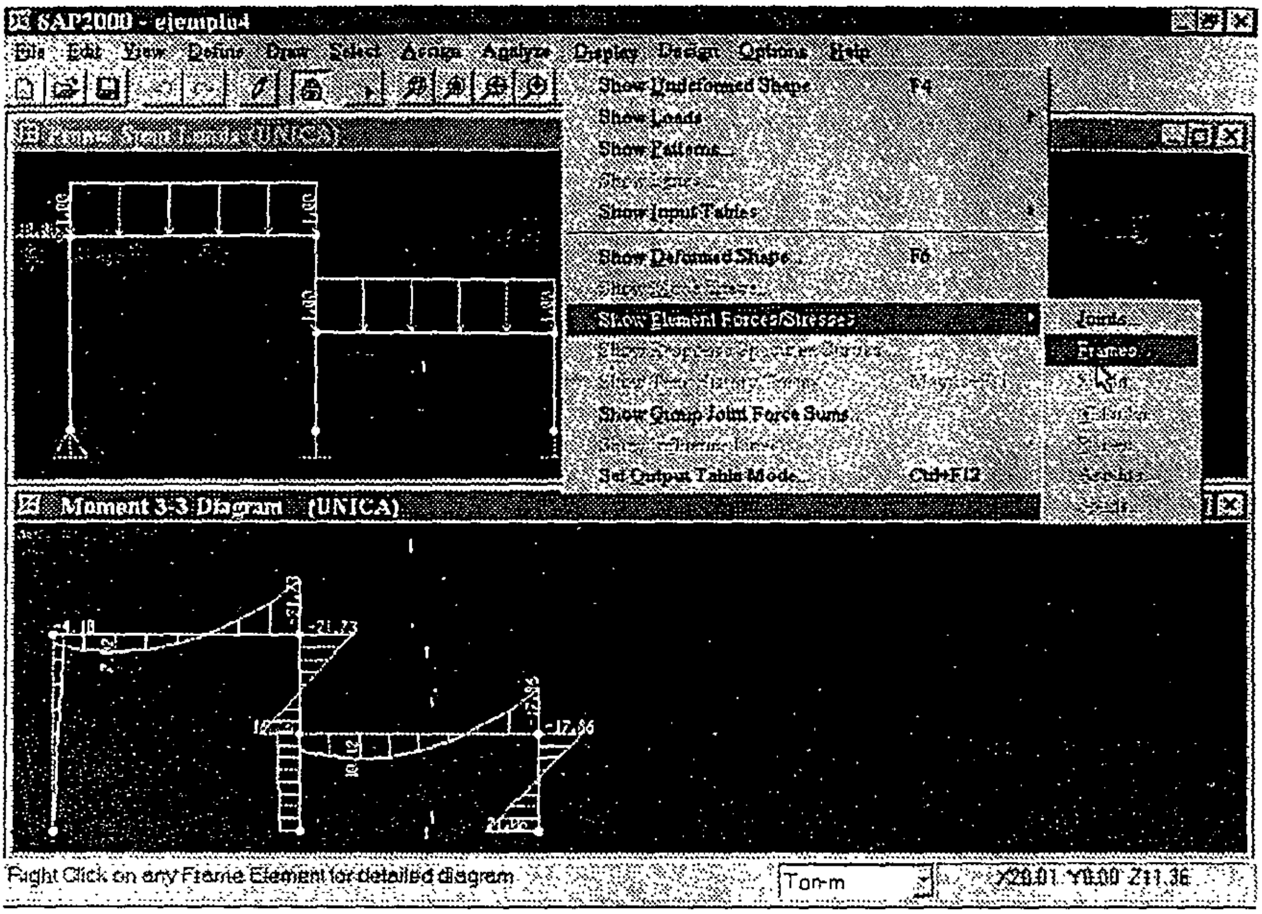


Figura 3.17. Salida obtenida con **Show Element Forces/Stresses** del menú **Display**

3.10 El menú Design

EL menú **Design** (ver figura 3.18) permite seleccionar algunas opciones de diseño, realizar el diseño (verificación) de elementos con la posibilidad de optimizar secciones, con la característica de producir salidas similares a las mostradas en las figuras 3.19 y 3.20 de entre otras.

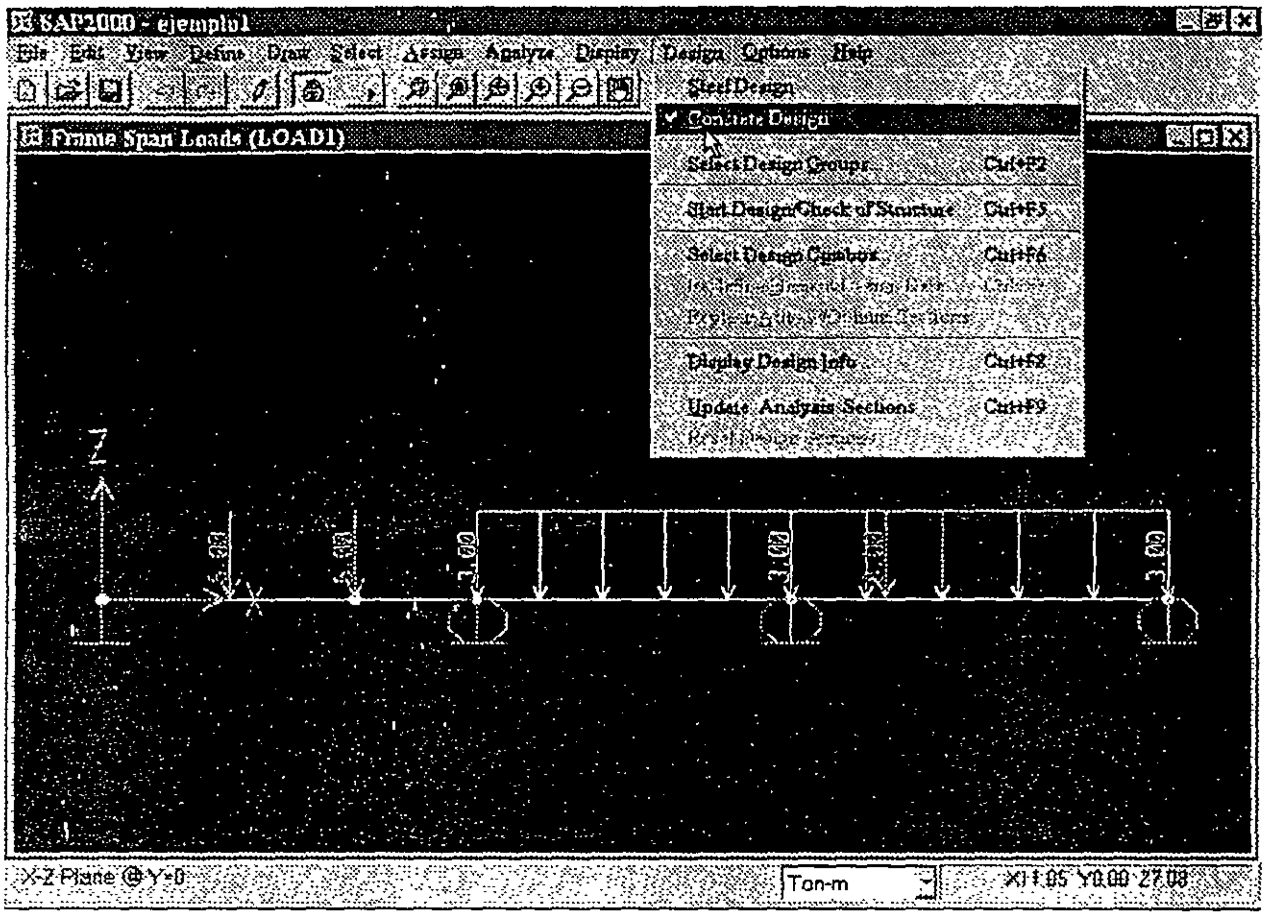


Figura 3.18. Opciones del menú **Design**.

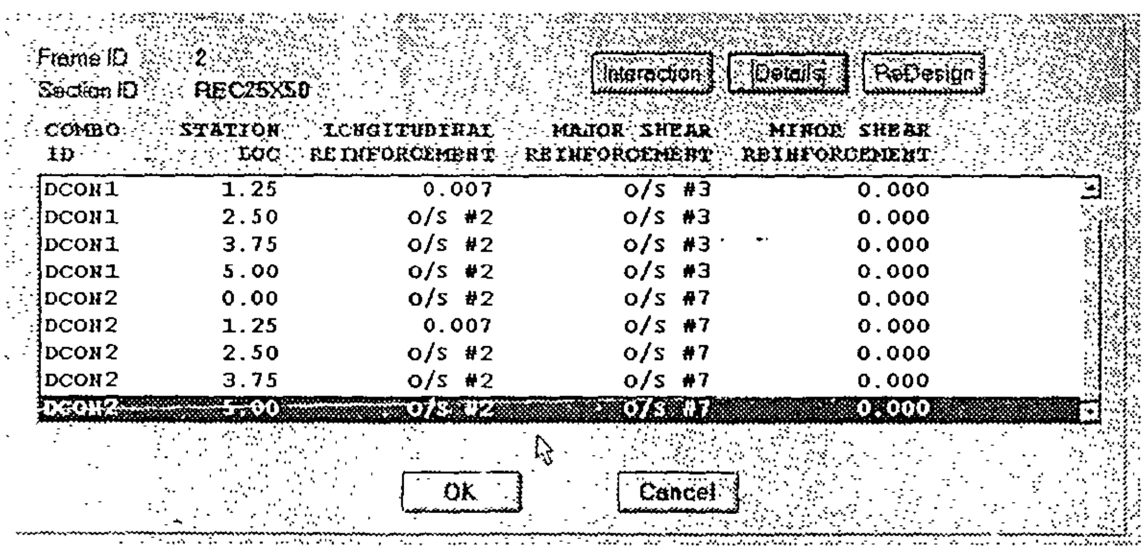


Figura 3.19. Algunos resultados del menú **Design**

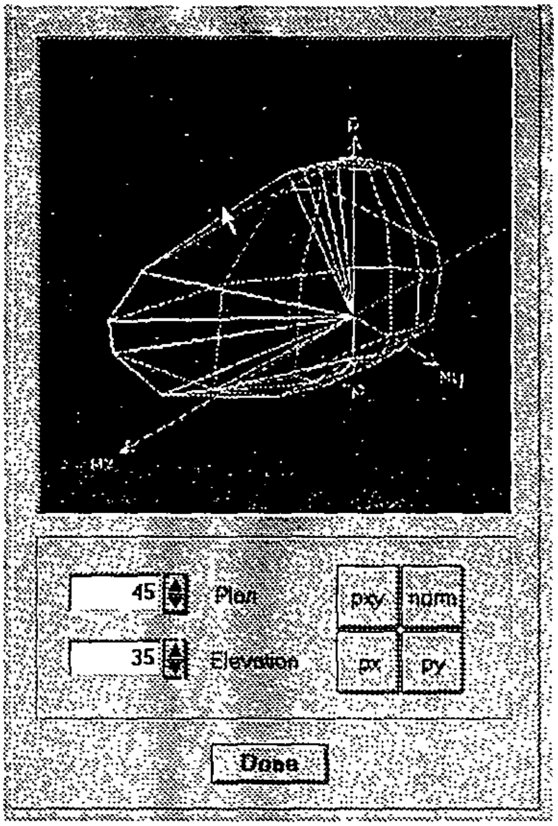
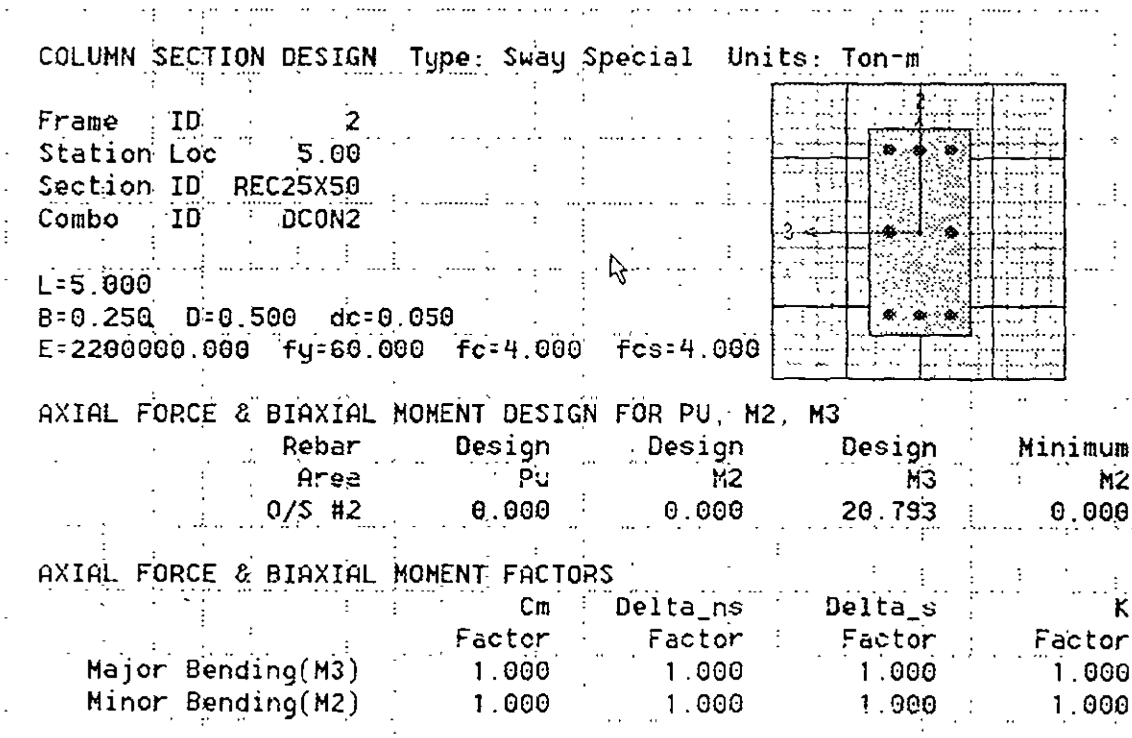


Figura 3.20. Algunos resultados del menú **Design**.



```
COLUMN SECTION DESIGN  Type: Sway Special  Units: Ton-m

Frame  ID          2
Station Loc     5.00
Section ID  REC25X50
Combo  ID     DCON2

L=5.000
B=0.250  D=0.500  dc=0.050
E=2200000.000  fy=60.000  fc=4.000  fcs=4.000

AXIAL FORCE & BIAXIAL MOMENT DESIGN FOR PU, M2, M3
                 Rebar      Design      Design      Design     Minimum
                  Area          Pu          M2          M3          M2
                 O/S #2      0.000       0.000      20.793       0.000

AXIAL FORCE & BIAXIAL MOMENT FACTORS
                             Cm     Delta_ns     Delta_s           K
                         Factor       Factor      Factor      Factor
   Major Bending(M3)      1.000        1.000       1.000       1.000
   Minor Bending(M2)      1.000        1.000       1.000       1.000
```

Figura 3.21. Algunos resultados del menú **Design**.

3.11 Los menús Options y Help

El menú **Options** (ver figura 3.22) permite por así decirlo controlar el tipo y características de la información que será mostrada en las diferentes áreas de presentación (colores, número de ventanas, etc.).

En este punto podemos mencionar que una vez que se realiza el análisis **SAP 2000** "bloquea" al modelo no permitiendo realizarle ninguna modificación por lo que solo es posible manejar los resultados (ver valores numéricos, gráficas, imprimirlos, etc.), para desbloquear al modelo y poder hacerle cambios se selecciona la opción **Lock Model** con esto ahora los resultados ya no están disponibles para poder tener acceso a ellos una vez realizados los cambios será necesario solicitar nuevamente la realización del análisis.

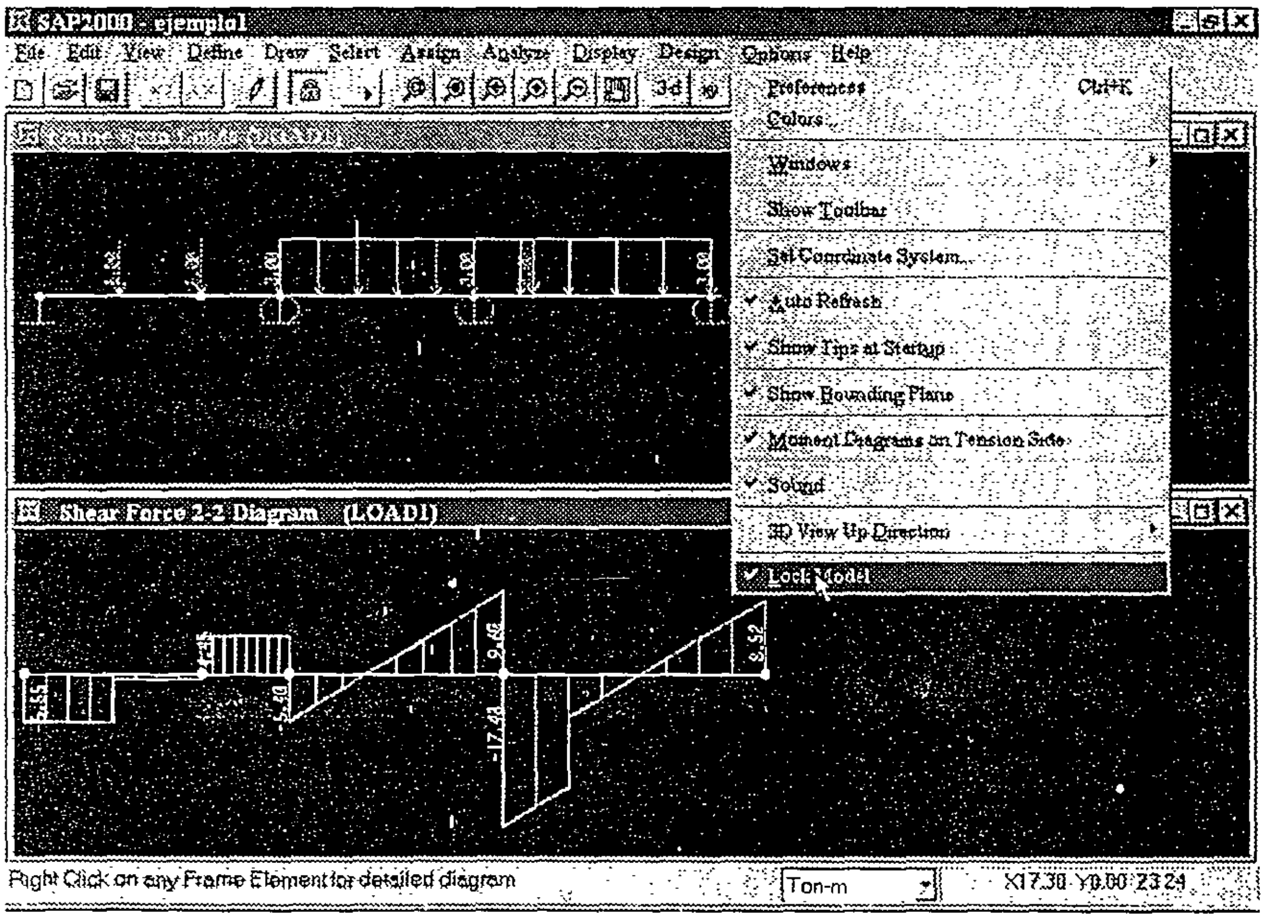


Figura 3.21. Opciones en el menú **Options** y desbloqueo del modelo

Se deja al lector que pruebe el efecto de las otras opciones del menú **Options** así como las del menú **Help**, las características de algunas de ellas se verán posteriormente en el desarrollo de algunos ejemplos.

GENERACION DE LA ESTRUCTURA

CAPÍTULO 4

4.1 INTRODUCCIÓN

En **SAP 2000** la generación de la estructura se entiende como la ubicación con respecto a un sistema de coordenadas (global) de los elementos barra, placa y sólido, la asignación de propiedades geométricas y elásticas a los elementos ya localizados, la introducción de apoyos, la definición y asignación de fuerzas a los nudos, barras y placas, la selección del tipo de análisis y resultados, por último, el dimensionamiento o revisión de elementos.

La forma de iniciar el programa **SAP 2000** ha sido descrita con anterioridad (ver inciso 3.2 del capitulo anterior), enseguida se recomienda elegir las unidades en que se introducirán los datos haciendo clic en la pestaña que se encuentra a la derecha del cuadro de unidades y seleccionándolas de la caja que muestra el programa (ver figura 4.1).

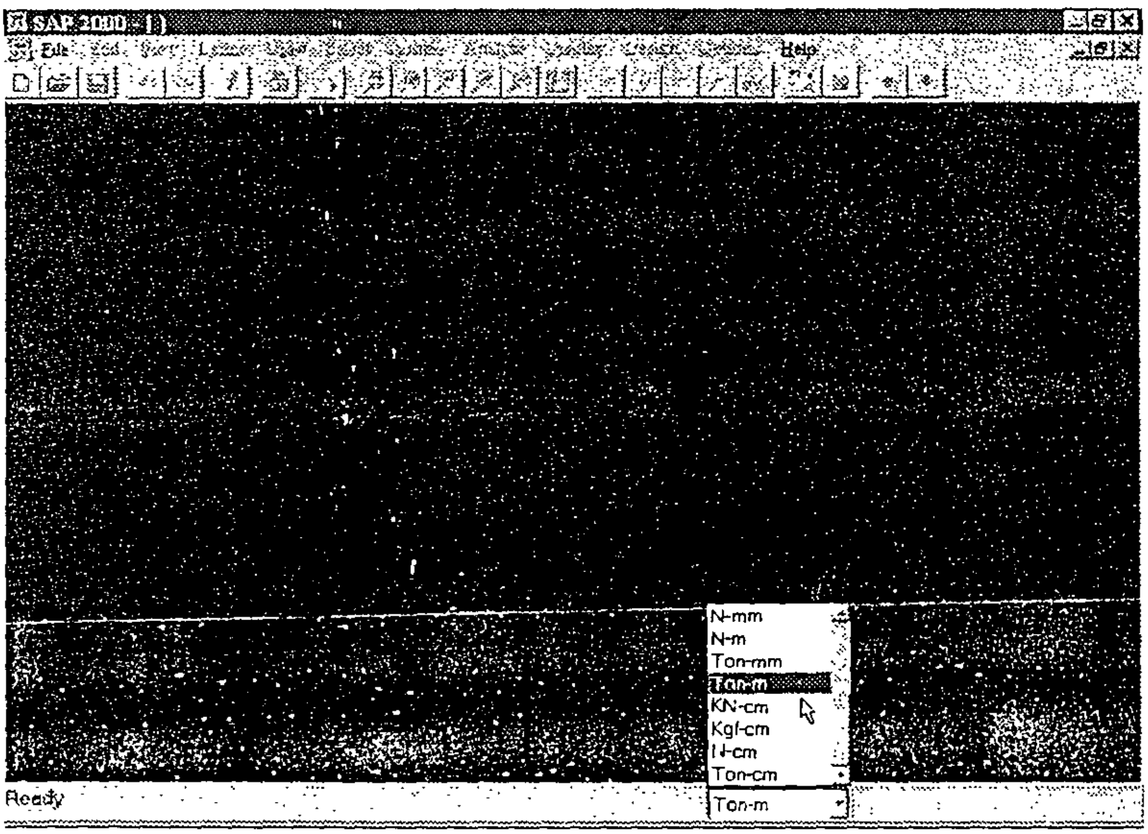


Figura 4.1 Selección de unidades.

SAP 2000 ofrece varias alternativas para introducir la topología de la estructura, aquí empezaremos por una de las más comunes que es introduciendo elemento por elemento, para ello se recomienda auxiliarnos de la malla (**grid**) que el programa nos proporciona por lo que se tendrá que ajustar la separación de las líneas que forman esa malla, seleccionemos **New Model** del menú **File**, enseguida el programa mostrará un cuadro en donde se especificarán las características de la malla como el número de espacios en cada dirección así como su separación los cuales se pueden modificar introduciendo valores particulares en los cuadros en fondo blanco haciendo clic en el que se quiera modificar (ver figura 4.2)

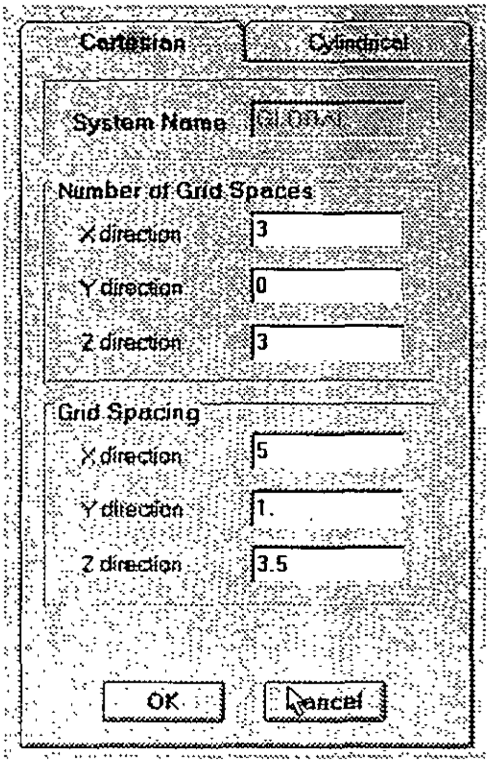


Figura 4.2 Ventana para definir las características de la malla auxiliar.

Una vez que se ha hecho clic en el botón OK el programa muestra la malla resultante en el área de dibujo (con fondo negro) dividiéndola en 2 cuadros mostrando en ellos una vista diferente de la malla (3 D y en el plano X-Y en Z=10.5), también puede observarse los ejes coordenados globales (ver figura 4.3).

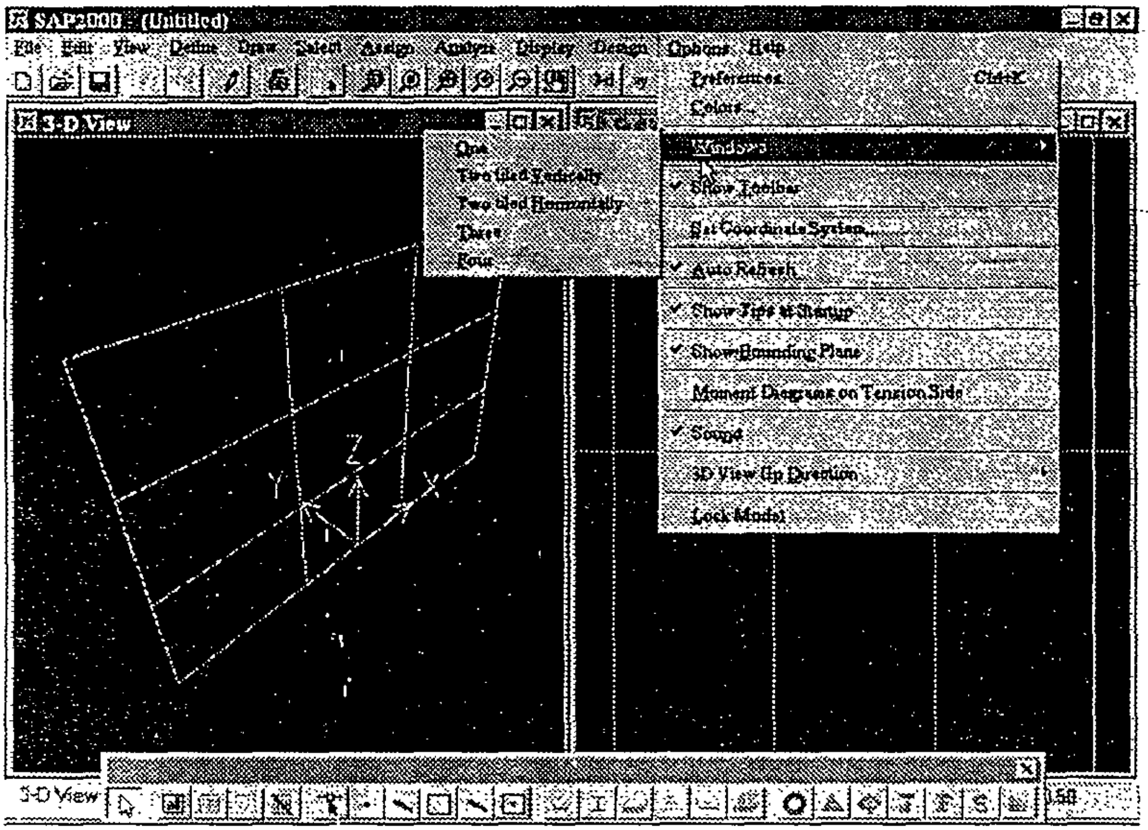


Figura 4.3 Imagen típica después de definir las características de la malla auxiliar.

Cada una de esas ventanas tiene en su extremo superior derecho los botones de minimizar, ventana completa y cerrar, el número y tipo de ventanas a mostrarse en la pantalla puede seleccionarse a través de la opción Windows del menú **Options**.

De las ventanas que se muestran, la ventana activa o en la que se muestran los resultados de los comandos que se elijan es aquella cuya barra de título está en color (generalmente diferente del gris), se activa una ventana haciendo clic en su interior.

La malla así creada tiene separación constante entre las líneas de una misma dirección, existen varias maneras de cambiar la separación entre cada línea de la malla, una de ellas es, después de seleccionar una vista en planta hacer dos clics seguidos en una de las líneas de la malla (con el botón izquierdo del ratón), enseguida se mostrará una ventana conteniendo información acerca de la posición de esas líneas con la opción de seleccionar la dirección de las líneas de la malla así como adicionar, mover y borrar líneas.

Haciendo clic en el cuadro en blanco e introduciendo el valor de la nueva posición de la línea y después de hacer clic en la opción **Add grid line** se ha introducido una nueva línea a la malla. Para modificar el valor de una línea se selecciona de la caja en gris haciendo clic izquierdo en la línea a modificar con lo cual se muestra en la caja en blanco y haciendo clic en esa caja se puede cambiar su valor, para que el cambio resulte efectivo después de modificar el valor de la caja en blanco se necesita hacer clic en el botón **Move**, las demás opciones complementan la modificación de la malla (ver figura 4.4). Desde luego que para que todos los cambios produzcan efecto es necesario hacer clic en el botón OK.

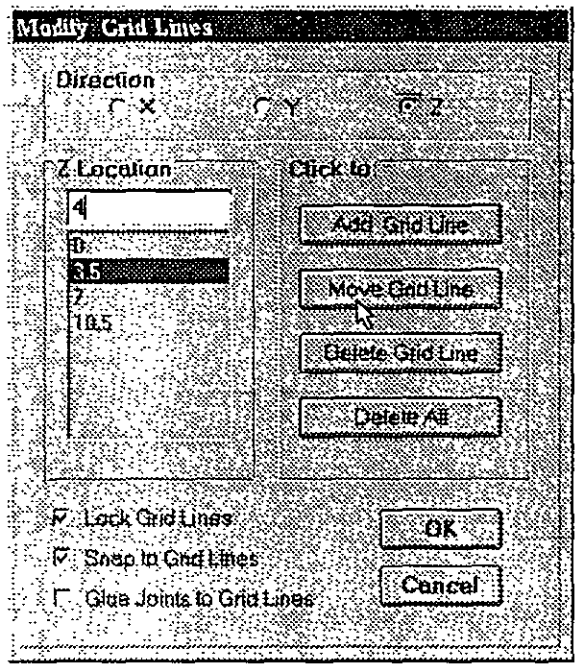


Figura 4.4 Modificación de la separación individual entre las líneas de la malla.

Otra manera para que se muestre el cuadro de la figura 4.4 es seleccionando **Edit Grid** del menú **Draw**, otro comando que resulta útil es la opción **Show Grid** del menú **View** con el cual se suprime o activa la aparición de la malla en área de dibujo.

Los datos de la estructura que se vayan introduciendo son almacenados en memoria volátil (RAM) por tal motivo se recomienda que con cierta frecuencia se graben en el disco duro (o en disco flexible), para ello se puede utilizar la opción **Save** o **Save As** del menú **File**, el programa asignará al nombre del archivo proporcionado por el usuario la extensión .SDB.

Ahora podemos introducir todos los elementos que componen a la estructura, a continuación se presenta una breve descripción lógica de las opciones de uso común así como de los comandos que nos permitirán la generación de la estructura en el orden mencionado al inicio de este capítulo, varios de los comandos fueron descritos en el capítulo anterior.

4.2 DESCRIPCIÓN GENERAL

La parte del proceso de modelación que consume más recursos (tiempo y esfuerzo) es la que concierne a la introducción de elementos (barra, placa, etc.), es por ello que el uso eficiente de los comandos del menú Draw y en combinación con algunos otros nos permitirá la generación de la topología (forma) de la estructura lo más pronto posible, como recomendaciones generales, se pueden mencionar las que se indican en los párrafos siguientes.

Procurar iniciar la geometría de la estructura a partir de alguna de las predefinidas que trae la librería del programa (vigas, marcos, etc.), enseguida realizar los cambios necesarios para ajustar esa geometría a la del modelo por analizar (adicionando o borrando algunos elementos, cambiándolos de posición, copiándolos, etc.)

Para la definición de elementos (barra, placa, etc.) auxiliarse de la malla (**grid**) cambiando la separación de las líneas de la malla para que sus intersecciones definan la mayor cantidad de coordenadas de los nudos de nuestro modelo procurando que con la nueva separación de las líneas de la malla los elementos resultantes tengan las características (dimensiones e inclinación) deseadas con lo que el usos de las opciones de dibujo rápido de elementos (con un solo clic, en lugar de dos clics) traerá algún ahorro y facilidad de creación o modificación del modelo.

Las características a ser mostradas en la pantalla (numeración, ejes locales, etc.) de los elementos que se van adicionando al modelo (nudos, barras, placas, etc.) pueden ser controladas mediante la opción **Set Elements** del menú **View** (ver figura 4.5). La información mostrada puede ser de utilidad, también es conveniente recordar que las características de algún elemento (nudo, barra, placa, etc.) pueden desplegarse seleccionándolo (clic izquierdo) y luego haciendo clic derecho, algunos de los elementos en la caja mostrada pueden ser modificados (ver figura 4.6).

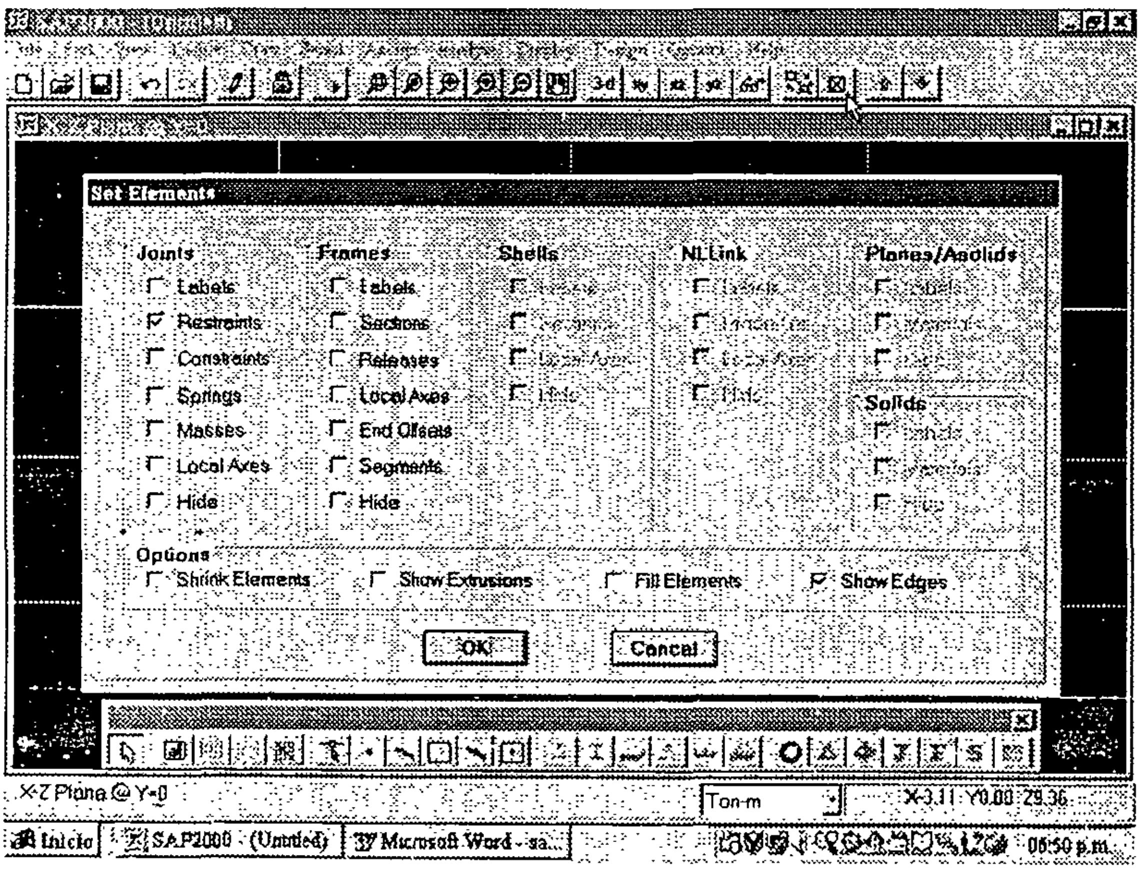


Figura 4.5 Selección de información a desplegarse en el área presentación del modelo.

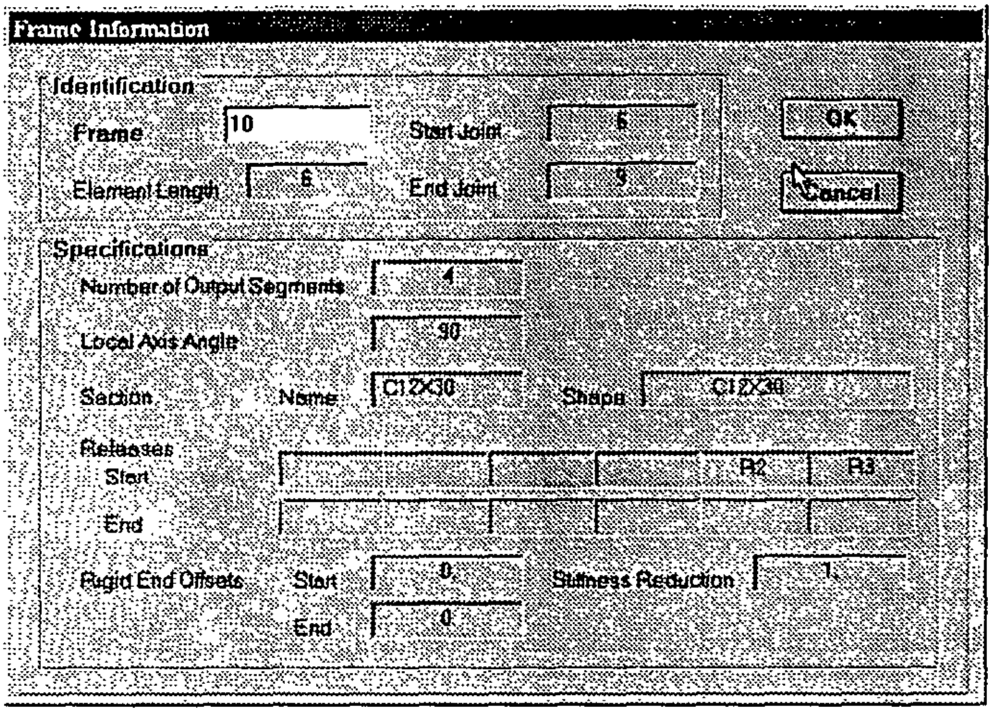


Figura 4.6 Características del elemento barra seleccionado (la información en el cuadro en blanco puede ser modificada directamente).

4.3 GENERACIÓN DE LA GEOMETRÍA

Una de las maneras de crear el archivo de datos o de modificar su contenido, es a través del editor gráfico cuyas opciones están contenidas principalmente en el menú **Draw** (ver figura 4.7), algunas de ellas se describen a continuación.

Draw Frame Element permite adicionar un nuevo elemento barra, para ello se hace clic primero en el punto extremo de la barra y luego en el opuesto.

Draw Shell Element permite adicionar un nuevo elemento placa, haciendo clic en los puntos extremos (vértices) sucesivos del elemento empezando por cualquiera de ellos se define la geometría de este elemento.

Quick Draw Frame element permite adicionar un elemento barra haciendo un solo clic en una línea de la malla auxiliar de dibujo que este delimitada por otras dos perpendiculares a la primera, esas líneas definen los límites del elemento, si se hace clic en cualquier punto de la zona delimitada por cuatro líneas de la malla o por cuatro nudos se adicionan dos elementos barra diagonales.

Quick Draw Shell element adiciona un nuevo elemento **shell** haciendo un solo clic en cualquier punto de la zona delimitada por cuatro líneas de la malla.

Comúnmente al seleccionar alguna de las opciones anteriores la forma del cursor cambia a una flecha vertical vacía hacia arriba, para cancelar o terminar la opción se hace clic en el primer icono de la barra flotante con lo que el cursor cambia a flecha inclinada llena.

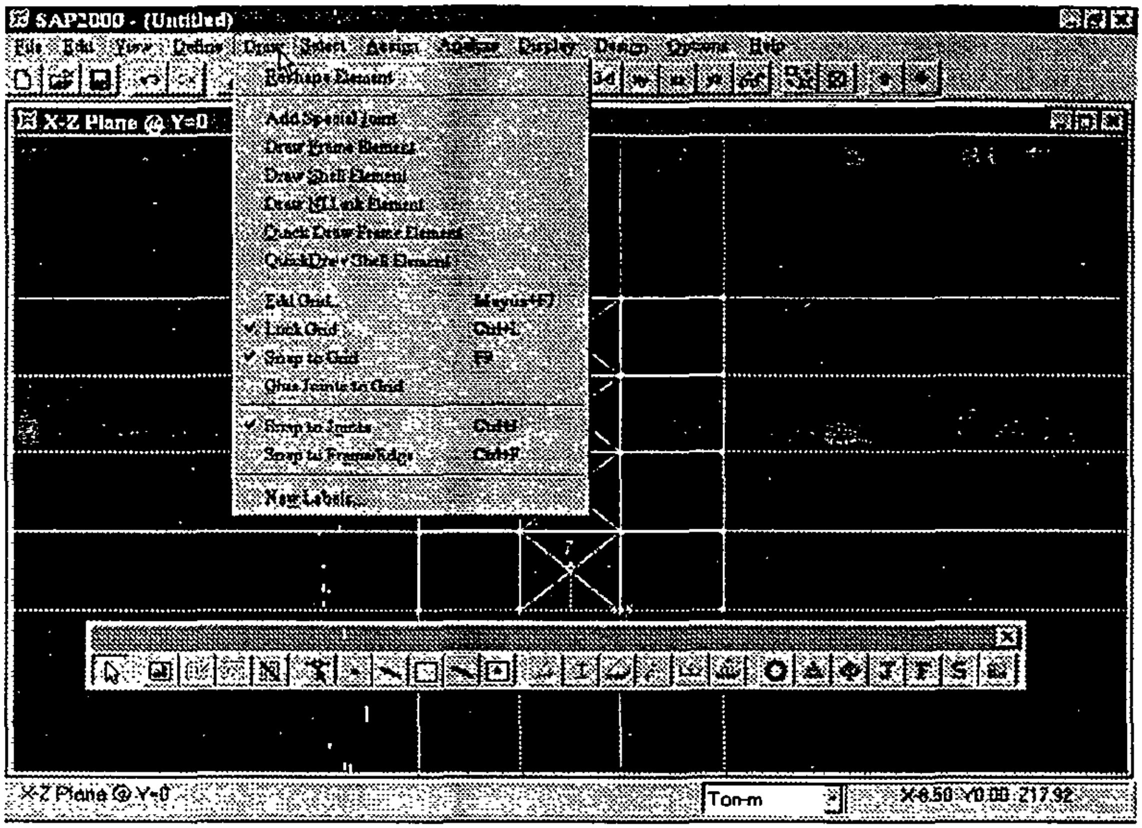


Figura 4.7 Opciones del menú **Draw**.

Con objeto de completar el modelo o realizar los cambios que se deseen, una vez que se han ubicado elementos, cuando sea posible se recomienda generar algunos otros realizando copias, giros, etc. de uno o varios de los que ya se tienen definidos.

Para tal efecto varias opciones se encuentran en el menú **Edit** (ver figura 4.8), pero para poder utilizar algunas de ellas es necesario seleccionar elementos (por ejemplo los que se van a copiar), para ello existen disponibles varias formas de seleccionar elementos, la más sencilla es hacer clic en el elemento a seleccionar (nudo o barra), el elemento seleccionado se muestra con línea discontinua, se puede anular la selección haciendo clic en un elemento seleccionado, también se pueden seleccionar elementos que queden totalmente contenidos en una ventana rectangular creada haciendo clic en uno de los vértices de la ventana y arrastrando el ratón hasta el vértice opuesto de la misma y soltando ahí, otras opciones de selección se encuentran en la opción **Select** del menú con el mismo nombre, desde luego que las acciones anteriores se pueden aplicar en repetidas ocasiones e inclusive combinar varias maneras de seleccionar y excluir (**Unselect**) elementos para lograr un resultado deseado.

Hecha la selección de algunos elementos (también puede ser uno o todos) se pueden llevar a cabo ciertas acciones con ellos dando como resultado posibles cambios a esos elementos y en general al modelo o estructura por analizar, cuando el efecto final no es el esperado se recomienda cancelar la acción, para ello se hace clic en el icono (**Un Do**) que está casi por debajo del menú **View**.

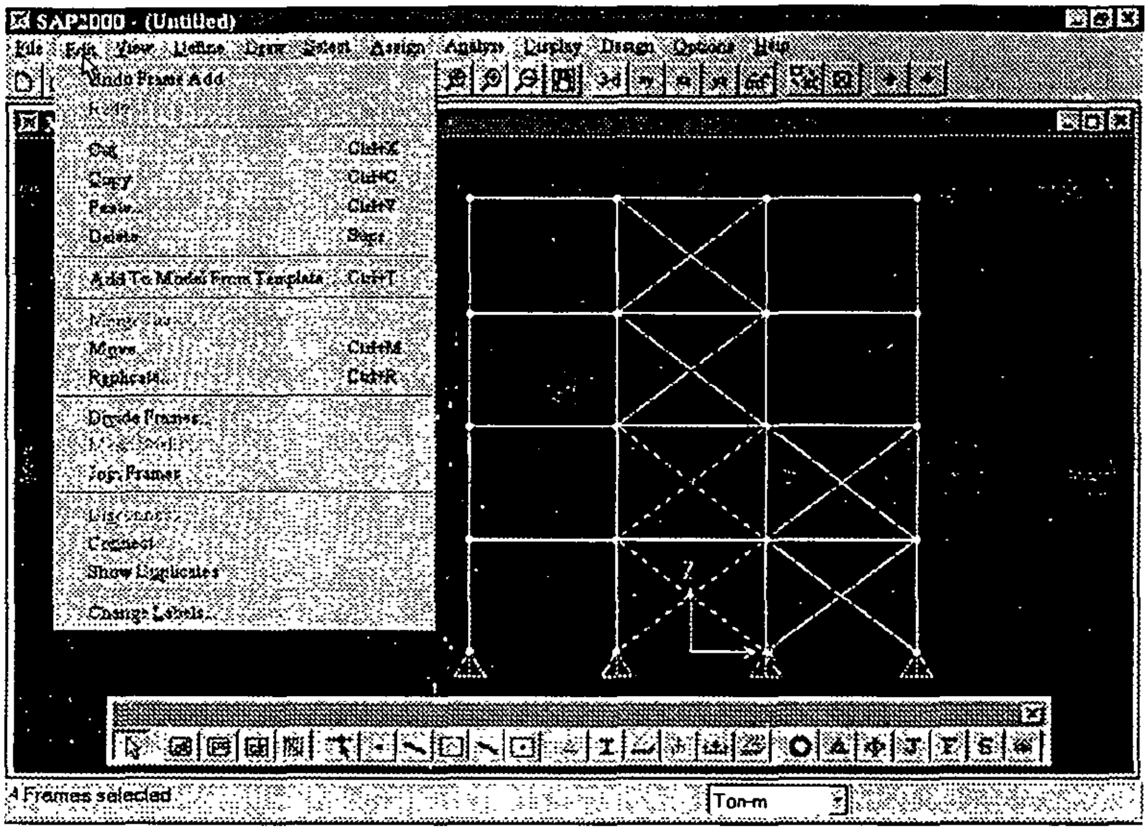


Figura 4.8 Opciones del menú **Edit**.

Cut y **Delete** Elimina los elementos seleccionados.

Copy copia los elementos seleccionados a una memoria temporal permitiendo ser insertados posteriormente, los elementos que actualmente se encuentran seleccionados no se suprimen, cuando se aplica nuevamente la opción **Copy** a una nueva selección, los elementos seleccionados anteriormente (si es que los había) se eliminan de la memoria temporal quedando los actualmente seleccionados.

Paste inserta los elementos que se almacenaron previamente en la memoria temporal mediante la opción **Copy**, al seleccionar esta opción se presenta una ventana en donde se puede especificar un incremento a todas las coordenadas de los nudos y a los nudos extremos de los elementos guardados previamente con la opción **Copy** (ver figura 4.9).

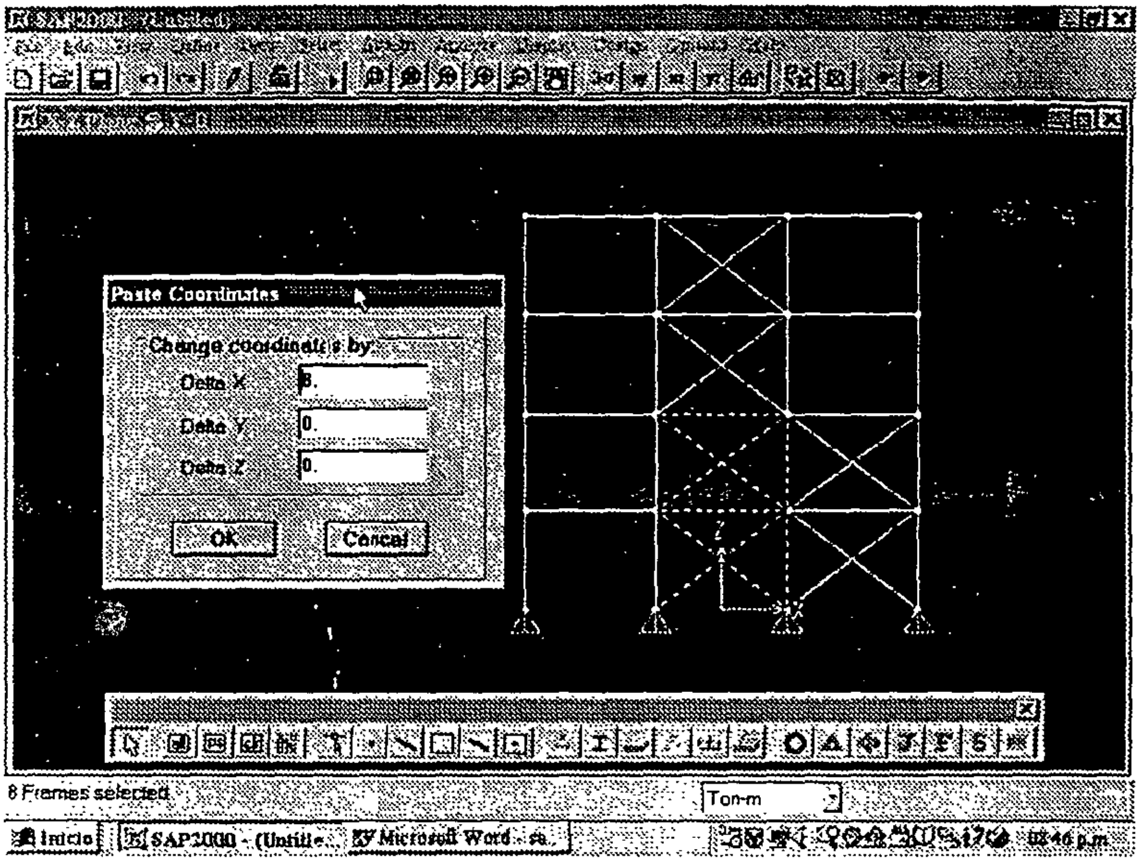


Figura 4.9 Opción **Paste** del menú **Edit**.

Move Una vez seleccionados algunos nudos este comando permite cambiar las coordenadas de los nudos seleccionados desplazándolos hacia nuevas posiciones obtenidas a partir de sus coordenadas actuales y de la información que el usuario proporcione en la ventana que se despliega cuando se elige esta opción (ver figura 4.10).

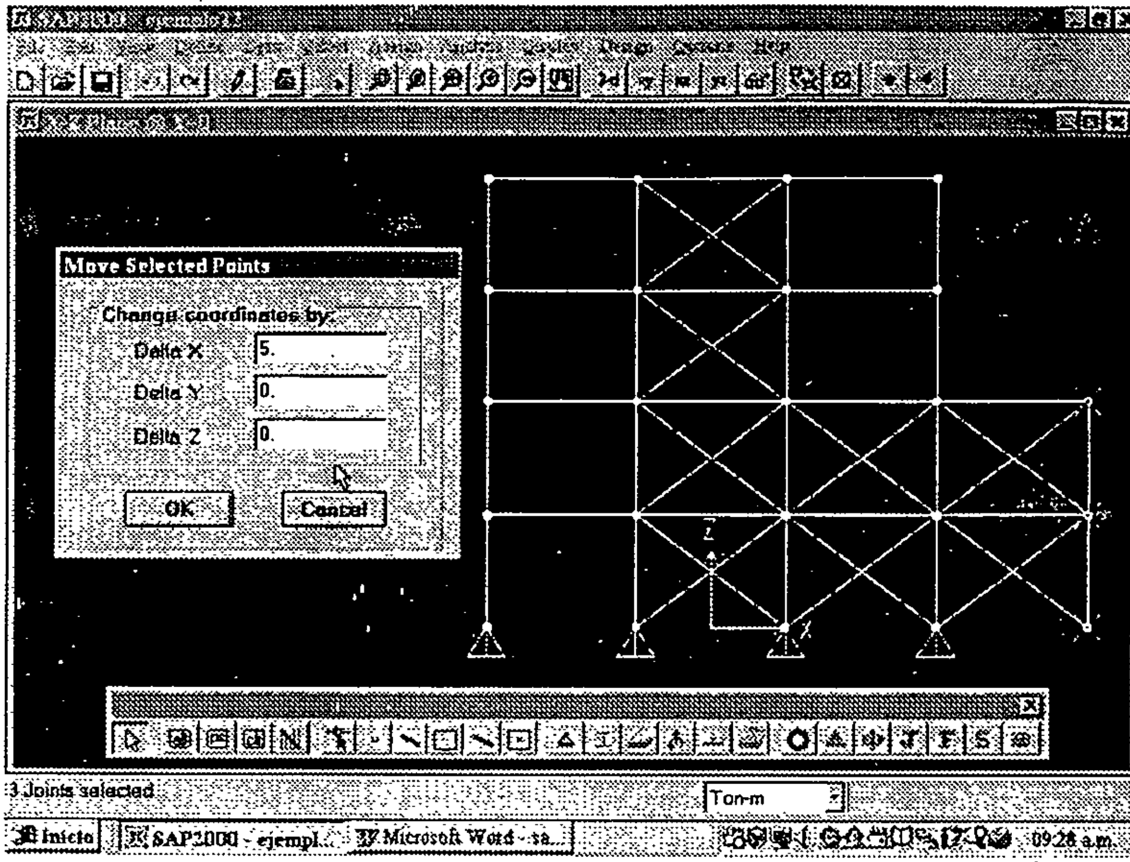


Figura 4.10 Información complementaria de la opción **Move** del menú **Edit**.

Replicate Una de las opciones más poderosas con la que se pueden realizar varios efectos es la opción **Replicate** del menú **Edit**, dentro de esta opción a su vez se encuentran disponibles otras 3, **Linear**, **Radial** y **Mirror.**

La opción **Linear** permite realizar varias copias de los elementos seleccionados, esas copias se pueden realizar en cualquiera de las direcciones x, y o z, por ejemplo si las copias se quieren realizar en dirección x se especifica un valor de x diferente de cero en la caja respectiva y cero en las demás (ver figura 4.11), desde luego se pueden especificar valores diferentes de cero con el efecto correspondiente, la opción **Radial** permite realizar copias en dirección radial (angular) especificando el eje alrededor del cual se van a hacer las copias así como el incremento en grados y el número de estas.

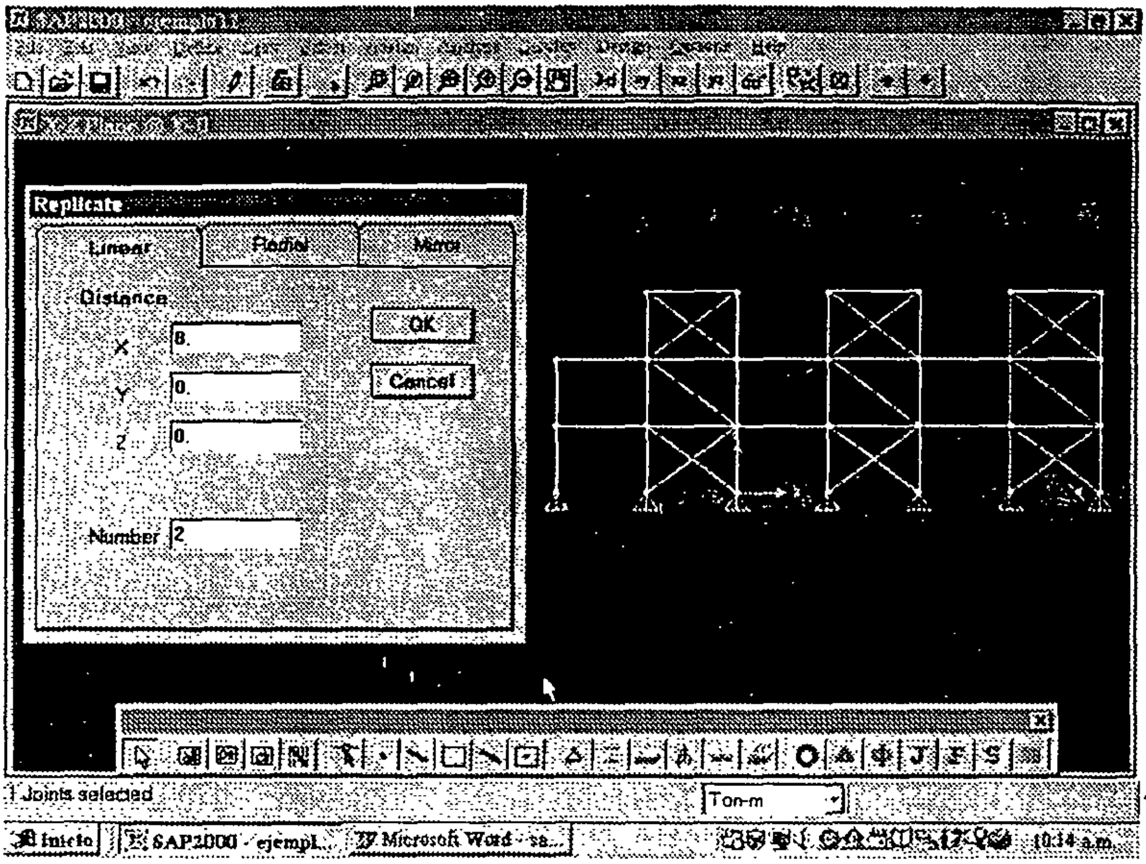


Figura 4.11 Efecto de **Linear** en la opción **Replicate** del menú **Edit**.

La opción **Mirror** permite realizar una copia tipo espejo de los elementos seleccionados especificando la posición del espejo mediante la selección de un plano (xy, yz o xz) y la distancia del origen a la posición del espejo (ver figura 4.12), esta opción resulta muy útil cuando se tiene una estructura simétrica ya que se introduce una parte de la misma y se genera la otra (parte simétrica) mediante la opción **Mirror**

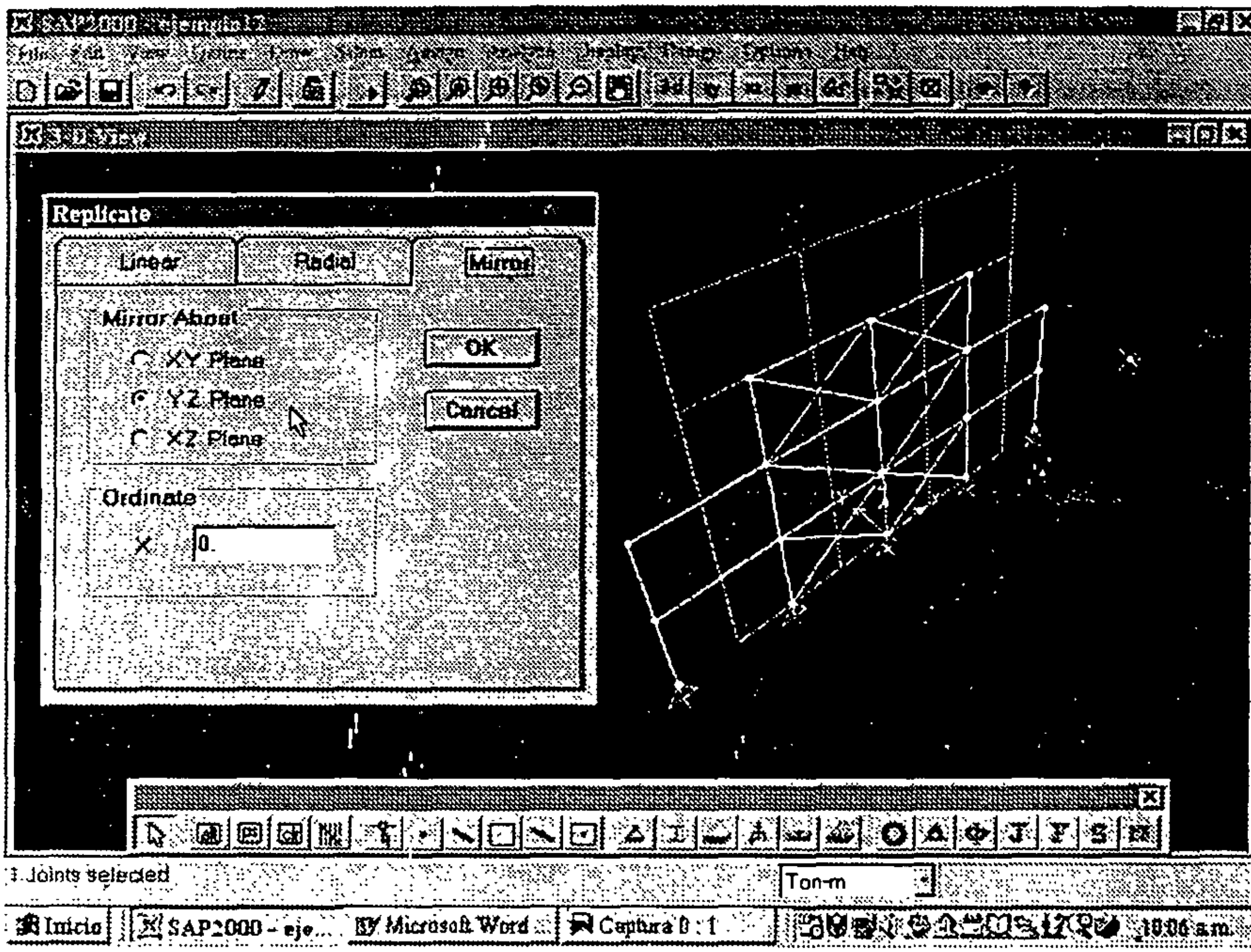


Figura 4.12 Efecto al seleccionar **Mirror** de la opción **Replicate** en el menú **Edit**.

4.4 DEFINICIÓN Y ASIGNACIÓN DE MATERIALES

En el menú Define en la opción **Materials** se podrán especificar las características de los materiales del cual estarán formados los elementos estructurales, en este menú se pueden especificar materiales tales como concreto, acerc y otros; después de seleccionar esta opción aparece el cuadro que se muestra en la figura 4.13, en donde como puede observarse mediante la opción **Modify/Show Materials**, se mostrarán con la posibilidad de modificar algunas caracteristicas del material que interviene para el análisis y el diseño de elementos (ver figura 4.14).

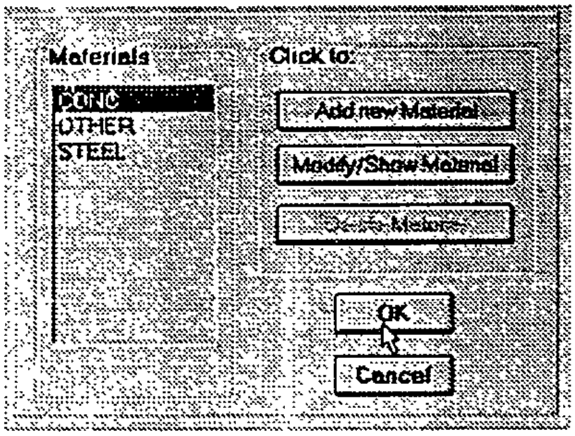


Figura 4.13 Ventana para definición de materiales.

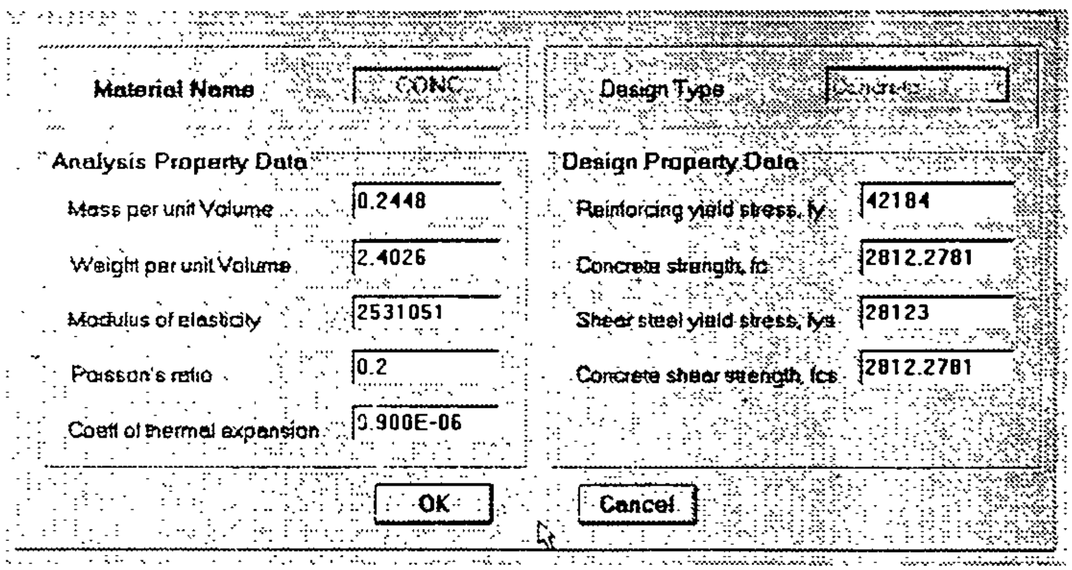


Figura 4.14 Ventana **Modifi/Show Material** de la opción **Define Materials**.

Add New material permite definir un nuevo material, se tendrá que especificar desde el nombre (**Material Name**), formado por un conjunto de hasta ocho caracteres el cual hará referencia a este material, se tendrán que proporcionar por lo menos los datos que se muestran en el cuadro análisis **Property Data**, sobre todo el módulo elástico y la relación de Poisson, en el caso de que se requiera considerar el peso propio en alguna condición de carga se tendrá que proporcionar el peso por unidad de volumen, si se va a solicitar alguna opción de análisis dinámico en donde se quiera considerar a la masa de la estructura repartida a lo largo de sus elementos será necesario proporcionar el valor de la masa por unidad de volumen, en el caso de que se requiera considerar el efecto de temperatura es necesario proporcionar el coeficiente de expansión térmica (ver figura 4.15).

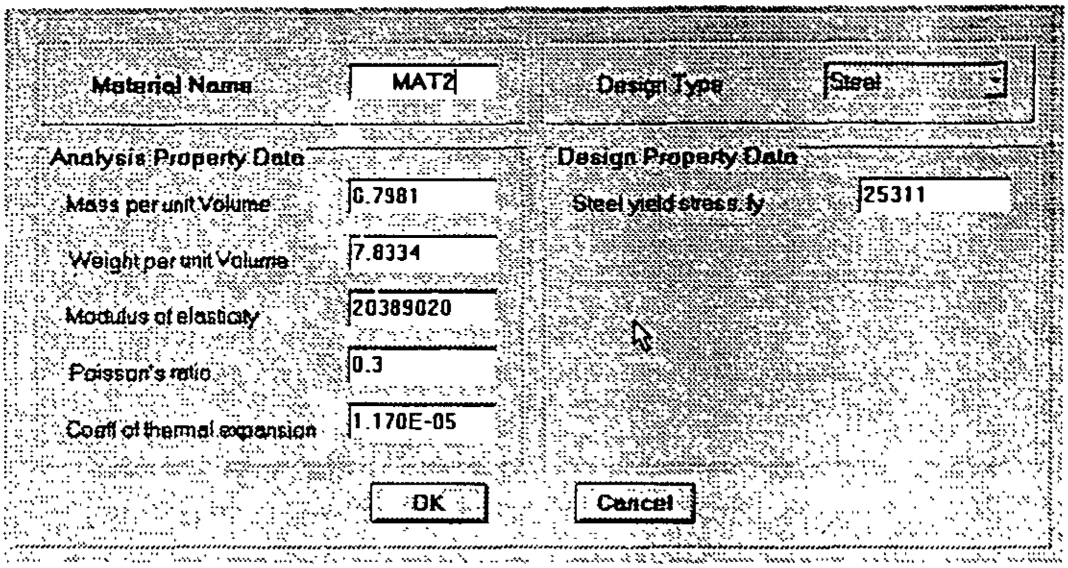


Figura 4.15 Opción **Add New** material de **Define Materials**

Se pueden definir varios materiales dependiendo de los que se requieran para especificar a los elementos en la estructura, la opción **Materials** del menú **Define** también permitirá eliminar algún material de los que se muestran en el cuadro **Materials** con excepción de los materiales **Conc** y **Steel**, para ello sólo se hace clic en el nombre del material a eliminar y luego en el botón **Delete Material**.

4.6 DEFINICION Y ASIGNACION DE PROPIEDADES GEOMETRICAS

En el menú **Define** también se encuentra presente la opción para definir características de las secciones transversales (**Frame Sections**) de los elementos que están presentes en la estructura por analizar. En la ventana correspondiente (ver figura 4.16), se tiene la opción **Import** para seleccionar las propiedades de una base de datos con extensión **PRO**, la versión educativa del programa **SAP2000** proporciona los archivos **Aisc.Pro**, **Cisc.Pro** y **Sections.Pro** de los dos primeros se pueden seleccionar algunas formas comunes, esos archivos se encuentran en la carpeta **SAP2000e**.

También se pueden definir las propiedades a partir de formas comunes mediante la opción **Add**, otras opciones que también se encuentran disponibles permiten modificar (**Modify/Show Section**) o eliminar alguna propiedad (**Delete Section**),

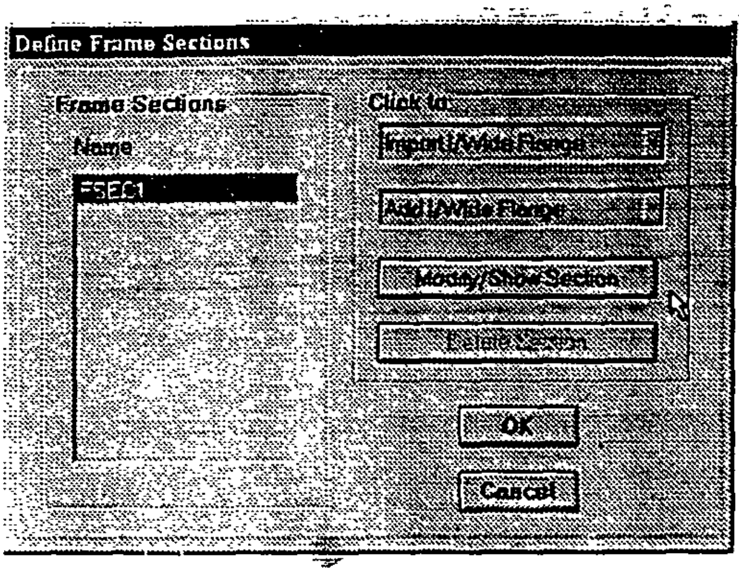


Figura 4.16 Opciones en **Define Frame Sections** del menú **Define**.

En la opción **Add** se tendrá que seleccionar la forma de la sección (rectangular, circular, tee, general, etc., ver figura 4.17).

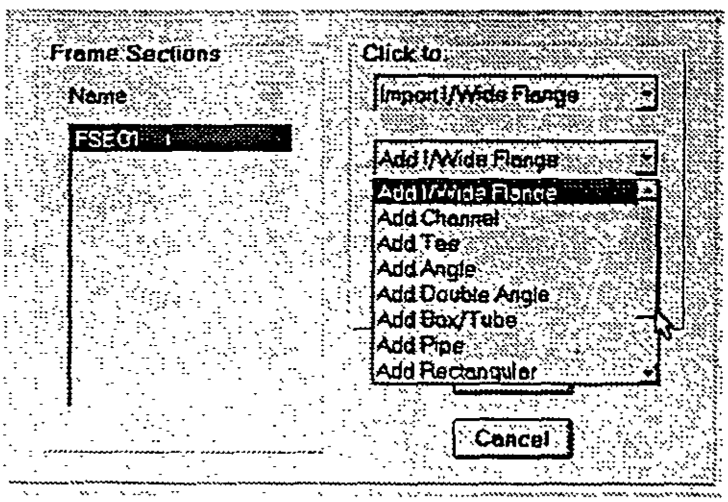


Figura 4.17 Selección de una forma predeterminada.

Una vez seleccionada la forma habrá que proporcionar algunas de las dimensiones de la misma con las cuales el programa obtiene de manera automática las propiedades geométricas de la forma definida (ver figura 4.18)

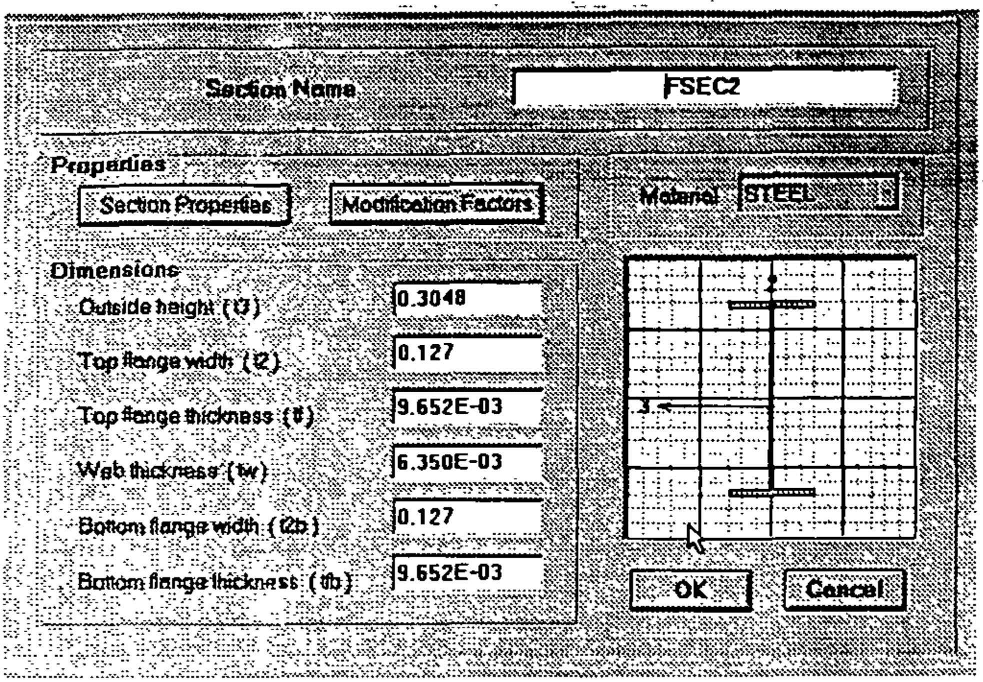


Figura 4.18 Especificación de las dimensiones de la forma de una sección transversal seleccionada.

El nombre de la sección se puede cambiar modificando el contenido del cuadro en blanco (**Section Name**), este nombre se utilizará para referencias posteriores (asignar esta sección transversal a uno o más elementos del modelo). Las características a modificar se presentan en el cuadro **Dimensions**, del cuadro **Material** se deberá seleccionar el material (los materiales se definieron previamente, ver párrafos anteriores) del cual esta o estará formada esa sección transversal. Una vez proporcionadas las dimensiones de la forma de la sección transversal se pueden mostrar sus propiedades geométricas (área, momentos de inercia, etc.) seleccionando la opción **Section Properties** del cuadro **Properties**, se pueden modificar (aumentar o disminuir en cierta proporción) algunas de esas propiedades modificando el factor correspondiente a la propiedad que se quiera modificar (el factor que se especifica es con respecto a la unidad) para ello habrá que seleccionar el botón **Modification Factors** del cuadro **Properties** y modificar el contenido del cuadro en blanco que corresponda a la propiedad que se quiere modificar.

Una vez definidas las distintas secciones de los diversos elementos estructurales habrá que indicar la sección transversal que corresponda al o a los elementos estructurales (la forma de la estructura ya se ha generado), primero se seleccionan los elementos que tiene una misma sección transversal, para ello se puede utilizar algún método de selección de la opción **Select del menú** con el mismo nombre, enseguida seleccionar **Sections** de la opción **Frame** en el menú **Assign,** con lo que aparece la ventana que se muestra en la figura 4.16, por último, en esa ventana se tendrá que seleccionar el nombre de la sección (la cual se definió previamente) del cuadro **Frame Sections,** después de hacer clic en el botón OK se asigna a los elementos seleccionados las características especificadas en la sección transversal seleccionada. La operación anterior se repetirá tantas veces como sea necesario para asignar secciones a todos los elementos que componen al modelo.

4.6 CONDICIONES DE FRONTERA, TIPOS DE APOYO

Para especificar los tipos de apoyo o condiciones de frontera de la elástica de la estructura primero se seleccionan aquellos nudos que tengan las mismas restricciones de desplazamiento, esto se hace con algunas de las opciones aplicables del menú **Select** y después seleccionar **Restraints** de la opción **Joint** del menú **Assign** (ver figura 4.19), desplegándose la ventana que se muestra en la figura 4.20, en ella se habrá de indicar el tipo de restricción que tendrán los nudos que se han seleccionado previamente, a menos que se modifiquen las direcciones 1, 2 y 3, corresponden a las direcciones globales X, Y y Z respectivamente, se puede seleccionar algún tipo de apoyo particular de uso común haciendo clic en alguno de ellos en el cuadro **Fast Restraints**. La operación anterior se puede aplicar en repetidas ocasiones para especificar completamente todos los nudos restringidos que tiene el modelo.

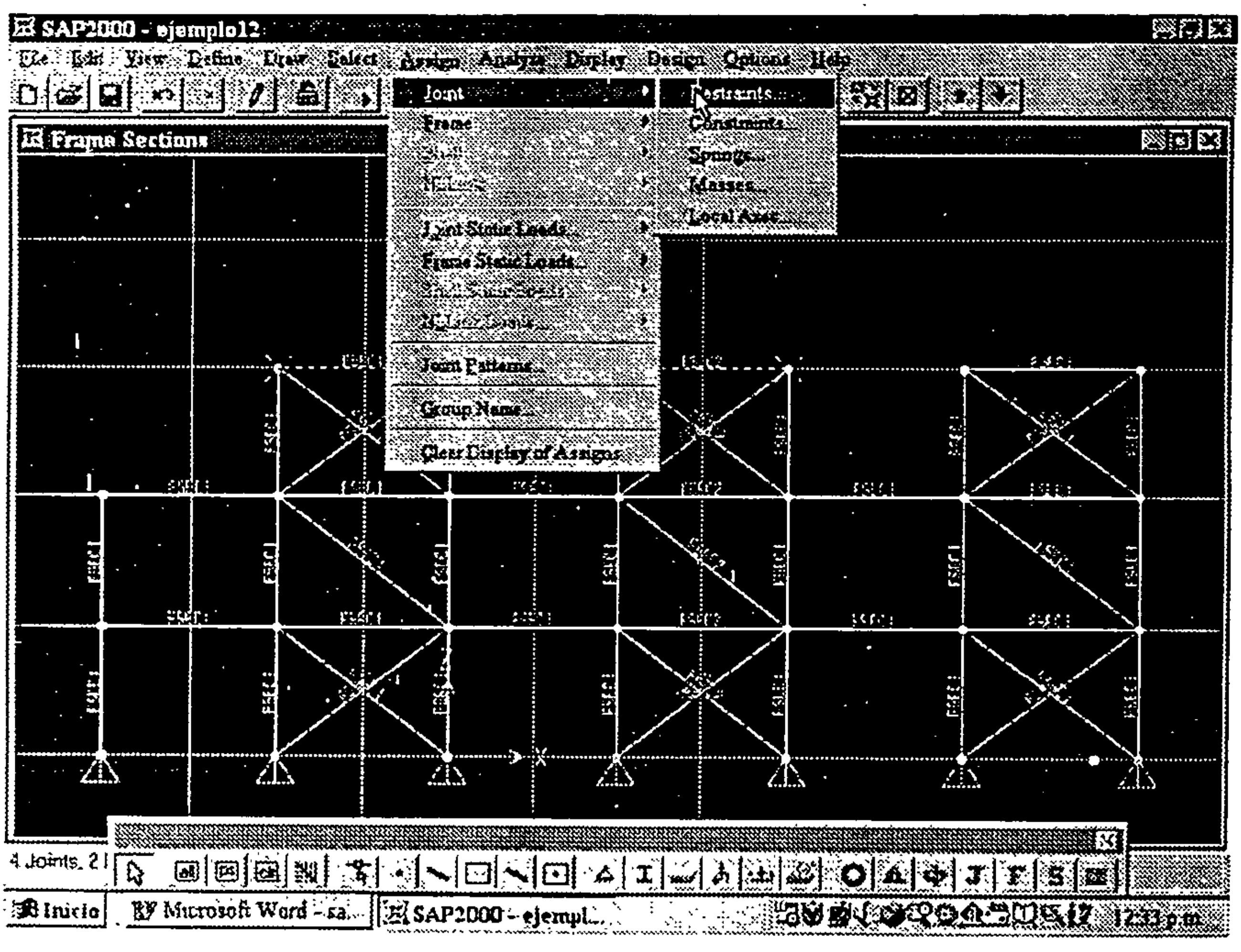


Fig. 4.19 Selección de restricciones.

Para cambiar las restricciones de algún nudo se requiere seleccionarlo y asignarle las nuevas restricciones, si se desea que ese nudo tenga posibilidad de desplazamiento lineal y angular en todas las direcciones habrá que dejar en blanco todos los cuadros del marco **Restrainsts in Local Directions** o bien hacer clic en el icono con un punto negro del marco **Fast Restraints**.

4.7 ASIGNACION DE FUERZAS Y COMBINACIONES

Para introducir diversos tipos de fuerza estática al modelo, primero habrá que definir condiciones de carga estática, para ello se selecciona la opción **Static Load Cases del menú Define** mostrándose la ventana de la figura 4.20, en ella se puede adicionar una nueva **(Add New Load)**, modificar características de una que existe (**Change Load**), o suprimir una condición de carga (**Delete Load**), resulta lógico que al menos se debe proporcionar una condición de carga.

El nombre de la condición se especifica en el cuadro en blanco debajo de **Load** y si se quiere considerar el peso propio en esa condición de carga se debe de proporcionar el valor de 1 en el cuadro en blanco debajo de Self **Weight Multiplier**, una vez que se han introducido los datos anteriores se puede seleccionar **Add...** para definir una nueva condición de carga o bien **Change...** para cambiar los datos de la condición de carga seleccionada (con fondo oscuro) por los datos de los cuadros en blanco.

Para modificar el nombre y el multiplicador del peso propio además de introducir el nuevo valor en los cuadros en blanco habrá que seleccionar la condición que se quiera modificar haciendo clic sobre ella, con lo que el fondo de la condición seleccionada cambia a obscuro y después hacer clic en el botón **Change Load** se realizan los cambios indicados ya que hasta que se ha hecho clic en este botón quedan registrados esos cambios es decir no basta modificar el contenido de las cajas en blanco.

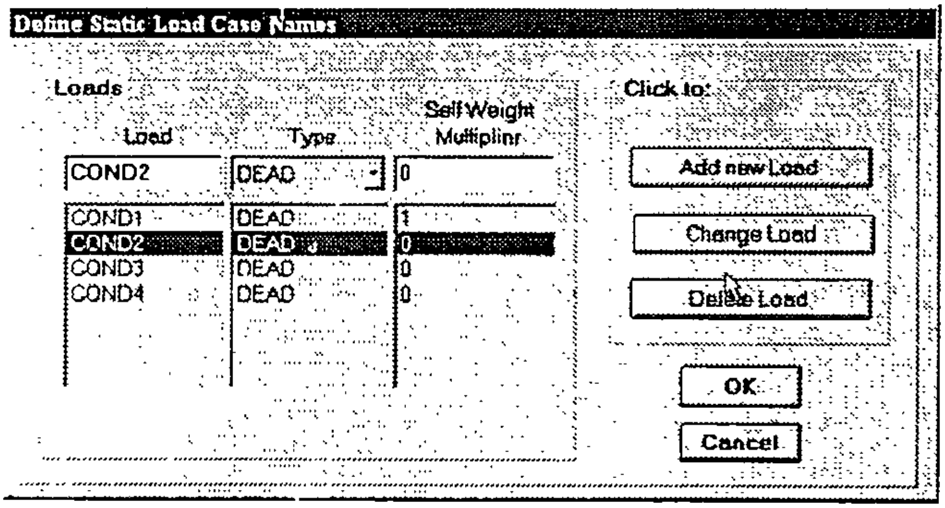


Fig. 4.20 Definición de condiciones de carga estática.

Para la asignación de fuerzas y o momentos a los nudos habrá que seleccionar aquellos nudos que tengan las mismas fuerzas y después seleccionar **Forces** de la opción **Joint Static Loads** en el menú **Asign** (ver figura 4.21)

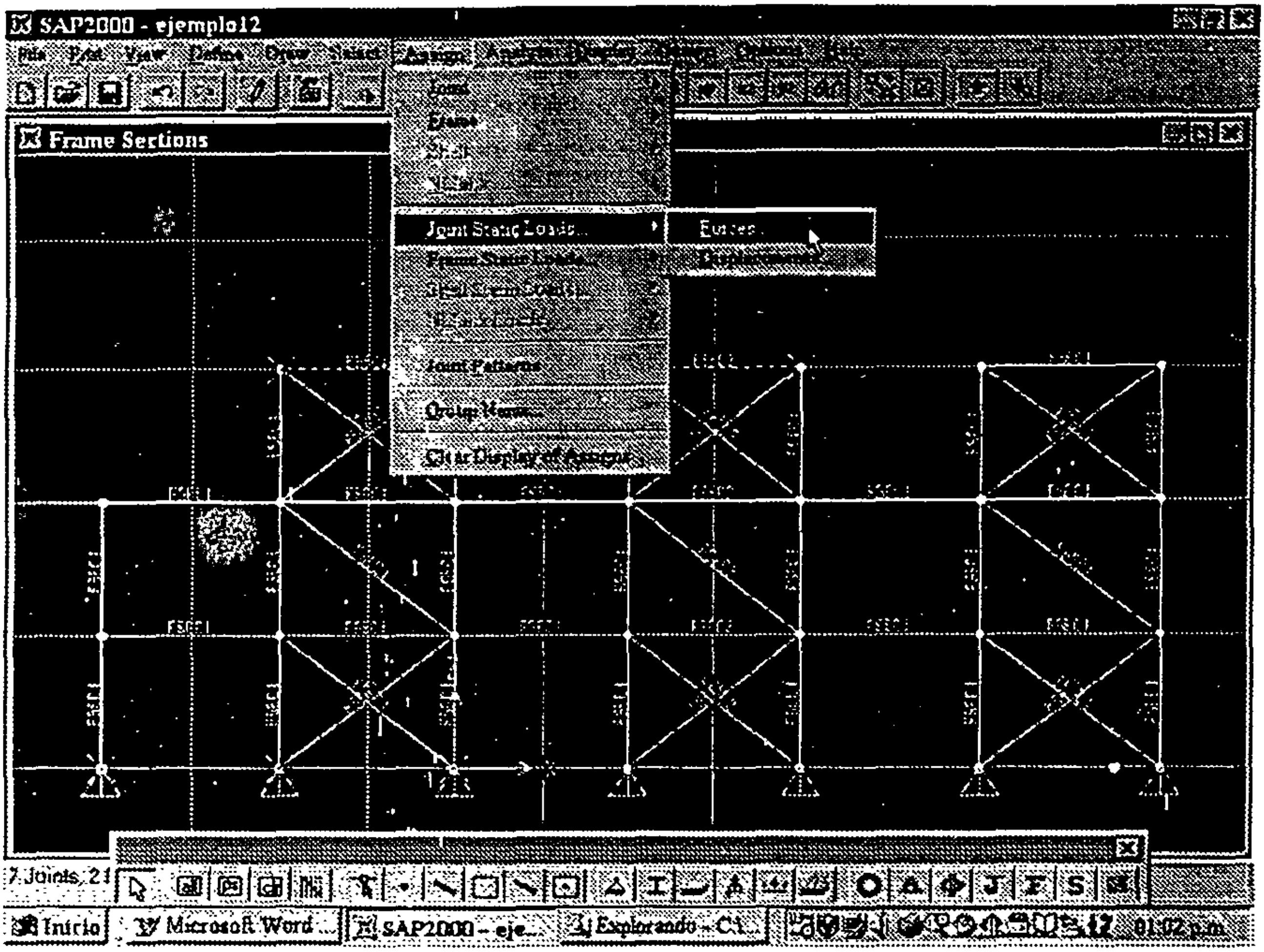


Fig. 4.21 Opción para asignar fuerzas a los nudos previamente seleccionados.

Enseguida se despliega la ventana mostrada en la figura 4.22, en ella se habrá de seleccionar del marco **Load Case Name** la condición en que se incluirán las fuerzas que se están especificando (por omisión aparece **LOAD1**), en el cuadro correspondiente a la dirección de la fuerza y o momento que actuará sobre los nudos seleccionados se introducirán los valores respectivos (en el marco Loads), también se encuentran disponibles las opciones:

> **Add To Existing Loads** (seleccionada por omisión), la cual adicionará a las fuerzas existentes en los nudos seleccionados las nuevas fuerzas que se están especificando, es decir si los nudos ya tenían fuerzas se les adicionarán las nuevas fuerzas cuyos valores se han introducido en los cuadros en blanco.
>
> **Replace Existing Loads** permitirá eliminar las fuerzas existentes en los nudos seleccionados remplazándolas por las que se están especificando en el marco **Loads**.
>
> **Delete Existing Loads** suprimirá las fuerzas existentes en los nudos seleccionados, independientemente de los valores que se están especificando en el marco **Loads**.

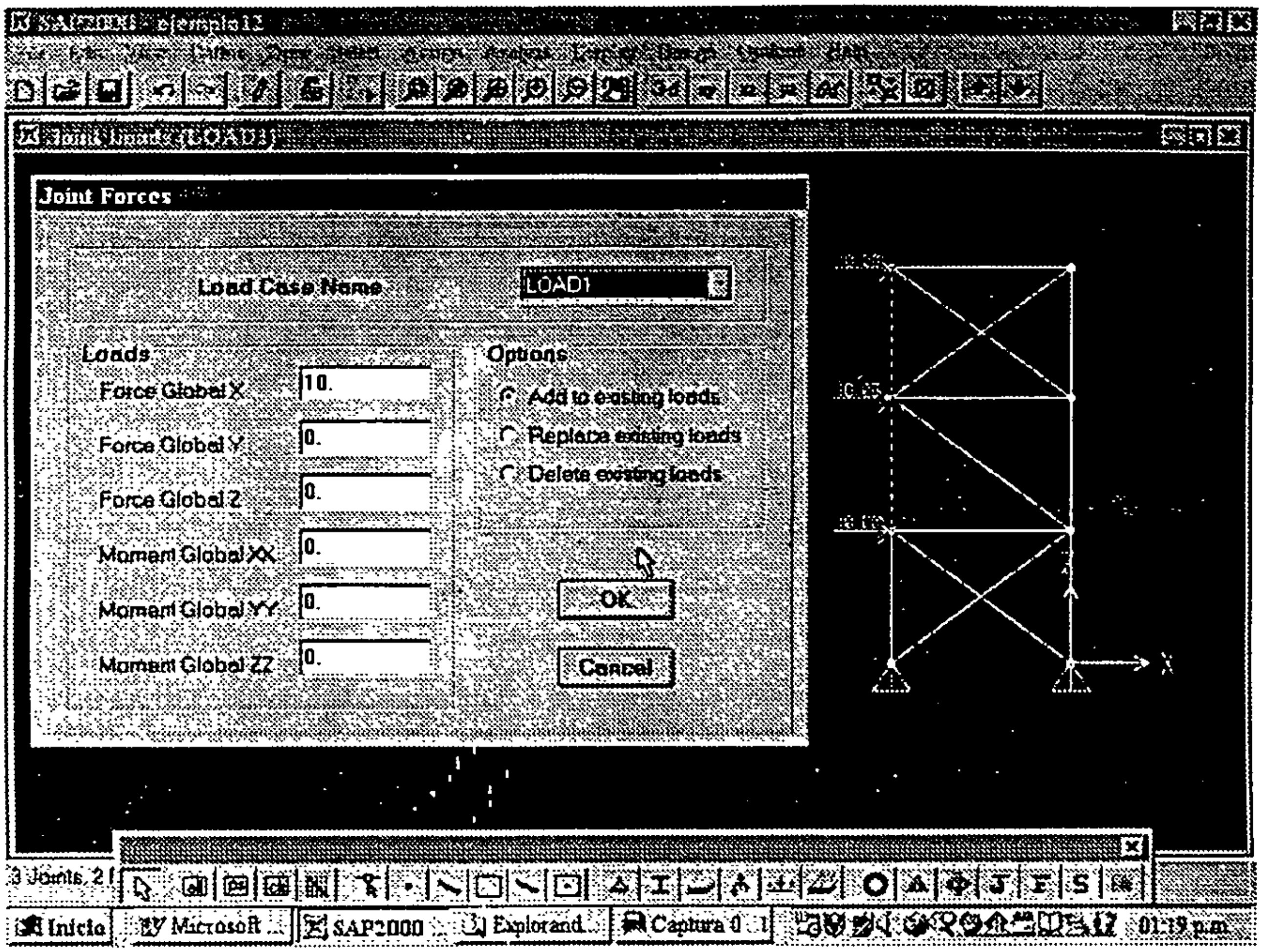


Fig. 4.22 Especificación de fuerzas en los nudos.

Para definir y asignar fuerzas a las barras primero se seleccionan las barras a las cuales se les asignarán las mismas fuerzas, después se selecciona el tipo de fuerza (uniforme, concentrada, variación lineal, etc.) de la opción **Frame Static Loads** del menú **Assign** (ver figura 4.23).

Por ejemplo para cargas puntuales y o uniformes en las barras se muestra la ventana de la figura 4.24, en donde se selecciona el nombre de la condición a donde se incluirán las fuerzas que se están especificando, así como el tipo de carga (fuerza o momento) así como la dirección en que actuarán y la opción a utilizar (**Add...**, **Replace...** y **Delete...**). En los cuadros en blanco del marco **Point Loads** se especifica el valor de las cargas concentradas así como la posición de cada una de ellas con respecto a la longitud del elemento, es decir si el valor de **Distance** es 0.5 indica que la carga está aplicada a la mitad del elemento, en el cuadro en blanco del marco **Uniform Load** se proporciona el valor correspondiente a la carga uniforme que actuará sobre el elemento. Pueden especificarse simultáneamente cargas concentradas y uniformes o sólo algún tipo de los anteriores.

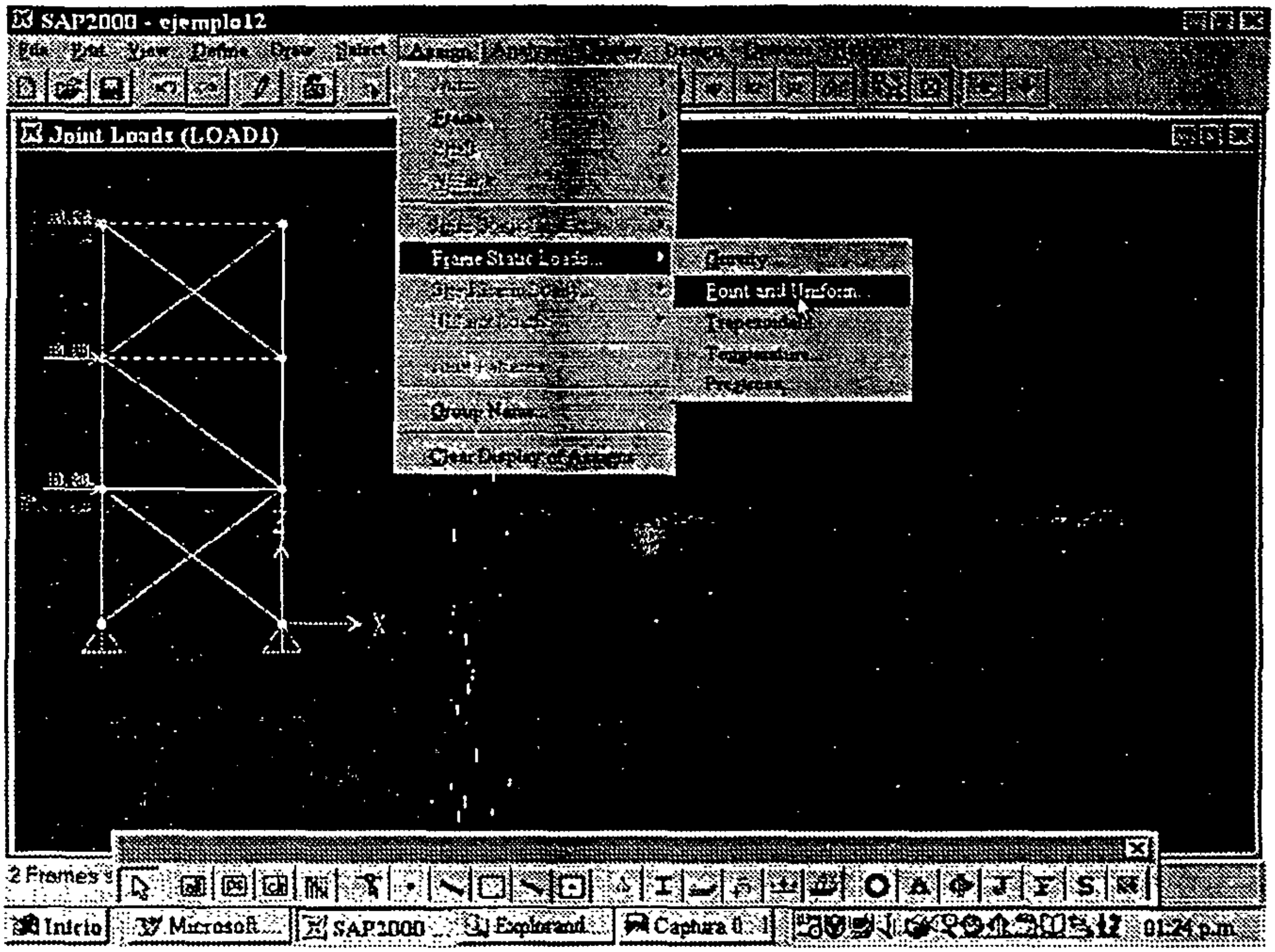


Fig. 4.23 Opción para introducir fuerzas en las barras.

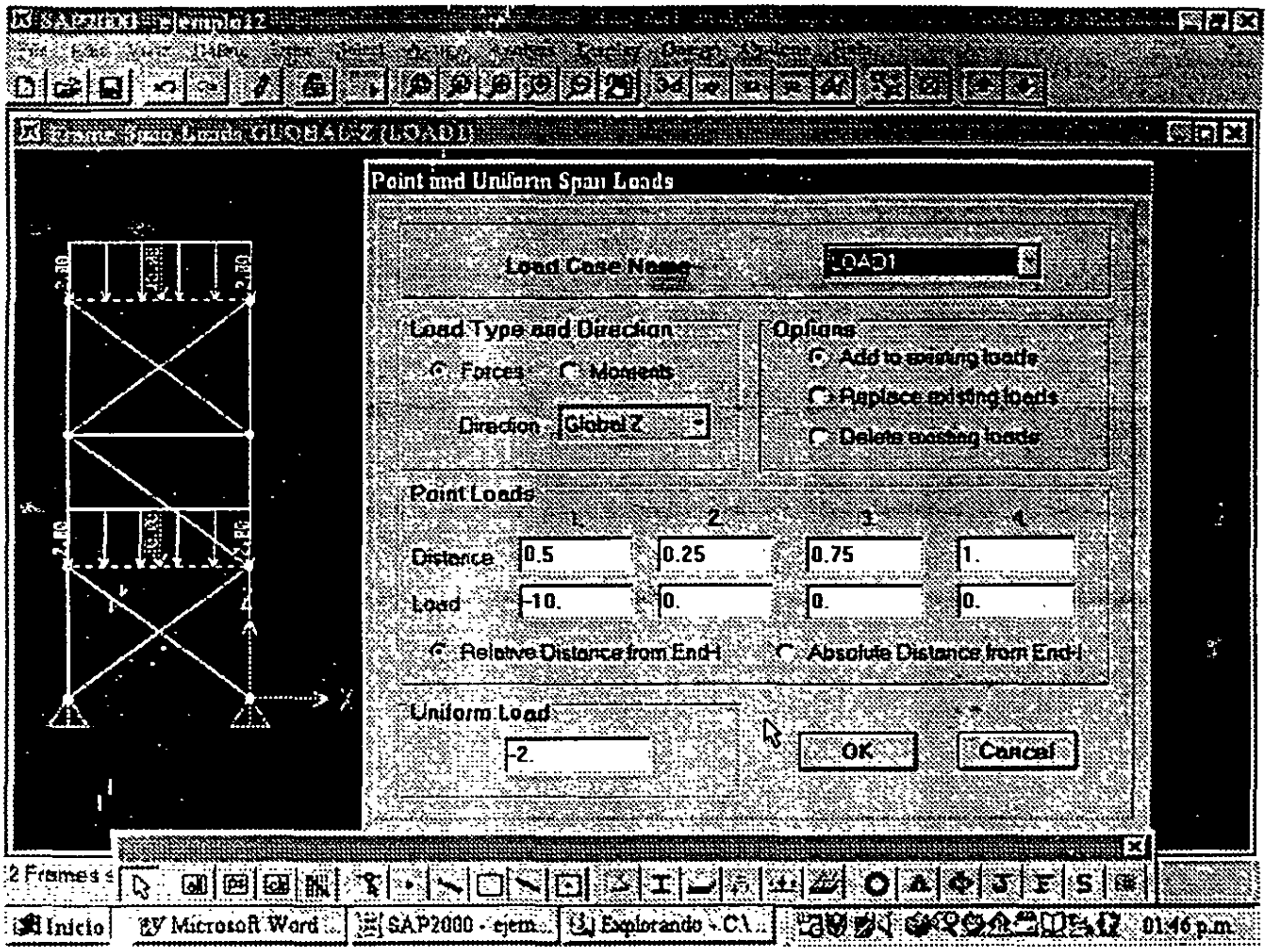


Fig. 4.24 Definición de fuerzas uniformes y/o concentradas en las barras.

Una vez que se especifican las fuerzas y se hace clic en el botón **OK** se ejecuta la opción seleccionada, en el caso de que esta sea adicionar o remplazar cargas, estas se muestran con sus características en el área de dibujo de la pantalla.

Definidas las condiciones de carga se pueden realizar combinaciones de las anteriores, es decir condiciones de cargas dependientes, para ello se selecciona la opción **Load Combinations** del menú **Define** mostrándose la ventana de la figura 4.25, con la posibilidad de adicionar, modificar y suprimir combinaciones de carga estas opciones se muestran en el marco **Combinations** las combinaciones que se tengan definidas hasta el momento

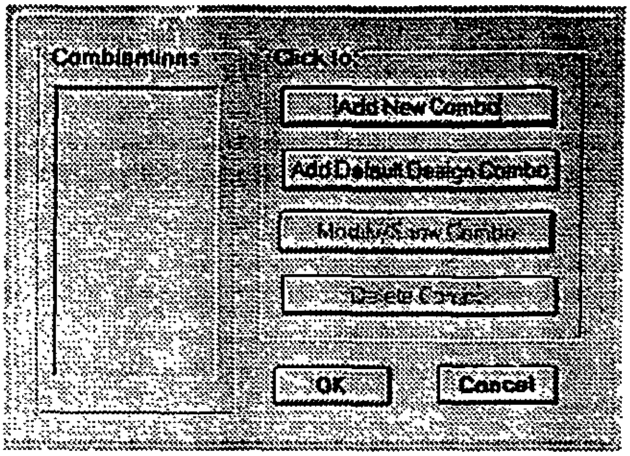


Fig. 4.25 Ventana para especificar y modificar combinaciones.

La opción para adicionar una nueva combinación despliega la ventana que se muestra en la figura 4.26, ahí se especificará el nombre, tipo y algún título para la combinación. Para definir las condiciones de carga que participarán en la combinación que se especifica, así como su respectivo factor de participación (con relación a la unidad, 1=100%) se selecciona el nombre y se modifica el valor en el cuadro en blanco debajo de **Scale Factor** en el marco **Define Combination** y después se hace clic en cualquiera de los botones **Add, Modify,** o **Delete**.

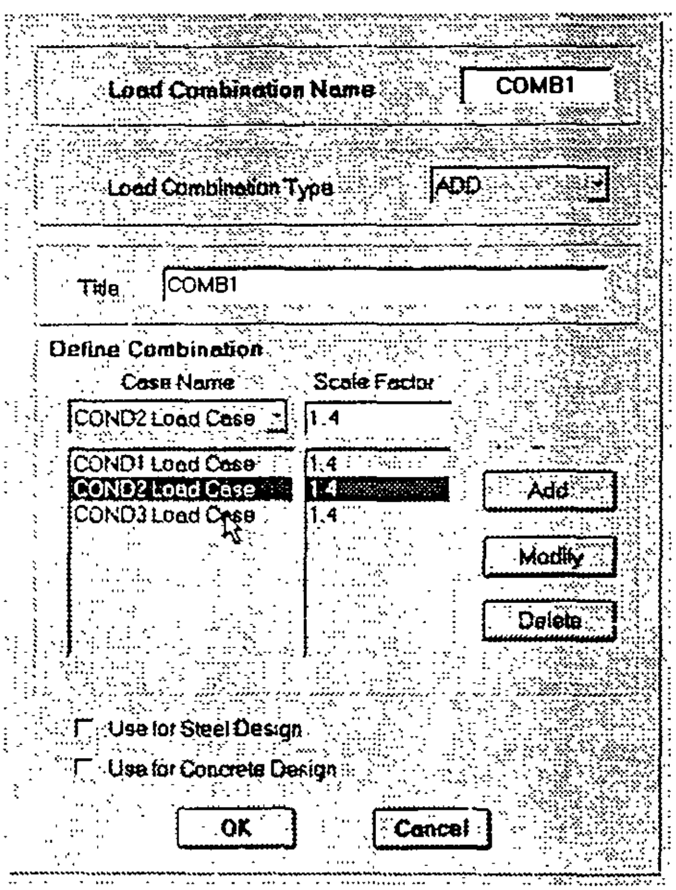


Fig. 4.26 Especificación de las características de una combinación.

Cuando se han especificado los datos de la combinación se hace clic en el botón **OK**.

Una vez que se han definido combinaciones se puede modificar sus características con la opción **Modify/Show Combo** o bien suprimir alguna combinación con la opción **Delete Combo**, cualquiera de estas opciones se selecciona haciendo clic sobre ella en el marco **Combinations**. Se pueden especificar tantas combinaciones como el problema de Análisis lo requiera.

4.8 OPCIONES DE ANALISIS SELECCIÓN DE RESULTADOS

Una vez que se han especificado completamente las caracteristicas geométricas, elásticas, condiciones de frontera y fuerzas se está en posibilidades de que el programa **SAP2000** realice el Análisis Estructural del modelo, sin embargo es conveniente especificar algunas opciones de Análisis, para ello se selecciona la opción **Set Options** del menú **Analyze**, desplegándose la ventana que se muestra en la figura 4.27.

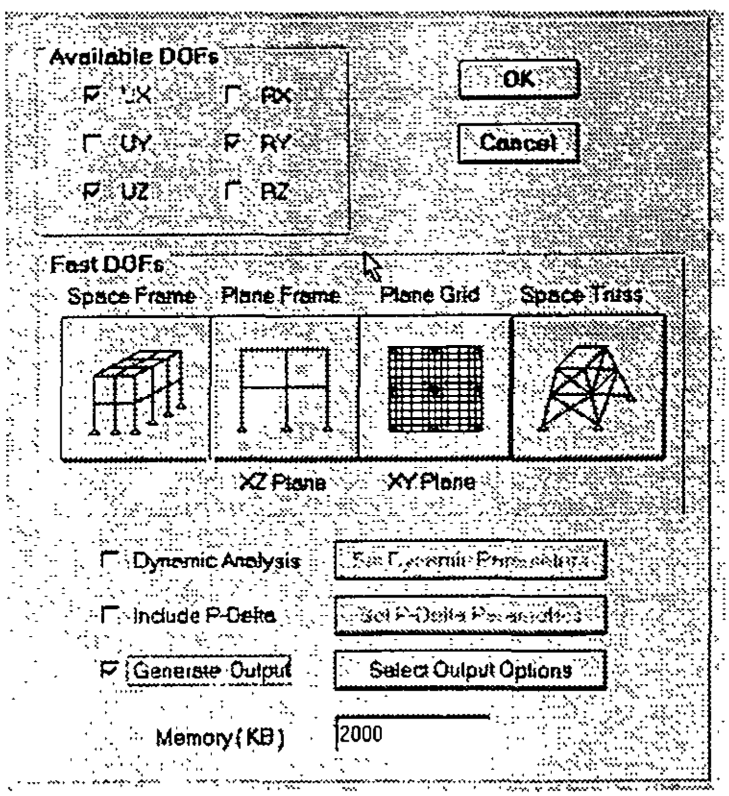


Figura 4.27 Selección de opciones de Análisis.

En ella se pueden seleccionar las componentes de desplazamiento independientes o grados de libertad que se considerarán para el análisis, **SAP2000** permite analizar estructuras en un espacio tridimensional por lo que cada nudo tiene la posibilidad de desplazarse lineal y angularmente en tres direcciones ortogonales, es decir en general posee 6 grados de libertad (a menos que se indique otra alternativa).

Si la estructura está contenida en un plano es conveniente indicar los grados de libertad que no intervienen en el Análisis con objeto de eliminar la posibilidad de inestabilidad en dirección perpendicular al plano de la estructura, disminuyendo además el tiempo de ejecución del Análisis, lo anterior se realiza desactivando grados de libertad en el marco **Available DOFs** o bien permitiendo que el programa lo realice dependiendo del tipo de estructura que se selecciona haciendo clic en alguno de los iconos que se muestran en el marco **Fast DOFs** y que corresponda con las características de la estructura que se vaya a analizar.

En la parte inferior de la ventana se muestran las opciones de Análisis Dinámico y efectos P-Delta, también se pueden seleccionar resultados que han de almacenarse en el archivo de salida (nombre.OUT), en el último renglón se muestra en un cuadro en blanco el valor de la memoria reservada para la solución del problema, este valor deberá aumentarse en caso de que no sea suficiente cuando se muestre el mensaje correspondiente durante el proceso de Análisis

La selección de resultados del Análisis se puede realizar haciendo clic en el cuadro en blanco a la izquierda de **Generate Output** (ver figura 4.27) y después de hacer clic en el botón **Select Output Options** se muestra la ventana de la figura 4.28.

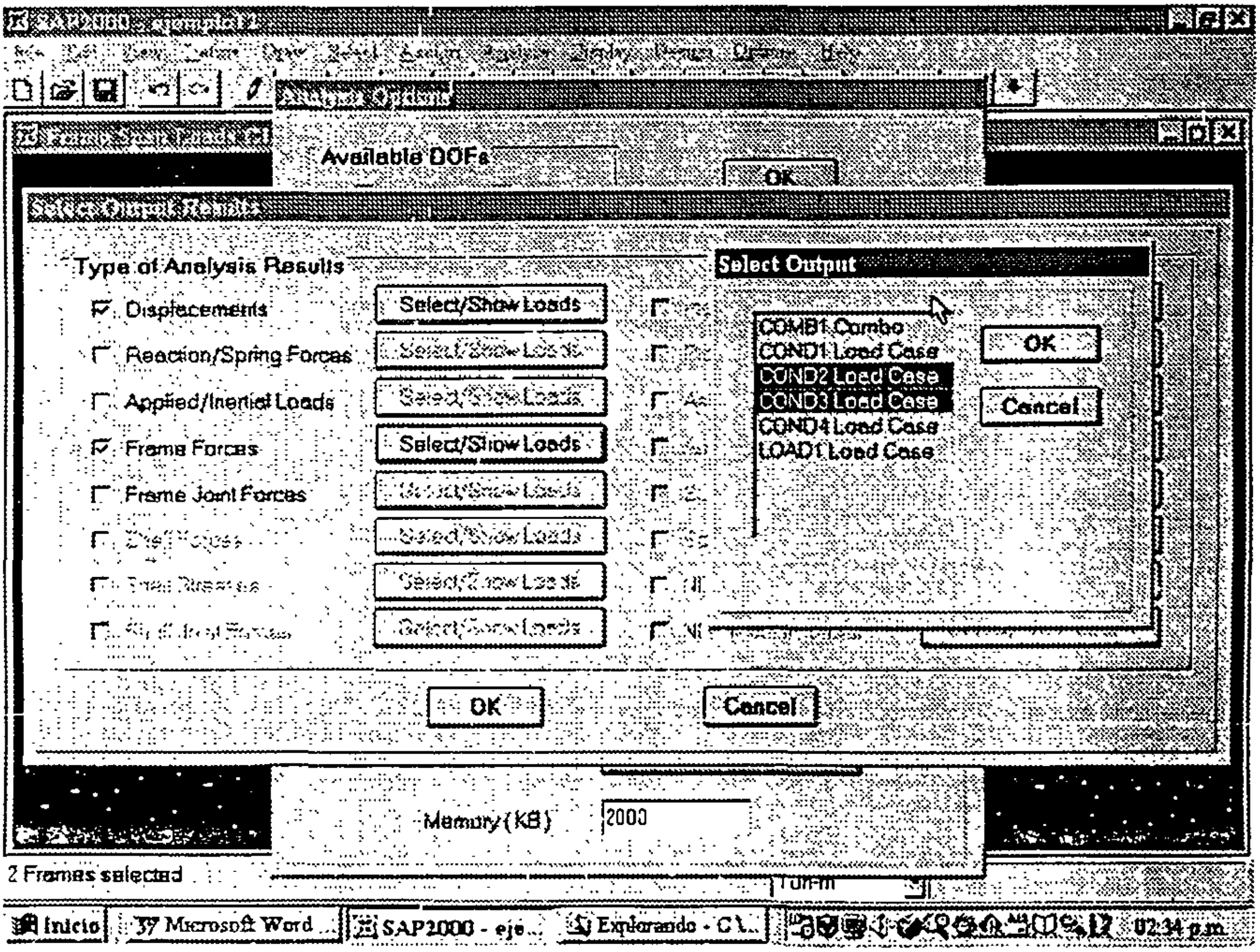


Figura 4.28 Selección de combinaciones de carga y tipos de resultados a incluirse en el archivo de salida.

En ella habrá que indicar los resultados que se incluirán en el archivo de salida haciendo clic en los cuadros en blanco y seleccionando para cada uno de esos resultados las condiciones de carga de los cuales se requieren los resultados seleccionados, lo anterior se logra haciendo clic en el botón **Select/Show Loads** correspondiente, con lo que se presentará una ventana mostrando las condiciones y combinaciones de Análisis que se han especificado para la estructura por analizar, en esa nueva ventana se deberán seleccionar las condiciones de carga para las que se requieren los resultados seleccionados, la condición o combinación de carga seleccionada se muestra con fondo oscuro, se puede seleccionar más de una combinación de carga arrastrando el ratón en el cuadro de selección.

Es necesario que se seleccione al menos una condición o combinación de carga para que los resultados se encuentren disponibles en el archivo de salida ya que de no hacerlo los resultados no se almacenarán (ver figura 4.28).

ANALISIS DE LA ESTRUCTURA

CAPÍTULO 5

5.1 ANALISIS DEL MODELO

Una vez que se han especificado las opciones de Análisis se puede solicitar la ejecución del mismo, seleccionando la opción **Run** del menú **Analyze** con lo que el análisis se efectúa y los resultados de esta fase se muestran en la ventana de la figura 5.1, en su extremo derecho se observa una barra de desplazamiento vertical que permite ver el contenido de la pantalla, al final de esta se muestra el botón **OK** haciendo clic en él se despliega en la mayoria de los casos la configuración deformada de la estructura para la primera condición de carga con lo que se está en posibilidades de tener acceso a varias opciones del menú **Display** las cuales nos mostrarán de manera gráfica y numérica algunos resultados del Análisis (desplazamientos, elementos mecánicos, etc.).

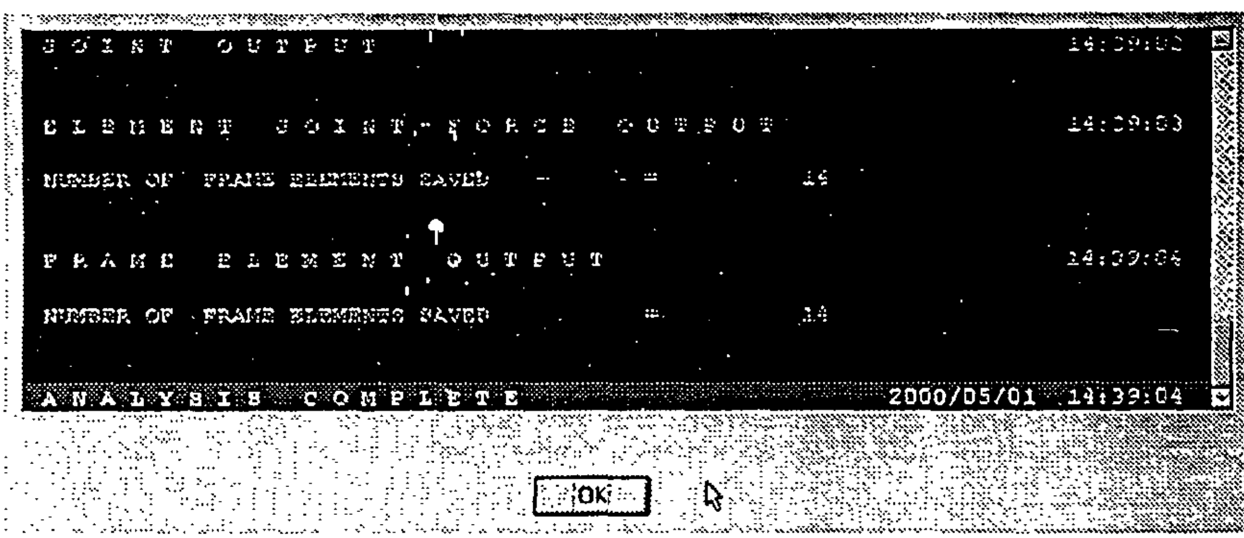


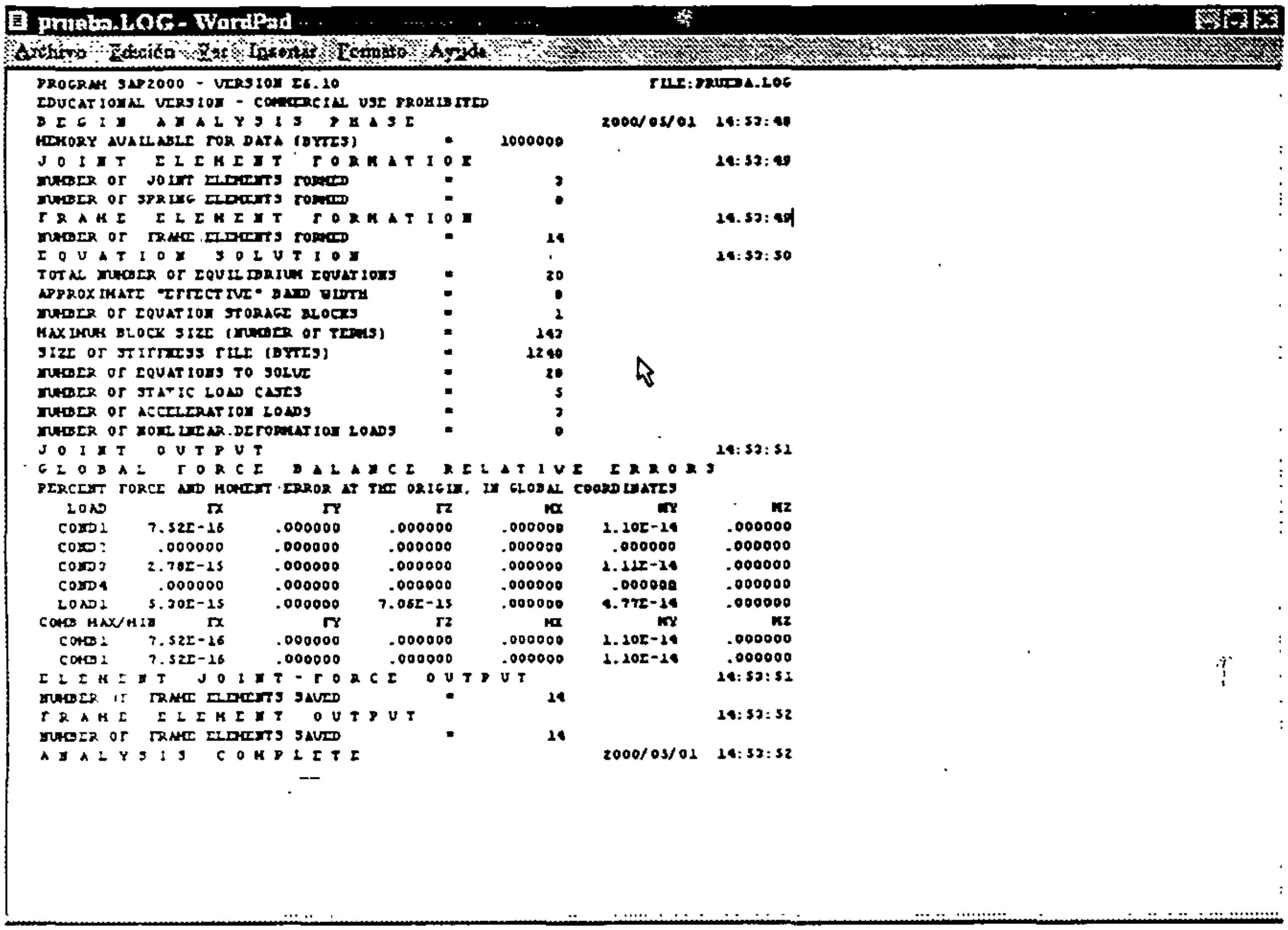
Figura 5.1 Ventana después de seleccionar la opción **Run** del menú **Analyze**

5.2 VERIFICACION DE ALGUNOS ELEMENTOS DEL PROCESO DE ANALISIS

Después de que el Análisis ha concluido se generan una serie de archivos con el mismo nombre que el archivo de datos pero con extensión diferente, algunos de los que se generan y que pueden ser de utilidad son:

El archivo nombre.LOG (ver figura 5.2), el cual contiene información de la fase de Análisis (memoria disponible, número de ecuaciones de equilibrio formadas, caracteristicas de la matriz de rigideces, balance de errores relativos y diversa información de salida).

El archivo nombre.$2k contiene los **datos de la estructura a analizar** como son: geometría, materiales, secciones, cargas, combinaciones, etc., tal y como se generaron por ejemplo mediante el editor gráfico del programa **SAP2000**, se puede recurrir a este archivo en el caso de que el archivo nombre.SDB sufra algún cambio que lo imposibilite para ser procesado por **SAP2000**.


prueba.LOG - WordPad
Archivo Edición Ver Insertar Formato Ayuda

```
PROGRAM SAP2000 - VERSION E6.10                                          FILE:PRUEBA.LOG
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED
B E G I N   A N A L Y S I S   P H A S E                            2000/03/01  14:53:48
MEMORY AVAILABLE FOR DATA (BYTES)            =     1000000
J O I N T   E L E M E N T   F O R M A T I O N                                  14:53:49
NUMBER OF  JOINT ELEMENTS FORMED             =           3
NUMBER OF SPRING ELEMENTS FORMED             =           0
F R A M E   E L E M E N T   F O R M A T I O N                                  14.53:49
NUMBER OF  FRAME ELEMENTS FORMED             =          14
E Q U A T I O N   S O L U T I O N                                              14:53:50
TOTAL NUMBER OF EQUILIBRIUM EQUATIONS        =          20
APPROXIMATE "EFFECTIVE" BAND WIDTH           =           8
NUMBER OF EQUATION STORAGE BLOCKS            =           1
MAXIMUM BLOCK SIZE (NUMBER OF TERMS)         =         143
SIZE OF STIFFNESS FILE (BYTES)               =        1240
NUMBER OF EQUATIONS TO SOLVE                 =          20
NUMBER OF STATIC LOAD CASES                  =           5
NUMBER OF ACCELERATION LOADS                 =           3
NUMBER OF NONLINEAR.DEFORMATION LOADS        =           0
J O I N T   O U T P U T                                                        14:53:51
G L O B A L   F O R C E   B A L A N C E   R E L A T I V E   E R R O R S
PERCENT FORCE AND MOMENT ERROR AT THE ORIGIN, IN GLOBAL COORDINATES
   LOAD         FX          FY          FZ          MX          MY          MZ
  COND1     7.52E-16     .000000     .000000     .000000    1.10E-14     .000000
  COND2      .000000     .000000     .000000     .000000     .000000     .000000
  COND3     2.78E-15     .000000     .000000     .000000    1.11E-14     .000000
  COND4      .000000     .000000     .000000     .000000     .000000     .000000
  LOAD1     5.30E-15     .000000    7.06E-15     .000000    4.77E-14     .000000
COMB MAX/MIN    FX          FY          FZ          MX          MY          MZ
  COMB1     7.52E-16     .000000     .000000     .000000    1.10E-14     .000000
  COMB1     7.52E-16     .000000     .000000     .000000    1.10E-14     .000000
E L E M E N T   J O I N T - F O R C E   O U T P U T                            14:53:51
NUMBER OF  FRAME ELEMENTS SAVED              =          14
F R A M E   E L E M E N T   O U T P U T                                        14:53:52
NUMBER OF  FRAME ELEMENTS SAVED              =          14
A N A L Y S I S   C O M P L E T E                                  2000/03/01  14:53:52
```

Figura 5.2 Contenido tipico del archivo nombre.LOG

El archivo nombre.EKO contiene una imagen o resultado del procesamiento de los datos contenidos en el archivo nombre.SDB generado mediante el editor gráfico del programa **SAP2000**, este archivo (nombre.EKO), contiene textos que indican las caracteristicas de los datos procesados por ejemplo. hay un titulo y encabezado para las coordenadas de los nudos seguido de éstas, es decir se despliega información respectiva para cada bloque de datos asi como los valores respectivos, únicamente se incluyen en este archivo los datos procesados.

En el caso de que se hayan seleccionado resultados para ser impresos éstos se encuentran en el archivo nombre OUT.

Es conveniente verificar algunas caracteristicas particulares del problema que se resolvió, por ejemplo que coincida el número total de grados de libertad que la estructura tiene con el número de ecuaciones de equilibrio que el programa formó y resolvió, también es conveniente verificar el número de elementos barra, placa, etc. que el programa procesó. Desde luego es conveniente verificar que todos los datos del problema fueron procesados por el módulo de Análisis, para todo lo anterior se recurre a revisar el contenido de los archivos mencionados en los párrafos anteriores, para tener acceso al contenido de esos archivos se puede recurrir a varios programas o procesadores de texto (por ejemplo Edit, Word Pad, Word, etc.)

SELECCION E INTERPRETACION DE RESULTADOS

CAPÍTULO 6

6.1 INTRODUCCION

Una vez que el Análisis se ejecuta sin que se hayan generado errores durante el mismo y después de hacer clic en el botón **OK** de la ventana que se muestra en la opción **Run** del menú **Analyze**, se pueden seleccionar varias opciones del menú **Display** que nos permitirán ver los diversos resultados tanto de manera gráfica como numérica, por ejemplo **Show Deformed Shape** nos mostrará la configuración deformada de la estructura para alguna condición de carga, **Show Element Forces/Stresses** nos mostrará el diagrama de elementos mecánicos, como puede verse en la figura 6.1, se encuentran disponibles algunas otras opciones para despliegue de resultados.

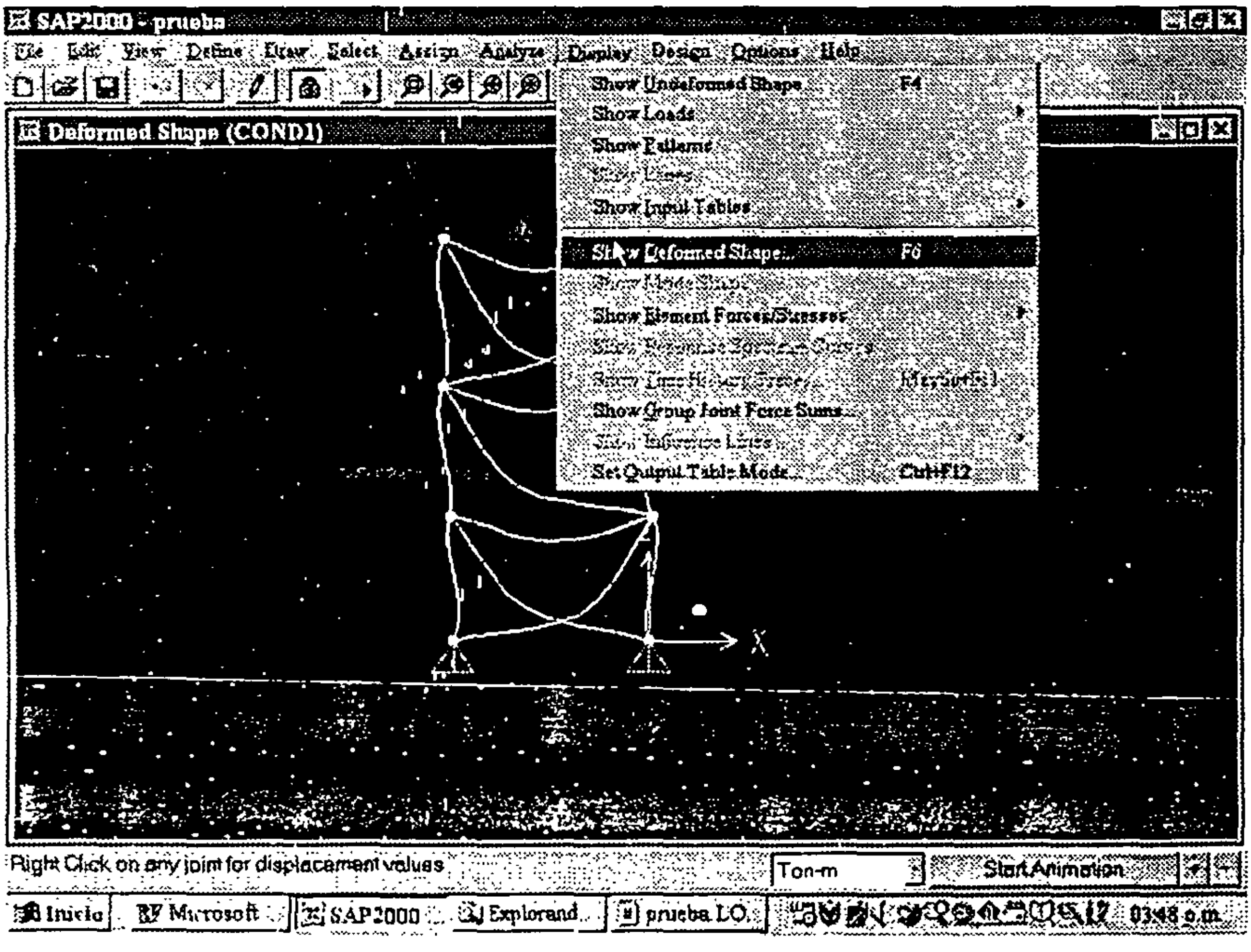


Figura 6.1 Opciones en el menú **Display**

6.2 VER LA ESTRUCTURA DEFORMADA

Para ello como se indicó en el párrafo anterior se selecciona la opción **Show Deformed Shape** del menú **Display** mostrándose la ventana de la figura 6.2, en esa ventana se selecciona del marco **Load** (parte superior de la ventana) la condición de carga de la cual se quiere ver la estructura deformada; en el marco **Scaling** se presentan dos opciones para la escala que se utilizará al desplegar la configuración deformada en caso de que se seleccione **Scale Factor** se presentará el factor de escala que se utilizará para tal fin, este factor mostrado en la caja en blanco puede ser modificado por el usuario, otras dos opciones se encuentran en el extremo inferior izquierdo de esa ventana, la primera de ellas es decir **Wire Shadow** mostrará además de la configuración deformada la no deformada, la última opción que es **Cubic Curve**, en caso de estar activada mostrará la configuración deformada ajustando una curva a esa configuración, en caso contrario sólo se dibujará la configuración deformada con lineas rectas.

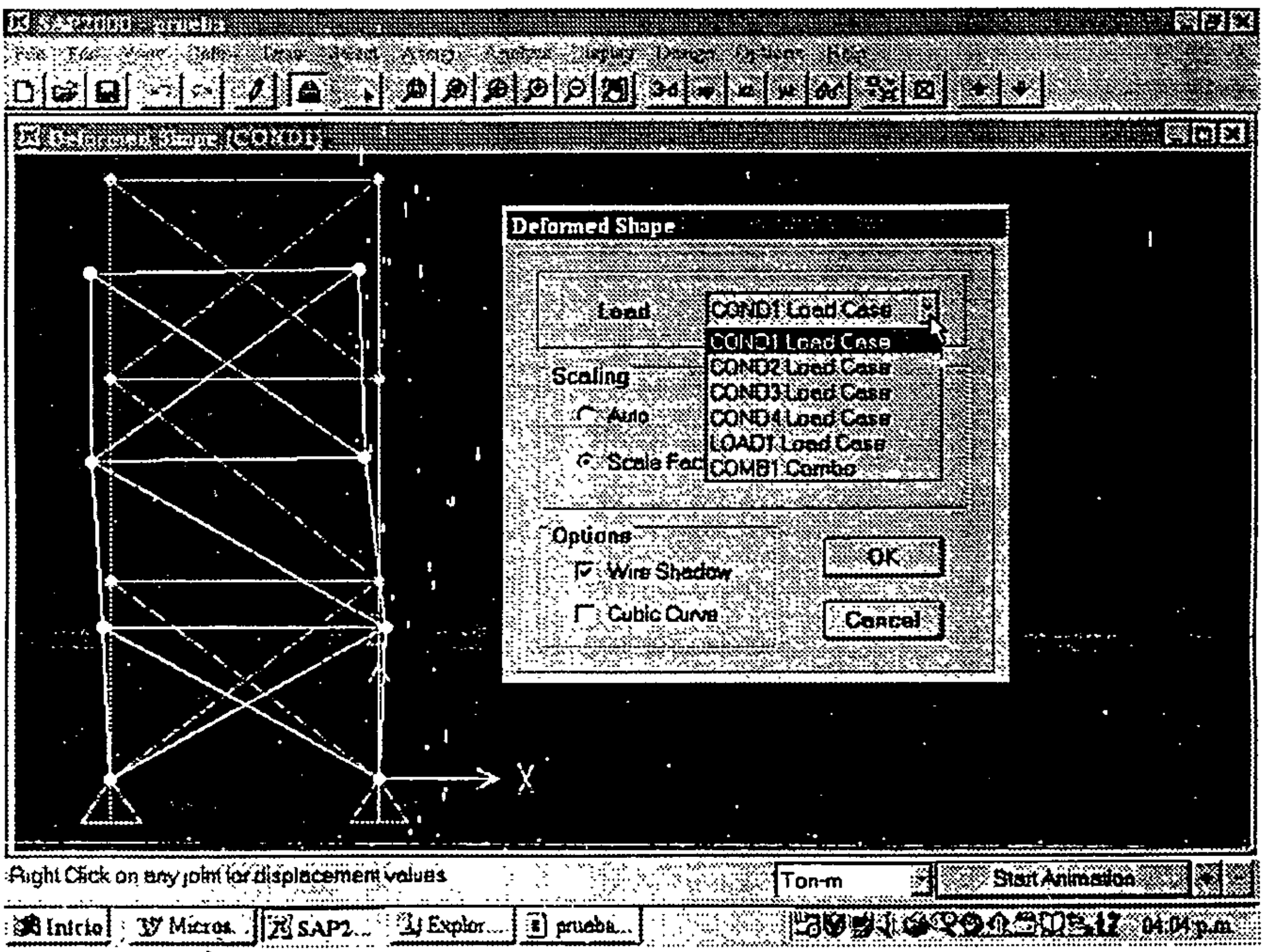


Figura 6.2 Selección de parámetros para despliegue de la configuración deformada.

Una vez mostrada la configuración deformada de la estructura se puede seleccionar algún nudo (p. ej. haciendo clic izquierdo) y después hacer clic derecho en el nudo seleccionado con lo cual se presentará una ventana conteniendo el valor de los desplazamientos de ese nudo (ver figura 6.3)

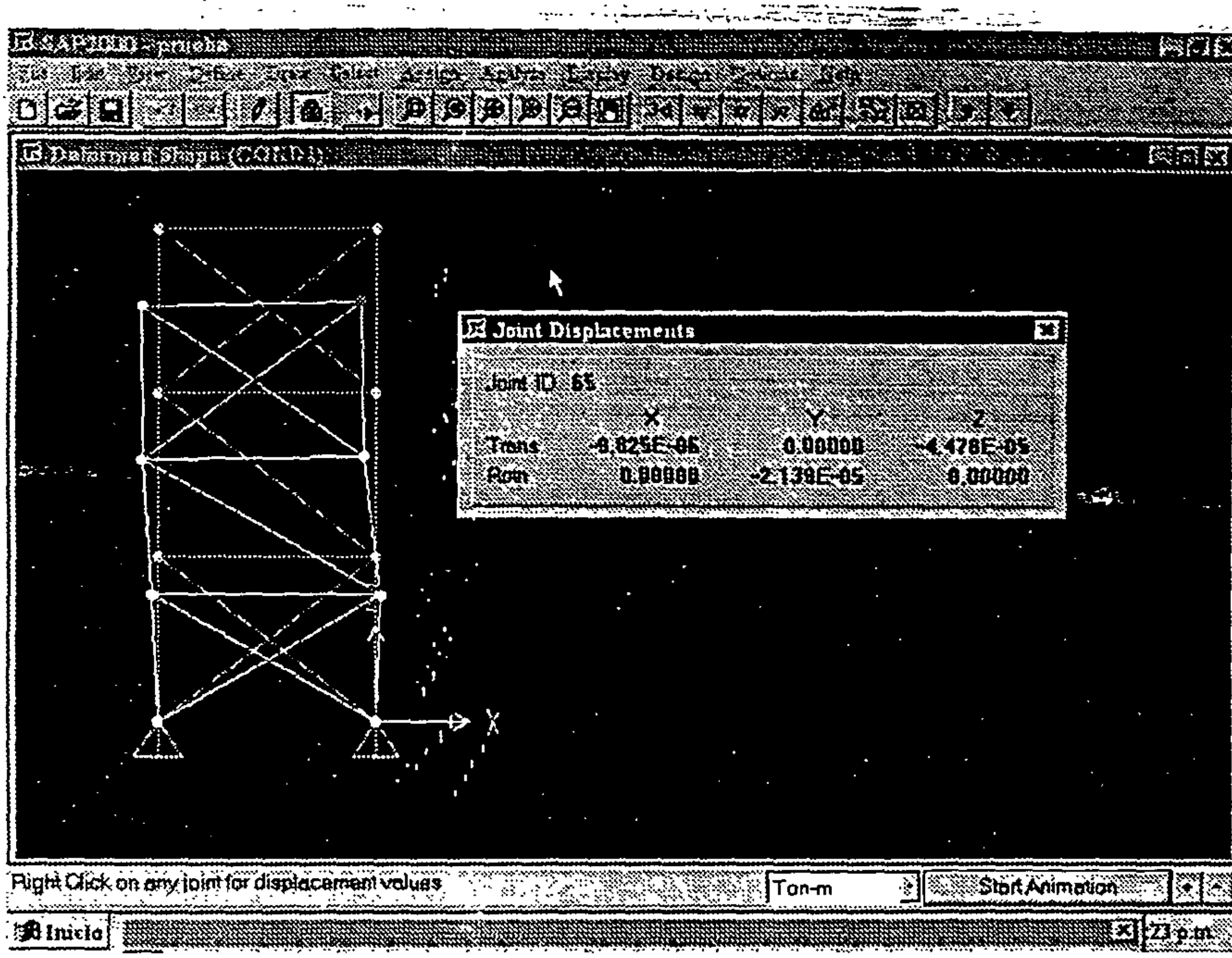


Figura 6.3 Valores del desplazamiento de un nudo seleccionado.

6.3 VER LOS DIAGRAMAS DE ELEMENTOS MECANICOS

Como se ha mencionado, **SAP2000** permite mostrar gráficamente los valores de algún elemento mecánico para determinada condición de carga, para ello se selecciona **Frames** de la opción **Show Element Forces/Stresses** en el menú **Display** presentando la ventana de la figura 6.4.

En el marco **Load** se selecciona la condición de carga y en **Component** se selecciona el tipo de elemento mecánico, las opciones en el marco **Scaling** producen el mismo efecto al caso de la configuración deformada, las opciones que se encuentran al final de la ventana nos permiten seleccionar si se desea un diagrama "lleno" y sin despliegue de valores del elemento mecánico o con valores en el diagrama

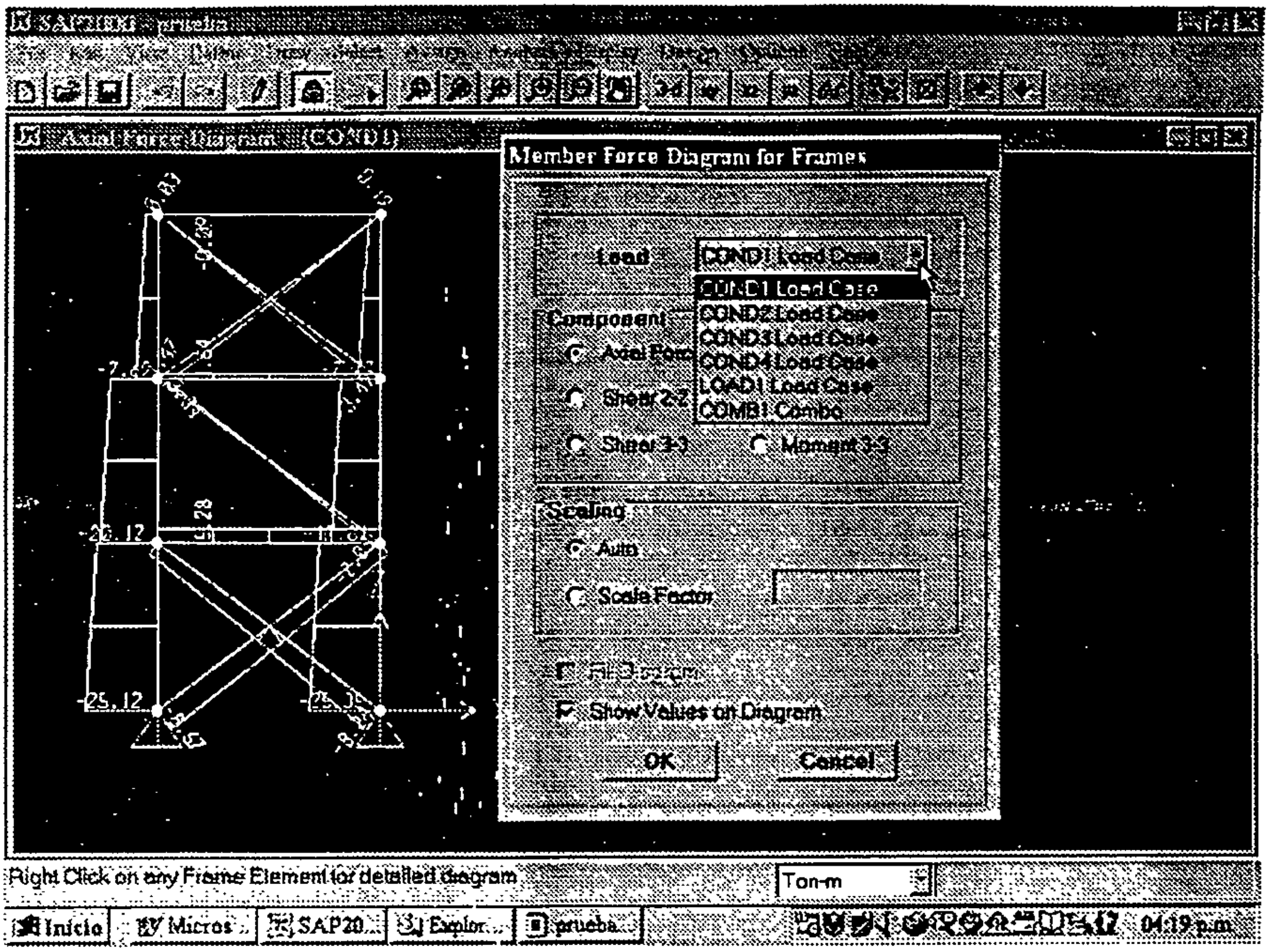


Figura 6.4 Selección de parámetros para despliegue de diagramas de elementos mecánicos.

Una vez mostrado el diagrama se puede seleccionar algún elemento barra haciendo clic sobre él y después de hacer clic derecho sobre el mismo se presenta una ventana mostrando el diagrama del elemento seleccionado, así como el valor del elemento mecánico en una sección transversal situada a la distancia que se muestra en el cuadro en blanco (ver figura 6.5), desplazando el puntero del ratón a lo largo del eje del elemento dentro de la ventana desplegada se muestra tanto la posición de la sección transversal como el valor respectivo del elemento mecánico, el contenido del cuadro puede ser modificado por el usuario desplegándose instantáneamente el valor del elemento mecánico que corresponda a la sección cuya posición se especificó en el cuadro en blanco.

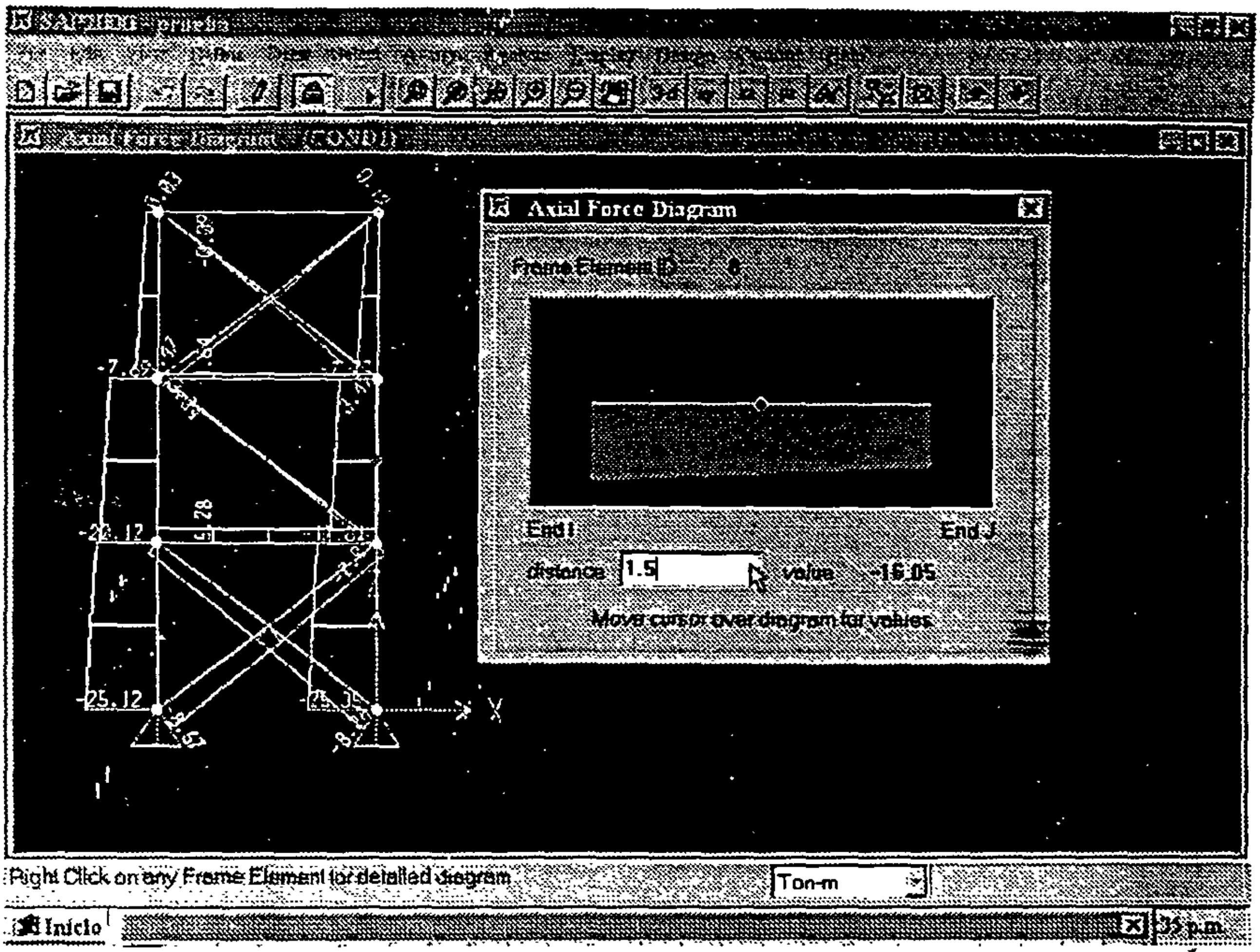


Figura 6.5 Diagrama de un elemento mecánico de una barra seleccionada.

6.4 VER LOS RESULTADOS DE DISEÑO

Algunas opciones de diseño se encuentran disponibles en el menú **Design** una vez realizado el Análisis se pueden tener acceso a ellas.

Como primer paso se seleccionará el tipo de diseño y características a utilizar, por ejemplo en el caso de diseño de concreto se tendrán que especificar algunas características de refuerzo lo cual se deberá de hacer en **Modify/Show Section** de la opción **Define Frame Sections** en el menú **Define**, seleccionando en la ventana que se despliega el botón **Reinforcement** para enseguida especificar el tipo de elemento (columna o viga), la configuración del refuerzo y las características de éste (ver figura 6.6)

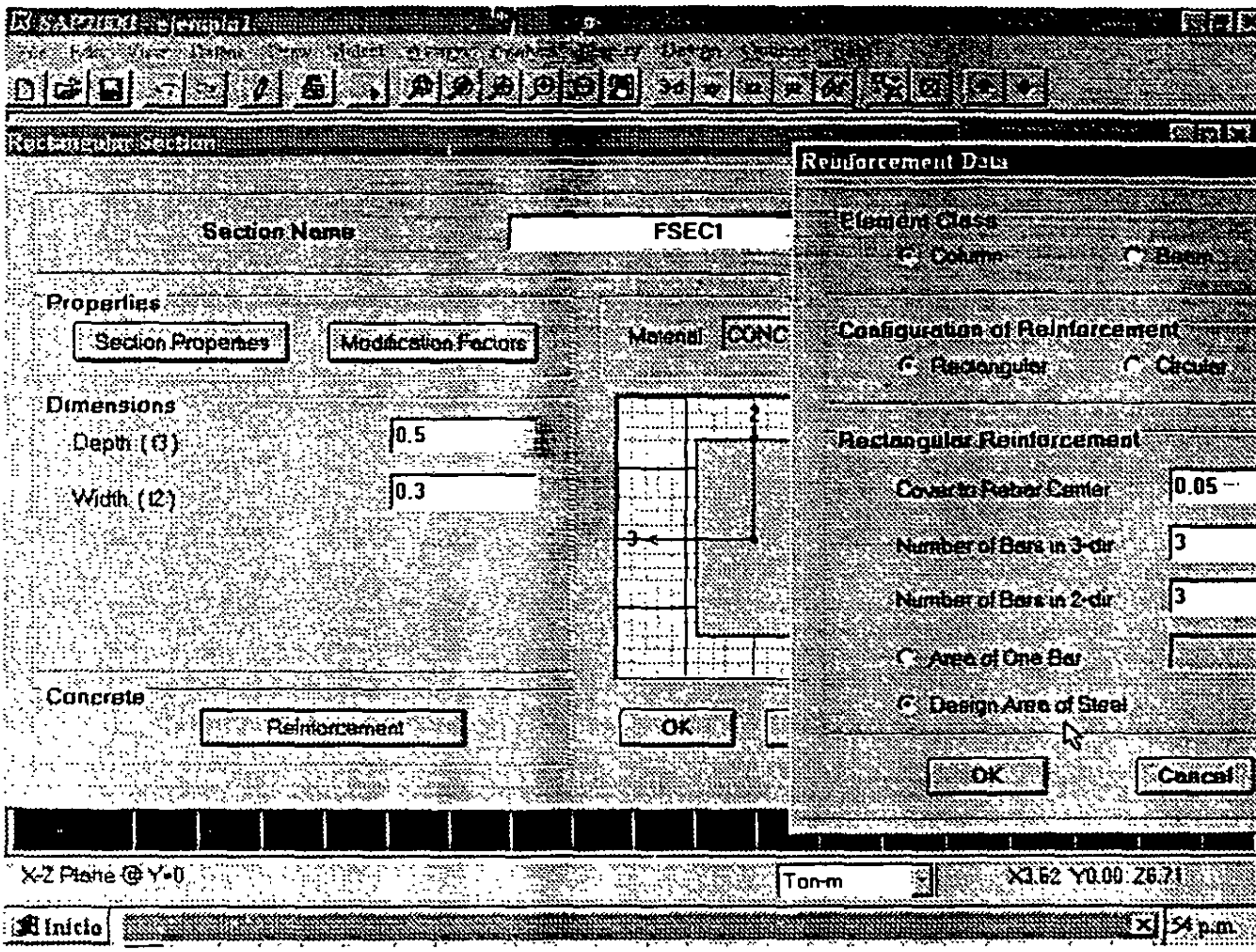


Figura 6.6 Caracteristicas para diseño de un elemento.

Como segundo paso se deberá especificar las combinaciones de carga que se utilizarán para verificar el diseño activando la que se quiera para ser usada en el diseño, esto se puede hacer en **Add** o **Modify/Show Combo** en la opción **Load Combinations** del menú **Define** (ver figura 6.7).

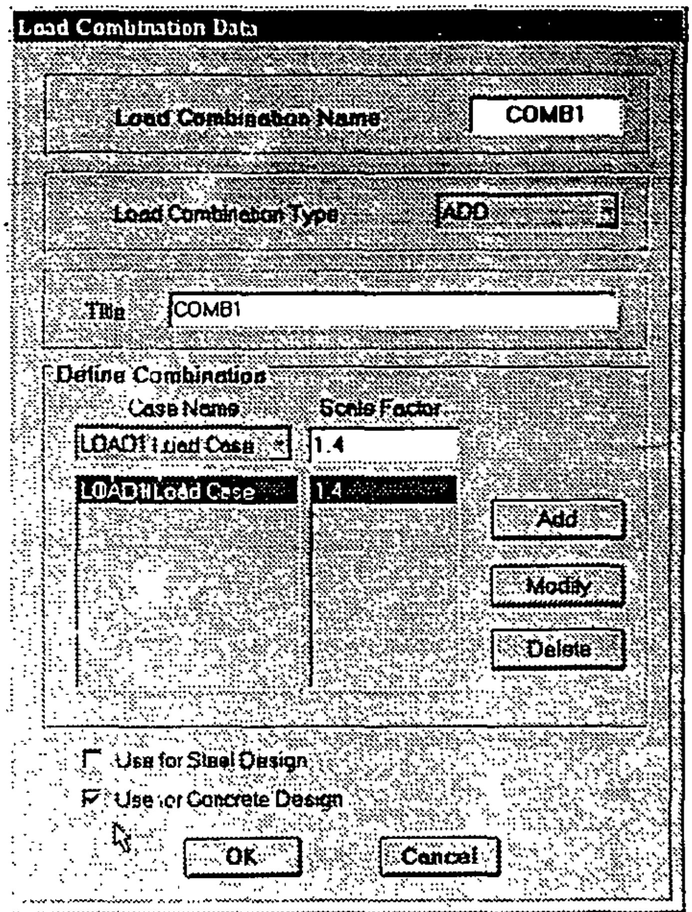


Figura 6.7 Especificación de combinaciones de carga para diseño.

Una vez realizado el Análisis, como tercer paso se seleccionarán las combinaciones de diseño para ello utiliza la opción **Select Design Combos** del menú **Design** (ver figura 6.8).

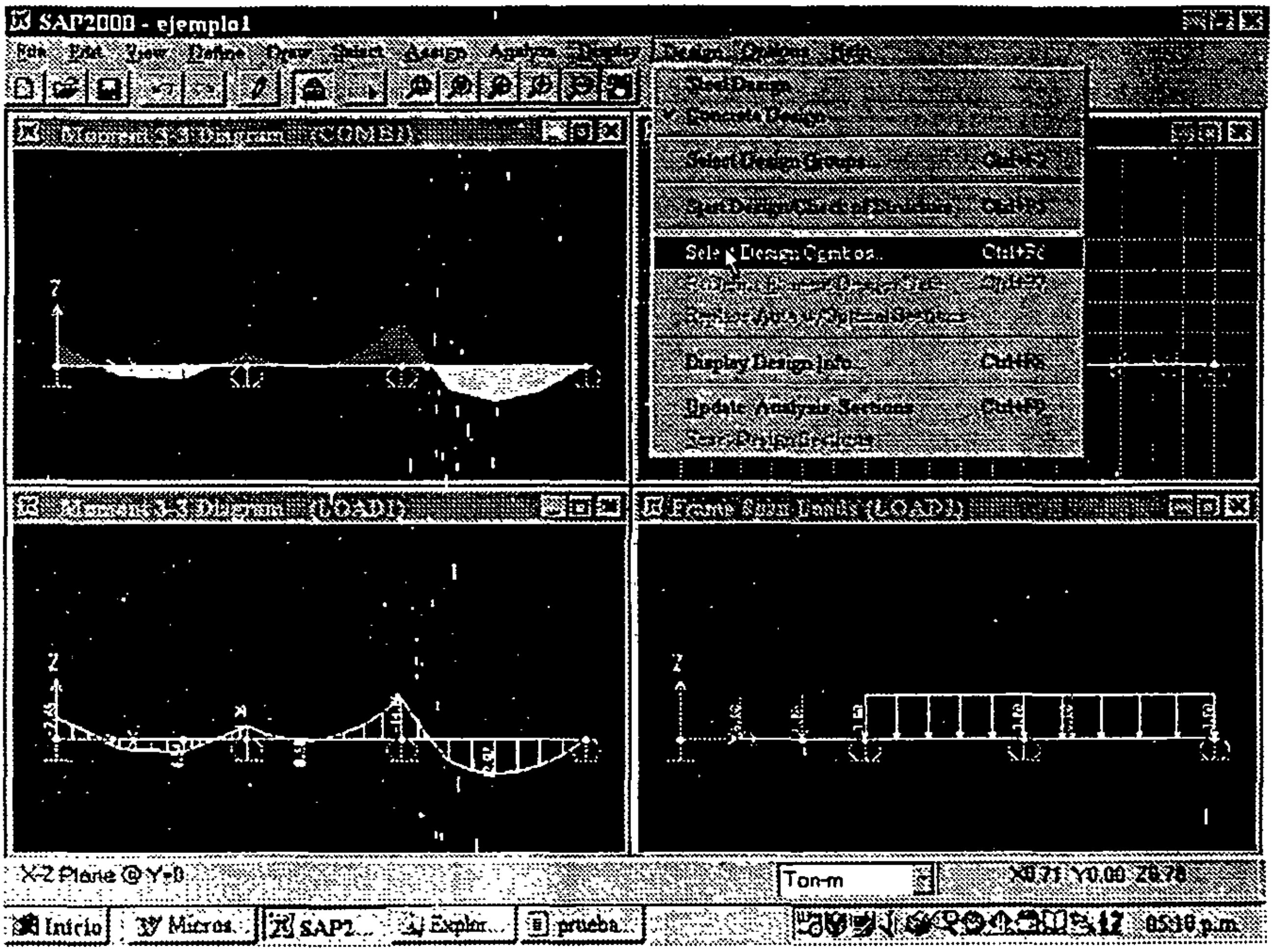


Figura 6.8 Algunas opciones del menú **Design**.

Como cuarto paso se seleccionará la opción **Start Design...** del menú **Design** (ver figura 6.8) con lo que se desplegarán algunos resultados del diseño, seleccionado una barra y después de hacer clic derecho sobre la misma se muestra una ventana similar a la de la figura 6.9.

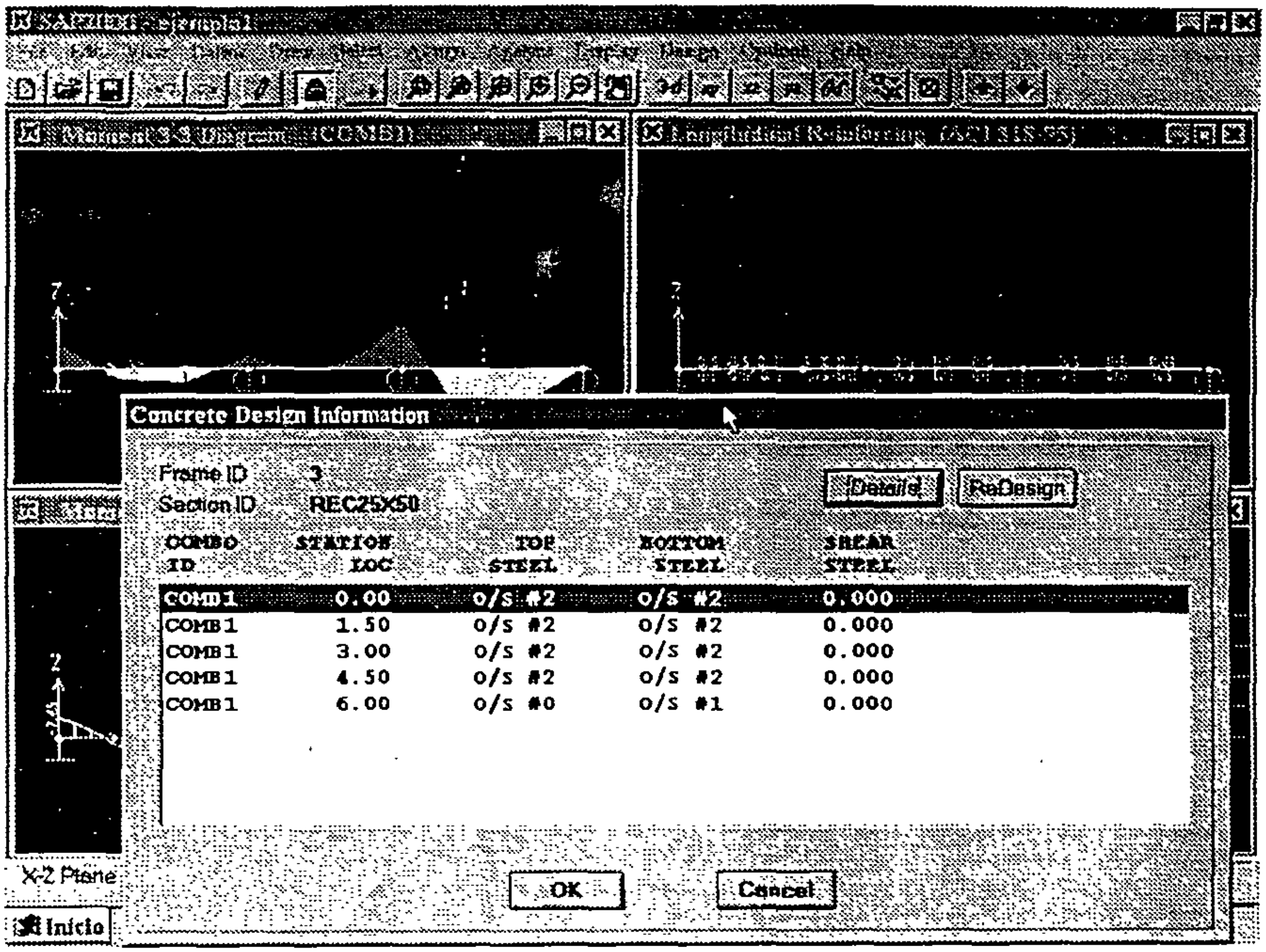


Figura 6.9 Resultados de diseño de un elemento seleccionado.

De ella se puede seleccionar el botón **Details** mostrándose información más detallada acerca de las características de diseño del elemento, se puede mostrar información diversa de la ventana arrastrando el mouse (botón izquierdo hacia alguna zona especifica de la ventana, ver figura 6.10)

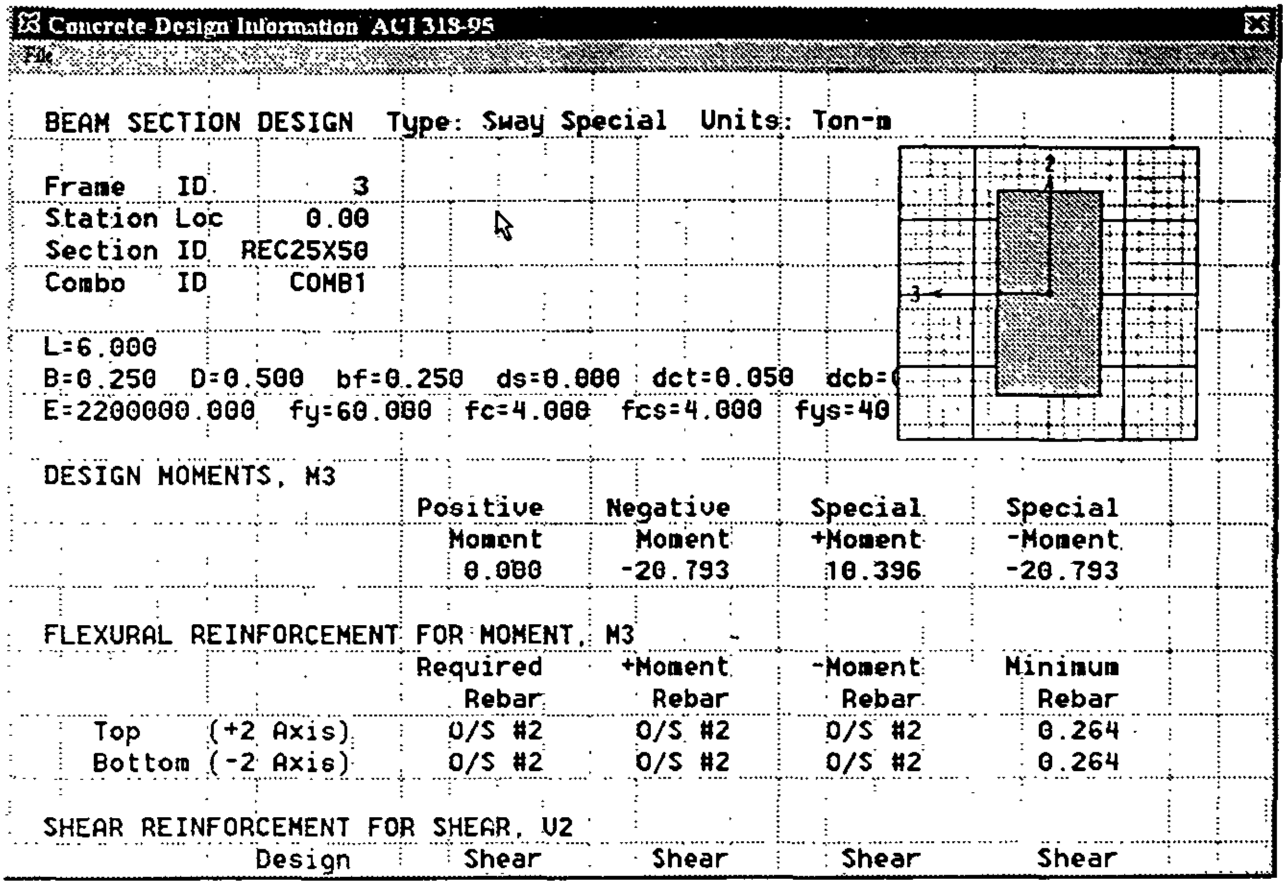


Figura 6.10 Detalle de los resultados de diseño de un elemento seleccionado.

6.4 OTRAS CARACTERISTICAS

El despliegue de reacciones puede ser seleccionado mediante **Joints** de la opción **Show Element...** del menú **Display** mostrándose la ventana de la figura 6.11, en donde se podrá seleccionar la condición de carga, después de hacer clic en **OK** se muestran las reacciones correspondientes a la condición de carga seleccionada (ver figura 6.11).

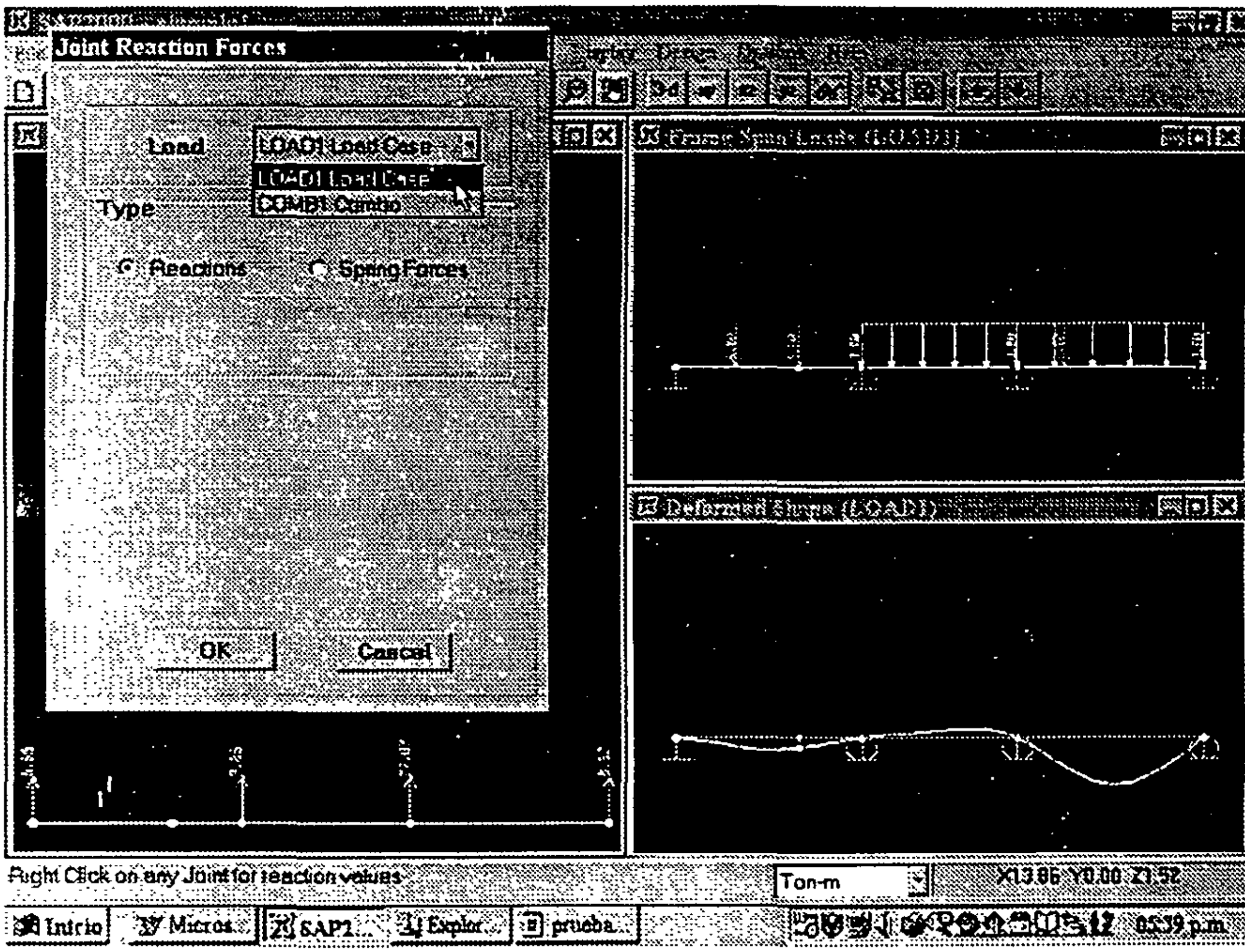


Figura 6.11 Ventana para la selección de reacciones.

Están disponibles en el menú **Display** algunas otras características relacionadas con el Análisis Dinámico como el dibujo de formas modales, espectros de respuesta y análisis de la respuesta en el tiempo y otras más.

Los resultados del Análisis se pueden almacenar en un archivo a manera de tablas para ello se selecciona la opción **Set Output Table Mode** del menú **Display** mostrándose una ventana en donde se seleccionarán las condiciones de carga de los resultados que se incluirán, después de hacer clic en el botón **OK** se pasa a una ventana con de título **Analysis Output Tables** (ver figura 6.12)

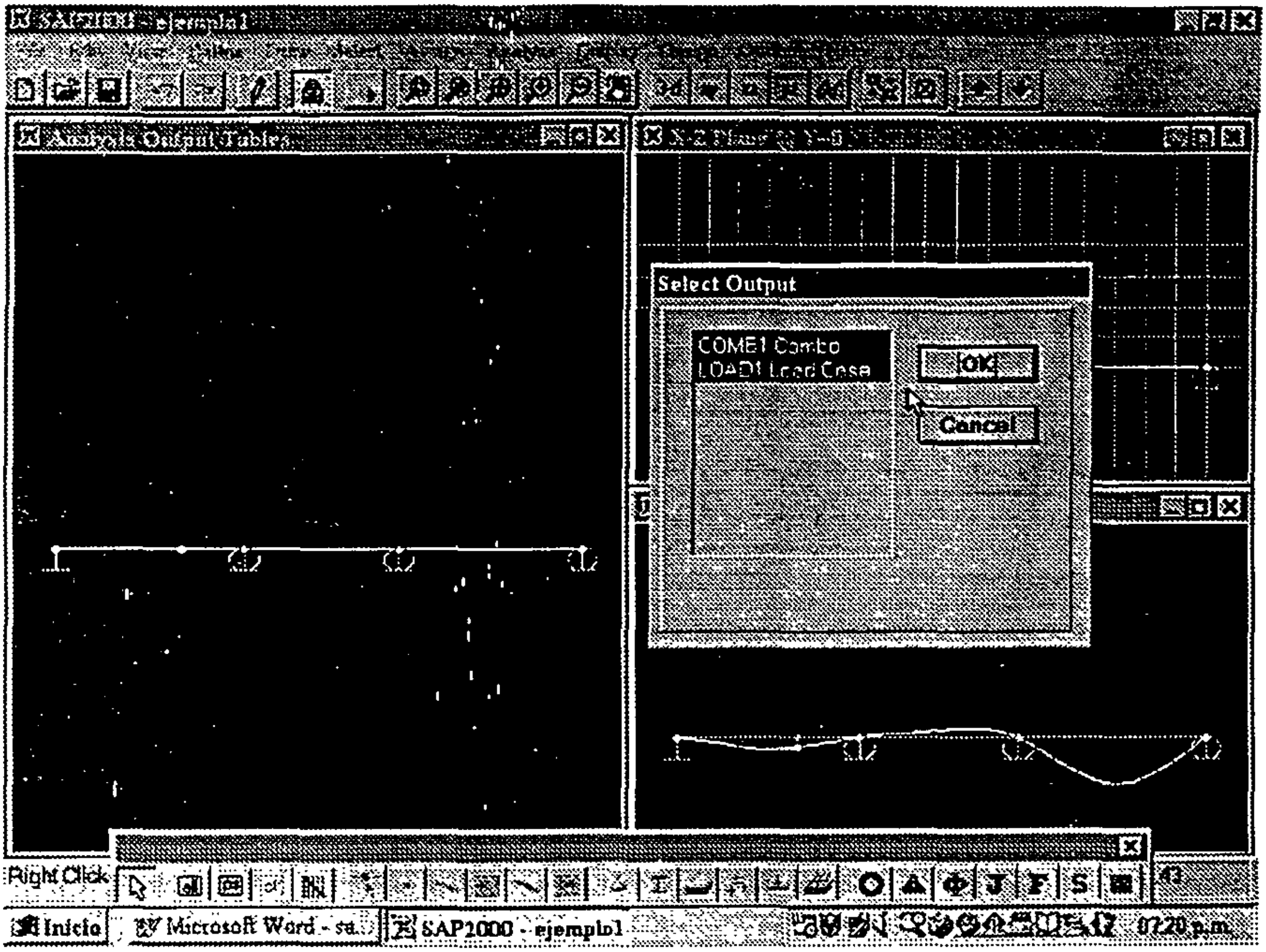


Figura 6.12 Selección de condiciones y para la generación de resultados en forma tabular.

En esa ventana se podrá seleccionar algún nudo o elemento después de hacer clic derecho en un nudo seleccionado, se desplegará una ventana conteniendo tanto los desplazamientos como las reacciones del nudo para las condiciones de carga seleccionadas (ver figura 6.13).

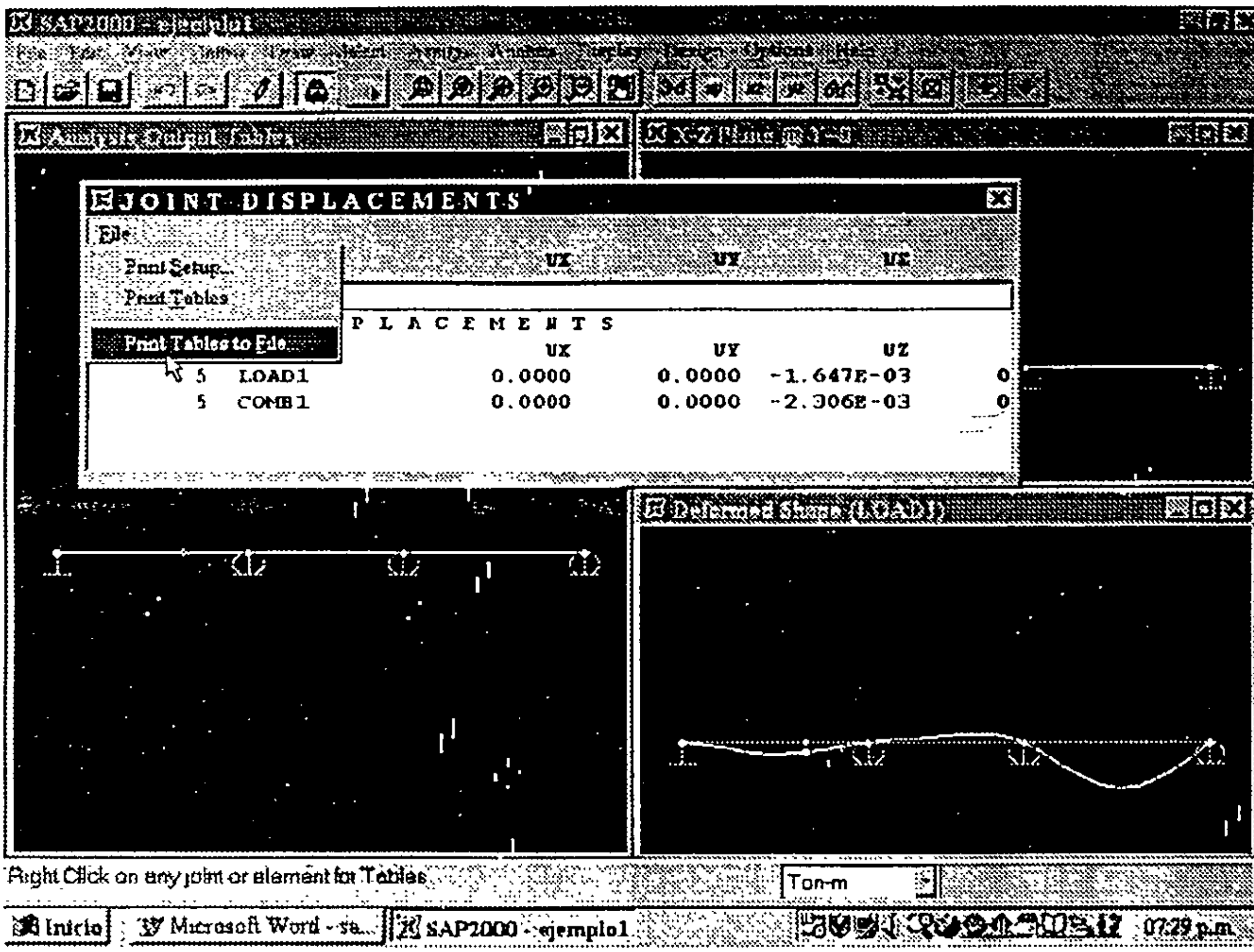


Figura 6.13 Ventana de resultados de un nudo seleccionado.

Si el elemento sobre el que se hace clic es una barra entonces la ventana que se despliega contiene los elementos mecánicos de las condiciones de carga seleccionadas (ver figura 6.14).

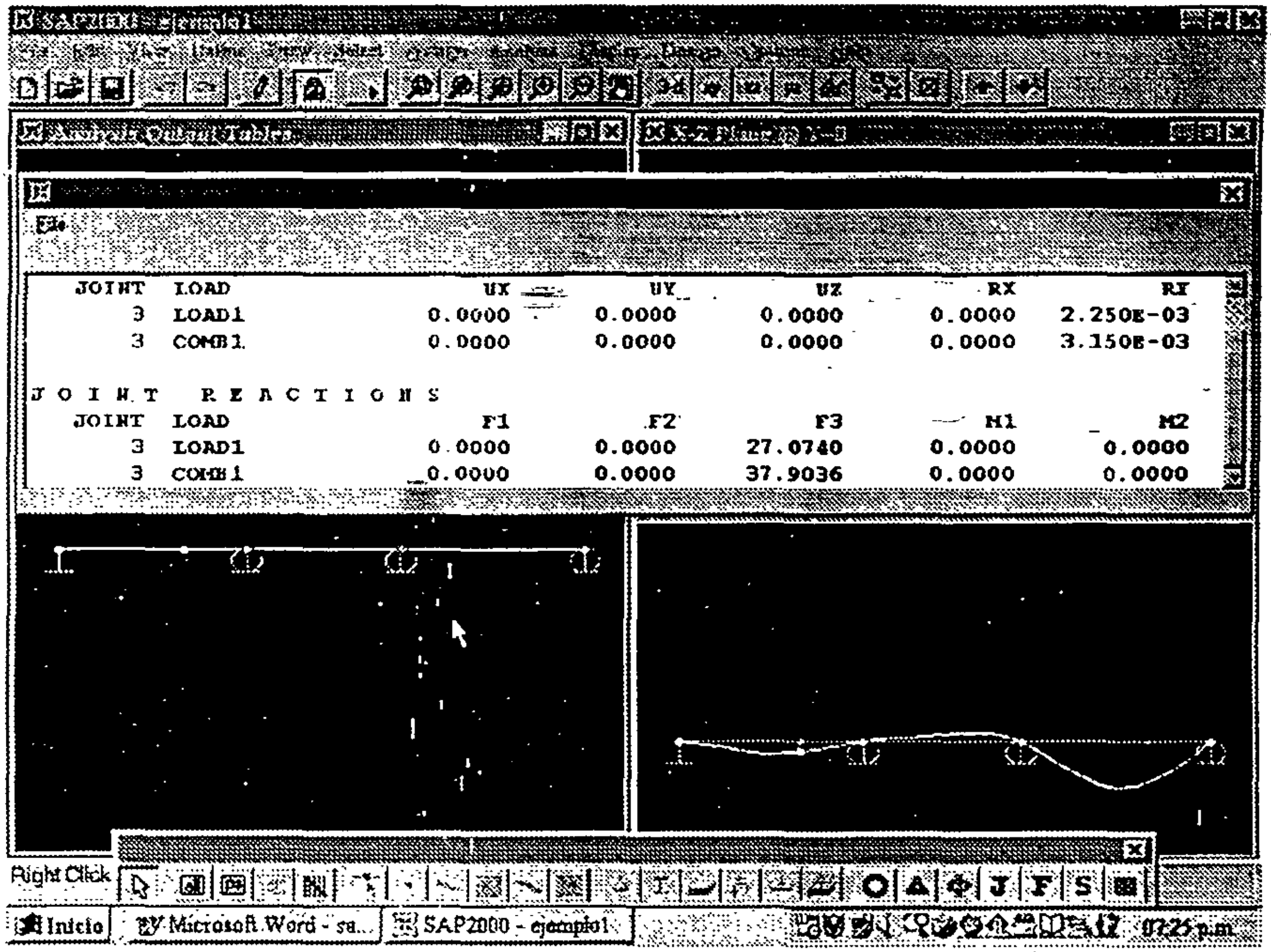


Figura 6.14 Ventana de resultados de una barra seleccionada

Tanto en la ventana de resultados de nudos como de barras en el extremo superior izquierdo de esta se encuentra la opción **File**, que permite el almacenamiento de los resultados contenidos en la tabla mostrada en un archivo, para ello después de hace clic en **File**, habrá que proporcionar en el cuadro en blanco el nombre del archivo y hacer clic en el botón guardar (ver figura 6.15).

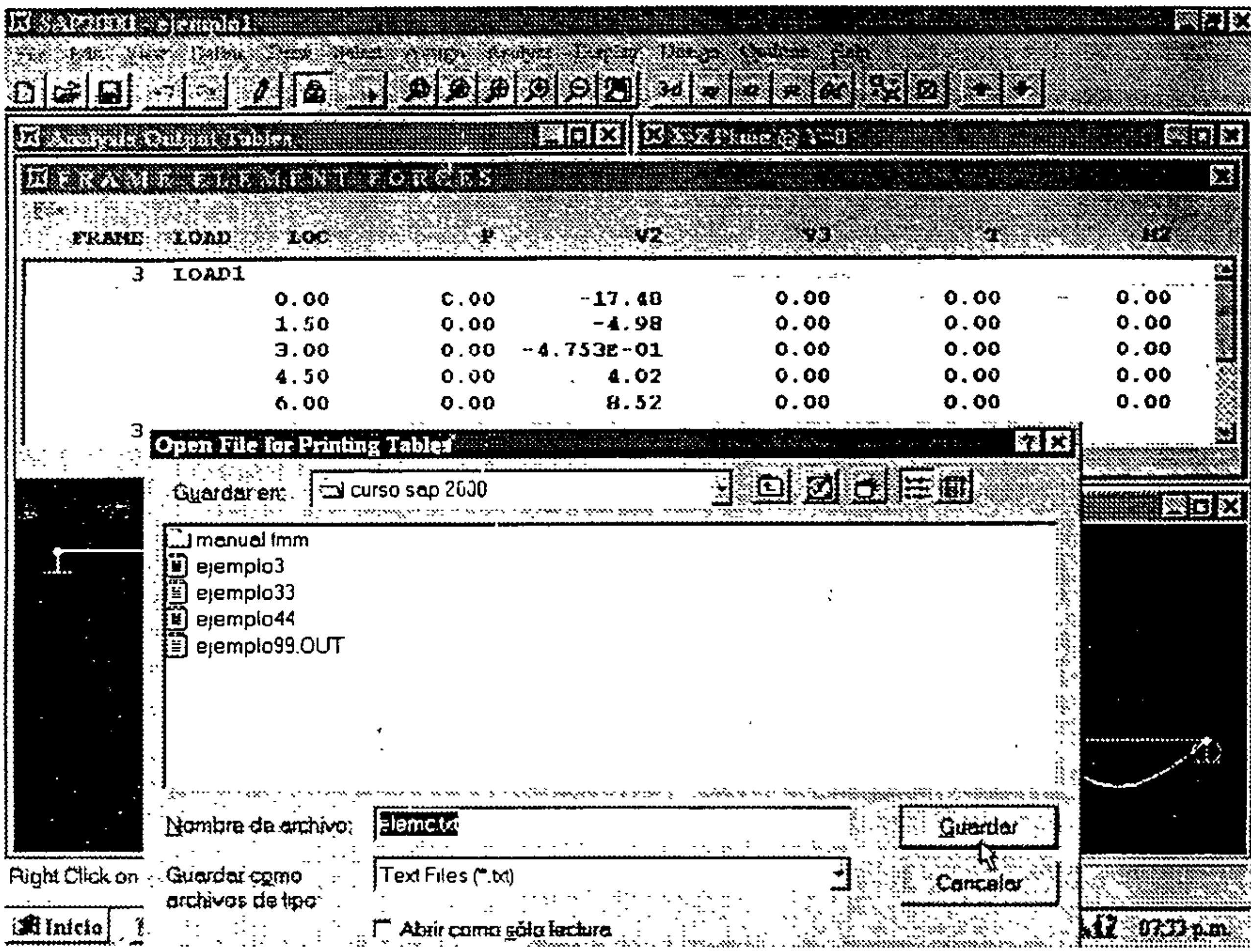


Figura 6.15 Almacenamiento de resultados en un archivo.

OPCIONES ADICIONALES

CAPÍTULO 7

7.1 INTRODUCCION

SAP2000 posee varias caracteristicas, con algunas de ellas se pueden modelar por ejemplo muros y losas mediante elementos **Shell**, las opciones abarcan desde la definición de materiales dibujo de elementos, definición de caracteristicas geométricas del elemento (**Shell Sections**) así como la asignación de las caracteristicas anteriores además de las fuerzas (uniformes, presión, etc.) a este tipo de elementos. La estructura y secuencia es muy parecida a la utilizada para los elementos barra, se recomienda consultar la ayuda en linea, los temas relacionados en los manuales o bien ver los ejemplos en la carpeta de ejemplos o en el disco.

En cuanto al elemento finito sólido; este no se puede generar utilizando el editor gráfico de la versión estudiantil (versión 6.1 ó 6 13), por lo que su definición y demás caracteristicas se tendrán que realizar mediante una serie de instrucciones que se adicionarán al archivo de datos mediante un editor, la misma recomendación hecha en el párrafo anterior es aplicable a este caso.

Una gran variedad de opciones para Análisis Dinámico está incluida en el programa **SAP2000**, para usar alguna de ellas se recomienda consultar los ejemplos que acompañan al presente instructivo o que se encuentran en el diskete, o bien los que se encuentran en el manual respectivo

También existe la posibilidad de Análisis de estructuras de puentes obteniendo mediante el programa por ejemplo: lineas de influencia, envolventes de elementos mecánicos, etc., lo anterior para varias condiciones de carga incluyendo uno o varios carriles con cargas vehiculares tipo o definidas por el usuario, la recomendación del párrafo anterior es igualmente aplicable.

Se recomienda consultar al autor ya que se encuentra en proceso un instructivo similar al presente para los fines mencionados en los párrafos anteriores.

7.2 VER EL ARCHIVO DE ENTRADA

Durante una sesión con el programa **SAP2000** las opciones **Save** y **Save as** del menú **File** permiten almacenar en un archivo con extensión **SDB** los datos de la estructura que se han introducido, al archivo así creado sólo se podrá acceder (para fines de este programa) mediante la opción **Open** del mismo menú, sin embargo los datos pueden ser almacenados en un archivo que pueda modificarse y ser reconocido por el programa **SAP2000** para ello se selecciona **SAP2000.S2K** de la opción **Export** en el menú **File** (ver figura 7.1).

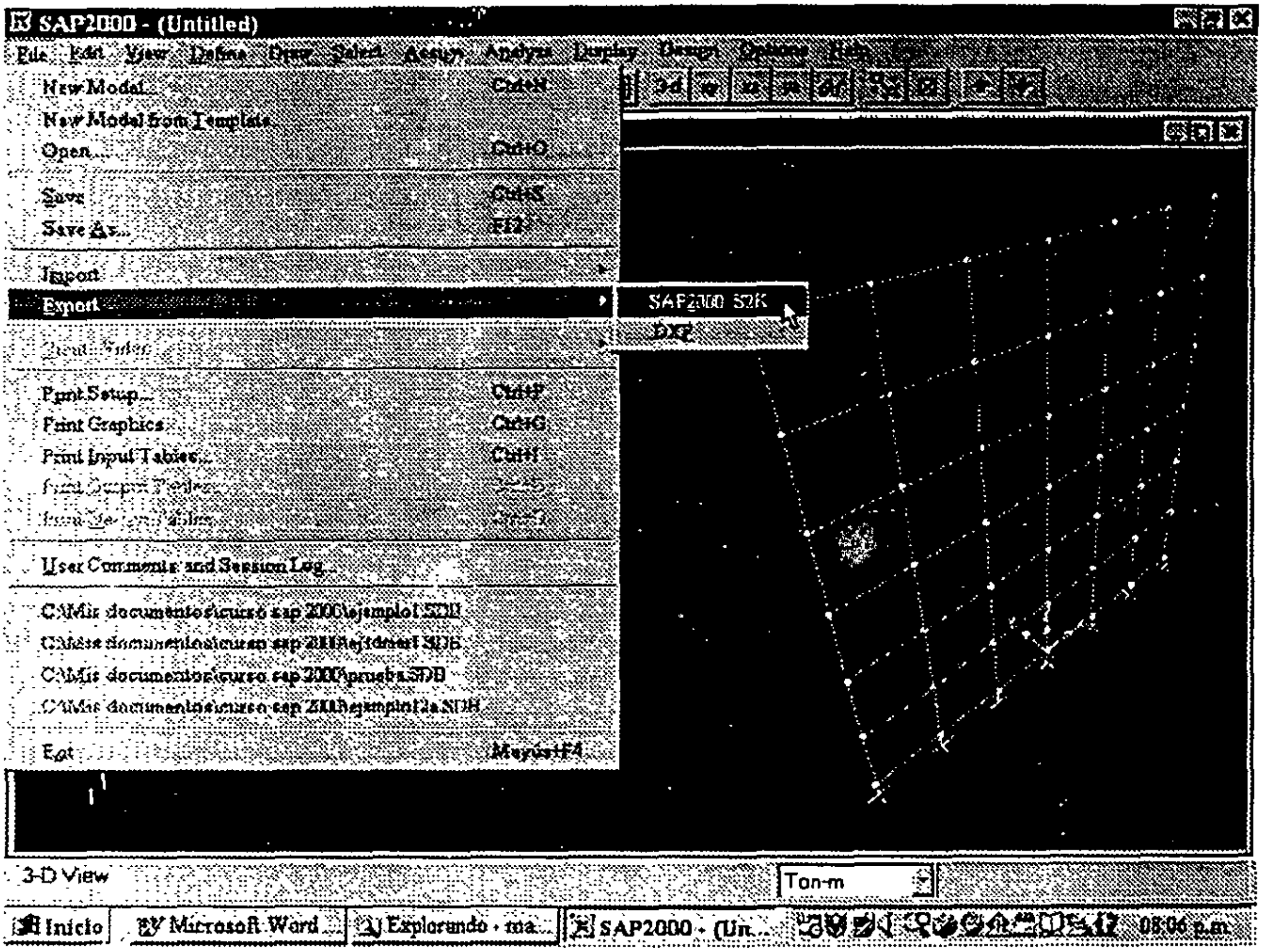


Figura 7.1 Almacenamiento de datos para poder realizar cambios al archivo.

El archivo extensión S2K puede ser modificado con la ayuda de algún editor (por ejemplo Edit, WordPad, etc.), el archivo resultante de la modificación deberá ser almacenado sin ningún caracter de control especial que se almacene en el mismo archivo, y con la misma extensión, si se usan algunos editores o procesadores de palabras se deberá tener especial cuidado de lo anterior, en caso de uso de esos procesadores se recomienda hacer varias copias de respaldo con objeto de no perder los cambios efectuados.

Una vez realizados los cambios, el contenido del archivo extensión S2K modificado podrá ser procesado por **SAP2000**, para ello se selecciona **SAP2000.S2K** de la opción **Import** en el menú **File**, para ambas opciones (**Export** e **Import**) será necesario proporcionar el nombre del archivo en el cuadro en blanco correspondiente de la ventana como la que se muestra en la figura 7.2.

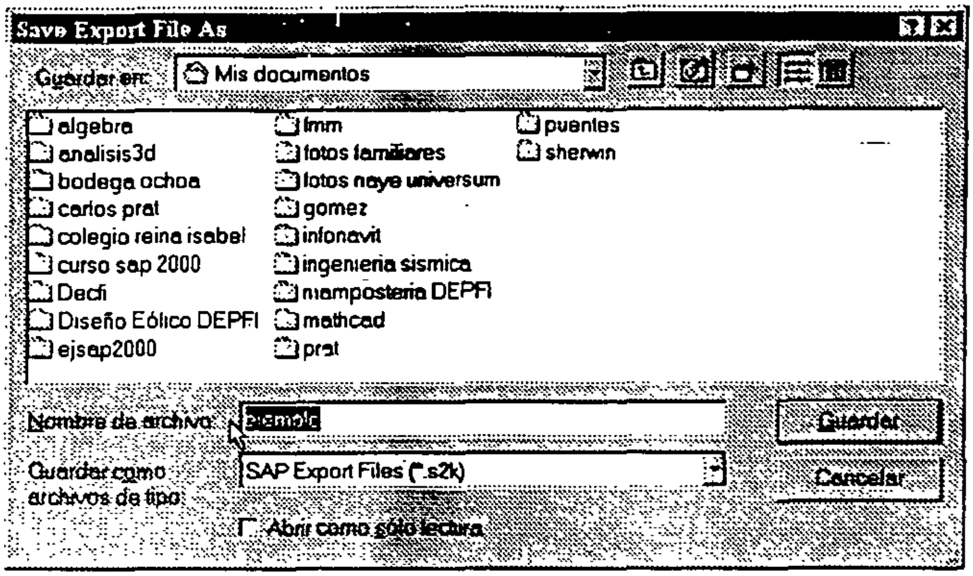


Figura 7.2 Ventana en la opción **Export** SAP2000.S2K.

7.3 VER EL ARCHIVO DE SALIDA

El contenido del archivo de resultados nombre.OUT indicado mediante **Generate Output** en la opción **Analysis Options** del menú **Analyze** se genera después de ejecutar la opción **Run** del menú del mismo nombre, el archivo así generado puede ser consultado mediante cualquier editor o procesador de palabras e inclusive por algunas hojas de cálculo, para ello se seleccionará la opción abrir (**Load** u **Open**) del programa que se vaya a utilizar con ese fin y especificar el nombre del archivo desde luego con extensión OUT (ver figura 7.3).

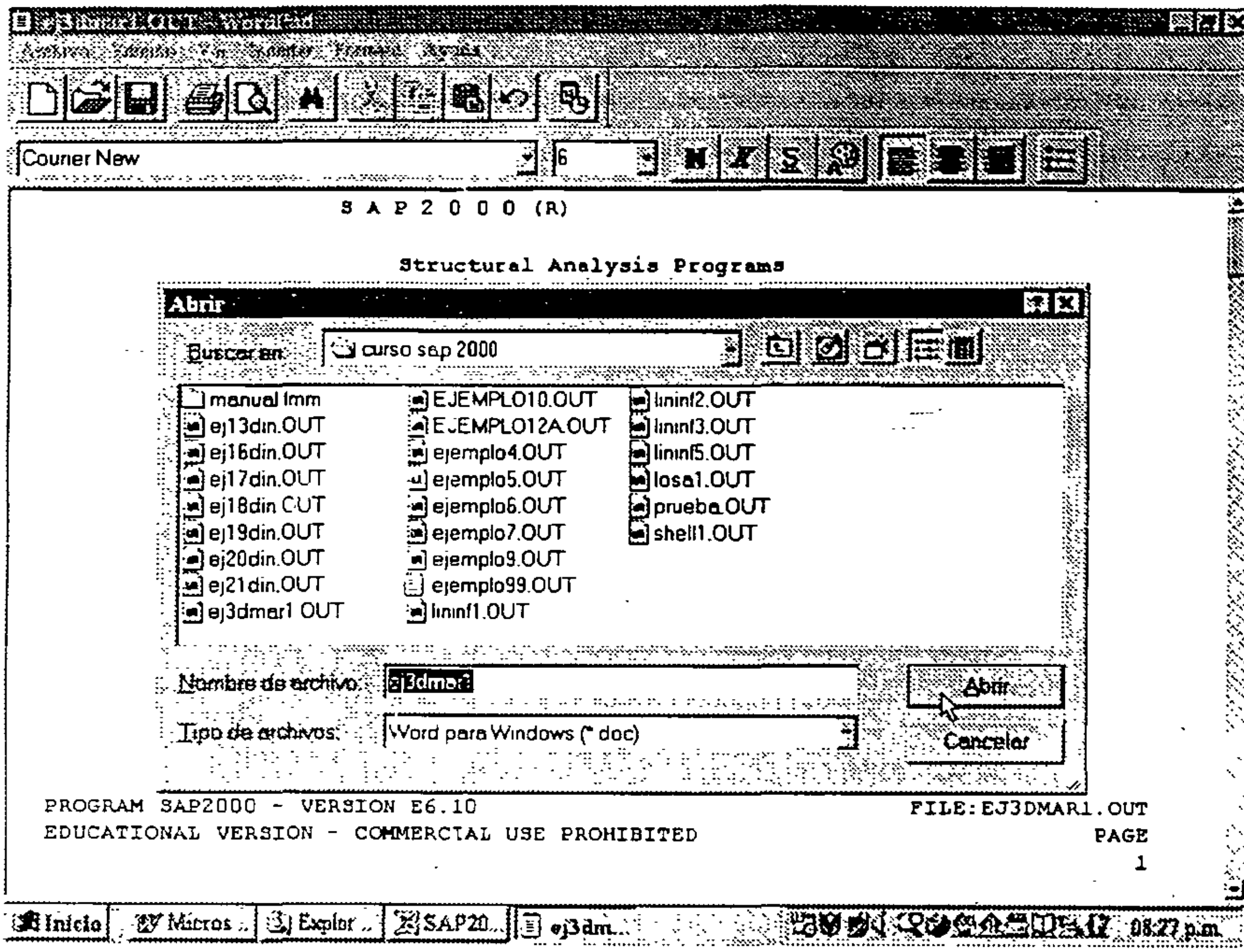


Figura 7.3 Acceso al archivo nombre.OUT mediante el programa **WordPad**.

7.4 RELACION CON AUTOCAD

La geometría de la estructura puede ser generada por AutoCAD realizando el dibujo de las barras (lineas) en una capa (**Layer**) de nombre Sap_frames (ver figura 7.4).

La geometría así generada se deberá exportar a un archivo extensión **dxf** como se muestra en la figura 7.5.

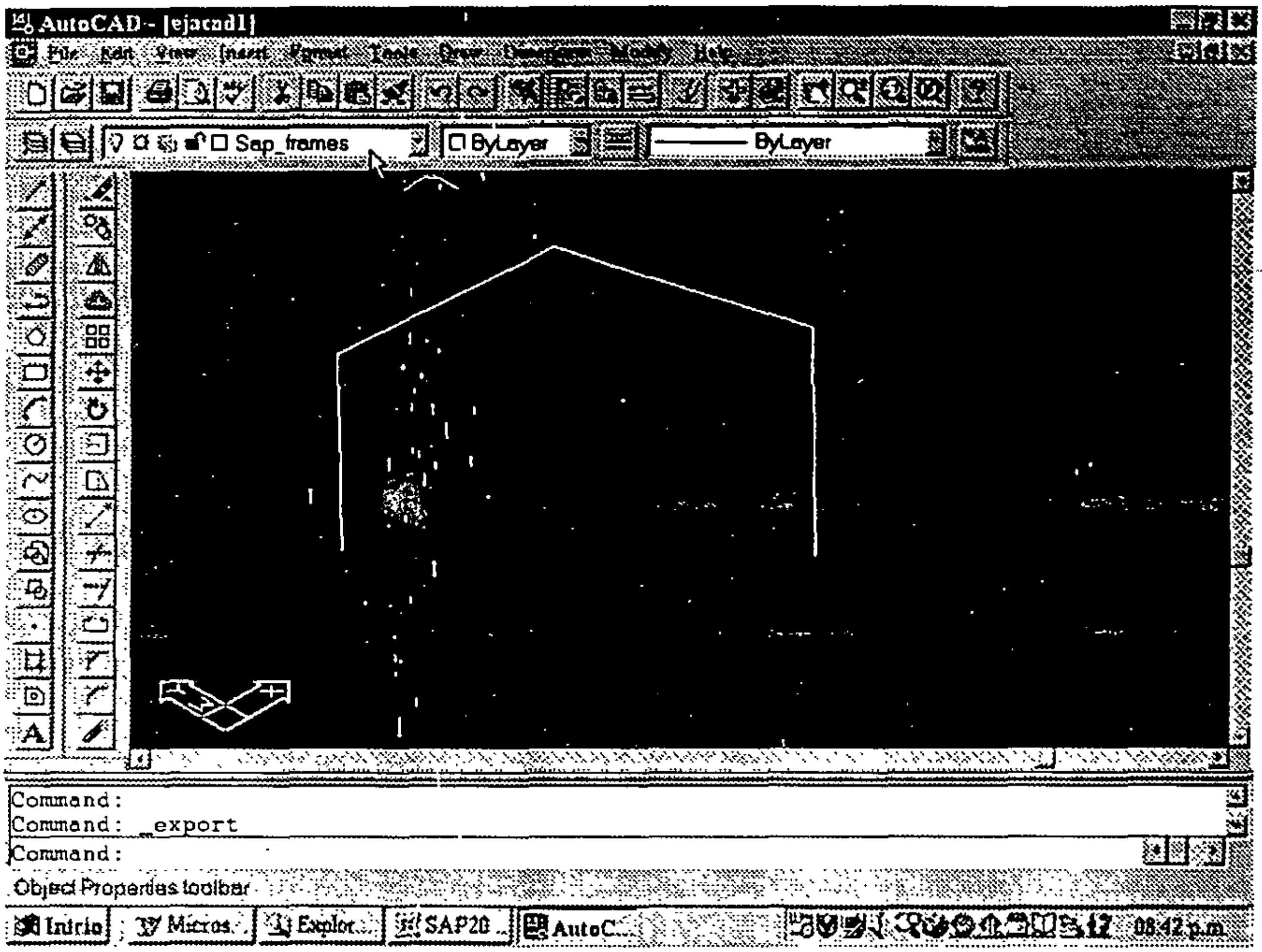


Figura 7.4 Geometría generada en AutoCAD.

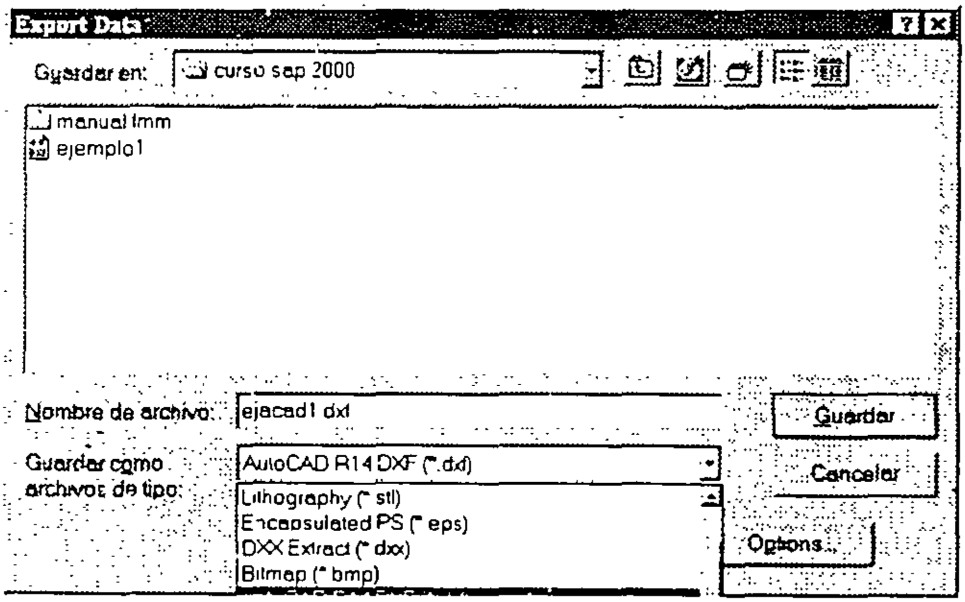


Figura 7.5 Exportando la geometría generada con AutoCAD a un archivo extensión **dxf**.

Para recuperar la información de un archivo **dxf,** se selecciona **.dxf** de la opción **Import** en el menú **File** (ver figura 7.6).

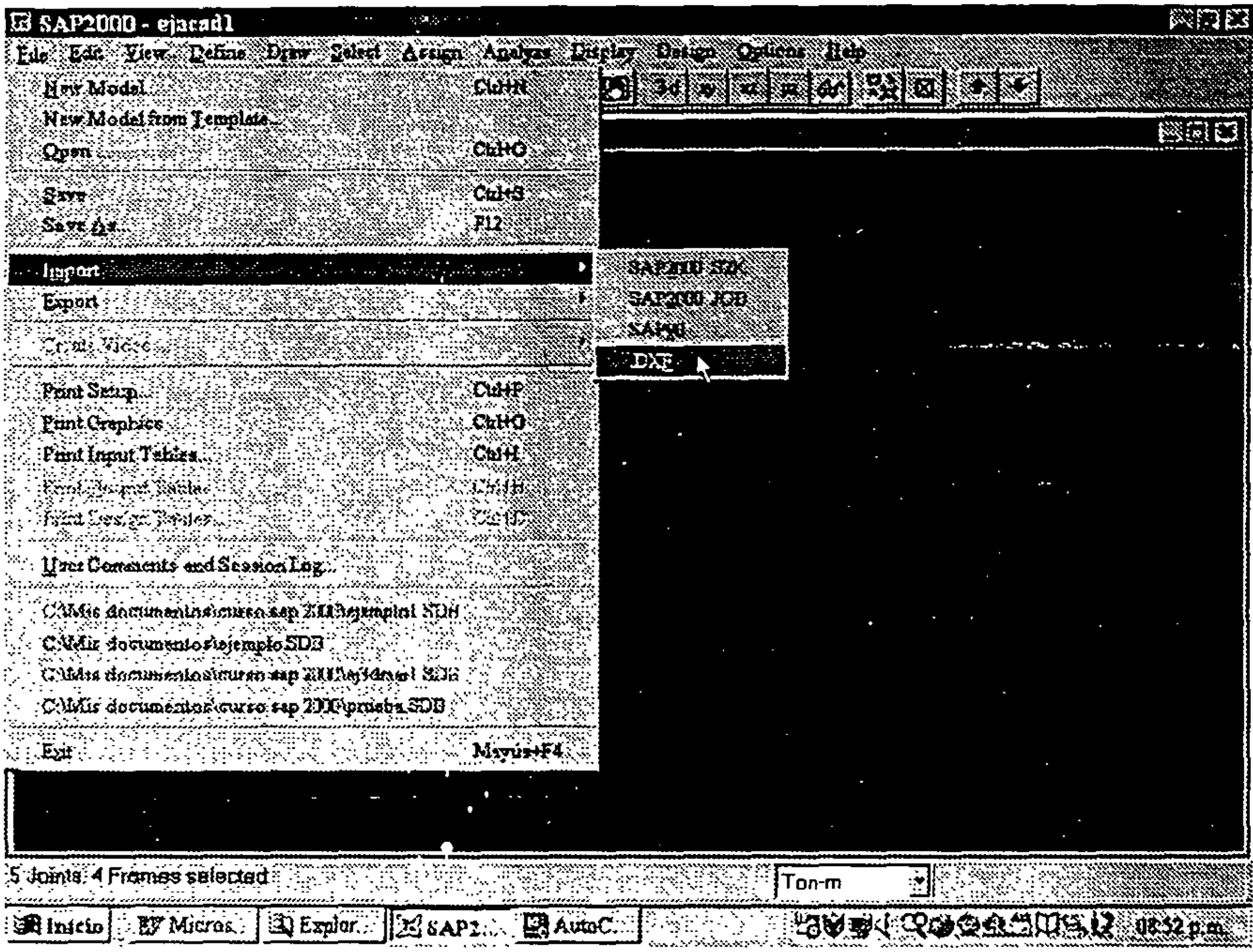


Figura 7.6 Importando datos de un archivo **.dxf**.

Desplegándose enseguida la ventana que se muestra en la figura 7.7, en donde se deberá especificar el nombre del archivo cuya extensión es **dxf** después de hacer clic en abrir se seleccionan de la ventana que se muestra en la figura 7.8 la dirección global y las unidades

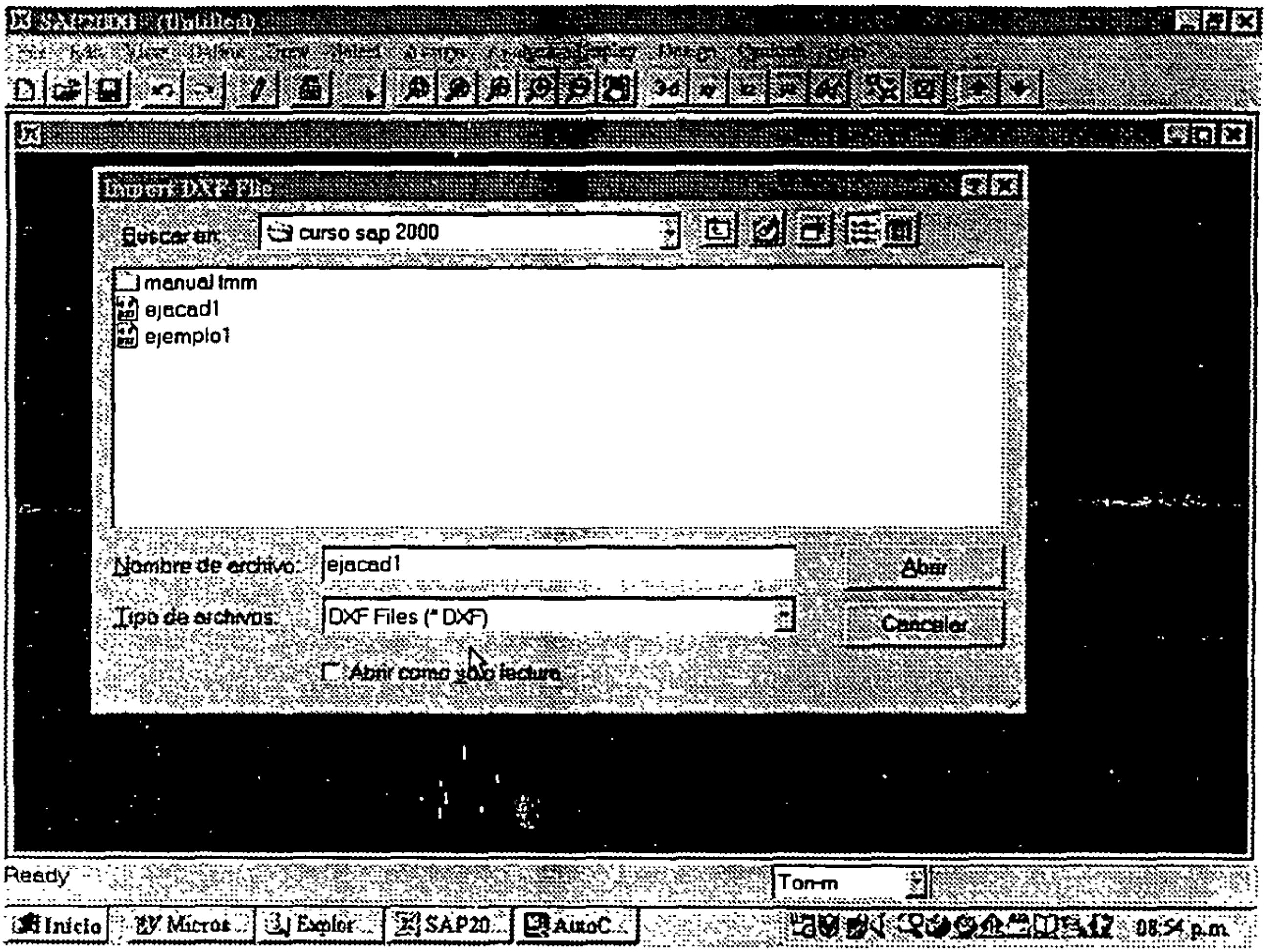


Figura 7.7 Ver.,ana para importar datos de un archivo **dxf**.

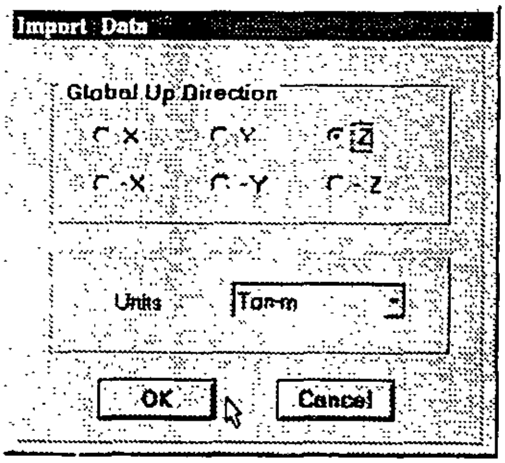


Figura 7.8 Indicando caracteristicas de los datos a importar de un archivo **dxf**.

EJEMPLOS E INTERPRETACION DE RESULTADOS

CAPÍTULO 8

8.1 INTRODUCCION

Durante la impartición del curso para uso y manejo del programa **SAP2000** tanto en la División de Ingeniería Civil, Topográfica y Geodésica como en la División de Educación Continúa de la Facultad de Ingeniería de la UNAM, se han desarrollado varios ejemplos típicos para el análisis de formas estructurales comunes (vigas continuas, marcos, armaduras, etc.) permitiendo al asistente practicar el uso de los comandos básicos tratados en los capítulos anteriores así como de algunos otros que no se han descrito o mencionado en este instructivo, por lo que sería conveniente que el lector interesado tuviera la oportunidad de asistir a alguno de esos cursos con objeto de despejar algunas dudas, desarrollar una mejor habilidad en el manejo del programa y adquirir una mejor comprensión de algunas de las opciones de Análisis así como de sus ventajas y limitaciones.

A continuación se presentan los listados (datos, resultados numéricos y gráficas) de algunos de los ejemplos que se han desarrollado durante los cursos que ha impartido el autor, los datos y resultados de otros más (incluyendo los que se listan a continuación) se encuentran en el disco que acompaña al presente instructivo, se sugiere que el interesado los consulte ya sea directamente (mediante algún editor) o procese los archivos de datos a través del programa **SAP2000**.

COMENTARIOS FINALES

CAPÍTULO 9

9.1 COMENTARIOS FINALES

Después de dar una primera visión hasta cierto punto con poca profundidad en algunos aspectos ya que se pretende que el presente instructivo inicie al lector interesado de una manera clara, rápida y sencilla en el uso de **SAP2000** pero a su vez lo motive para que explore y profundice en otras opciones que están disponibles en el programa, los comentarios finales tienen por objeto eliminar algunas posibilidades de error en los datos proporcionados al programa así como mejorar la interpretación de resultados, estos son:

- Verificar la geometría y características de los materiales, para ello se sugiere almacenar los datos en un archivo con extensión S2K y revisarlos con algún editor con objeto de detectar posibles errores en las características de los materiales y dimensiones de los elementos.

- Verificar las condiciones y combinaciones de carga bajo las cuales se realizará el análisis del modelo, lo anterior con objeto de detectar posibles omisiones o duplicidad de cargas (por ejemplo peso propio).

- Verificar los grados de libertad y las condiciones de apoyo o restricción de los nudos especificados en el archivo de datos los cuales deberán ser acordes con el número de ecuaciones que se forman y resuelven.

- Una vez realizado el Análisis se deberán de verificar e interpretar los resultados, el equilibrio se deberá satisfacer en todo momento, se recomienda que manualmente se verifique éste, por lo menos de manera global (suma de fuerzas externas y reacciones), sin embargo no está por demás verificar el equilibrio de algunos elementos de la estructura de manera aislada (nudos y barras), por ejemplo verificando el equilibrio de algún entrepiso (suma de cortantes en las columnas de entrepiso con el cortante externo en la misma dirección)

- Se deberá verificar la forma de los diagramas de elementos mecánicos la cual tendrá que corresponder con el tipo de cargas, por ejemplo, si en una barra existe carga uniforme el diagrama de cortantes deberá presentar variación lineal y el de momentos variación parabólica.

- La configuración deformada que la estructura presente para alguna condición de carga deberá ser consistente con las condiciones de apoyo del modelo analizado así como con las caracteristicas de las fuerzas contenidas en esa condición.

- Cuando sea posible se tratará de relacionar los resultados obtenidos con el programa con los que resulten de la aplicación de algún método aproximado, por ejemplo para cargas laterales en un marco se puede utilizar alguno de los métodos aproximados como el del factor o el de Bowman.

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP – 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

ANÁLISIS Y DISEÑO INTEGRADO DE ESTRUCTURAS POR EL MÉTODO DE ELEMENTOS FINITOS

EJEMPLO DE APLICACIÓN

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

SAP2000®

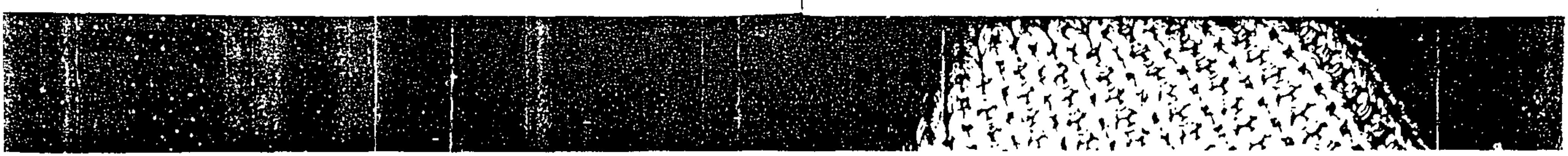
Análisis y Diseño Integrado de Estructuras por el Método de Elementos Finitos

EJEMPLOS DE APLICACION

COMPUTERS & STRUCTURES INC

Computers and Structures, Inc.
Berkeley, California, USA

Version 6.1
September 1997



DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000 THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMP-TIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Indice

Ejemplo 1	Pórtico Bidimensional bajo Carga Estática............	1
Ejemplo 2	Pórtico Bidimensional con Carga de un Espectro de Respuesta	13
Ejemplo 3	Pórtico Bidimensional Análisis de Historia en el Tiempo..	20
Ejemplo 4	Diseño en Acero de un Pórtico Bidimensional..........	32
Apéndice A	Descripción de los Iconos de la Barra de Herramientas..	A1
Apéndice B	Descripción de los Iconos de la Barra Flotante..........	B1

4. Edite la geometría de la malla y presione el botón OK para cerrar la plantilla

Sugerencia: *Finalizada la edición de la malla, se puede hacer click con el botón derecho del mouse sobre las columnas para verificar si éstas tienen la longitud apropiada. Esta es una manera muy práctica de obtener información sobre cualquier nudo o elemento de la estructura.*

Edición de Apoyos

El siguiente paso es el cambio de los apoyos de la estructura de la opción por defecto que corresponde a nudos articulados, a la opción de nudos rígidos que tenemos en este caso.

1. Seleccione el icono **Pointer Tool** de la barra de herramientas flotante.
2. Marque un área rectangular que abarque los tres nudos en la base de la estructura.

Sugerencia: *Se puede observar la barra de estado para ver el número y tipo de elementos que han sido seleccionados*

3. Seleccione el icono **Assign Joint Restraints** de la barra de herramientas flotante para asignar empotramiento en los apoyos de la estructura. Se pueden asignar también otras características de los nudos desde el menú **Assign**.

Definición de la Sección Transversal de los Elementos

1. Seleccione primeramente todas las secciones transversales que van a emplearse en el pórtico. Desde el menú **Define** seleccione la opción **Frame Sections**. Luego importe los perfiles de acero mostrados en la Figura 1-1.

Nota: ***Se puede seleccionar más de una sección a la vez de la lista Section Selection. Para ello presione la tecla Ctrl mientras se efectúa la selección.***

2. Bajo el menú **Select** encontrará varias formas de seleccionar nudos y elementos. Para este problema son útiles los modos de selección **Pointer/Window** e **Intersecting Line**.
3. Una vez seleccionados los elementos del pórtico deseados, se podrán asignar las secciones de acero correspondientes a través del botón **Assign Frame Sections** que está ubicado en la barra de herramientas flotante

Asignación de Cargas

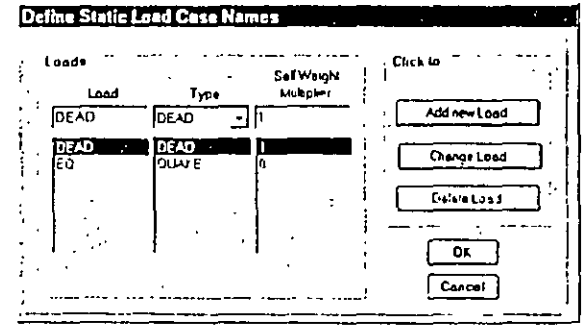


Figura 1-2 Plantilla con los nombres de las condiciones de carga estática

1. El primer paso al ingresar las cargas es definir las condiciones de carga estática. Para ello ingrese al menú **Define** y seleccione la opción **Static Load Cases**

- DEAD puede usarse para las cargas verticales por peso propio de las vigas, manteniendo el indicador **Self Weight Multiplier** con el valor 1, SAP2000 agregará el peso propio de las vigas

2. Defina una condición de carga lateral estática llamada EQ para la carga de sismo. Asigne esta carga lateral como una carga del tipo **QUAKE**. Esto permitirá al programa efectuar automáticamente las combinaciones de carga a ser empleadas por el módulo de diseño del SAP2000. Además asigne al parámetro **Self Weight Multiplier** el valor cero.
3. Las cargas verticales mostradas en la Figura 1-1 pueden asignarse a las vigas seleccionando todas las vigas utilizando el botón **Assign Frame Span Loads** de la barra de herramientas flotante.
4. Las cargas laterales estáticas necesitan ingresarse seleccionando individualmente cada nudo y empleando el botón **Assign Joint Loads**.

Importante: ***Asegúrese de que esté añadiendo las cargas a la condición de carga correspondiente.***

Creación de Diafragmas de Piso

Crear diafragmas de piso y especificar la masa del piso sólo en la dirección X son técnicas comúnmente usadas para reducir el tamaño del problema. Por otro lado, al añadir diafragmas el comportamiento del modelo se asemeja al de un edificio con diafragmas rígidos.

1 Repita los siguientes pasos para cada piso

- Seleccione todos los nudos del piso
- Entre al menú **Assign** y seleccione la opción **Joint ... Constraints**.
- Seleccione **Add Diaphragm** de la caja de opciones.
    - En la plantilla **Diaphragm Constraint** ingrese un nombre para el diafragma del primer piso. En este caso usaremos el nombre DIA1.
    - Seleccione la opción **Z-axis constraint**. Esta opción define un diafragma perpendicular al eje-Z.
    - Presione el botón OK
- Presione el botón OK para finalizar la operación.
- Repita estos pasos para los demás pisos usando diferentes nombres en cada uno

2 La masa de todos los pisos es la misma. Luego seleccione un nudo en cada piso.

3 Cambie las unidades en que se van a ingresar los datos a Kip-in, puesto que la masa indicada en la Figura 1-1 está dada en esas unidades.

4 Del menu **Assign** seleccione la opción **Joint ... Masses**

- Ingrese la masa de cada piso en la dirección del eje coordenado local 1 (que en éste caso coincide con la dirección del Eje Global X).
- Todos los demás valores son cero

5 Retorne las unidades a Kip-ft.

Propiedades de los Materiales

Por último, antes de efectuar el análisis de la estructura, deberemos verificar que la asignación de las propiedades de los materiales es la correcta.

1. Desde el menú **Define** seleccione la opción **Materials**.
2. En la plantilla **Materials** seleccione STEEL y presione el botón MODIFY/SHOW MATERIAL.

- En la plantilla **Material Property Data** verifique que las propiedades del material sean las correctas. Recuerde que los valores son reportados en las unidades con las que se este trabajando en este momento

Efectuando el Análisis

Una vez que los datos han sido ingresados, es tiempo para correr el modelo y revisar los resultados.

1. Grabe el modelo.
2. Especifique los parámetros para el análisis seleccionando la opción **Analyze** del menú **Set Options**
    - En la plantilla **Analysis Options** seleccione **Plane Frame Analysis** para reducir el tamaño del problema y por tanto reducir el tiempo de cálculo.
    - Presione el botón OK para aceptar los cambios realizados.
3. Seleccione la opción **Run** del menú **Analyze** para proceder al análisis la estructura.

    Nota: ***Una vez concluido el análisis Ud. podrá revisar los resultados completos en la pantalla antes de presionar el botón OK. Esta será su primera verificación para ver si existe algún problema en el modelo.***

Usando los Resultados

Verificación de los Resultados

Una vez que se ha analizado el modelo se debe verificar si los resultados son correctos y que sus valores son del orden y magnitud a los esperados.

Verificación del Modelo:

1. Verifique que el cortante total en la base es igual a la carga lateral total para la condición de carga EQ.
    - Seleccione el grupo de elementos del pórtico que están ubicados en el primer nivel así como los nudos en la base de la estructura.
    - Desde el menú **Assign** seleccione **Group Names**.
    - Asigne a este grupo de elementos un nombre representativo por ejemplo BASE SHEAR
    - Seleccione el botón ADD NEW GROUP NAME y presione el botón OK.
    - En el menú **Display** seleccione la opción **Show Group Joint Force Sums** y elija el grupo previamente creado.

2 Observe la deformada de la estructura y cree una animación de la misma bajo cargas verticales y laterales para asegurarse de que el comportamiento del modelo es el esperado.

- Ingrese al menú Display y seleccione Show Deformed Shape y seleccione la condición de carga en la que este interesado. También seleccione la opción Wire Shadow, así podrá ver la geometría no-deformada de la estructura al mismo tiempo. Vea las Figuras 1-3 y 1-4 para las formas deformadas de la estructura. Haga click con el botón derecho del mouse sobre cualquier nudo para observar los desplazamientos y rotaciones correspondientes.

- Genere una animación de la deformada presionando el botón START ANIMATION ubicado en la parte inferior de la barra de estado (Para esto se necesita que se encuentre activa una ventana conteniendo la deformada de la estructura). La animación así creada puede salvarse como una archivo *.AVI para verse después desde el menú File. (Vea la Ayuda En-línea bajo el ítem "Export an AVI file".)

  Intente esto : ***Presione los botones + y – ubicados junto al botón Animate y vea lo que le sucede a la deformada de le estructura.***

- Presione el botón STOP ANIMATION cuando haya terminado de observar la animación.

Si los procedimientos antes descritos muestran que la información ingresada aparenta ser correcta, podemos entonces avanzar hacia procedimientos más avanzados de revisión de los resultados.

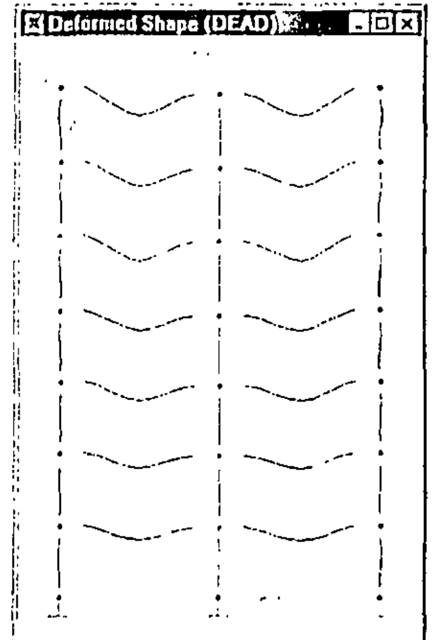


Figura 1-3 Deformada de la Estructura para Cargas Verticales

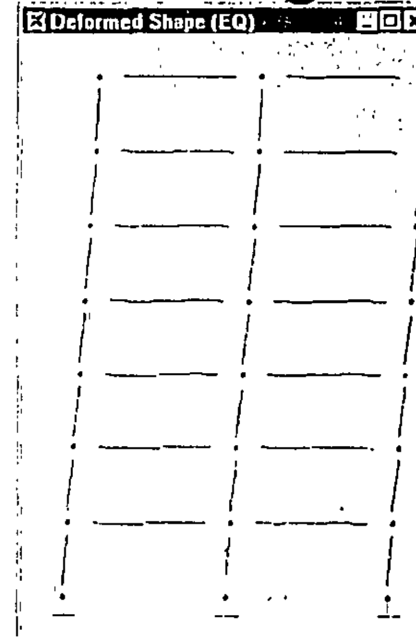


Figura 1-4 Deformada de la Estructura para Cargas Laterales

Comportamiento Estructural

En muchas ocasiones se desea verificar si la estructura se encuentra dentro de determinados límites de comportamiento, tales como rangos de esfuerzo especificados por algún código de diseño. SAP2000 hace todas estas verificaciones automáticamente cuando los elementos son diseñados. (Las opciones de diseño del SAP2000 serán discutidas con mayor detalle en los siguientes ejemplos)

1. Los elementos estructurales pueden diseñarse desde al menú Design y seleccionando la opción Start Design/Check of Structure.

   - Los elementos del pórtico mostrarán en este momento colores que representan el nivel del esfuerzo en cada elemento. Asimismo en la parte inferior de cada elemento se muestra un valor numérico representativo del nivel de esfuerzo presente en el elemento. Un valor 1 por ejemplo significa 100% esforzado.

- Para tener más información sobre el diseño de los elementos entre al menú **Design** y seleccione la opción **Display Design Info**.

2. Se puede también ver la información del diseño de cada elemento, o inclusive asignar secciones transversales alternativas, haciendo click con el botón derecho del mouse sobre un elemento
   - De la ventana que se muestre se puede seleccionar el botón DETAILS para apreciar información detallada de la sección bajo cada una de las combinaciones de carga empleadas en el diseño.
   - También se puede rediseñar el elemento después de cambiar sus parámetros de diseño, longitud efectiva, factor K o propiedades a la sección, presionando el botón REDESIGN.
3. Si se ha seleccionado una nueva sección la cual se quiere utilizar en el diseño final de la estructura, únicamente ingrese al menú **Design** y seleccione la opción **Update Analysis Sections** para reanalizar la estructura con las nuevas secciones seleccionadas.

> *Nota: Puede ser necesario el uso del botón Refresh Window de la barra de herramientas para actualizar la información en la ventana activa luego de haber efectuado cambios en los parámetros de diseño*

Observando e Imprimiendo Resultados

A menudo se necesita disponer de una copia impresa de los resultados de los análisis obtenidos con el SAP2000 Existen diferentes formas de obtenerlos:

1 Se pueden elegir los resultados que nos interesan con la opción **Generate Output** ubicada en la plantilla **Analysis Options**. El botón **Select Output Options** que aparece permitirá seleccionar cuantos y cuales de los resultados queremos imprimir Estos resultados son escritos en un archivo de texto con el mismo nombre de nuestro archivo de datos, pero con la extensión *.OUT.

2 Los datos ingresados así como la mayor parte de los resultados generados también pueden verse a partir del menú **Display** .

3 Desde el menú **File** se puede optar por imprimir ya sea Gráficos , Tablas con los Datos Ingresados ó Tablas con los Resultados del Análisis y Diseño de elementos.

> *Sugerencia: Si existen elementos ó nudos seleccionados al momento de generar la impresión de resultados, únicamente se imprimirá la información correspondiente a dichos elementos. De lo contrario, la impresión se generará para todos los elementos y nudos del modelo.*

4. El análisis efectuado por el SAP2000 genera dos archivos de salida. El archivo *filename*.EKO que incluye toda la información empleada en el análisis; y el archivo *filename*.OUT, que

contiene los resultados del análisis así como los resultados específicamente seleccionados en el menú **Analyze ...Set Options**.

> Recuerde: ***Es un buen hábito generar la salida primeramente a un archivo de texto antes de enviarlo directamente a la impresora. Esto nos permite revisar previamente la información usando cualquier editor de textos, sin tener que hacer frente a enormes pilas de papel***

Comentarios Finales

Como habrá podido observar, SAP2000 es una poderosa herramienta para el análisis estructural que puede usarse en una gran variedad de problemas Sin embargo, es muy importante entender los principios de ingeniería sobre los cuales este programa ha sido creado.

La mayoría de los trabajos en ingeniería se inician con sencillos anteproyectos para posteriormente madurar en complejos proyectos de análisis/diseño. Esto hace que sea muy importante decidir desde un inicio las herramientas apropiadas de forma tal que no sea necesario cambiar de programas a la mitad de un proyecto SAP2000 trata de satisfacer la mayor parte de las necesidades que un diseñador puede tener durante el desarrollo de un proyecto.

Las características que SAP2000 ofrece en el proceso de diseño incluyen:

- La capacidad de diseñar pequeños ó grandes proyectos sin necesidad de aprender a usar un nuevo programa
- La capacidad de diseñar elementos de concreto y acero en un mismo programa.
- Algoritmos de cálculo rápidos que permiten al usuario dedicar mayor tiempo en la modelación del problema y optimización del diseño de elementos estructurales.
- La habilidad para modificar y mejorar el diseño fácilmente.
- Existen probablemente tantas formas de modelar una estructura como ingenieros existen. Sin embargo, puede encontrar útiles algunas de las siguientes ideas :
- Comience con un modelo básico de la estructura y trate de entenderlo antes de añadir más detalles Será más sencillo corregir problemas en el sistema estructural adoptado cuando el modelo es aún simple
- Asegúrese de que la estructura pueda construirse y que se comportará en la manera en que la hemos modelado Si no puede ser construida en esa forma, es necesario entender el efecto del proceso constructivo en el comportamiento final de la estructura.
- Documente detalladamente su diseño incluyendo información sobre las consideraciones asumidas, áreas que deban revisarse e incluso sobre información que aún es requerida. Pare ello use el editor de textos **User Comments and Session Log** que se encuentra dentro del menú **File**. Este editor de textos incorporado en el programa, le permitirá que dichas anotaciones y comentarios formen parte del modelo.

- Experimente con sistemas estructurales alternativos. SAP2000 ha sido diseñado para efectuar cálculos numéricos rápidamente, permitiendo utilizar mayor tiempo en el mejoramiento de nuestros diseños
- Así como hay un tiempo asignado para la revisión general al final de un proyecto, no hay razón por la que no deba haber un tiempo para revisar el proyecto desde sus inicios

EJEMPLO 2

Pórtico Bidimensional con Carga de un Espectro de Respuesta

Descripción

Este ejemplo es una continuación del Ejemplo 1. En esta sección mostraremos como incorporar un Espectro de Respuesta en el análisis de un pórtico bidimensional. La base para definir el Espectro de Respuesta será el espectro del código UBC94S2 el cual esta incluido en el SAP2000.

Aspectos Significativos del Modelo y del SAP2000

- Uso del comando Help para obtener instrucciones sobre las opciones del SAP2000.
- Incorporación de una carga proveniente de un Espectro de Respuesta..
- Adecuar la escala del Espectro de Respuesta para su uso en el diseño.

Definiendo el Espectro de Respuesta

Un Espectro de Respuesta es la máxima respuesta de un sistema excitado en su base por una función aceleración-tiempo. Esta función se expresa en términos de la frecuencia natural de la estructura y del amortiguamiento del sistema. El Espectro de Respuesta del código UBC94S2 que vamos a emplear en este ejemplo es suministrado con SAP2000 y no es necesario definirlo por separado. Si hubiese la necesidad de definir un Espectro de Respuesta distinto, se puede usar la ayuda en línea para obtener instrucciones que indican paso a paso como efectuar esta tarea.

Ayuda En-línea

Recuerde: ***Podrá utilizar alguno de los métodos siguientes para obtener información sobre cualquiera de las funciones del SAP2000***

1 Del menú Help seleccione **Search for Help on.**

2 Con el plantilla **Index** seleccionada:

- En el Area 1 escriba 'Define . Ud. vera en el Area 2 una lista de todos los tópicos disponibles que comiencen con la palabra 'define'. Uno de esos tópicos es 'Define Response Spectrum Functions', que es el tópico del cual necesitamos obtener ayuda. Haga doble click en la línea con la frase 'Define Response Spectrum Functions' para que el programa muestre la información correspondiente.

3 Alternativamente, seleccione el indicador Find para buscar una palabra clave en cualesquiera de los tópicos disponibles en la Ayuda En-línea.

- Si es la primera vez que usa la opción Find de la Ayuda En-línea del SAP2000, aparecerá una plantilla denominada Find Setup Wizard.
    - Presione el botón NEXT para aceptar el criterio para construir la base de datos de búsqueda.
    - Presione el botón FINISH para construir la base de datos.
- En el Area 1 escriba 'Response Spectrum'
- En el Area 3 encontrará nuevamente la opción 'Define Response Spectrum Functions' la cual puede seleccionarse para obtener la información de ayuda correspondiente.

Nota: ***Se puede encontrar mayor información sobre el uso de la Ayuda En-línea, en la documentación de Windows. También puede ejecutar el archivo WINHELP32.HLP ubicado en C:\WINDOWS\HELP.***

Definiendo el Espectro de Respuesta

1 Si el modelo esta protegido (locked), use el botón **Lock/Unlock Model** para remover la protección y poder efectuar cambios en el modelo

2. Ajuste las unidades a Kip-ft.
3. Ingrese al menú Define y seleccione la opción Response Spectrum Case.
4. Presione el botón ADD NEW SPECTRA en la plantilla Response Spectra.
5. En la plantilla Response Spectrum Case Data:
    - Especifique el amortiguamiento asociado al Espectro de Respuesta colocando en la casilla Damping el valor, para nuestro caso: 0.05 (5 %)
    - Seleccione UBC94S2 para la dirección U1, así como un factor de escala 32.2ft/sec^2 en la casilla Scale Factor. Este factor de escala es usado por el Espectro de Respuesta debido a que el espectro UBC94S2 esta normalizado al valor de la aceleración de la gravedad g
    - El resto de los valores por defecto son aceptables.
    - Presione el botón OK para aceptar los cambios hechos en ambas plantillas.

Efectuando el Análisis

Una vez que se han realizado las modificaciones, es tiempo de analizar el modelo y echar una mirada a los resultados del Espectro de Respuesta.

1. Grabe el modelo.
2. En el menú **Analyze** seleccione la opción **Set Options**.
    - Marque la casilla **Dynamic Analysis.**
    - Presione el botón SET DYNAMIC PARAMETERS y modifique el número de modos de vibración a ser considerados en el análisis en la opción Number of Modes. Para nuestro caso 7. El resto de valores por defecto son aceptables.
    - Presione el botón OK en ambas plantillas para aceptar los cambios.

    Nota: ***Se debe decidir cuantos modos de vibración deben considerarse en el análisis para obtener resultados adecuados. Para ello hay muchos criterios a tomar en cuenta, pero para una estructura sencilla como la que estamos analizando puede considerarse satisfactorio un numero de modos igual al numero de pisos.***
3. Seleccione la opción **Run Minimized** del menú **Analyze** para analizar la estructura.

    Nota: ***La opción Run Minimized es sumamente útil cuando se tienen modelos grandes que requieren mayor tiempo para ser analizados. Esta opción permite al SAP2000 correr en un segundo plano, permitiendo continuar trabajando con otros programas. Otra ventaja de esta opción es que nos brinda un botón para cancelar la ejecución del análisis en caso necesario.***

Verificación de los Resultados

1. Verifique si las formas modales y períodos de vibración son los esperados.

   - Del menú **Display** seleccione la opción **Show Mode Shape** y elija el modo en que esta interesado. Pude también seleccionar la opción **Wire Shadow** para ver al mismo tiempo la forma no-deformada de la estructura. Observe las Figuras 2-1 a la 2-4 y note que el número de modo y período de vibración correspondiente están indicados en el título de la ventana.

   Nota: *Se puede apreciar los modos de vibración subsecuentes presionando los botones + y −, próximos al botón START ANIMATION.*

2. Es útil ver el cortante en la base producido por el análisis del Espectro de Respuesta.

   - Usando el grupo BASE SHEAR que fue definido en el Ejemplo 1 observe el cortante en la base de la estructura debido al Espectro de Respuesta. Se puede apreciar que este cortante es considerablemente mayor que el debido a la condición de carga estática.

3. Se puede verificar el desplazamiento de un nudo debido al Espectro de Respuesta.

   - Del menú **Display** seleccione **Show Deformed Shape.**
     - En la plantilla **Deformed Snape** seleccione la condición de carga para el análisis espectral.
     - Presione el botón OK.
   - Haga click con el botón derecho del mouse sobre un nudo del nivel superior de la estructura para ver su correspondiente desplazamiento en la dirección global X.

4. Verifique la participación de la masa de la estructura para ver si se ha incluido el en la solución del problema el número de modos suficiente. Para ello se debe revisar el archivo de texto filename.OUT empleando un editor de textos como el WordPad de Windows.

   - Minimice el programa SAP2000.
   - Inicie el programa WordPad o cualquier otro editor de textos.
   - En WordPad abra el archivo *filename*.OUT. Donde *filename* es el nombre del archivo usado al grabar este ejemplo.
     - Busque la sección titulada MODAL PARTICIPATING MASS RATIOS como se muestra en la Figura 2-5.
     - Bajo la columna CUMULATIVE SUM encontrará que los Modos de Vibración 1 al 7 incluyen el 100% de la participación de masa. Lo cual significa que los 7 modos empleados en el análisis fueron suficientes.

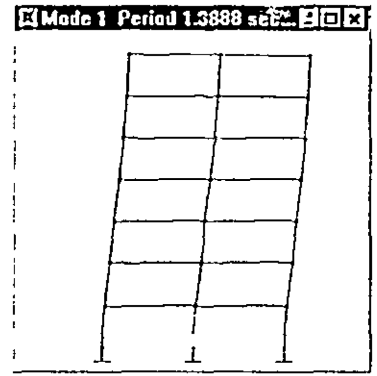


Figura 2-1 Forma Modal y Período de Vibración 1

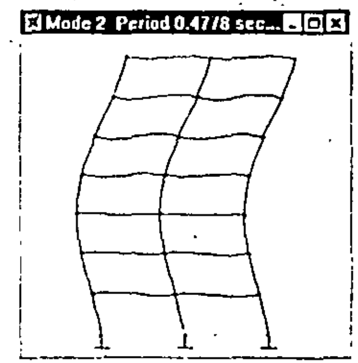


Figura 2-2 Forma Modal y Período de Vibración 2

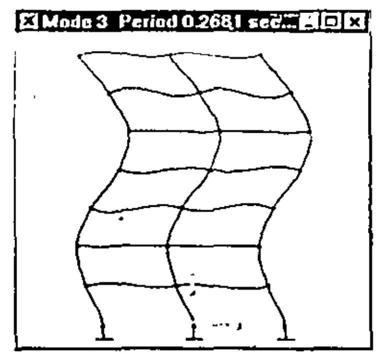


Figura 2-3 Forma Modal y Período de Vibración 3

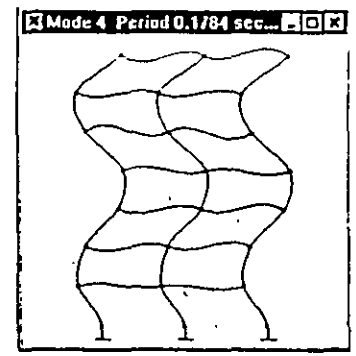


Figura 2-4 Forma Modal y Período de Vibración 4

MODAL PARTICIPATING MASS RATIOS

MODE	PERIOD	INDIVIDUAL MODE (PERCENT) UX	UY	UZ	CUMULATIVE SUM (PERCENT) UX	UY	UZ
1	1.388750	79.6359	0.0000	0.0000	79.6359	0.0000	0.0000
2	0.477933	11.5761	0.0000	0.0000	91.2120	0.0000	0.0000
3	0.268126	4.3033	0.0000	0.0000	95.5144	0.0000	0.0000
4	0.178439	2.1229	0.0000	0.0000	97.6373	0.0000	0.0000
5	0.133678	1.4077	0.0000	0.0000	99.0450	0.0000	0.0000
6	0.107637	0.6592	0.0000	0.0000	99.7043	0.0000	0.0000
7	0.090778	0.2957	0.0000	0.0000	100.0000	0.0000	0.0000

Figura 2-5 Bloque de Participación de la Masa del archivo de salida

Modificando la Escala del Espectro de Respuesta

Algunos códigos de diseño permiten modificar la escala del Espectro de Respuesta de manera que el cortante en la base del análisis espectral sea igual al cortante en la base del análisis empleando cargas sísmicas estáticas

En este sentido para obtener el nuevo factor de escala para el Espectro de Respuesta tenemos que :

1. Dividir el Cortante en la Base producido por la Carga Sísmica Estática por el Cortante en la Base obtenido del Análisis Espectral, y multiplicar dicho número por 32.2 ft/seg^2 para obtener el nuevo factor de escala del Espectro de Respuesta.
2. Substituir el nuevo factor de escala en el Espectro de Respuesta.
3. Efectuar nuevamente el análisis para obtener las nuevas fuerzas en los elementos bajo la acción del Espectro de Respuesta escalado

Comentarios Finales

Un análisis empleando el Espectro de Respuesta introduce un nivel de complejidad mayor, que requiere que el ingeniero verifique cuidadosamente los resultados, y tenga muy presente las consideraciones hechas al crear el modelo. Algunos de los aspectos a considerar durante un análisis espectral son:

- Entender completamente el comportamiento estático del modelo antes de efectuar un análisis dinámico.
- Tener un conocimiento cabal y racional de los aspectos involucrados al escalar los resultados del análisis dinámico para obtener un cortante en la base similar al que se obtiene al efectuar un análisis por cargas sísmicas estáticas.
- La ventaja de la mayor rapidez del Análisis Espectral en comparación con el Análisis de Historia en el Tiempo es en muchos casos sustancial. En el diseño, el espectro de respuesta puede incluso proveer aun mayores ventajas debido a que no se deben efectuar verificaciones para diferentes intervalos de tiempo. Sin embargo, es necesario tener presente las limitaciones del Análisis Espectral frente al mayor refinamiento que se obtiene al efectuar un Análisis de Historia en el Tiempo.

EJEMPLO 3

Pórtico Bidimensional
Análisis de Historia en el Tiempo

Descripción

Este ejemplo continua el análisis del pórtico bidimensional visto en los Ejemplos 1 y 2, añadiendo en este caso una carga de sismo especificada con un acelerograma en la base de la estructura. El registro de aceleraciones a utilizarse se muestra en la Figura 3-1 y corresponde a la componente N-S del sismo ocurrido en El Centro en 1940. Los resultados del análisis de historia en el tiempo se utilizarán para generar un Espectro de Respuesta que luego se empleará para reanalizar la estructura a manera de comparación.

Aspectos Significativos del Modelo y del SAP2000

- Respuesta de historia en el tiempo de una excitación en la base.
- Gráficas de los resultados del análisis de Historia en el Tiempo.
- Gráficas de un Espectro de Respuesta a partir de los resultados de la Historia en el Tiempo.
- Importación del Espectro de respuesta para su uso en el análisis.

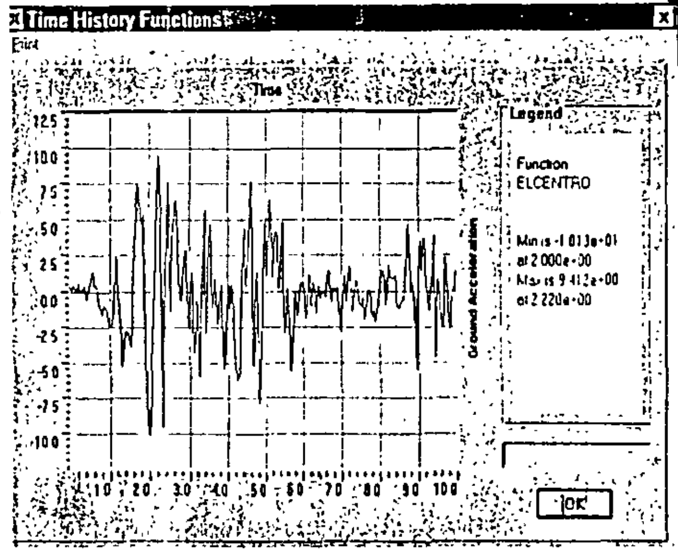


Figura 3-1 Acelerograma de entrada del Sismo de El Centro 1940 (ft/sec^2)

Definición del Concepto Historia en el Tiempo

El término Historia en el Tiempo define un registro de aceleraciones del terreno a determinados intervalos de tiempo para una excitación sísmica específica en una dirección determinada. El registro es usualmente normalizado y en consecuencia necesita multiplicarse por la aceleración de la gravedad ó por algún otro factor correspondiente.

1. Desde el menú **Define** seleccione **Time History Functions**.
2. Seleccione el botón ADD FUNCTION FROM FILE.
   - Presione el botón **Open File** y seleccione el archivo ELCENTRO ubicado en el subdirectorio EXAMPLES del directorio SAP2000.
   - Cambie el nombre de la función a ELCENTRO para que sea fácil de reconocer.
   - El formato de este archivo es de tres pares de columnas de datos por renglón. El primer par de datos la primera columna de cada par es el tiempo y la segunda es la aceleración.
     - Ingrese 3 Puntos por Línea.

- Seleccione la opción **Time and Function Values** (en archivo de datos).
- Presione el botón OK.

- Presione el botón OK para aceptar los datos ingresados.

3. Del menú **Define** seleccione la opción **Time History Cases** para definir los parámetros específicos para el análisis de Historia en el Tiempo de nuestro modelo

- Seleccione el botón ADD NEW HISTORY.
- Para adicionar amortiguamiento al sistema presione el botón MODIFY/SHOW MODAL DAMPING, ingrese 0.05 (5%) para todos los modos y presione OK.
- Ingrese en el ítem **Number of Output Time Steps** el valor 500.
- Ingrese 0.02 (sec) en el ítem **Output Time Step Sizes**. Estos parámetros nos darán 10 segundos del registro sísmico para el análisis de historia en el tiempo.
- De la Lista **Analysis Type** seleccione la opción **Linear**.
- En el área **Load Assignment** :
  - Seleccione ACC DIR1 para el parámetro **Load**
  - Seleccione la opción ELCENTRO para el parámetro **Function**
  - Para el parámetro **Scale Function** ingrese la aceleración de la gravedad que es de 386.4 in/sec^2 si trabaja en Kip-in o 32.2 ft/sec^2 si trabaja en Kip-ft.
  - Para los parámetros **Arrival Time** y **Angle** asigne valor cero.
  - Presione el botón **ADD** para agregar esta carga al modelo, y presione el botón OK para aceptar los datos que acaba de ingresar.
- Presione el botón OK en ambas plantillas para aceptar las adiciones al modelo.

De esta manera hemos ingresado toda la información que necesitamos para efectuar el Análisis de Historia en el Tiempo

> Nota: *Por lo general es una buena idea correr el modelo cada vez que se hace un cambio ó adición importante al modelo. Esto permite detectar errores y ahorra tiempo en el diseño final.*

Efectuando el Análisis

1. Grabe el modelo.
2. Ajuste los parámetros para el análisis seleccionando **Analyze** del menú **Set Options**.
   - Verifique que los parámetros en **Dynamic Analysis** son los mismos del Ejemplo 2.
3. Seleccione en el menú **Analyze** la opción **Run** para analizar la estructura.

Usando los Resultados

Verificación de los Resultados

Una vez que ha corrido el modelo se deben verificar que los resultados obtenidos sean del orden y magnitud a lo esperado.

1. Verifique el Cortante en la Base producido en el Análisis de Historia en el Tiempo.
   - Desde el menú **Display** seleccione la opción **Show Time History Traces**.
     - De la plantilla **Time History Display Definition** presione el botón DEFINE FUNCTIONS
     - En la plantilla **Time History Functions** seleccione Add Base Functions y marque solamente la opción Base Shear X.
     - Presione OK para regresar a la plantilla **Time History Display Definition**.
     - Adicione la función Base Shear X a la lista **Plot Functions**.
     - Presione el botón DISPLAY para ver una gráfica del cortante en la base en la dirección global X como función del tiempo. Vea la Figura 3-2.

   > Nota: *También se puede generar la gráfica del cortante en la base seleccionando Add Group Summation Forces en lugar de Add Base Functions y seleccionando el grupo de elementos BASE SHEAR definido en el Ejemplo 1.*

2. También se puede verificar el desplazamiento de un nudo ante una excitación de Historia en el Tiempo, para ello:
   - Escoja un nudo y desde el menú **Display** seleccione la opción **Show Time History Traces**.

- Presione el botón DEFINE FUNCTIONS de la plantilla **Time History Functions**. Seleccione el nombre del nudo de la lista y presione el botón MODIFY/SHOW TH FUNCTION.
- En la plantilla Time History Joint Function seleccione DISPL para el parámetro **Vector Type** y UX para el parámetro **Vector Direction**.
- Presione el botón OK para aceptar los cambios.
- Presione los botones OK para regresar a la plantilla **Time History Display Definition**.
- Añada el nudo de la lista **List of Functions** a la lista **Plot Functions** y remueva de esta última la función Base Shear X.
- Presione el botón DISPLAY para ver el desplazamiento del nudo con respecto al tiempo. Vea Figura 3-3.

- Se puede también definir una función nodal directamente en la plantilla **Time History Display Definition** sin haber seleccionado previamente el nudo.
  - En la plantilla **Time History Display Definition** presione el botón DEFINE FUNCTIONS y en la plantilla **Time History Functions** seleccione la opción Add Joint Disps/Forces.
  - En la plantilla Time History Joint Function ingrese el nombre del nudo (ID).
  - Seleccione **Vector Type** y **Vector Direction**.
  - Presione los botones OK para regresar a la plantilla **Time History Display Definition** en donde encontrará la nueva función nodal en el recuadro **List of Functions**.

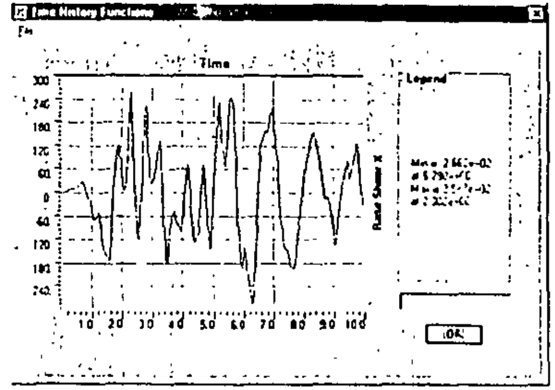


Figura 3-2 Cortante en la Base – Kips

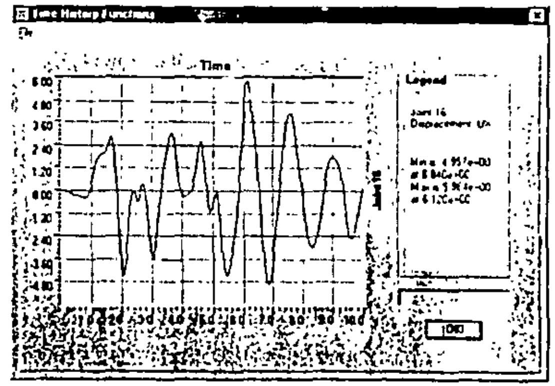


Figura 3-3 Historia en el Tiempo del Desplazamiento del Nivel Superior – In

Creación de un Espectro de Respuesta

Lo primero que se tiene que hacer en este punto es crear un Espectro de Respuesta a partir de los resultados del Análisis de Historia en el Tiempo. Los datos generados de esta forma deben imprimirse en archivo de texto para poder editadarse en un formato que pueda ser leído por el SAP2000.

Gráficas del Espectro de Respuesta

1. Seleccione un nudo en la base de la estructura
2. Del menú **Display** seleccione la opción **Show Response Spectrum Curves**. Esta opción aparece solamente cuando se ha seleccionado un nudo.
3. En la plantilla **Response Spectrum Generation** encontrará el nombre del nudo que fue seleccionado
   - Bajo el ítem **Define** asigne el valor X para el parámetro **Vector Direction**.
   - Bajo el ítem **Axes** seleccione **Period** para el parámetro **Abscissa** y **PSA** (Seudoaceleración Espectral) para el parámetro **Ordinate**.
   - Bajo el ítem **Options** seleccione **Arithmetic** tanto para **Abscissa** como para **Ordinate**. Para el parámetro **Ordinate** asigne el factor de escala 1/g (g=32.2 ft/sec^2) es decir, 0.03106 sec^2/ft si las unidades en que se trabaja son kip-ft.

     Nota: *El factor de escala es usado para normalizar el Espectro de Respuesta. El registro para el análisis de historia en el tiempo que se utilizó para generar el Espectro de Respuesta estaba normalizado a la aceleración de la gravedad (g) por lo que necesitamos dividir el espectro por la misma cantidad para obtener valores normalizados.*
   - Bajo el ítem **Period** seleccione para las frecuencias los parámetros **Default** y **Structural**. Estos parámetros son usados en la generación del Espectro de Respuesta, las frecuencias del tipo **Default** son una serie de frecuencias predeterminadas que son típicamente de interés en las estructuras; las frecuencias del tipo **Structural** son las frecuencias naturales de la estructura.
   - Bajo el ítem **Damping** mantenga el valor del amortiguamiento de 0.05 para el parámetro **Damping Value**. Como se ha asumido que la estructura tiene 5% de amortiguamiento no será necesario emplear otros valores para el amortiguamiento del sistema.
   - Presione el botón **DISPLAY** cuando se halla terminado.
4. Ahora Ud. podrá apreciar la gráfica del Espectro de Respuesta para el sismo de El Centro para un amortiguamiento de 5%. Ver Figura 3-4.

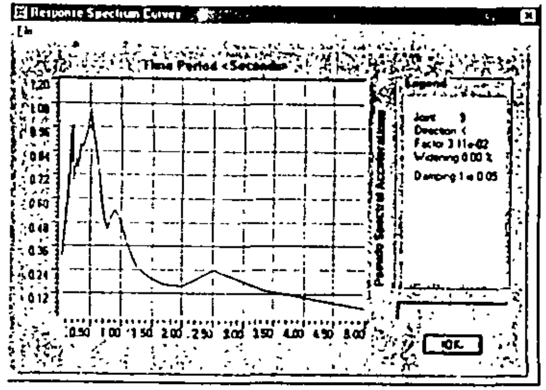


Figura 3-4 Espectro de Respuesta del Análisis de Historia en el Tiempo

5. En la plantilla **Response Spectrum Curves** seleccione la opción **Print Tables to File**. Esta opción generará un archivo que tiene dos columnas de datos. La primera de ellas es el período, y la segunda es su correspondiente Seudoacelaración (PSA).
   - Grabe el archivo con el nombre RS-ELCEN.TXT

Edición de Tablas.

El siguiente paso consiste en hacer algunas pequeñas modificaciones al archivo de texto RS-ELCEN.TXT para que tenga un formato que pueda ser leído por el SAP2000. Esto se debe a que cuando el archivo original es creado se le agrega información aclaratoria que permite al usuario interpretar y entender fácilmente su contenido. Esta información extra debe removerse.

1. Con un editor de textos como WORDPAD o NOTEPAD abra el archivo RS-ELCEN.TXT.
   - Seleccione todo el texto que se muestra resaltado en la **Figura 3-5** y remuévalo.
   - Grabe el archivo RS-ELCEN.TXT como un archivo de texto con el mismo nombre.
2. Ahora que el archivo tiene tan solo las columnas con los períodos y sus correspondientes Seudoacceleraciones, podrá ser leído directamente por el SAP2000

```
SAP2000 v6.06 File: TUTORIAL1 Kip-ft Units PAGE 1
May 19, 1997 17:26

S P E C T R U M   D A T A

Joint      .  9
Direction  .  XT
Factor     .  0.03
Widening   .  0.5

Pseudo Spectral Accelerations vs Time Period <Seconds>

              DAMPING
              0.0500
    0.0303  3.0866E-01
    0.0357  3.2657E-01
    0.0400  3.1139E-01
    0.0455  3.2217E-01
    0.0500  3.2205E-01
    0.0555  3.3098E-01
    0.0606  3.4265E-01
    0.0667  3.7431E-01
```

Figure 3-5 Archivo de salida con el Espectro de Respuesta generado

Lectura de los Datos del Espectro

Ahora que se tienen datos en un formato que el SAP2000 puede leer, necesitamos indicar al programa la ubicación del archivo así como la forma en que éste contiene la información.

1. Si el modelo esta protegido presione el botón **Lock/ Unlock Model** en la barra de herramientas. Al hacer esto se remueve la protección sobre el modelo y nos permitirá realizar las modificaciones.
2. Del menú **Define** seleccione **Response Spectrum Functions**.
3. En la plantilla **Response Spectrum Functions** presione el botón **Add Function from File**.
   - Asigne al espectro el nombre RSELCEN
   - Presione el botón **Open File** y seleccione el archivo RS-ELCEN.TXT en **Pick File**.
   - Mantenga el parámetro **Number Of Points Per Line** en el valor 1 puesto que únicamente hay un par de datos por renglón para definir el Espectro de Respuesta.
   - Seleccione la opción **Period and Acceleration Values**.
   - Presione el botón OK para cerrar las plantillas.

4. Del menú **Define** seleccione **Response Spectrum Cases**.
5. En la plantilla Response Spectrum presione el botón ADD NEW SPECTRA.
   - Asigne al parámetro **Modal Damping** el valor 0.05.
   - En el área **Input Response Spectra** seleccione RSELCEN para la dirección U1 y asígnele el factor de escala 32.2 ft/sec^2.
   - Los demás valores por defecto son aceptables.
   - Presione el botón OK para cerrar las plantillas.

Efectuando el Análisis

Una vez que se han hecho las modificaciones es tiempo de correr el modelo y revisar los resultados obtenidos

1. Grabe el modelo.
2. Seleccione **Run Minimized** del menú **Analyze** para analizar la estructura.

Revisando los Resultados

Lo primero que debe hacerse es la revisión de la máxima deflexión en la parte superior de la estructura así como el cortante en la base tanto para el análisis espectral como para el análisis de Historia en el Tiempo. La comparación de estos resultados nos permitirá ver que tan bién funciona la metodología previamente descrita. Al final de esta sección encontrará los resultados del análisis por cargas sísmicas estáticas, del análisis espectral y del análisis de historia en el tiempo.

Deflexiones de acuerdo al Análisis Espectral

1. Del menú Display seleccione Display **Deformed Shape.**
   - En la plantilla Deformed Shape seleccione la condición de carga para el análisis espectral.
   - Presione el botón OK.
2. Haga click con el botón derecho del mouse sobre un nudo ubicado en el nivel superior para ver su desplazamiento en la dirección del eje global X

Cortante en la Base en el Análisis Espectral

Usando el grupo BASE SHEAR que fue creado en el Ejemplo 1, observe los valores del cortante en la base de la estructura debido al Análisis Espectral.

Deflexión y Cortante en la Base en el Análisis de Historia en el Tiempo

1. Usando el método descrito en la primera parte de esta guía grafique la deflexión en el nivel superior de la estructura.
2. Ahora remueva ese nudo de la lista **Plot Functions** y en su lugar grafique la función **Base Shear X** ubicada en **List of Functions** de la plantilla **History Display Definitions.**

	Carga Lateral Estática	**Espectro de Respuesta**	**Historia en el Tiempo**
Max Deflexión	1.65 in	5.5 in	5.8 in
Max Cortante en la Base	72 5 Kips	302 Kips	325 Kips

Tabla 3-1 Comparación de los Resultados del Análisis de Cargas Laterales

Comentarios Finales

Como habrá podido apreciar el Análisis de Historia en el Tiempo involucra un mayor tiempo de cómputo que el Análisis con Espectro de Respuesta. Debe observarse sin embargo que ambos métodos de análisis brindan resultados similares. En este sentido es sumamente importante que el ingeniero entienda las ventajas y limitaciones de cada método para poder utilizarlos de la manera más adecuada y efectiva.

EJEMPLO 4

Diseño en Acero de un Pórtico Bidimensional

Descripción

Este ejemplo es un introducción al uso de las poderosas herramientas que posee el SAP2000 para el diseño de una estructura una vez concluido su análisis estructural. Se dará énfasis en esta parte a módulos de diseño en acero empleando como ejemplo la estructura analizada en el Ejemplo 1.

Aspectos Importante del Modelo y del SAP2000

- Creación de zonas rígidas en los elementos.
- Selección Automática de grupos
- Cambio de propiedades en elementos
- Designación de elementos por grupos
- Inclusión del efecto P-Delta en el análisis
- Visualización de los resultados
- Auto Selección de secciones

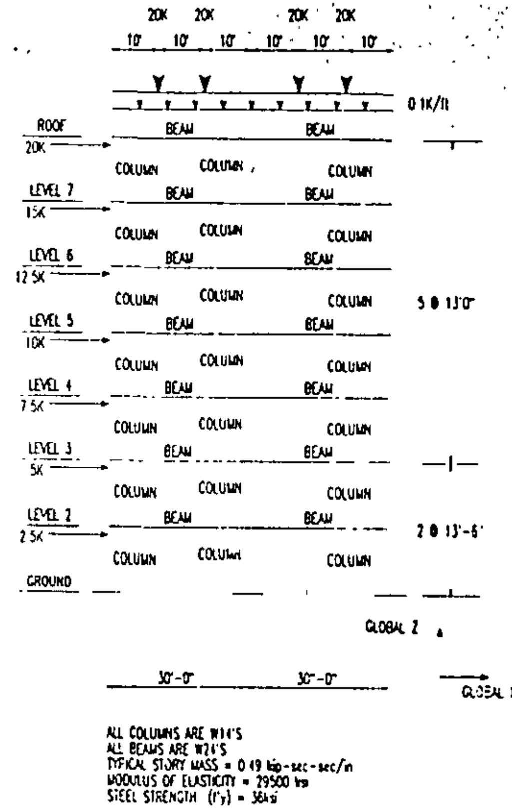


Figura 4-1 Pórtico Bidimensional a diseñarse

Creando el Modelo en el SAP2000

Es posible emplear el modelo desarrollado en el Ejemplo 1 efectuando pequeñas modificaciones.

Materiales

Lo primero que se debe hacer es especificar las propiedades de los materiales

1. Verifique que las unidades estén en Kip-in.
2. Entre al menú **Define** y seleccione la opción **Materials**.
3. Elija STEEL para el parámetro **Material** y presione el botón MODIFY/SHOW MATERIAL
4. Especifique el Esfuerzo de Fluencia ajustando el parámetro **Steel Yield Stress fy** en 36 Ksi.
5. Especifique el Módulo de Elasticidad ajustando el parámetro **Modulus of Elasticity E** en 29,500 Ksi
6. Presione los botones OK para aceptar los cambios y cerrar las plantillas

Cargas

1. En el ejemplo 1 se asignó el peso propio de la estructura así como una serie de cargas concentradas y distribuidas a la condición de carga DEAD. (Vea Figura 4-2 para la nueva lista Static Loads Case). En este ejemplo asignaremos una condición de carga para la carga viva y otra para el peso propio de los elementos. Es una buena práctica incluir una condición de carga para el peso propio de la estructura con el fin de seguir de cerca el proceso de optimización estructural. Las cargas son separadas en carga muerta, carga viva y carga transversal de sismo de tal manera que el SAP2000 pueda generar automáticamente las combinaciones de carga.
   - Para el caso de carga DEAD especifique el valor del multiplicador **Self Weight Multiplier** en cero.
   - Agregar una condición de carga para el peso propio y nómbrela SELF, asígnele el tipo DEAD como parámetro a **Type** y ajuste el multiplicador **Self Weight Multiplier** en 1.
   - Agregar otra condición de carga estática llamada LIVE y asígnele el tipo LIVE como parámetro a **Type**.
2. Añada las mismas cargas correspondientes a la carga DEAD a la condición de carga LIVE. Esto significa que cada viga de la estructura tiene cargas idénticas para carga muerta y viva. (Puede revisar el ejemplo 1 para ver las instrucciones de como ingresar las una cargas)

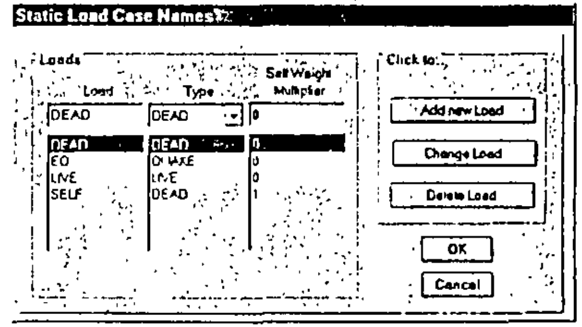


Figura 4-2 - Condiciones de Carga Estática

Definiendo un Grupo de Auto-Selección

La opción de Auto-Selección del SAP2000 es una forma muy efectiva de diseñar estructuras. Al definir un grupo de secciones denominado grupo de Auto-Selección, el programa puede diseñar cada elemento del pórtico escogiendo de entre las secciones especificadas en ese grupo. Por ejemplo se puede definir un grupo de Auto-Selección denominado COLUMNS con únicamente perfiles W14, y otro grupo llamado BEAM con perfiles W24. El programa de esta manera diseñará las secciones tipo COLUMNS empleando solamente secciones W14 y las secciones tipo BEAM empleando únicamente secciones tipo W24.

Lo primero que debemos hacer es definir un grupo de Auto-Selección que incluya únicamente secciones tipo columna. Esencialmente lo que estamos haciendo en esta etapa es darle al programa una lista de secciones de entre las cuales puede elegir al momento de diseñar los elementos del pórtico. El programa por su parte seleccionará la sección mas eficiente de entre ese grupo.

Una vez se haya concluido el diseño preliminar es momento de refinarlo, para ello los grupos BEAM y COLUMN serán reemplazados por secciones optimizadas elegidas de entre el grupo de Auto-Selección. Este proceso asignará a los elementos secciones que serán empleadas tanto en el análisis como en el diseño, esto hará mucho más fácil el cambio de secciones que necesiten ser modificadas.

Nota: ***La opción de Auto-Selección funciona solo en pórticos de acero.***

1. Desde el menú **Define** y seleccione la opción **Frame Sections**.
2. Importe a la plantilla **Frame Sections** todas las secciones de acero comprendidas entre W14x61 y W14x283.
   - Seleccione en la caja de diálogo la opción IMPORT I/WIDE FLANGE.

- Busque y seleccione la sección W14x283
- Manteniendo presionada la tecla SHIFT haga click con el botón izquierdo del mouse sobre W14x61 y presione el botón OK. Esta operación permite seleccionar todas las secciones entre la sección W14x283 y la sección W14x61.

3. Borre de plantilla **Frame Sections** cualquier sección que pudiera estar duplicada.

   Recuerde: *No es posible borrar una sección que este en uso. De esta manera el programa asegura que todos los elementos tienen asignada secciones existentes.*

4. Desde la plantilla **Frame Section** añada una sección **Auto Select**. Esta sección se ubicará en la parte inferior de la lista **Add**.

   - Cambie el nombre en **Auto Section Name** a COLUMN.
   - De la lista **Auto Selections** elija y remueva usando el botón **Remove** todas las secciones exceptuando los perfiles W14. Esto significa que todos los elementos que tengan una sección tipo COLUMN serán diseñados empleando alguno de los perfiles W14 de entre la lista Auto Selections (Vea Figura 4-3).

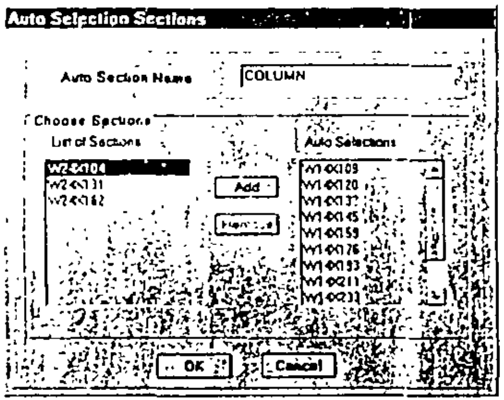


Figura 4-3 Definiendo el grupo de Auto-Selección COLUMN

5. Siguiendo las instrucciones dadas en los pasos 2 al 4:

   - Importe todas las secciones entre la W24x55 y la W24x162.
   - Asigne un grupo de Auto-Selección denominado BEAM con perfiles W24 únicamente

6. Finalmente, selecciones todos los elementos verticales del pórtico y asígnele la sección tipo COLUMN. Luego seleccione todos los elementos horizontales y asígnele la sección tipo BEAM (Vea el ejemplo 1 para las instrucciones de como asignar secciones a los elementos del pórtico).

   Nota: *Se puede por supuesto seleccionar una sección específica tanto para el diseño como para el análisis en lugar de emplear la opción de Auto-Selección. Para ello simplemente se necesita asignar a los elementos del pórtico una sección de acero adecuada y diseñarla de acuerdo a lo descrito en el ejemplo 1. Esta sección puede ser ya sea una sección definida por el usuario o bien una sección elegida de entre las secciones predeterminadas.*

Efectuando el Análisis

Una vez que se han ingresado los datos es tiempo de correr el modelo y revisar los resultados.

1. Grabe el modelo.
2. Asigne los parámetros para el diseño entrando al menú **Analyze** y seleccionando la opción **Set Options**
   - En la plantilla **Analysis Options** seleccione el tipo de análisis **Plane Frame** para reducir el tamaño de la solución y en consecuencia reducir el tiempo de análisis.
   - Marque la opción **Include P-Delta**.
   - Presione el botón SET P-DELTA PARAMETERS para ajustar los parámetros del análisis.
     - Asigne a **Maximum Iterations** el valor 10.
     - Incluya las condiciones de carga muerta DEAD y SELF, en la combinación P-Delta, ambas con factores de carga igual a 1.
     - Incluya la condición de carga LIVE con un factor de carga 1.

     Nota: ***Los factores de carga a emplearse deben ser los correspondientes a las combinaciones de carga que se usen en el diseño de la estructura y que produzcan los máximos efectos en la misma. Deben incluirse además el efecto de las cargas laterales.***

     - El resto de valores por defecto es aceptable.
     - Presione los botones OK para aceptar los cambios y cerrar las plantillas.
3. Entre al menú **Analyze** y seleccione la opción **Run** para analizar la estructura.

Nota: *Debido a que hemos asignado grupos de secciones y no secciones específicas para nuestro diseño, el SAP2000 elegirá las propiedades de las secciones más convenientes para generar la matriz de rigidez y efectuar el análisis estructural. Una vez que se ha efectuado el primer análisis y diseño, se puede instruir al programa para realizar el análisis con las propiedades de las secciones designadas.*

Diseño de Secciones

Una vez que se ha efectuado el análisis y revisado sus resultados, se podrán establecer los parámetros necesarios para el diseño en acero de la estructura.

Selección del Código de Diseño

La información resultante del análisis es empleada para efectuar una verificación de las secciones empleando un código de diseño

1. Desde el menú **Options** seleccione **Preferences**
2. En la plantilla **Preferences**, bajo el ítem **Steel** seleccione el código de diseño que desee emplear. En este caso se empleará el código AISC-ASD-89.
   - Emplee el mismo archivo **Section Properties file** que fue usado para importar las secciones de acero.

Combinaciones de Carga y Diseño

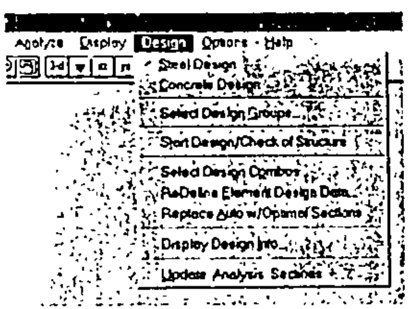


Figura 4-5 Opciones en el Menú de Diseño

Una vez que se ha seleccionado el código a utilizar en el diseño de los elementos se deben verificar las combinaciones de carga a emplearse.

1. Primeramente, en el menú **Design** asegúrese que exista la contraseña en el ítem **Steel Design**. Esto le indica al SAP2000 que debe efectuarse un diseño de secciones en acero.
2. En el menú Design elija la opción **Select Design Combos**.
   - Revise las combinaciones de carga generadas bajo la lista **Design Combos**, seleccionando las combinaciones y presionando el botón SHOW.
   - Si existen otras combinaciones de carga que desee incluir en el diseño, puede agregarlas usando para ello la opción Select Design Combos del menú **Define**.
3. Inicie el Diseño/Revisión de la estructura desde el menú **Design** seleccionando el ítem **Start** Design/Check of Structure.
   - Cada uno de los elementos será diseñado empleando la sección más eficiente de entre las correspondiente a su grupo de Auto-Selección
   - El SAP2000 mostrará automáticamente el porcentaje del nivel de esfuerzos existente en cada elemento con relación al máximo esfuerzo admisible.
   - Para mayor comodidad el programa asignará colores a cada uno de los elementos, los cuales muestran el nivel de esfuerzo presente en cada miembro usando una escala gráfica de colores/esfuerzos ubicada en la parte inferior de la ventana.

   Nota: ***Si desea verificar el diseño de un número determinado de elementos, podrá realizarlo simplemente con seleccionarlos y ejecutar la opción Start Design/Check of Structure.***

Revisión de los Resultados y Rediseño

Una vez efectuado el Diseño/Revisión es tiempo de verificar si los resultados son correctos. SAP2000 brinda al usuario una serie de herramientas para ello.

1. Haga click con el botón derecho del mouse sobre cualquier elemento para ver los resultados de su diseño. El elemento que haya sido seleccionado parpadeará para su fácil identificación.
2. En la plantilla **Steel Stress Check Information** encontrará una lista de las combinaciones de carga empleadas para verificar la sección en varios puntos a lo largo del elemento. (Vea Figura 4-5)
   - Una de las combinaciones de carga estará resaltada cuando abra esta plantilla. Esta es la combinación que controla el diseño del elemento.
   - Junto a cada combinación de carga hay un indicador de la ubicación a lo largo del elemento donde fue efectuada la verificación, seguida por la relación de esfuerzo para la interacción de momento y cortante.

Sugerencia : *Se puede cambiar el número de puntos a lo largo del elemento en los cuales las fuerzas de sección son reportadas. Para ello seleccione los elementos y desde el menú Assign elija la opción Frame Output Segments para cambiar el número de segmentos. Es necesario ejecutar nuevamente el análisis del modelo para obtener los resultados*

3. Seleccionando cualesquiera de las combinaciones de carga y presionando el botón **Details** se mostrarán los resultados del análisis para ese elemento así como las ecuaciones que gobiernan su diseño de acuerdo al código empleado. (Ver Figura 4-6)

4. Al presionar el botón **ReDesign** se presentará la plantilla **Element Overwrite Assignments.** En esta plantilla se puede elegir de entre varias opciones:

   Nota: *Si se efectúan cámbios en la plantilla Element Overwrite Assignments empleando el botón ReDesign, será necesario presionar el botón Refresh Window de la barra de herramientas para ver los resultados del diseño actualizados en la ventana activa .*

   - Seleccionar otra sección para ver el cambio en los esfuerzos en el elemento.

   Nota: En el modo Auto-Selección, esta sección puede emplearse para ensamblar una nueva matriz de rigidez si se elige la opción "Update Analysis Sections". Esto último se llevará acabo una vez que se ejecute nuevamente el Diseño/Revisión.

   - Clasificar los elementos por tipo **Moment Resisting Element** o **Brace.**
   - Sobrescribir los factores de diseño tales como longitud efectiva y la relación de longitud no arriostrada.
   - Elija la opción **Overwrite Allowable Stresses** para sobrescribir los esfuerzos admisibles empleados en el diseño de la sección.
   - Cuando halla terminado de modificar los parámetros de diseño presione el botón **OK.**

   Nota: ***Al cambiar la información de la plantilla ReDesign, el SAP2000 automáticamente recalculará los esfuerzos de diseño de acuerdo a la nueva información y actualizará la información en la plantilla Steel Stress Check Information. Para más instrucciones sobre como actualizar las secciones para el análisis refiérase a la sección "Re-Analizando".***

5. Para usar la sección elegida en el Re-Diseño en el siguiente análisis estructural, es necesario entrar al menú **Design** y seleccionar la opción **Update Analysis Sections.** Esta opción reemplaza las secciones empleadas inicialmente para formar la matriz de rigidez de la estructura, por las nuevas secciones dándonos mayor precisión en los cálculos.

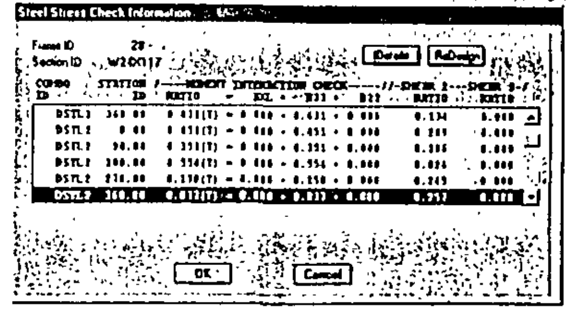


Figura 4-5 Verificación de Esfuerzos en las secciones de Acero para las Combinaciones de Carga Especificadas

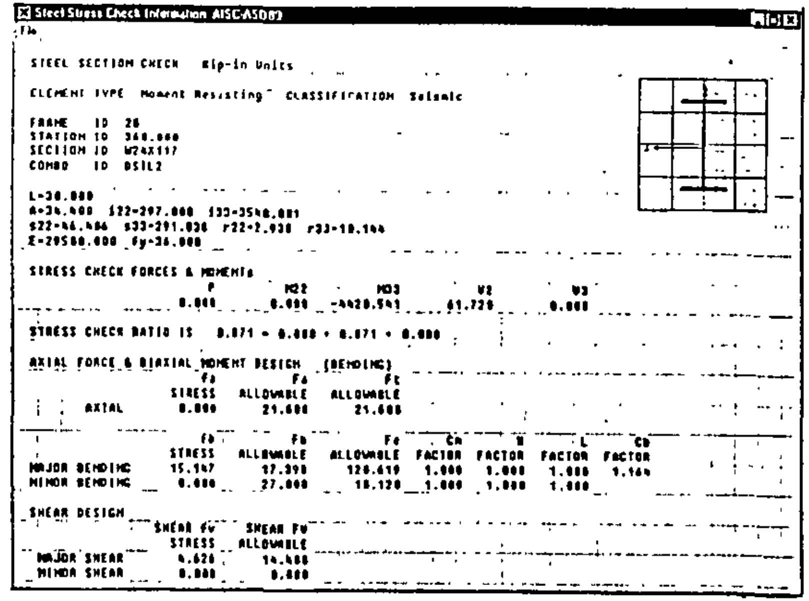


Figura 4-6 Información detallada del diseño en acero de un elemento tipo viga

6 También se pueden observar los resultado del diseño de manera gráfica en la pantalla. Para ello ingrese al menú **Design** y seleccione la opción **Display Design Info**. Los resultados se mostrarán en forma gráfica en la parte inferior derecha de los elementos del pórtico.

Nota: ***Las secciones empleadas en el análisis estructural son mostradas en la parte superior izquierda de cada elemento. Por otra parte, toda la información correspondiente al diseño de los elementos es mostrada en la parte inferior derecha de los mismos.***

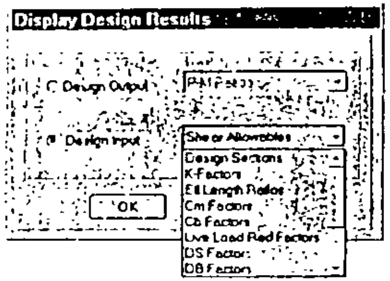


Figura 4-7 Plantilla de salida resultados

7. Se pueden imprimir los resultados del diseño ingresando al menú **File** y seleccionando la opción **Print Design Tables**. Para imprimir los resultados de un número limitado de elementos, primeramente selecciónelos y enseguida efectúe la opción **Print Design Tables**.

Edición de las Propiedades de la Sección

En la Figura 4-1 podrá observar que las vigas del pórtico tienen cargas concentradas ubicadas a los tercio del claro. Si asumimos que dichas cargas son transmitidas por otros miembros, entonces podemos considerar que las vigas tienen soporte lateral en los puntos de carga. El modelo tal como ha sido creado no toma en cuenta este hecho, por lo que las vigas están sobredimensionadas. Para diseñar las vigas más eficientemente se deben editar sus propiedades de diseño.

1. Seleccione todas las vigas de la estructura. Para ello puede usar el modo de selección **Intersecting Line Select Mode**

2. Desde el menú **Design** escoja la opción **ReDefine Element Design Data**.

   - En la plantilla **Element Overwrite Assignments** seleccione el ítem **Unbraced Length Ratio, L22** y asígnele el valor 0.33. Esto introduce un soporte lateral a las vigas a cada 1/3 del claro (arriostramiento contra el pandeo en el eje local 1-2), en lugar de la opción por defecto que corresponde a elementos arriostrados solo en los extremos. (Vea Figura 4-8)

   - Presione el botón OK luego de ingresar la nueva información. El SAP2000 automáticamente efectuará el Diseño/Revisión del modelo actualizado.

3. Podrá apreciar ahora que las dimensiones de las secciones de las vigas son menores.

Sugerencia: ***Se puede conocer el peso de la estructura obteniendo la suma de cargas nodales de los elementos del grupo BASE SHEAR. Esta es una manera rápida de ver si los cambios efectuados producen una estructura más eficiente. También se puede obtener el peso total de la estructura empleando el archivo filename.EKO.***

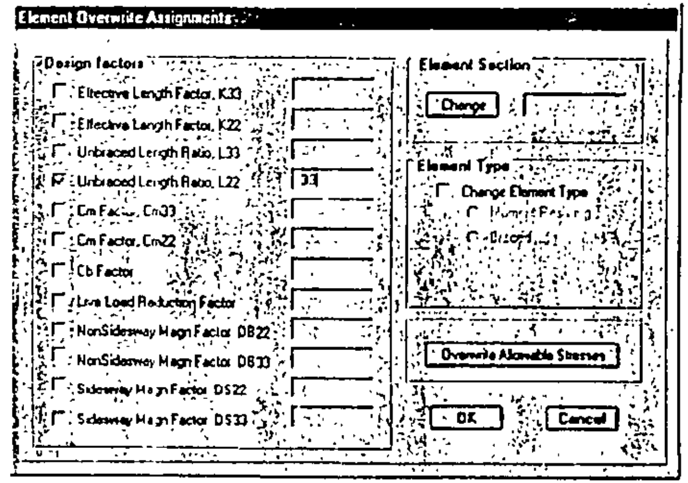


Figura 4-8 Plantilla de sobre-escritura de parámetros de diseño de elementos tipo viga

Re-Analizando

El primer análisis empleó propiedades de sección aproximadas para genera la matriz de rigidez de la estructura. Es por ello que el modelo se debe reanalizar a manera de un proceso iterativo para asegurarnos que el análisis toma en cuenta las propiedades y secciones actualizadas de los elementos.

1 Una vez que se halla terminado de modificar las secciones estructurales que se van a emplear, ingrese al menú Design y elija la opción Update Analysis Sections. Luego efectúe nuevamente el análisis empleando las últimas propiedades de sección.

2 Lleve acabo nuevamente el diseño de los elementos para ver si hay cambios.

3 Una vez que se encuentre satisfecho con las secciones elegidas, ingrese al menú Design y elija la opción Replace Auto w/ Optimal Sections. Esta opción asigna las secciones definitivas ya sean las óptimas o bien aquellas seleccionadas por el usuario tanto para el análisis como para el diseño, y reemplaza las propiedades de sección preliminares tomadas de los grupos de auto selección BEAM y COLUMN.

Diseño de acuerdo al LRFD

La metodología empleada por el LRFD es esencialmente la misma que usa el ASD. Sin embargo las combinaciones de carga así como la ecuaciones de verificación de los elementos son efectuadas empleando el código LRFD, por lo que los resultados e información resultante es distinta. Para efectuar el Diseño/Revisión de acuerdo al código LRFD, es necesario cambiar algunos parámetros de entrada.

1 Ingrese los nuevos factores de carga para el análisis P-Delta.

2 Seleccione de la plantilla Preferences el código de diseño en acero AISC-LRFD93.

3 Rediseñe las secciones de acero

Opciones Avanzadas

Definición de Grupos de Elementos para el Diseño

Algunas veces puede encontrará útil esta opción al diseñar elementos en estructuras aporticadas. Esta opción permite diseñar todos los elementos de un grupo usando únicamente una sección. La ventaja de este método de diseño es que reduce el número de secciones diferentes a emplearse. Por ejemplo se pueden agrupar las columnas o las vigas de dos o tres pisos del pórtico dentro de un mismo grupo de diseño, permitiéndonos usar una sola sección para dicho conjunto de elementos.

1. Reasigne las secciones del tipo Auto-Selección a los elementos del pórtico.
2. Agrupe los elementos del 3er piso hacia abajo en un grupo denominado BOTTOM.
3. Agrupe los elementos entre los pisos 3 y 5 en un grupo llamado MIDDLE.
4. Asigne los elementos restante al grupo TOP.

5. Efectúe nuevamente el análisis del modelo.
6. En el menú Design seleccione la opción **Select Design Group**. En esta parte se le indica al programa que se diseñe un grupo de elementos empleando la sección más ligera que satisfaga los requerimientos y esfuerzos admisibles en todos los elementos.
   - Incluya en la lista Design Groups los grupos de elementos TOP, MIDDLE y BOTTOM. Al hacer esto indicamos que estos grupos serán diseñados con la sección más eficiente de las secciones del grupo Auto-Selección.

     Nota: ***Si no hay grupos en la lista "Design Group list", cada uno de los elementos de la estructura serán diseñados individualmente.***
   - Cuando presione el botón OK , El SAP2000 automáticamente diseñará las secciones de acero y mostrará los resultados en la ventana activa
7. Compare los resultados del diseño anteriormente efectuado con el diseño por grupos para ver como afecta este hecho en las secciones seleccionadas.

Zonas Rígidas

La estructura que hemos venido estudiando, ha sido analizada y diseñada considerando que los elementos se extienden completamente de nudo a nudo, sin tomar en cuenta las dimensiones propias de las secciones transversales de los elementos. Si bien es cierto que ésta no es una mala consideración, el SAP2000 permite efectuar análisis aun más precisos mediante la introducción de zonas rígidas en el modelo. Las zonas rígidas definen una región en la conexión entre viga y columna, en la cual los elementos no sufren deformaciones por flexión. Se genera así esencialmente una zona rígida en la conexión. Esta área puede ser tan grande como el usuario especifique, pero usualmente se considera igual al peralte del miembro (o una fracción del mismo) al que se esta llegando en ese nudo

1. Seleccione todos los elementos del pórtico.
2. Del menú Assign seleccione la opción **Frame... End Offsets**.
   - En la plantilla **End Offset** seleccione la opción **Update Lengths From Current Connectivity**. Esta opción hace que el programa calcule automáticamente las dimensiones de las zonas rígidas a considerarse en cada nudo.
   - Ingrese el valor 1 para **Rigid Zone Factor**. Esto significa que el 100% de la "longitud potencial" de zona rígida deberá considerarse en el análisis.
   - Presione el botón OK.
3. Si se especifica la opción **Element Shrink** de la plantilla **Set Elements**, y se observa en la pantalla activa, se podrá apreciar una serie de líneas en cada nudo que indican la asignación de zonas rígidas en los elementos.

Recuerde : *Se necesita restablecer la opción End Offsets cada vez que las secciones de los elementos sean modificadas.*

Nota: *Los momentos y cortantes en las vigas y columnas van a ser ligeramente diferentes en aquellos casos en los que no se toman en cuenta las zonas rígidas. Esto se debe a que la introducción de las mismas reduce la longitud flexible de los elementos.*

Comentarios Finales

Las herramientas de diseño del SAP2000 son muy útiles en el diseño de estructuras aporticadas. Sin embargo hay algunos puntos que se deben tener presentes

1. Asegúrese que toda la información de diseño sea correcta. Los valores por defecto que usa el programa no son necesariamente los correctos (p ej. K y Longitudes no Arriostradas de los elementos). Se puede usar la plantilla **Display Design Results** para ver esta información en los elementos del pórtico De manera conveniente, es posible apreciar el análisis de las secciones al mismo tiempo que la información del diseño.
2. Verifique que las combinaciones de carga de diseño que el programa ha proporcionado sean las correctas y adecuadas para el tipo de estructura en particular que se este analizando. Sin no lo son, añada las combinaciones de carga que desea utilizar en el diseño
3. Verifique los resultados del diseño en puntos claves, para asegurarse que los resultados del diseño guardan relación con los resultados esperados.
4. Verifique que los factores de carga en el análisis P-Delta son los correctos.
5. Rediseñe la estructura toda vez que efectúe cambios en el modelo. Esto permite ver si las secciones empleadas son aún aceptables.
6. Emplee grupos de elementos para determinar el peso total de la estructura. (Revise el Ejemplo 1 para tener instrucciones de como efectuar este paso)
7. Emplee grupos de elementos en el diseño para reducir el número de las diferentes secciones a utilizarse en la estructura.
8. El archivo *filename*.EKO contiene la información del peso total de cada uno de las secciones (perfiles) empleadas en el diseño. Esta información nos permite estimar costos de una manera preliminar.

Apéndice A – Descripción de los Iconos de la Barra de Herramientas

Icono	Nombre del Control	Permite
	New Model	Iniciar un nuevo modelo
	Open *.SDB file	Abrir un archivo existente del SAP2000.
	Save Model	Grabar el modelo activo.
	Undo	Deshacer el último cambio.
	Redo	Revierte el último Deshacer.
	Refresh Window	Regenera la ventana activa con información actualizada
	Lock/Unlock Model	Protege el modelo contra cambios de datos.
	Run Analysis	Efectúa el Análisis.
	Zoom	Zoom en la estructura del área determinada con el mouse.
	Restore Full View	Restaura la vista total del modelo.
	Restore Previous View	Restaura la vista anterior del modelo.
	Zoom In	Zoom in en el modelo. (Acercamiento)
	Zoom Out	Zoom out en el modelo. (Alejamiento)
	Pan	Mueve dinámicamente la estructura en cualquier dirección.
	Show 3-d view	Muestra vista 3-d del modelo.
	Show 2-d View of X-Y/r-Θ Plane	Vista 2-d del modelo paralela al plano X-Y/r-Θ.
	Show 2-d View of X-Z/r-Z Plane	Vista 2-d del modelo paralela al plano X-Z/r-Z .
	Show 2-d View of Y-Z/Θ-Z Plane	Vista 2-d del modelo paralela al plano X-Z o una vista desarrollada del plano r-Z plane.
	Perspective Toggle	Muestra vista 3-d en perspectiva.
	Shrink Elements	Contrae los elementos para facilitar la visualización de la conectividad.
	Set Element	Ajusta la visibilidad de los elementos y sus propiedades
	Up One Gridline	Visualiza el siguiente nivel superior en una malla en la vista en planta 2-d .
	Down One Gridline	Visualiza el siguiente nivel inferior en una malla en la vista en planta 2-d

Apéndice B – Descripción de Iconos de la Barra de Herramientas Flotante

Icono	Nombre del Control	Permite
	Pointer Tool	Selecciona elementos individualmente o en cajas.
	Select All	Selecciona todos los elementos en un gráfico.
	Restore Previous Selection	Restaura elementos previamente seleccionados.
	Clear Selection	Libera todos los elementos seleccionados.
	Set Intersecting Line Select Mode	Selecciona los elementos interceptados por una línea.
	Reshape Element	Mueve elementos tomándolos en su parte central y redimensionalos seleccionando sus extremos
	Add Special Joint	Añade manualmente un nudo.
	Draw Frame Element	Dibuja un elemento tipo frame al definir la ubicación de sus nudos extremos.
	Draw Shell Element	Dibuja un elemento tipo shell al definir la ubicación de sus esquinas.
	Quick Draw Frame Element	Dibuja un elemento tipo frame usando una malla.
	Quick Draw Shell Element	Dibuja un elemento tipo shell usando una malla.
	Assign Joint Restraints	Asigna restricciones en los nudos
	Assign Frame Sections	Asigna secciones y materiales a elementos frame
	Assign Shell Sections	Asigna secciones y materiales a elementos shell.
	Assign Joint Load	Asigna cargas concentradas nodales.
	Assign Frame Span Loading	Asigna cargas en elementos tipo frame.
	Assign Shell Uniform Loading	Asigna cargas en elementos tipo shell.
	Show Undeformed Shape	Muestra la geometría original del modelo.
	Display Static Deformed Shape	Muestra la geometría deformada de la estructura.
	Display Mode Shapes	Muestra formas de modo y periodos de vibración.
	Display Element Forces/Stresses	Muestra resultados del análisis – Fuerzas y Esfuerzos.
	Set Output Table Mode	Muestra tablas con los resultados del análisis.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 WEB TUTORIAL 1

DETAILED TUTORIAL INCLUDING PUSHOVER ANALYSIS

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col Centro, México D.F., APDO Postal M-2285 • Tels 5521 4021 al 24, 5623.2910 y 5623.2971 • Fax 5510 0573

SAP2000®

Integrated
Finite Elements Analysis
and
Design of Structures

SAP2000 Web Tutorial 1

DETAILED TUTORIAL INCLUDING
PUSHOVER ANALYSIS

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

Issue Date: June 1998
Revision Number : 0
Revision Date: N/A

COPYRIGHT

Further information may be obtained from:

Computers and Structures, Inc
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *support@csiberkeley.com*
web: *www.csiberkeley.com*

Table Of Contents

A. Introduction **1**

B. Description of Building Model **3**

C. Tutorial Part 1 - Creating, Analyzing and Designing the Basic Model **8**
1a. Setting up the model geometry starting from a template 8
1b. Setting up the model geometry starting from scratch, including restraints 18
2. Relabeling joint, frame and shell elements 28
3. Defining material properties 30
4. Defining frame sections 32
5. Defining shell sections 34
6. Assigning groups 35
7. Assigning frame sections 41
8. Assigning shell sections 47
9. Assigning frame end releases 48
10. Defining static load cases 51
11. Assigning frame static loads 53
12. Assigning shell static loads 61
13. Assigning joint static loads 63
14. Assigning joint masses 67
15. Assigning diaphragm constraints 70
16. Static and dynamic analysis (not pushover) 71
17. Reviewing mode shapes 74
18. Reviewing deformed shapes 75
19. Reviewing element forces and stresses 77
20. Performing a steel design stress check 81

D. Tutorial Part 2 - Pushover Analysis **86**
1. Defining hinge properties (pushover) 86
2. Assigning hinge properties (pushover) 90
3. Viewing generated hinge properties (pushover) 94
4. Defining static pushover cases 98
5. Running the pushover analysis 105
6. Displaying the pushover deformed shape and the sequence of hinge formation 106
7. Displaying frame element forces at each step of the pushover 110
8. Displaying the pushover and capacity spectrum curves 112

E. Final Comments **123**

DISCLAIMER

A. Introduction

This tutorial is quite detailed. It is intended to introduce and demonstrate many of the capabilites of SAP2000. Because we are trying to demonstrate as many different capabilities as reasonable, the example problem is not necessarily created and the results are not necessarily reviewed in the most efficient and expedient manner. Often with computer programs, what is efficient for one person may not be the best method for the next person. It is assumed that once introduced to the SAP2000 capabilities and methods in this tutorial, users will decide which methods work best for them in their particular circumstances. Following is an outline of this tutorial (see the Table of Contents for a more complete outline):

A. Introduction

B. Description of Building Model

C. Tutorial Part 1 - Creating and Analyzing the Basic Model

D. Tutorial Part 2 - Pushover Analysis

E. Discussion of Additional Pushover Cases

If you are not interested in the pushover portion of the tutorial, you can skip parts D and E.

If you are only interested in the pushover tutorial, then you can read parts A and B, skip part C, and then open the already created model (without the pushover data) named Sapwb01c.sdb, which is supplied with this tutorial, and start with Step D.

Note: You must have SAP2000 Version 6.20 or later to read files Sapwb01c.sdb and Sapwb01d.sdb. These files are not compatible with earlier versions of SAP2000.

Finally, if you are not interested in working through the tutorial at this time, but want to see the results of the pushover analysis, then you can read parts A and B, skip part C and the first five steps of part D, and then open the already created model (with the pushover data) named Sapwb01d.sdb, which is supplied with this tutorial. You should then run the linear static and dynamic analysis by clicking Run on the Analyze menu. Next select Run Static Pushover from the Analysis menu to perform the pushover analysis. The results are now ready for viewing. See steps 6, 7 and 8 in part D for information on viewing results. See part E for additional discussion of results. Note that for real problems (as contrasted with tutorial problems) you should always run and review the results of a basic static analysis (and maybe also dynamic analysis, at least for mode shapes) to verify that your model is behaving as you intend.

SAP2000 has extensive online help that can help answer many of your questions. One of the most useful ways to access the online help is to press the F1 key on the keyboard from within most dialog boxes. Pressing the F1 key will bring up context sensitive help pertaining to the dialog box that is open. You may find it useful to use this option throughout this tutorial to get more information.

If you require further technical assistance concerning this tutorial, or other aspects of SAP2000, you can contact CSI by phone at (510) 845-2177, or by e-mail at support@csiberkeley.com.

B. Description of Building Model

The example problem for this tutorial is a steel-frame building two bays wide by two bays deep, and two stories high. The plans and elevation in Figure B-1 show the basic dimensions and member sizes for the model. The building has a moment frame lateral force-resisting system in the X-direction and a braced frame lateral force-resisting system in the Y-direction. All steel is Fy = 36 ksi. The second floor is concrete over metal deck, and is assumed to be a rigid diaphragm in the model. The roof deck has no concrete, so it is not modeled as rigid.

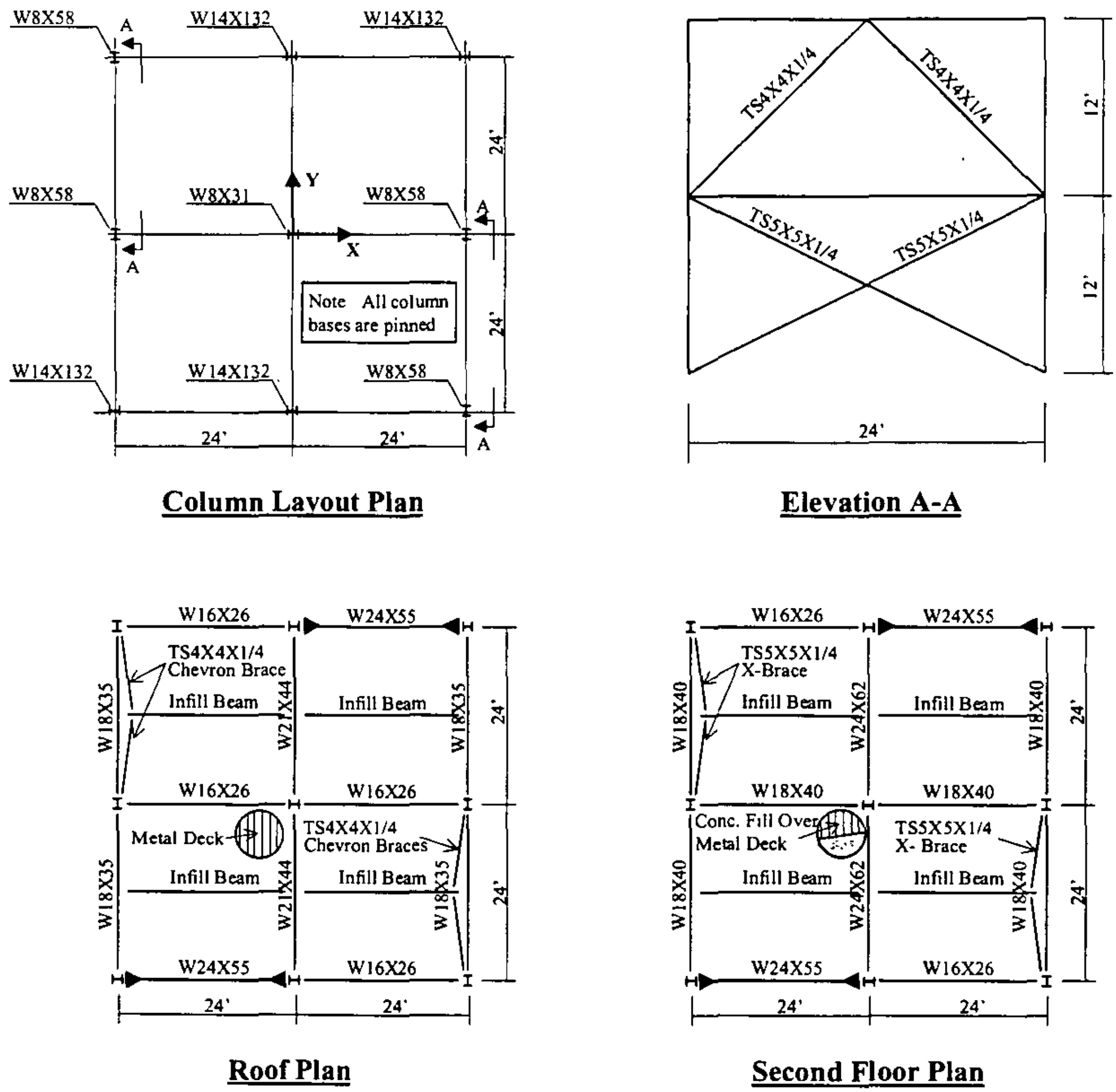
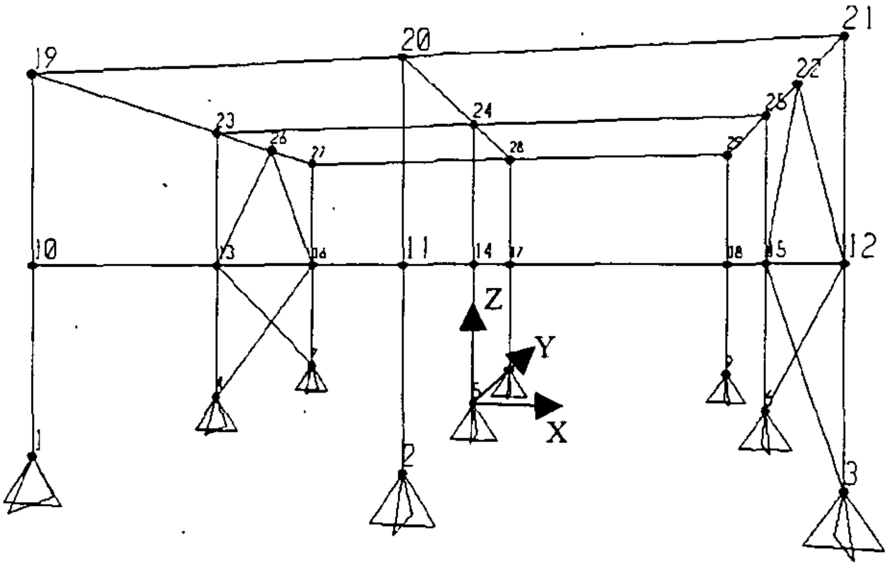
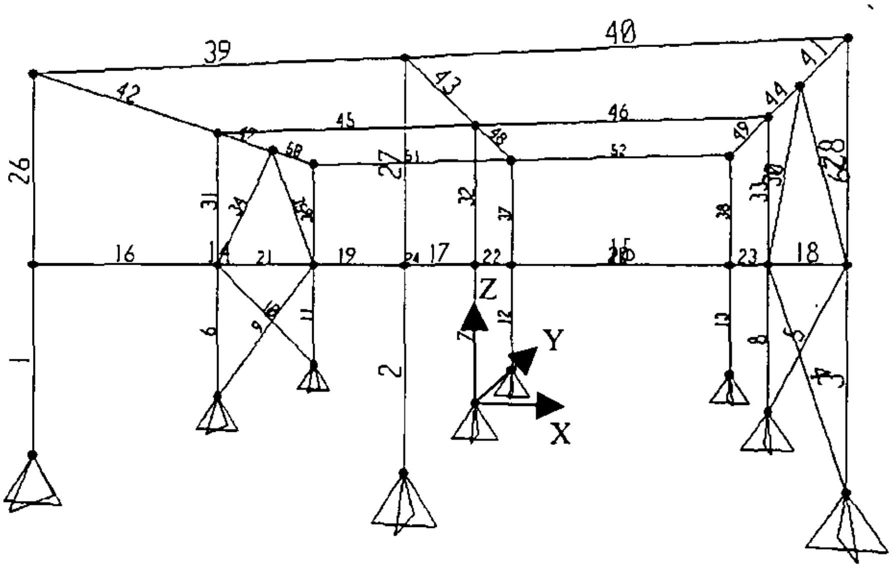


Figure B-1: Example Building Plans and Braced Frame Elevation

Figure B-2 shows perspective views of the computer model that include joint labels and frame element labels. Figure B-3 includes plan views of the computer model with frame element labels. Note that the infill beams are not specifically included in the computer model. These figures may be useful for reference when you are working through the tutorial.

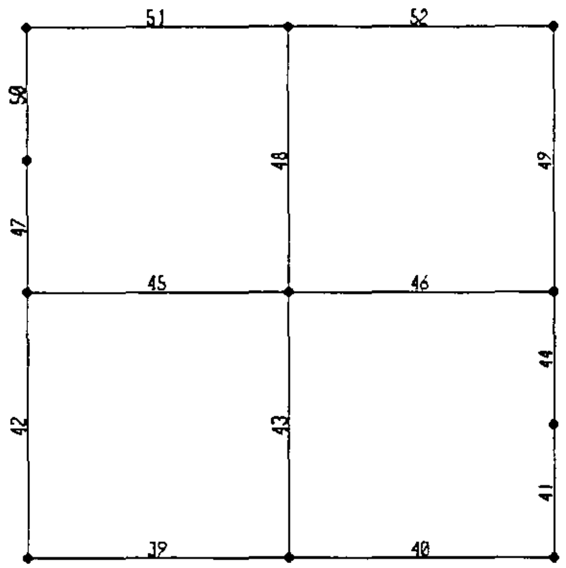


Perspective View Showing Joint Labels

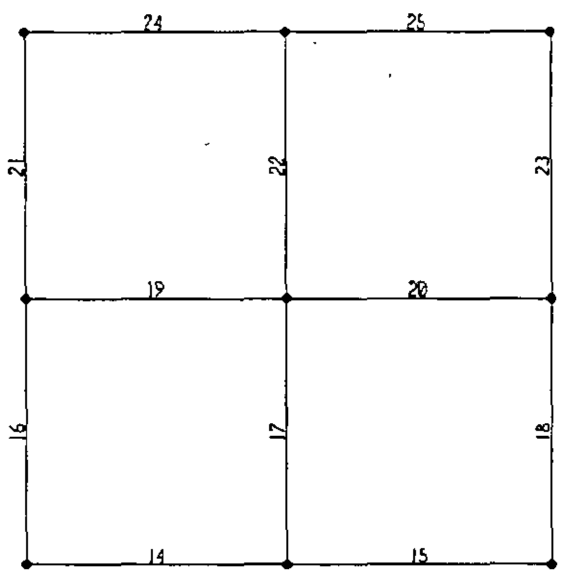


Perspective View Showing Frame Element Labels

Figure B-2: Views of Computer Model Showing Joint and Frame Element Labels



Roof Level



Second Floor Level

Figure B-3: Plan Views of Computer Model Showing Frame Element Labels

The following assumptions are used for dead and live loads:

Roof:	DL = 40 psf LL = 20 psf
Second Floor:	DL = 80 psf LL = 50 psf
Perimeter Wall:	DL = 20 psf

These loads are assumed to include the self-weight of the structural elements. Figure B-4 shows the beam span loads that are applied to each beam in the SAP2000 model.

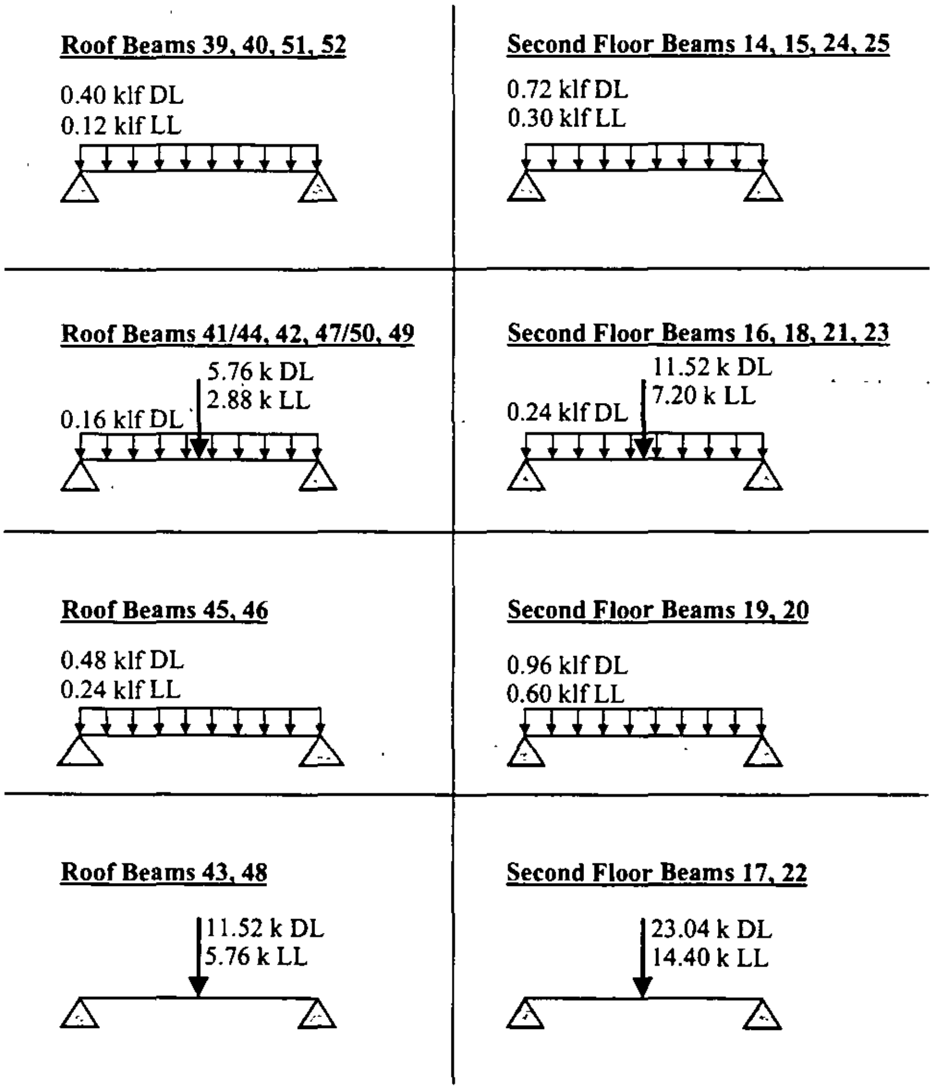


Figure B-4: Beam Span Loads Used In SAP2000 Model

For this model, all mass is input at the joints. Table B-1 defines the joint masses used in the model. The bottom portion of the table defines the masses, the top portion of the table defines which joints are referred to as the center, corner and edge joints. Note that no mass is assigned to joints 22 and 26 at the roof level. These are the joints at the top of the chevron braces.

Joint Labels at Center, Edge and Corner Joints			
Level	**Center Joint**	**Edge Joints**	**Corner Joints**
Roof	24	20, 23, 25, 28	19, 21, 27, 29
Second	14	11, 13, 15, 17	10, 12, 16, 18
Joint Masses at Center, Edge and Corner Joints (k-sec^2 / ft)			
Level	**Center Joint**	**Edge Joints**	**Corner Joints**
Roof	0.72	0.48	0.30
Second	1.45	0.90	0.55

Table B-1: Joint Masses Used In Computer Model

The lateral earthquake loads are assumed to be 17 kips (0.0074ksf) at the roof level and 16 kips at the second level for the X-direction (moment frame direction), and 26 kips (0.0113 ksf) at the roof level and 23 kips at the second level for the Y-direction (braced frame direction). These forces are assigned as shell static uniform loads at the roof level and as joint loads at the second level. The second level forces are broken down into joint loads as shown in Table B-2.

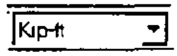
Joint Loads at Center, Edge and Corner Joints (k) For Second Level			
Earthquake Direction	**Center Joint** 14	**Edge Joints** 11, 13, 15, 17	**Corner Joints** 10, 12, 16, 18
EQX	2.67	2.00	1.33
EQY	3.83	2.88	1.92

Table B-2: Joint Forces Used In Computer Model For Static Earthquake Loading At Second Floor Level In X and Y Direction

C. Tutorial Part 1 - Creating, Analyzing and Designing the Basic Model

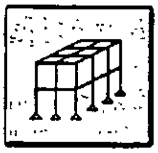
This tutorial steps through setting up the model geometry both from a built-in template (step 1a), and from scratch (step 1b). It is not necessary to do both of these steps, in order to complete the tutorial; one or the other is sufficient. However, it may be helpful to see the process and techniques used in completing both steps. We recommend that to obtain additional information during the course of this tutorial you refer liberally to the online help available in SAP2000 by pressing the F1 key from within almost any dialog box to obtain context-sensitive help.

Step 1a: Setting Up the Model Geometry From a Template

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model from Template....** This displays the Model Templates dialog box.

3. In this dialog box:

   - Click on the Space Frame template.  This will display the Space Frame dialog box (see Figure C-1).

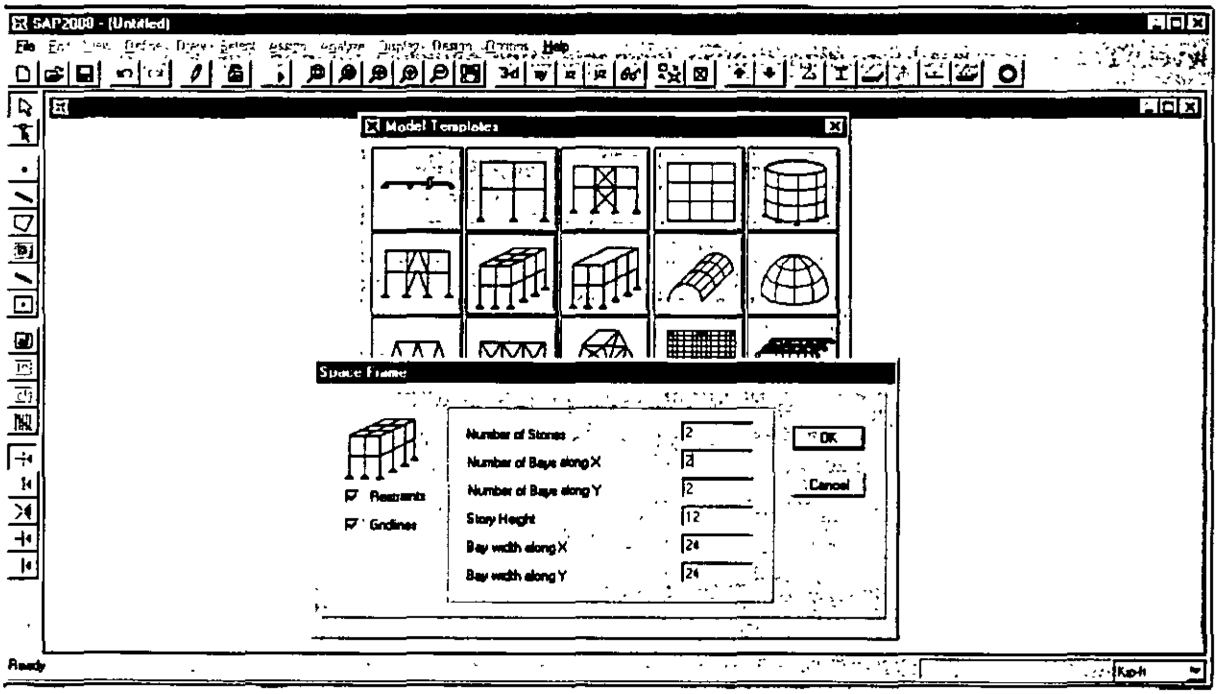


Figure C-1: Space Frame Dialog Box

   - In this dialog box:

     ✓ Change the Number of Bays along X to 2.

✓ Check the Restraints box if it is not already checked.

✓ Check the Gridlines box if it is not already checked.

✓ Accept the remainder of the default values.

✓ Click the **OK** button.

4. The screen will refresh and display a 3-D and a 2-D (XY Plane @ Z=24) view of the model in vertically tiled adjoining windows.

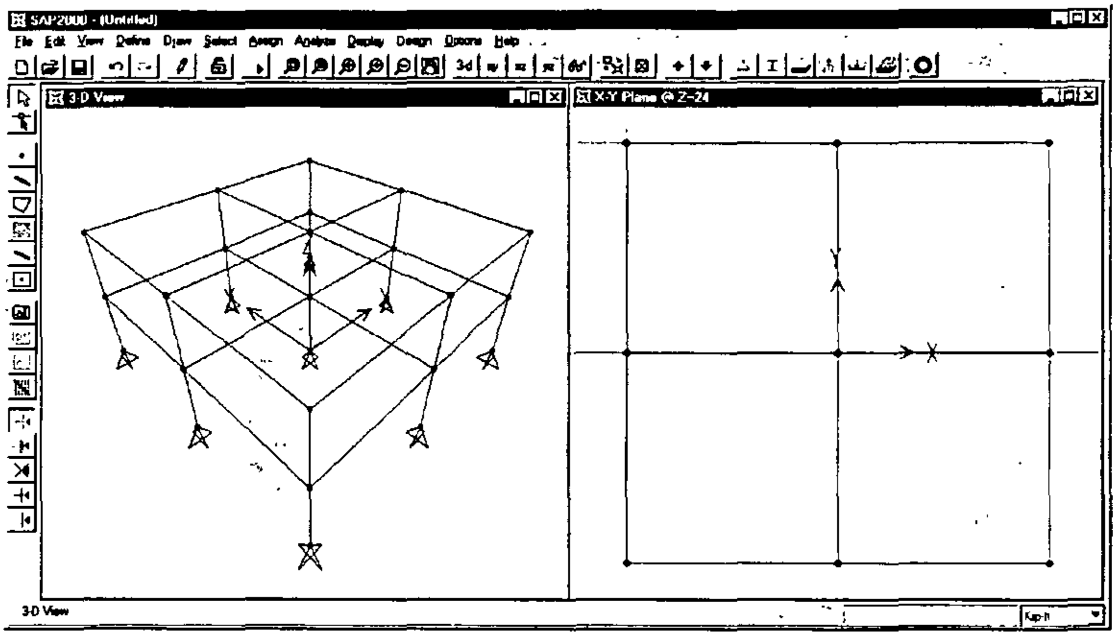


Figure C-2: Initial Screen From Space Frame Template

5. Note that the default restraints provided when the Restraints box is checked in the Space Frame dialog box are pinned supports.

6. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

7. Click the **yz 2D View** button yz on the main toolbar to change the view to an elevation in the YZ plane. Note that the title of the window reads YZ Plane @ X=24. This same title also occurs on the left-hand side of the status bar at the bottom of the SAP2000 window.

8. Click the roof level beam on the left side of the elevation to select it. From the **Edit** menu, click **Divide Frames...** to display the Divide Selected Frames dialog box.

9. Accept the default values in this dialog box and click the **OK** button. The roof beam is divided into two beams, thus providing a node for the top of the chevron brace.

10. Click the **Draw Frame Element** button on the side toolbar, or select **Draw Frame Element** from the **Draw** menu. The program is now in Draw Mode.

Note: If you hold the pointer over a toolbar button for a few seconds, a text box including the name of the button will appear.

11. Make sure that the **Snap to Joints and Grid Points** button on the side tool bar is depressed. Place the mouse pointer on the joint labeled "A" in Figure C-3 and click on the left mouse button. Note that when the mouse pointer is near the joint, it snaps to the joint, and a text box that says "Grid Intersection" appears. This is the effect of the **Snap to Joints and Grid Points** feature.

Note: Other snap options included in SAP2000 include Snap to Midpoints and Ends, Snap to Element Intersections, Snap to Perpendicular, and Snap to Lines and Edges. The buttons for these features are located on the side toolbar just below the ***Snap to Joints and Grid Points*** *button. Note that all of the snap features can also be accessed through the* ***Snap To...*** *option on the* ***Draw*** *menu. For more information on the snap capabilities of SAP2000, refer to the topic titled Snap Tools in the SAP2000 online help. Click on the* ***Help*** *menu and select the* ***Search for Help on...*** *option to access the online help in SAP2000.*

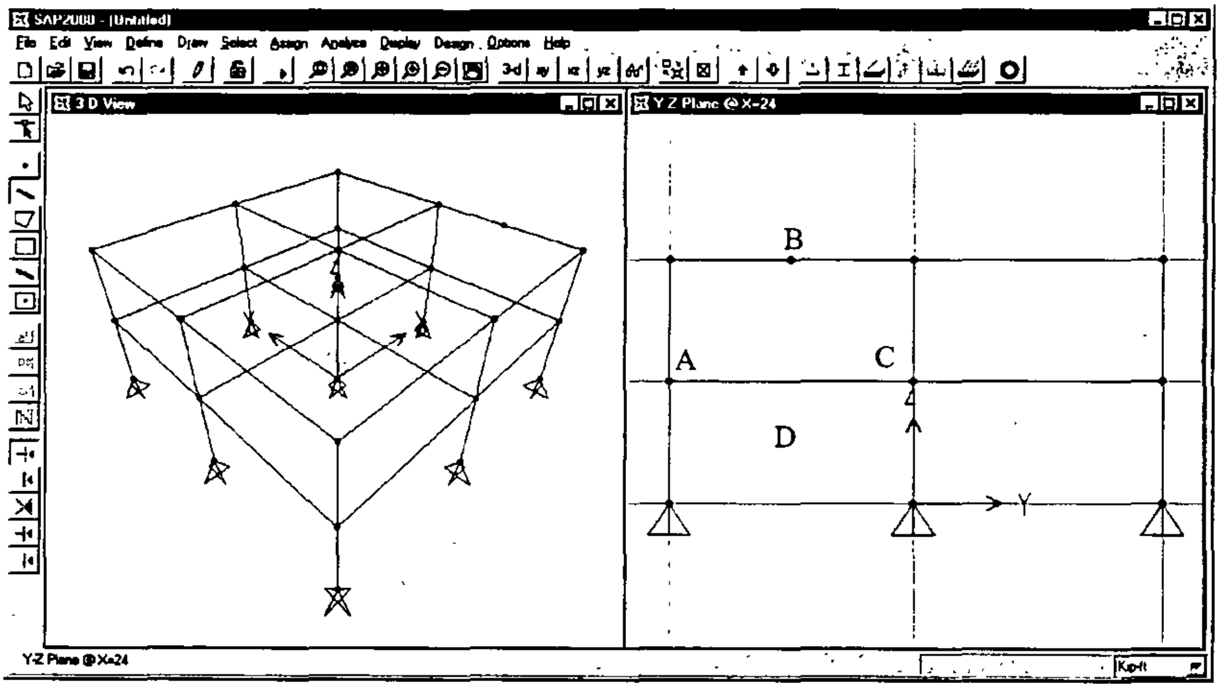


Figure C-3: Joint Labels For Drawing Chevron Brace

12. Place the mouse pointer near the joint labeled "B" in Figure C-3 and note that in this case the Snap To option text box just says "Point" because the joint does not occur at a grid intersection. Click on the left mouse button to draw the chevron brace element.

    Note: If you wanted to, at this point you could just click on point C in Figure C-3 to draw the second brace element. If we were to do that, the start point for the brace would be at the top (point B) and the bottom point would be at the bottom (point C). Though not necessarily a problem, this would be inconsistent with how we input the first brace. In this example we will opt for consistency.

    Note: If you wanted to, at this point you could move the mouse pointer into the 3D View window, and select the second joint for the next frame element there. Try moving the mouse pointer over the 3D View window, but for this example don't actually click to define the second frame member.

13. Press the Enter key on the keyboard to stop drawing the second frame member. Note that the **Draw Frame Element** button is still depressed, i.e., the program is still in Draw Mode and ready to draw another frame element.

14. Click on the joint labeled "C" and then the joint labeled "B" in Figure C-3, and then press the Enter key on the keyboard to draw the second chevron brace member.

15. Click the **Quick Draw Frame Element** button on the side toolbar. Then click in the area labeled "D" in Figure C-3 to enter the X-braces. The model now appears as shown in Figure C-4.

    Note: The X-braces could also have been entered using the same technique that was employed for the chevron braces.

    *Note: Both the **Draw Frame Element** and the **Quick Draw Frame Element** options are also available on the **Draw** menu.*

    *Note: The **Quick Draw Frame Element** option works two different ways. You can click on a grid segment to quickly draw a single frame element between the two adjacent perpendicular grid lines. Alternatively, you can click in a space bounded by four grid lines to quickly draw a cross brace as was done here.*

16 Click the **Down One Gridline** button on the main toolbar twice to display the YZ elevation at X=-24. Note the window title changes to Y-Z Plane @ X=-24. The model appears as shown in Figure C-5.

17. We will use a different method to locate these chevron braces. Click the **Snap to Midpoints and Ends** button to activate this snapping option. Click the **Draw Frame Element** button on the side toolbar, or select **Draw Frame Element** from the **Draw** menu. The program is now in Draw Mode.

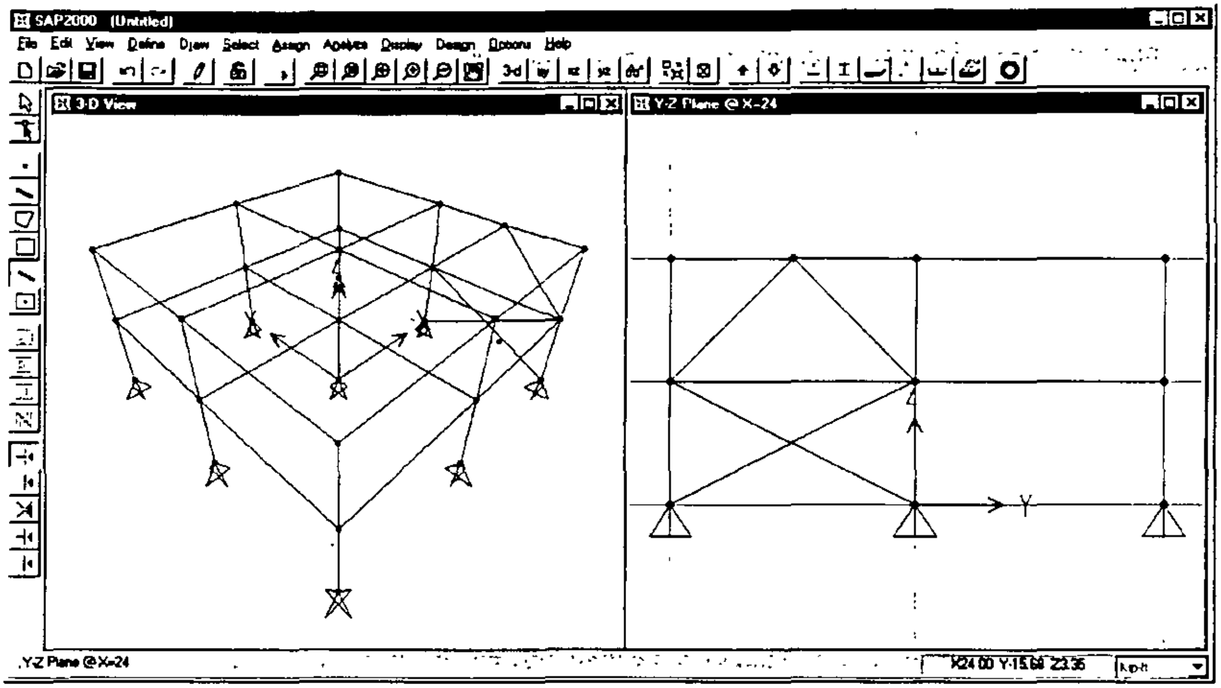


Figure C-4: Model After Inputting First Set of Braces

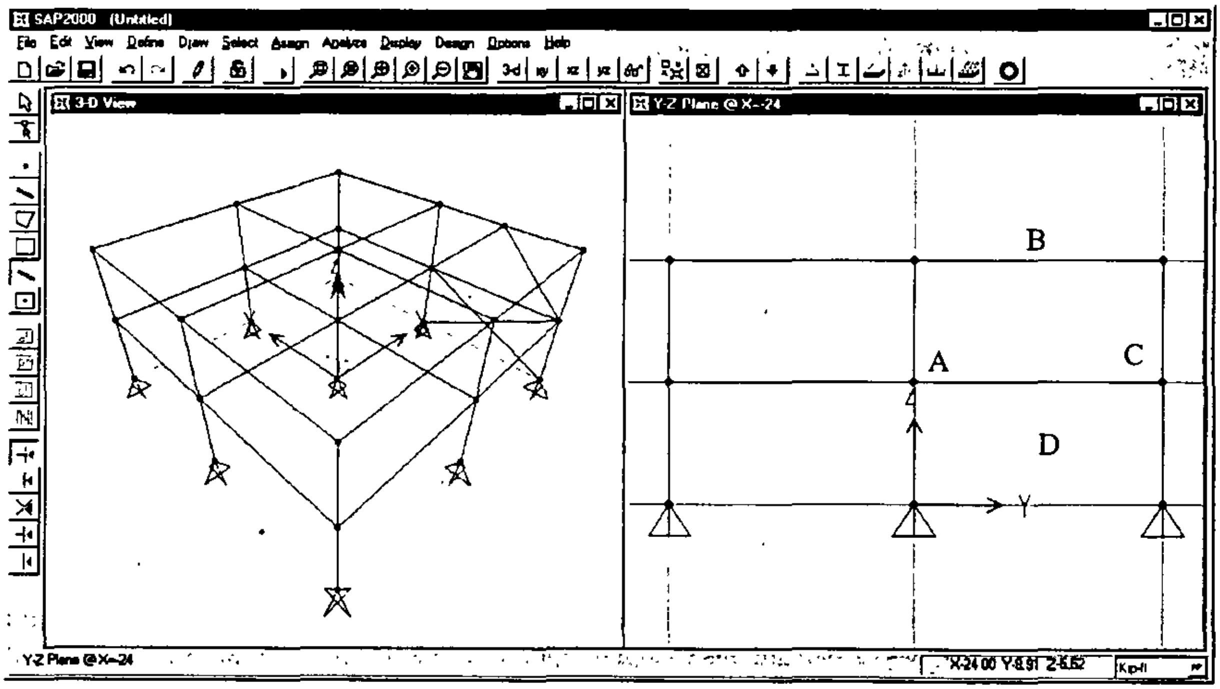


Figure C-5: Model Ready To Draw Second Set of Braces

18. Place the mouse pointer on the joint labeled "A" in Figure C-5 and click on the left mouse button.

19. Move the mouse pointer over to the center of the beam element at the point labeled "B" in Figure C-5. When the Snap To feature text box appears saying "Midpoint" click the left mouse button to input the brace element. Then press the Enter key on the keyboard.

20. Click the **Snap to Midpoints and Ends** button to deactivate this snapping option.

21. Click on the joint labeled "C" and then the joint labeled "B" in Figure C-5, and then press the Enter key on the keyboard to draw the second chevron brace member.

22. Click the **Quick Draw Frame Element** button on the side toolbar. Then click in the area labeled "D" in Figure C-5 to enter the X-braces.

23. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

24. Click the roof beam at the top of the braced frame to select it. Also click the joint at the top of the chevron brace (center of braced frame roof beam) to select it.

25. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.

26. In this dialog box:

    - Select the Break At Intersections With Selected Frames and Joints option.

    - Click the **OK** button.

Note: This completes inputting of the frame element geometry. Now we will input shell element geometry. We will use shell elements to model the roof diaphragm. We will demonstrate three different options for inputting the shell elements.

27. Click the **xy 2D View** button xy on the main toolbar to change the view to a plan in the XY plane. Note that the title of the window reads XY Plane @ Z=24.

28. Click the **Quick Draw Rectangular Shell Element** button on the side toolbar (or select **Quick Draw Rectangular Shell Element** from the **Draw** menu).

29. Click in the area labeled "A" in Figure C-6 to input the first shell element. Note that a quick Shell element is drawn by clicking in a grid space, bounded by four grid lines.

30. Note that just the outline of the shell element is shown. Sometimes when working with shell elements it is easier if you can view the shell element filled in. Click the **Set Elements**

button on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

31. Check the box labeled Fill Elements and click the **OK** button to display the shell elements filled.

32. Click in the area labeled "B" in Figure C-6 to input a second shell element. Note that this element will be reshaped in a subsequent step.

33. Click the **Draw Rectangular Shell Element** button on the side toolbar (or select **Draw Rectangular Shell Element** from the **Draw** menu). Make sure that the **Snap to Joints and Grid Points** button on the side tool bar is selected (depressed).

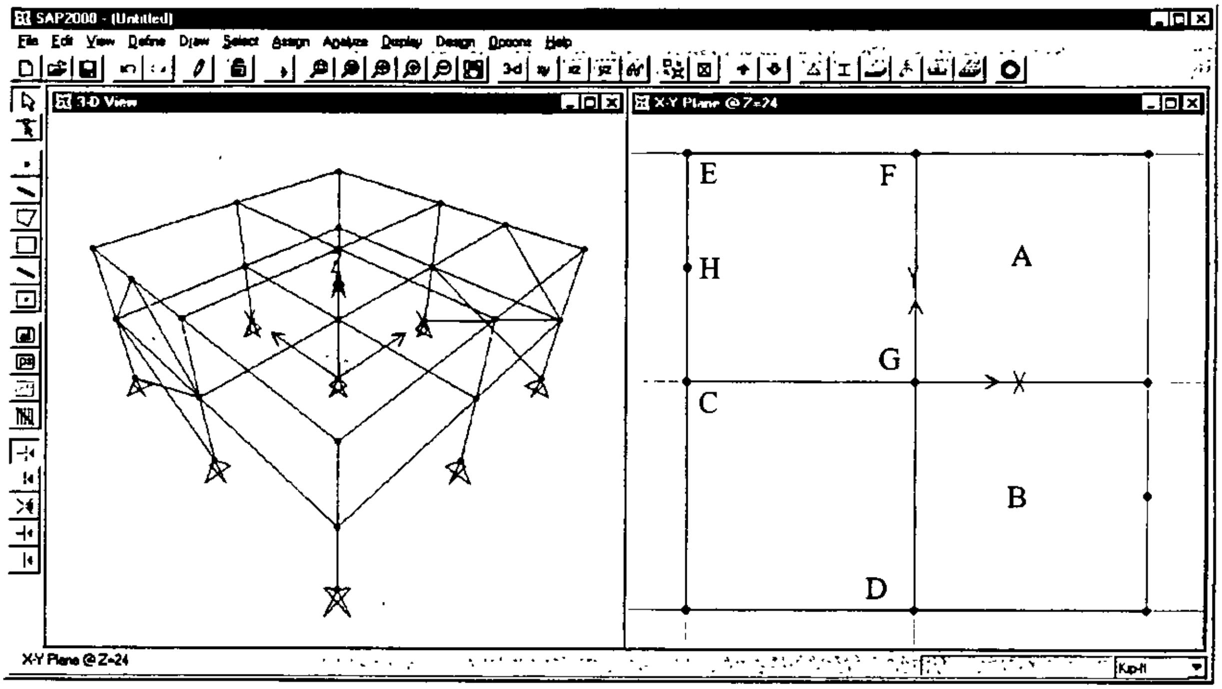


Figure C-6: Shell Element Input

34. Click on the point labeled "C" in Figure C-6 and then click the point labeled "D" to input the next shell element. Note that a rectangular Shell element is drawn by clicking to define two corners that are diagonally opposite of each other.

35. Click the **Draw Quadrilateral Shell Element** button on the side toolbar (or select **Draw Quad Shell Element** from the **Draw** menu).

36. Click on the points labeled "E", "F", "G" and "H" in Figure C-6, in that order, to draw a quadrilateral shell element.

37. Click on the points labeled "H", "G", "C" and "H" in Figure C-6, in that order, to draw a triangular shell element.

Note: Shell elements may be either four-sided, or three-sided. In general, four-sided elements are recommended.

38. Click the **Reshaper** button on the side toolbar (or select **Reshape Element** from the **Draw** menu).

39. Click once on the shell element in the lower right-hand corner (drawn in item 29 in the area labeled "B" in Figure C-6) to highlight it. Note that member end handles appear on the shell element as shown in Figure C-7.

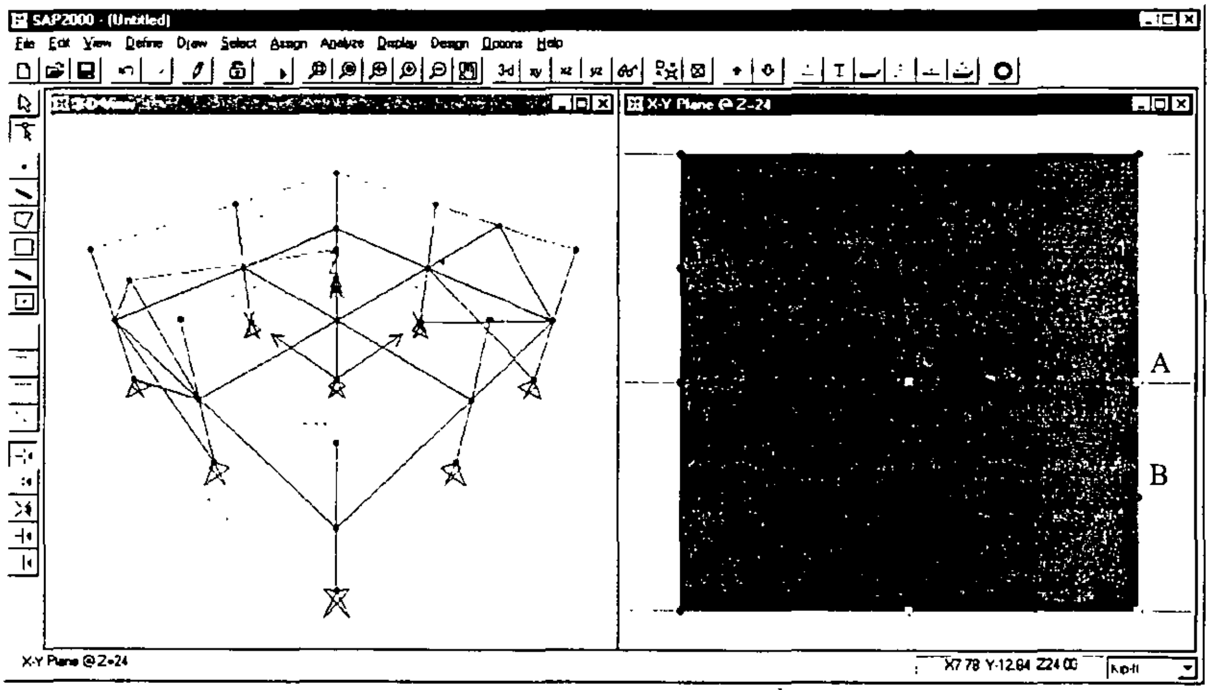


Figure C-7: Reshape Shell Element

40. Click on the point labeled "A" in Figure C-7, and while holding down the left mouse button, drag the member end handle to point "B".

41. Click the **Refresh Window** button on the main toolbar (or select **Refresh Window** from the **View** menu) to see the results of reshaping the element.

42. .Click the **Draw Quadrilateral Shell Element** button on the side toolbar (or select **Draw Quad Shell Element** from the **Draw** menu). Then click on the points labeled "C", "A", "B" and "C" in Figure C-7, in that order, to draw a triangular shell element.

43. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

44. Now we will return the shell element view to unfilled elements. Click the **Set Elements** button [⊠] on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

45. Uncheck the box labeled Fill Elements and click the **OK** button to display the shell elements not filled. The display should appear as shown in Figure C-8.

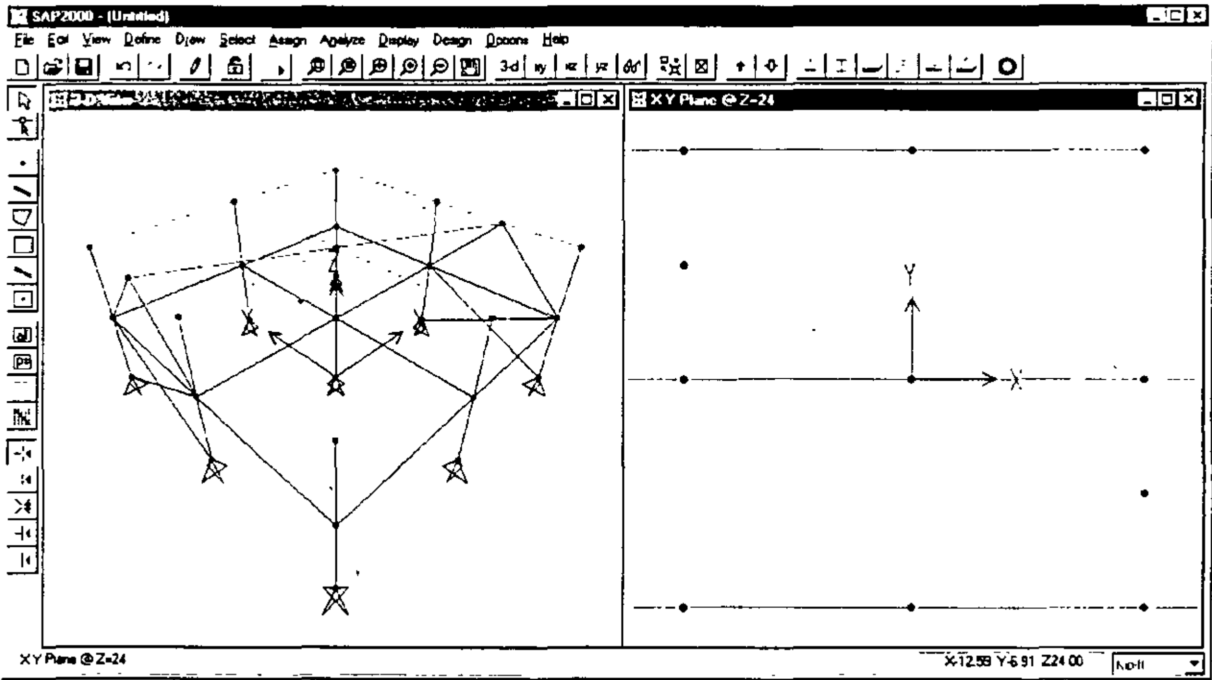


Figure C-8: Model With All Geometry Input

46. From the **File** menu choose **Save** and input a new name to save your file.

Note: It is a good idea to save your file often.

This completes the input of the model geometry. Now you can do one of the following:

- If you started the model from a template, and do not want to try starting the model from scratch, then skip to Step 2.
- If you started the model from a template and now want to try starting it from scratch, then continue on to Step 1b.
- If you started the model from scratch, and do not want to try starting the model from a template, then skip to Step 2.

- If you started the model from scratch, and now want to try starting it from a template, then return to the beginning of Step 1a.

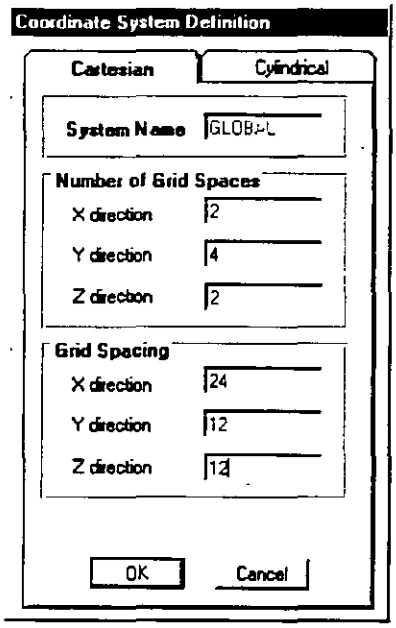
- If you want to stop working on the tutorial for now, and close SAP2000, make sure that you have saved your file as outlined in item 46, and then from the **File** menu select **Exit**.

Step 1b: Setting Up the Model Geometry From Scratch

This tutorial steps through setting up the model geometry both from a built-in template (step 1a), and from scratch (step 1b). It is not necessary to do both of these steps in order to complete the tutorial; one or the other is sufficient. However, each of them has some unique information.

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model....** This displays the Coordinate System Definition dialog box.



3. In this dialog box:

- Select the Cartesian tab.
- Set the Number of Grid Spaces in X direction to 2.
- Set the Number of Grid Spaces in Y direction to 4.
- Set the Number of Grid Spaces in Z direction to 2.
- Set the Grid Spacing in the X direction to 24.
- Set the Grid Spacing in the Y direction to 12.
- Set the Grid Spacing in the Z direction to 12.
- Click the OK button to accept the grid definition.

4. The screen will refresh and display a 3-D and a 2-D (XY Plane @ Z=24) view in vertically-tiled adjoining windows.

5. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

6. Click the **Quick Draw Frame Element** button on the side toolbar (or select **Quick Draw Frame Element** from the **Frame** menu).

   Note: If you hold the pointer over a toolbar button for a few seconds, a text box including the name of the button will appear.

7. In the window labeled X-Y Plane @ Z=24, to enter a beam element click on a grid line. The **Quick Draw Frame Element** tool will then create a beam element on that grid line spanning between the closest perpendicular grid lines on either side of the point where you clicked.

Thus click the ten locations designated by an O in Figure C-9 to draw some of the roof level beams.

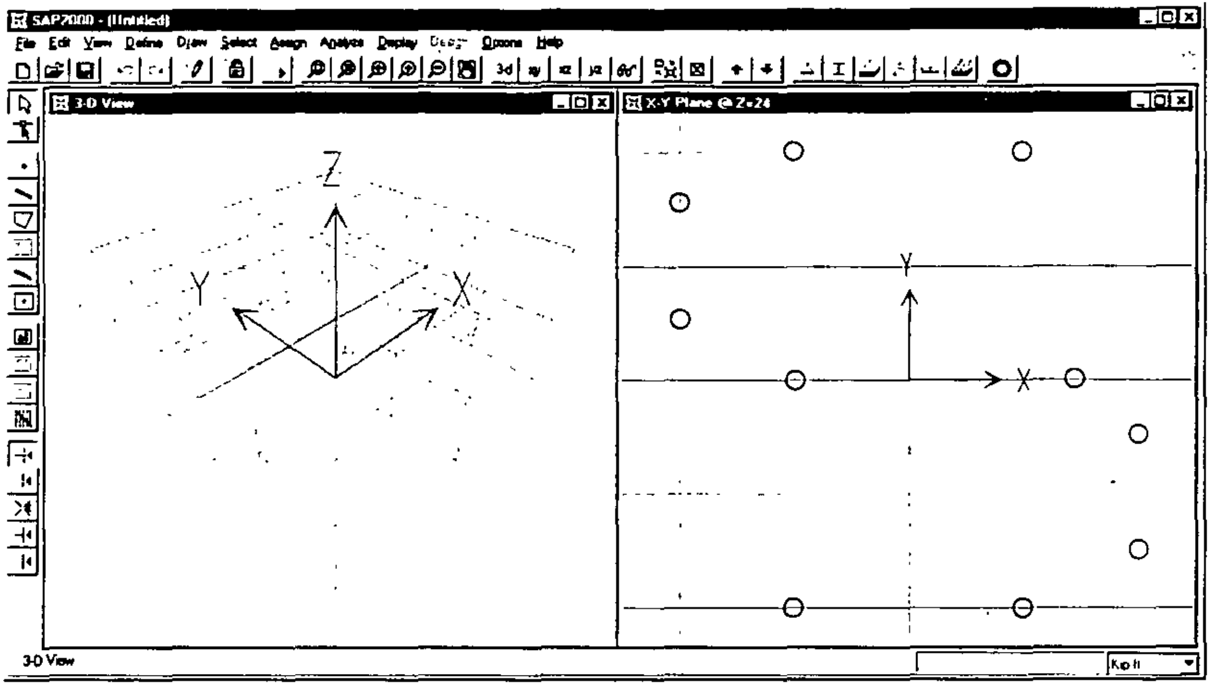


Figure C-9: Quick Drawing Roof Level Beams

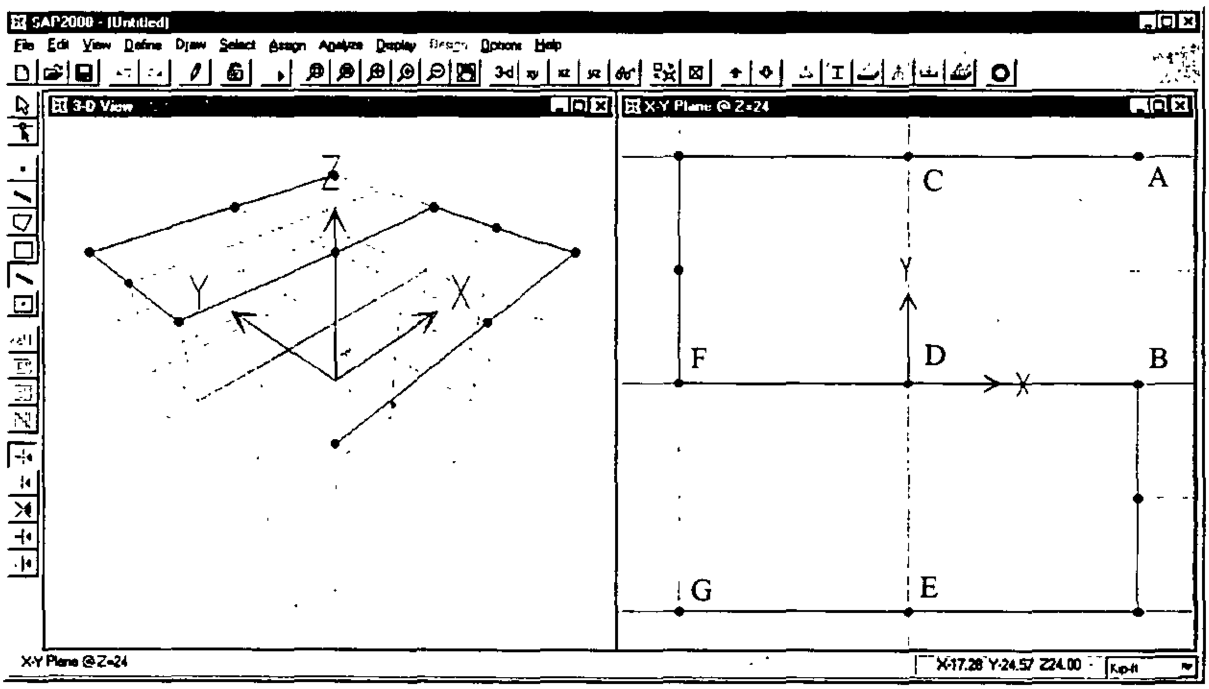


Figure C-10: Drawing Remaining Roof Level Beams

Note: If you miss slightly when clicking on a grid line, the program may assume you were attempting to input two diagonal braces in the bay bounded by the grid lines. If this happens, simply click the Undo button on the main toolbar twice, or select Undo Frame Add from the Edit menu twice.

8. Click the **Draw Frame Element** button on the side toolbar. The program is now in Draw Mode.

9. Make sure that the **Snap to Joints and Grid Points** button on the side tool bar is depressed. Place the mouse pointer on the joint labeled "A" in Figure C-10 and click on the left mouse button. Note that when the mouse pointer is near the joint, it snaps to the joint, and a text box that says "Grid Intersection" appears. This is the effect of the **Snap to Joints and Grid Points** feature.

   *Note: Other snap options included in SAP2000 include Snap to Midpoints and Ends, Snap to Element Intersections, Snap to Perpendicular, and Snap to Lines and Edges. The buttons for these features are located on the side toolbar just below the **Snap to Joints and Grid Points** button. Note that all of the snap features can also be accessed through the **Snap To...** option on the **Draw** menu. For more information on the snap capabilities of SAP2000, refer to the topic titled Snap Tools in the SAP2000 online help. Click on the **Help** menu and select the **Search for Help on...** option to access the SAP2000 online help.*

10. Place the mouse pointer near the joint labeled "B" in Figure C-10 and Click on the left mouse button to draw the roof beam element.

    *Note: We couldn't use the **Quick Draw Frame Element** tool to draw this beam because the beam crosses a grid line.*

11. By default the program is now ready to draw another frame element starting from point "B". Press the Enter key on the keyboard to stop from drawing the second frame member at this location.

12. Place the mouse pointer on the joint labeled "C" in Figure C-10 and click on the left mouse button. Click in sequence on joints "D" and "E" and then press the Enter key to draw the next two roof beams.

13. We will now edit the grid lines so that we can use the **Quick Draw Frame Element** tool to enter the last roof beam that will span from point "F" to point "G" in Figure C-10.

14. From the **Draw** menu select **Edit Grid....** This displays the Modify Grid Lines dialog box.

15. In this dialog box:

    - Click the Y option in the Direction area.

- Highlight -12 in the Y Location list box and click the **Delete Grid Line** button.

- Highlight 12 in the Y Location list box and click the **Delete Grid Line** button.

- Click the **OK** button.

16. Click the **Quick Draw Frame Element** button on the side toolbar (or select **Quick Draw Frame Element** from the **Frame** menu).

17. Click on the grid line between points "F" and "G" in Figure C-10 to enter the last roof beam. The model now appears as shown in Figure C-11.

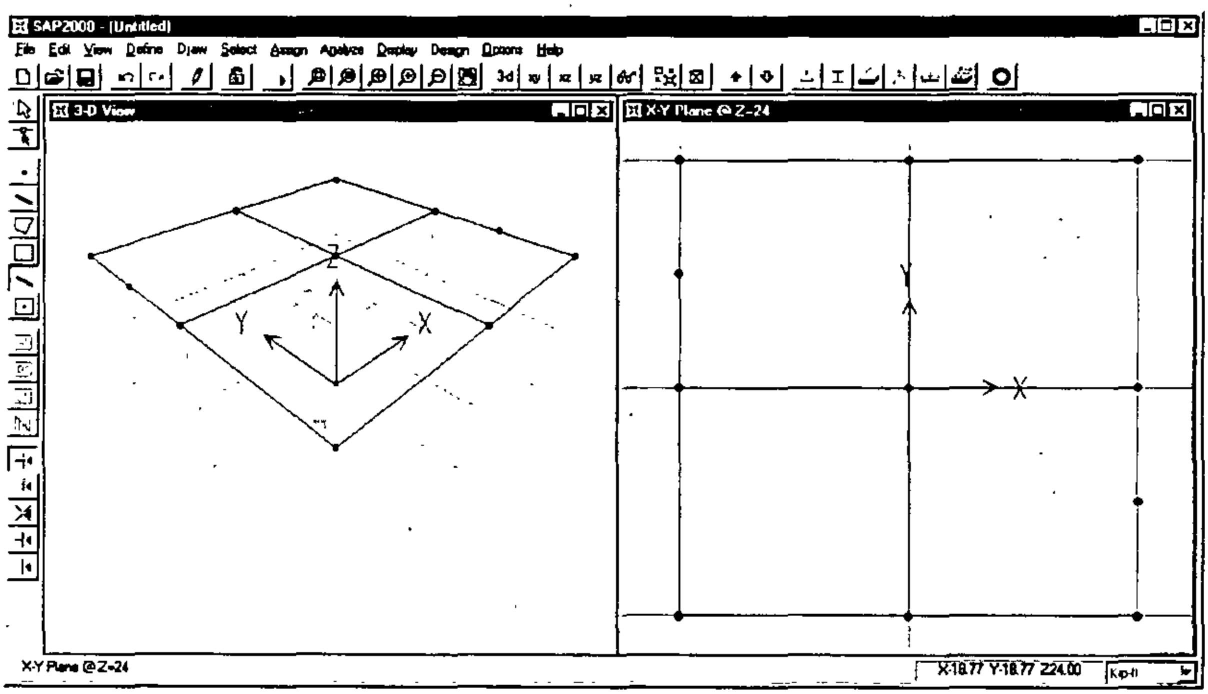


Figure C-11: Model After All Roof Level Beams Have Been Drawn

18. Select all of the roof level frame elements and joints in the X-Y Plane @ Z=24 by "windowing." To do this:

- Click the **Pointer** button on the side toolbar to activate the Select Mode.

- Move the pointer above and to the left of the frame elements and joints.

- Click and hold the left mouse button.

- While holding, move the pointer below and to the right of the frame elements and joints. A "rubber-band" window will show the region selected.

- Release the left mouse button to select all elements in this window.

19. From the **Edit** menu, choose **Copy**.

20. From the **Edit** menu, choose **Paste**. This will display the Paste Coordinates dialog box.

21. In this dialog box input **0** for Delta X, **0** for Delta Y and **-12** for Delta Z.

22. Click the **OK** button and the geometry for the roof level is copied to the second level.

23. Click the **Down One Gridline** button on the main toolbar to display the plan view at the second floor level, Z=12. Note the window title changes to X-Y Plane @ Z=12. The model now appears as shown in Figure C-12.

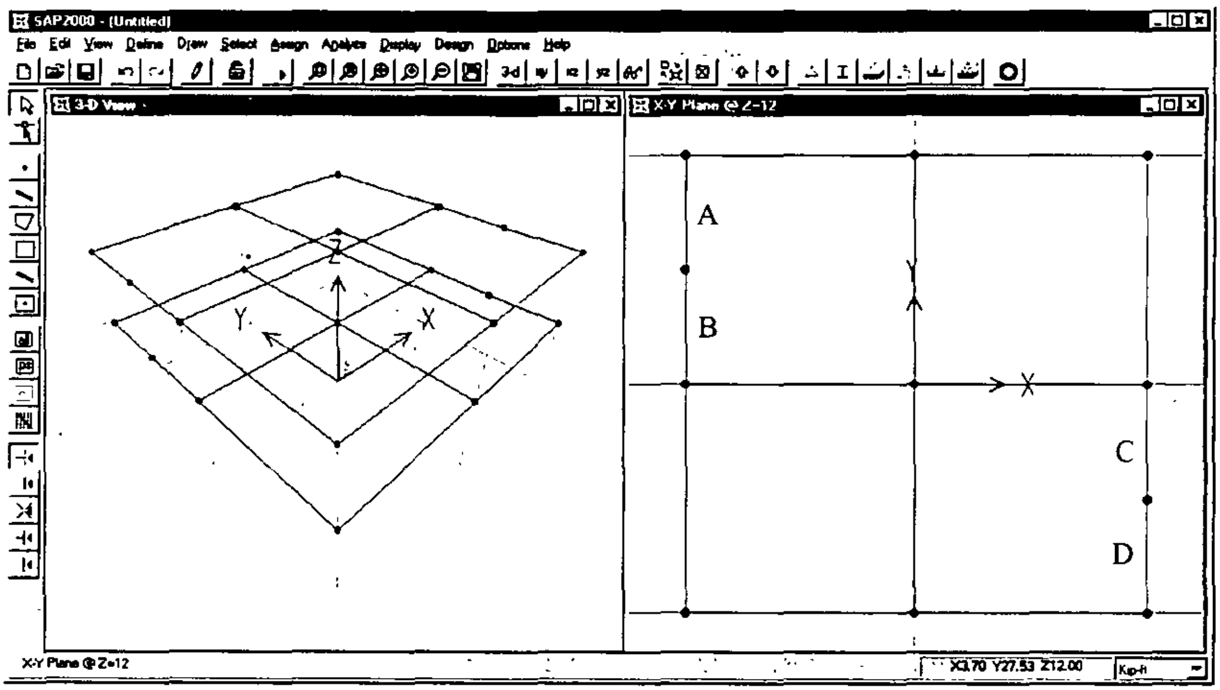


Figure C-12: Model After All Roof Level Beams Copied To Second Level

Note: The second level beams at the braced frames do not need a joint at the center to receive a brace. Thus we will delete the center joints.

24. Click the second level beams labeled "A" and "B" in Figure C-12 to select them.

25. From the **Edit** menu select **Join Frames** to combine these two elements into a single element and remove unused joints left over from the joining process.

26. Click anywhere on the combined frame member to select it. Note that the entire member is highlighted thus verifying that the members have been joined.

27. Click the **Clear Selection** button [clr] on the side toolbar (or select **Clear Selection** from the **Select** menu) to deselect the beam.

28. Click the second level beams labeled "C" and "D" in Figure C-12 to select them. From the **Edit** menu select **Join Frames** to combine these two elements into a single element.

29. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active. Note when the window is active, its title bar will be highlighted.

30. Click the **yz 2D View** button [yz] on the main toolbar to change the view to an elevation in the YZ plane. Note that the title of the window reads YZ Plane @ X=24. This same title also occurs on the left-hand side of the status bar at the bottom of the SAP2000 window. The model appears as shown in Figure C-13.

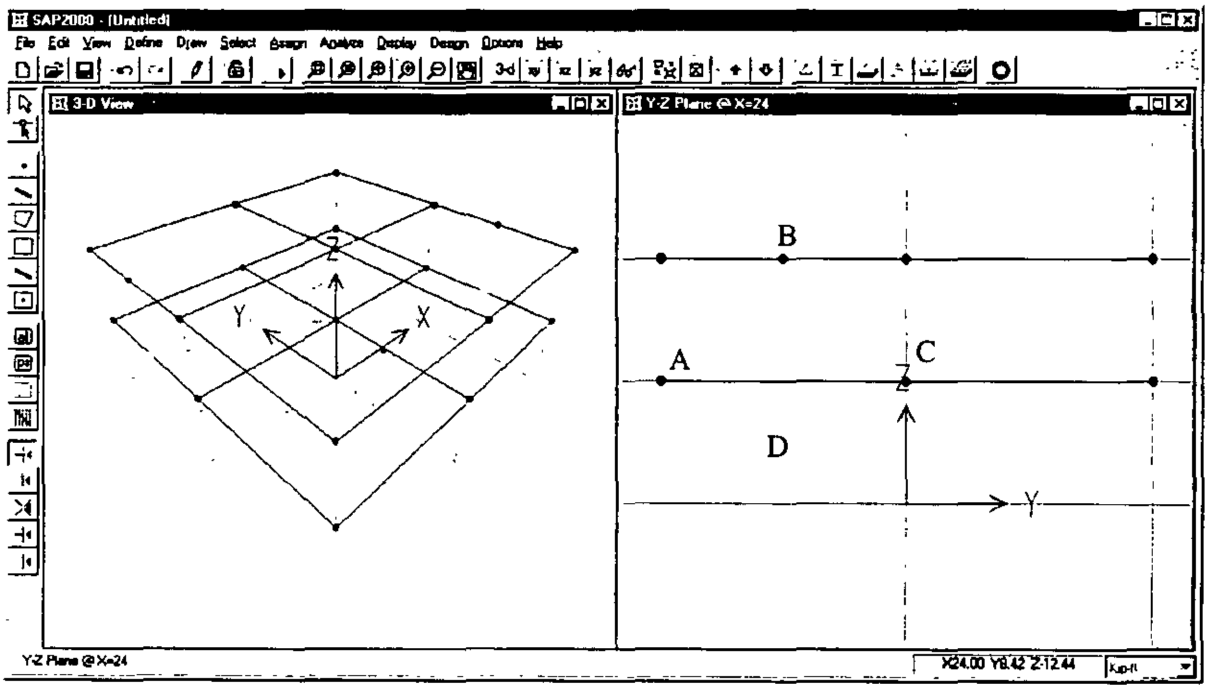


Figure C-13: Y-Z Elevation

31. Click the **Draw Frame Element** button on the side toolbar. The program is now in Draw Mode.

32. Make sure that the **Snap to Joints and Grid Points** button on the side tool bar is depressed. Place the mouse pointer on the joint labeled "A" in Figure C-13 and click on the left mouse button. Note that when the mouse pointer is near the joint, it snaps to the joint,

and a text box that says "Grid Intersection" appears. This is the effect of the Snap to Joints and Grid Points feature.

33. Place the mouse pointer near the joint labeled "B" in Figure C-13 and note that in this case the Snap To option text box just says "Point" because the joint does not occur at a grid intersection. Click on the left mouse button to draw the chevron brace element.

    Note: If you wanted to, at this point you could just click on point C in Figure C-12 to draw the second brace element. If we were to do that, the start point for the brace would be at the top (point B) and the bottom point would be at the bottom (point C). Though not necessarily a problem, this would be inconsistent with how we input the first brace. In this example we will opt for consistency.

    Note: If you wanted to, at this point you could move the mouse pointer into the 3D View window, and select the second joint for the next frame element. Try moving the mouse pointer over the 3D View window, but for this example don't actually click to define the second frame member.

34. Press the Enter key on the keyboard to stop drawing the second frame member. Note that the **Draw Frame Element** button is still depressed, i.e., the program is still in Draw Mode and ready to draw another frame element.

35. Click on the joint labeled "C" and then the joint labeled "B" in Figure C-13, and then press the Enter key on the keyboard to draw the second chevron brace member.

36. Click the **Quick Draw Frame Element** button on the side toolbar. Then click in the area labeled "D" in Figure C-13 to enter the X-braces.

    *Note: The **Quick Draw Frame Element** option works two different ways. You can click on a grid segment to quickly draw a quick single frame element between the two adjacent perpendicular grid lines. Alternatively, you can click in a space bounded by four grid lines to draw a cross brace as was done here.*

    Note: The X-braces could also have been entered using the same technique that was employed for the chevron braces.

    *Note: Both the **Draw Frame Element** and the **Quick Draw Frame Element** options are also available on the **Draw** menu.*

37. Click the **Down One Gridline** button on the main toolbar twice to display the elevation view at X=-24. Note the window title changes to Y-Z Plane @ X=-24.

38. Repeat steps 32 through 36 to draw the second set of braces. Note that the second set of braces occurs on the right-hand side of the elevation. The model now appears as shown in Figure C-14.

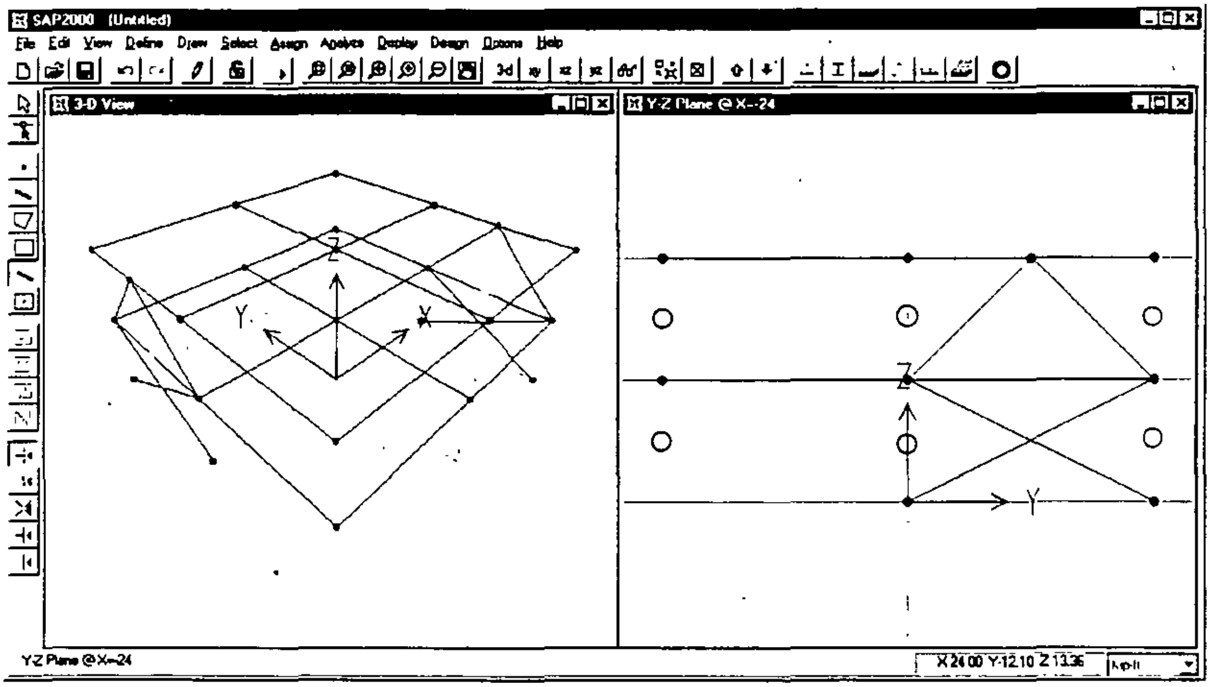


Figure C-14: Model After Braces Have Been Drawn

39. Now we will draw the columns. Click the six column grid line locations designated with an O in Figure C-14 to define the first line of columns.

40. Click the **Pointer** button to exit Draw Mode and enter Select Mode. Click on the six columns just entered to select them.

    Note: A message on the left-hand side of the status bar at the bottom of the SAP2000 window tells you how many of each type of element are currently selected.

41. From the **Edit** menu, choose **Replicate**. This will display the **Replicate** dialog box.

42. In this dialog box select the **Linear** tab, input **24** for X, **0** for Y and **0** for Z. Input **2** for the number.

43. Click the **OK** button and the geometry for the columns will be replicated twice at a 24-foot spacing.

    Note: In addition to linear replication, the ***Replicate*** *option also allows radial replication and mirroring. Refer to the topic replicate in the online help for more information. The online help can be accessed by clicking on the* ***Help*** *menu and selecting the* ***Search for Help on...*** *option or you can access it by pressing the F1 key on the keyboard when the Replicate dialog box is open.*

44. Now we will apply the base restraints. Click in the window labeled Y-Z Plane @ X=-24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

45. From the **View** menu select **Set 2D View....** This will display the Set 2D View dialog box.

46. Make sure the **X-Y plane** option button is selected and enter 0 in the **Z** = edit box.

47. Click the **OK** button, and the window will now display the X-Y Plane @ Z=0.

48. Select all of the joints in the X-Y Plane @ Z=0 by "windowing." To do this:

    - Click the **Pointer** button on the side toolbar left side of the screen.
    - Move the pointer above and to the left of the support joints.
    - . Click and hold the left mouse button.
    - While holding, move the pointer below and to the right of the support joints. A "rubber-band" window will show the region selected.
    - Release the left mouse button to select all elements (joints) in this window.

49. From the **Assign** menu, choose **Joint**, and then **Restraints...**from the submenu. This will display the Joint Restraints dialog box.

50. In this dialog box:

    - Click the pinned base fast restraint button to set all translational degrees of freedom (U1, U2 and U3) as restrained.
    - Click the **OK** button

51. Click in the window labeled 3-D View to make sure it is active.

52. Click the **Refresh Window** button on the main toolbar (or select **Refresh Window** from the **View** menu) to see the restraints in the 3-D View window.

53. Click in the plan view window currently labeled Joint Restraints to make sure it is active.

54. Click the **Show Undeformed Shape** button to reset the view and to return the window label to X-Y Plane @ Z=0.

55. Click the **Up One Gridline** button on the main toolbar twice to display the elevation view at Z=24.

56. To finish defining the model geometry, complete items 25 through 46 in Step 1a , i.e., the previous step. When finished, return to this point.

This completes the input of the model geometry from scratch. Now you can do one of the following:

- If you started the model from scratch, and do not want to try starting the model from a template, then go on to Step 2.
- If you started the model from scratch, and now want to try starting it from a template, then return to the beginning of Step 1a.
- If you want to stop working on the tutorial for now, and close SAP2000, make sure that you have saved your file as outlined in item 46 of Step 1a, and then from the **File** menu select **Exit**.

Step 2: Relabeling Joint, Frame and Shell Elements

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

3. In this dialog box:

   - In the Joints area check the Labels box.
   - In the Frames area check the Labels box.
   - In the Shells area check the Labels box.
   - Click the **OK** button.

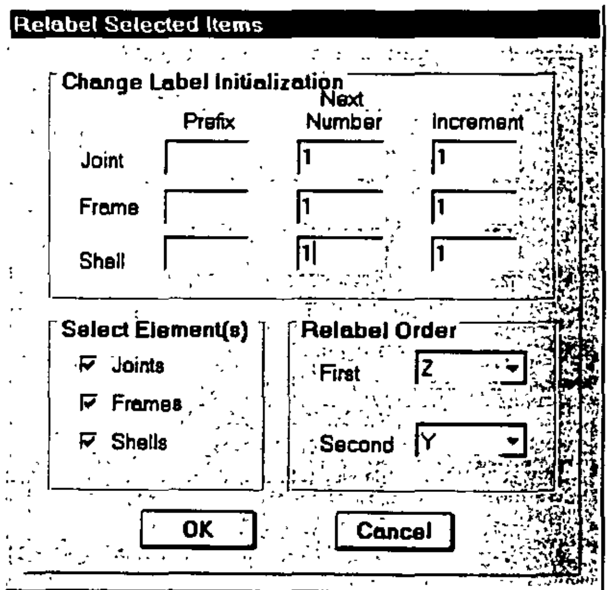
   Note: We are turning on the element labels so that we can see the effect of the element relabeling. It is not necessary to turn on element labels in order to relabel them.

4. From the **Select** menu choose **Select**, and then **All** from the submenu. All elements in the model (joints, frame elements and shell elements) are selected.

5. From the **Edit** menu choose **Change Labels** to display the Relabel Selected Items dialog box.

6. In this dialog box:

   - Press the F1 key to display the context-sensitive online help for this dialog box.
   - When finished reading the online help, click the "X" in the top right-hand corner of the Help window, or select Exit from the File menu on the Help window to close it.
   - In the Change Label Initialization area of the dialog box type **1** in the Next Number edit boxes for Joint, Frame and Shell elements.



- Accept the rest of the default values in the dialog box.

- Click the **OK** button to make the changes.

  Note: It is not typically necessary to renumber the elements. It is done here to illustrate the process, and to make sure all tutorial users have the same numbering system, whether they started the model from a template, or from scratch, since we will refer to joint and frame elements by their labels later in this tutorial.

  Note: Joint, Frame and Shell element labels can be given alphanumeric prefixes. These prefixes can be input in the Prefix edit boxes on the Relabel Selected Items dialog box.

  Note: It is not necessary to select all elements to relabel. You could select only a few elements, of any type, and relabel only the selected elements.

7. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

8. In this dialog box:

   - In the Joints area uncheck the Labels box.

   - In the Frames area uncheck the Labels box.

   - In the Shells area uncheck the Labels box.

   - Click the **OK** button.

9. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes relabeling the joint, frame and shell elements.

Step 3: Defining Material Properties

In this tutorial we will use default material properties, except that we will not use the self-weight and self mass. In this step we will first review the default material properties (items 1 through 14) and then we will change the material properties to set the self-weight and self mass to zero (items 15 through 21).

1. From the **Define** menu choose **Materials....** This displays the Define Materials dialog box.

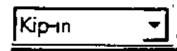
2. Highlight CONC in the Materials area and click the **Modify/Show Material** button. This will display the Material Property Data dialog box.

   Note: To add a new material property, click the Add New Material button.

3. Note the material properties shown, and notice that they are in units of kips and feet.

4. Click the **Cancel** button to exit the Material Property Data dialog box without making any changes.

5. Click the **Cancel** button to exit the Define Materials dialog box without making any changes.

6. Click the drop down box in the status bar to change the units to kip-in. 

7. From the **Define** menu choose **Materials....** This again displays the Define Materials dialog box.

8. Highlight CONC in the Materials area, if it is not already highlighted, and click the **Modify/Show Material** button. This displays the Material Property Data dialog box for the material named CONC.

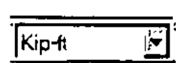
9. Note the material properties shown, and notice that they are in units of kips and inches.

10. Click the **Cancel** button to exit the Define Materials dialog box without making any changes.

11. Highlight STEEL in the Materials area and click the **Modify/Show Material** button. This will display the Material Property Data dialog box.

12. Note the material properties shown, and notice that they are in units of kips and inches.

13. Click the **Cancel** button twice to exit the Material Property Data dialog box and the Define Materials dialog box without making any changes.

14. Click the drop down box in the status bar to change the units back to kip-ft. 

15. From the **Define** menu choose **Materials....** This displays the Define Materials dialog box.

16. Highlight CONC in the Materials area and click the **Modify/Show Material** button. This will display the Material Property Data dialog box.

17. In this dialog box:

    - In the Analysis Property Data area type **0** in the Mass Per Unit Volume edit box.

    - In the Analysis Property Data area type **0** in the Weight Per Unit Volume edit box.

    - Click the **OK** button.

18. Highlight STEEL in the Materials area and click the **Modify/Show Material** button. This will display the Material Property Data dialog box.

19. In this dialog box:

    - In the Analysis Property Data area type **0** in the Mass Per Unit Volume edit box.

    - In the Analysis Property Data area type **0** in the Weight Per Unit Volume edit box.

    - Click the **OK** button.

20. Click the **OK** button to close the Define Materials dialog box.

21. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the review and definition of material properties.

Step 4: Defining Frame Sections

We will use nine different wide flange sections (W24X62, W24X55, W21X44, W18X40, W18X35, W16X26, W14X132, W8X58 and W8X31) and two different structural tube sections (TS5X5X1/4 and TS4X4X1/4) for this model. We will use the structural sections data file provided with SAP2000. The file is called SECTIONS.PRO and resides in the same directory as SAP2000.

1. From the **Define** menu choose **Frame Sections....** This will display the Define Frame Sections dialog box.

2. In this dialog box:

   - Click on the Import drop-down box.

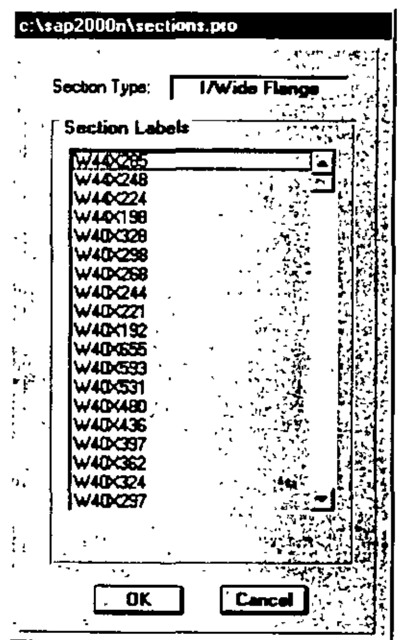
   - Click on Import I/Wide Flange. This will display the Section Property File dialog box.

   - In this dialog box:

     ✓ Locate the SECTIONS.PRO data file. It is typically located in the directory where you installed SAP2000.

     ✓ Open the SECTIONS.PRO data file by highlighting it and clicking on the Open button or by double clicking on the file name.

     ✓ This will display a dialog box that includes a scrolling list box showing all the I/wide flange sections available in the data file. The title bar of this box displays the full path to the data file.



*Note: In a SAP2000 session you only have to locate and open the SECTIONS.PRO data file once. You have the option to select another data file at any time by choosing **Preferences** in the **Options** menu and selecting the Steel tab.*

✓ In the list box:

➢ Use the Scroll buttons to locate the W24X62 frame section. Click once on this section to highlight it.

➢ Use the Scroll buttons to locate the W24X55 frame section. While holding down the Ctrl key on the keyboard, click once on this section to highlight it and add it to the selection.

Note: Frame sections may be selected one at a time or they may be selected in groups as shown in this example. To select multiple frame sections that are next to each other, hold down the Shift key and click on the first and last frame element in the group. To select multiple frame sections that are not next to each other, hold down the Ctrl key and click each section you want to select, as is done here.

- ➢ Use the Scroll buttons to locate the W21X44, W18X40, W18X35, W16X26, W14X132, W8X58 and W8X31 frame sections. While holding down the Ctrl key on the keyboard, click once on each section to highlight it and add it to the selection.

- ➢ Click the **OK** button. This will display the I/Wide Flange Section dialog box which shows a schematic view of the last selected section (W8X31), section dimensions, and STEEL as the default material type.

- ✓ Click the **OK** button. This will close the I/Wide Flange Section dialog box and return to the Define Frame Sections dialog box. Notice that in the dialog box the labels of the selected frame elements are added to the default section name (FSEC1) in the Frame Section area.

- Click on the Import drop-down box.

- Click on Import Box/Tube. This will display a scrolling list box showing all the box/tube sections available in the data file.

- In this dialog box:

  - ✓ Use the Scroll buttons to locate the TS5X5X1/4 frame section. Click once on this section to highlight it.

  - ✓ Use the Scroll buttons to locate the TS4X4X1/4 frame section. While holding down the Ctrl key on the keyboard, click once on this section to highlight it and add it to the selection.

  - ✓ Click the **OK** button. This will display the Box/Tube Section dialog box that shows a schematic view of the last selected section (TS4X4X1/4), section dimensions, and STEEL as the default material type.

- Click the **OK** button. This will close the Box/Tube Section dialog box and return to the Define Frame Sections dialog box. Notice that in the dialog box the labels of the selected frame elements are added to the previously defined frame sections.

3. Click the **OK** button to close the Define Frame Sections dialog box.

This completes the definition of frame sections.

Step 5: Defining Shell Sections

We will use one shell section property in the model to model the roof diaphragm. We will assume the roof diaphragm to be equivalent to a 1" thick concrete diaphragm. This is not a recommended way to approximate a metal deck diaphragm, it is just a simple and expedient method to use for this example.

1. From the **Define** menu choose **Shell Sections....** This will display the Define Shell Sections dialog box.

2. In this dialog box:

   - Click the **Add New Section** button. This will display the Shell Sections dialog box.

   - In this dialog box:

     ✓ Type ROOF in the Section Name edit box.

     ✓ Accept the default material CONC.

     ✓ In the thickness area type 0.0833 in both the Membrane and Bending edit boxes.

     ✓ In the Type area, select the Membrane option.

     Note: In general we recommend using the Shell type option for shell elements.

     ✓ Click the **OK** button.

3. Click the **OK** button to close the Define Shell Sections dialog box.

4. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the definition of shell sections.

Step 6: Assigning Groups

Groups can be a powerful tool for selecting elements for both assignments and display. In this step we will create nine groups, as shown in the table below:

Group Name	Description
ROOF	All roof level joints, frame elements and shell elements
2ND	All second level joints, frame elements and shell elements
COLS	All column elements
FRCOLS	All moment frame columns
BRCOLS	All braced frame columns
BRACE1	All braces between the first and second level
BRACE2	All braces between the second level and the roof level
FRMGIRD	All moment frame girders
BASE	All bottom level columns and support joints

Note: By default the program creates a group named ALL which includes all elements in the model (joint elements included).

1. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Pointer** button on the side toolbar and select all of the elements in the X-Y Plane @ Z=24 by "windowing."

   Note: To add more joints and/or elements to an already assigned group, first select the group, then select more joints and/or elements, and finally assign them to the group. Group assignment always replaces the existing elements in that group.

3. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

4. In this dialog box:

   - Type **ROOF** in the edit box at the top of the Groups area.

   - Click the **Add New Group Name** button to define a group named ROOF.

     *Note: A common error is to forget to click the **Add New Group Name** button before pressing the **OK** button.*

   - Click the **OK** button to assign the selected elements to the group named ROOF.

5. Click the **Down One Gridline** button on the main toolbar to display the plan view at Z=12. Note the window title changes to X-Y Plane @ Z=12.

6. Select all of the elements in the X-Y Plane @ Z=12 by "windowing."

7. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

8. In this dialog box:

   - Type **2ND** in the edit box at the top of the Groups area.
   - Click the **Add New Group Name** button to define a group named 2ND.
   - Click the **OK** button to assign the selected elements to the group named 2ND.

9. Click on the **xz** button on the main toolbar to view an elevation in the X-Z plane. Note the title of the window is probably X-Z Plane @ Y=24; it depends on the sequence of steps you followed in creating the model. If the window title is not X-Z Plane @ Y=24, click the **Up One Gridline** button on the main toolbar or the **Down One Gridline** button on the main toolbar until the title of the window is X-Z Plane @ Y=24.

10. Click on the six column elements in this view to select them.

    Note: SAP2000 will display the number and type of elements you have selected on the left-hand side of the status bar at the bottom of the SAP2000 window.

    *Note: You can click again on a single selected element to deselect it. If you want to deselect all elements at once, then click the **Clear Selection** button on the side toolbar, or choose **Clear Selection** from the **Select** menu.*

11. Click the **Down One Gridline** button on the main toolbar to display the elevation at Y=0. Note the window title changes to X-Z Plane @ Y=0.

12. Select the bottom level columns by intersection. To do this:

    - Click the **Pointer** button on the side toolbar.
    - Click the **Set Intersecting Line Select Mode** button on the side toolbar.
    - Move the pointer to the left of the columns you want to select.
    - Click and hold the left mouse button.
    - While holding, move the pointer to the right of the members you want to select. A "rubber band" will show the intersecting line.
    - Release the left mouse button to select all members that intersect this line.

13. Click the **Set Intersecting Line Select Mode** button again and select the top level columns.

14. Click the **Down One Gridline** button on the main toolbar to display the elevation at Y=-24. Note the window title changes to X-Z Plane @ Y=-24.

15. Use the Intersecting Line mode to select the six column elements in this elevation. There should now be a total of 18 frame elements selected. You can confirm this by looking on the left-hand side of the status bar at the bottom of the SAP2000 window.

16. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

17. In this dialog box:

    - Type **COLS** in the edit box at the top of the Groups area.
    - Click the **Add New Group Name** button to define a group named COLS.
    - Click the **OK** button to assign the selected elements to the group named COLS.

18. Now we will assign the moment frame columns to a group. Click on the left-hand column and the center column (4 elements total) to select them.

19. Click the **Up One Gridline** button on the main toolbar twice to display the elevation at Y=24. Note the window title changes to X-Z Plane @ Y=24.

20. Click on the right-hand column and the center column (4 elements total) to select them. There should now be eight frame elements selected.

21. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

22. In this dialog box:

    - Type **FRCOLS** in the edit box at the top of the Groups area.
    - Click the **Add New Group Name** button to define a group named FRCOLS.
    - Click the **OK** button to assign the selected elements to the group named FRCOLS.

23. Now we will assign the braced frame columns to a group. Click on the **yz** button yz on the main toolbar to view an elevation in the Y-Z plane. If necessary, click the **Up One Gridline** button on the main toolbar until the title of the window is Y-Z Plane @ X=24.

24. Click on the four column elements at the braced frame to select them.

25. Click the **Down One Gridline** button on the main toolbar twice to display the elevation at X=-24.

26. Click on the four column elements at the braced frame to select them. There should now be eight frame elements selected.

27. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

28. In this dialog box:

    - Type **BRCOLS** in the edit box at the top of the Groups area.

    - Click the **Add New Group Name** button to define a group named BRCOLS.

    - Click the **OK** button to assign the selected elements to the group named BRCOLS.

29. Now we will assign the bottom level braced frame braces to a group. Click on two bottom level braces to select them.

30. Click the **Up One Gridline** button on the main toolbar twice to display the elevation at X=24.

31. Click on two bottom level braces to select them. There should now be four frame elements selected.

32. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

33. In this dialog box:

    - Type **BRACE1** in the edit box at the top of the Groups area.

    - Click the **Add New Group Name** button to define a group named BRACE1.

    - Click the **OK** button to assign the selected elements to the group named BRACE1.

34. Now we will assign the top level braced frame braces to a group. Click on two top level braces to select them.

35. Click the **Down One Gridline** button on the main toolbar twice to display the elevation at X=-24.

36. Click on two top level braces to select them. There should now be four frame elements selected.

37. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

38. In this dialog box:

 - Type **BRACE2** in the edit box at the top of the Groups area.

 - Click the **Add New Group Name** button to define a group named BRACE2.

 - Click the **OK** button to assign the selected elements to the group named BRACE2.

39. Now we will assign the moment frame girders to a group. Click on the **xz** button on the main toolbar to view an elevation in the X-Z plane. Note the title of the window is X-Z Plane @ Y=24.

40. Click on the roof level and second level beam on the right side of this elevation to select them.

41. Click the **Down One Gridline** button on the main toolbar twice to display the elevation at Y=-24.

42. Click on the roof level and second level beam on the left side of this elevation to select them. There should now be four frame elements selected.

43. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

44. In this dialog box:

 - Type **FRMGIRD** in the edit box at the top of the Groups area.

 - Click the **Add New Group Name** button to define a group named FRMGIRD.

 - Click the **OK** button to assign the selected elements to the group named FRMGIRD.

45. Finally we will assign the lower level columns, braces and the base joints to a group. This group can be used in the Group Joint Force Sum option (on the **Display** menu) to display the base shear. Click the **Perspective Toggle** button on the main toolbar. A perspective view of the X-Z elevation is displayed.

46. Click the **Set Intersecting Line Select Mode** button and select all of the bottom level columns. Note that in doing so you will also select the bottom level braces.

47. Click on the **xy** button on the main toolbar to view an elevation in the X-Y plane. Note the title of the window is X-Y Plane @ Y=12.

48. Click the **Down One Gridline** button on the main toolbar to display the plan at Z=0.

49. Select all of the joints at this level by "windowing". There should now be 9 joints and 13 frame elements selected.

50. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

51. In this dialog box:

   - Type **BASE** in the edit box at the top of the Groups area.
   - Click the **Add New Group Name** button to define a group named BASE.
   - Click the **OK** button to assign the selected elements to the group named BASE.

   Note: The BASE group will be useful for determining base shears, overturning moments, total vertical loads.

52. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assignment of groups.

Step 7: Assigning Frame Sections

Refer to Figure B-1 for the beam, column and brace element sections.

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

3. In this dialog box:

   - In the Frames area check the Labels box.

   - In the Shells area check the Hide box.

   - Click the **OK** button.

4. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active. Note when the window is active, its title bar will be highlighted.

5. Click the **Up One Gridline** button ⇧ on the main toolbar twice to display the roof level plan at Z=24.

6. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

7. In this dialog box:

   - In the Frames area check the Labels box.

   - In the Shells area check the Hide box.

   - Click the **OK** button.

8. In the window labeled X-Y Plane @ Z=24, click on frame elements 40, 45, 46 and 51 to select them.

   Note: You can refer to Figures B-2 and B-3, as well as the screen, to identify the frame element numbers.

9. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

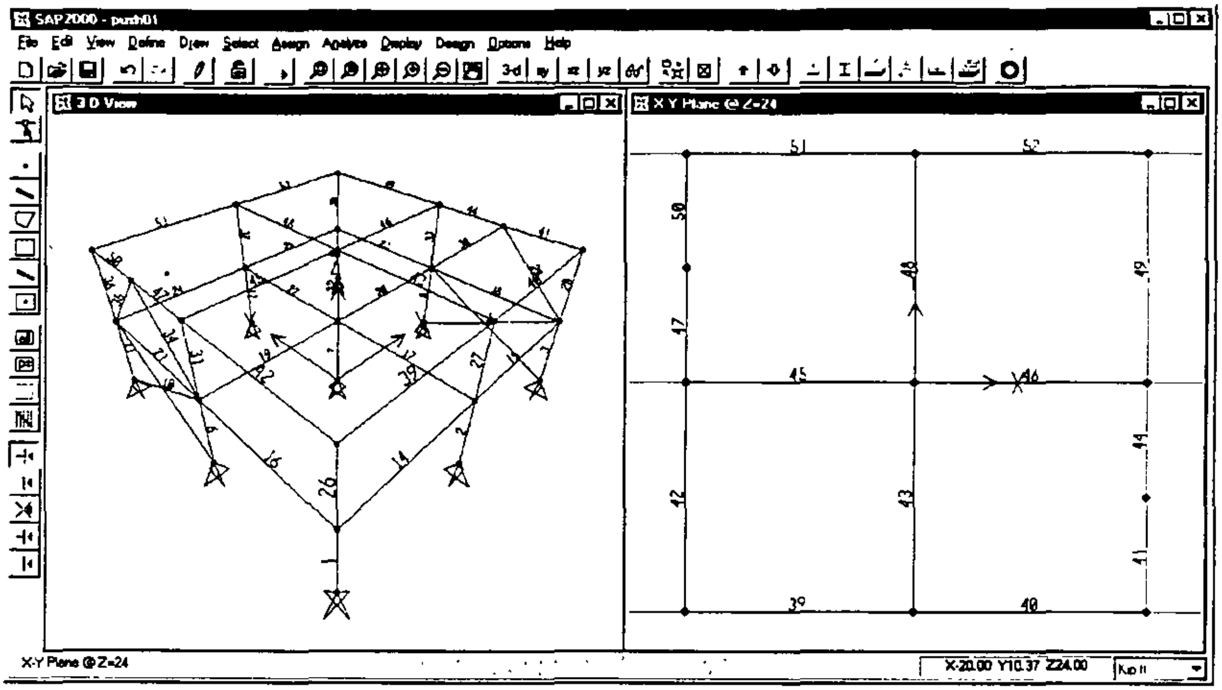


Figure C-15: Frame Element Labels

10. In the Frame Sections area click on the W16X26 name once to highlight it and then click the **OK** button. This assigns the W16X26 property to the selected frame elements. Note the other frame elements have the default FSEC1 assigned.

11. Click the **Show Undeformed Shape** button to clear the display of the assigned sections so that you can see the frame element labels again.

12. In the plan view, click on frame elements 41, 42, 44, 47, 49 and 50 to select them.

13. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

14. In the Frame Sections area click on the W18X35 name once to highlight it and then click the **OK** button.

15. Click the **Show Undeformed Shape** button to clear the display of the assigned sections so that you can see the frame element labels again.

16. In the plan view, click on frame elements 43 and 48 to select them.

17. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

18. In the Frame Sections area click on the W21X44 name once to highlight it and then click the **OK** button.

19. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

20. Click on the group name FRMGIRD once to highlight it and then click the **OK** button.

21. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

22. In the Frame Sections area click on the W24X55 name once to highlight it and then click the **OK** button.

23. Click the **Show Undeformed Shape** button to clear the display of the assigned sections so that you can see the frame element labels again.

24. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active.

25. Click the **Down One Gridline** button on the main toolbar to display the second floor plan at Z=12.

26. From the **Select** menu select **Select** and then **Labels** from the submenu. This will display the Select by Labels dialog box.

27. Click the Element Type drop-down box and select Frame.

    - Click the Element Type drop-down box and select Frame.
    - Type **15** in the Start Label edit box, **24** in the End Label edit box and **9** in the Increment edit box.
    - Click the **OK** button.

28. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

29. In the Frame Sections area click on the W16X26 name once to highlight it and then click the **OK** button.

30. In the plan view, click on frame elements 16, 18, 19, 20, 21 and 23 to select them.

31. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

32. In the Frame Sections area click on the W18X40 name once to highlight it and then click the **OK** button.

33. In the plan view, click on frame elements 17 and 22 to select them.

34. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

35. In the Frame Sections area click on the W24X62 name once to highlight it and then click the **OK** button.

36. From the **Assign** menu select **Clear Display of Assigns** to clear the display of frame assigns from the plan view.

37. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

38. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

39. In this dialog box:

    - In the Frames area uncheck the Labels box.

    - Click the **OK** button.

40. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

41. Click on the group name BRACE1 once to highlight it and then click the **OK** button.

42. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

43. In the Frame Sections area click on the TS5X5X1/4 name once to highlight it and then click the **OK** button.

44. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

45. Click on the group name BRACE2 once to highlight it and then click the **OK** button.

46. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

47. In the Frame Sections area click on the TS4X4X1/4 name once to highlight it and then click the **OK** button.

48. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

49. Click on the group name FRCOLS once to highlight it and then click the **OK** button.

50. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

51. In the Frame Sections area click on the W14X132 name once to highlight it and then click the **OK** button.

52. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

53. Click on the group name BRCOLS once to highlight it and then click the **OK** button.

54. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

55. In the Frame Sections area click on the W8X58 name once to highlight it and then click the **OK** button.

56. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

57. Click on the group name COLS once to highlight it and then click the **OK** button.

58. From the **Select** menu select **Deselect** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

59. In this dialog box:

    - Click on the group name FRCOLS once to select (highlight) it.

    - Hold down the Ctrl key on the keyboard and click on the BRCOLS group name once to highlight it and add it to the selection.

    - Click the **OK** button.

60. From the **Assign** menu select **Frame** and then **Sections...** from the submenu. This will display the Define Frame Sections dialog box.

61. In the Frame Sections area click on the W8X31 name once to highlight it and then click the **OK** button.

62. From the **Assign** menu select **Clear Display of Assigns** to clear the display of frame assigns from the 3-D view.

63. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assignment of frame section properties.

Step 8: Assigning Shell Sections

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

3. In this dialog box:

   - In the Shells area uncheck the Hide box.

   - Click the **OK** button.

4. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active. Note when the window is active, its title bar will be highlighted.

5. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

6. In this dialog box:

   - In the Shells area uncheck the Hide box.

   - Click the **OK** button.

7. Click the **Up One Gridline** button on the main toolbar to display the roof level plan at Z=24.

8. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

9. Click on the group name ROOF once to highlight it and then click the **OK** button.

10. From the **Assign** menu select **Shell** and then **Sections...** from the submenu. This will display the Define Shell Sections dialog box.

11. In the Shell Sections area click on the ROOF name once to highlight it and then click the **OK** button.

12. From the **Assign** menu select **Clear Display of Assigns** to clear the display of shell assigns from the plan view.

13. Click the **Save Model** button on the main toolbar to save the file.

This completes the assignment of shell section properties.

Step 9: Assigning Frame End Releases

We will release the M3 and M2 moment at the ends of all non-moment frame girders, and release M3, M2 and torsional moment at the ends of all braces.

1. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

3. Click on the group name ROOF once to highlight it, then, holding down the Ctrl key, click on the Group Named 2ND to add it to the selection. Click the **OK** button.

4. From the **Select** menu select **Deselect** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

5. Click on the group name FRMGIRD once to highlight it and then click the **OK** button.

6. From the **Assign** menu select **Frame** and then **Releases...** from the submenu. This will display the Frame Releases dialog box.

   Note: The end releases will be applied incorrectly to the roof beams at the braced frames as a result of this selection. We will fix that in items 8 through 18.

7. In this dialog box:

   - Check the Start and End boxes for Moment M22 (Minor) and Moment M33 (Major).

   - Click the **OK** button to apply the releases.

8. Note that releases are applied at the center of the braced frame roof beams. This is not correct since the braced frame roof beams are continuous over the top of the chevron brace.

9. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

10. In this dialog box check the Local Axes box and the Labels box in the Frames area and click the OK button. The local axes arrows appear as shown in Figure C-16.

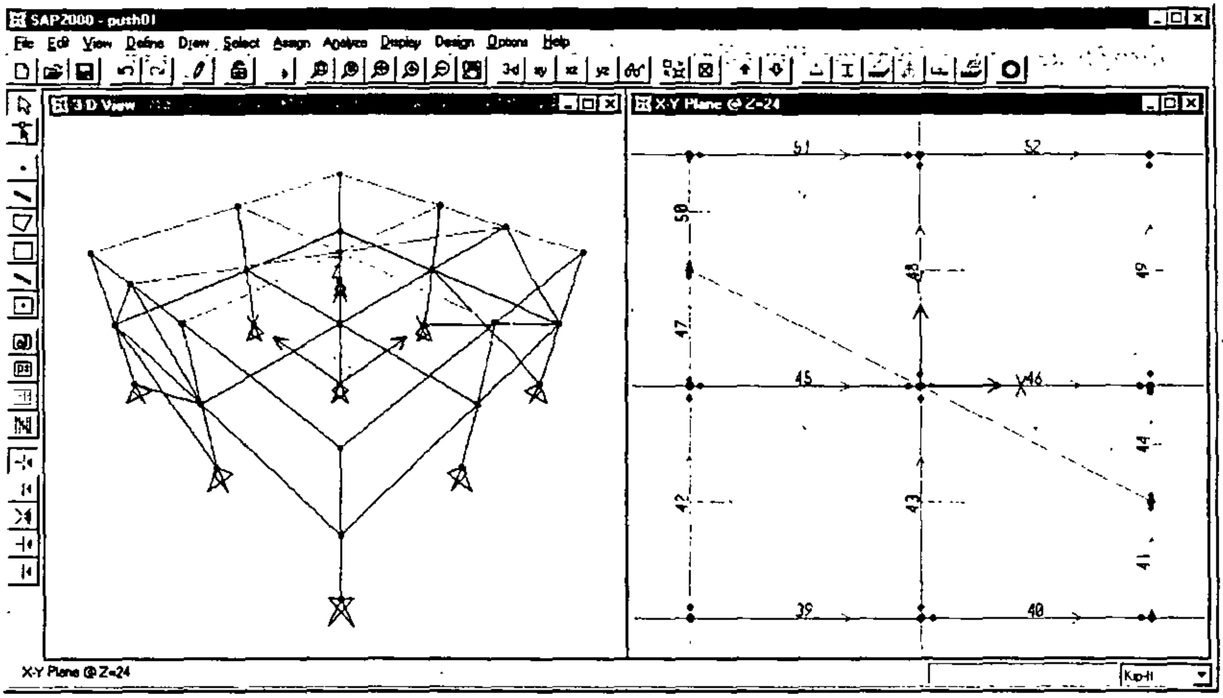


Figure C-16: Frame Element Local Axes Arrows

Note: The local axes arrows are color-coded red, white and blue. The red arrow is always local axis 1, the white arrow is always local axis 2, and the blue arrow is always local axes 3. In the plan view, local axis 2 is pointing straight up and thus we can not see that arrow.

Note: For frame elements, local axis 1 always points along the longitudinal axis of the member from the start joint to the end joint. Thus we can use the local axes to tell which end is the start end for any frame element.

11. Click on frame elements 41 and 47 to select them. Note that for these elements, moments will be released at the start end only.

12. From the **Assign** menu select **Frame** and then **Releases...** from the submenu. This will display the Frame Releases dialog box.

13. In this dialog box:

    - Check the Start box for Moment M22 (Minor) and Moment M33 (Major).
    - Click the **OK** button to apply the releases.

14. Click on frame elements 44 and 50 to select them. Note that for these elements, moments will be released at the end joint only.

15. From the **Assign** menu select **Frame** and then **Releases...** from the submenu. This will display the Frame Releases dialog box.

16. In this dialog box:

    - Check the End box for Moment M22 (Minor) and Moment M33 (Major).

    - Click the **OK** button to apply the releases.

17. The member end releases have now been corrected. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

18. In this dialog box uncheck the Local Axes box and the Labels box in the Frames area and click the OK button.

19. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

20. Click on the group name BRACE1 once to highlight it, then, holding down the Ctrl key, click on the Group Named BRACE2 to add it to the selection. Click the **OK** button.

21. From the **Assign** menu select **Frame** and then **Releases...** from the submenu. This will display the Frame Releases dialog box.

22. In this dialog box:

    - Check the Start and End boxes for Moment M22 (Minor), Moment M33 (Major) and the Start box for Torsion.

    Note: You can apply a torsional release at one end of the frame element or the other, but not both simultaneously. If you applied a torsion release to both ends, the frame element would be free to spin about its longitudinal axis, and thus the structure would be unstable.

    - Click the **OK** button to apply the releases.

23. From the **Assign** menu select **Clear Display of Assigns** to clear the display of frame releases from the plan view.

24. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assignment of frame element end releases.

Step 10: Defining Static Load Cases

We will define five static load cases. They are dead load, live load, earthquake in the X-direction, earthquake in the Y-direction and a load pattern to be used in the pushover analysis.

1. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.

2. In this dialog box:

   - Type **DL** in the Load edit box.
   - Select Dead from the Type drop-down box.
   - Type **0** in the Self Weight Multiplier edit box.
   - Click the **Change Load** button.

     Note: In this example, if we had wanted the program to automatically include the self-weight of the structural members, we would have specified a non zero weight per unit volume in the Material data, and we would have put a self-weight multiplier of 1 on the DL load case only.

   - Type **LL** in the Load edit box.
   - Select Live from the Type drop-down box.
   - Type **0** in the Self Weight Multiplier edit box.
   - Click the **Add New Load** button.
   - Type **EQX** in the Load edit box.
   - Select Quake from the Type drop-down box.
   - Type **0** in the Self Weight Multiplier edit box.
   - Click the **Add New Load** button.
   - Type **EQY** in the Load edit box.
   - Select Quake from the Type drop-down box.
   - Type **0** in the Self Weight Multiplier edit box.
   - Click the **Add New Load** button.

- Type **PUSHPAT** in the Load edit box.
- Select OTHER from the Type drop-down box.
- Type **0** in the Self Weight Multiplier edit box.
- Click the **Add New Load** button.

3. Click the **OK** button.
4. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the definition of static load cases.

Step 11: Assigning Frame Static Loads

Note that the beam span loading is indicated in Figure B-4.

1. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

3. In this dialog box check the Labels box in the Frames area, check the Hide box in the Shells area and click the **OK** button.

4. Click the **Perspective Toggle** button 6σ on the main toolbar. A perspective view of the X-Y plan is displayed.

5. From the **View** menu select **Set Limits....** This displays the Set limits dialog box.

6. In this dialog box type **23** in the Min edit box in the Set Z-Axis Limits area and click the **OK** button to change the limits such that only the roof beam elements show.

7. Select roof beams 39, 40, 51 and 52 by clicking on them.

8. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

9. In this dialog box:

   - Select DL from the Load Case Name drop-down box.

   - In the Load Type and Direction area, select the Forces option, and select Global Z from the drop-down Direction box.

   - In the Options area, select the Add to Existing Loads option.

   - In the Uniform Load area, type **-0.40**, or just type **-.4**.

   Note: Take care to make sure you enter the minus sign with the load. The load is input in the global Z direction, thus the minus sign indicates a downward load.

   - Click the **OK** button to apply the load.

10. Click the **Restore Previous Selection** button ps on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

11. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

12. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area, type **-.12**. Don't forget the minus sign.
    - Click the **OK** button to apply the load.

13. Select roof beams 41, 42, 44, 47, 49 and 50 by clicking on them.

14. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

15. In this dialog box:

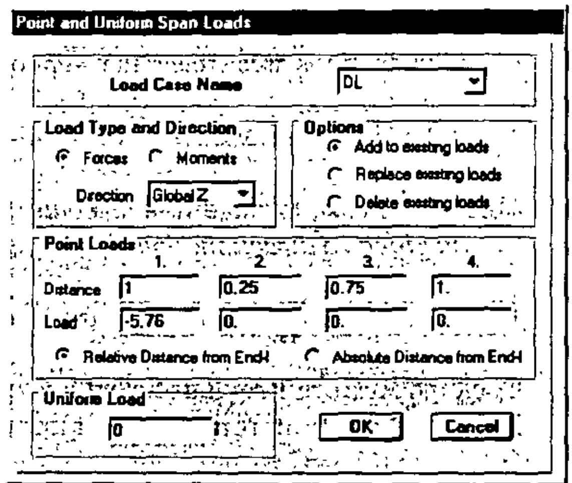
    - Select DL from the Load Case Name drop-down box.
    - In the Uniform Load area, type **-.16**.
    - Click the **OK** button to apply the load.

Note: The point load for beams 41/44 and 47/50 can either be input as a joint load, or as a beam span load at the end of a beam. We will input it as a beam span load.

16. Select roof beams 41 and 47 by clicking on them.

17. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

18. In this dialog box:



    - Select DL from the Load Case Name drop-down box.
    - In the Options area, select the Add to Existing Loads option.
    - In the Point Loads area there are four sets of Distance and Load boxes labeled 1 through 4. In the first set of boxes input **1** in the Distance box and **-5.76** in the Load box. Make sure the Relative Distance From End I option is selected.

- In the Uniform Load area, type **0**.
- Click the **OK** button to apply the load.

19. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

20. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

21. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Point Loads area type **1** in the first Distance box and **-2.88** in the first Load box.
    - Click the **OK** button to apply the load.

22. Select roof beams 42 and 49 by clicking on them.

23. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

24. In this dialog box:

    - Select DL from the Load Case Name drop-down box.
    - In the Point Loads area type **.5** in the first Distance box and **-5.76** in the first Load box.
    - Click the **OK** button to apply the load.

25. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

26. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

27. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Point Loads area type **-2.88** in the first Load box.
    - Click the **OK** button to apply the load.

28. Select roof beams 45 and 46 by clicking on them.

29. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

30. In this dialog box:

    - Select DL from the Load Case Name drop-down box.
    - In the Point Loads area type **0** in the first Distance box and **0** in the first Load box.
    - In the Uniform Load area, type **-.48**.
    - Click the **OK** button to apply the load.

    *Note: Because of the perspective view, you will not see this load after it is applied. To see it, click the **3-D View** button.* 3-d *To return to the perspective view, click the **xy 2-D View** button* xy *and then click the **Perspective Toggle** button.* 6σ

31. Click the **Restore Previous Selection** button ps on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

32. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

33. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area, type **-.24**.
    - Click the **OK** button to apply the load.

34. Select roof beams 43 and 48 by clicking on them.

35. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

36. In this dialog box:

    - Select DL from the Load Case Name drop-down box.
    - In the Point Loads area type **.5** in the first Distance box and **-11.52** in the first Load box.
    - In the Uniform Load area, type **0**.

- Click the **OK** button to apply the load.

37. Click the **Restore Previous Selection** button [ps] on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

38. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

39. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Point Loads area type **-5.76** in the first Load box.
    - Click the **OK** button to apply the load.

40. From the **View** menu select **Set Limits....** This displays the Set limits dialog box.

41. In this dialog box type **11** in the Min edit box and **13** in the Max edit box in the Set Z-Axis Limits area and click the **OK** button to change the limits such that only the second floor beam elements show.

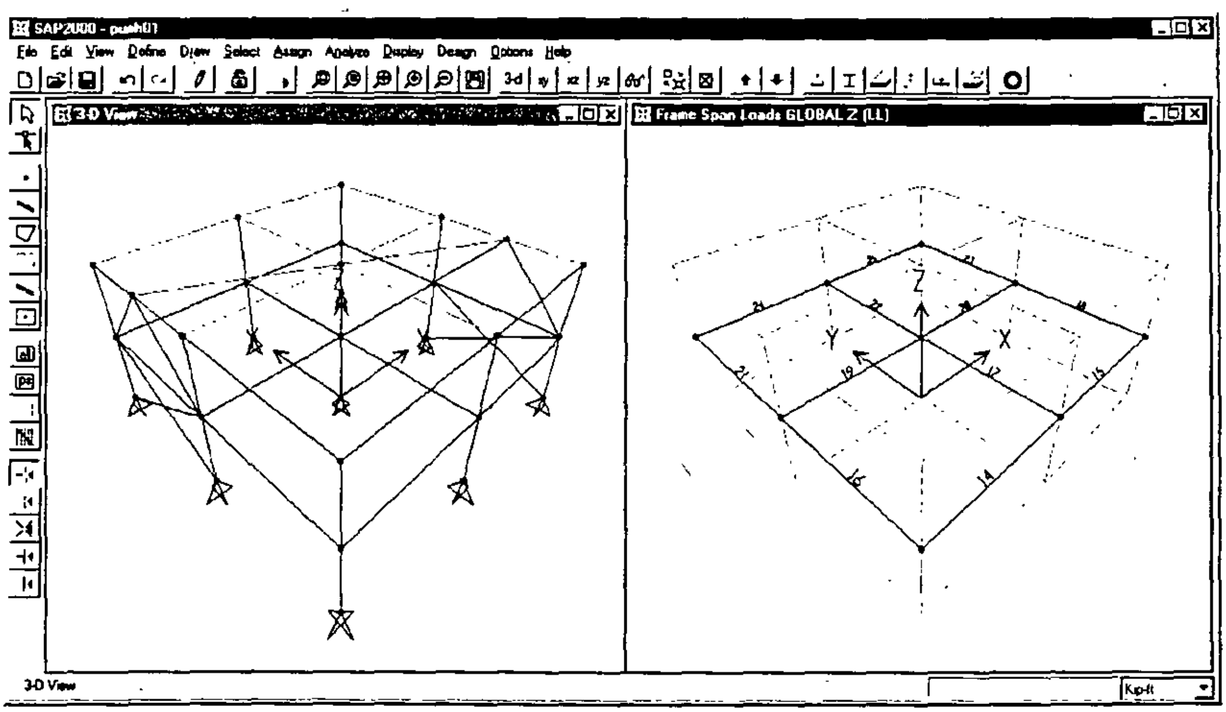


Figure C-17: 3-D View With Limits Set To Show Second Level

42. Click on the **3-D View** button 3-d to change the view to a three dimensional view. See Figure C-17.

43. Select second level beams 14, 15, 24 and 25 by clicking on them.

44. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

45. In this dialog box:

   - Select DL from the Load Case Name drop-down box.
   - In the Point Loads area type **0** in the first Distance box and **0** in the first Load box.
   - In the Uniform Load area, type **-.72**.
   - Click the **OK** button to apply the load.

46. Click the **Restore Previous Selection** button ps on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

47. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

48. In this dialog box:

   - Select LL from the Load Case Name drop-down box.
   - In the Uniform Load area, type **-.3**.
   - Click the **OK** button to apply the load.

49. Select second level beams 16, 18, 21 and 23 by clicking on them.

50. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

51. In this dialog box:

   - Select DL from the Load Case Name drop-down box.
   - In the Point Loads area type **.5** in the first Distance box and **-11.52** in the first Load box.
   - In the Uniform Load area, type **-.24**.
   - Click the **OK** button to apply the load.

52. Click the **Restore Previous Selection** button  on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

53. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

54. In this dialog box:

   - Select LL from the Load Case Name drop-down box.
   - In the Point Loads area type **-7.2** in the first Load box.
   - In the Uniform Load area, type **0**.
   - Click the **OK** button to apply the load.

55. Select second level beams 19 and 20 by clicking on them.

56. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

57. In this dialog box:

   - Select DL from the Load Case Name drop-down box.
   - In the Point Loads area type **0** in the first Distance box and **0** in the first Load box.
   - In the Uniform Load area, type **-.96**.
   - Click the **OK** button to apply the load.

58. Click the **Restore Previous Selection** button  on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

59. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

60. In this dialog box:

   - Select LL from the Load Case Name drop-down box.
   - In the Uniform Load area, type **-0.6**.
   - Click the **OK** button to apply the load.

61. Select second level beams 17 and 22 by clicking on them.

62. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

63. In this dialog box:

    - Select DL from the Load Case Name drop-down box.
    - In the Point Loads area type **0.5** in the first Distance box and **-23.04** in the first Load box.
    - In the Uniform Load area, type **0**.
    - Click the **OK** button to apply the load.

64. Click the **Restore Previous Selection** button [ps] on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

65. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu. This will display the Point and Uniform Span Loads dialog box.

66. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Point Loads area type **-14.4** in the first Load box.
    - Click the **OK** button to apply the load.

67. From the **View** menu select **Set Limits....** This displays the Set limits dialog box.

68. In this dialog box click the **Show All** button in the Set Z-Axis Limits area and click the **OK** button to change the limits such that the entire model is displayed.

69. From the **Assign** menu select **Clear Display of Assigns** to clear the display of beam span loading. Note that the frame element labels remain because they were turned on using the Set Elements command.

70. Click the **xy 2-D View** button [xy] on the main toolbar to return to a plan view. Note the the window title is X-Y Plane @ Z=24.

71. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assignment of frame static loads.

Step 12: Assigning Shell Static Loads

The roof level lateral earthquake loads are assigned as shell static loads.

1. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

3. In this dialog box uncheck the Hide box in the Shells area and click the **OK** button.

4. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

5. Click on the group name ROOF once to highlight it then click the **OK** button.

6. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu. This will display the Shell Uniform Loads dialog box.

7. In this dialog box:

   - Select EQX from the Load Case Name drop-down box.
   - In the Uniform Load area type **.0074** in the Load box and select Global X from the drop-down Dir box.
   - Select Add To Existing Loads in the Options area.
   - Click the **OK** button to apply the load.

8. Click the **Restore Previous Selection** button ps on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

9. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu. This will display the Shell Uniform Loads dialog box.

10. In this dialog box:

    - Select EQY from the Load Case Name drop-down box.
    - In the Uniform Load area type **.0113** in the Load box and select Global Y from the drop-down Dir box.
    - Click the **OK** button to apply the load.

11. Click the drop down box in the status bar to change the units to lb-ft. lb-ft Observe that the shell load shown on the screen is now 11.30 instead of 0.01. Sometimes it is convenient to change units when viewing input loads and output results.

12. Click the drop down box in the status bar to change the units back to kip-ft. Kip-ft

13. From the **Assign** menu select **Clear Display of Assigns** to clear the display of shell loading.

14. Click the **Save Model** button on the main toolbar to save the file.

This completes the assignment of shell static loads.

Step 13: Assigning Joint Static Loads

The second level lateral earthquake loads (load cases EQX and EQY) are assigned as joint static loads. Also the static load pattern to be used in the pushover analysis, PUSHPAT is assigned as joint static loads.

Note the static load pattern for the pushover analysis could be any combination of joint, frame and shell loading.

1. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Down One Gridline** button on the main toolbar to display the second floor plan at Z=12.

3. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

4. In this dialog box uncheck the Labels box in the Frames area, check the Labels box in the Joints area and click the **OK** button. Note we are leaving the Restraints box in the Joints area checked.

5. Click joint 14 to select it.

6. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

7. In this dialog box:

   - Select EQX from the Load Case Name drop-down box.

   - In the Options area, select the Add to Existing Loads option.

   - In the Loads area type **2.67** in the Force Global X edit box.

   - Click the **OK** button to apply the load.

8. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

9. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

10. In this dialog box:

    - Select EQY from the Load Case Name drop-down box.

- In the Loads area type **0** in the Force Global X edit box.
- In the Loads area type **3.83** in the Force Global Y edit box.
- Click the **OK** button to apply the load.

11. Click on joints 11, 13, 15 and 17 to select them.

12. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

13. In this dialog box:

    - Select EQX from the Load Case Name drop-down box.
    - In the Options area, select the Add to Existing Loads option.
    - In the Loads area type **2** in the Force Global X edit box.
    - In the Loads area type **0** in the Force Global Y edit box.
    - Click the **OK** button to apply the load.

14. Click the **Restore Previous Selection** button PS on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

15. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

16. In this dialog box:

    - Select EQY from the Load Case Name drop-down box.
    - In the Loads area type **0** in the Force Global X edit box.
    - In the Loads area type **2.88** in the Force Global Y edit box.
    - Click the **OK** button to apply the load.

17. Click on joints 10, 12, 16 and 18 to select them.

18. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

19. In this dialog box:

- Select EQX from the Load Case Name drop-down box.
- In the Options area, select the Add to Existing Loads option.
- In the Loads area type **1.33** in the Force Global X edit box.
- In the Loads area type **0** in the Force Global Y edit box.
- Click the **OK** button to apply the load.

20. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

21. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

22. In this dialog box:

    - Select EQY from the Load Case Name drop-down box.
    - In the Loads area type **0** in the Force Global X edit box.
    - In the Loads area type **1.92** in the Force Global Y edit box.
    - Click the **OK** button to apply the load.

23. We will now define the PUSHPAT load pattern that will be used in the pushover analysis. It will be a triangular distribution of load over the height of the building. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

    Note: Our purpose here is to input an inverted triangular loading pattern over the height of the building. Thus we will put 1 kip loads at 9 second level joints (9 kips total) and 2 kip loads at 9 roof level joints (18 kips total). We could assign the 9 kips at the second level and 18 kips at the roof level to the joints on a tributary area basis, or some other basis, but we will not consider any such refinement in this example.

24. Click on the group name 2ND and then click the **OK** button.

25. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

26. In this dialog box:

    - Select PUSHPAT from the Load Case Name drop-down box.

- In the Loads area type **1** in the Force Global X edit box.
- In the Loads area type **0** in the Force Global Y edit box.
- Click the **OK** button to apply the load.

27. Click the **Up One Gridline** button on the main toolbar to display the roof level plan at Z=24.

28. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

29. Click on the group name ROOF and then click the **OK** button.

30. Click on joints 22 and 26 to deselect them.

31. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu. This will display the Joint Forces dialog box.

32. In this dialog box:

    - Select PUSHPAT from the Load Case Name drop-down box.
    - In the Loads area type **2** in the Force Global X edit box.
    - Click the **OK** button to apply the load.

33. From the **Assign** menu select **Clear Display of Assigns** to clear the display of joint loads. Note that the joint element labels remain because they were turned on using the Set Elements command.

34. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assignment of joint static loads.

Step 14: Assigning Joint Masses

We will add joint masses consistent with those shown in Table B-1.

1. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click joint 24 to select it.

3. From the **Assign** menu select **Joint** and then **Masses...** from the submenu. This will display the Joint Masses dialog box.

4. In this dialog box:

   - Type **.72** in the Direction 1 edit box in the Masses in Local Directions area.

   - Type **.72** in the Direction 2 edit box in the Masses in Local Directions area.

   - In the Options area, select the Add to Existing Masses option.

   - Click the **OK** button to apply the mass.

   Note: If we wanted to obtain vertical direction mode shapes we should include a Direction 3 mass as well.

5. Click joints 20, 23, 25 and 28 to select them.

6. From the **Assign** menu select **Joint** and then **Masses...** from the submenu. This will display the Joint Masses dialog box.

7. In this dialog box:

   - Type **.48** in both the Direction 1 edit box and the Direction 2 edit box in the Masses in Local Directions area.

   - Click the **OK** button to apply the mass.

8. Click joints 19, 21, 27 and 29 to select them.

9. From the **Assign** menu select **Joint** and then **Masses...** from the submenu. This will display the Joint Masses dialog box.

10. In this dialog box:

    - Type **.3** in both the Direction 1 edit box and the Direction 2 edit box in the Masses in Local Directions area.

- Click the **OK** button to apply the mass.

11. Click the **Down One Gridline** button on the main toolbar to display the second floor plan at Z=12.

12. Click joint 14 to select it.

13. From the **Assign** menu select **Joint** and then **Masses...** from the submenu. This will display the Joint Masses dialog box.

14. In this dialog box:

    - Type **1.45** in both the Direction 1 edit box and the Direction 2 edit box in the Masses in Local Directions area.

    - Click the **OK** button to apply the mass.

15. Click joints 11, 13, 15 and 17 to select them.

16. From the **Assign** menu select **Joint** and then **Masses...** from the submenu. This will display the Joint Masses dialog box.

17. In this dialog box:

    - Type **.9** in both the Direction 1 edit box and the Direction 2 edit box in the Masses in Local Directions area.

    - Click the **OK** button to apply the mass.

18. Click joints 10, 12, 16 and 18 to select them.

19. From the **Assign** menu select **Joint** and then **Masses...** from the submenu. This will display the Joint Masses dialog box.

20. In this dialog box:

    - Type **.55** in both the Direction 1 edit box and the Direction 2 edit box in the Masses in Local Directions area.

    - Click the **OK** button to apply the mass.

21. From the **Assign** menu select **Clear Display of Assigns** to clear the display of joint masses. Note that the joint element labels remain because they were turned on using the Set Elements command.

22. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

23. In this dialog box uncheck the Labels box in the Joints area and click the OK button. This will turn off the display of joint labels.

24. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assignment of joint masses.

Step 15: Assigning Diaphragm Constraints

We will assign a diaphragm constraint at the second level to simulate a rigid diaphragm.

1. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Select all elements at the second level by "windowing".

3. From the **Assign** menu choose **Joint**, and then **Constraints...**from the submenu. This will display the Constraints dialog box.

4. In this dialog box:

   - In the Click To area, click the drop-down box and select Add Diaphragm. This will display the Diaphragm Constraint dialog box.

   - In this dialog box:

     ✓ Type **2NDDIA** in the Constraint Name edit box.
     ✓ Select the Z Axis option in the Constraint Axis area.
     ✓ Click the **OK** button.

   - Click the **OK** button to exit the Constraints dialog box and assign the second floor level diaphragm constraint.

   Note: The joints change colors indicating the constraint has been assigned.

5. From the **Assign** menu select **Clear Display of Assigns** to clear the display of joint constraints.

6. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assigning of diaphragm constraints.

Step 16: Static and Dynamic Analysis (Not Pushover)

In this step we will set the analysis options for the static and dynamic analysis (not pushover) and run the analysis.

1. From the **Analyze** menu select **Set Options....** This will display the Analysis Options dialog box.

2. In this dialog box:

   - Check the Dynamic Analysis check box.

   - Click on the **Set Dynamic Parameters** button. This will display the Dynamic Analysis Parameters dialog box.

   - In this dialog box:

     ✓ Type **6** in the Number Of Modes edit box
     ✓ In the Type of Analysis area select the Eigenvectors option.
     ✓ Accept the other default values in the dialog box.
     ✓ Click the **OK** button.

   - Check the Generate Output check box.

     *Note: The Generate Output option creates output that will be stored in the *.out file where * represents your SAP2000 filename.*

   - Click on the **Set Output Options** button. This will display the Select Output Results dialog box.

   - In this dialog box:

     ✓ Check the Displacements check box

     ✓ Click on the **Select/Show Loads** button adjacent to the Displacements check box. This displays the Select Output dialog box.

     ✓ Click on the EQX load case to highlight it. Hold down the control key and click on the EQY load case to add it to the selection.

     ✓ Click the **OK** button to close the Select Output dialog box.

     ✓ Click the **OK** button to close the Select Output Results dialog box.

   - Click the **OK** button again to exit the Analysis Options dialog box:

*Note: It is important to note that when displacements or force are specified to be printed in the *.out files using the Generate Output option,* ***data for each and every joint or element is printed.*** *You have no control over this. For larger problems this can lead to very large and unwieldy output files. See the notes at the ends of steps 18 and 19 for information on other printing options.*

Note: If you are running a large model, you may want/need to increase the memory allocated to SAP2000 above the default 2000 Kb. You can do this in the Analysis options dialog box.

3. Click the Run Analysis button on the main toolbar.

Note: You can also click the ***Analyze*** *menu and select* ***Run*** *or select* ***Run Minimized*** *to run the analysis. Run minimized will perform the execution in the background, i.e., it will allow you to minimize SAP2000 while the analysis is being carried out. It also provides a* ***Cancel*** *button that allows you to stop a run that is in progress.*

4. A window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display as shown in Figure C-18.

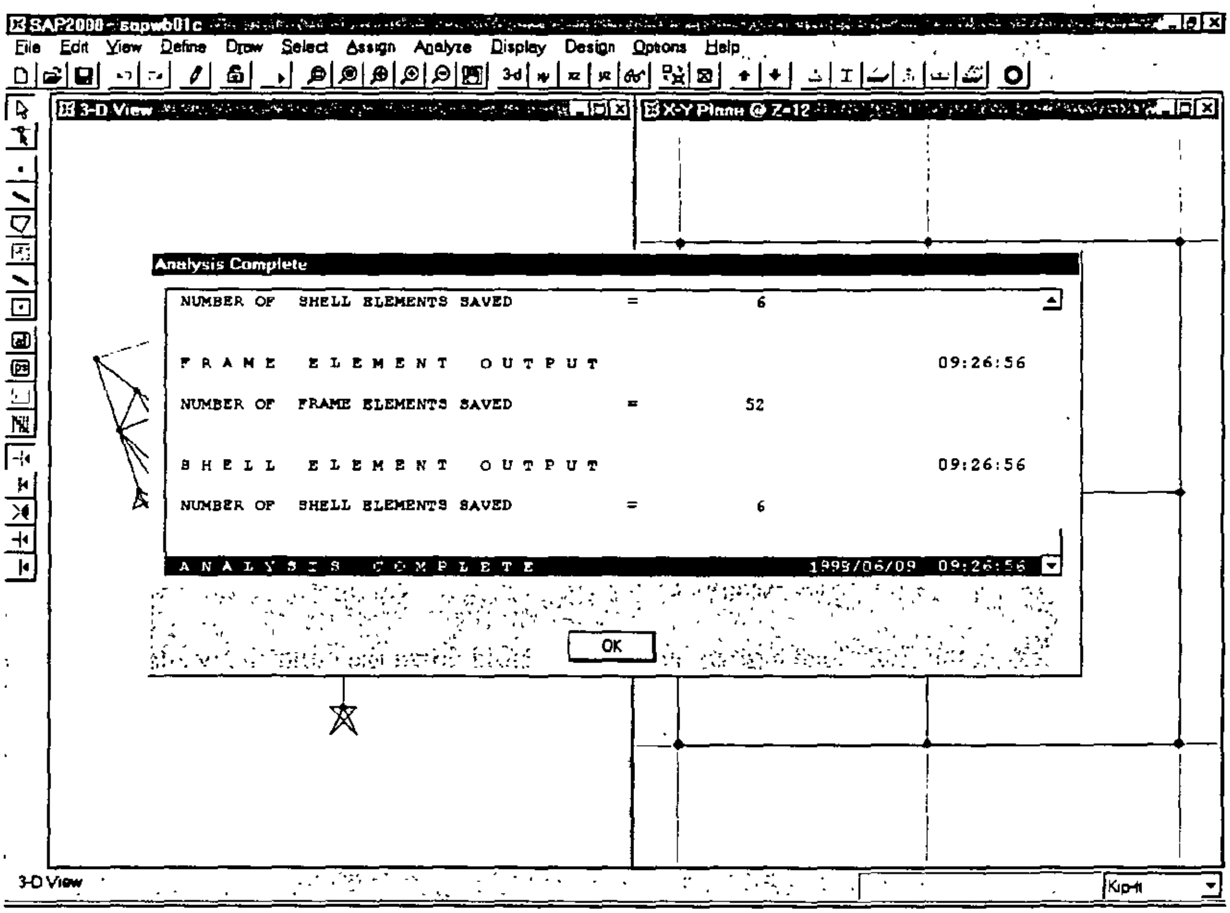


Figure C-18: Screen Message When Analysis Is Complete

5. Use the scroll bar to review the analysis messages and check for any error or warning messages (there should be none).

   *Note: The information in the scrolling analysis window can also be found in the *.log file, where the * represents your filename.*

6. Click the **OK** button in the Analysis window to close it. Note that the 3-D window now shows the first mode shape.

This completes the static and dynamic analysis.

Step 17: Reviewing Mode Shapes

In this step we will review the mode shapes.

1. Note that the first mode shape is currently displayed in the 3-D View window. Click on this window to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Display Mode Shape** button on the main toolbar, (or select **Show Mode Shape**... from the **Display** menu). The Mode Shape dialog box appears.

3. In this dialog box:

   - Set the mode number to 1 in the Mode Number area.

   - Set the scale factor to 1 in the Scale Factor area.

   - Select (i.e., check) the Wire Shadow and Cubic Curve options in the Options area.

   - Click the **OK** button to redisplay the first mode.

4. Click the **Start Animation** button Start Animation, located in the status bar at the bottom of the SAP2000 window, to animate the mode shape.

5. Click the **Right Arrow** button ⇨, located in the status bar at the bottom of the screen, to view the next mode shape.

   Note: When viewing mode shapes, the right and left arrow buttons, located in the status bar at the bottom of the screen, provide an easy way to view the next (right arrow), or previous (left arrow) mode shape.

6. Using the Right Arrow button, review modes 2 through 6. Note that mode 1 is predominantly in the X-direction; we will use this information when creating pushover load cases.

   *Note: Additional information on mode shapes including modal periods and frequencies, modal participation factors and modal participating mass ratios can be found in the *.OUT file.*

7. After viewing mode 6, click the **Stop Animation** button Stop Animation, located in the status bar at the bottom of the SAP2000 window, to stop the mode shape animation.

8. Click the **Show Undeformed Shape** button to clear the display of mode shapes.

This completes the review of displaying mode shapes.

Step 18: Reviewing Deformed Shapes

In this step we will demonstrate methods for reviewing the deformed shapes and displacements.

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Display Static Deformed Shape** button, (or select **Show Deformed Shape...** from the **Display** menu). The Deformed Shape dialog box appears.

3. In this dialog box:

   - Select the EQX Load Case from the Load drop-down box.

   - Select the Auto scaling option in the Scaling area.

   - Check both the Wire Shadow and the Cubic curve boxes in the Options area.

   - Click the **OK** button to display the deformed shape.

4. Click the **Start Animation** button [Start Animation], located in the status bar at the bottom of the SAP2000 window, to animate the deformed shape.

5. Click the **Stop Animation** button [Stop Animation], located in the status bar at the bottom of the SAP2000 window, to stop the deformed shape animation.

6. Click the **Right Arrow** button [⇨], located in the status bar at the bottom of the screen, to view the next deformed shape, based on the EQY static load.

   Note: When viewing deformed shapes, the right and left arrow buttons, located in the status bar at the bottom of the screen, provide an easy way to view the next (right arrow), or previous (left arrow) deformed shape. You can easily cycle through all of the load cases in this manner.

7. Right click on any joint in the 3-D view to bring up a dialog box with displacements for all six degrees of freedom (UX, UY, UZ, RX, RY, RZ) for that joint. When done viewing these displacements, right click another joint, or left click anywhere to close the dialog box.

8. From the **Display** menu select **Set Output Table Mode...** to display the Select Output dialog box.

9. All of the load cases in the Select Output dialog box should be selected (higlighted). If they are not, click on each one while holding down the Ctrl key to select them. Click the **OK** button to accept this selection.

   Note: If we wanted to, we could select only one, or a few load cases, instead of all of them.

10. Right click any joint to get a table of displacements for the joint. Note you can print this table by selecting **Print** from its associated **File** menu. When done viewing this table click the "X" in the upper right-hand corner to close it.

    Note: You can also right click a frame element or a shell element to get a table of forces for that element.

    *Note: You can also print joint displacements from the graphic interface without having to first view them in a table. With this method you can control the elements for which the output is provided (printed). After the analysis has been run, select all of the joints for which you want displacement output. (Note that if you don't select any elements, then this feature will give you output for all elements.) Then select **Print Output Tables...** from the **File** menu. Fill in the resulting Print Output Tables dialog box, remembering to select the load cases for which you want output, and click the **OK** button. Note that with the Print Output Tables feature you have the option of printing to the printer or to a file. If you print to a file, an ASCII file is created which you can review in a text editor or word processor.*

11. Click the **Show Undeformed Shape** button to clear the display of deformed shapes.

This completes the review of deformed shapes and displacements.

Step 19: Reviewing Forces and Stresses

In this step we will demonstrate methods for reviewing the forces and stresses in frame and shell elements.

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Member Force Diagram for Frames** button| **F** |, (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.

3. In this dialog box:

   - Select the EQX Load Case from the Load drop-down box.

   - Select the Moment 3-3 option in the Component area.

   - Select the Auto scaling option in the Scaling area.

   - Check the Fill Diagram box.

   - Click the **OK** button to display the moment diagrams as filled shapes.

4. Right click on any element to display a dialog box that allows you to see the moment at any point along the element. When done viewing, right click on another element or left click anywhere to close the dialog box.

5. Click the **Member Force Diagram for Frames** button| **F** |, again (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.

6. In this dialog box:

   - Uncheck the Fill Diagram box.

   - Check the Show Values on Diagram box.

   - Click the **OK** button to display the moment diagrams with critical values shown.

7. Right click on any element to display a dialog box that allows you to see the moment at any point along the element. When done viewing, right click on another element or left click anywhere to close the dialog box.

8. Click the **Element Force/Stress Contours for Shells** button **S**, (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Shells...** from the submenu). The Element Force Stress/Contours for Shells dialog box appears.

9. In this dialog box:

   - Select the EQX Load Case from the Load drop-down box.
   - Select the Forces option button.
   - Select the F12 component by checking its box.
   - Accept the rest of the default values.
   - Click the **OK** button to display the shell element forces.

10. Suppose we now want to see the Y-direction force transferred from the roof diaphragm to joints 19, 23, 26 and 27 in static load case EQY. To do this we will use the Group Joint Force Sum feature. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active. Note when the window is active, its title bar will be highlighted.

11. Click the **Up One Gridline** button on the main toolbar to display the roof level plan at Z=24.

12. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

13. In this dialog box check the Labels box in the Joints area, uncheck the Hide box in the Shells area if it is checked, check the Labels box in the Shells area and click the OK button. This will turn on the display of joint labels and shell labels.

14. Click on joints 19, 23, 26 and 27 and on shells 2, 4 and 6.

15. From the **Assign** menu choose **Group Name....** This will display the Assign Group dialog box.

16. In this dialog box:

   - Type **ROOFSHR** in the edit box at the top of the Groups area.
   - Click the **Add New Group Name** button to define a group named ROOFSHR.
   - Click the **OK** button to assign the selected elements to the group named ROOFSHR.

17. From the **Display** menu select **Set Output Table Mode...** to display the Select Output dialog box.

18. All of the load cases in the Select Output dialog box should be selected (higlighted). If they are not, then click on each one while holding down the Ctrl key to select them. Click the **OK** button to accept this selection.

    Note: If we wanted to, we could select only one, or a few load cases, instead of all of them.

19. From the **Display** menu select **Show Group Joint Force Sums...** to display the Select Groups dialog box.

20. In that dialog box click on the ROOFSHR group to highlight it and click the **OK** button.

    Note: We could select more than one group at this time if we wanted to.

21. The group joint force sum for the ROOFSHR group is displayed for each load case. The Y-direction force transferred from the roof diaphragm to joints 19, 23, 26 and 27 in static load case EQY is the FY force for load EQY.

    *Note: We can use groups to define section cuts through shell elements (and frame elements) at any location and then use the Group Joint Force Sum feature to see the forces acting at that section cut. To do this, imagine a section cut through the structure. The section cut may be through the entire structure or through a portion of the structure. Select all of the elements that the section cut passes through, and select all of the joints connected to those elements on one side of the section cut. Note you must select both the shell and/or frame elements <u>and</u> the joints. Define a group that includes all of the selected items. Use the **Show Group Joint Force Sum** option on the **Display menu** to show the forces at the section cut.*

22. We can display the base shear using the BASE group we previously defined. From the **Display** menu select **Show Group Joint Force Sums...** to display the Select Groups dialog box. In that dialog box click on the BASE group to highlight it and click the **OK** button.

23. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

24. In this dialog box uncheck the Labels box in the Joints area, uncheck the Labels box in the Shells area and click the OK button. This will turn off the display of joint labels and shell labels.

25. We can display output tables of element forces on the screen.

26. From the **Display** menu select **Set Output Table Mode...** to display the Select Output dialog box.

27. In the Select Output dialog box click on the EQX Load Case to highlight it and then hold down Ctrl key and click on the EQY Load Case to highlight it. Click the OK button to accept this selection.

28. Right click any frame or shell element to get a table of forces for that element.

    Note: You can also right click a joint to get a table of joint displacements.

    Note: You can also print frame and shell element forces from the graphic interface without having to first view them in a table. With this method you can control the elements for which the output is provided (printed). After the analysis has been run, select all of the frame and/or shell elements for which you want force output. (Note that if you don't select any elements, then this feature will give you output for all elements.) Then select ***Print Output Tables...*** *from the* ***File*** *menu. Fill in the resulting Print Output Tables dialog box, remembering to select the load cases for which you want output, and click the* ***OK*** *button. Note that with the Print Output Tables feature you have the option of printing to the printer or to a file. If you print to a file, an ASCII file is created which you can review in a text editor or word processor.*

This completes the review of forces and stresses.

Step 20: Performing a Steel Design Stress Check

Now we will perform a steel design stress check.

1. From the **Options** menu, click **Preferences...** to display the Preferences dialog box.

2. In this dialog box:

   - Select the Steel tab.

   - Click the Steel Design Code drop-down box to review the available options. Select the AISC-ASD89 option.

   - Click the **OK** button to close the Preferences dialog box.

3. Click on the **Design** menu. Note that the **Steel Design** option on the menu is checked indicating a steel design will be done.

   Note: Either the Steel Design or Concrete Design option can be checked on the Design menu, but not both at the same time.

4. On the **Design** menu choose **Select Design Combos**. The Design Load Combinations Selection dialog box is displayed.

   Note: SAP2000 automatically creates appropriate load combinations for the selected design code. For this model, using AISC-ASD89, these ten default load combinations will include DL, DL + LL, DL + LL + EQX, DL + LL - EQX, DL + LL + EQY, DL + LL – EQY, DL +EQX, DL – EQX, DL + EQY and DL – EQY.

5. In this dialog box:

   - Click on the DSTL1 combination in the Design Combos box to highlight it.

   - Click the **Show** button to review the combination.

     *Note: Clicking the **Remove** button when a Design Combo is highlighted will remove that combination from the Design Combo box and put it in the List of Combos box. Only the design combinations in the Design Combo box are in the design stress check.*

     *Note: You can also add your own combinations to the design combinations. To define a new load combination, click **Load Combinations...** on the **Define** menu. To add the new load combination to the design list, highlight the new load combination name, that will be in the List of Combos box in the Design Load Combinations Selection dialog box, and click the **Add** button.*

   - Review the other combinations in a similar manner.

- Press the **OK** button twice to close all dialog boxes.

6. Click in the window labeled 3-D View to make sure it is active.

7. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

8. In this dialog box check the Hide box in the Shells area and click the OK button. This will turn off the display of shells.

9. On the **Design** menu choose **Start Design/Check of Structure**. The design check of the structure proceeds.

10. When the design is complete, the member stress ratios are displayed on the structure as shown in Figure C-19. Note that many of the beam stress ratios are quite high. This has occurred because by default SAP2000 assumes that the beams are unsupported for their full length. For simplicity we will assume that the unsupported length for all beams is two feet, except for the roof beam at the braced frasme where it is one foot. This makes an unbraced length ratio of 0.0833 for all beams. We will make this change and rerun the design.

Note: The assumption that the unsupported length for all beams is 2 feet may not be realistic in a real building.

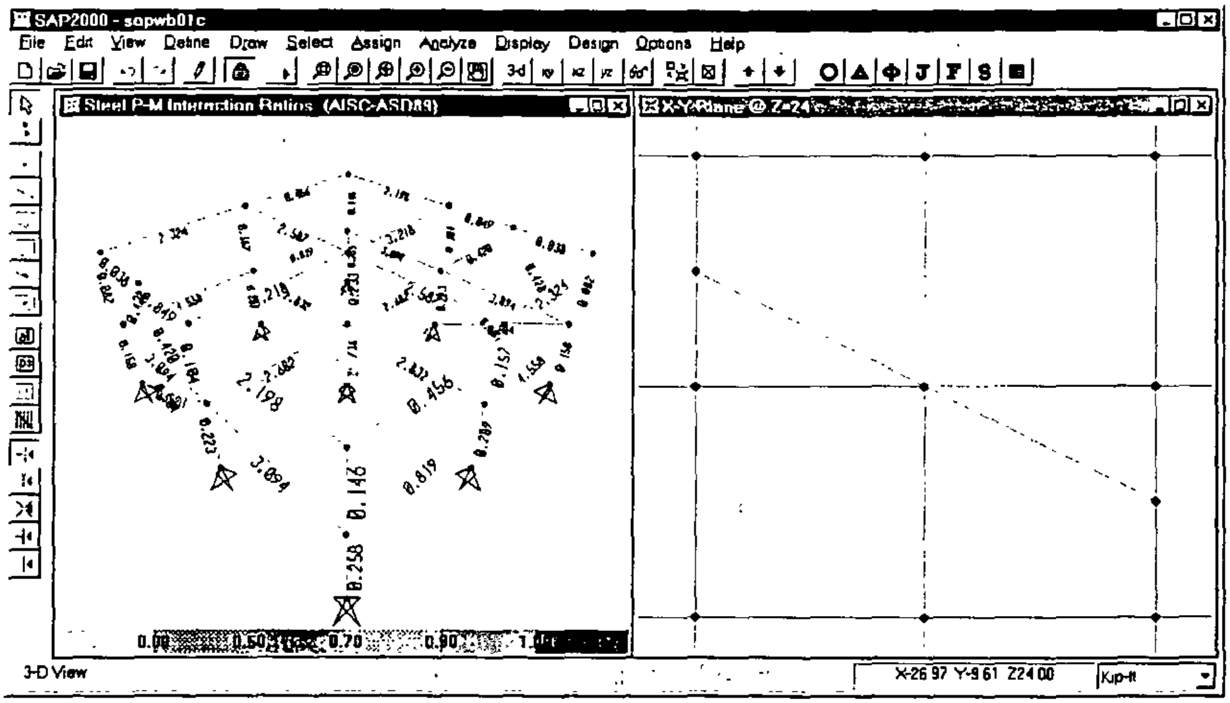


Figure C-19: Initial Member Stress Ratios

11. From the **Select** menu select **Select** and then **Groups...** from the submenu. This will display the Select Groups dialog box.

12. Click on the group name ROOF once to highlight it, then, holding down the Ctrl key, click on the group named 2ND to add it to the selection. Click the **OK** button.

13. On the **Design** menu choose **Redefine Element Design Data**. The Element Overwrite Assignments dialog box is displayed.

14. In this dialog box:

    - In the Assignment Options area check the box labeled Unbraced Length Ratio (Minor, LTB) and then type **.0833** in the Unbraced Length Ratio (Minor, LTB) edit box.

    - Click the **OK** button to close the Preferences dialog box.

15. On the **Design** menu choose **Start Design/Check of Structure**. The second design check of the structure proceeds.

16. When the design is complete, the member stress ratios are displayed on the structure as shown in Figure C-20.

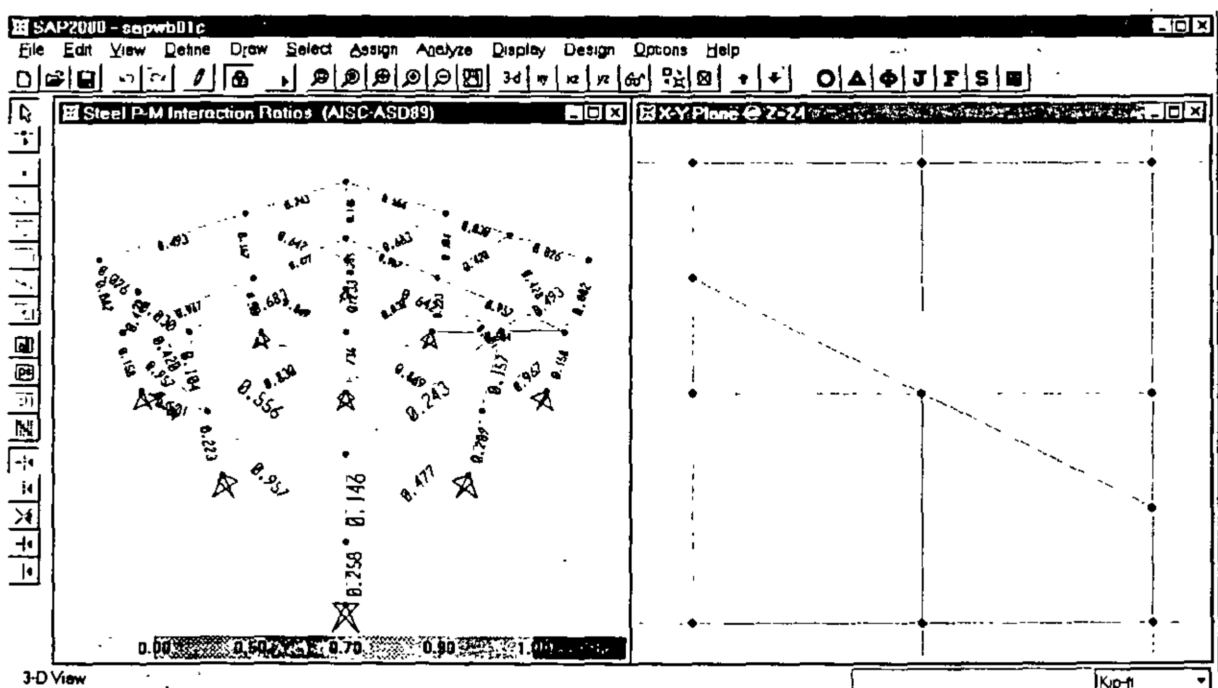


Figure C-20: Final Member Stress Ratios

17. Right click on any member to bring up the Steel Stress Check Information dialog box for that member. This dialog box shows details of the interaction ratio for each design load combination at each design station along the element.

18 To get additional detail on any item in this dialog box, highlight the item and click the Details button. A detailed form with design information, similar to that shown in Figure C-21 will appear. Note you can print this form by clicking the associated File menu and selecting **Print**. Click the "X" in the upper right-hand corner of this detailed form to close it.

Note: If this form is too big to fully fit on your screen, you may have to increase your screen resolution. Typically, it should fit if your resolution is 800 x 600 and you use small fonts or 1024 x 768 and you use large fonts.

```
Steel Stress Check Information  AISC-ASD89
File

STEEL SECTION CHECK    Units: Kip-ft

Frame ID: 14  Station Loc: 24.000  Section ID: W24X55
Element Type:  Moment Resisting   Classification:  Compact

L=24.000
A=0.113  i22=0.001  i33=0.065
s22=0.005  s33=0.066  r22=0.112  r33=0.761
E=4176000.000  fy=5184.000

P-M33-M22 Demand/Capacity Ratio is   0.477 = 0.000 + 0.477 + 0.000

STRESS CHECK FORCES & MOMENTS
                        P        M33          M22          U2          U3
  Combo   DSTL3      0.000    -144.323      0.000      20.638       0.000

AXIAL FORCE & BIAXIAL MOMENT DESIGN   (H2-1)
                       fa          Fa          Ft
                   Stress   Allowable   Allowable
  Axial             0.000    2855.653    3110.400

                       fb          Fb          Fe       Cm        K        L       Cb
                   Stress   Allowable   Allowable   Factor   Factor   Factor   Factor
  Major Bending  2177.078    3421.440   21604.658    1.000    1.000    1.000    1.000
  Minor Bending     0.000    3888.000   67114.541    1.000    1.000    0.083

SHEAR DESIGN
                       fv          FV      Stress
                   Stress   Allowable       Ratio
  Major Shear     319.208    2073.600       0.115
  Minor Shear       0.000    2073.600       0.000
```

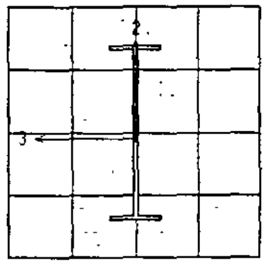


Figure C-21: Detailed Steel Stress Check Information

19. Clicking on the Redesign button will take you to the Element Overwrite Assignments dialog box discussed in item 13.

20. Click the **OK** button to close the Steel Stress Check Information dialog box.

21. Now we will confirm the unbraced length ratios that were used. From the **Design** menu, select **Display Design Info...** to display the Display Design Results dialog box.

22. In this dialog box:

   - Select the Design Input option.

   - In the design input drop-down box select Unbraced L_ratios.

   - Click the **OK** button to display the unsupported length ratios. They are displayed as Major Unbraced Length Ratio, Minor Unbraced Length Ratio.

23. If you wanted to print a picture of the model with the unbraced length ratios displayed you could do so by selecting **Print Graphics** from the **File** menu now.

24. If you wanted to print design information to the printer, or to a file, you could do so by selecting **Print Design Tables...** from the **File** menu and completing the resulting dialog box. Note that if you select members prior to entering this dialog box, you have the option of printing output for the selected members only.

25. Click the **Show Undeformed Shape** button to clear the display of unsupported length ratios.

26. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

27. In this dialog box uncheck the Hide box in the Shells area and click the **OK** button. This will turn the display of shells back on.

The completes the review of performing a steel design stress check. This also completes Part 1 of this tutorial.

D. Tutorial Part 2 – Pushover Analysis

This portion of the tutorial explores the pushover analysis capabilities of SAP2000. It is assumed that you have basic knowledge of the pushover concept, and that you are familiar with the ATC-40 and FEMA-273 documents.

If you did not complete part C of the tutorial, and thus create the basic model, then you should open the file named Sapwb01c.sdb that was supplied with this tutorial. Note that in a typical analysis you should have, at the very least, run and reviewed the results of a static analysis to verify the behavior of your model prior to running a pushover analysis.

SAP2000 has extensive online help for the pushover analysis. Press F1 from within a pushover related dialog box to get context sensitive help related to the dialog box. To see all of the help items related to pushover analysis, select **Search For Help On...** from the **Help** menu, select the Index Tab, highlight the Pushover index entry and click the **Display** button.

Step 1: Defining Hinge Properties (Pushover)

In this tutorial we will use the default hinge properties, so it is not necessary to define any new properties. In this step we will review the process of viewing already defined hinge properties, and will discuss the process of defining new hinge properties. This step can be skipped if you do not want to review the hinge properties.

Background: There are three types of hinge properties in SAP2000. They are default hinge properties, user-defined hinge properties and generated hinge properties. Only default hinge properties and user-defined hinge properties can be assigned to frame elements. When these hinge properties are assigned to a frame element, the program automatically creates a different generated hinge property for each and every hinge.

Default hinge properties can not be modified. They also can not be viewed because the default properties are section dependent. The default properties can not be fully defined by the program until the section that they apply to is identified. Thus to see the effect of the default properties, the default property should be assigned to a frame element, and then the resulting generated hinge property should be viewed. The built-in default hinge properties are typically based on FEMA-273 and/or ATC-40 criteria.

User-defined hinge properties can be either be based on default properties or they can be fully user-defined. When user-defined properties are based on default properties, the hinge properties can not be viewed because, again, the default properties are section dependent. When user-defined properties are not based on default properties, then the properties can be viewed and modified.

The generated hinge properties are used in the analysis. They can be viewed, but they can not be modified. Generated hinge properties have an automatic naming convention of LabelH#, where Label is the frame element label, H stands for hinge, *and # represents the hinge number. The program starts with hinge number 1 and increments the hinge number by one for each consecutive hinge applied to the frame element. For example if a frame element label is F23, the generated hinge property name for the second hinge applied to the frame element is F23H2.*

The main reason for the differentiation between defined properties (in this context, defined means both default and user-defined) and generated properties is that typically the hinge properties are section dependent. Thus it would be necessary to define a different set of hinge properties for each different frame section type in the model. This could potentially mean that a very large number of hinge properties would need to be defined by the user. To simplify this process, the concept of default properties is used in SAP2000. When default properties are used, the program combines its built-in default criteria with the defined section properties for each element to generate the final hinge properties. The net effect of this is that you do significantly less work defining the hinge properties because you don't have to define each and every hinge.

1. From the **Define** menu choose **Hinge Properties....** This will display the Define Frame Hinge Properties dialog box.

2. Note there are four default hinge properties defined. They are Default-M3, Default-P, Default-PMM, and Default-V2. Highlight the Default-M3 property. Note that the **Modify/Show Property** button is inactive because you can not view or modify default hinge properties.

3. Check the Show Generated Props check box. If there were generated hinge properties they would now appear in the Defined Hinge props list box. However, since we haven't yet assigned any hinge properties, the program hasn't generated any, and thus none appear in the list box.

4. Highlight the Default-V2 hinge property and click on the **Define New Property** button. The Frame Hinge Property Data dialog box appears.

   *Note: When you highlight a property in the Defined Hinge Props area of the Define Frame Hinge Properties dialog box, and then click the **Define New Property** button, the new property will default to being the same as the highlighted property.*

5. Note that the Hinge Type is Shear V2, and that the associated Default check box is checked.

6. Type **USER** in the Property Name edit box.

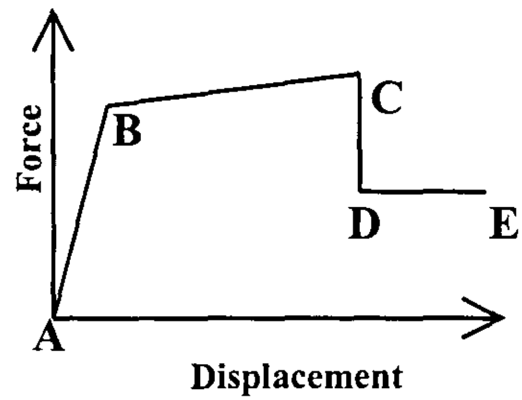
7. Click on the Default check box to uncheck it, and then click the **Modify/Show For V2** button. This displays the Frame Hinge Property Data For USER dialog box.

8. In this dialog box:

- In the Force-Displacement spreadsheet and diagram area:

  - ✓ Note that the symmetric check box is checked and that the hinge property force-displacement diagram is symmetric. In the diagram, the axes are shown in red. The horizontal axis is displacement and the vertical axis is force.

    Note: When the symmetric check box is checked, the upper portion (points B-, C-, D- and E-) of the input spreadsheet for the hinge force-displacement properties is gray. You can not edit these values; they are automatically picked up from symmetry with points B, C, D and E.



  - ✓ Change the Force/Yield value for point C from 1.25 to 1.5 by typing **1.5** into the appropriate spreadsheet cell and then clicking in any other spreadsheet cell.

  - ✓ Note the change in shape of the force-displacement diagram. The diagram is scaled to fit within the plot area.

  - ✓ Note that the Force/Yield value for point C- has changed also from -1.25 to -1.5.

  - ✓ Uncheck the symmetric check box by clicking it. Note that points B- through E- are no longer gray. Also note that in the Acceptance Criteria area, the Negative column is no longer gray.

  - ✓ Change the Force/Yield value for point C- from -1.5 to -2 and click in another spreadsheet cell. Again note the change in shape of the force-displacement diagram.

- In the Scaling area note that the Calculate Yield Force and Calculate Yield Displacement check boxes are checked by default. This means that the program will automatically calculate these values. We could, for example, overwrite the yield force by unchecking the Calculate Yield Force check box and typing in an appropriate value in the Yield Moment edit box. For this tutorial we will accept the default and have the program calculate the yield values.

- In the Acceptance Criteria area we will also accept the default values. Note that since the Symmetric box is not checked both Positive and Negative values are input. If the Symmetric box was checked, only Positive values would be input (the Negative column would be gray and inactive).

  Note: The acceptance criteria are input as the ratio of displacement over yield displacement, similar to the force-displacement spreadsheet.

- In the Type area select the Force-Displacement option. This controls whether the data in the spreadsheet and the acceptance criteria are interpreted by the program as force-displacement or stress-strain. The stress-strain option is only available for axial and shear hinges (uncoupled).

  Note: When the Type option is set to stress-strain, a hinge length must be input. The hinge length can be input as an absolute length or a relative length. The relative length is relative to the clear length of the frame element between rigid end offsets. If the stress-strain option is chosen, the program internally transforms the data into a force-displacement format for analysis. The force is calculated as the stress times the frame member axial area (σA) for axial hinges and the stress times the frame member shear area (σA_V) for shear hinges. The displacement is calculated as the strain times the hinge length (εL).

- Click the **OK** button three times to accept the USER hinge property definition.

- We will now delete the USER hinge property since we are not going to use it.

- From the **Define** menu choose **Hinge Properties....** This will display the Define Frame Hinge Properties dialog box.

- Highlight the USER hinge property and click on the **Delete Property** button. Click the **Yes** button when the program asks if it is OK to delete Frame Hinge USER from list.

- Click the **OK** button to accept the change in hinge properties and exit the Define Frame Hinge Properties dialog box.

  *Note: If you click the **Cancel** button the property will not be deleted.*

This completes the review of defining hinge properties.

Step 2: Assigning Hinge Properties (Pushover)

We will assign Default-PMM hinges to each end of the moment frame columns and the braced frame columns. We will assign Default-M3 hinges to each end of the moment frame beams. We will assign Default-P to the center of each brace.

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. From the **Select** menu choose **Select**, and then **Groups...** from the submenu. This will display the Select Groups dialog box.

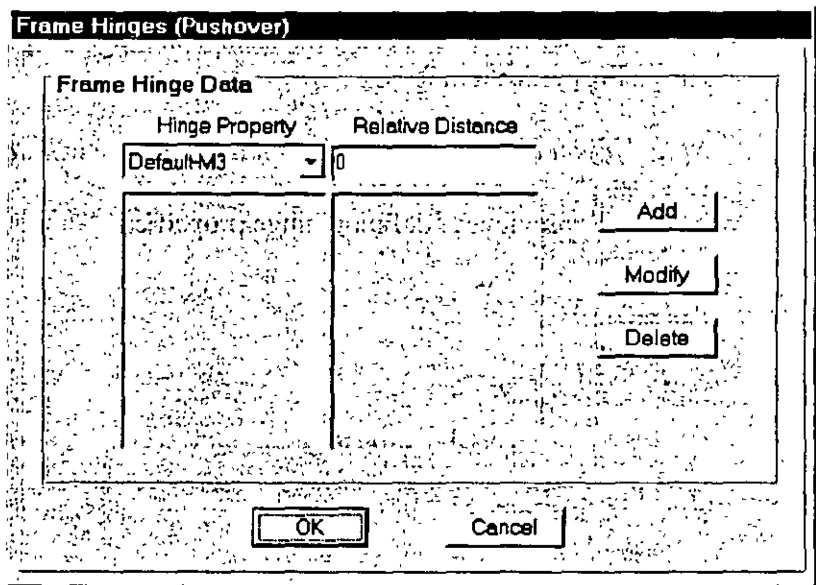
3. Highlight the group named FRCOLS by clicking on it.

4. Hold down the CTRL key and highlight the group named BRCOLS by clicking on it to add it to the selection.

5. Click the **OK** button to select all of the elements in the FRCOLS and BRCOLS groups.

6. From the **Assign** menu choose **Frame**, and then **Hinges (Pushover)...** from the submenu. This will display the Frame Hinges (Pushover) dialog box.

7. In this dialog box:



   - Select Default-PMM in the Hinge Property drop-down box.

   - Type **0** in the Relative Distance edit box.

     Note: Relative distance is measured from the I-end of the beam, and is based on the clear length between rigid end offsets. A "0" relative distance indicates the hinge is at the I-end of the beam. A "1" relative distance indicates the hinge is at the j-end of the beam. Hinges may be located anywhere along the length of the beam.

   - Click the **Add** button to add a PMM hinge at the I-end (relative distance equals 0) end of the beam.

- Type **1** in the Relative Distance edit box.

- Click the **Add** button to add a PMM hinge at the J-end (relative distance equals 1) end of the beam.

- Click the **OK** button to assign the hinge properties. The model now appears as shown in Figure D-1.

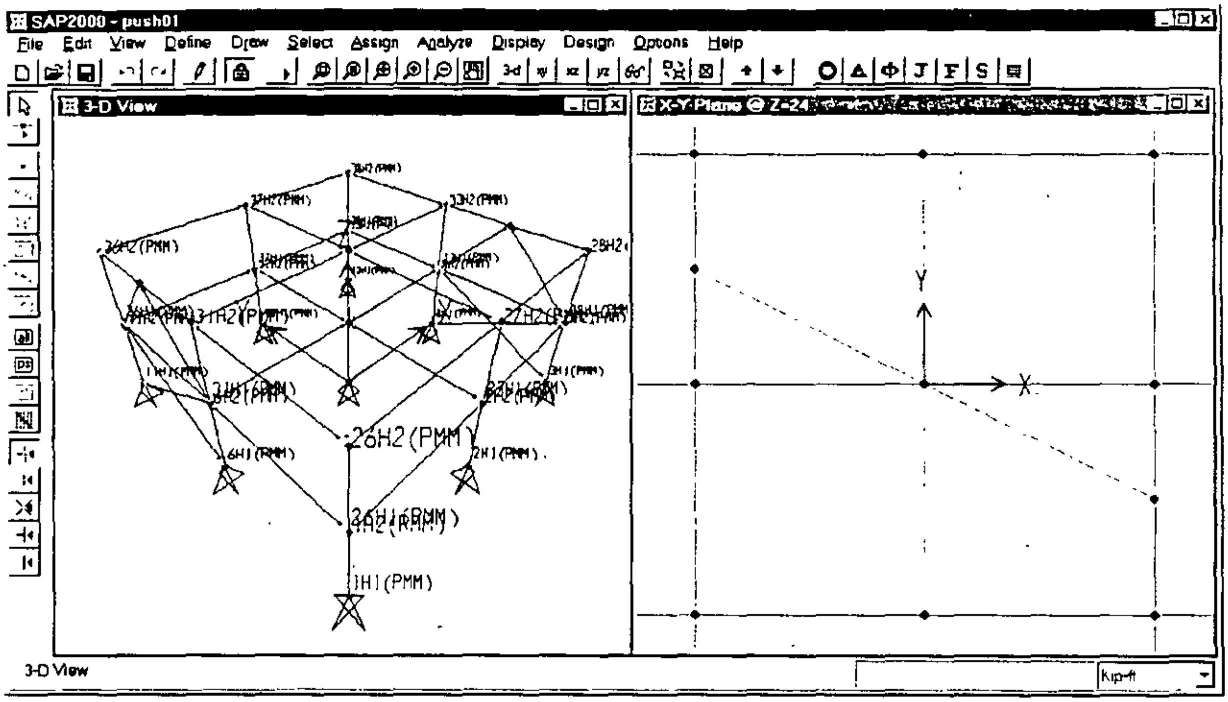


Figure D-1: Model After Assigning Default-PMM Hinges To Columns

Note: Generated hinge properties have an automatic naming convention of LabelH#, where Label is the frame element label, H stands for hinge, and # represents the hinge number. The program starts with hinge number 1 and increments the hinge number by one for each consecutive hinge applied to the frame element. For example if a frame element label is F23, the generated hinge property name for the second hinge applied to the frame element is F23H2. These are the numbers shown on the screen. The item in parenthesis next to the hinge number is the defined hinge that the generated properties are based on. On the screen the Default-P, Default-V2, Default-V3, and Default-PMM are shortened to P, V2, V3 and PMM.

8. From the **Select** menu choose **Select**, and then **Groups...** from the submenu. This will display the Select Groups dialog box.

9. Highlight the group named FRMGIRD by clicking on it and click the **OK** button.

10. From the **Assign** menu choose **Frame**, and then **Hinges (Pushover)...** from the submenu. This will display the Frame Hinges (Pushover) dialog box.

11. In this dialog box:

    - Select Default-M3 in the Hinge Property drop-down box.
    - Type **0** in the Relative Distance edit box.
    - Click the **Add** button.
    - Type **1** in the Relative Distance edit box.
    - Click the **Add** button.
    - Click the **OK** button to assign the hinge properties.

12. From the **Select** menu choose **Select**, and then **Groups...**from the submenu. This will display the Select Groups dialog box.

13. Highlight the group named BRACE1 by clicking on it.

14. Hold down the CTRL key and highlight the group named BRACE2 by clicking on it to add it to the selection.

15. Click the **OK** button to select all of the elements in the BRACE1 and BRACE2 groups.

16. From the **Assign** menu choose **Frame**, and then **Hinges (Pushover)...** from the submenu. This will display the Frame Hinges (Pushover) dialog box.

17. In this dialog box:

    - Select Default-P in the Hinge Property drop-down box.
    - Type **.5** in the Relative Distance edit box.
    - Click the **Add** button to add a P hinge at the center (relative distance equals 0.5) of the beam.
    - Click the **OK** button to assign the hinge properties.

18. From the **Assign** menu select **Clear Display of Assigns** to clear the display of hinge assignments.

19. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assigning of pushover hinge properties.

Step 3: Viewing Generated Hinge Properties (Pushover)

This section will demonstrate the process to view generated hinge properties. This step can be skipped if you do not want to view generated hinge properties.

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

3. In this dialog box check the Rel / Hinge box in the Frames area and click the **OK** button. This will turn the display of hinges on.

4. Note that the hinge at the top of column element 1 is labeled 1H2. (Column element 1 is a lower level column located at X=-24 ft, Y=-24 ft.) We will view the properties for this PMM hinge.

   Note: In the hinge label 1H2, the 1 indicates the hinge is applied to frame element 1, the H indicates it is a pushover hinge, and the 2 indicates it is the second hinge applied to that frame element. This is an automatic naming convention used by the program for generated hinges.

5. From the **Define** menu choose **Hinge Properties....** This will display the Define Frame Hinge Properties dialog box.

6. Check the Show Generated Props check box. The generated hinge properties now appear in the Defined Hinge props list box.

7. In the All Hinge Props area, click on the hinge labeled 1H2 to highlight it and then click the **Modify/Show Property** button. This will display the Frame Hinge Property Data dialog box.

8. Note that since this is a default PMM hinge, the P-M2-M3 box is checked, but it is gray and inactive. The program also automatically checks the Axial P, Moment M2 and Moment M3 boxes. All of these boxes are gray and inactive since the user can only view generated hinge properties not change them.

9. Click the **Modify/Show For PMM** button to display the Frame Hinge Property Data For 1H2 dialog box (see Figure D-2).

10. In this dialog box:

    - You can not edit any of the values because this is a generated hinge.

    - The spreadsheet is filled with M/M_Y and θ/θ_Y values.

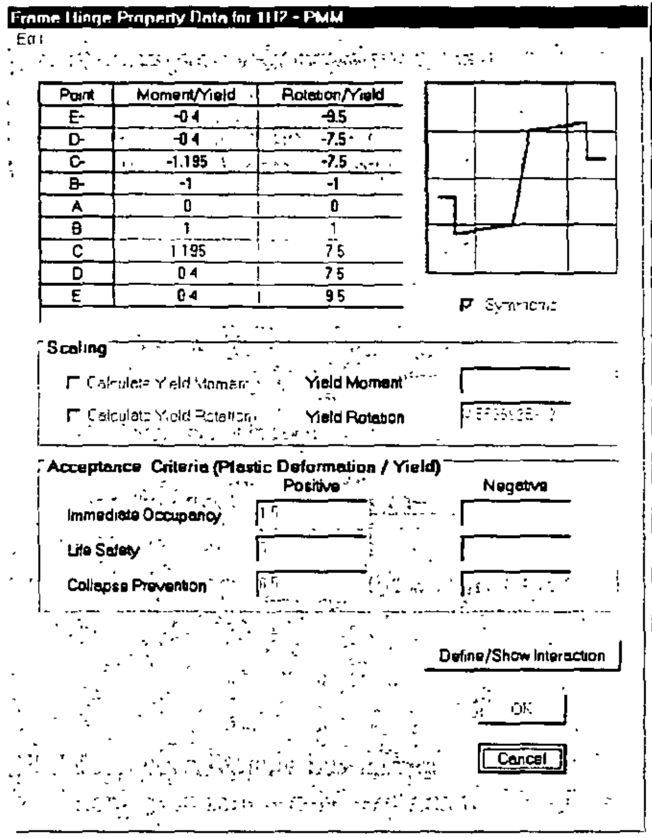


Figure D-2: Frame Hinge Property Data For Generated Pushover Hinge 1H2

- The default hinge properties are symmetric.

- In the scaling area the Yield Rotation is filled in and the Yield Moment is gray and inactive. TheYield Rotation was calculated by the program based on the associated frame section property. The Yield Moment is inactive because this is a PMM hinge and there is no single yield moment, there is a yield (interaction) surface instead. If this were just a moment hinge then there would be a value in the Yield Moment box.

- The acceptance criteria are θ/θ_Y values.

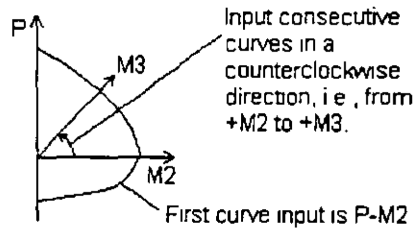
- Click the **Define/Show Interaction** button to display the Frame Hinge Interaction Surface dialog box. In this dialog box which is mostly inactive because you are viewing a generated hinge property, note:

  ✓ The User Definition option is used.

  ✓ The Doubly Symmetric About M2 and M3 check box is checked.

  ✓ Five curves are used to define the interaction surface.

Note: The Number of Curves edit box indicates the number of equally spaced P-M curves that will be used to define the interaction surface. If the Doubly Symmetric About M2 and M3 box is checked, then the curves are equally spaced between 0 and 90 degrees. If the Doubly Symmetric About M2 and M3 box is not checked, then the curves are equally spaced between 0 and 360 degrees.



Construction of User-Defined Interaction Surface

✓ Click the **Define/Show Surface** button to display the Interaction Surface Definition dialog box.

✓ In this dialog box:

➢ The spreadsheet shows normalized values of axial load, P, and moment, M. There is one P column and five M columns corresponding to the number entered in the Number of Curves edit box in the previous dialog box. Use the scroll bars to see the other M columns.

Note: The axial load is normalized by the maximum axial load value. The moments are normalized by the maximum moment value for all of the moment curves. Thus the maximum value in the spreadsheet for both axial load and moment is 1.

➢ The diagram shows the P versus M curves. Click in an M column in the spreadsheet to see that particular P versus M curve. The M curve at Angle 0 corresponds to +M2 bending. The M curve at Angle 90 corresponds to +M3 bending.

➢ In the scaling area, the scale factors for both axial load, P, and moment, M, are shown. Note that the M scale factor applies to all of the M curves.

Note: Since both P and M are normalized to 1, the P and M scale factors are the maximum values of P and M, respectively.

11. Click the **Cancel** button five (5) times to exit all of the Hinge Property dialog boxes.

12. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box. In this dialog box uncheck the Rel / Hinge box in the Frames area and click the **OK** button to turn off the display of hinges.

This completes the viewing of generated hinge properties.

Step 4: Defining Static Pushover Cases

For this tutorial example we will define six static pushover cases. The first will apply the gravity load to the structure, and the other five will apply different distributions of lateral load to the structure.

1. From the **Define** menu choose **Static Pushover Cases....** This will display the Define Static Pushover Cases dialog box.

2. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

3. In this dialog box:

   - Type GRAV in the Pushover Case Name edit box.

   - In the Options Area select the Push To Load Level Defined By Pattern option button.

     Note: Typically the Push To Load Level Defined By Pattern option is used to apply gravity load, and the Push To Displacement Of option is used to apply lateral pushes.

   - In the Load Pattern do the following:

     ✓ Select DL from the Load drop-down box.

     ✓ Type **1** in the Scale Factor edit box.

     ✓ Click the **Add** button.

     ✓ Select LL from the Load drop-down box.

     ✓ Type **0.25** in the Scale Factor edit box.

     ✓ Click the **Add** button.

   - Accept all of the other default values. Note that the default Control Joint, joint 19, occurs at the roof. The dialog box appears as shown in Figure D-3.

   - Click the **OK** button to complete this pushover case definition.

   Note: There are five control parameters in the right-hand side of the Options area. These parameters are used to control the pushover analysis. For most analyses the default values are sufficient.

   The Minimum Saved Steps and Maximum Saved Steps provide control over the number of points actually saved in the pushover analysis. The default values are

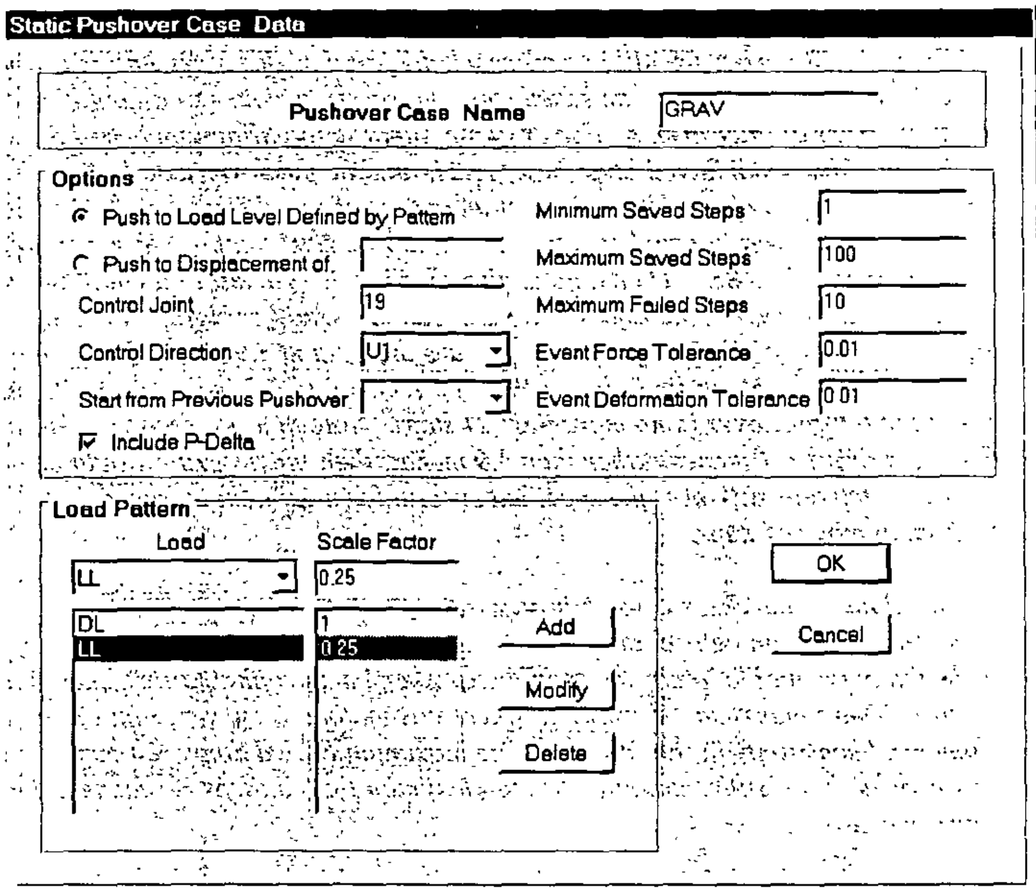


Figure D-3: Static Pushover Case Data Dialog Box

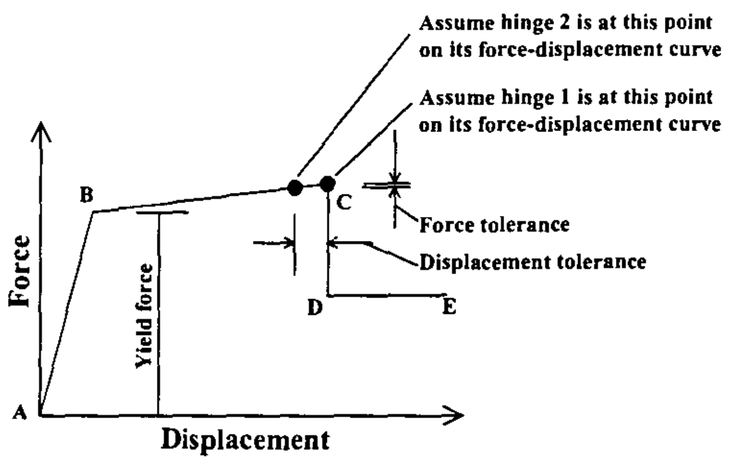
adequate in most cases. If the minimum number of steps saved is too small, you may not have enough points to adequately represent the pushover curve. If the minimum and maximum number of saved steps is too large, then the analysis may consume a considerable amount of disk space, and it may take an excessive amount of time to display results.

The program automatically determines the spacing of pushover steps to be saved as follows. The maximum step length is equal to total force goal or total displacement goal divided by the specified Minimum Saved Steps. The program starts by saving steps at this increment. If a significant event occurs at a step length less than this increment, then the program will save the step too and pick up with the maximum increment from there. For example, suppose the Minimum Saved Steps and Maximum Saved Steps are set at 20 and 30 respectively, and the pushover is to be to a displacement of 10 inches. The maximum increment of saved steps will be 10 / 20 = 0.5 inches. Thus, data is saved at 0.5, 1, 1.5, 2, 2.5 inches. Suppose that a significant event occurs at 2.7 inches. Then data is also saved at 2.7 inches, and continues on from there being saved at 3.2, 3.7, 4.2, 4.7, 5.2, 5.7, 6.2, 6.7, 7.2, 7.7, 8.2, 8.7, 9.2, 9.7 and 10.0 inches.

The Maximum Saved Steps controls the number of significant events for which data will be saved. The program will always reach the force or displacement goal within the specified number of maximum saved steps, however, in doing so it could have to skip saving steps at later events. For example, suppose the Minimum Saved Steps is set to 20, the Maximum Saved Steps is set to 21, and the pushover is to be to a displacement of 10 inches. The maximum increment of saved steps will be 10 / 20 = 0.5 inches. Thus, data is saved at 0.5, 1, 1.5, 2, 2.5 inches. Suppose that a significant event occurs at 2.7 inches. Then data is also saved at 2.7 inches, and continues on from there being saved at 3.2 and 3.7 inches. Suppose another significant event occurs at 3.9 inches. The program will not save the data at 3.9 inches because if it did it would not be able to limit the maximum increment to 0.5 inches and still get through the full pushover in no more than 21 steps. Note that if a second significant event occurred at 4.1 inches rather than 3.9 inches, then the program would be able to save the step and still meet the specified criteria for maximum increment and maximum number of steps.

The Maximum Failed Steps is used, if necessary, to declare failure (i.e., non-convergence) in a run before it reaches the specified force or displacement goal. The program may be unable to converge on a step when catastrophic failure occurs in the structure. There may also be instances where it is unable to converge on a step due to numerical sensitivity in the solution. The Maximum Failed Steps is a cumulative counter through the entire analysis. If the Maximum Failed Steps is reached, the analysis stops.

The Event Force Tolerance and the Event Deformation Tolerance are ratios that are used to determine when an event actually occurs for a hinge. Consider the figure that shows the location of two hinges on their force-displacement plots. Hinge 1 has reached an event location. For hinge 2, if both the Event Force Tolerance and the Event Displacement Tolerance are met, then the hinge is within event tolerance and it too will be treated as part of the event. In



the figure, if the Force Tolerance divided by the Yield Force is less than the Event Force Tolerance specified in the Static Pushover Case Data, and the Displacement Tolerance divided by the horizontal distance from B to C is less than the Displacement Event Tolerance specified in the Static Pushover Case Data, then hinge 2 will be treated as part of the event. When determining the Force Tolerance Ratio, *the denominator is always the yield force. When determining the Displacement Tolerance Ratio, the denominator is the horizontal length of the portion of the force-displacement curve that the hinge is currently on. In the figure, hinge 2 is on the B-C portion of the curve, thus we used the B-C horizontal length in the denominator of the Displacement Tolerance Ratio.*

4. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

5. In this dialog box:

   - Accept the default Pushover Case Name, PUSH2.

   - In the Options area, select GRAV from the Start From Previous Pushover drop-down box.

   - In the Options area, check the Include P-Delta box if it is not already checked.

   - In the Options area, accept the Push to Displacement Of value of 0.96 feet.

     Note: The Push To Displacement Of value defaults to 0.04 times the Z coordinate of the highest joint in the model. Note that this may lead to very large displacements if the base of the model is not at Z=0. You can change this value, if necessary, by typing a new value in the edit box.

   - Accept all of the other default values in the Options Area.

   - In the Load Pattern do the following:

     ✓ Select PUSHPAT from the Load drop-down box.

     ✓ Type **1** in the Scale Factor edit box.

     ✓ Click the **Add** button.

   - Click the **OK** button to complete this pushover case definition.

6. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

7. In this dialog box:

   - Accept the default Pushover Case Name, PUSH3.

   - In the Options area, select GRAV from the Start From Previous Pushover drop-down box.

   - In the Options area, check the Include P-Delta box if it is not already checked.

- Accept all of the other default values in the Options Area.
- In the Load Pattern do the following:
  - ✓ Select acc dir X from the Load drop-down box.
  - ✓ Type **1** in the Scale Factor edit box.
  - ✓ Click the **Add** button.
- Click the **OK** button to complete this pushover case definition.

8. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.
9. In this dialog box:
   - Accept the default Pushover Case Name, PUSH4.
   - In the Options Area do the following:
     - ✓ Type **.5** in the Push To Displacement Of edit box.
     - ✓ Select U2 from the Control Direction drop-down box.
     - ✓ Select GRAV from the Start From Previous Pushover drop-down box.
     - ✓ Accept all of the other default values in the Options Area.
   - In the Load Pattern do the following:
     - ✓ Select acc dir Y from the Load drop-down box.
     - ✓ Type **1** in the Scale Factor edit box.
     - ✓ Click the **Add** button.
   - Click the **OK** button to complete this pushover case definition.
10. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.
11. In this dialog box:
    - Accept the default Pushover Case Name, PUSH5.

- In the Options area, select GRAV from the Start From Previous Pushover drop-down box.

- In the Options area, check the Include P-Delta box.

- Accept all of the other default values in the Options Area.

- In the Load Pattern do the following:

  ✓ Select MODE from the Load drop-down box.

  ✓ Type **1** in the Scale Factor edit box.

  ✓ Click the **Add** button. The Select Mode Number dialog box appears.

  ✓ Type in **1** for the mode number (corresponding to the first mode in the X-direction), and click the **OK** button.

- Click the **OK** button to complete this pushover case definition.

12. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

13. In this dialog box:

- Accept the default Pushover Case Name, PUSH6.

- In the Options area, select GRAV from the Start From Previous Pushover drop-down box.

- In the Options area, check the Include P-Delta box.

- Accept all of the other default values in the Options Area.

- In the Load Pattern do the following:

  ✓ Select acc dir X from the Load drop-down box.

  ✓ Type **1** in the Scale Factor edit box.

  ✓ Click the **Add** button.

  ✓ Select acc dir Y from the Load drop-down box.

  ✓ Type **1** in the Scale Factor edit box.

  ✓ Click the **Add** button.

- Click the **OK** button to complete this pushover case definition.

14. Click the **OK** button to exit the Define Static pushover Cases dialog box.

15. Click the **Save Model** button on the main toolbar.

This completes the definition of static pushover cases.

Step 5: Running the Pushover Analysis

1. On the **Analyze** menu select **Run Static Pushover**.

   Note: To run a pushover analysis, you must first have pushover hinges and pushover load cases defined, at least a static analysis run, and, if steel members with Auto sections, or concrete members whose reinforcing is to be designed by the program are included, you must have run the design portion of the program.

2. A window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display as shown in Figure D-4.

   *Note: Most of the information in the scrolling analysis window is appended to the *.log file that was created when the original analysis was run.*

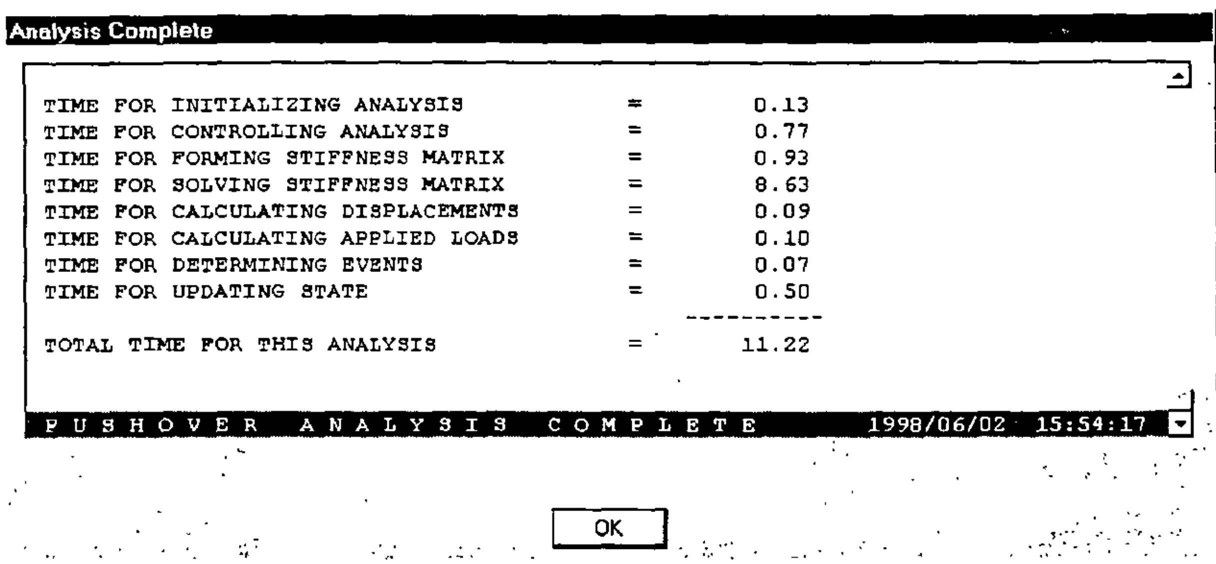


Figure D-4: Screen Message When Pushover Analysis Is Complete

3. Use the scroll bar to review the analysis messages and check for any error or warning messages (there should be none).

   Note: One of the items you will see in the scrolling window is labeled Analysis Complete. As each pushover load case is running you will see the value continually changing. It is a measure of what percent of your force or displacement goal you have reached. When the Analysis Complete reaches 1, you have reached your goal. Note that the analysis may not to reach its goal (i.e., an Analysis complete of 1) because the structure catastropically fails earlier or because of numerical sensitivities.

4. Click the **OK** button in the Analysis window to close it.

This completes running the pushover analysis.

Step 6: Displaying the Pushover Deformed Shape and the Sequence of Pushover Hinge Formation

1. From the **Options** menu select **Windows** and the select **One** from the submenu. The display changes to one window.

   Note: With this option you can select to have from one to four windows on the screen at the same time. Each of the windows can be showing a completely different view.

2. Click the **3-D View** button 3-d on the main toolbar to show the default 3-D view.

3. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box. In this dialog box check the Hide box in the Shells area and click the **OK** button. This will turn off the display of shell elements.

4. Click the **Display Static Deformed Shape** button ▲, or from the **Display** menu select **Show Deformed Shape...** to display the Deformed Shape dialog box.

5. In this dialog box:

   - Select Push2 Static Push from the Load drop-down box.
   - Select Auto in the Scaling area.
   - Check both the Wire Shadow and the Cubic Curve options in the Options area.
   - Click the **OK** button.

6. The deformed shape will appear. Note that the title of the window includes the information "Deformed Shape (PUSH2 – Step 0)". Thus we are currently viewing the deformed shape at the start of the pushover. Since the PUSH2 pushover was started from the GRAV pushover, we are also viewing the deformed shape at the end of the GRAV pushover.

   Note: Recall that the PUSH2 pushover load case is the PUSHPAT static load pattern.

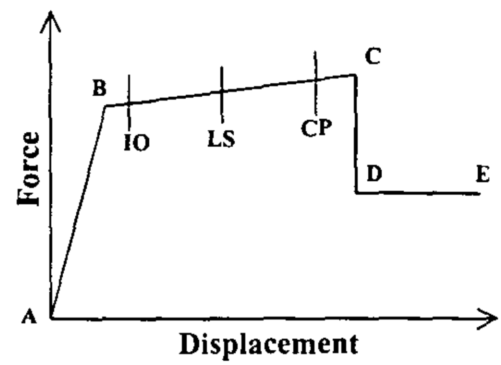
7. Right click on any second level or roof level joint to see the displacement for that joint. The displacements will be very small, as is expected when only the gravity load is applied.

8. Click the **Right Arrow** button ⇨ located on the right-hand side of the status bar at the bottom of the screen to view the deformed shape of the next step (Step 1) in the pushover.

   Note: When viewing the pushover deformed shape and sequence of hinge formation, the right and left arrow buttons, located in the status bar at the bottom of the screen, provide an easy way to view the deformed shape for the next (right arrow), or previous (left arrow) pushover step.

9. Note that the title of the window includes the information "Deformed Shape (PUSH2 – Step 1)", and the building moves slightly in the X-direction. You can again right-click on any joint to see its displacement. You can click the **Left Arrow** button to return to Step 0 of the pushover.

10. Continue clicking the **Right Arrow** button until the first colored hinges appear. The first colored hinges should appear in Step 4, as illustrated in Figure D-5. The color of the hinges indicates the state of the hinge, i.e, where it is along its force displacement curve. The legend for the hinge colors is included at the bottom of the screen. The points B, IO, LS, CP, C, D and E are illustrated in the figure. When hinges first appear, they are at point B on the force-displacement curve.



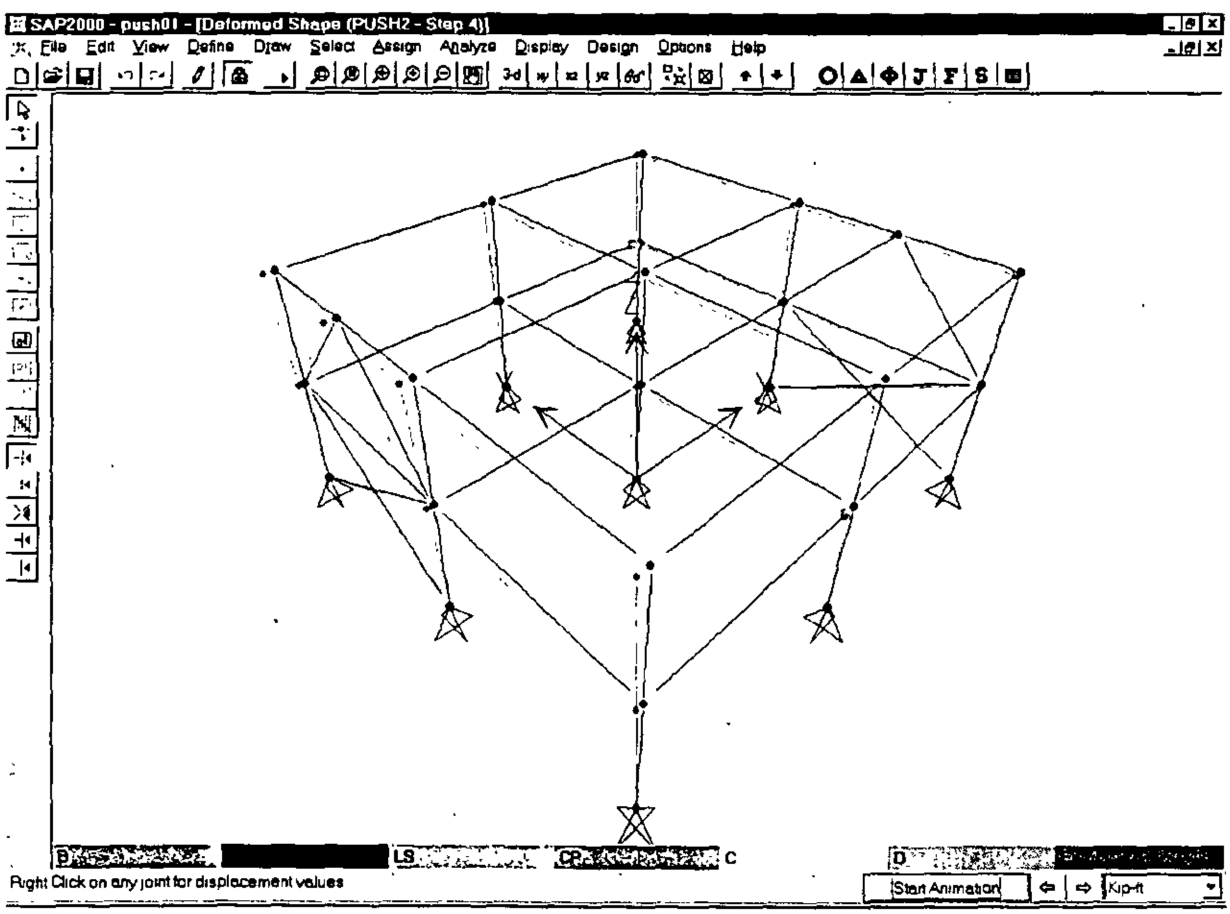


Figure D-5: First Pushover Hinge Yielding

11. Continue clicking the **Right Arrow** button ⇨ to step through the sequence of hinge formation in the pushover to the last step. Note how the colors of the hinges change as the pushover proceeds.

Note: To change the color coding for the hinges, From the Options menu, select Colors, and then select the Output tab. The color-coding for the pushover hinge state is controlled by the colors in the area labeled Contours. There are ten contour colors; these contour colors also are used in displaying stress contours for shell elements. Assume the color in the top box, next to the label "Min", is designated "Box 1", and the color in the bottom box, next to the label "Max", is designated "Box 10". Then the color coding for pushover hinge states is as defined in the table.

Point	Color Box
B	Box 2
IO	Box 3
LS	Box 4
CP	Box 5
C	Box 6
D	Box 7
E	Box 8

12. At the last pushover step right click one of the roof level joints to see its displacement. It will be about 0.96 feet which is consistent with the displacement goal for the PUSH2 pushover load case.

13. Click the **Start Animation** button Start Animation, located in the status bar at the bottom of the SAP2000 window, to animate the deformed shape at the last step. When done viewing the animation, click the **Stop Animation** button. Stop Animation

Note: In this instance, the animation is only for the particular load step, not for the entire pushover. The purpose of the animation is to make the behavior at that particular load step more apparent. You can create a video of the entire pushover using the Create Video... option on the File menu, however, this option will not be available in the SAP2000 6.20 Beta version.

14. Click the **Display Static Deformed Shape** button ▲, or from the **Display** menu select **Show Deformed Shape...** to display the Deformed Shape dialog box.

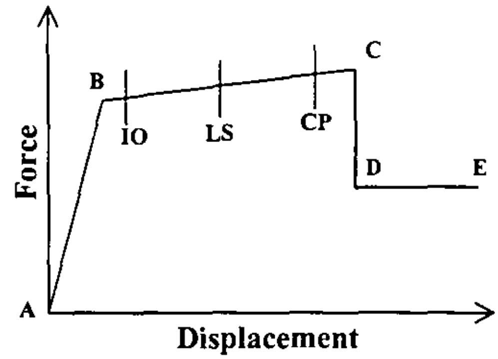
15. In this dialog box:

- Select Push4 Static Push from the Load drop-down box. Recall this is the uniform acceleration in the Y-direction.

- Click the **OK** button.

16. Click the **Right Arrow** button ⇨ to step through the entire sequence of hinge formation in the pushover. You will note that through the sequence of steps the deformation reverses directions several times. The reason for this is dicussed in the note below.

Note: When a hinge reaches point C on its force-displacement curve (see figure below), that hinge must begin to drop load. Within the program, the way load is dropped from a hinge that has reached point C is that the pushover force (base shear) is reduced until the force in that hinge is consistent with the force at point D. As the *force is dropped, all elements unload, and the displacement is reduced. Once the yielded hinge reaches the Point D force level, the pushover force (base shear) is again increased and the displacement begins to increase again. This behavior is apparent when displaying the deformed shapes and force diagrams (moment, shear, etc.) for each step of the pushover, when viewing a video created for the pushover, and when displaying the force-displacement plot of the pushover.*



17. Click the **Show Undeformed Shape** button to clear the display of deformed shape for the pushover.

18. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box.

19. In this dialog box:

- Uncheck the Shrink Elements box in the options area (if it is checked).

- Click the **OK** button.

This completes the review of the pushover deformed shape and the sequence of pushover hinge formation. You may want to step through the deformed shapes for the other pushover load cases before proceeding on to the next step.

Step 7: Displaying Frame Element Forces at Each Step of the Pushover

1. Click the **Member Force Diagram for Frames** button **F**, or from the **Display** menu select **Show Element Forces/Stresses**, and then **Frames...** to display the Member Force Diagram for Frames dialog box.

2. In this dialog box:

   - Select Push2 Static Push from the Load drop-down box. Recall this is the PUSHPAT static load pattern.

   - In the Component area select the Moment 3-3 option.

   - In the Scaling area select the Auto option.

   - Check the Fill Diagram box. Note if the Show Values on Diagram box is checked, you will have to uncheck it before you can check the Fill Diagram box.

     Note: The frame element forces for each step of the pushover can be displayed either with the filled diagram or with the values shown on the diagram.

   - Click the **OK** button. The display appears as shown in Figure D-6.

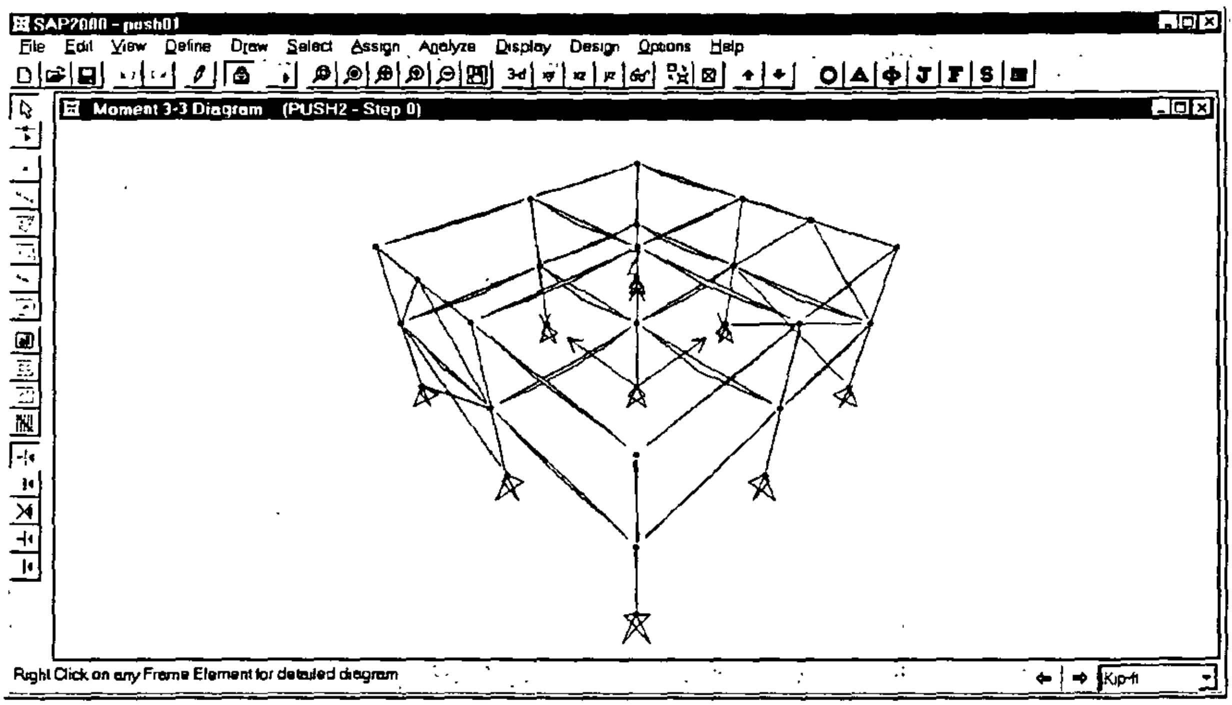


Figure D-6: Pushover Forces (M3-3) at Step 0

3. Note that the title of the window includes the information "Moment 3-3 Diagram (PUSH2 – Step 0)". Thus we are currently viewing the M33 moments at the start of the pushover. Since the PUSH2 pushover was started from the GRAV pushover, we are also viewing the M33 moments at the end of the GRAV pushover.

4. Note you can right click on any frame element for a detailed diagram. When done viewing a detailed diagram, right click on another element to view its detailed diagram, or left click anywhere to finish viewing detailed diagrams.

5. Click the **Right Arrow** button ⇨ located on the right-hand side of the status bar at the bottom of the screen to view the M-33 diagram at the next step (Step 1) of the pushover.

   Note: When viewing the pushover forces, the right and left arrow buttons, located in the status bar at the bottom of the screen, provide an easy way to view the forces for the next (right arrow), or previous (left arrow) pushover step.

6. Continue clicking the **Right Arrow** button ⇨ to step through the moment diagrams for each step of the pushover. You can right click an element at any step for a detailed diagram.

7. Click the **Show Undeformed Shape** button to clear the display of element forces for the pushover.

This completes the review of the frame element forces at each step of the pushover. You may want to step through the element forces for other force components and for other pushover load cases before proceeding on to the next step.

Step 8: Displaying Pushover and Capacity Spectrum Curves

1. From the **Display** menu select **Show Static Pushover Curve...** to display the Pushover Curve dialog box.

   Note: If this dialog box does not fully fit on your screen then you may want to increase the resolution of your screen. The dialog box should fully fit on the screen if your resolution is 800 x 600 with small fonts, or 1024 x 768 with large fonts.

2. If no plot is visible in the plot area, then click the **Display** button at the bottom of the form.

   *Note: If at any time the plot area display is not visible in the Pushover curve dialog box, click the **Display** button.*

3. For practice, press the F1 key on the keyboard to see context sensitive on-line help pertaining to this dialog box.

4. When finished with the online help click the "X" in the upper right-hand corner of the Help window to close it, or choose **Exit** from the **File** menu on the Help window.

5. Notice that in the Plot Type area, the Base Shear vs Control Displacement option is selected.

   Note: The base shear that is plotted in the Base Shear vs Control Displacement plot is the resultant base shear. The displacement plotted is the displacement in the control direction (not resultant) at the control joint.

6. The dialog box currently appears as shown in Figure D-7. Note the following:

   - The Demand Spectrum area and the Damping Parameters area are gray and inactive. These areas will become active when the Capacity Spectrum option is chosen in the Plot Type area.

   - There are four boxes just below the plot area. These boxes are the Cursor Location, the Performance Point (V, D), the Performance Point (Sa, Sd), the Performance Point (Teff, βeff). When the Base Shear vs Control Displacement option is chosen in the Plot Type area, only the Cursor Location box is active. Place the cursor (mouse pointer) anywhere over the plot and the coordinates of the pointer will appear in the Cursor Location box. The Performance Point boxes are only filled in when the Capacity Spectrum option is chosen in the Plot Type area.

   - If we wanted to include some notes with any printed graphic output, we could type those notes in the Additional Notes For printed Output edit box.

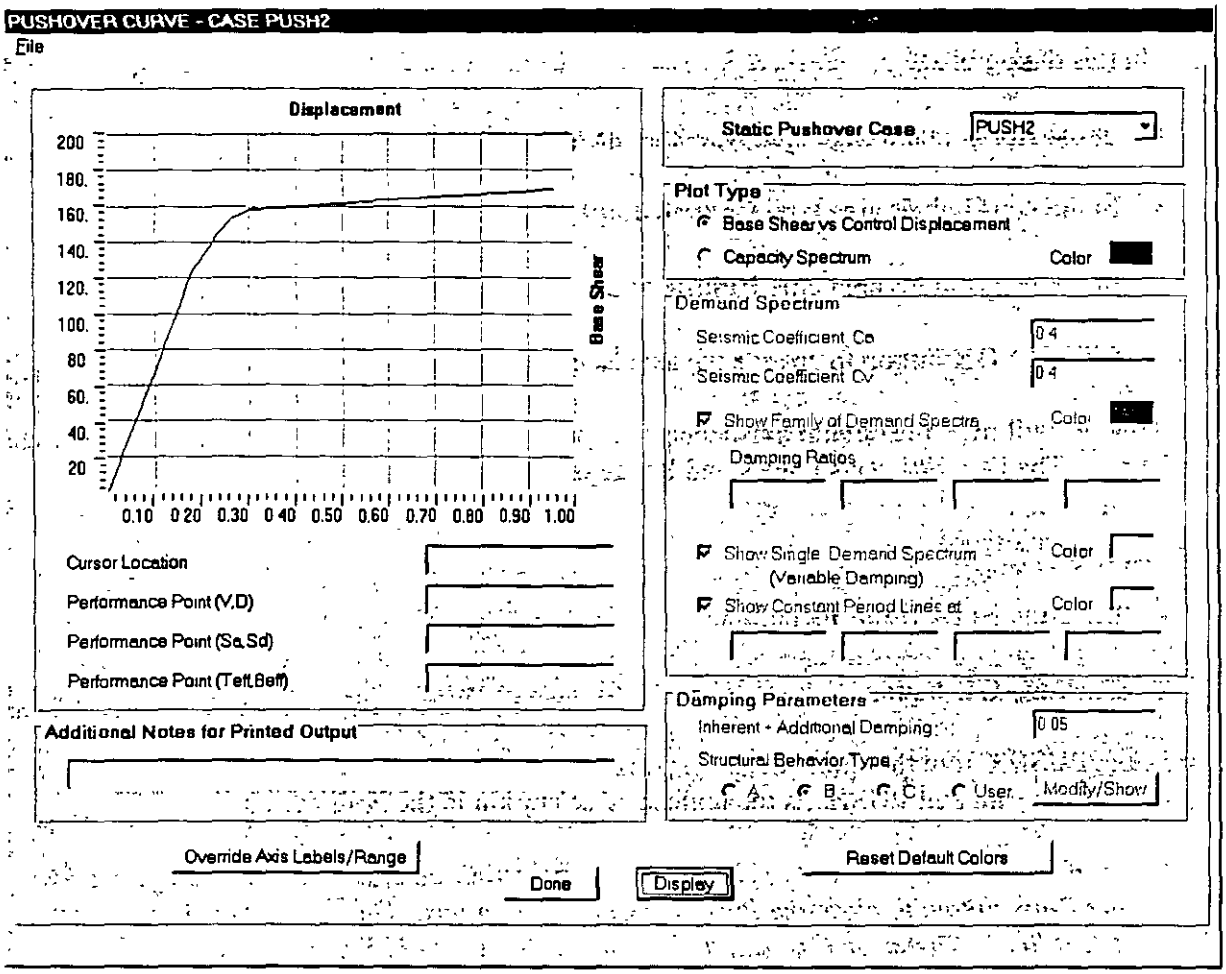


Figure D-7: Pushover Curve For Pushover Load Case PUSH2

7. Suppose we want to change the color of the base shear versus displacement curve from the default green color to blue. To do this click on the green color box in the Plot Type area to open the Color dialog box.

8. In this dialog box:

    - Click on one of the blue colored boxes.
    - Click the **OK** button to change the color.
    - If necessary, click the **Display** button to redisplay the plot.

9. To change the curve color back to the default green, click the **Reset Default Colors** button.

10. Now we will override the axis labels and range. The axis labels and range will appear on the screen and on any graphic output. Click the **Override Axis Labels/Range** button. The Override Axis Labels and Range dialog box appears.

11. In this dialog box:

   - In the Horizontal Range area type **2** in the Max edit box.
   - In the Axis Labels area type **Control Joint Displacement** in the Horizontal edit box.
   - In the Axis Labels area type **Resultant Base Shear** in the Vertical edit box.
   - Click the **OK** button to change the axis labels and range.

12. Now we will reset the default axis labels and range. Click the **Override Axis Labels/Range** button. The Override Axis Labels and Range dialog box appears.

13. In this dialog box:

   - Click the **Reset Defaults** button.
   - Click the **OK** button.

14. Select PUSH3 from the Static Pushover Case drop-down box. The plot changes to that for PUSH3. Recall that PUSH3 is the uniform acceleration in the X-direction.

   *Note: If at any time you want to check and see the definition of a pushover load case click the **Done** button to close the Pushover Curve dialog box. Then, from the **Define** menu select **Static Pushover Cases...**, and when the Define Static Pushover Cases dialog box appears, select the pushover case you are interested in, and click the **Modify/Show Pushover** button.*

15. Review the base shear versus control displacement curves for the other pushover load cases.

16 Select PUSH2 from the Static Pushover Case drop-down box.

17. Select the Capacity Spectrum option in the Plot Type area. As shown in Figure D-8, the plot changes and the Demand Spectrum area and the Damping Parameters area are now active.

   Note: When the capacity spectrum option is chosen, the pushover curve is displayed in ADRS (Acceleration-Displacement Response Spectrum) format. Refer to ATC-40 for a discussion of this format which is essentially a plot of spectral acceleration versus spectral displacement. In SAP2000, the force-displacement pushover curve is converted to the ADRS format by converting the resultant base shear to a spectral acceleration , Sa, and the control displacement in the control direction to a spectral displacement, Sd, generally based on equations 8-1 thru 8-4 in ATC-40.

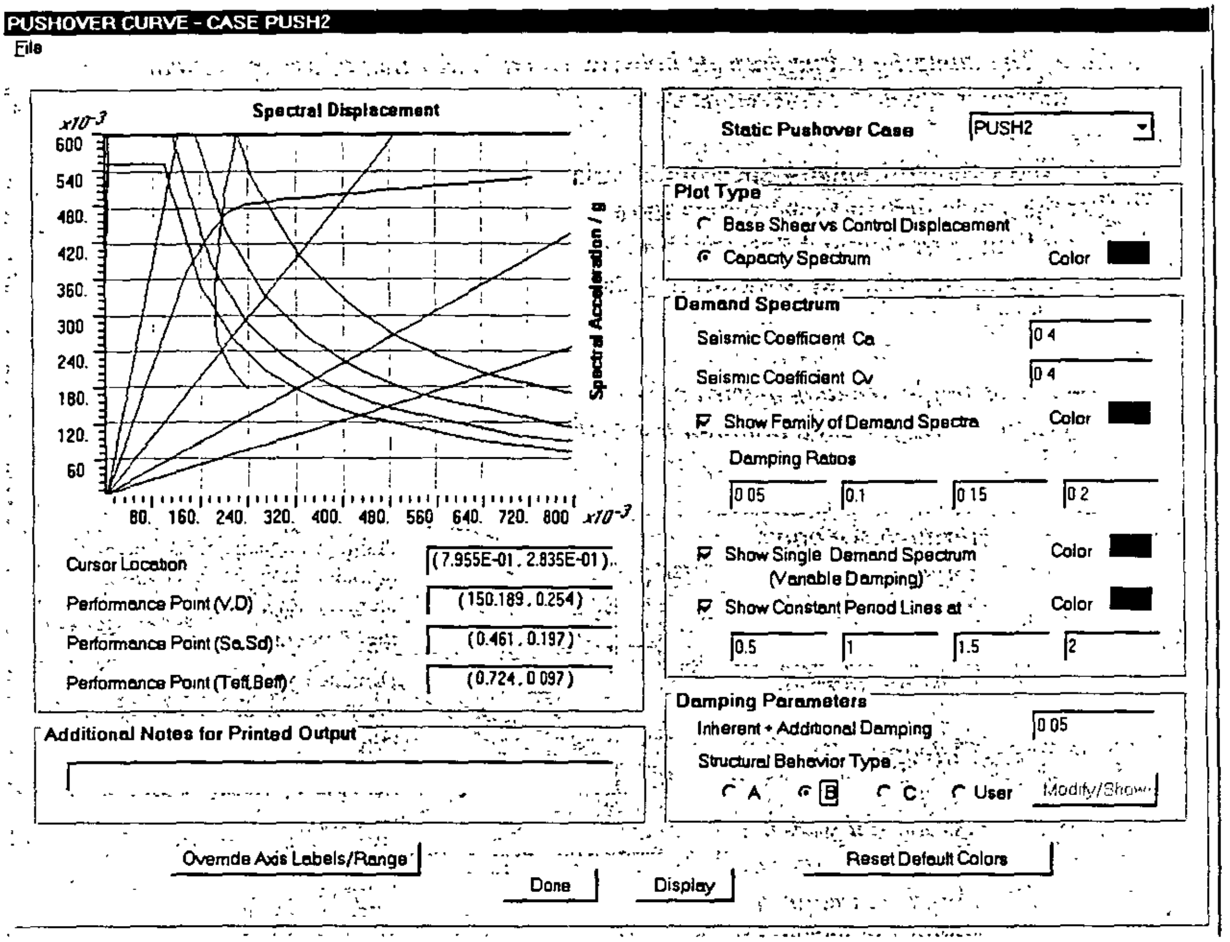


Figure D-8: Capacity spectrum For Pushover Load Case PUSH2

18. In the Demand Spectrum area note the following:

- The Seismic Coefficient Ca and the Seismic Coefficient Cv both default to 0.4. Each of these values can be changed by typing a new value the appropriate edit box. These values control the shape of the 5% damped spectrum. Refer to ATC-40, Chapter 4, for a discussion of, and appropriate values for, Ca and Cv.

- The Show Family of Demand Spectra check box is checked indicating that the family of demand spectra, with up to four different damping ratios, is shown on the plot. The color box adjacent to the Show Family of Demand Spectra check box is red (by default) indicating that the family of demand spectra is shown in red. You can change the color by clicking on the color box.

- There are four edit boxes labeled Damping Ratios. The numbers entered in these boxes are 0.05, 0.1, 0.15 and 0.2. These are the default damping ratios for the family of demand spectra. You can change any of these values by typing a new value in the appropriate edit box. The damping values do not have to be input in order. If you leave a Damping Ratio box blank, or enter a zero (0), then that curve will not be plotted.

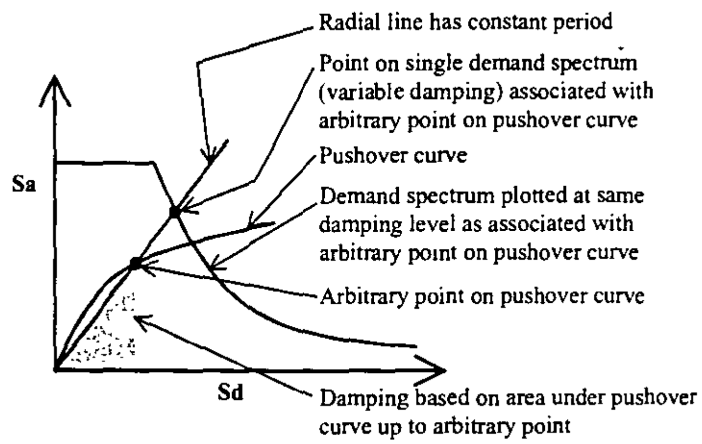
Note: The damping ratios must be between 0 and 1. A value of 5% of critical damping should be entered as 0.05, not 5.

- The Show Single Demand Spectrum (Variable Damping) check box is checked indicating that the single demand spectrum is shown on the plot. The color box adjacent to the Show Family of Demand Spectra check box is yellow (by default) indicating that the single demand spectrum is shown in yellow. You can change the color by clicking on the color box.

Note: The single demand spectrum (variable damping) curve is constructed by doing the following for each point on the ADRS pushover curve:

1. *Draw a radial line through the point on the ADRS pushover curve. This is a line of constant period.*

2. *Calculate the damping associated with the point on the curve based on the area under the curve up to that point.*



3. *Construct the demand spectrum, plotting it for the same damping level as associated with the point on the pushover curve.*

4. *The intersection point of the radial line and the associated demand spectrum represents a point on the Single Demand Spectrum (Variable Damping) curve.*

This method is similar to the method called Procedure B in Chapter 8 of ATC-40 except it does not make the simplifying assumption that the post yield stiffness remains constant. It is essentially the method referred to as "exact" in the discussion of the method called Procedure C in Chapter 8 of ATC-40.

- The Show Constant period Lines At check box is checked indicating that up to four different constant period lines are shown on the plot. The color box adjacent to the Show Constant period Lines At check box is gray (by default) indicating that the constant period lines are shown in gray. You can change the color by clicking on the color box.

Note: In the ADRS format, lines of constant period show up as straight lines radiating from the origin.

- There are four edit boxes associated with the Show Constant period Lines At check box. The numbers entered in these boxes are 0.5, 1, 1.5 and 2. These are the default periods for the lines of constant period. You can change any of these values by typing a new value in the appropriate edit box. The periods do not have to be input in order. If you leave a Period box blank, or enter a zero (0), then that line will not be plotted.

19. In the Damping Parameters area note the following:

- The Inherent + Additional Damping box defaults to 0.05. This value can be changed by typing a new value in the edit box.

  Note: To help you understand what value to input in the Inherent + Additional Damping box, refer to ATC-40 equation 8-8. The β_0 term in this equation is automatically included by the SAP2000 analysis method. The 5% inherent viscous damping term can be specified in the Inherent/Additional Damping edit box as 0.05. If there is additional viscous damping provided in the structure, perhaps by viscous dampers that are not specifically included in the model, then this damping should also be included in the Inherent/Additional Damping edit box. Thus if the damping inherent in the structure is assumed to be 5% of critical damping, and dampers which provide an additional 7% of critical damping are assumed to be added to the structure (although they are not actually in the model), then the value input in the Inherent/Additional Damping edit box should be 0.12, since 0.05 + 0.07 = 0.12.

  If dampers modeled with NLLink elements are included in the model, then for the pushover analysis, the program treats them as linear elements. Their stiffness is based on the linear effective stiffness (KE) and the damping is based on the linear effective damping coefficient (CE). The program uses the linear effective damping coefficient for the damper, together with the effective period to calculate damping which is internally added to the specified Inherent/Additional Damping term.

- There are four Structural Behavior Type options: A, B, C and User. Structural Behavior type B, which should be currently selected, is the default. Structural Behavior Types A, B and C, which define a kappa (κ) factor that reduces the assumed damping, are taken directly from ATC-40. The User option allows input of other values of kappa (κ).

  Note: Refer to ATC-40 Section 8.2.2.1.1 for a discussion of structural behavior types. The structural behavior type is used to specify a kappa (κ) factor (see Figure 8-15 in ATC-40) that modifies (reduces) the calculated area of the hysteresis loops to account for assumed pinching of the loops.

20. The performance point in Sa, Sd coordinates is shown in the Performance Point (Sa, Sd) box below the plot. The units for Sa are always g; the current units for Sd are feet. Note you could also estimate the value of the performance point by holding the mouse pointer over the performance point (intersection of the ADRS pushover curve and the single demand spectrum (variable damping)) and reading the value in the Cursor Location box.

    Note: The location of the mouse pointer is specified in the Cursor Location box as (Horizontal axis location, Vertical Axis location). Thus when looking at the Performance Point using the mouse pointer, you will read (Sd, Sa), which is switched from how it is specified in the Performance Point (Sa, Sd) box.

21. In the Demand Spectrum area, type **.6** into the Seismic Coefficient Cv edit box. Click the mouse somewhere on the pushover curve dialog box outside of the Seismic Coefficient Cv edit box to enter the change. Note how both the plot and the Performance Point information below the plot change. Note that the performance point in Sa, Sd coordinates changes.

22. Type **.4** into the Seismic Coefficient Cv edit box. Click the mouse on the plot to enter the change.

23. Uncheck the Show Family of Demand Spectra box. The family of demand spectra are removed from the plot.

24. Check the Show Family of Demand Spectra box to show the family of demand spectra again.

25. In the Damping Ratios edit boxes, type **.3** in the last box. Click the mouse on the plot to enter the change. Note that the lowest demand spectra got even lower when we increased the damping ratio from 0.2 to 0.3.

26. Type **.2** in the last Damping Ratios edit box and click the mouse on the plot to return the form to its original state.

27. In the Damping Ratios edit boxes, delete the value in the second edit box and click the mouse on the plot to enter the change. Note that only three demand spectra now appear on the plot, one with 5% damping, one with 15% damping and one with 20% damping.

28. Type **.1** in the second Damping Ratios edit box and click the mouse on the plot to return the form to its original state.

29. Uncheck the Show Single Demand Spectrum (Variable Damping) box. The single demand spectrum is removed from the plot.

30. Check the Show Single Demand Spectrum (Variable Damping) box to show the single demand spectrum again.

31. Uncheck the Show Constant Period Lines At box. The constant period lines are removed from the plot.

32. Check the Show Constant Period Lines At box to show the period lines again.

33. In the fourth (and last) edit box below the Show Constant Period Lines At check box, type **3**. Click the mouse on the plot to enter the change. Note that the last period line became flatter.

34. Type **2** in the fourth edit box below the Show Constant Period Lines At check box and click the mouse on the plot to return the form to its original state.

35. In the Damping Parameters area, type **.2** in the Inherent + Additional Damping edit box. Click the mouse on the plot to enter the change. The relative location of the single demand spectrum (variable damping) curve changes and the performance point location changes.

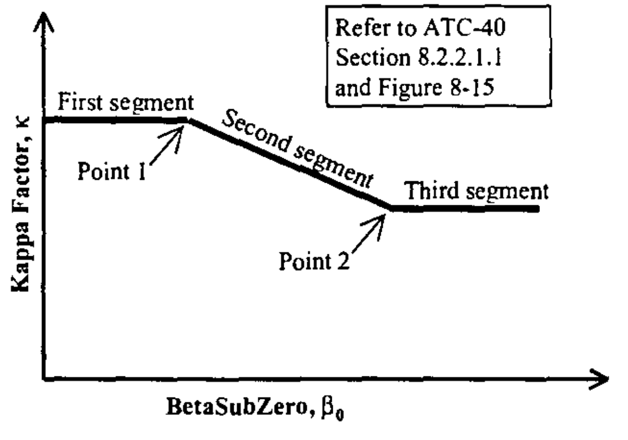
36. Type **0.05** in the Inherent + Additional Damping edit box and click the mouse on the plot to return the form to its original state.

37. In the Damping Parameters area, click the Structural Behavior Type A option and observe the change in the relative location of the single demand spectrum (variable damping) curve and the performance point.

38. Click the Structural Behavior Type C option again observing the change in the relative location of the single demand spectrum (variable damping) curve and the performance point.

39. Click the Structural Behavior Type - User option, and note that the adjacent **Modify/Show** button becomes available. Click the **Modify/Show** button to display the Override Structural Behavior Type dialog box.

40. In this dialog box:



   - In the Point 1 area type **20** in the Beta Sub Zero edit box and type **.6** in the Kappa Factor edit box.
   - In the Point 2 area type **45** in the Beta Sub Zero edit box and type **.6** in the Kappa Factor edit box.
   - Click the **OK** button and observe the change in the relative location of the single demand spectrum (variable damping) curve and the performance point.

41. Click the Structural Behavior Type B option to return the form to its original state.

42. From the File menu at the top of the Pushover Curve dialog box select Display Tables. A table similar to that shown in Figure D-9 appears.

PUSHOVER CAPACITY/DEMAND COMPARISON

File

Step	Teff	βeff	Sd(C)	Sa(C)	Sd(D)	Sa(D)	ALPHA	PF*Ø
0	0.756	0.050	4.002E-04	0.000	0.247	0.529	1.000	1.000
1	0.756	0.046	0.039	0.083	0.252	0.541	0.955	1.246
2	0.756	0.049	0.078	0.167	0.248	0.532	0.955	1.241
3	0.756	0.050	0.116	0.250	0.247	0.530	0.955	1.240
4	0.756	0.050	0.150	0.321	0.247	0.530	0.955	1.239
5	0.792	0.083	0.188	0.368	0.226	0.441	0.953	1.241
6	0.796	0.086	0.193	0.374	0.225	0.435	0.953	1.242
7	0.839	0.117	0.231	0.403	0.216	0.376	0.952	1.244
8	0.875	0.136	0.269	0.432	0.215	0.344	0.951	1.246
9	0.902	0.145	0.303	0.457	0.216	0.326	0.950	1.248
10	0.931	0.161	0.331	0.469	0.215	0.305	0.948	1.252
11	0.932	0.161	0.332	0.469	0.215	0.305	0.947	1.252
12	0.976	0.187	0.368	0.474	0.214	0.276	0.943	1.260
13	1.018	0.208	0.404	0.478	0.214	0.254	0.940	1.266
14	1.058	0.223	0.440	0.483	0.217	0.238	0.936	1.270
15	1.096	0.233	0.477	0.487	0.221	0.226	0.934	1.274
16	1.133	0.240	0.513	0.491	0.225	0.215	0.931	1.278
17	1.167	0.247	0.549	0.495	0.230	0.207	0.929	1.281
18	1.201	0.252	0.586	0.498	0.234	0.199	0.927	1.283

Figure D-9: Table For Capacity Spectrum

43. In this table note the following:

- Step identifies the step number in the pushover curve.
- Teff is the effective period at the associated step.
- βeff is the effective damping at the associated step.
- Sd(C) and Sa(C) define a point on the ADRS capacity curve for the associated step.
- Sd(D) and Sa(D) define a point on the single demand spectrum (variable damping) curve for the associated step.
- Alpha is the factor used in converting the base shear to spectral acceleration at the associated step.
- PF*ϕ is the factor used in converting the displacement to spectral displacement at the associated step.
- To print this table, click on the File menu at the top of the table and select either **Print Tables**, to print to a printer, or **Print Tables To File...**, to print the table to a file.

44. Click the "X" in the upper right-hand corner of the table to close it.

45. Select the Base Shear vs Control Displacement option in the Plot Type area.

46. From the File menu at the top of the Pushover Curve dialog box select Display Tables. A table similar to that shown in Figure D-10 appears.

PUSHOVER CURVE

File

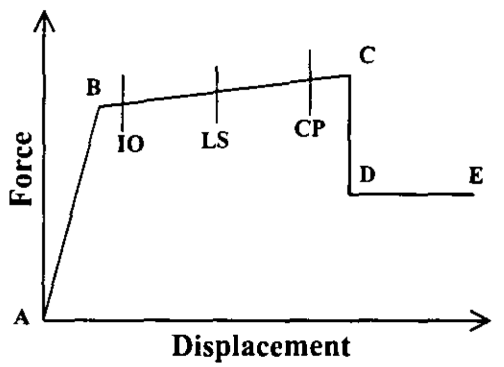
Step	Displacement	Base Shear	A-B	B-IO	IO-LS	LS-CP	CP-C	C-D	D-E	>E	TOTAL
0	4.002E-04	0.0000	48	0	0	0	0	0	0	0	48
1	0.0484	28.3708	48	0	0	0	0	0	0	0	48
2	0.0964	56.7417	48	0	0	0	0	0	0	0	48
3	0.1444	85.1127	48	0	0	0	0	0	0	0	48
4	0.1856	109.4811	46	2	0	0	0	0	0	0	48
5	0.2336	125.1036	46	2	0	0	0	0	0	0	48
6	0.2398	127.1146	44	4	0	0	0	0	0	0	48
7	0.2878	136.8057	44	4	0	0	0	0	0	0	48
8	0.3358	146.4968	44	4	0	0	0	0	0	0	48
9	0.3780	154.9516	42	4	2	0	0	0	0	0	48
10	0.4148	158.5392	41	5	2	0	0	0	0	0	48
11	0.4154	158.5750	40	6	2	0	0	0	0	0	48
12	0.4634	159.4686	40	4	4	0	0	0	0	0	48
13	0.5114	160.3621	40	4	4	0	0	0	0	0	48
14	0.5594	161.2557	40	4	4	0	0	0	0	0	48
15	0.6074	162.1492	40	2	6	0	0	0	0	0	48
16	0.6554	163.0428	40	0	8	0	0	0	0	0	48
17	0.7034	163.9364	40	0	8	0	0	0	0	0	48
18	0.7514	164.8299	40	0	8	0	0	0	0	0	48
19	0.7994	165.7235	40	0	8	0	0	0	0	0	48
20	0.8474	166.6170	40	0	8	0	0	0	0	0	48
21	0.8954	167.5106	40	0	6	2	0	0	0	0	48

Figure D-10: Table For Pushover Curve

47. In this table note the following:

- Step identifies the step number in the pushover curve.

- Displacement and Base Shear define a point on the pushover curve for the associated step.



- A-B, B-IO, IO-LS, LS-CP, CP-C, C-D, D-E, >E all identify the total number of hinges within each of these ranges on their associated force-displacement curves.

- TOTAL is the total number of pushover hinges in the structure.

- To print this table, click on the File menu at the top of the table and select either **Print Tables,** to print to a printer, or **Print Tables To File...**, to print the table to a file.

48. You can also print graphic plots of the pushover curve and/or capacity spectrum curve as follows:

- Set up the plot on the screen (either in Base Shear vs Control Displacement, or in Capacity Spectrum format) as you want to print it.

- If you want to change the axis labels or the axis range, click the Override Axis Labels/Range button and make the desired changes.

- If you want to have additional notes printed on the graphics plot, type those notes in the Additional Notes For Printed Output edit box.

- To print the graphics, select **Print Graphics** from the **File** menu at the top of the Pushover Curve dialog box.

Note: Another technique that can be used is to create a screen plot of the entire dialog box which you can then print from another program such as Paint, Microsoft Word for Windows, or any other program that supports graphics.

*To create the screen plot, press the Alt and Print Screen keys on your keyboard. This will send a copy of the active window to the clipboard. (Note that if you just press the Print Screen key, without the Alt key, you will send a picture of the entire screen to the clipboard.) Open your other program that supports graphics and paste the picture into that program. Usually the command to paste the picture is called **Paste**, and it can be found on a menu called **Edit**. Once you have pasted the picture into a file created in your program that supports graphics, you can print from there.*

49. Click the "X" in the upper right-hand corner of the table to close it.

50. Click the **Done** button to close the Pushover Curve dialog box.

This completes the review of displaying the pushover and capacity spectrum curves.

E. Final Comments

This tutorial, together with the SAP2000 online help provides extensive documentation of all of the pushover analysis features available in SAP2000. It is intended that you can use this tutorial as a continuing reference for SAP2000 pushover analyses.

As previously noted, you can obtain context-sensitive online help from within any pushover dialog box by pressing the F1 key while that dialog box is open. The names of each of the basic pushover analysis topics covered in the online help are listed below.

- Nonlinear Static Pushover Analysis
- Define Frame Hinge Properties
- Frame Hinge Property Data
- Frame Hinge Property Data For XXX
- Frame Hinge Interaction Surface
- Interaction Surface Definition
- Define Static Pushover Cases
- Static Pushover Case Data
- Assign Frame Hinges (Pushover)
- Pushover Curve
- Override Axis Labels And Range
- Override Structural Behavior Type

The Nonlinear Static Pushover Analysis topic gives an overview of the SAP2000 pushover analysis capabilities. The other topics provide context-sensitive help for dialog boxes of the same name.

Finally, it is emphasized that the SAP2000 documentation for the pushover is not intended to, and does not, document the pushover analysis method, but rather is intended to document the pushover analysis capabilities of SAP2000. For information regarding the pushover analysis method you should refer to the ATC-40 and FEMA-273 documents which are referenced below.

ATC, 1996
Seismic Evaluation and Retrofit of Concrete Buildings, Volume 1, ATC-40 Report, Applied Technology Council, Redwood City, California.

FEMA, 1997
NEHRP Guidelines for the Seismic Rehabilitation of Buildings, Developed by the Building Seismic Safety Council for the Federal Emergency Management Agency (Report No. FEMA 273), Washington, D.C.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 WEB TUTORIAL 2
QUICK PUSHOVER ANALYSIS TUTORIAL

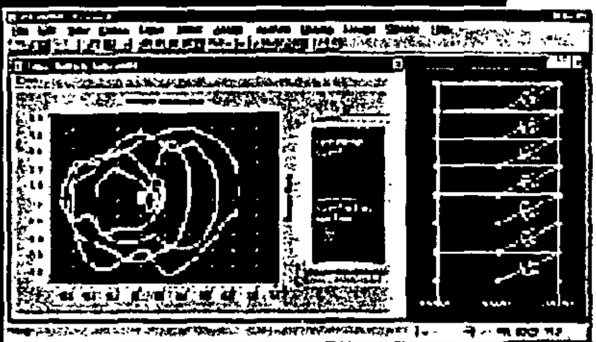
EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels 5521.4021 al 24, 5623 2910 y 5623.2971 • Fax: 5510.0573

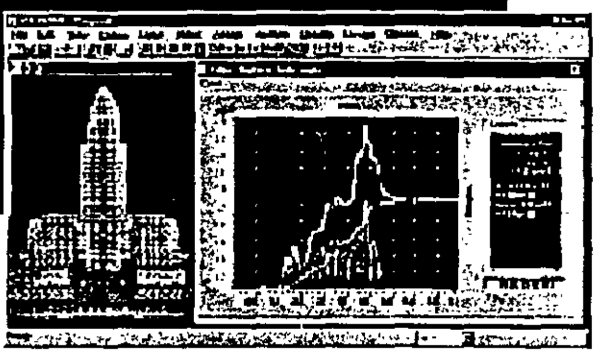
STATIC PUSHOVER
BASE ISOLATORS
VISCOUS DAMPERS
STRUCTURAL POUNDING

SAP2000 NONLINEAR

SAP2000/NL-PUSH NONLINEAR ELEMENTS

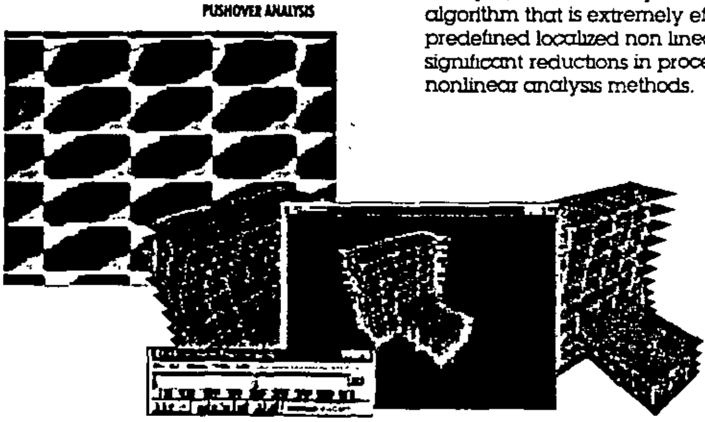


FORCE-DEFORMATION RELATIONSHIPS



ENERGY VS. TIME DISPLAYS

PUSHOVER ANALYSIS



PLAYING AVI FILE

SAP2000 Nonlinear extends the capabilities of the PLUS version to include nonlinear analysis options

The Frame Plastic Hinge Element for use with Static Pushover Analysis
- Axial, flexural, shear and torsional hinge
- Axial load-biaxial moment interaction
- Multilinear behavior including softening
- P-delta option

Pushover Analysis Option
- Static pushover analysis starting from gravity loads
- Force or displacement control
- User-defined lateral load patterns
- Effective damping computations
- Capacity-spectrum computation
- Demand-spectrum comparisons

The Nonlinear Link Element
- For use with the dynamic time history analysis option
- Link may be placed between any two joints or from joint to ground
- Viscous damper with nonlinear exponent on velocity term
- Gap (compression only) and Hook (tension only)
- Uniaxial plasticity (all 6 degrees of freedom)
- Base isolator with biaxial plasticity behavior
- Base isolator with friction and/or pendulum behavior
- AVI file option for creating real time displays of nonlinear deformation behavior
- Force versus deformation plots of nonlinear systems for energy dissipation studies

The Wilson FNA Method
The SAP2000 nonlinear time history analysis uses the new numerical integration technique known as the Wilson FNA (Fast Nonlinear Analysis) Method. The procedure uses an iterative vector superposition algorithm that is extremely efficient for analyzing structures with predefined localized non linearities. The method has demonstrated significant reductions in processing times when compared with other nonlinear analysis methods.

SAP2000®

Integrated
Finite Elements Analysis
and
Design of Structures

SAP2000 Web Tutorial 2

QUICK PUSHOVER ANALYSIS TUTORIAL

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

Issue Date: June 1998
Revision Number : 0
Revision Date: N/A

COPYRIGHT

Further information may be obtained from:

Computers and Structures, Inc
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *support@csiberkeley.com*
web: *www.csiberkeley.com*

Table Of Contents

Introduction 1
1. Assigning hinge properties (pushover) 1
2. Defining static pushover cases 4
3. Running the pushover analysis 6
4. Displaying the pushover deformed shape and the sequence of hinge formation 7
5. Displaying frame element forces at each step of the pushover 10
6. Displaying the pushover and capacity spectrum curves 11

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Introduction

This is a quick tutorial that introduces some of the pushover analysis features available in SAP2000. For a more detailed pushover analysis tutorial refer to the SAP2000 Web Tutorial 1. The basic model used in this tutorial, Sapwb02.sdb, is developed from scratch in SAP2000 Web Tutorial 1.

SAP2000 has extensive online help for the pushover analysis. Press F1 from within a pushover related dialog box to get context sensitive help related to the dialog box. To see all of the help items related to pushover analysis, select **Search For Help On...** from the **Help** menu, select the Index Tab, highlight the Pushover index entry and click the **Display** button.

To begin this tutorial, open the file named Sapweb02.sdb using the **Open...** command from the **File** menu.

> *Note: It may be helpful to save the file with a different name so that you retain a copy of the original file. You can do this by selecting* ***Save As...*** *from the* ***File*** *menu.*

Step 1: Assigning Hinge Properties (Pushover)

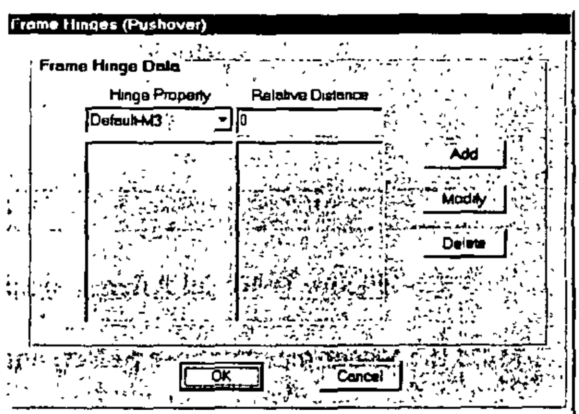
We will assign Default-PMM hinges to each end of the moment frame columns and the braced frame columns. We will assign Default-M3 hinges to each end of the moment frame beams. We will assign Default-P to the center of each brace.

1. Click in the window labeled 3-D View to make sure it is active. Note when the window is active, its title bar will be highlighted.

2. From the **Select** menu choose **Select**, and then **Groups...**from the submenu. This will display the Select Groups dialog box.

3. Highlight the group named FRCOLS (frame columns) by clicking on it.

4. Hold down the CTRL key and highlight the group named BRCOLS (braced frame columns) by clicking on it to add it to the selection.



5. Click the **OK** button to select all of the elements in the FRCOLS and BRCOLS groups.

6. From the **Assign** menu choose **Frame**, and then **Hinges (Pushover)...** from the submenu. This will display the Frame Hinges (Pushover) dialog box.

7. In this dialog box:

- Select Default-PMM in the Hinge Property drop-down box.

- Type **0** in the Relative Distance edit box.

  Note: Relative distance is measured from the I-end of the frame element, and is based on the clear length between rigid end offsets. A "0" relative distance indicates the hinge is at the I-end of the frame element. A "1" relative distance indicates the hinge is at the j-end of the frame element. Hinges may be located anywhere along the length of the frame element.

- Click the **Add** button to add a PMM hinge at the I-end (relative distance equals 0) end of the frame element.

- Type **1** in the Relative Distance edit box.

- Click the **Add** button to add a PMM hinge at the J-end (relative distance equals 1) end of the frame element.

- Click the **OK** button to assign the hinge properties. The model now appears as shown in Figure 1.

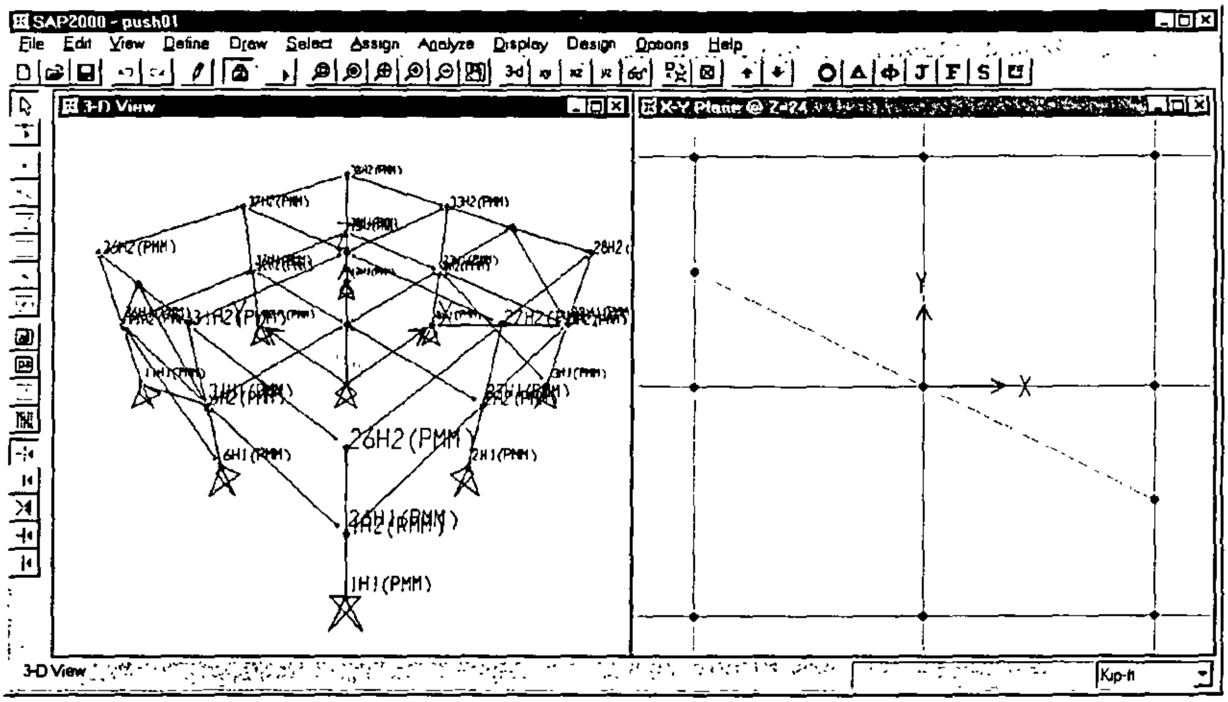


Figure 1: Model After Assigning Default-PMM Hinges To Columns

8. From the **Select** menu choose **Select**, and then **Groups...** from the submenu. This will display the Select Groups dialog box.

9. Highlight the group named FRMGIRD by clicking on it and click the **OK** button.

10. From the **Assign** menu choose **Frame**, and then **Hinges (Pushover)...** from the submenu. This will display the Frame Hinges (Pushover) dialog box.

11. In this dialog box:

    - Select Default-M3 in the Hinge Property drop-down box.
    - Type **0** in the Relative Distance edit box.
    - Click the **Add** button.
    - Type **1** in the Relative Distance edit box.
    - Click the **Add** button.
    - Click the **OK** button to assign the hinge properties.

12. From the **Select** menu choose **Select**, and then **Groups...**from the submenu. This will display the Select Groups dialog box.

13. Highlight the group named BRACE1 by clicking on it.

14. Hold down the CTRL key and highlight the group named BRACE2 by clicking on it to add it to the selection.

15. Click the **OK** button to select all of the elements in the BRACE1 and BRACE2 groups.

16. From the **Assign** menu choose **Frame**, and then **Hinges (Pushover)...**from the submenu. This will display the Frame Hinges (Pushover) dialog box.

17. In this dialog box:

    - Select Default-P in the Hinge Property drop-down box.
    - Type **.5** in the Relative Distance edit box.
    - Click the **Add** button to add a P hinge at the center (relative distance equals 0.5) of the beam.
    - Click the **OK** button to assign the hinge properties.

18. From the **Assign** menu select **Clear Display of Assigns** to clear the display of hinge assignments

19. Click the **Save Model** button on the main toolbar, or select **Save** from the **File** menu to save the file.

This completes the assigning of pushover hinge properties.

Step 2: Defining Static Pushover Cases

For this tutorial example we will define two static pushover cases. The first will apply the gravity load to the structure, and the other will apply lateral load to the structure.

1. From the **Define** menu choose **Static Pushover Cases....** This will again display the Define Static Pushover Cases dialog box.

2. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

3. In this dialog box:

   - Type **GRAV** in the Pushover Case Name edit box.

   - In the Options Area select the Push To Load Level Defined By Pattern option button.

     Note: Typically the Push To Load Level Defined By Pattern option is used to apply gravity load, and the Push To Displacement Of option is used to apply lateral pushes.

   - In the Load Pattern do the following:

     ✓ Select DL from the Load drop-down box.

     ✓ Type **1** in the Scale Factor edit box.

     ✓ Click the **Add** button.

     ✓ Select LL from the Load drop-down box.

     ✓ Type **0.25** in the Scale Factor edit box.

     ✓ Click the **Add** button.

   - Accept all of the other default values. Note that the default Control Joint, joint 19, occurs at the roof. The dialog box appears as shown in Figure 2.

   - Click the **OK** button to complete this pushover case definition.

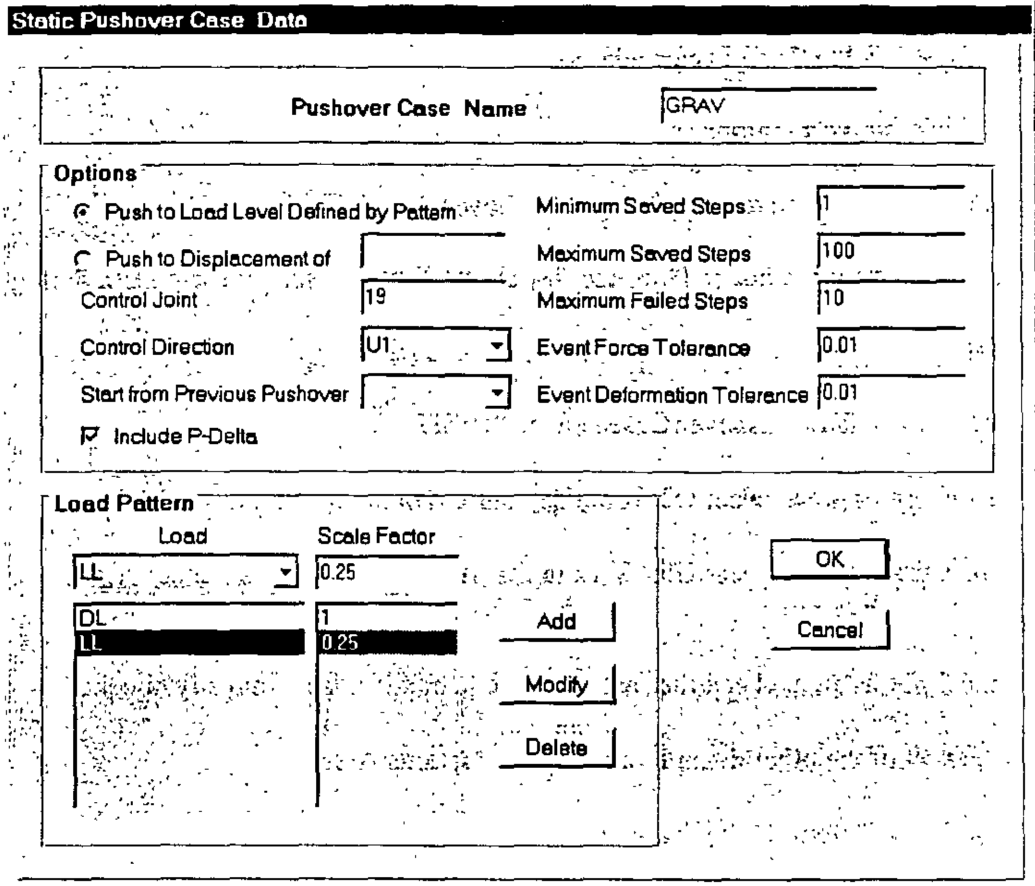


Figure 2: Static Pushover Case Data Dialog Box

4. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

5. In this dialog box:

- Accept the default Pushover Case Name, PUSH2.
- In the Options area, select GRAV from the Start From Previous Pushover drop-down box.
- In the Options area, check the Include P-Delta box if it is not already checked.
- Accept all of the other default values in the Options Area.
- In the Load Pattern do the following:

  ✓ Select PUSHPAT from the Load drop-down box.

  Note: PUSHPAT is a previously defined static load case that applies an inverted triangular pattern of load over the height of the structure.

- ✓ Type **1** in the Scale Factor edit box.
- ✓ Click the **Add** button.

- Click the **OK** button to complete this pushover case definition.

6. Click the **Add New Pushover** button to display the Static Pushover Case Data dialog box.

7. In this dialog box:

- Accept the default Pushover Case Name, PUSH3.
- In the Options area, select U2 from the Control Direction drop-down box.
- In the Options area, select GRAV from the Start From Previous Pushover drop-down box.
- In the Options area, check the Include P-Delta box if it is not already checked.
- Accept all of the other default values in the Options Area.
- In the Load Pattern do the following:
  - ✓ Select acc dir Y from the Load drop-down box.

    Note: The acc dir y load pattern provides a uniform acceleration in the Y direction, that is, a lateral load that is proportional to the mass.

  - ✓ Type **1** in the Scale Factor edit box.
  - ✓ Click the **Add** button.
- Click the **OK** button to complete this pushover case definition.

8. Click the **OK** button to exit the Define Static pushover Cases dialog box.

9. Click the **Save Model** button on the main toolbar.

This completes the definition of static pushover cases.

Step 3: Running The Pushover Analysis

1. On the **Analyze** menu select **Run Static Pushover**. (In SAP2000 Version 6.20 Beta, you will need to run the static analysis first using the **Run** option on the **Analyze** menu.)

Note: To run a pushover analysis, you must first have pushover hinges and pushover load cases defined, at least a static analysis run, and, if steel members with Auto sections, or concrete members whose reinforcing is to be designed by the program are included, you must have run the design portion of the program.

2. A window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display as shown in Figure 3.

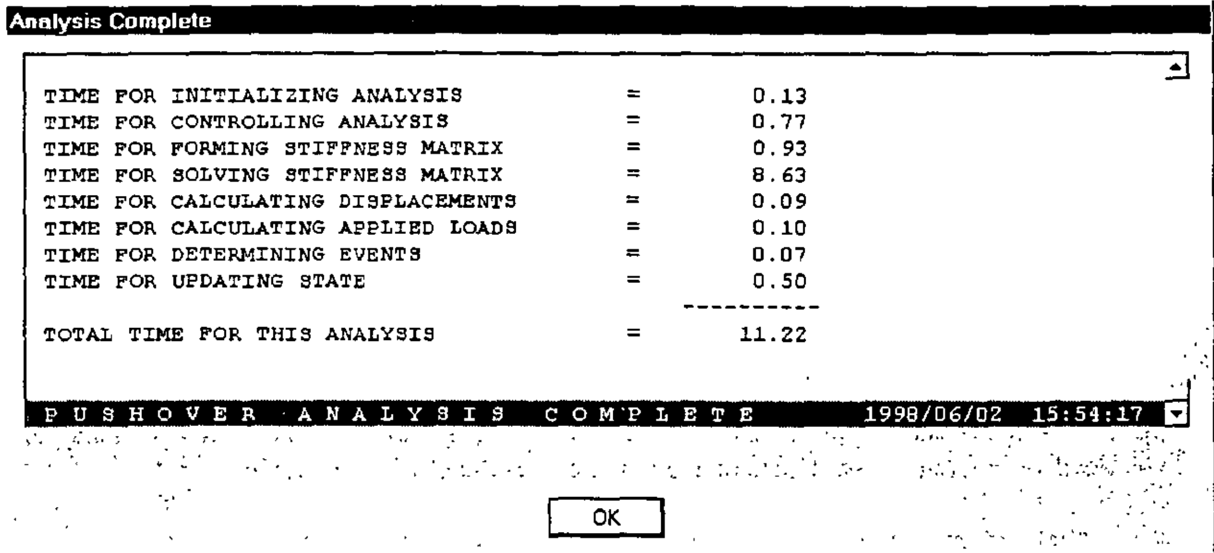


Figure 3: Screen Message When Pushover Analysis Is Complete

3. Use the scroll bar to review the analysis messages and check for any error or warning messages (there should be none).

4. Click the **OK** button in the Analysis window to close it.

This completes running the pushover analysis.

Step 4: Displaying the Pushover Deformed Shape and the Sequence of Pushover Hinge Formation

1. From the **Options** menu select **Windows** and the select **One** from the submenu. The display changes to one window.

   Note: With this option you can select to have from one to four windows on the screen at the same time. Each of the windows can be showing a completely different view.

2. Click the **Show Undeformed Shape** button to clear the display of the first mode shape.

3. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu). This displays the Set Elements Dialog box. In this dialog box check the Hide box in the Shells area and click the **OK** button. This will turn off the display of shell elements.

4. Click the **Display Static Deformed Shape** button, or from the **Display** menu select **Show Deformed Shape...** to display the Deformed Shape dialog box.

5. In this dialog box:

   - Select Push2 Static Push from the Load drop-down box.

   - Select Auto in the Scaling area.

   - Check both the Wire Shadow and the Cubic Curve options in the Options area.

   - Click the **OK** button.

6. The deformed shape will appear. Note that the title of the window includes the information "Deformed Shape (PUSH2 – Step 0)". Thus we are currently viewing the deformed shape at the start of the pushover. Since the PUSH2 pushover was started from the GRAV pushover, we are also viewing the deformed shape at the end of the GRAV pushover.

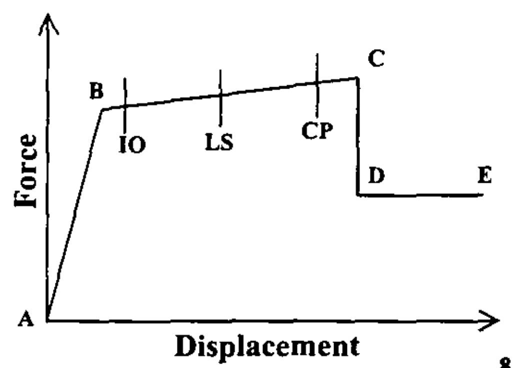
7. Right click on any second level or roof level joint to see the displacement for that joint. The displacements will be very small, as is expected when only the gravity load is applied.

8. Click the **Right Arrow** button ⇨ located on the right-hand side of the status bar at the bottom of the screen to view the deformed shape of the next step (Step 1) in the pushover.

   Note: When viewing the pushover deformed shape and sequence of hinge formation, the right and left arrow buttons, located in the status bar at the bottom of the screen, provide an easy way to view the deformed shape for the next (right arrow), or previous (left arrow) pushover step.

9. Note that the title of the window includes the information "Deformed Shape (PUSH2 – Step 1)", and the building moves slightly in the X-direction. You can again right-click on any joint to see its displacement. You can click the **Left Arrow** button ⇦ to return to Step 0 of the pushover.

10. Continue clicking the **Right Arrow** button ⇨ until the first colored hinges appear. The first colored hinges should appear in Step 4, as illustrated in Figure 4. The color of the hinges indicates the state of the hinge, i.e, where it is along its force displacement curve. The legend for the hinge colors is included at the bottom of the



screen. The points B, IO, LS, CP, C, D and E are illustrated in the figure. When hinges first appear, they are at point B on the force-displacement curve.

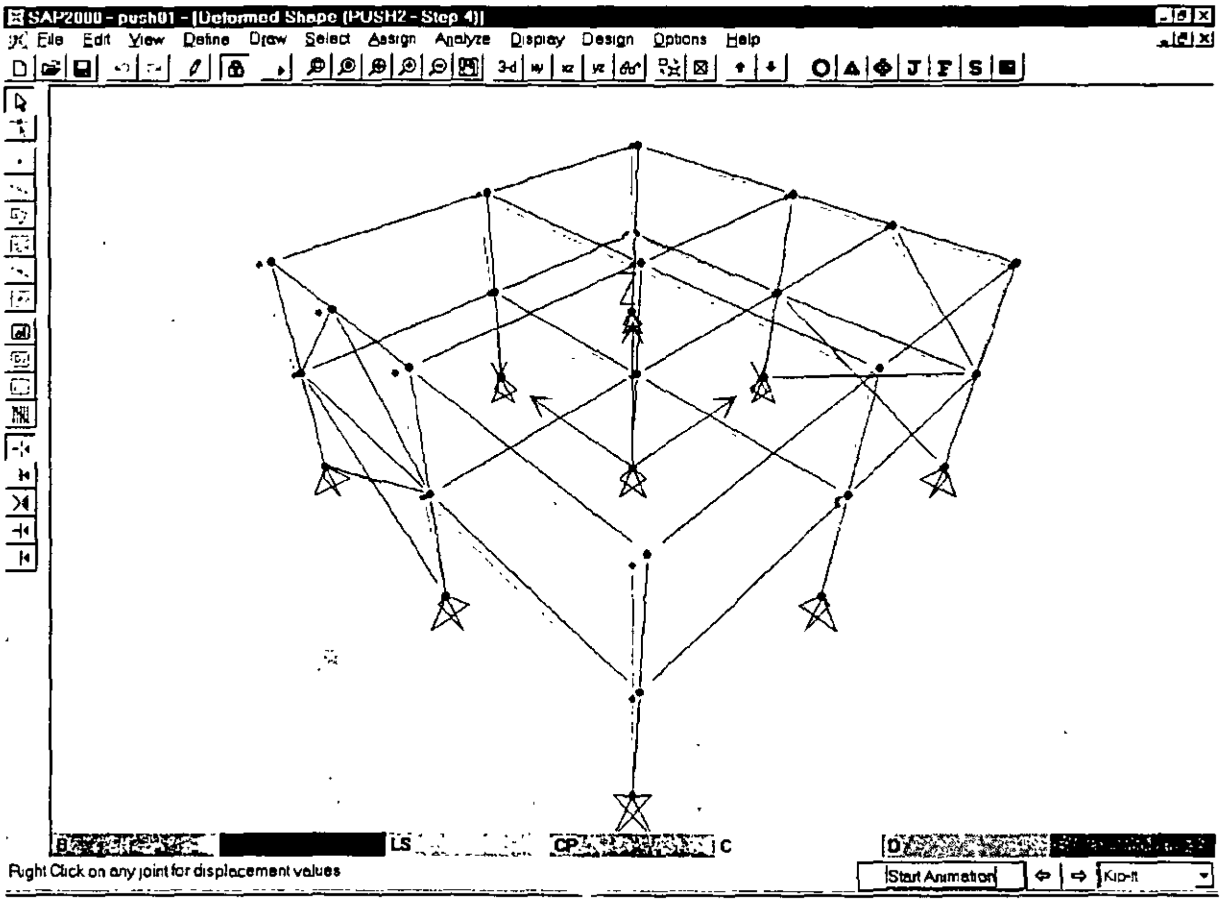


Figure 4: First Pushover Hinge Yielding

11. Continue clicking the **Right Arrow** button ⇨ to step through the sequence of hinge formation in the pushover to the last step. Note how the colors of the hinges change as the pushover proceeds.

Note: To change the color coding for the hinges, From the Options menu, select Colors, and then select the Output tab. The color-coding for the pushover hinge state is controlled by the colors in the area labeled Contours. There are ten contour colors; these contour colors also are used in displaying stress contours for shell elements. Assume the color in the top box, next to the label "Min", is designated "Box 1", and the color in the bottom box, next to the label "Max", is designated "Box 10". Then the color coding for pushover hinge states is as defined in the table.

Point	Color Box
B	Box 2
IO	Box 3
LS	Box 4
CP	Box 5
C	Box 6
D	Box 7
E	Box 8

12. Click the **Display Static Deformed Shape** button ▲, or from the **Display** menu select **Show Deformed Shape...** to display the Deformed Shape dialog box.

13. In this dialog box:

- Select Push3 Static Push from the Load drop-down box.
- Click the **OK** button.

14. Click the **Right Arrow** button ⇨ to step through the sequence of hinge formation

15. Click the **Show Undeformed Shape** button to clear the display of deformed shape for the pushover.

This completes the review of the pushover deformed shape and the sequence of pushover hinge formation.

Step 5: Displaying Frame Element Forces at Each Step of the Pushover

1. Click the **Member Force Diagram for Frames** button **F**, or from the **Display** menu select **Show Element Forces/Stresses**, and then **Frames...** to display the Member Force Diagram for Frames dialog box.

2. In this dialog box:

   - Select Push2 Static Push from the Load drop-down box.
   - In the Component area select the Moment 3-3 option.
   - In the Scaling area select the Auto option.
   - Check the Fill Diagram box. Note if the Show Values on Diagram box is checked, you will have to uncheck it before you can check the Fill Diagram box.

     Note: The frame element forces for each step of the pushover can be displayed either with the filled diagram or with the values shown on the diagram.

   - Click the **OK** button. The display appears as shown in Figure 5.

3. Click the **Right Arrow** button ⇨ located on the right-hand side of the status bar at the bottom of the screen to view the M-33 diagram at the next step (Step 1) of the pushover.

   Note: When viewing the pushover forces, the right and left arrow buttons, located in the status bar at the bottom of the screen, provide an easy way to view the forces for the next (right arrow), or previous (left arrow) pushover step.

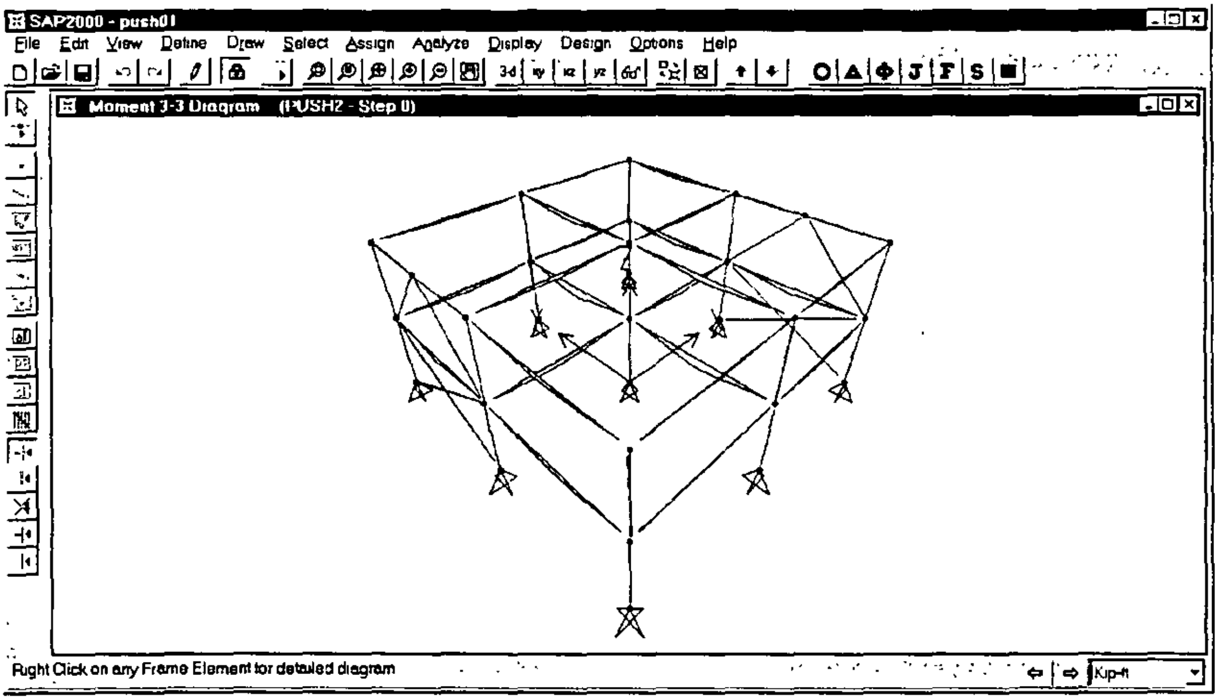


Figure 5: Pushover Forces (M3-3) at Step 0

4. Continue clicking the **Right Arrow** button ⇨ to step through the moment diagrams for each step of the pushover. You can right click an element at any step for a detailed diagram.

5. In this dialog box:

   - Select Push3 Static Push from the Load drop-down box.
   - In the Component area select the Axial Force option.
   - Click the **OK** button.

6. Click the **Right Arrow** button ⇨ to step through the axial force diagrams for the pushover.

7. Click the **Show Undeformed Shape** button to clear the display of element forces for the pushover.

This completes the review of the frame element forces at each step of the pushover.

Step 6: Displaying Pushover and Capacity Spectrum Curves

1. From the **Display** menu select **Show Static Pushover Curve...** to display the Pushover Curve dialog box.

Note: If this dialog box does not fully fit on your screen then you may want to increase the resolution of your screen. The dialog box should fully fit on the screen if your resolution is 800 x 600 with small fonts, or 1024 x 768 with large fonts.

2. If no plot is visible in the plot area, then click the **Display** button at the bottom of the form.

   *Note: If at any time the plot area display is not visible in the Pushover curve dialog box, click the **Display** button.*

3. Notice that in the Plot Type area, the Base Shear vs Control Displacement option is selected.

   Note: The base shear that is plotted in the Base Shear vs Control Displacement plot is the resultant base shear. The displacement plotted is the displacement in the control direction (not resultant) at the control joint.

4. Currently we are looking at the pushover curve for the PUSH2 pushover load case. The dialog box appears as shown in Figure 6.

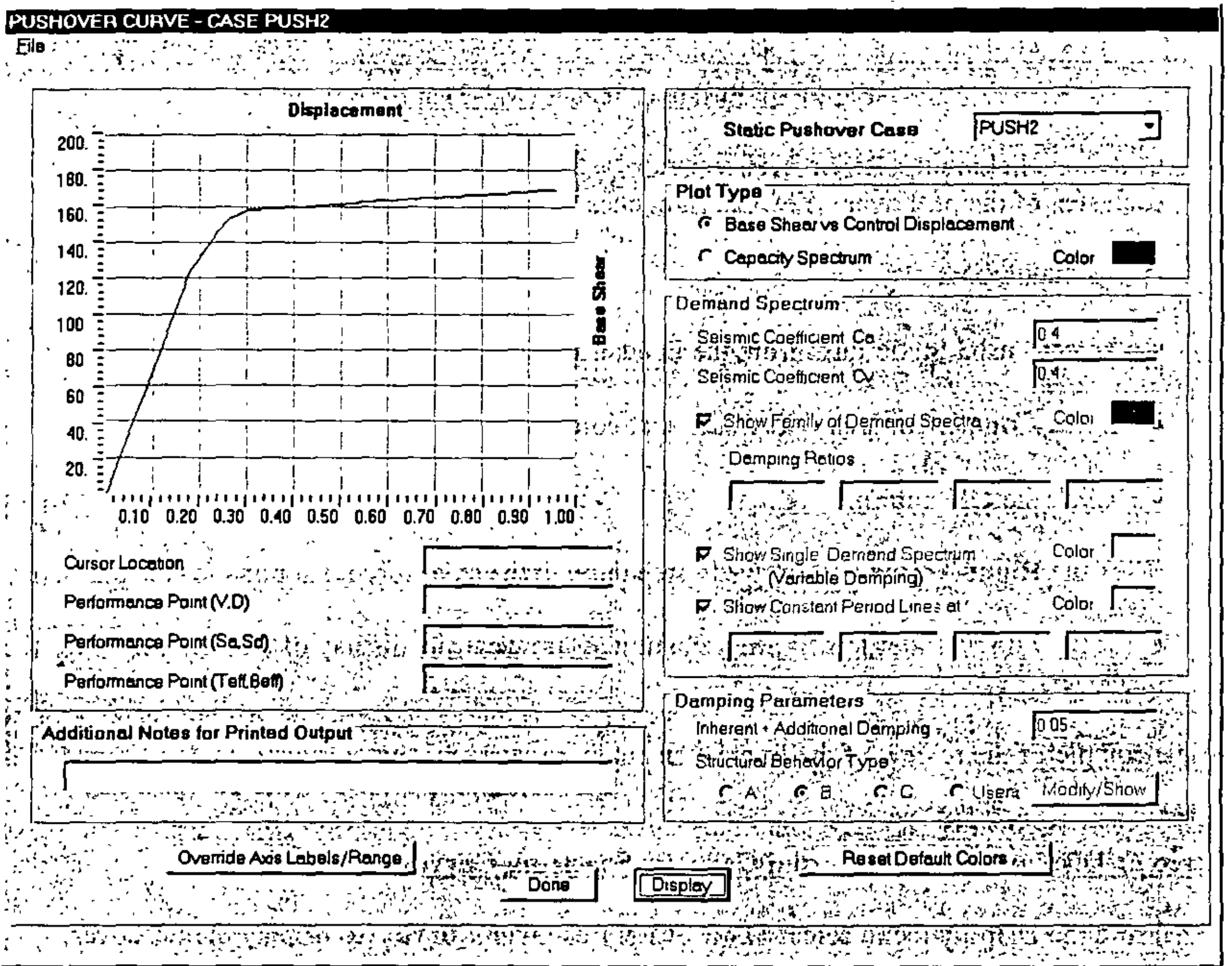


Figure 6: Pushover Curve For Pushover Load Case PUSH2

PUSHOVER CURVE

File

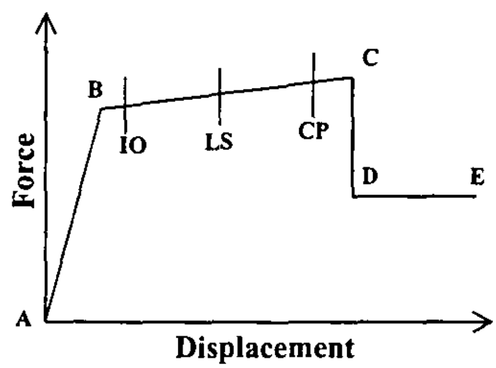
Step	Displacement	Base Shear	A-B	B-IO	IO-LS	LS-CP	CP-C	C-D	D-E	>E	TOTAL
0	4.002E-04	0.0000	48	0	0	0	0	0	0	0	48
1	0.0484	28.3708	48	0	0	0	0	0	0	0	48
2	0.0964	56.7417	48	0	0	0	0	0	0	0	48
3	0.1444	85.1127	48	0	0	0	0	0	0	0	48
4	0.1856	109.4811	46	2	0	0	0	0	0	0	48
5	0.2336	125.1036	46	2	0	0	0	0	0	0	48
6	0.2398	127.1146	44	4	0	0	0	0	0	0	48
7	0.2878	136.8057	44	4	0	0	0	0	0	0	48
8	0.3358	146.4968	44	4	0	0	0	0	0	0	48
9	0.3780	154.9516	42	4	2	0	0	0	0	0	48
10	0.4148	158.5392	41	5	2	0	0	0	0	0	48
11	0.4154	158.5750	40	6	2	0	0	0	0	0	48
12	0.4634	159.4686	40	4	4	0	0	0	0	0	48
13	0.5114	160.3621	40	4	4	0	0	0	0	0	48
14	0.5594	161.2557	40	4	4	0	0	0	0	0	48
15	0.6074	162.1492	40	2	6	0	0	0	0	0	48
16	0.6554	163.0428	40	0	8	0	0	0	0	0	48
17	0.7034	163.9364	40	0	8	0	0	0	0	0	48
18	0.7514	164.8299	40	0	8	0	0	0	0	0	48
19	0.7994	165.7235	40	0	8	0	0	0	0	0	48
20	0.8474	166.6170	40	0	8	0	0	0	0	0	48
21	0.8954	167.5106	40	0	6	2	0	0	0	0	48

Figure 7: Table For Pushover Curve

5. From the File menu at the top of the Pushover Curve dialog box select Display Tables. The table shown in Figure 7 appears.

6. In this table note the following:

   - Step identifies the step number in the pushover curve.
   - Displacement and Base Shear define a point on the pushover curve for the associated step.
   - A-B, B-IO, IO-LS, LS-CP, CP-C, C-D, D-E, >E all identify the total number of hinges within each of these ranges on their associated force-displacement curves.
   - TOTAL is the total number of pushover hinges in the structure.



7. Click the "X" in the upper right-hand corner of the table to close it.

8. Select the Capacity Spectrum option in the Plot Type area. As shown in Figure 8, the plot changes and the Demand Spectrum area and the Damping Parameters area are now active.

9. From the File menu at the top of the Pushover Curve dialog box select Display Tables. The table shown in Figure 9 appears.

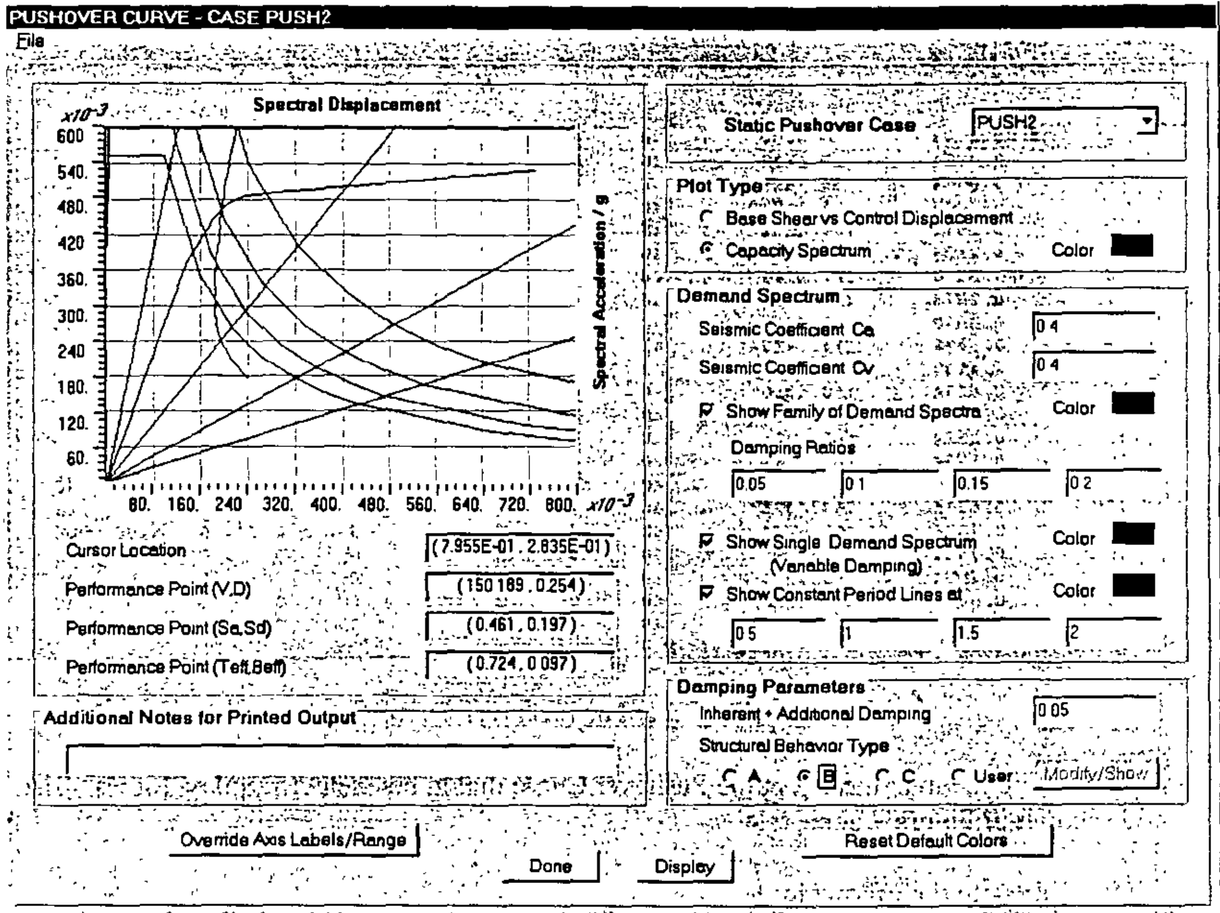


Figure 8: Capacity spectrum For Pushover Load Case PUSH2

PUSHOVER CAPACITY/DEMAND COMPARISON

File

Step	Teff	Beff	Sd(C)	Sa(C)	Sd(D)	Sa(D)	ALPHA	PF*Ø
0	0.756	0.050	4.002E-04	0.000	0.247	0.529	1.000	1.000
1	0.756	0.046	0.039	0.083	0.252	0.541	0.955	1.246
2	0.756	0.049	0.078	0.167	0.248	0.532	0.955	1.241
3	0.756	0.050	0.116	0.250	0.247	0.530	0.955	1.240
4	0.756	0.050	0.150	0.321	0.247	0.530	0.955	1.239
5	0.792	0.083	0.188	0.368	0.226	0.441	0.953	1.241
6	0.796	0.086	0.193	0.374	0.225	0.435	0.953	1.242
7	0.839	0.117	0.231	0.403	0.216	0.376	0.952	1.244
8	0.875	0.136	0.269	0.432	0.215	0.344	0.951	1.246
9	0.902	0.145	0.303	0.457	0.216	0.326	0.950	1.248
10	0.931	0.161	0.331	0.469	0.215	0.305	0.948	1.252
11	0.932	0.161	0.332	0.469	0.215	0.305	0.947	1.252
12	0.976	0.187	0.368	0.474	0.214	0.276	0.943	1.260
13	1.018	0.208	0.404	0.478	0.214	0.254	0.940	1.266
14	1.058	0.223	0.440	0.483	0.217	0.238	0.936	1.270
15	1.096	0.233	0.477	0.487	0.221	0.226	0.934	1.274
16	1.133	0.240	0.513	0.491	0.225	0.215	0.931	1.278
17	1.167	0.247	0.549	0.495	0.230	0.207	0.929	1.281
18	1.201	0.252	0.586	0.498	0.234	0.199	0.927	1.283

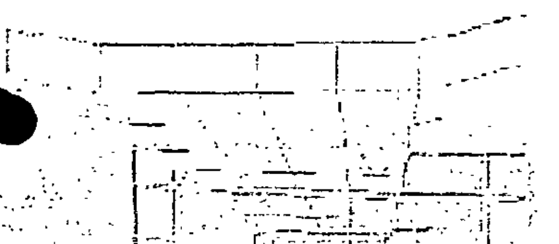
Figure 9: Table For Capacity Spectrum

10. Click the "X" in the upper right-hand corner of the table to close it.

11. Select Push3 from the Static Pushover Case drop-down box to view the capacity spectrum for the Push3 pushover load case.

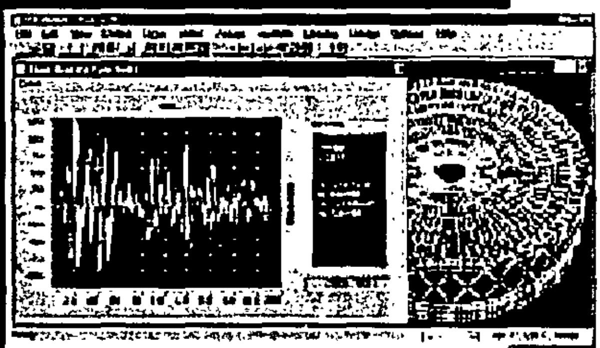
11. Click the **Done** button to close the Pushover Curve dialog box.

This completes the review of displaying the pushover and capacity spectrum curves, and it completes this tutorial. If you want further information on the SAP2000 pushover analysis features, then refer to the SAP2000 Web Tutorial 1 and to the online help.

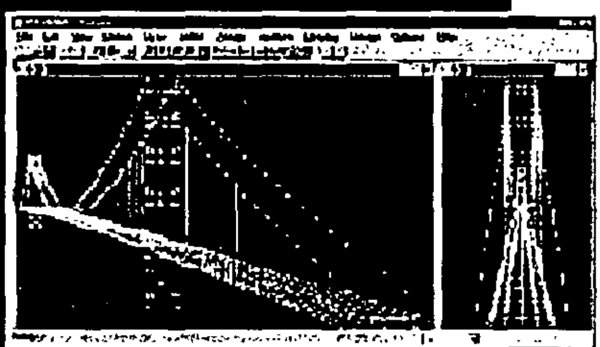


UNLIMITED CAPACITY
SOLID ELEMENTS
TIME HISTORY
BRIDGE ANALYSIS

SAP2000 PLUS



TIME HISTORY ANALYSIS



BRIDGE ANALYSIS

SAP2000 PLUS ELEMENTS

SAP2000 Plus extends the capabilities of the standard version with unlimited capacity and additional analytical capabilities

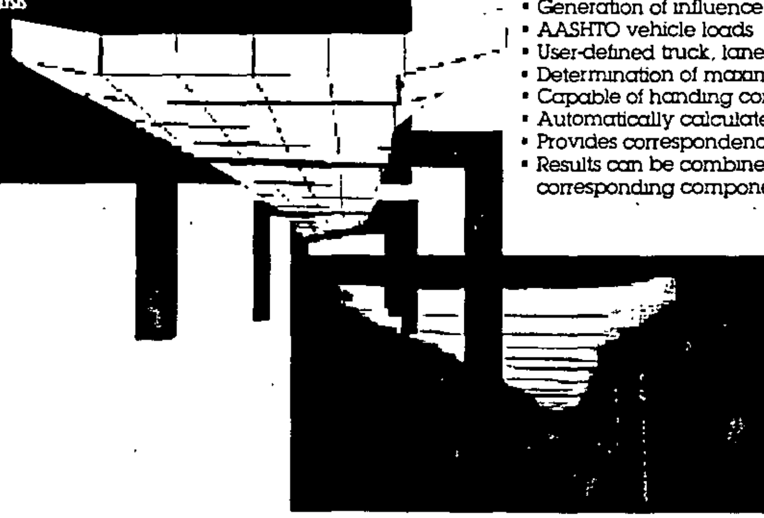
PLANE Element
- 3 to 9 nodes plane stress or plane strain element
- Orthotropic material properties
- Gravity, thermal, surface pressure and pressure gradient loading

ASOLID Element
- 3 to 9 nodes axisymmetric element
- Orthotropic material properties
- Gravity, thermal, surface pressure and pressure gradient loading

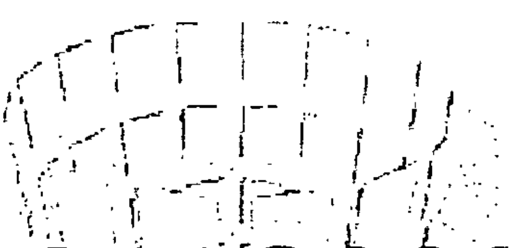
SOLID Element
- Three dimensional 8 node brick element
- Anisotropic material properties
- Gravity, thermal, surface pressure and pressure gradient loading

Dynamic Time History Analysis
- Ground acceleration excitation
- Multiple base excitation
- Load forcing functions
- Transient or steady-state
- Multiple time history cases
- Sequential history cases
- Time history Windows AVI file
- Graphic displays of nodal and element time history records
- Functions vs time or function vs function displays
- Generation of response spectrum curves for any joint acceleration component
- Results can be combined with other loads for enveloping or step by step steel and concrete design

Bridge Analysis
- Moving load analysis
- Generation of influence lines and forces envelopes
- AASHTO vehicle loads
- User-defined truck, lane and train loads
- Determination of maximum and minimum displacements and reactions
- Capable of handing complex lane geometries
- Automatically calculates all possible permutations of traffic loads
- Provides correspondence between response components
- Results can be combined with other loads for enveloping or corresponding components for steel and concrete design



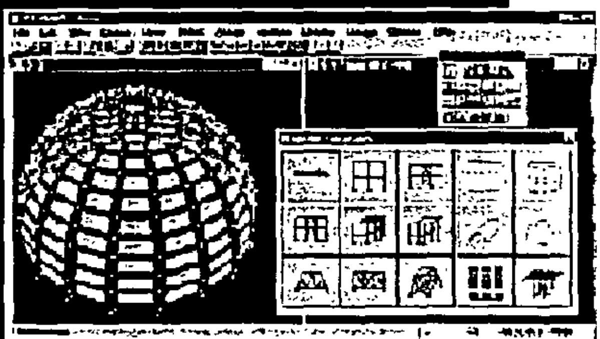
BRIDGE ANALYSIS



THE MOST POWERFUL IS NOW ALSO THE MOST FRIENDLY

SAP2000 STANDARD

THE STANDARD ELEMENTS

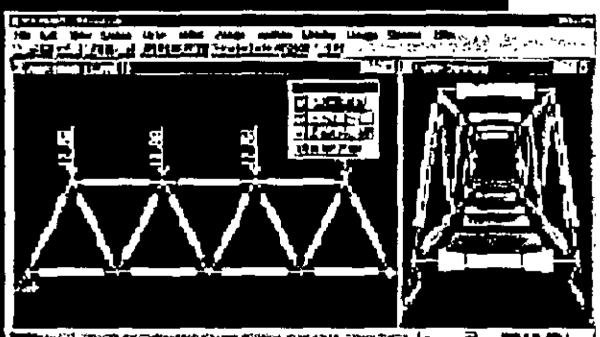


TEMPLATES

The 2D and 3D Beam and Truss Element
- Multiple non-prismatic segments over element length
- Point, uniform and trapezoidal loading in any direction
- Temperature and thermal gradient loading
- Prestress loading
- Automated end offset evaluation
- Moment and shear releases
- Built-in steel sections

The 3D Shell Element
- General quadrilateral or triangular element
- Orthotropic materials
- Six degrees of freedom per joint
- Shell, plate or membrane action
- Thick shell option
- Gravity, uniform, pressure, temperature and thermal gradient loading

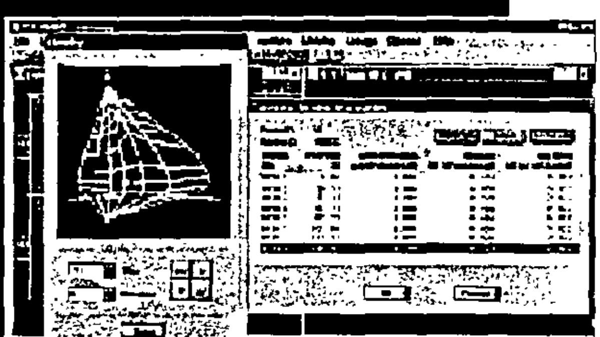
The Spring Element
- Joint to ground (support) spring
- Global and skewed springs
- Coupled 6x6 user-defined spring stiffness option (for foundation modeling)



FRAME ELEMENTS

Analytical Options
- Static and/or dynamic response spectrum analysis
- P-delta analysis with either static or dynamic analysis
- Blocked active column equation solver
- Automated fast profile optimization
- Generalized joint constraint options including: rigid bodies, diaphragms, rods and welds
- Applied force and applied displacement loading
- Gravity, pressure and thermal loading
- Eigen analysis with an accelerated subspace iteration algorithm
- Ritz analysis for fast predominant mode evaluation
- Multiple response spectrum cases in single run
- Modal combination by the SRSS, the CQC or the GMC (Gupta) method
- Directional combinations by the ABS or the SRSS method
- Static and dynamic response combinations and envelopes

Analysis Output Display Options
- 3D perspective graphical displays of undeformed and deformed structural geometries
- Static deformed shapes and mode shapes
- Loading diagrams
- Bending moment and shear force diagrams
- Stress contours
- Animation of deformed shapes and mode shapes
- Animated stress contours
- Multiple windows displaying different parameters
- Instantaneous graphical and tabulated output details for specific joint or element with right button click
- Static and dynamic load combinations and envelopes



CONCRETE DESIGN

Steel and Concrete Design
- Fully interactive and graphical steel and concrete frame member design
- AISC (ASD and LRFD), ACI, British and Eurocodes
- Design for static and dynamic loads
- Ductile and non-ductile design
- Member-grouping for design envelopes
- Detailed onscreen design information with right button click
- Steel member selection and optimization
- Biaxial moment-axial load column interaction diagrams

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 WEB TUTORIAL 2
SAFE PROBLEM 1

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

SAFE Problem 1

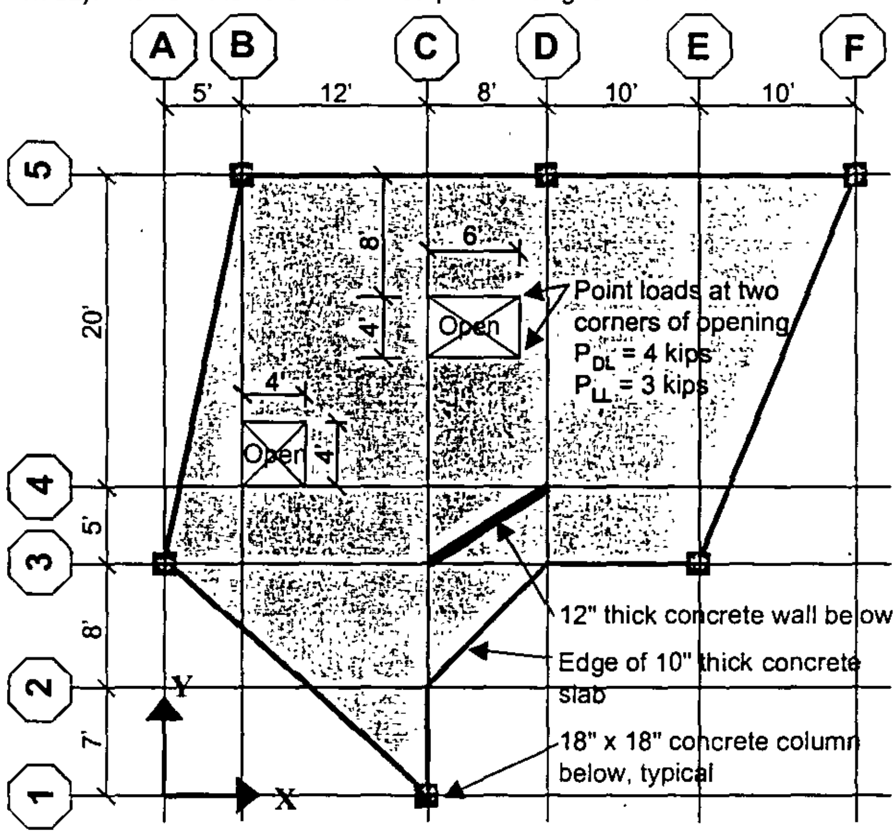
Irregular Flat Slab

Concrete Slab Properties
E = 4000 ksi, Poissons Ratio = 0.2, Unit Weight = 150 pcf
f'c = 4 ksi, fy = 60 ksi
Assume default cover for reinforcing steel

Concrete Columns and Wall Properties
E = 4000 ksi, Poissons Ratio = 0.2, Height =12'

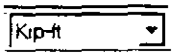
Slab Uniform Loading
Dead Load: Slab self weight plus 50 psf superimposed DL
Live Load: 100 psf typical, 250 psf between column lines 3, 5, E and F

To Do
Review the elastic and cracked slab vertical displacements under unfactored DL + LL loading. Review the factored M_{XX} slab moments (1.4DL + 1.7LL). Review the calculated X-strip reinforcing for ACI 318-95.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

SAFE Problem 1 Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model...**. This displays the Grid Definition dialog box.

3. In this dialog box:

- Click the **Edit Grid** button to display the Define X Grid dialog box.

- In this dialog box:

  ➢ In the Display area verify that the X Grid option is selected.

  ➢ In the Display Grid As area verify that the Ordinates option is selected.

  ➢ Fill in the Grid ID and Coordinate spreadsheet columns as shown in Figure 1-1.

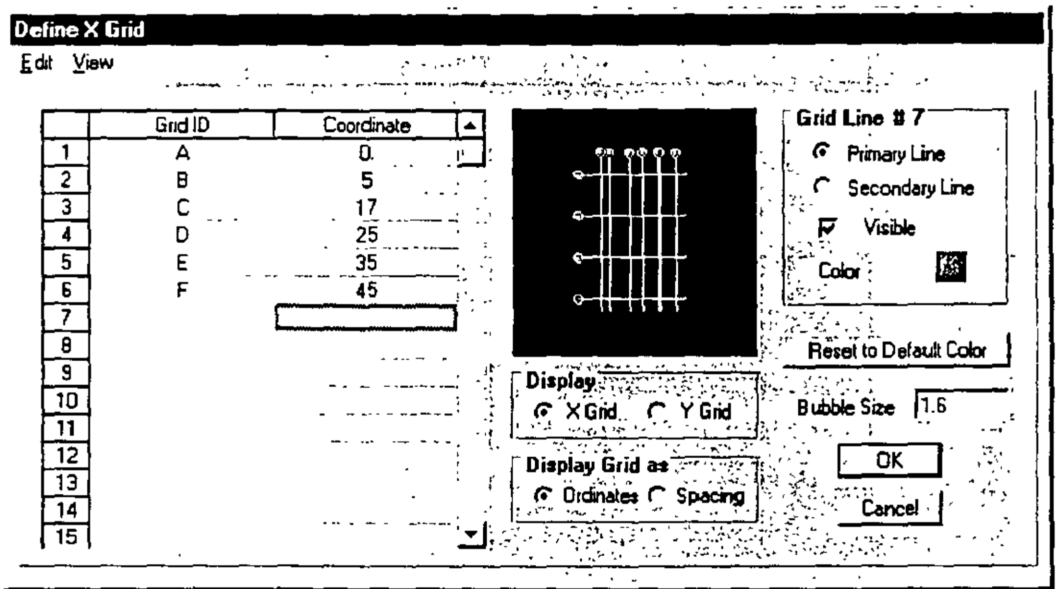


Figure 1-1: X Grid Definition

  ➢ In the Display select the Y Grid option. Note that the title of the dialog box changes to Define Y Grid.

  ➢ Fill in the Grid ID and Coordinate spreadsheet columns as shown in Figure 1-2.

  ➢ Click the **OK** button to complete the grid definition. The screen will refresh and display two views of the model (grid lines at this point) in vertically tiled adjoining windows. The left-hand window is a structural layer plan view in the X-Y plane and the right-hand window is a structural layer 3-D view.

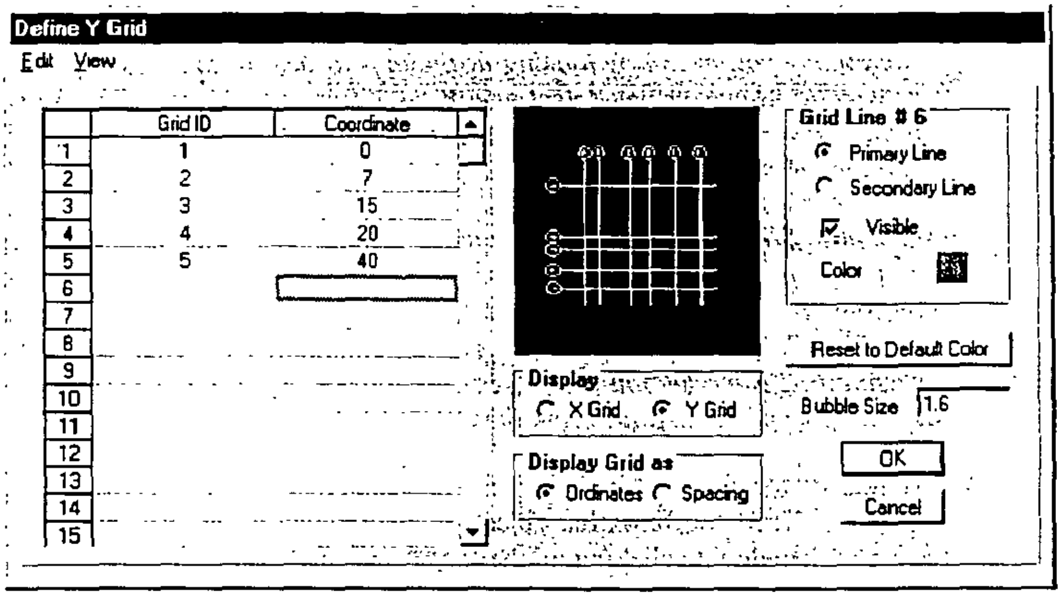


Figure 1-2: Y Grid Definition

Note: In SAFE information can be viewed on three different layers. They are the Structural Layer,. X-Design Strip Layer and Y-Design Strip Layer. The Structural Layer includes information about slab geometry, loads and boundary conditions. The X-Design Strip Layer is used to define the extent of the X-direction design strips. The Y-Design Strip Layer is used to define the extent of the Y-direction design strips. You can select the layer to be shown from the View menu.

4. Click in the window titled Structural Layer Plan View to make sure it is active. The window is active when its title bar is highlighted.

5. Verify that the **Snap To Points** button on the side toolbar is depressed thus enabling this snapping option.

*Note: You could also verify that the Snap To Points option is active by clicking the **Draw** menu, selecting **Snap To** and verifying that there is a check mark next to **Points** on the submenu. If the check mark exists then the option is active. If the check mark is not there, then click points and then check mark will appear indicating that the option is now active. This menu item is like a toggle switch. You can click on it to turn it on and off. The check mark tells you whether it is on (checked) or off (not checked).*

6. Click the **Draw Quadrilateral Area Object** button on the side toolbar (or select **Draw Quadrilateral Area Object** from the **Draw** menu). Note that you are now in Draw mode and that the shape of the mouse pointer has changed.

Note: There are two distinct modes in SAFE: Select mode and Draw Mode. Select mode, the default mode, is used for assigning properties and loads and for display of assignments and results. Draw mode is used to draw geometry. The mouse pointer is different depending on which mode you are in. In Select mode the mouse pointer looks *like this . In Draw mode the mouse pointer looks like this . You can always tell which mode you are in by looking at the appearance of the mouse pointer. You enter draw mode whenever you select one of the drawing tools either on the side toolbar or on the Draw menu. Clicking the Pointer button on the side toolbar will switch you from the Draw mode back to Select mode.*

Background: SAFE models are made up of area objects, line objects and point objects. Area objects are used to define slabs, uniformly distributed loads and soil (uniformly distributed) supports. Line objects are used to define beams, line loads and wall supports. Point objects are used to define point loads and column (point) supports. To create a SAFE model you draw these three types of objects (area, line and point) and assign appropriate definitions to them.

7. We will define the slab (not including the openings that will be addressed later) using three area objects. Draw the first area object by single clicking on grid intersections B-5, F-5 and E-3 and then double clicking on the fourth and final grid intersection at A-3.

   *Note: When you place the pointer near one of the grid intersections a dot will appear on the grid intersection with a title box that says "Grid Intersection". This indicates that you are within the tolerance of the Snap To Point feature and that if you click the mouse you will snap to the grid intersection. You can control the tolerance for snapping by selecting the **Options** menu, clicking **Preferences...**, selecting the Dimensions Tab and setting the Screen Snap To Tolerance.*

   Note: Double clicking on the last grid intersection completes the area object. Area objects can either be three-sided (triangle) or four-sided (quadrilateral). If you double click on the third point you will get a three-sided area object. Double clicking on the fourth point gives a four-sided area object. Instead of double clicking on the last point you can single click on the last point and then press the Enter key on the keyboard to complete the area object. This method works on both three-sided and four-sided area objects.

8. Draw a second area object by single clicking on grid intersections A-3, C-3 and C-1 and then pressing the Enter key on the keyboard.

9. Draw a third area object by single clicking on grid intersections C-3 and D-3 and then double clicking on the third and final grid intersection at C-2.

10. Click the **Draw Line Object** button on the side toolbar (or select **Draw Line Object** from the **Draw** menu).

11. Draw a line object by clicking on grid intersection C-3 and then double clicking on grid intersection D-4 to complete the line object. This line object will be used to define the wall support.

    Note: We could also single click on grid intersection D-4 and then press the Enter key on the keyboard to complete the line object.

12. Click the **Draw Point Object** button on the side toolbar (or select **Draw Point Object** from the **Draw** menu).

13. Draw a point object by double clicking on grid intersection D-5. This point object will be used to define a column support. The model appears as shown in Figure 1-3.

    Note: We could also single click on grid intersection D-5 and then press the Enter key on the keyboard to complete the point object.

    Note: It is not necessary to draw point objects for the other five columns because those point objects already exist. They were created when the area objects defining the slab were drawn. When area objects are drawn, point objects are automatically created at the corners of the area object. Similarly, when line objects are drawn point objects are automatically created at the ends of the line object.

    If you do not define a point object as either a support or a load then the program will ignore that object when the analysis is run. Thus you need not worry about any extraneous point objects that may have been created by the program at the corners of area objects or ends of line objects.

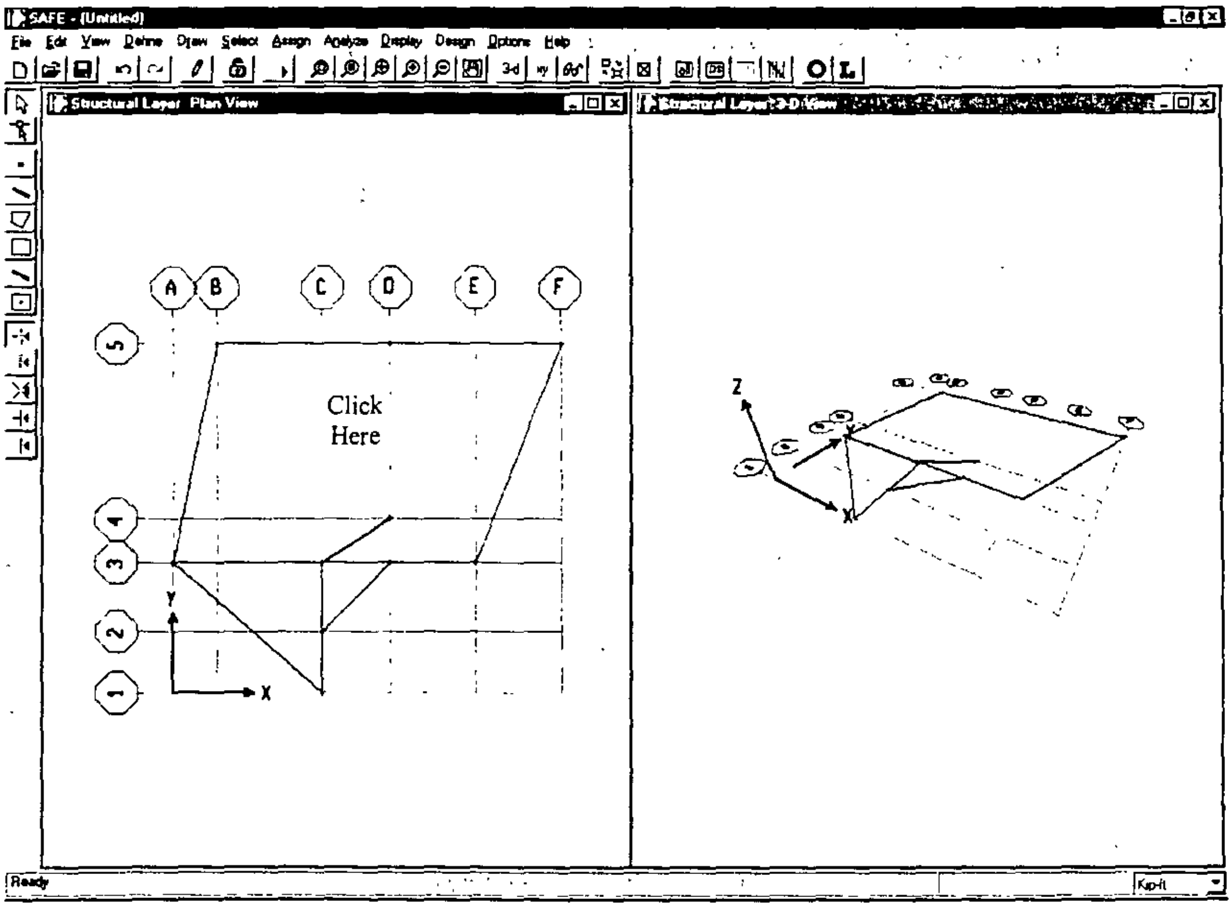


Figure 1-3: Model After Step 13

14. Now we will draw the slab opening between column lines C and D. Click the **Quick Draw Rectangular Area Object** button on the side toolbar (or select **Quick Draw Rectangular Object** from the **Draw** menu).

15. Click once in the area between column lines C, D, 4 and 5 that is labeled "Click Here" in Figure 1-3. A rectangular area object is created covering the area bounded by column lines C, D, 4 and 5.

16. Click the **Pointer/Select** button to switch from Draw mode to Select mode. The model now appears as shown in Figure 1-4.

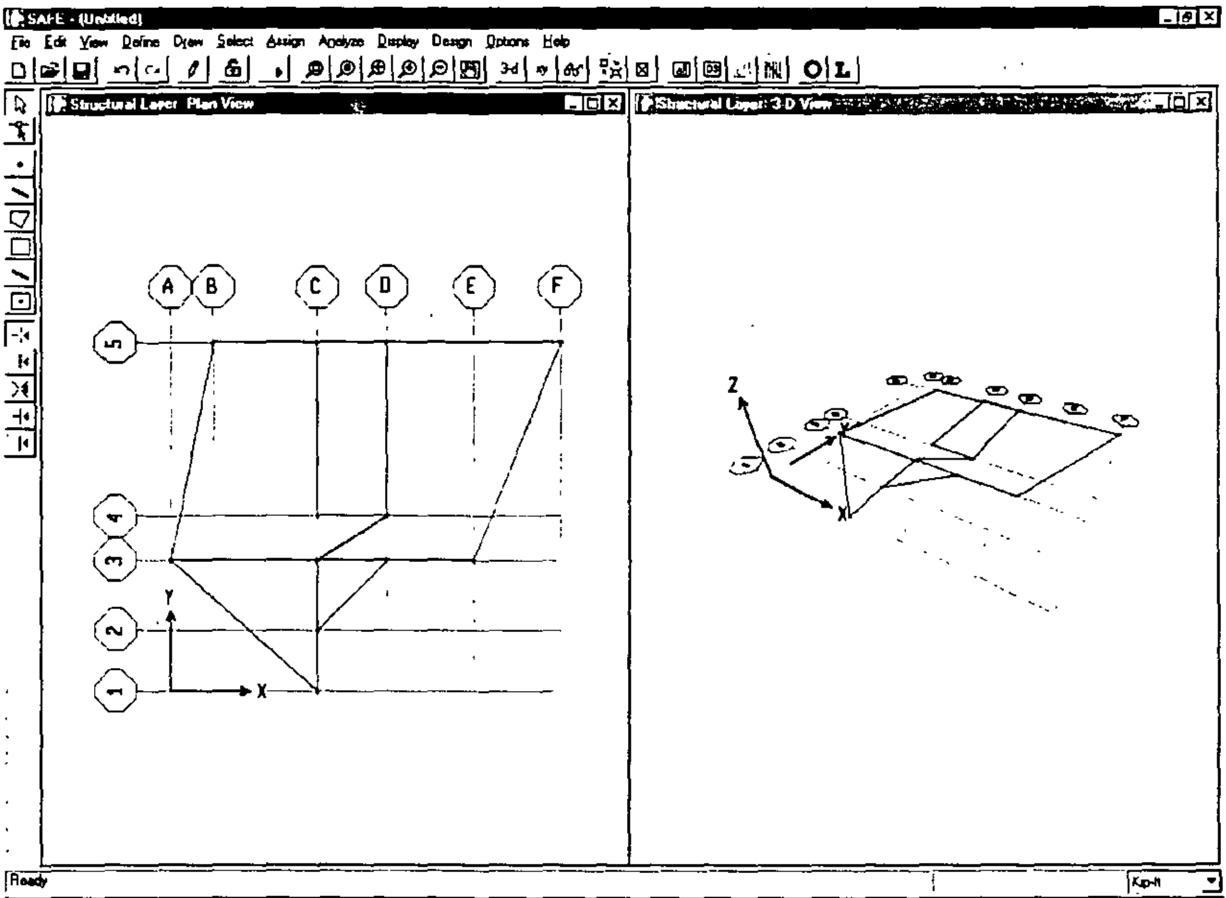


Figure 1-4: Model After Step 16

17. Right click once in the area object just drawn using the Quick Draw feature to display the Rectangular Slab Information dialog box.

18. In this dialog box:

- In the Locate Slab area verify that the By Edges option is selected.

- In the Identification and Location area type **17** in the Xmin edit box.

- In the Identification and Location area type **23** in the Xmax edit box.
- In the Identification and Location area type **28** in the Ymin edit box.
- In the Identification and Location area type **32** in the Ymax edit box.
- Click the **OK** button. The area object is resized.

Note: Alternatively we could have created additional grid lines and used these grid lines to define this area object

19. Left click the resized area object to select it.

20. From the Assign menu select Opening. This area is now designated as an opening and the model appears as shown in Figure 1-5.

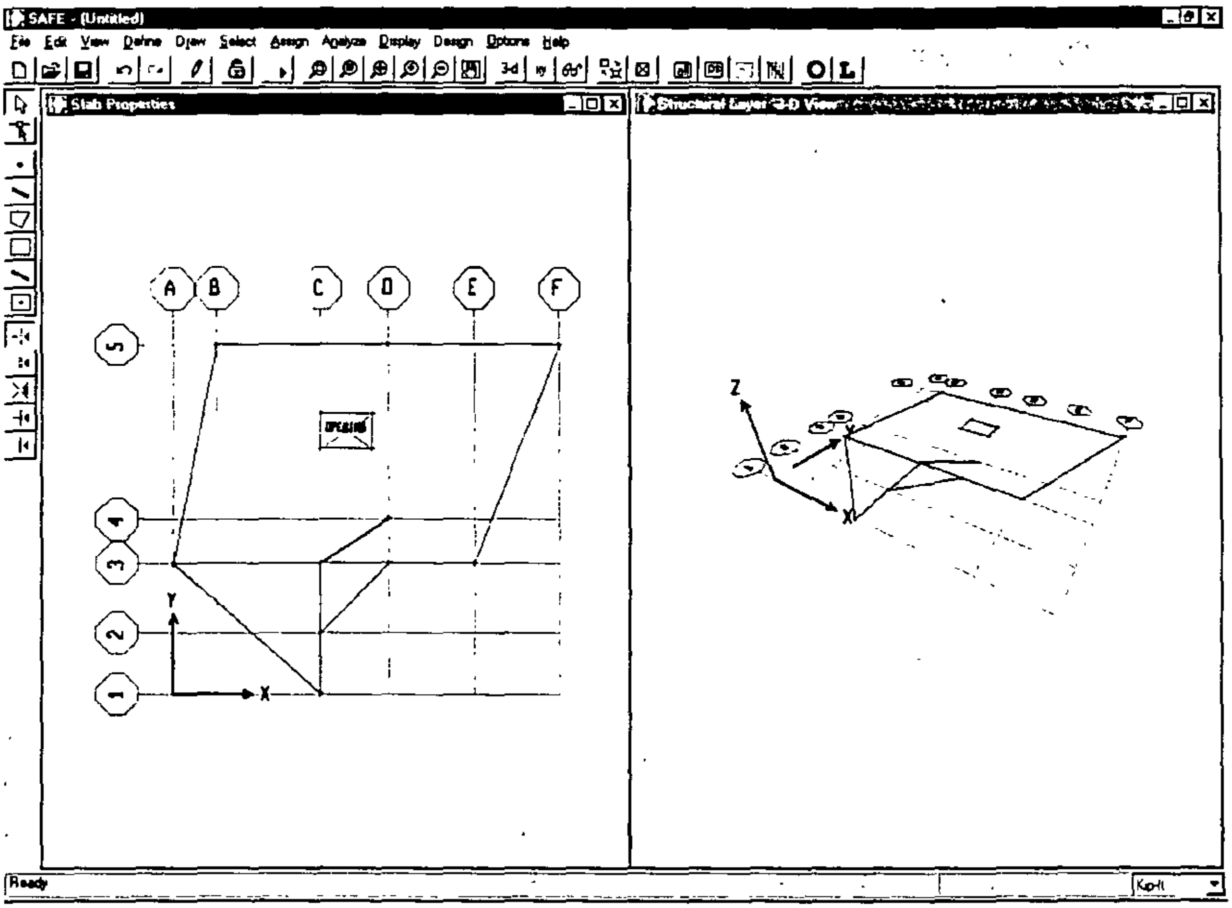


Figure 1-5: Model After Step 20

21. Now we will draw the slab opening between column lines B and C. We will use secondary grid lines to assist us in drawing the opening.

22. From the **Edit** menu select **Edit Grid...** to display the Define X Grid dialog box.

23. In this dialog box:

- In the Display area verify that the X Grid option is selected.
- In the Display Grid As area verify that the Ordinates option is selected.
- Click in the 7th row of the spreadsheet.
- In the Grid Line #7 area select the Secondary Line option.

  Note: Secondary grid lines do not have Grid ID bubbles. We will not specify a Grid ID for these secondary lines.

- Type **9** in the Coordinate column of the 7th row of the spreadsheet.
- In the Display area select the Y Grid option.
- Click in the 6th row of the spreadsheet.
- Verify that the Secondary line option is selected in the Grid Line #6 area.
- Type **24** in the Coordinate column of the 6th row of the spreadsheet.
- Click the **OK** button. The model now appears as shown in Figure 1-6.

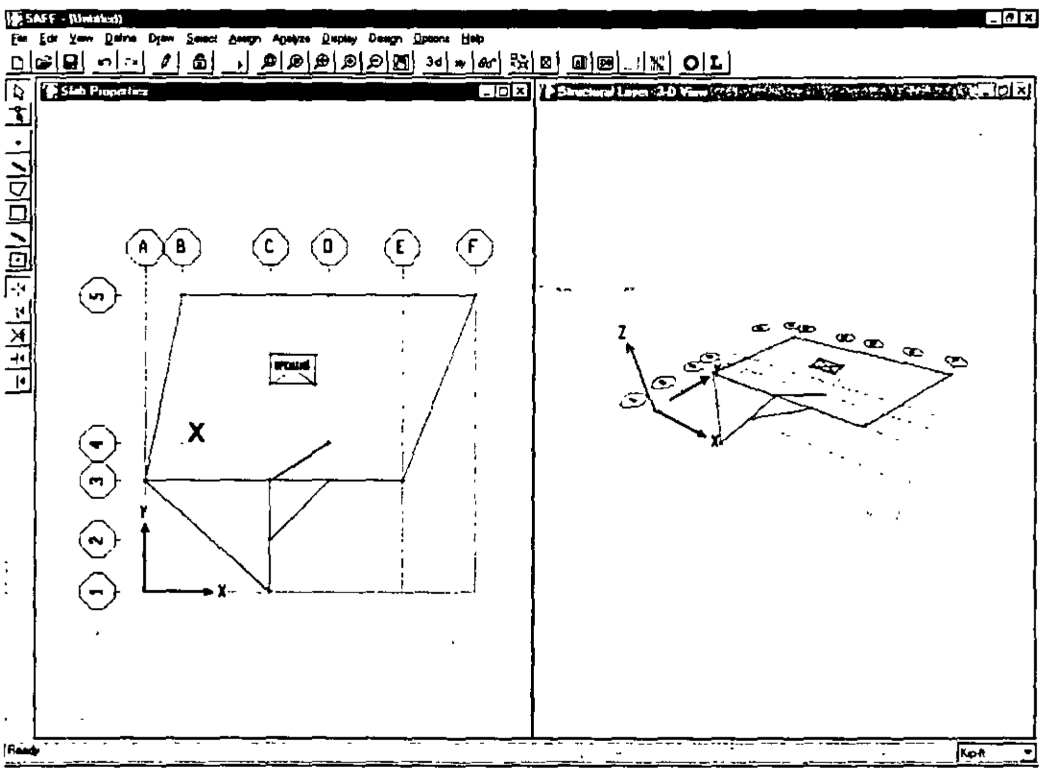


Figure 1-6: Model After Step 23

24. Click the **Quick Draw Rectangular Area Object** button on the side toolbar (or select **Quick Draw Rectangular Object** from the **Draw** menu).

25. Click in the area designated "X" in Figure 1-6 to draw the area object.

26. Click the **Pointer/Select** button to switch from Draw mode to Select mode.

27. Click the new area object to select it.

28. From the Assign menu select Opening. This area is now designated as an opening.

29. Click the drop down box in the status bar to change the units to kip-in. Kip-in

30. Click on the point objects at B-5, D-5, F-5, E-3, C-1 and A-3 to select them. When you have selected all six point objects look at the left-hand side of the status bar at the bottom of the SAFE window and verify that it says "6 Joints selected".

31. From the **Assign** menu select **Column Supports...** to display the Support Properties dialog box.

32. In this dialog box:

    - Highlight COL1 in the Support Props area.

    - Click the **Modify/Show Property** button to display the Column Support Property Data dialog box.

    - In this dialog box:

        - In the Define Column By area verify that the Rectangular Properties option is selected.

        - In the Properties Below Slab area type **4000** in the Modulus of Elasticity edit box.

        - In the Properties Below Slab verify that Poisson's Ratio is 0.2.

        - In the Properties Below Slab area type **18** in the X Dimension edit box.

        - In the Properties Below Slab area type **18** in the Y Dimension edit box.

        - In the Properties Below Slab area type **144** in the Column Height edit box.

        - Click the **OK** button twice to exit all dialog boxes.

33. Select the line object between grid intersections C-3 and D-4 by clicking on it.

34. From the **Assign** menu select **Wall Supports...** to display the Support Properties dialog box.

35. In this dialog box:

- Highlight WALL1 in the Support Props area.
- Click the **Modify/Show Property** button to display the Wall Support Property Data dialog box.
- In this dialog box:
  - ➢ In the Define Wall By area verify that the Dimensions option is selected.
  - ➢ In the Properties Below Slab area type **4000** in the Modulus of Elasticity edit box.
  - ➢ In the Properties Below Slab verify that Poisson's Ratio is 0.2.
  - ➢ In the Properties Below Slab verify that the Thickness is 12.
  - ➢ In the Properties Below Slab area type **144** in the Column Height edit box.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

36. Select the three area objects that define the slab by clicking in each of them. Do not select the area objects that define the slab openings.

37. From the **Assign** menu select **Slab Properties...** to display the Slab Properties dialog box.

38. In this dialog box:

- In the Slab Property area highlight the property named SLAB1 and the click the **Modify/Show Property** button to display the Slab Property Data dialog box.
- In this dialog box:
  - ➢ Accept the default Property Name, SLAB1.
  - ➢ In the Analysis Property Data area:
    - ✓ Type **4000** in the Modulus of Elasticity edit box.
    - ✓ Verify Poisson's ratio is 0.2.

      Note: We will verify the unit weight later.
    - ✓ Type **10** in the Bending Thickness (X) edit box.
    - ✓ Type **10** in the Bending Thickness (Y) edit box.
    - ✓ Type **10** in the Twisting Thickness edit box.

✓ Verify that the Slab Type option is selected.

✓ Verify that the Thick Plate check box is checked.

Note: Checking the Thick Plate check box tells the program to consider slab shear deformations in the analysis. If this box is not checked then slab shear deformations are not included in the analysis.

➢ In the Design Property Data area:

✓ Type **10** in the Thickness edit box.

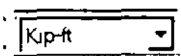
✓ Accept the X Cover Top, Y Cover Top, X Cover Bottom and Y Cover Bottom default values.

✓ Verify the Concrete Strength, f'c is 4.

✓ Verify the Reinforcing Yield Stress, fy is 60.

✓ Verify that the No Design check box is <u>not</u> checked.

➢ Click the **OK** button twice to assign this slab property.

39. Click the drop down box in the status bar to change the units to kip-ft. 

40. From the **Define** menu select Slab Properties... to display the Slab Properties dialog box.

41. In this dialog box:

- In the Slab Property area highlight the property named SLAB1 and the click the **Modify/Show Property** button to display the Slab Property Data dialog box.

- In this dialog box:

  ➢ Verify that the unit weight is 0.15.

  ➢ Click the **OK** button twice to exit all dialog boxes.

42. From the **Define** menu select **Static Load Cases...** to display the Static Load Case Names dialog box.

43. In this dialog box:

- Type **DL** in the Load edit box.

- Type **3** in the Long Term Deflection Multiplier edit box, if it is not already entered.

Note: The multiplier on long term deflections is only active for cracked analysis, it is not active for normal analysis. The longterm deflection multiplier is intended to *account for the effects of creep and shrinkage. A longterm deflection multiplier of 1 essentially means that the longterm deflection is equal to the immediate deflection, that is, there is no additional deflection due to creep and shrinkage for that load case. A longterm multiplier of three means that the longterm deflection is equal to three times the immediate deflection, that is, the additional deflection due to creep and shrinkage is equal to twice the immediate deflection for that load case.*

- Click the **Modify Load** button.
- Type **LL** in the Load edit box.
- Select Live from the Type drop-down box.
- Type **0** in the Self Weight Multiplier edit box.
- Type **1** in the Long Term Deflection Multiplier edit box.
- Click the **Add New Load** button.
- Click the **OK** button.

44. Click the Restore Previous Selection button PS on the main toolbar to again select the three area objects that define the slab (not including the area objects that define the slab openings).

45. From the **Assign** menu select **Surface Loads...** to display the Surface Loads dialog box..

46. In this dialog box:

    - Verify that the DL load case is selected in the Load Case Name drop-down box.
    - Type **.05** in the Load per Area edit box.

      Note: Since the current units are kips and feet, 0.05 is equivalent to 50 psf.

      Note: In SAFE, positive vertical loads act downward.

    - Click the **OK** button to assign the uniform slab load.

47. Click the Restore Previous Selection button PS on the main toolbar to again select the three area objects that define the slab (not including the area objects that define the slab openings).

48. From the **Assign** menu select **Surface Loads...** to display the Surface Loads dialog box.

49. In this dialog box:

    - Select the LL load case from the Load Case Name drop-down box.

- Type **.1** (equivalent to 100 psf) in the Load per Area edit box.
- Click the **OK** button to assign the uniform slab load.

50. Click the **Show Undeformed Shape** button to remove the displayed surface loads.
51. Click the **Draw Quadrilateral Area Object** button on the side toolbar (or select **Draw Quadrilateral Area Object** from the **Draw** menu).
52. Draw an area object by single clicking on grid intersections E-5 and F-5 and then double clicking on grid intersection E-3.
53. Click the **Pointer/Select** button to switch from Draw mode to Select mode.
54. Click on the just drawn area object to select it.
55. From the **Assign** menu select **Surface Loads...** to display the Surface Loads dialog box.
56. In this dialog box:
    - Select the LL load case from the Load Case Name drop-down box.
    - Type **.15** (equivalent to 150 psf) in the Load per Area edit box.

      Note: This area load will be added to the area load of 0.1 ksf already assigned to give a total load of 0.25 ksf (250 psf).
    - Click the **OK** button to assign the uniform slab load.
57. Click the **Show Undeformed Shape** button to remove the displayed surface loads.
58. Click on the two points at the corners of the opening between column lines C and D where the point loads are to be assigned (corners nearest column line D) to select them.
59. From the **Assign** menu select **Point Loads...** to display the Point Loads dialog box.
60. In this dialog box:
    - Verify that the DL load case is selected in the Load Case Name drop-down box.
    - In the Loads area type **4** in the Vertical Load edit box.
    - Click the **OK** button to assign the point loads.
61. Click the **3D View** button 3-d on the main toolbar to get a better view of the applied point loads.
62. Click the **2D View** button xy on the main toolbar to return to a plan view.

63. Click the Restore Previous Selection button on the main toolbar to again select the two joints again.

64. From the **Assign** menu select **Point Loads...** to display the Point Loads dialog box.

65. In this dialog box:

    - Select the LL load case from the Load Case Name drop-down box.
    - In the Loads area type **3** in the Vertical Load edit box.
    - Click the **OK** button to assign the point loads.

66. Click the **Perspective Toggle** button on the main toolbar to get a better view of the applied point loads.

67. Click the **Perspective Toggle** button on the main toolbar to return to a plan view.

68. Click the **Show Undeformed Shape** button to remove the displayed point loads.

69. From the **Define** menu select **Load Combinations...** to display the Load Combinations dialog box.

70. In this dialog box:

    - Click the **Add New Combo** button to display the Load Combination Data dialog box.
    - In this dialog box:
        - ➢ Type **DISPL** (short for displacement) in the Load Combination Name edit box.
        - ➢ Type **DL + LL For Displacements** in the Title edit box.
        - ➢ In the Case Name drop-down box verify the DL Load Case is selected.
        - ➢ Verify that 1 is entered in the Scale Factor edit box.
        - ➢ Click the **Add** button.
        - ➢ Select LL Load Case from the Case Name drop-down box.
        - ➢ Click the **Add** button.
        - ➢ Click the **OK** button twice to exit all dialog boxes.

71. From the **Analyze** menu select **Set Options** to display the Analysis Options dialog box.

72. In this dialog box:

- In the Analysis Type area select the Normal and Cracked Deflections option.

- Accept the default maximum mesh dimension of 4 feet.

  Note: To automatically mesh the model the program first develops a mesh that is broken at each grid line, at the edge of any area objects, at the ends of each line object and at each point object. If the dimension of any elements meshed in this manner exceeds the specified maximum mesh dimension then those elements are subdivided such that the maximum mesh dimension is no longer exceeded.

  Note: In general, the maximum mesh size should be based on the span length. It is good practice to have at least four elements in a span, and generally it is not necessary to have more than eight elements in a span. Thus if the typical span length is less than sixteen feet, it may be advisable to reduce the default maximum mesh size to something less than 4 feet.

  *Note: The mesh automatically generated by the program can be viewed by clicking the **Set Objects** button ⊠ on the main toolbar (or selecting **Set Object Options...** from the **View** menu) to display the Set Objects dialog box, checking the Show Mesh check box in the Options area, and clicking the **OK** button. You can turn off the display of the mesh by unchecking this box.*

- Click the **OK** button.

73. Click the **Run Analysis** button on the main toolbar (or select **Run Analysis** from the **Analyze** menu).

74. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

75. Click in the plan view window to make sure it is active.

76. Click the **Show Static Deformed Shape** button on the main toolbar (or select **Show Deformed Shape...** from the **Display** menu) to display the Deformed Shape dialog box.

77. In this dialog box:

    - Select DISPL Combo from the Load drop-down box.

    - In the Display Options area verify that the Displacement Contours check box is checked.

    - In the Display Options area select the Elastic option.

    - Click the **OK** button to display the deformed shape.

78. Move the mouse cursor over the slab elements and read the displacement values on the left-hand side of the status bar at the bottom of the SAFE window.

79. Click the **Show Static Deformed Shape** button on the main toolbar (or select **Show Deformed Shape** from the **Display** menu) to display the Deformed Shape dialog box.

80. In this dialog box:

   - Select DISPL Combo from the Load drop-down box if it is not already selected.
   - In the Display Options area verify that the Displacement Contours check box is checked.
   - In the Display Options area select the Long Term Cracked option.
   - Click the **OK** button to display the deformed shape.

81. Move the mouse cursor over the slab elements and read the displacement values on the left-hand side of the status bar at the bottom of the SAFE window.

   Note: Cracked long term deflections could be compared with elastic deflections in side-by-side windows.

82. From the **Options** menu select **Preferences...** and then select the Concrete Tab.

83. In this dialog box:

   - Verify that ACI 318-95 is selected in the Concrete Design Code drop-down box.
   - Verify that in the Reinforcement Reults Units area the Sq-in and Sq-in/ft option is selected.
   - Click the **OK** button.

84. On the **Design** menu choose **Select Design Combos** to display the Design Load Combinations Selection dialog box.

85. In this dialog box:

   - In the Design Combos list box click on DCON2 to highlight it and then click the **Show** button.
   - Note that this load combination is defined as 1.4DL + 1.7LL.

     *Note: SAFE automatically creates design load combinations based on the selected design code and on the static load types. In this case where the defined static load types are DEAD and LIVE (see **Static Load Cases** on the **Define** menu) and the design code is ACI 318-95, the program automatically creates two load cases. They are 1.4DL (DCON1) and 1.4DL + 1.7LL (DCON2).*

   - Click the **OK** button twice to exit all dialog boxes.

86. On the **Design** menu select **Start Design**.

87. When the design is complete the calculated X-strip reinforcement is displayed in square inches for each of the design strips. Note that you can run your cursor over this reinforcement and the required reinforcement is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

    Note: If you do not specify at least one slab design strip then the program will automatically create a slab design strip for each row/column of the mesh. Since we did not create any slab design strips in this example, the program automatically created the design strips shown now and they correspond to the mesh.

88. From the **Design** menu select **Display Slab Design Info...** to display the Slab Reinforcing dialog box.

89. In this dialog box:

    - In the Choose Strip Direction area verify that the X Direction Strip option is chosen.
    - In the Reinforcing Display Type select the Show Number of Bars option and then select #6 from the associated drop-down box.
    - Click the **OK** button.

90. The calculated X-strip reinforcement is now displayed showing the required number of #6 bars for each of the design strips. Note that you can run your cursor over this reinforcement and the required reinforcement in square inches is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

91. Click the **Show Slab Forces** button S on the main toolbar (or select **Show Slab Forces...** from the **Display** menu) to display the Slab Forces dialog box.

92. In this dialog box:

    - Select DISPL Combo from the Load drop-down box.
    - Verify that the Mxx option is selected in the component area.
    - Click the **OK** button.

93. The X-direction moments are now displayed. Note that you can run your cursor over the moment contours and the moment at the cursor location is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 WEB TUTORIAL 2
SAFE PROBLEM 2

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels· 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

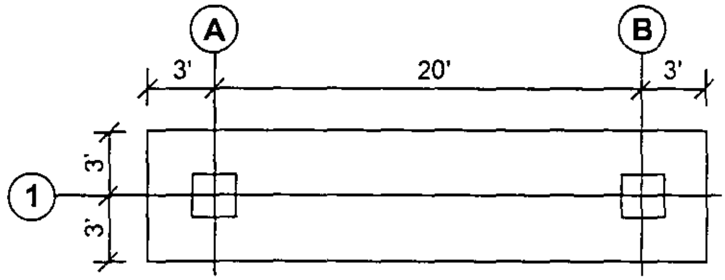
SAFE Problem 2

Combined Footing

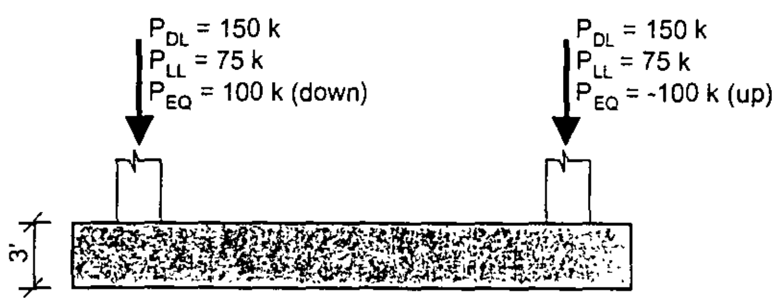
Concrete Footing Properties
E = 3600 ksi, Poissons Ratio = 0.2, Unit Weight = 150 pcf
f'c = 4 ksi, fy = 60 ksi
X direction top rebar cover = 3" (to rebar centroid)
Y direction top rebar cover = 2" (to rebar centroid)
X direction bottom rebar cover = 3.5" (to rebar centroid)
Y direction bottom rebar cover = 4.5" (to rebar centroid)
Concrete columns are 18" by 18".

Soil Properties
Subgrade Modulus = 200 kcf

To Do
Use the ACI 318-95 code. Run the analysis using the uplift iteration option. Review the soil pressure distribution. Review the elastic vertical footing displacements under unfactored DL + LL + EQ loading. Review the factored footing moments. Review the calculated footing reinforcing.



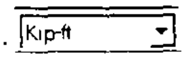
PLAN



ELEVATION

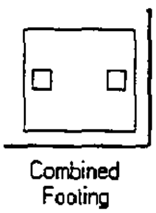
Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

SAFE Problem 2 Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Slab Templates dialog box.

3. In this dialog box:

   - Click on the **Combined Footing** button 

 to display the Combined Footing dialog box.

   - In this dialog box:

     - Type **3** in the Left Edge Distance, Right Edge Distance, Top Edge Distance and Bottom Edge Distance edit boxes (4 different edit boxes total).

     - In the Spacing area verify that the X option is selected.

     - Type **20** in the Spacing edit box.

     - For Load 1 type **150** in the P_{DEAD} edit box and type **75** in the P_{LIVE} edit box.

       Note: When the combined footing is defined in the X-direction (i.e., the X option is selected in the Spacing area) then Load 1 is the left column loading and Load 2 is the right column loading. When the combined footing is defined in the Y-direction then Load 1 is the bottom column loading and Load 2 is the top column loading.

     - For Load 2 type **150** in the P_{DEAD} edit box and type **75** in the P_{LIVE} edit box.

     - Type **3** in the Footing Thickness edit box.

     - Type **200** in the Soil Modulus edit box.

     - Type **1.5** in the Load Size (Square) edit box. Since the units are kips and feet the 1.5 is equivalent to 18 inches. The dialog box now appears as shown in Figure 2.1.

     - Click the **OK** button.

4. From the **Define** menu select **Static Load Cases...** to display the Static Load Case Names dialog box.

5. In this dialog box:

   - Type **EQ** in the Load edit box.

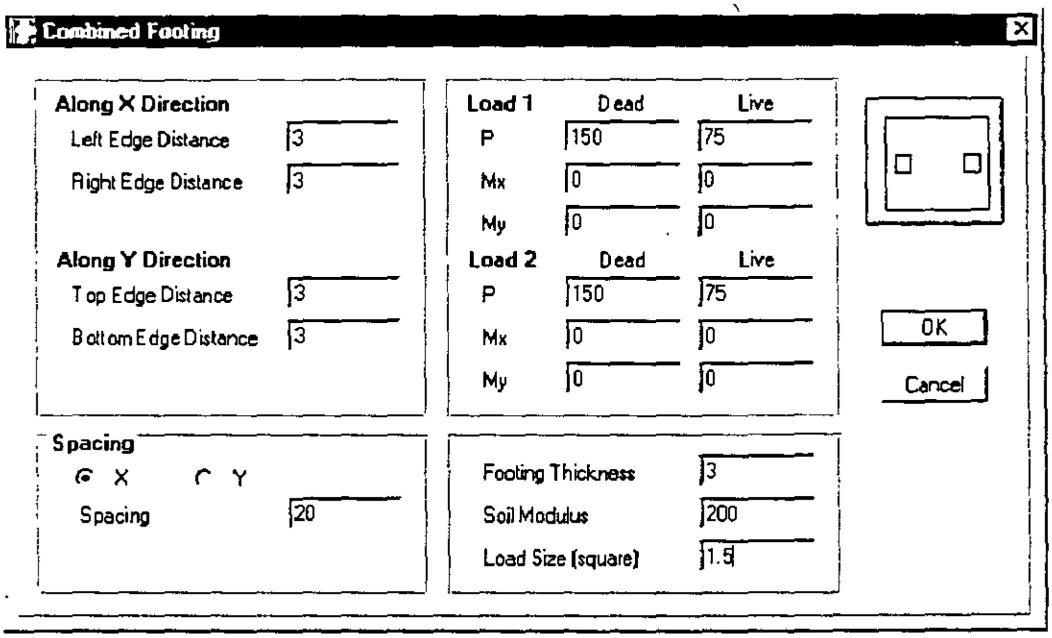


Figure 2-1: Combined Footing Dialog Box

- Select QUAKE from the Type drop-down box.
- Type **0** in the Self Weight Multiplier edit box.
- Type **1** in the Long Term Deflection Multiplier edit box.
- Click the **Add New Load** button.
- Click the **OK** button.

6. From the **Define** menu select **Load Combinations...** to display the Load Combinations dialog box.

7. In this dialog box:

- Click the **Add New Combo** button to display the Load Combination Data dialog box.
- In this dialog box:
  - ➢ Type **DISPL** (short for displacement) in the Load Combination Name edit box.
  - ➢ Type **DL + LL + EQ For Displacements** in the Title edit box.
  - ➢ In the Case Name drop-down box verify the DL Load Case is selected.
  - ➢ Verify that 1 is entered in the Scale Factor edit box.
  - ➢ Click the **Add** button.

> Select LL Load Case from the Case Name drop-down box.

> Click the **Add** button.

> Select EQ Load Case from the Case Name drop-down box.

> Click the **Add** button.

> Click the **OK** button twice to exit all dialog boxes.

8. Click on the point object (joint element) at grid intersection A-1 to select it. When the joint is selected look at the left-hand side of the status bar at the bottom of the SAFE window and verify that it says "1 Joints selected".

9. From the **Assign** menu select **Point Loads...** to display the Point Loads dialog box.

10. In this dialog box:

    - Select the EQ load case from the Load Case Name drop-down box.

    - In the Loads area type **100** in the Vertical Load edit box.

      Note: In SAFE, positive vertical loads act downward.

    - In the Size of Load area type **1.5** in both the X Dimension and Y Dimension edit boxes.

    - Click the **OK** button to assign this point load.

11. Click on the point object (joint element) at grid intersection B-1 to select it.

12. From the **Assign** menu select **Point Loads...** to display the Point Loads dialog box.

13. In this dialog box:

    - Verify that the EQ load case is selected in the Load Case Name drop-down box.

    - In the Loads area type **-100** in the Vertical Load edit box. The minus sign is used because this earthquake load is acting up.

    - In the Size of Load area verify that both the X Dimension and Y Dimension are 1.5.

    - Click the **OK** button to assign this point load.

14. Click the **3D View** button 3-d on the main toolbar to get a better view of the applied point loads.

15. Click the **2D View** button xy on the main toolbar to return to a plan view.

16. Click the **Show Undeformed Shape** button to remove the displayed point loads.

17. Right click on the footing to display the Rectangular Slab Information dialog box.

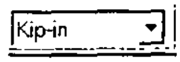
18. In this dialog box:

- In the Specifications area note that the Slab Property is called FOOTING. This slab property was created and applied by the combined footing template.

- In the Specifications area note that the Support is SOIL. This support property was created and applied by the combined footing template.

- We will check these properties and if necessary modify them to meet our own requirements in the following steps.

- Click the **Cancel** button to close the dialog box.

19. Click the drop down box in the status bar to change the units to kip-in. 

20. From the **Define** menu select **Slab Properties...** to display the Slab Properties dialog box.

21. In this dialog box:

- In the Slab Property area highlight the property named FOOTING and then click the **Modify/Show Property** button to display the Slab Property Data dialog box.

- In this dialog box:

  ➤ In the Analysis Property Data area:

    ✓ Verify that the Modulus of Elasticity is 3600.

    ✓ Verify Poisson's ratio is 0.2.

    Note: We will verify the unit weight later.

    ✓ Verify that the Bending Thickness (X), Bending Thickness (Y) and Twisting Thickness are all 36.

    Note: These values will all be 36 because we entered the footing thickness in the Combined Footing dialog box in Step 3.

    ✓ Verify that the Slab Type option is selected.

    ✓ Verify that the Thick Plate check box is checked.

  Note: Checking the Thick Plate check box tells the program to consider slab shear deformations in the analysis. If this box is not checked then slab shear deformations are not included in the analysis.

  ➤ In the Design Property Data area:

✓ Verify that the Thickness is 36.

✓ Type **3** in the X Cover Top (to Centroid) edit box.

✓ Type **2** in the Y Cover Top (to Centroid) edit box.

✓ Type **3.5** in the X Cover Bottom (to Centroid) edit box.

✓ Type **4.5** in the Y Cover Bottom (to Centroid) edit box.

✓ Verify the Concrete Strength, f'c is 4.

✓ Verify the Reinforcing Yield Stress, fy is 60.

✓ Verify that the No Design check box is <u>not</u> checked.

➢ Click the **OK** button twice.

22. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

23. From the **Define** menu select Slab Properties... to display the Slab Properties dialog box.

24. In this dialog box:

- In the Slab Property area highlight the property named FOOTING and then click the **Modify/Show Property** button to display the Slab Property Data dialog box.

- In this dialog box:

  ➢ Verify that the unit weight is 0.15.

  ➢ Click the **OK** button twice to exit all dialog boxes.

25. From the **Define** menu select Soil Supports... to display the Support Properties dialog box.

26. In this dialog box:

- In the Support Props area highlight the property named SOIL and then click the **Modify/Show Property** button to display the Soil Support Property Data dialog box.

- In this dialog box:

  ➢ Verify that the subgrade modulus is 200.

  Note: This value will be 200 because we entered the soil subgrade modulus in the Combined Footing dialog box in Step 3.

  ➢ Click the **OK** button twice to exit all dialog boxes.

27. From the **Options** menu select **Preferences...** and then select the Concrete Tab.

28. In this dialog box:

- Verify that ACI 318-95 is selected in the Concrete Design Code drop-down box.
- Verify that in the Reinforcement Results Units area the Sq-in and Sq-in/ft option is selected.
- Click the **OK** button.

29. On the **Design** menu choose **Select Design Combos** to display the Design Load Combinations Selection dialog box.

30. In this dialog box:

- In the Design Combos list box click on DCON2 to highlight it and then click the **Show** button.
- Note that this load combination is defined as 1.4DL + 1.7LL. Also note that the Use For Design check box is checked.

  Note: SAFE automatically creates design load combinations based on the selected design code and on the static load types. In this case where the defined static load types are DEAD and LIVE and QUAKE (see ***Static Load Cases*** *on the* ***Define*** *menu) and the design code is ACI 318-95, the program automatically creates six load cases. They are 1.4DL (DCON1) and 1.4DL + 1.7LL (DCON2), 1.05DL + 1.275LL + 1.4025EQ (DCON3), 1.05DL + 1.275LL - 1.4025EQ (DCON4), 0.9DL + 1.43 EQ (DCON5), and 0.9DL - 1.43 EQ (DCON6).*

  If we wanted to add an additional design load combination we could define it by selecting ***Load Combinations...*** *from the* ***Define*** *menu and then clicking the* ***Add New Combo*** *button, to display the Load Combination data dialog box. In this dialog box we can define the load combination and we can check the Use For Design check box if we want the load combination to be used in design.*

- Click the **OK** button twice to exit all dialog boxes.

31. From the **Analyze** menu select **Set Options** to display the Analysis Options dialog box.

32. In this dialog box:

- In the Analysis Type area select the Iterative For Uplift option.
- Accept the default values in the Uplift Iteration Parameters area.

  Note: The default Convergence Tolerance, 0.001, should provide sufficient accuracy for most problems. The default Maximum Number of Iterations, 10, may not *provide sufficient accuracy for many problems. Additional iterations may be required. These terms are defined in Step 35.*

- Accept the default maximum mesh dimension of 4 feet.

  Note: To automatically mesh the model the program first develops a mesh that is broken at each grid line, at the edge of any area objects, at the ends of each line object and at each point object. If the dimension of any elements meshed in this manner exceeds the specified maximum mesh dimension then those elements are subdivided such that the maximum mesh dimension is no longer exceeded.

  Note: In general, the maximum mesh size should be based on the span length. It is good practice to have at least four elements in a span, and generally it is not necessary to have more than eight elements in a span. Thus if the typical span length is less than sixteen feet, it may be advisable to reduce the default maximum mesh size to something less than 4 feet.

  Note: The mesh automatically generated by the program can be viewed by clicking the ***Set Objects*** *button ⊠ on the main toolbar (or selecting* ***Set Object Options...*** *from the* ***View*** *menu) to display the Set Objects dialog box, checking the Show Mesh check box in the Options area, and clicking the* ***OK*** *button. You can turn off the display of the mesh by unchecking this box.*

- Click the **OK** button.

33. Click the **Run Analysis** button on the main toolbar (or select **Run Analysis** from the **Analyze** menu).

34. When the initial analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window. The program will then proceed to do the iterations for uplift.

35. When the iterations for uplift are complete an Uplift Analysis Status window similar to that shown in Figure 2-2 will appear giving information on how well the uplift iterations converged.

Uplift Analysis Status

Combo	ConErr	ConTol	Iterations	MaxIters
DISPL	0.0000	0.0010	1	10
DCON1	0.0000	0.0010	1	10
DCON2	0.0000	0.0010	1	10
DCON3	0.0000	0.0010	1	10
DCON4	0.0000	0.0010	1	10
DCON5	0.0670	0.0010	10	10
DCON6	0.0670	0.0010	10	10

Figure 2-2: Uplift Analysis Status Using a Maximum of 10 Iterations

*Note: There are five columns in the Uplift Analysis Status window labeled Combo, ConErr, ConTol, Iterations and MaxIters respectively. The data in this window is to be read in rows. The Combo column identifies the load combination. The ConErr column is the convergence error at the end of the uplift iterations. The convergence error is defined as the maximum tensile stress divided by the average stress where the average stress is P_{TOTAL} / A_{TOTAL}. The ConTol column is the user-defined convergence tolerance, in other words, the allowable or acceptable convergence error. This value is defined by the Convergence Tolerance item input in the Uplift Iteration Parameters area in the Analysis Options dialog box (**Analyze** menu > **Set Options**). The MaxIters column is the user-defined maximum number of iterations performed before the uplift iteration is stopped, even though it has not converged. This value is defined by the Maximum Number of Iterations item input in the Uplift Iteration Parameters area in the Analysis Options dialog box (**Analyze** menu > **Set Options**). The Iterations column tells how many iterations were actually performed for the specified load combination. Uplift analysis iterations are performed until either the convergence tolerance is satisfied, or the specified maximum number of iterations is reached.*

36. The Uplift Analysis Status window indicates that the convergence tolerance has not been satisfied for load combinations DCON5 and DCON6. To see the effect of this nonconvergence we will view the soil pressures.

37. Click the "X" in the upper right-hand corner of the Uplift Analysis Status window to close it.

38. Click the **Show Joint Reactions** button on the main toolbar (or select **Show Reaction Forces...** from the **Display** menu) to display the Joint Forces dialog box.

39. In this dialog box:

    - Select DCON5 Combo from the Load drop-down box.
    - In the Type of Load area select the Soil Pressure option.
    - Click the **OK** button to display the soil pressure diagram.

40. Run the mouse cursor over the soil pressure diagram contour plot and read the values from the left-hand side of the status bar at the bottom of the SAFE window. Note that the minimum value is approximately -0.145 ksf (tension). If the iterations had fully converged, that is the convergence error had reached zero, then this minimum value would be 0 ksf, that is, no tension. We will run the analysis again using more iterations to improve this convergence.

41. Click the Lock/Unlock Model button on the main toolbar to unlock the model.

42. From the **Analyze** menu select **Set Options** to display the Analysis Options dialog box.

43. In this dialog box:

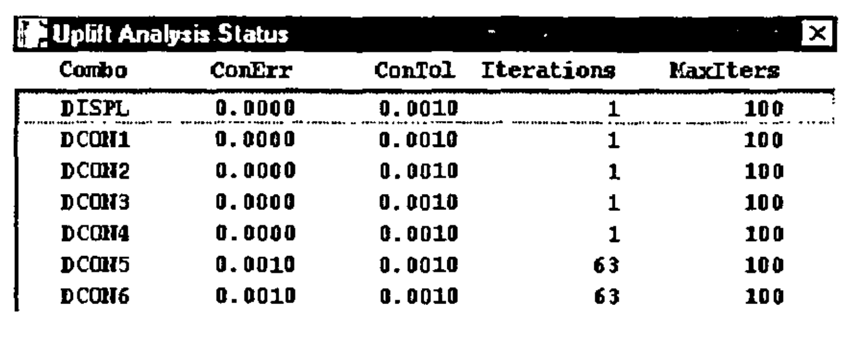
- In the Uplift Iteration Parameters area type **100** in the Maximum Number of Iterations edit box.

- Click the **OK** button.

44. Click the **Run Analysis** button on the main toolbar (or select **Run Analysis** from the **Analyze** menu).

45. When the initial analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window. The program will then proceed to do the iterations for uplift.

46. When the iterations for uplift are complete an Uplift Analysis Status window similar to that shown in Figure 2-3 will appear giving information on how well the uplift iterations converged.



Combo	ConErr	ConTol	Iterations	MaxIters
DISPL	0.0000	0.0010	1	100
DCON1	0.0000	0.0010	1	100
DCON2	0.0000	0.0010	1	100
DCON3	0.0000	0.0010	1	100
DCON4	0.0000	0.0010	1	100
DCON5	0.0010	0.0010	63	100
DCON6	0.0010	0.0010	63	100

Figure 2-2: Uplift Analysis Status Using a Maximum of 100 Iterations

47. Notice that all load combinations are converged, and that it took 63 iterations for load combinations DCON5 and DCON6 to converge.

48. Click the "X" in the upper right-hand corner of the Uplift Analysis Status window to close it.

49. Click the **Show Joint Reactions** button on the main toolbar (or select **Show Reaction Forces...** from the **Display** menu) to display the Joint Forces dialog box.

50. In this dialog box:

    - Select DCON5 Combo from the Load drop-down box.

    - In the Type of Load area select the Soil Pressure option.

    - Click the **OK** button to display the soil pressure diagram.

51. Run the mouse cursor over the soil pressure diagram contour plot and read the values from the left-hand side of the status bar at the bottom of the SAFE window. Note that the

minimum value is approximately -0.002 ksf (tension). This is a very good approximation to zero tension.

52. Click in the plan view window to make sure it is active.

53. Click the **Show Static Deformed Shape** button on the main toolbar (or select **Show Deformed Shape...** from the **Display** menu) to display the Deformed Shape dialog box.

54. In this dialog box:

    - Select DISPL Combo from the Load drop-down box.

    - In the Display Options area verify that the Displacement Contours check box is checked.

    - Click the **OK** button to display the deformed shape.

55. Move the mouse cursor over the slab elements and read the displacement values on the left-hand side of the status bar at the bottom of the SAFE window. Note that these displacements are in feet.

56. Click the drop down box in the status bar to change the units to kip-in. Kip-in

57. Move the mouse cursor over the slab elements and read the displacement values on the left-hand side of the status bar at the bottom of the SAFE window. Note that these displacements are in inches.

58. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

59. Click the **Show Slab Forces** button on the main toolbar (or select **Show Slab Forces...** from the **Display** menu) to display the Slab Forces dialog box.

60. In this dialog box:

    - Select DCON3 Combo from the Load drop-down box.

    - In the Component area select the Mxx option.

    - Click the **OK** button.

61. The X-direction moments are now displayed. Note that you can run your cursor over the moment contours and the moment at the cursor location is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

62. If you want to view slab force results for other force components or for other load combinations then repeat steps 55 through 57.

63. On the **Design** menu select **Start Design**.

64. When the design is complete the calculated X-strip reinforcement is displayed in square inches. Note that you can run your cursor over this reinforcement and the required reinforcement is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

Note: If you do not specify at least one slab design strip then the program will automatically create a slab design strip for each row/column of the mesh. Since we did not create any slab design strips in this example, the program automatically created the design strips shown now and they correspond to the mesh.

65. From the **Design** menu select **Display Slab Design Info...** to display the Slab Reinforcing dialog box.

66. In this dialog box:

- In the Choose Strip Direction area verify that the X Direction Strip option is chosen.

- In the Reinforcing Display Type select the Show Number of Bars option and then select #6 from the associated drop-down box.

- Click the **OK** button.

67. The calculated X-strip reinforcement is now displayed showing the required number of #6 bars. Note that you can run your cursor over this reinforcement and the required reinforcement in square inches is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

68. From the **Design** menu select **Display Slab Design Info...** to display the Slab Reinforcing dialog box.

69. In this dialog box:

- In the Choose Strip Direction select the Y Direction Strip option.

- In the Reinforcing Display Type select the Show Number of Bars option and then select #4 from the associated drop-down box.

- Click the **OK** button.

70. The calculated Y-strip reinforcement is now displayed showing the required number of #4 bars for each of the design strips. Note that you can run your cursor over this reinforcement and the required reinforcement in square inches is displayed on the left-hand side of the status bar at the bottom of the SAFE window.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 WEB TUTORIAL 2
SAFE PROBLEM 3

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521 4021 al 24, 5623 2910 y 5623.2971 • Fax: 5510.0573

SAFE Problem 3

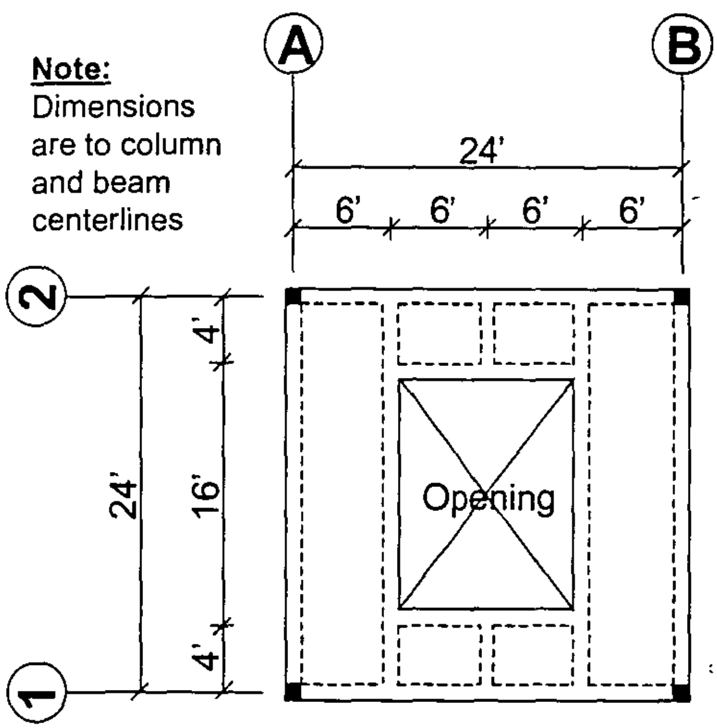
One-Way Beam-Slab System With Opening

Design Information

E = 3600 ksi, Poissons Ratio = 0.2, Unit Weight = 150 pcf
f'c = 4 ksi, fy = 60 ksi
X direction top rebar cover = 2.5" (to rebar centroid)
Y direction top rebar cover = 1.5" (to rebar centroid)
X direction bottom rebar cover = 1.5" (to rebar centroid)
Y direction bottom rebar cover = 2.5" (to rebar centroid)
All columns are 12" by 12".
Column height below is 10 feet.
All beams are 12" wide by 18" deep (12" deep below slab).
Design beams as rectanglar beams (not T-beams).
Slab is 6" thick.
Dead load in addition to self-weight is 35 psf.
Live load is 100 psf.

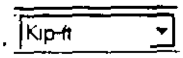
To Do

Use the ACI 318-95 code. Review the factored beam moments. Review the calculated beam flexural reinforcing.



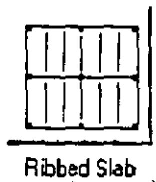
Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

SAFE Problem 3 Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Slab Templates dialog box.

3. In this dialog box:

   - Click on the **Ribbed Slab** button 

 to display the Ribbed Slab dialog box.
   - In this dialog box:

     ➢ In the Along X Direction area verify that 0.5 is entered in the Left Edge Distance and Right Edge Distance edit boxes.

     ➢ In the Along X Direction area type **1** in the Number of Spans edit box.

     ➢ In the Along X Direction area verify that 24 is entered in the Spacing edit box.

     ➢ In the Along Y Direction area verify that 0.5 is entered in the Left Edge Distance and Right Edge Distance edit boxes.

     ➢ In the Along Y Direction area type **1** in the Number of Spans edit box.

     ➢ In the Along Y Direction area verify that 24 is entered in the Spacing edit box.

     ➢ Type **.5** in the Slab Thickness edit box.

     ➢ Verify that 1 is entered in the Column Size (square) edit box.

     ➢ Type **10** in the Column Height Below edit box.

     ➢ In the Load area type **.035** in the Dead Load (Additional) edit box. Since the units are kips and feet, .035 is equivalent to 35 psf.

     ➢ In the Load area type **.1** in the Live Load edit box. Since the units are kips and feet, .1 is equivalent to 100 psf.

     ➢ Leave the Create Live Load Patterns check box unchecked.

     Note: Checking the Create Live Load Patterns check box causes the program to create all sorts of automatically generated skip loading patterns for the live load. These patterns are automatically added to the defined static load cases and are identified with a Load Type called Pattern Live. In addition, these live load patterns are included in the load combinations automatically created for design.

- In the Joist area type **6** in the Spacing edit box.

- In the Joist area verify that the Y Direction option is selected This means that the joists will be defined parallel to the Y- axis..

  Note: We will define and assign actual beam/joist sizes later.

- Click the **OK** button.

4. From the **Edit** menu select **Edit Grid...** to display the Define X Grid dialog box.

5. In this dialog box:

   - In the Display area select the Y Grid option.

   - In the Display Grid As area verify that the Ordinates option is selected.

   - Click in the 3rd row of the spreadsheet.

   - In the Grid Line #3 area select the Secondary Line option.

     Note: Secondary grid lines do not have Grid ID bubbles. We will not specify a Grid ID for these secondary lines.

   - Type **4** in the Coordinate column of the 3rd row of the spreadsheet.

   - Type **20** in the Coordinate column of the 4th row of the spreadsheet.

   - Click the **OK** button. The model now appears as shown in Figure 3-1.

6. Delete the center joist (beam) by clicking on it to select it and then pressing the Delete key on the keyboard.

7. Verify that the **Snap to Points** button on the side toolbar is depressed. Alternatively, you could click on the **Draw** menu, select **Snap To** and verify there is a check mark next to **Points** on the submenu.

8. Click the **Snap to Intersections** button on the side toolbar. After you click on the button it will remain depressed indicating that this snap option is active. Alternatively, you could click on the **Draw** menu, select **Snap To** and click on Intersections in the submenu to activate this snap option.

   *Note: The snap options buttons act like toggle switches. You can use them to turn snap options on or off. If a snap option button appears depressed then that snap option is active. Clicking any snap option button toggles it between active(depressed) and inactive (not depressed). Similarly, the snap option items on the **Draw** menu act like toggle switches. If a snap option on the Snap To submenu has a check mark next to it* *then that snap option is active. Clicking a snap option on the Snap To submenu toggles the state of that option.*

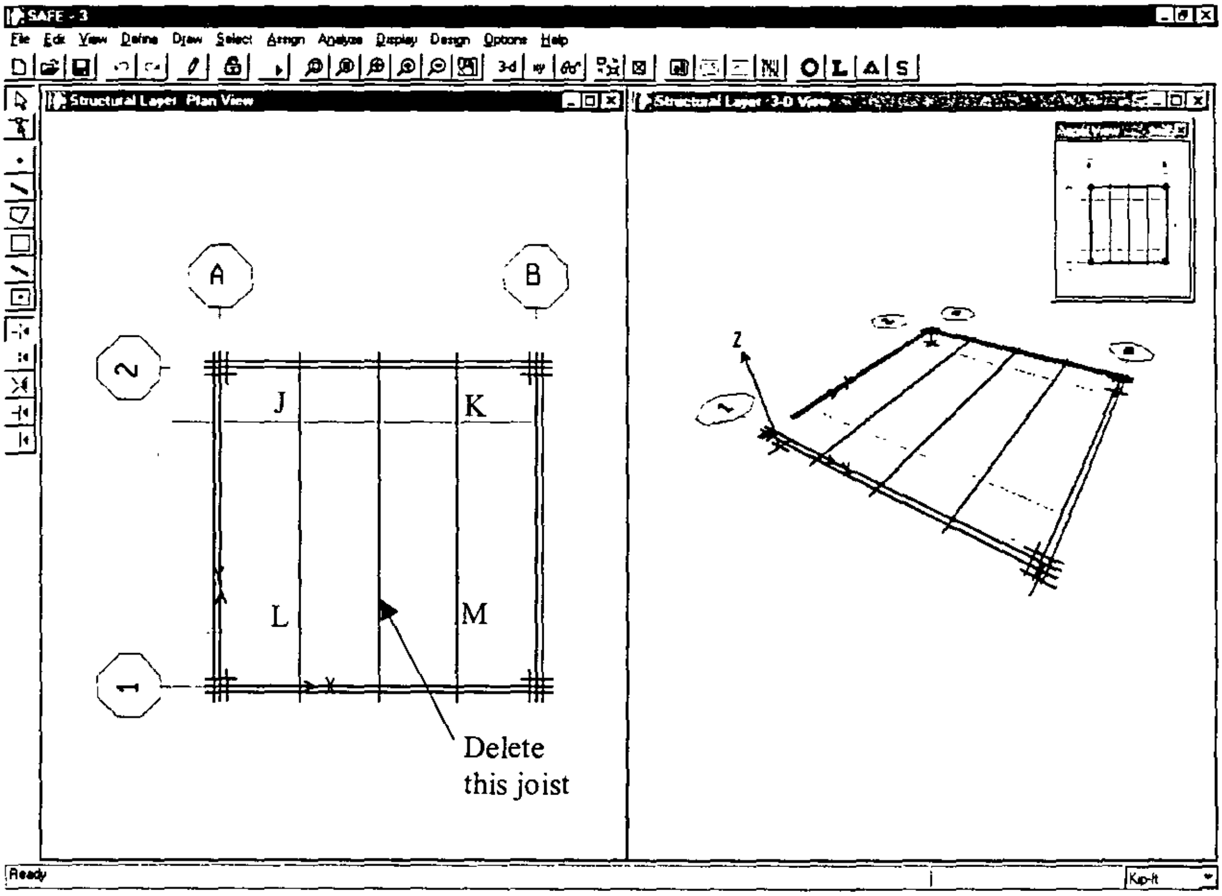


Figure 3-1: Model After Step 5

9. Click the **Draw Line Object** button on the side toolbar (or select **Draw Line Object** from the **Draw** menu).

10. Draw a line object by clicking on the point labeled J in Figure 3-1 and then double clicking on the point labeled K to complete the line object.

    Note: With the Intersection snap option activated the mouse cursor will snap to the intersection of the beam and grid line at points J and K.

    Note: We could also single click on the point labeled K and then press the Enter key on the keyboard to complete the line object.

11. Draw a line object by clicking on the point labeled L in Figure 3-1 and then double clicking on the point labeled M to complete the line object. The model now appears as shown in Figure 3-2.

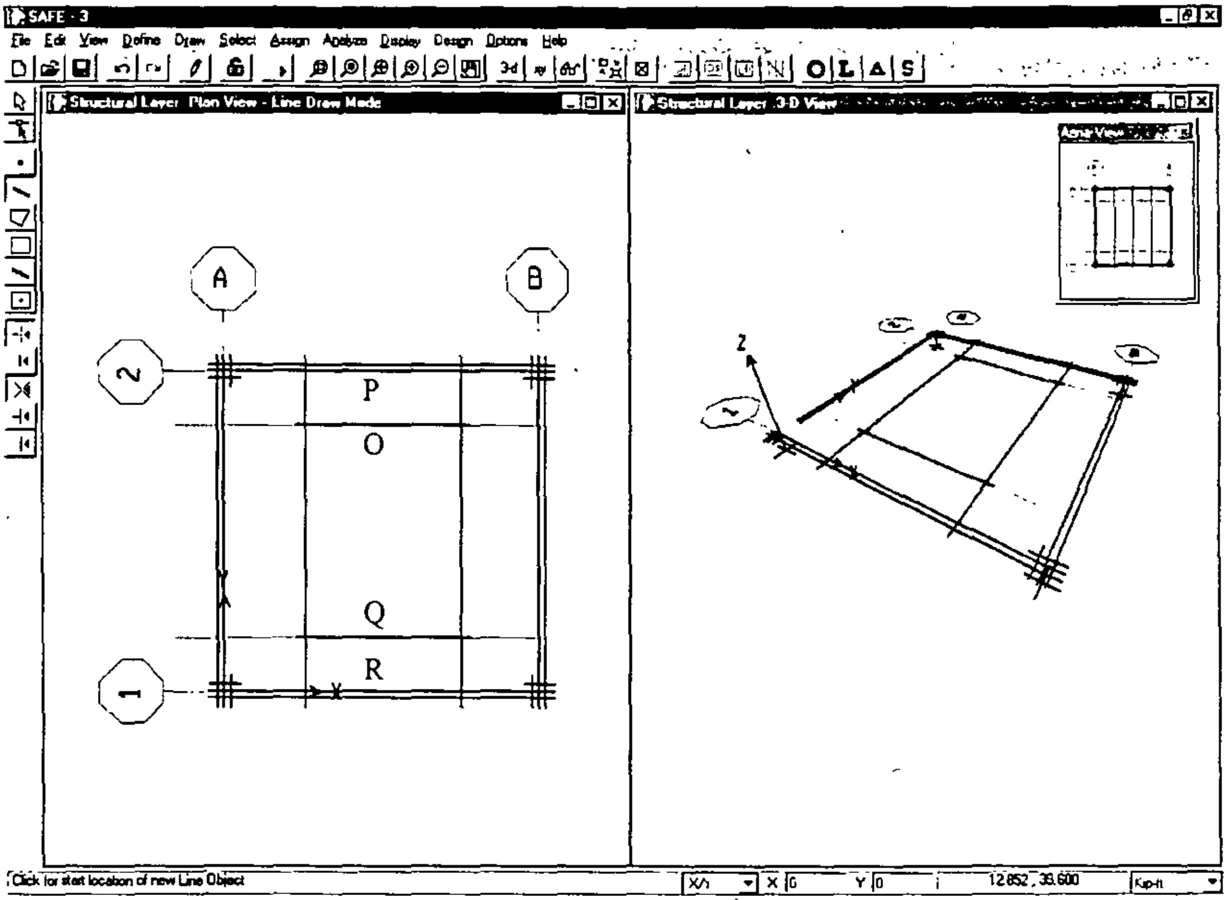


Figure 3-2: Model After Step 11

12. Click the **Snap to Intersections** button on the side toolbar to deactivate this snap option.

13. Click the **Snap to Middle and Ends** button on the side toolbar to activate this snap option.

14. Run the mouse cursor along the beam near the point labeled O in Figure 3-2 until the cursor snaps to the midpoint of the beam and a text box which says Mid Point appears. Left click the mouse to start drawing a new beam from this midpoint location.

15. Move the mouse up to the point labeled P in Figure 3-2. Again the mouse cursor will snap to the midpoint of the beam. Double click at point P to finish drawing this beam.

16. Similarly, click on the point labeled Q in Figure 3-2 and then double click on the point labeled R to draw another beam. Note that both points Q and R are at the midpoints of beams. The model now appears as shown in Figure 3-3.

17. Now we will draw the area object that will define the slab opening. Click the **Draw Rectangular Area Object** button on the side toolbar (or select **Draw Rectangular Object** from the **Draw** menu).

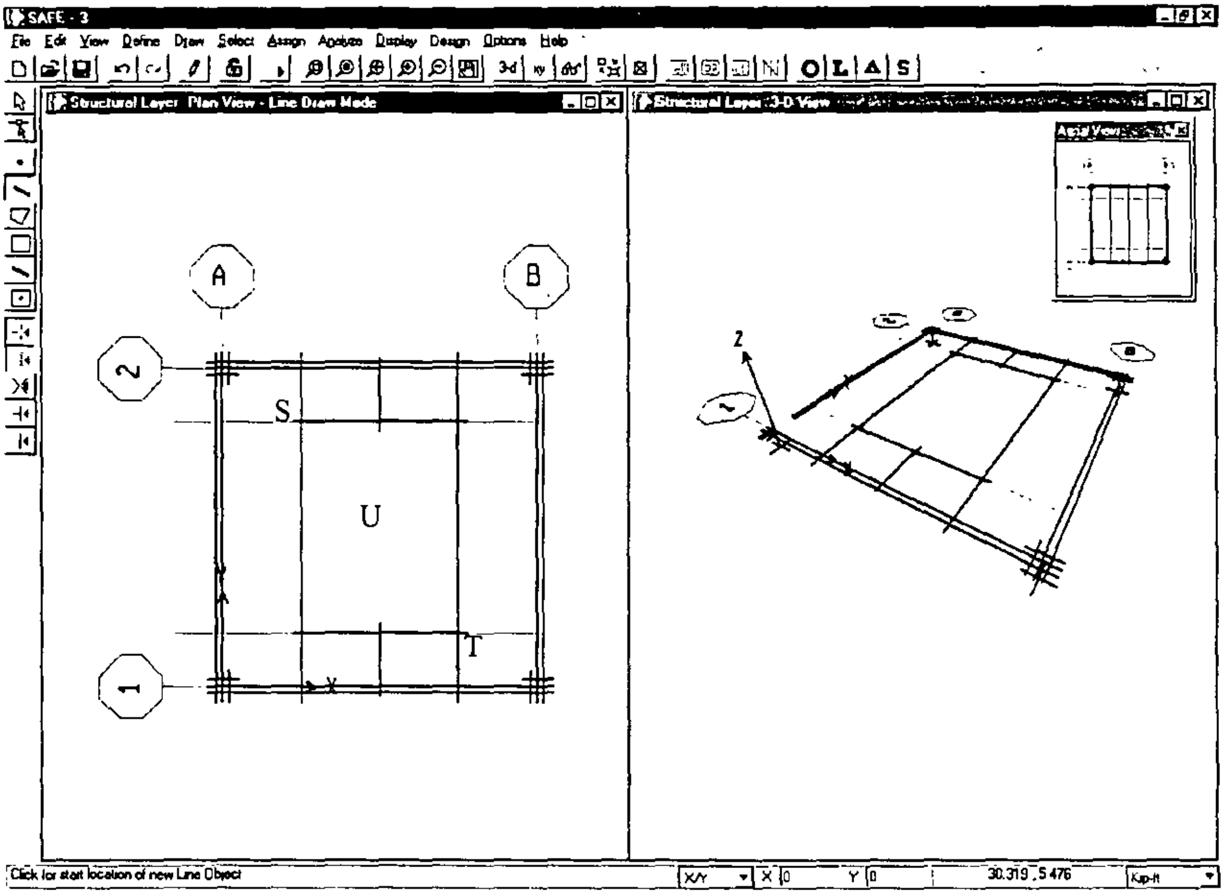


Figure 3-3: Model After Step 16

18. Single click on the point labeled S in Figure 3-3 and then double click on the point labeled T to draw the area object.

19. Click the **Pointer/Select** button to switch from Draw mode to Select mode.

20. Right click near the point labeled U in Figure 3-3 to display the Rectangular Slab Information dialog box for the just drawn area object.

21. In this dialog box:

    - In the Locate Slab area select the By Center option.

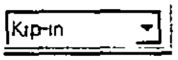
    - In the Identification and Location area verify that Center X and Center Y are both 12. These are the X and Y coordinates of the center of the area object.

    - In the Identification and Location area type **11** in the X Dimension and type **15** in the Y Dimension edit boxes.

    - Click the **OK** button.

22. Left click near the point labeled U in Figure 3-3 to select the just dimensioned area object.

23. From the Assign menu select Opening. This area is now designated as an opening.

24. Click the drop down box in the status bar to change the units to kip-in. 

25. From the **Define** menu select **Slab Properties...** to display the Slab Properties dialog box.

26. In this dialog box:

    - In the Slab Property area highlight the property named SLAB and then click the **Modify/Show Property** button to display the Slab Property Data dialog box.

    - In this dialog box:

      ➢ In the Analysis Property Data area:

        ✓ Verify that the Modulus of Elasticity is 3600.

        ✓ Verify Poisson's ratio is 0.2.

        Note: We will verify the unit weight later.

        ✓ Verify that the Bending Thickness (X), Bending Thickness (Y) and Twisting Thickness are all 6.

        Note: These values will all be 6 because we entered the footing thickness in the Ribbed Slab dialog box in Step 3.

        ✓ Verify that the Slab Type option is selected.

        ✓ Verify that the Thick Plate check box is checked.

      Note: Checking the Thick Plate check box tells the program to consider slab shear deformations in the analysis. If this box is not checked then slab shear deformations are not included in the analysis.

      ➢ In the Design Property Data area:

        ✓ Verify that the Thickness is 6.

        ✓ Type **2.5** in the X Cover Top (to Centroid) edit box.

        ✓ Type **1.5** in the Y Cover Top (to Centroid) edit box.

        ✓ Type **1.5** in the X Cover Bottom (to Centroid) edit box.

        ✓ Type **2.5** in the Y Cover Bottom (to Centroid) edit box.

- ✓ Verify the Concrete Strength, fc is 4.
- ✓ Verify the Reinforcing Yield Stress, fy is 60.
- ✓ Verify that the No Design check box is not checked.

➢ Click the **OK** button twice.

27. From the **Define** menu select **Beam Properties...** to display the Beam Properties dialog box.

28. In this dialog box:

- Click the drop-down box in the Click To area and click on Add Rectangular Beam to display the Property Data For Rectangular Beam dialog box.
- In this dialog box:

➢ Type **XBEAM** in the Beam Property name edit box.

➢ In the Analysis Property Data area:

- ✓ Verify that the Modulus of Elasticity is 3600.
- ✓ Verify Poisson's ratio is 0.2.

Note: We will verify the unit weight later.

- ✓ Verify the width is 12.
- ✓ Type **18** in the Depth edit box.

➢ In the Design Property Data area:

- ✓ Verify the width is 12.
- ✓ Type **18** in the Depth edit box.
- ✓ Type **2.5** in the Cover Top (to Centroid) edit box.
- ✓ Type **1.5** in the Cover Bottom (to Centroid) edit box.
- ✓ Verify the Concrete Strength, fc is 4.
- ✓ Verify the Reinforcing Yield Stress, fy is 60.
- ✓ Verify the Shear Steel Yield Stress, fys is 60.
- ✓ Verify the Concrete Shear Strength, fcs is 4.

- ✓ Verify that the No Design check box is <u>not</u> checked.

- ➢ Click the **OK** button.

- Click the drop-down box in the Click To area and click on Add Rectangular Beam to display the Property Data For Rectangular Beam dialog box.

- In this dialog box:

  - ➢ Type **YBEAM** in the Beam Property name edit box.

  - ➢ In the Analysis Property Data area:

    - ✓ Verify that the Modulus of Elasticity is 3600.

    - ✓ Verify Poisson's ratio is 0.2.

      Note: We will verify the unit weight later.

    - ✓ Verify the width is 12.

    - ✓ Type **18** in the Depth edit box.

  - ➢ In the Design Property Data area:

    - ✓ Verify the width is 12.

    - ✓ Type **18** in the Depth edit box.

    - ✓ Type **1.5** in the Cover Top (to Centroid) edit box.

    - ✓ Type **2.5** in the Cover Bottom (to Centroid) edit box.

    - ✓ Verify the Concrete Strength, fc is 4.

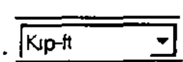
    - ✓ Verify the Reinforcing Yield Stress, fy is 60.

    - ✓ Verify the Shear Steel Yield Stress, fys is 60.

    - ✓ Verify the Concrete Shear Strength, fcs is 4.

    - ✓ Verify that the No Design check box is <u>not</u> checked.

  - ➢ Click the **OK** button twice.

29. Click the drop down box in the status bar to change the units to kip-ft. 

30. From the **Define** menu select Slab Properties... to display the Slab Properties dialog box.

31. In this dialog box:

- In the Slab Property area highlight the property named SLAB and then click the **Modify/Show Property** button to display the Slab Property Data dialog box.
- In this dialog box:
  - ➢ Verify that the unit weight is 0.15.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

32. From the **Define** menu select Beam Properties... to display the Beam Properties dialog box.

33. In this dialog box:

- In the Beam Property area highlight the property named XBEAM and then click the **Modify/Show Property** button to display the Property Data For Rectangular Beam dialog box.
- In this dialog box:
  - ➢ Verify that the unit weight is 0.15.
  - ➢ Click the **OK** button.
- In the Beam Property area highlight the property named YBEAM and then click the **Modify/Show Property** button to display the Property Data For Rectangular Beam dialog box.
- In this dialog box:
  - ➢ Verify that the unit weight is 0.15.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

34. Click in the plan view window titled Slab Properties to make sure it is active. When the window is active its title bar is highlighted.

35. Click the **Select All** button  on the main toolbar to select all elements.

36. From the **Assign** menu select **Beam Properties...** to display the Beam Properties dialog box.

37. In this dialog box:

- In the Beam Property list box click on YBEAM to highlight it.
- Click the **OK** button. The model now appears as shown in Figure 3-4.

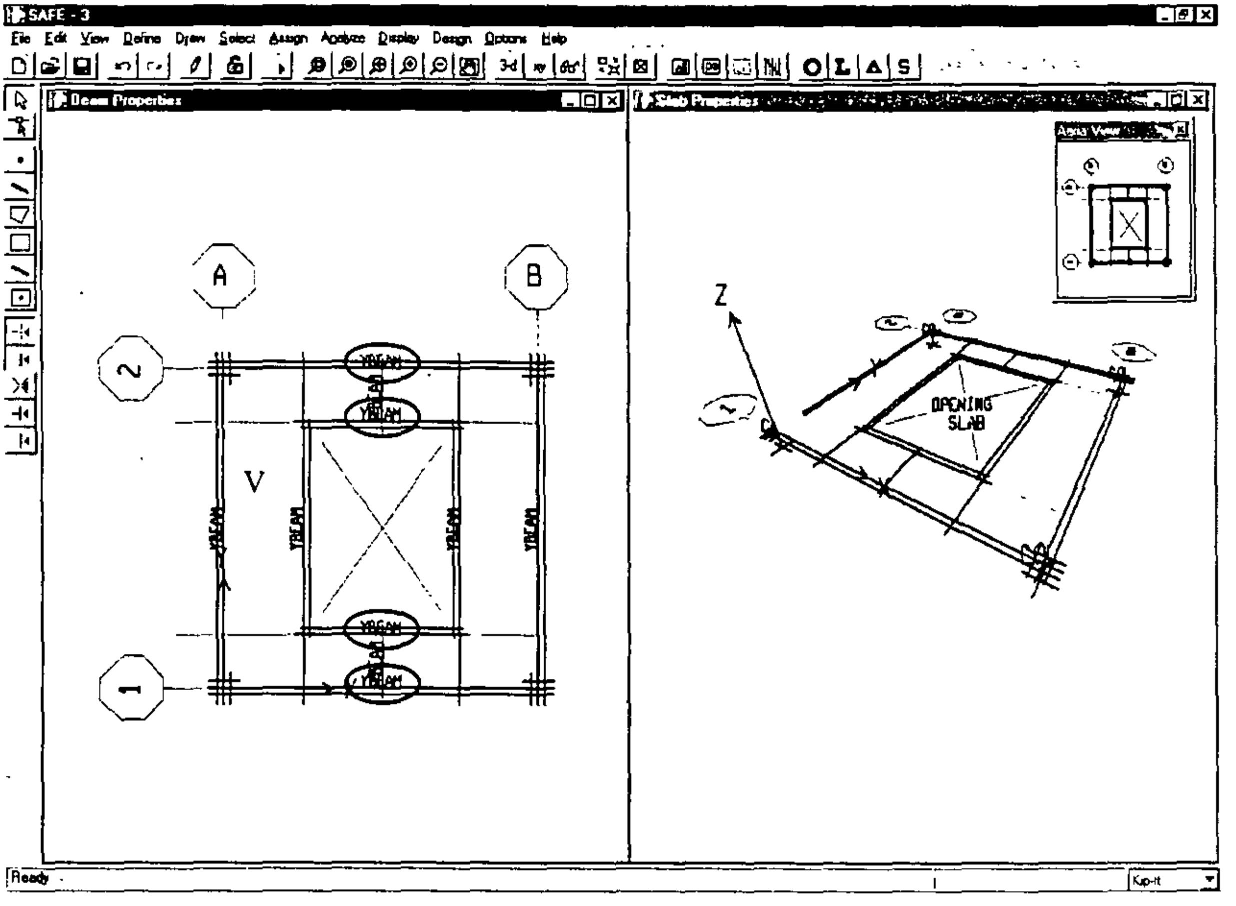


Figure 3-4: Model After Step 37

38. Select the four X-direction beams that are circled in Figure 3-4 by clicking on them.

39. From the **Assign** menu select **Beam Properties...** to display the Beam Properties dialog box.

40. In this dialog box:

    - In the Beam Property list box click on XBEAM to highlight it.
    - Click the **OK** button.

41. Right click somewhere on the slab area object not in the opening, for example at the point labeled V in Figure 3-4 to display the Rectangular Slab Information dialog box.

42. In this dialog box:

    - Note that the Slab Property is SLAB and recall that we have already defined these properties.

- Note that the Load Case drop-down box says DEAD and that the associated w/area edit box says 0.035. This is the dead load we initially input (see step 3).
- Select LIVE from the Load Case drop-down box and note that the associated w/area edit box says 0.1. This is the live load we initially input (see step 3).
- Click the **OK** button.

43. From the **Options** menu select **Preferences...** and then select the Concrete Tab.

44. In this dialog box:

    - Verify that ACI 318-95 is selected in the Concrete Design Code drop-down box.
    - Verify that in the Reinforcement Results Units area the Sq-in and Sq-in/ft option is selected.
    - Click the **OK** button.

45. On the **Design** menu choose **Select Design Combos** to display the Design Load Combinations Selection dialog box.

46. In this dialog box:

    - In the Design Combos list box click on DCON2 to highlight it and then click the **Show** button.
    - Note that this load combination is defined as 1.4DL + 1.7LL. Also note that the Use For Design check box is checked.

      *Note: SAFE automatically creates design load combinations based on the selected design code and on the static load types. In this case where the defined static load types are DEAD and LIVE (see **Static Load Cases** on the **Define** menu) and the design code is ACI 318-95, the program automatically creates two load cases. They are 1.4DL (DCON1) and 1.4DL + 1.7LL (DCON2).*

      *If we wanted to add an additional design load combination we could define it by selecting **Load Combinations...** from the **Define** menu and then clicking the **Add New Combo** button, to display the Load Combination data dialog box. In this dialog box we can define the load combination and we can check the Use For Design check box if we want the load combination to be used in design.*

    - Click the **OK** button twice to exit all dialog boxes.

47. From the **Analyze** menu select **Set Options** to display the Analysis Options dialog box.

48. In this dialog box:

- In the Analysis Type area verify that the Normal option is selected.
- Accept the default maximum mesh dimension of 4 feet.

  Note: To automatically mesh the model the program first develops a mesh that is broken at each grid line, at the edge of any area objects, at the ends of each line object and at each point object. If the dimension of any elements meshed in this manner exceeds the specified maximum mesh dimension then those elements are subdivided such that the maximum mesh dimension is no longer exceeded.

  Note: In general, the maximum mesh size should be based on the span length. It is good practice to have at least four elements in a span, and generally it is not necessary to have more than eight elements in a span. Thus if the typical span length is less than sixteen feet, it may be advisable to reduce the default maximum mesh size to something less than 4 feet.

  In this example we are accepting the default maximum mesh of four feet. This means that the slab spanning between beams will be modeled using two elements which is less than the four elements recommended in the previous paragraph. If the accuracy of the slab results (moments, etc.) were important to us then we might want to reduce the maximum mesh size to 1.5 feet.

  *Note: The mesh automatically generated by the program can be viewed by clicking the **Set Objects** button ⊠ on the main toolbar (or selecting **Set Object Options...** from the **View** menu) to display the Set Objects dialog box, checking the Show Mesh check box in the Options area, and clicking the **OK** button. You can turn off the display of the mesh by unchecking this box.*

- Click the **OK** button.

49. Click the **Run Analysis** button on the main toolbar (or select **Run Analysis** from the **Analyze** menu).

50. When the initial analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

51. Click the **Show Beam Forces** button on the main toolbar (or select **Show Beam Forces...** from the **Display** menu) to display the Beam Forces dialog box.

52. In this dialog box:

    - Select DCON2 Combo from the Load drop-down box.
    - In the Component area select the Moment option.

- In the Display options area check the Show Values check box. Note that if the Fill Diagram check box is checked you will have to uncheck it before you can check the Show Values box.

- Click the **OK** button to display the moment diagrams.

53. Run the mouse cursor over the beams and read the moment values from the left-hand side of the status bar at the bottom of the SAFE window.

54. If you want to view beam force results for other force components then repeat steps 51 through 53.

55. On the **Design** menu select **Start Design**. When the design is complete the calculated X-strip reinforcement is displayed in square inches.

56. From the **Design** menu select **Display Beam Design Info...** to display the Beam Reinforcing dialog box.

57. In this dialog box:

    - In the Type of Reinforcing area verify that the Flexural option is chosen.

    - In the Reinforcing Values area verify that the Show Rebar Values at Controlling Station check box is checked.

    - In the Reinforcing Values area verify that the Show Rebar Values at Every Station check box is not checked.

    - Accept the other default values in this dialog box.

    - Click the **OK** button.

58. Run the mouse cursor over the beams and read the flexural steel values from the left-hand side of the status bar at the bottom of the SAFE window.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "A"

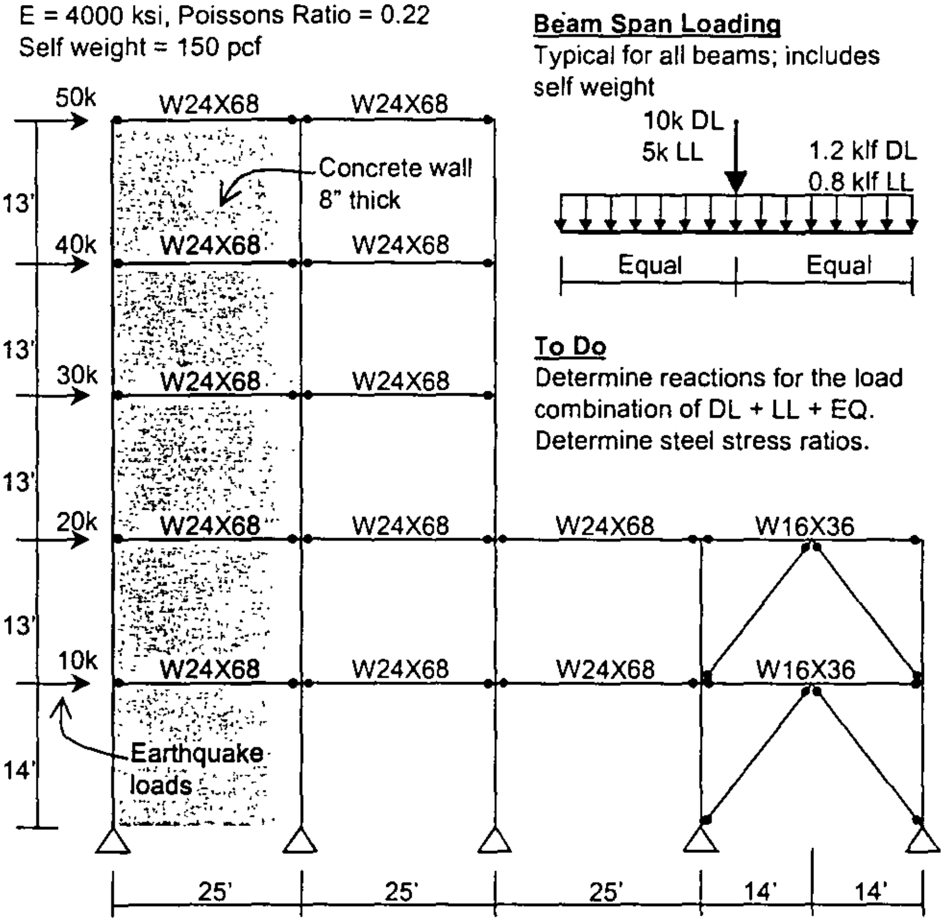
EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Problem A

Concrete Wall and Steel Frame

Steel
Fy = 36 ksi, E = 29500 ksi, Poisson's Ratio = 0.3
Columns: W10X49, typical - pinned base
Beams: As noted, pinned ends except continuous over top of brace
Assume all W24X68 beams are braced at 1/3 points
Assume W16X36 beams braced at center only
Beams at concrete wall are not embedded in wall
Braces: TS6X6X1/4, pinned ends
Design Code: AISC-ASD89

Concrete
E = 4000 ksi, Poissons Ratio = 0.22
Self weight = 150 pcf



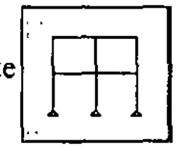
To Do
Determine reactions for the load combination of DL + LL + EQ.
Determine steel stress ratios.

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem A Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box:

   - Type **5** in the Number of Stories edit box.
   - Type **4** in the Number of Bays edit box.
   - Type **13** in the Story Height edit box.
   - Type **25** in the Bay Width edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Labels box in the Joints area.
   - Check the Labels box in the Frames area.
   - Click the **OK** button.

8. Select column elements 18, 19, 20, 23, 24 and 25 and beam elements 38, 39, 40, 43, 44 and 45. Press the delete key on the keyboard to delete these elements.

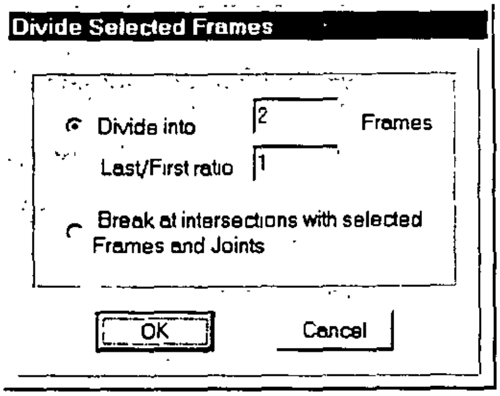
   *Note: You could select the elements by clicking on each one individually, by "windowing", by using the Intersecting Line Select Mode, or by using the Select By Labels option (Select menu > **Select** > **Labels**).*

9. Click the **Refresh Window** button to refresh the drawing.

10. From the **Draw** menu choose **Edit Grid...** to display the Modify Grid Lines dialog box.

11. In this dialog box:

- Verify that the X option is chosen in the Direction area.
- Check the Glue Joints To Grid Lines box.
- Click on the 50 grid line in the X Location list box to highlight it. Note that the 50 appears in the X Location edit box.
- Change the 50 in the X Location edit box to **53** and click the **Move Grid Line** button.
- Select the Z option in the Direction area.
- Click on the 0 grid line in the Z Location list box to highlight it. Note that the 0 appears in the Z Location edit box.
- Change the 0 in the Z Location edit box to **-1** and click the **Move Grid Line** button.
- Click the **OK** button.



12. Select beams 41 and 42.
13. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.
14. Fill in the dialog box as shown in the figure (typically the dialog box will default to these values) and click the **OK** button.
15. Verify that the **Snap to Joints and Grid Points** button on the side tool bar is depressed.
16. Click the **Draw Frame Element** button on the side toolbar, or select **Draw Frame Element** from the **Draw** menu.
17. Draw the first brace element as follows:
    - Place the mouse pointer on joint 19. When the text box saying "Grid Intersection" appears click the left mouse button once.
    - Move the mouse pointer to joint 31. When the text box saying "Point" appears click the left mouse button once.
    - Press the Enter key on the keyboard.
18. Click on joint 25 and then joint 31, and press the Enter key to draw the second brace element.
19. Click on joint 20 and then joint 32, and press the Enter key to draw the third brace element.

20. Click on joint 26 and then joint 32, and press the Enter key to draw the fourth and final brace element.

21. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

22. Click the drop down box in the status bar to change the units to kip-in.

23. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

24. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

25. In this dialog box:

    - Type **0** in the Mass per Unit Volume edit box.
    - Type **0** in the Weight per Unit Volume edit box.
    - Type **29500** in the Modulus of Elasticity edit box.
    - Type **.3** in the Poisson's Ratio edit box, if it is not already entered.
    - Type **36** in the Steel Yield Stress, Fy edit box, if it is not already entered.
    - Click the **OK** button.

26. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

27. In this dialog box:

    - Type **4000** in the Modulus of Elasticity edit box.
    - Type **.22** in the Poisson's Ratio edit box
    - Accept the other values in the dialog box.
    - Click the **OK** button.

28. Click the **OK** button to close the Define Materials dialog box.

29. Click the drop down box in the status bar to change the units to kip-ft.

30. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

31. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

32. In this dialog box:

- Verify **0.15** is entered in the Weight per Unit Volume edit box.
- Click the **OK** button.

33. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

34. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

35. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.

36. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X68 section.
    - Scroll down to the W16X36 section, and click on it while holding down the Ctrl key on the keyboard.
    - Scroll down to the W10X49 section, and click on it while holding down the Ctrl key on the keyboard.
    - Click the **OK** button twice to return to the Define Frame sections dialog box.

37. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import Box/Tube item.

38. A dialog box appears with a list of all structural tube sections in the database. In this dialog box:

    - Scroll down and click on the TS6X6X1/4 section.
    - Click the **OK** button three times to exit all dialog boxes.

39. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

40. In the Click To area, click the **Add New Section** button to display the Shell Sections dialog box.

41. In this dialog box:

    - Type **WALL** in the Section Name edit box.
    - In the Thickness area type **.6667** in both the Membrane and the Bending edit boxes.

- Verify that the Shell option is selected in the Type area.
- Click the **OK** button.

42. Click the **OK** button to close the Define Shells dialog box.

43. Select all of the beams except for the braced frame beams (i.e., select beams 26 through 37 and do not select beams 46 through 49). The Intersecting Line Selection option could be useful for this.

    *Note: To use the Intersecting Line Selection option, click the **Set Intersecting Line Select Mode** button on the side tool bar. Then click the left mouse button at the top of one beam bay, and while holding down the left mouse button drag the mouse to the bottom of the beam bay. A "rubberband line" will appear and all elements that this "rubberband line" passes through will be selected. Release the left mouse button to make the selection.*

44. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.

45. In this dialog box check both the Start and the End boxes for Moment 33 (Major) and then click the **OK** button.

46. Select beam elements 46 and 48.

47. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.

48. In this dialog box check the Start box for Moment 33 (Major) and then click the OK button.

49. Select beam elements 47 and 49.

50. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.

51. In this dialog box check the End box for Moment 33 (Major) and then click the OK button.

52. Select all of the braces (i.e., select braces 50 through 53).

53. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.

54. In this dialog box check both the Start and the End boxes for Moment 33 (Major) and then click the OK button.

55. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

56. In this dialog box:

- Type **DL** in the Load Edit box.
- Click the **Change Load** button.
- Type **LL** in the Load Edit box.
- Select Live from the Type drop-down box.
- Type **0** in the Self Weight Multiplier edit box.
- Click the **Add New Load** button.
- Type **EQ** in the Load Edit box.
- Select Quake from the Type drop-down box.
- Click the **Add New Load** button.
- Click the **OK** button.

57. From the **Define** menu select **Load Combinations...** to display the Define Load Combinations dialog box.

58. In this dialog box:

- Click the **Add New Combo** button to display the Load Combination Data dialog box.
- In this dialog box:
    - ➢ Type **ALL** in the Load Combina.·on Name edit box.
    - ➢ Select ADD from the Load Combination Type drop-down box if it is not already selected.
    - ➢ Type **DL + LL + EQ** in the Title edit box.
    - ➢ Select DL Load Case in the Case Name drop down box (if it is not already selected) and type **1** in the Scale Factor edit box (if it is not already there).
    - ➢ Click the **Add** button.
    - ➢ Select LL Load Case in the Case Name drop down box.
    - ➢ Click the **Add** button.
    - ➢ Select EQ Load Case in the Case Name drop down box.
    - ➢ Click the **Add** button.

➢ Click the **OK** button twice.

59. Select beams 26 through 37.

60. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

61. In this dialog box:

- Select DL from the Load Case Name drop-down box.
- In the Point Loads area type **.5** in the first Distance edit box and type **-10** in the first Load edit box
- Type **-1.2** in the Uniform Load area edit box.
- Click the **OK** button.

62. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

63. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

64. In this dialog box:

- Select LL from the Load Case Name drop-down box.
- In the Point Loads area type **-5** in the first Load edit box
- Type **-.8** in the Uniform Load area edit box.
- Click the **OK** button.

65. Select beams 46 through 49.

66. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

67. In this dialog box:

- Select DL from the Load Case Name drop-down box.
- In the Point Loads area type **0** in the first Distance edit box and type **0** in the first Load edit box
- Type **-1.2** in the Uniform Load area edit box.

- Click the **OK** button.

68. Click the **Restore Previous Selection** button [ps] on the side toolbar (or select Get Previous Selection from the Select menu).

69. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

70. In this dialog box:

    - Select LL from the Load Case Name drop-down box.

    - Type **-.8** in the Uniform Load area edit box.

    - Click the **OK** button.

71. Select joints 31 and 32.

72. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

73. In this dialog box:

    - Select DL from the Load Case Name drop-down box.

    - Type **-10** in the Force Global Z edit box in the Loads area.

    - Click the **OK** button.

74. Click the **Restore Previous Selection** button [ps] on the side toolbar (or select Get Previous Selection from the Select menu).

75. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

76. In this dialog box:

    - Select LL from the Load Case Name drop-down box.

    - Type -5 in the Force Global Z edit box in the Loads area.

    - Click the **OK** button.

77. Select joints 2, 3, 4, 5 and 6 by "windowing".

78. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

79. In this dialog box:

- Select EQ from the Load Case Name drop-down box.
- Type **10** in the Force Global X edit box in the Loads area.
- Type **0** in the Force Global Z edit box in the Loads area.
- Click the **OK** button.

80. Select joints 3, 4, 5 and 6 (not 2) by "windowing".
81. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
82. In this dialog box:
    - Verify Add To Existing Loads is selected in the Options area.
    - Click the **OK** button.
83. Select joints 4, 5 and 6 (not 2 and 3) by "windowing".
84. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
85. In this dialog box click the **OK** button.
86. Select joints 5 and 6 (not 2, 3 and 4).
87. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
88. In this dialog box click the **OK** button.
89. Select joint 6 (not 2, 3, 4 and 5).
90. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
91. In this dialog box click the **OK** button.
92. Select beams 26 through 37.
93. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
94. In this dialog box:
    - Click on W24X68 in the Frame Sections area to highlight it.

- Click the **OK** button.

95. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments so that you can see the frame element labels again.

96. Select beams 46 through 49.

97. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

98. In this dialog box:

    - Click on W16X36 in the Frame Sections area to highlight it.

    - Click the **OK** button.

99. Select all of the columns. An easy way to do this is to "window" each of the column lines separately.

    Note: To "window" a column line, left click the mouse above and to the left of the column line. While holding the left mouse button down, drag the mouse so that it is below and to the right of the column line. A "rubberband window" will appear surrounding the column line. Release the left mouse button to select all elements that are fully enclosed by the "rubberband window".

100. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

101. In this dialog box:

    - Click on W10X49 in the Frame Sections area to highlight it.

    - Click the **OK** button.

102. Select the four brace elements.

103. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

104. In this dialog box:

    - Click on TS6X6X1/4 in the Frame Sections area to highlight it.

    - Click the **OK** button.

105. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

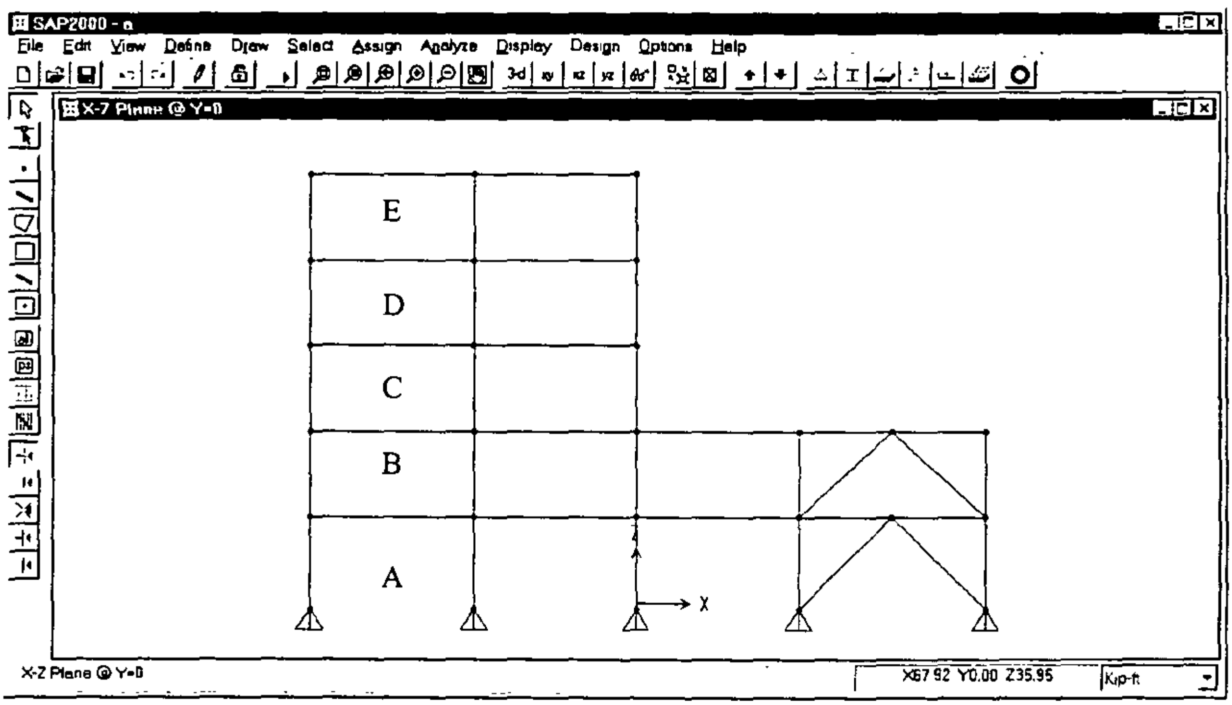
106. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

107. In this dialog box:

- Uncheck the Labels box in the Joints area.
- Uncheck the Labels box in the Frames area.
- Click the **OK** button.

108. Click the Quick Draw Rectangular Shell Element button on the side toolbar.

109. Click the mouse pointer once in the areas labeled A, B, C, D and E in the figure below to draw the shell elements.



110. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

111. Select the five shell elements just entered by clicking on them.

112. From the **Assign** menu select **Shell** and then select **Sections...** from the submenu to display the Define Shell Sections dialog box.

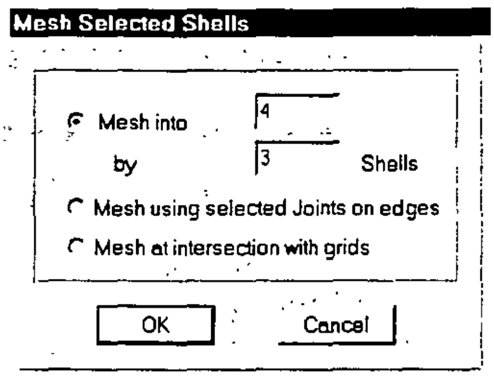
113. In this dialog box:

- Click once on the WALL item in the Shell Sections area to highlight it.
- Click the **OK** button.

114. Click the **Show Undeformed Shape** button to remove the displayed shell section assignments.

115. Click the **Restore Previous Selection** button on the side toolbar (or select Get Previous Selection from the Select menu).

116. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.



117. Fill out this dialog box as shown in the figure to mesh each shell into twelve elements (4 by 3) and click the **OK** button.

118. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

119. In this dialog box:

- Check the Hide box in the Shells area.

- Check the Hide box in the Joints area.

- Check the Sections box in the Frames area.

- Click the **OK** button.

120. Select all of the W24X68 beam sections (12 total).

121. From the **Design** menu select **Redefine Element Design Data...** to display the Element Overwrite Assignments dialog box.

122. In this dialog box:

- Check the Unbraced Length Ratio (Minor, LTB) box and type **.3333** in the associated edit box.

- Click the **OK** button.

123. Select all of the W16X36 beam sections (4 total).

124. From the **Design** menu select **Redefine Element Design Data...** to display the Element Overwrite Assignments dialog box.

125. In this dialog box:

- Check the Unbraced Length Ratio (Minor, LTB) box and type **.5** in the associated edit box.

- Click the **OK** button.

126. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

127. In this dialog box:

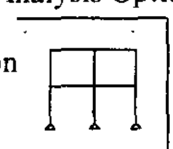
    - Uncheck the Hide box in the Joints area.
    - Check the Restraints box in the Joints area.
    - Uncheck the Sections box in the Frames area.
    - Uncheck the Hide box in the Shells area.
    - Click the **OK** button.

128. From the **Options** menu select **Preferences...** to display the Preferences dialog box.

129. In this dialog box.

    - Click on the Steel Tab
    - Select AISC-ASD89 from the Steel Design Code drop-down box if it is not already selected.
    - Click the **OK** button.

130. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
    - Click the **OK** button.

131. Click the **Run Analysis** button ▸ to run the analysis.

132. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

133. Click the **Joint Reaction Forces** button J on the main toolbar to display the Joint Reaction Forces dialog box.

134. In this dialog box:

    - Select All Combo from the Load drop-down box.
    - Verify that the Reactions option is selected in the Type area.

- Click the **OK** button.

135. The reactions are displayed on the screen. You can right click on any joint to see the reactions at that joint or you can just read the reactions on the screen. If the text is too small to read, you can zoom in, or you can change the minimum font size as described in the note below.

*Note: To change the minimum font size select **Preferences** from the **Options** menu and make sure the Dimensions Tab is selected. In the Minimum Graphic font Size edit box input a new size, maybe 5 or 6 points. Click the **OK** button. Click the Refresh Window button located on the main toolbar to see the changes.*

136. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.

137. When the design check completes, the stress ratios are displayed.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "B"

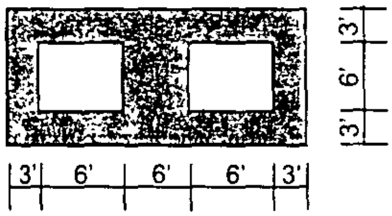
EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Problem B

Concrete Wall

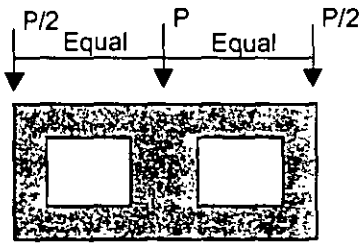
Concrete
E = 3600 ksi, Poissons Ratio = 0.2

To Do
Model wall using shell elements. Determine shear axial force and moment in Pier A, and determine total shear, moment and axial force at the sixth level for all piers combined.

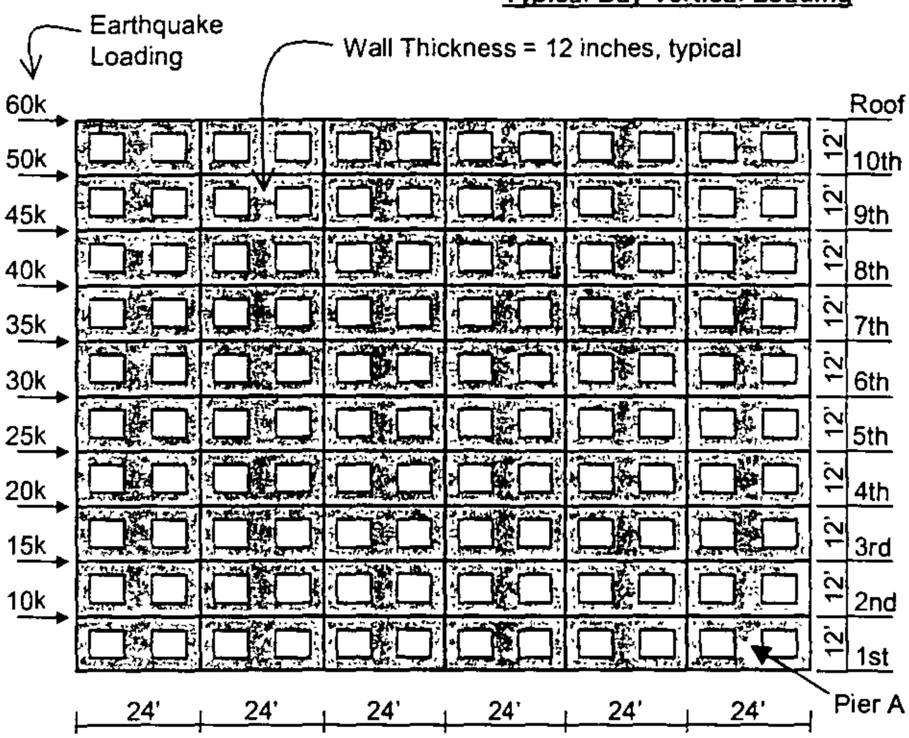


Typical Bay Dimensions

P (DL) = 21.6 k, P (LL) = 7.2 k



Typical Bay Vertical Loading



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem B Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Shear Wall** template button to display the Shear Wall dialog box.

4. In this dialog box

   - Type **8** in the Number of Spaces Along X edit box.
   - Type **4** in the Number of Spaces Along Z edit box.
   - Type **3** Space Width Along X edit box.
   - Type **3** Space Width Along Z edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Labels box in the Joints area.
   - Check the Labels box in the Shells area.
   - Click the **OK** button.

8. Select shell elements 6, 7, 10, 11, 22, 23, 26 and 27 by clicking on them.

9. Press the Delete key on the keyboard to delete these elements.

10. Click the **Refresh Window** button to refresh the drawing.

11. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

12. In this dialog box:

    - Type **DL** in the Load Edit box.

- Click the **Change Load** button.
- Type **LL** in the Load Edit box.
- Select Live from the Type drop-down box.
- Type **0** in the Self Weight Multiplier edit box.
- Click the **Add New Load** button.
- Type **EQ** in the Load Edit box.
- Select Quake from the Type drop-down box.
- Click the **Add New Load** button.
- Click the **OK** button.

13. From the **Define** menu select **Load Combinations...** to display the Define Load Combinations dialog box.

14. In this dialog box:

- Click the **Add New Combo** button to display the Load Combination Data dialog box.
- In this dialog box:
  - Type **ALL** in the Load Combination Name edit box.
  - Select ADD from the Load Combination Type drop-down box if it is not already selected.
  - Type **DL + LL + EQ** in the Title edit box.
  - Select DL Load Case in the Case Name drop down box (if it is not already selected) and type **1** in the Scale Factor edit box (if it is not already there).
  - Click the **Add** button.
  - Select LL Load Case in the Case Name drop down box.
  - Click the **Add** button.
  - Select EQ Load Case in the Case Name drop down box.
  - Click the **Add** button.
  - Click the **OK** button twice.

15. Select joints 10, 25 and 45.

16. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

17. In this dialog box:

    - Select DL from the Load Case Name drop-down box.
    - Type **-10.8** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.

18. Select joint 25.

19. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box. Click the **OK** button in this dialog box.

20. Select joints 10, 25 and 45.

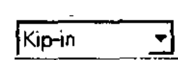
21. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

22. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - Type **-3.6** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.

23. Select joint 25.

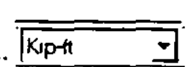
24. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box. Click the **OK** button in this dialog box.

25. Click the drop down box in the status bar to change the units to kip-in. 

26. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

27. In this dialog box:

    - Verify that the modulus of elasticity is 3600 and poisson's ratio is 0.2.
    - Click the **OK** button twice to exit the dialog boxes.

28. Click the drop down box in the status bar to change the units to kip-ft. 

29. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box. Click the **Modify/Show Section** button to display the Shell Sections dialog box.

30. In this dialog box:

    - Accept all of the default values.
    - Click the **OK** button twice to exit the dialog boxes.

31. Click the **Select All** button on the side tool bar.

32. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

33. In this dialog box:

    - Click the Linear Tab if it is not already selected.
    - In the Distance area type **24** in the X edit box.
    - Type **5** in the Number edit box.
    - Click the **OK** button

34. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

35. In this dialog box:

    - Check the Hide box in the Joints area.
    - Uncheck the Labels box in the Shells area.
    - Click the **OK** button.

36. Click the **Select All** button on the side tool bar.

37. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

38. In this dialog box:

    - Click the Linear Tab if it is not already selected.
    - In the Distance area type **0** in the X edit box.
    - Type **12** in the **Z** edit box.
    - Type **9** in the Number edit box.
    - Click the **OK** button.

39. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

40. In this dialog box:

    - Uncheck the Hide box in the Joints area.
    - Check the Labels box in the Joints area.
    - Check the Restraints box in the Joints area.
    - Check the Fill Elements box in the Options area.
    - Click the **OK** button.

41. Select joint 10. You may need to zoom in to distinguish it.

42. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

43. In this dialog box:

    - Select EQ from the Load Case Name drop-down box.
    - Type **10** in the Force Global X edit box in the Loads area.
    - Type **0** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.

44. Select joint 243.

45. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

46. In this dialog box:

    - Type **15** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

47. Select joint 427.

48. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

49. In this dialog box:

    - Type **20** in the Force Global X edit box in the Loads area.

- Click the **OK** button.

50. Select joint 611.

51. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

52. In this dialog box:

    - Type **25** in the Force Global X edit box in the Loads area.

    - Click the **OK** button.

53. Select joint 795.

54. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

55. In this dialog box:

    - Type **30** in the Force Global X edit box in the Loads area.

    - Click the **OK** button.

56. Select joint 979.

57. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

58. In this dialog box:

    - Type **35** in the Force Global X edit box in the Loads area.

    - Click the **OK** button.

59. Select joint 1163.

60. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

61. In this dialog box:

    - Type **40** in the Force Global X edit box in the Loads area.

    - Click the **OK** button.

62. Select joint 1347.

63. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

64. In this dialog box:

    - Type **45** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

65. Select joint 1531.

66. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

67. In this dialog box:

    - Type **50** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

68. Select joint 1715.

69. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

70. In this dialog box:

    - Type **60** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

71. Click the **Show Undeformed Shape** button to remove the displayed joint force assignments.

72. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

73. In this dialog box:

    - Check the Labels box in the Shells area.
    - Click the **OK** button.

74. Zoom in on the pier labeled Pier A in the problem statement.

75. Select joints 208, 212 and 218.

76. Select shell elements 138 and 142.

77. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

78. In this dialog box:

- Type **PIERA** in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

79. From the **View** menu select **Restore Full View**.

80. Select all joints level with the bottom of the sixth floor windows by "windowing" (joints 972, 973, 980, etc., 49 joints total).

81. Select all shell elements level with the bottom half of the sixth floor windows by using the intersecting line selection method (shell elements 730, 738, 742, etc., 24 shells total).

82. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

83. In this dialog box:

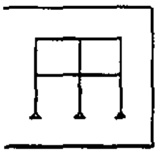
- Type 6TH in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

84. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

85. In this dialog box:

- Uncheck the Labels box in the Joints area.
- Uncheck the Labels box in the Shells area.
- Click the **OK** button.

86. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
- Click the **OK** button.

87. Click the **Run Analysis** button ▸ to run the analysis.

88. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

89. From the **Display** menu select **Show Group Joint Force Sums** to display the Select Groups dialog box.

90. In this dialog box:

    - Click on the group named 6TH to highlight it.

    - Hold down the Ctrl key on the keyboard and click on the group named PIERA to add it to the selection

    - Click the **OK** button.

91. When finished viewing the group joint force sums press the "X" in the upper right-hand corner of the Group joint Force Summation window to close it.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "C"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

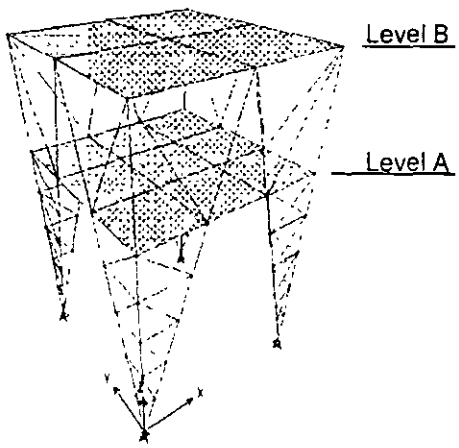
Problem C

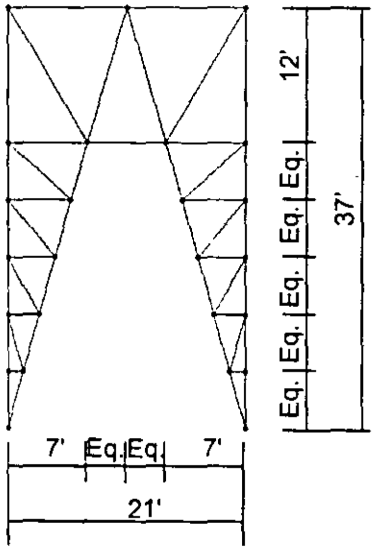
Truss Frame

Steel Frame
E =29000 ksi, Poissons Ratio = 0.3
All steel members are L4x4 angles, Fy = 36ksi
Base is pinned

Diaphragms
Concrete diaphragms are 8" thick with a unit weight of 150 pcf
Model as rigid diaphragm at Levels A and B
Additional dead load at each diaphragm is 50 psf
Live load at each diaphragm is 100 psf

To Do
Size steel members for DL + LL using AISC - ASD89
Determine the first three modes of vibration





Typical Elevation
(All four sides are the same)

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem C Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model...**. This displays the Coordinate System Definition dialog box.

3. In this dialog box

   - Select the Cartesian Tab.
   - In the Number of Grid Spaces area type **0** in the X direction edit box.
   - In the Number of Grid Spaces area type **0** in the Y direction edit box.
   - In the Number of Grid Spaces area type **0** in the Z direction edit box.
   - Click the **OK** button.

4. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

5. In this dialog box:

   - Verify that the X option is selected in the Direction area.
   - Type **7** in the X Location edit box and click the **Add Grid Line** button.
   - Type **10.5** in the X Location edit box and click the **Add Grid Line** button.
   - Select the Z option in the Direction area.
   - Type **25** in the Z Location edit box and click the **Add Grid Line** button.
   - Type **37** in the Z Location edit box and click the **Add Grid Line** button.
   - Click the **OK** button.

6. Click in the window titled X-Y Plane @ Z=0 to make sure it is active. The window is highlighted when it is active.

7. Click the **xz 2D View** button xz to change the view to an X-Z elevation. Note that the title of the window changes to X-Z Plane @ Y=0.

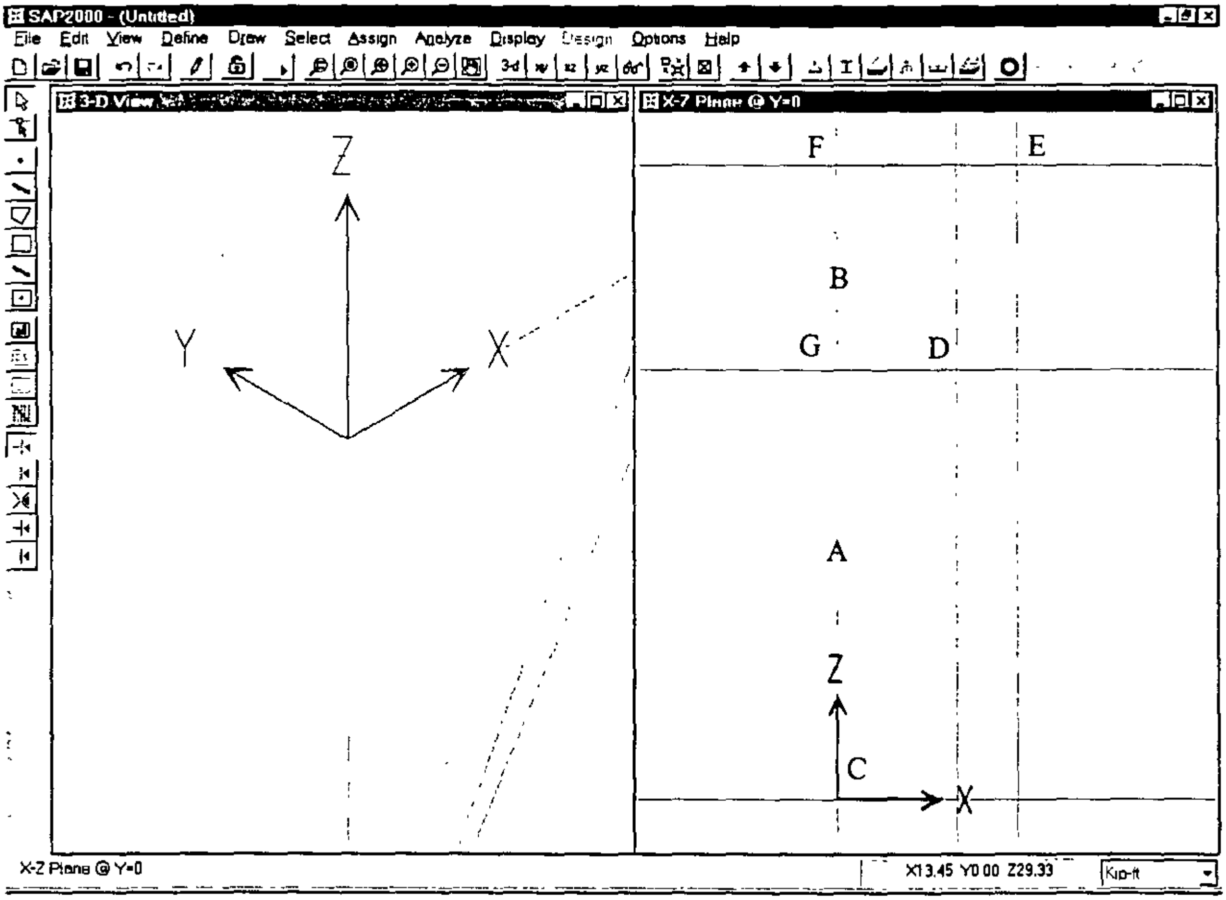


Figure C-1: Initial Grid Layout In X-Z Plane

8. Click the **Quick Draw Frame Element** button on the side toolbar or select **Quick Draw Frame Element** from the **Draw** menu.

9. Click on the grid line at the point labeled "A" in Figure C-1 to enter a frame element.

10. Click on the grid line at the point labeled "B" in Figure C-1 to enter another frame element.

11. Click the **Draw Frame Element** button on the side toolbar or select **Draw Frame Element** from the **Draw** menu.

12. Click on the points labeled "C", "D", and "E" in Figure C-1, in that order, and then press the Enter key on the keyboard to draw two more frame elements.

13. Click on the point labeled "F" and then double click the point labeled "E" in Figure C-1 to draw the next frame element.

Note: You could have single-clicked the point labeled "E" in Figure C-1 and then pressed the Enter key on the keyboard to finish drawing the frame element.

14. Click on the point labeled "G" and then double click the point labeled "D" in Figure C-1 to draw the next frame element.

15. Click on the point labeled "D" and then double click the point labeled "F" in Figure C-1 to draw the next frame element.

16. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

17. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

18. In this dialog box:

    - Check the Labels box in the Joints area.
    - Check the Labels box in the Frames area.
    - Check the Fill Elements box.
    - Click the **OK** button. The screen appears as shown in Figure C-2.

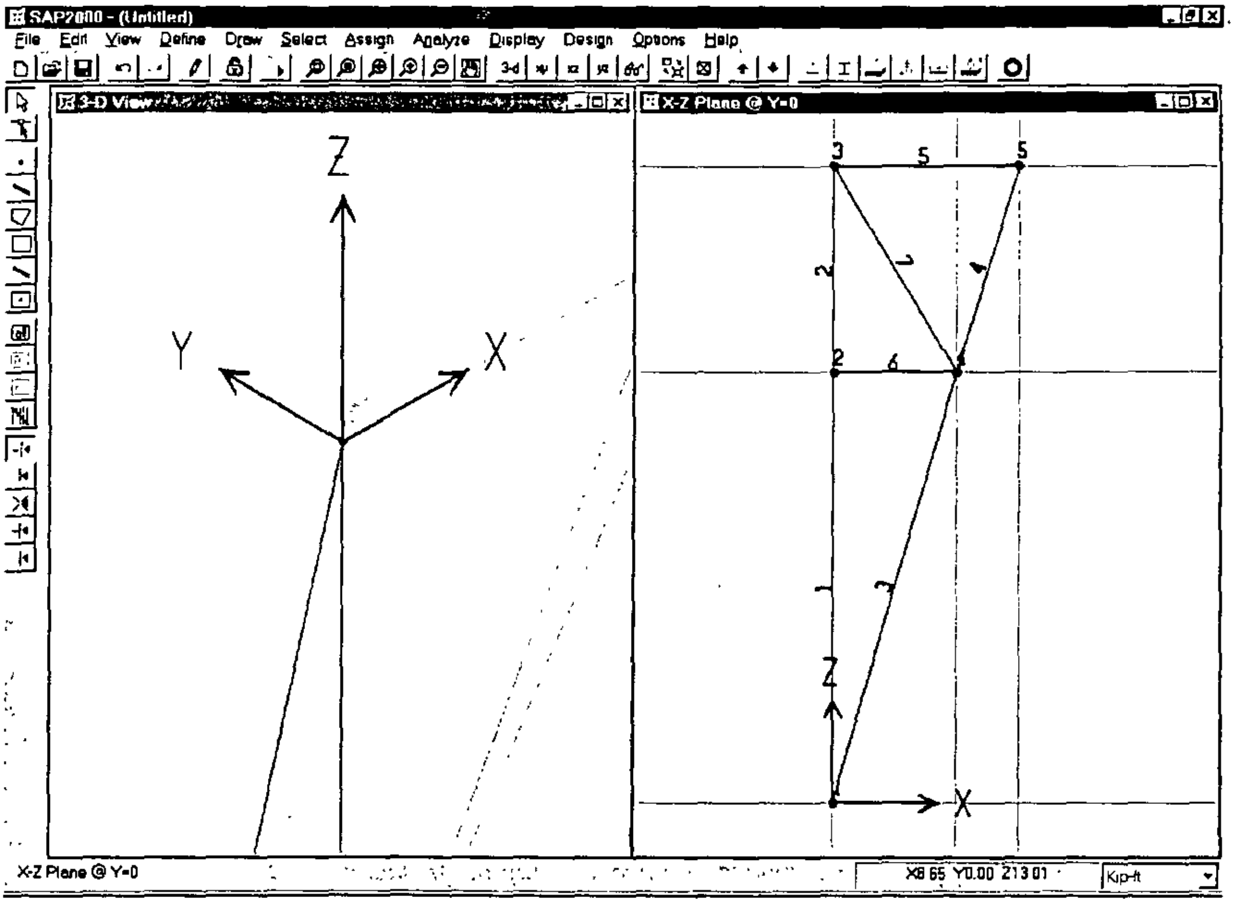
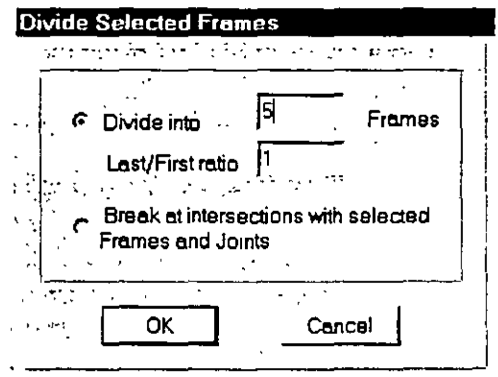


Figure C-2: Screen As It Appears After Step 18

19. Click on frame elements 1 and 3 to select them.

20. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.



21. Verify that this dialog box is filled out as shown in the adjacent figure and click the **OK** button.

22. Click the **Draw Frame Element** button on the side toolbar or select **Draw Frame Element** from the **Draw** menu.

23. Click on joint 9 and then double click on joint 13 to draw a frame element.

24. Click on joint 8 and then double click on joint 12 to draw a frame element.

25. Click on joint 7 and then double click on joint 11 to draw a frame element.

26. Click on joint 6 and then double click on joint 10 to draw a frame element.

27. Click on joint 13 and then double click on joint 2 to draw a frame element.

28. Click on joint 12 and then double click on joint 9 to draw a frame element.

29. Click on joint 11 and then double click on joint 8 to draw a frame element.

30. Click on joint 10 and then double click on joint 7 to draw a frame element.

31. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

32. Click in the 3D View window to activate it.

33. From the **View** menu select **Refresh View** to rescale the view.

34. Click in the Window labeled X-Z Plane @ Y=0 to activate it.

35. Click the **Select All** button on the side toolbar to select all elements.

36. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

37. In this dialog box:

    - Select the Mirror Tab.
    - In the Mirror About area select Y-Z plane.
    - In the Ordinate area type **10.5** in the X edit box.

- Click the **OK** button to proceed with the replication.

38. Click the **Draw Frame Element** button on the side toolbar or select **Draw Frame Element** from the **Draw** menu.

39. Click on joint 4 and then double click on joint 16 to draw a frame element.

    *Note: If the font size is too small for you to read the joint labels use the following procedure to increase the font size. From the **Options** menu select **Preferences**, click on the Dimensions Tab if it is not already visible, type in a new (larger) font size in the Minimum Graphic Font Size edit box (usually about 6 points is sufficient), click the **OK** button and then click the **Refresh Window** button on the main toolbar.*

    Note: If you still have difficulty reading a particular joint label you can always right click the joint to bring up a dialog box that gives you information about the joint.

40. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

41. Click the **Select All** button on the side toolbar to select all elements.

42. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

43. In this dialog box:

    - Select the Radial Tab.
    - In the Rotate About area select the Z Axis option.
    - In the Increment Data area verify that the Angle is 90 and the Number is 1.
    - Click the **OK** button to proceed with the replication.

44. Click the **Restore Previous Selection** button on the side toolbar.

45. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

46. In this dialog box:

    - Verify the Linear Tab is selected.
    - In the Distance area type **21** in the Y edit box.
    - Verify that 0 is entered in the X and Z edit boxes.
    - Verify that 1 is entered in the Number edit box.
    - Click the **OK** button to proceed with the replication.

47. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.

48. Click the **yz 2D View** button to change the view to an Y-Z elevation. Note that the title of the window changes to Y-Z Plane @ X=0.

49. Select all of the elements in the Y-Z plane @ X=0 by "windowing".

50. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

51. In this dialog box:

    - Verify the Linear Tab is selected.
    - In the Distance area type **21** in the X edit box.
    - In the Distance area type **0** in the Y edit box.
    - Verify that 0 is entered in the Z edit box.
    - Verify that 1 is entered in the Number edit box.
    - Click the **OK** button to proceed with the replication.

52. Click the **xy 2D View** button to change the view to an X-Y plan. Note that the title of the window changes to X-Y Plane @ Z=0.

53. Select the four joints at this level either by "windowing" or by clicking on them individually.

54. From the **Assign** menu, choose **Joint**, and then **Restraints...**from the submenu. This will display the Joint Restraints dialog box.

55. In this dialog box:

    - Verify that the Translation 1, Translation 2 and Translation 3 boxes are checked.
    - Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.
    - Click the **OK** button.

56. Click the **Show Undeformed Shape** button to reset the window display from joint restraints to undeformed geometry. Note that the window title changes.

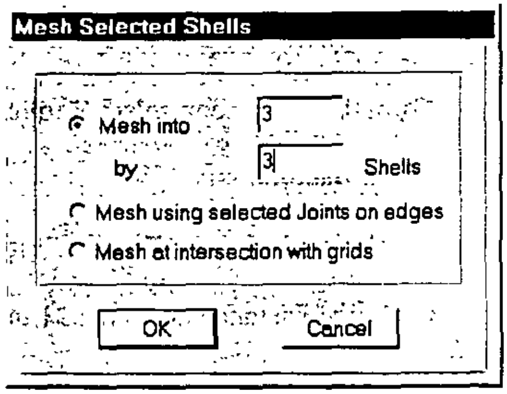
57. Click the **Up One Gridline** button on the main toolbar to display the elevation view at Z=25.

58. Click the **Draw Rectangular Shell Element** button on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.

59. Click on joint 32 and then joint 14 to draw a shell element over the entire structure.

60. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

61. Click on the shell element to select it.



62. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

63. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.

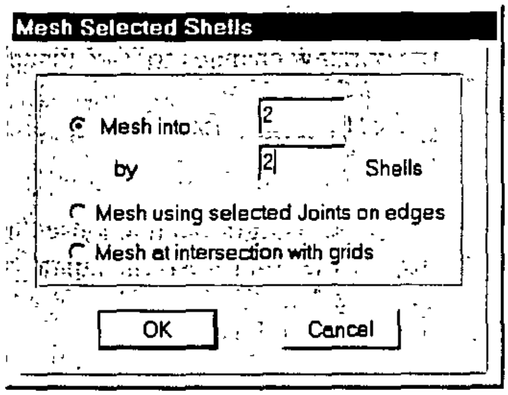
64. Click the **Up One Gridline** button on the main toolbar to display the elevation view at Z=37.

65. Click the **Draw Rectangular Shell Element** button on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.

66. Click on joint 33 and then joint 15 to draw a shell element over the entire structure.

67. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

68. Click on the shell element to select it.



69. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

70. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.

71. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

72. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

73. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.

74. In this dialog box:

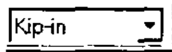
    - Type **DL** in the Load edit box.
    - Click the **Change Load** button

- Type **LL** in the Load edit box.
- Select LIVE from the Type drop-down box.
- Type **0** in the Self weight Multiplier box.
- Click the **Add New Load** button.
- Click the **OK** button.

75. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

76. In this dialog box:

- Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
- In this dialog box:
  - ➢ Verify that the Mass per Unit Volume is 4.658E-03.
  - ➢ Verify that the Weight per Unit Volume is 0.15.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

77. Click the drop down box in the status bar to change the units to kip-in. 

78. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

79. In this dialog box:

- Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
- In this dialog box:
  - ➢ Verify that the Modulus of Elasticity is 29000.
  - ➢ Verify that Poisson's ratio is 0.3.
  - ➢ Verify that the steel yield stress is 36.
  - ➢ Click the **OK** button twice to exit the dialog boxes.

80. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

81. In this dialog box:

- Click the drop-down box that says Import I/Wide Flange and select the Import Angle option.

- If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.

- A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

  - Scroll down and highlight the L4x4x3/4 by clicking on it.

  - Hold down the Shift key on the keyboard and click on the L4x4x1/4 angle. All of the L4x4 angles will now be selected (seven total).

  - Click the **OK** button twice to return to the Define Frame sections dialog box.

- Click the drop-down box that says Add I/Wide Flange and select the Add Auto Select option to display the Auto Selection Sections dialog box.

- In this dialog box:

  - Highlight all of the angles in the List of Sections list box by clicking on the top angle, pressing and holding down the shift key on the keyboard, and clicking on the bottom angle.

  - Click the **Add** button to add the angles to the Auto selections list box.

  - Click the **OK** button twice to exit all dialog boxes.

82. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

83. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

84. In this dialog box:

- Click the **Modify/Show Section** button to display the Shell Sections dialog box.

- In this dialog box:

  - Verify the Material specified is CONC.

  - In the Thickness area type **.6667** in both the Membrane and Bending edit boxes.

  - Verify that the Shell option is chosen in the Type area.

  - Click the **OK** button twice to exit all dialog boxes.

85. Click in the 3D View window to make sure it is active.

86. Click the **Select All** button on the side toolbar to select all elements.

87. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

88. In this dialog box:

    - Highlight the AUTO1 section.
    - Click the **OK** button.

89. Click the **Select All** button on the side toolbar to select all elements.

90. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

91. In this dialog box:

    - Verify DL is selected in the Load Case Name drop-down box.
    - In the Uniform Load area type **-.05** (50 psf) in the Load edit box.
    - In the Uniform Load area verify that the Dir item is set to Global Z.
    - Click the **OK** button.

92. Click the **Select All** button on the side toolbar to select all elements.

93. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

94. In this dialog box:

    - Select LL in the Load Case Name drop-down box.
    - In the Uniform Load area type **-.1** (100 psf) in the Load edit box.
    - Click the **OK** button.

95. Click the **Show Undeformed Shape** button to remove the display of the shell static loads.

96. Click in the window labeled X-Y Plane @ Z=37 to activate it.

97. Select all of the elements in the plan view by "windowing"

98. From the **Assign** menu select **Joints** and then **Constraints...** from the submenu to display the Constraints dialog box.

99. In this dialog box:

- In the Click To area click the drop-down box that says Add Body and select Add Diaphragm to display the Diaphragm Constraint dialog box.
- In this dialog box:
  - ➢ Type **ROOF** in the Constraint Name edit box.
  - ➢ Verify that the Z Axis option is selected in the Constraint Axis area.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

100. Click the **Show Undeformed Shape** button to remove the display of the joint constraints and reset the window display.

101. Click the **Down One Gridline** button to move to the X-Y Plane @ Z=25.

102. Select all of the elements in the plan view by "windowing"

103. From the Assign menu select Joints and the Constraints... from the submenu to display the Constraints dialog box.

104. In this dialog box:

- In the Click To area click the drop-down box that says Add Body and select Add Diaphragm to display the Diaphragm Constraint dialog box.
- In this dialog box:
  - ➢ Type **SECOND** in the Constraint Name edit box.
  - ➢ Verify that the Z Axis option is selected in the Constraint Axis area.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

105. Click the **Show Undeformed Shape** button to remove the display of the joint constraints and reset the window display.

106. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- Check the Dynamic Analysis check box.
- Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

- In this dialog box:
  - ➢ Type 3 in the Number of Modes edit box.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

107. From the **Options** menu select **Preferences...** to display the Preferences dialog box.

108. In this dialog box:
   - Select the Steel Tab.
   - Select AISC-ASD89 from the Steel Design Code drop-down box.
   - Click the **OK** button.

109. Click the **Run Analysis** button to run the analysis.

110. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

111. From the **Design** menu select **Start Design/Check of Structure** to initiate the design. The design proceeds and when it is complete P-M interaction ratios are displayed.

112. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

113. In this dialog box:
   - Check the Sections box in the Frames area.
   - Click the **OK** button. The sections chosen by the program are displayed.

   Note: You may want to zoom in using the Rubber Band Zoom button on the main toolbar to see the chosen sections better.

114. Click the **Show Undeformed Shape** button to remove the display of frame sections and interaction values.

115. If you have zoomed in for a better view of the chosen sections, then click the Restore Full View button on the main toolbar.

116. Click the **Select All** button on the side toolbar to select all elements.

117. From the **Design** menu select **Replace Auto W/ Optimal Sections** to update the frame sections from Auto sections to the chosen angle sizes. Click **OK** when it says it will unlock the model and asks if it is OK to update.

118. Click the **Run Analysis** button to run the analysis using the optimal sections.

119. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window. Note that the 3-D window now shows the first mode shape.

120. Click the **Start Animation** button [Start Animation], located in the status bar at the bottom of the SAP2000 window, to animate the mode shape.

121. Click the **Right Arrow** button [⇨], located in the status bar at the bottom of the screen, to view the second mode shape.

122. Click the **Right Arrow** button [⇨] again to view the third mode shape.

123. Click the **Stop Animation** button [Stop Animation], located in the status bar at the bottom of the SAP2000 window, to stop the mode shape animation.

124. From the **Design** menu select **Start Design/Check of Structure** to initiate a final design check of the structure based on the analysis results using the optimal sections. The design proceeds and when it is complete the final P-M interaction ratios are displayed.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "D"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

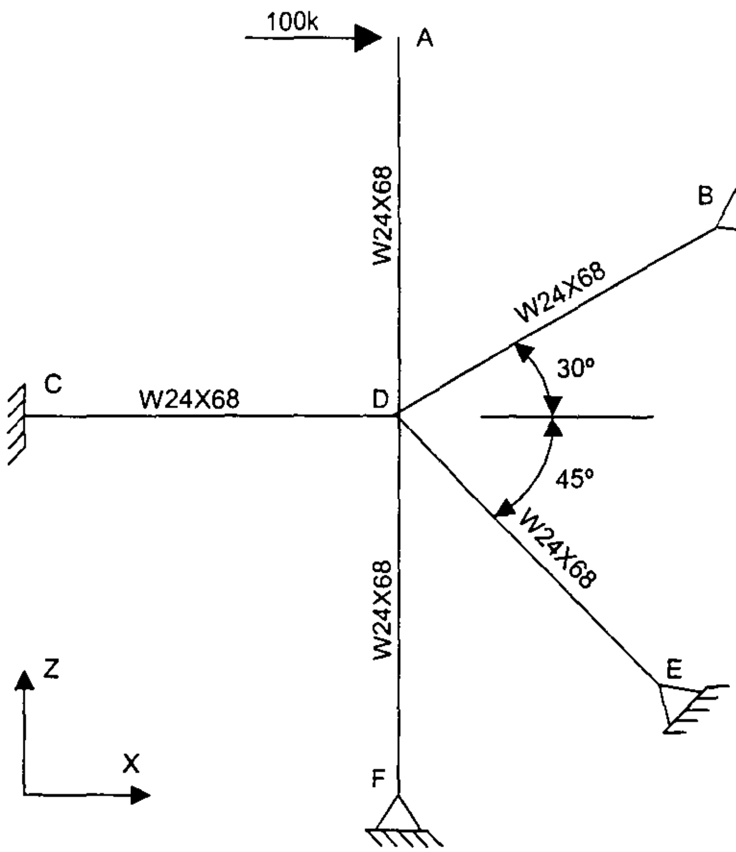
Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510 0573

Problem D

Inclined Supports

Steel
E =29000 ksi, Poissons Ratio = 0.3
All members are 10 feet long.

To Do
Determine support reactions.
Determine X-direction displacements
at joints A and B.

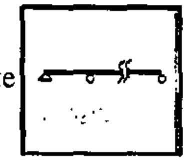


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem D Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

4. In this dialog box

   - Type **2** in the Number of Spans edit box.
   - Type **10** in the Span Length edit box.
   - Uncheck the Restraints box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Labels box in the Frames area.
   - Click the **OK** button.

8. Click the drop down box in the status bar to change the units to kip-in. Kip-in

9. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

10. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

11. In this dialog box:

    - Verify **29000** is entered in the Modulus of Elasticity edit box.
    - Verify **.3** is entered in the Poisson's Ratio edit box.
    - Accept the other default values.
    - Click the **OK** button to exit all dialog boxes.

12. Click the drop down box in the status bar to change the units to kip-ft. 

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

14. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

15. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.

16. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X68 section.
    - Click the **OK** button three times.

17. Select frame elements 1 and 2.

18. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

19. Highlight W24X68 in the Frame Sections area and click the **OK** button.

20. Click the **Show Undeformed Shape** button  to remove the displayed frame section assignments.

21. Select frame element 2 by clicking on it.

22. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

23. In this dialog box:

    - Click the Radial Tab.
    - Choose the Y Axis option in the Rotate about area. Note it will rotate about the Y-axis at the origin.
    - Type **45** in the Angle edit box in the Increment Data area.
    - Type **1** in the Number edit box in the Increment Data area.
    - Click the **OK** button.

24. Select frame element 2 by clicking on it.

25. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

26. In this dialog box:

    - Click the Radial Tab.
    - Type **90** in the Angle edit box in the Increment Data area.
    - Click the **OK** button.

27. Select frame element 2 by clicking on it.

28. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

29. In this dialog box:

    - Click the Radial Tab.
    - Type **270** in the Angle edit box in the Increment Data area.
    - Click the **OK** button.

30. Select frame element 2 by clicking on it.

31. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

32. In this dialog box:

    - Click the Radial Tab.
    - Type **330** in the Angle edit box in the Increment Data area.
    - Click the **OK** button.

33. Select frame element 2 by clicking on it.

34. Press the Delete key on the keyboard to delete this member.

35. Click the **Refresh Window** button to refresh the drawing.

36. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

37. In this dialog box:

    - Check the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

38. Select joint 4.

39. From the **Assign** menu select **Joint** and then **Local Axes...** from the submenu to display the Joint Local Axis dialog box.

40. In this dialog box:

    - Type **-45** in the about Y' edit box.

    - Press the **OK** button.

41. Select joint 7.

42. From the **Assign** menu select **Joint** and then **Local Axes...** from the submenu to display the Joint Local Axis dialog box.

43. In this dialog box:

    - Type **-120** in the about Y' edit box.

    - Press the **OK** button.

44. Select joint 1.

45. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

46. In this dialog box:

    - Check all six boxes in the Restraints in Local Directions area.

    - Click the **OK** button.

47. Select joints 4 and 5.

48. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

49. In this dialog box:

    - In the Restraints in Local Directions area uncheck the three Rotation boxes and leave the three Translation boxes checked.

    - Click the **OK** button.

50. Select joint 7.

51. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

52. In this dialog box:

    - In the Restraints in Local Directions area uncheck the Translation 3 box and leave the Translation 1 and Translation 2 boxes checked.

    - Click the **OK** button.

53. Select joint 6.

54. From the **Assign** menu select **Joint Static Loads...** and then **Forces**... from the submenu to display the Joint Forces dialog box.

55. In this dialog box:

    - Type **100** in the Force Global X edit box in the Loads area.

    - Click the **OK** button.

56. Click the **Show Undeformed Shape** button to remove the displayed joint force assignments.

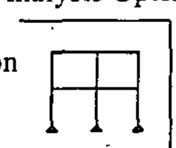
57. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

58. In this dialog box:

    - Uncheck the Labels box in the Joints area.

    - Click the **OK** button.

59. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.

    - Click the **OK** button.

60. Click the **Run Analysis** button to run the analysis.

61. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

62. Right click on the joints labeled A and B in the problem statement to see their displacements.

63. Click the **Joint Reaction Forces** button J to display the Joint Reaction Forces dialog box.

64. In this dialog box:

- Verify that the Reactions option is selected in the Type area.

- Click the **OK** button.

65. The reactions are displayed on the screen. You can right click on any joint to see the reactions at that joint or you can just read the reactions on the screen. If the text is too small to read, you can zoom in, or you can change the minimum font size as described in the note below.

    Note: To change the minimum font size select ***Preferences*** *from the* ***Options*** *menu and make sure the Dimensions Tab is selected. In the Minimum Graphic font Size edit box input a new size, maybe 5 or 6 points. Click the* ***OK*** *button.*

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "E"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510 0573

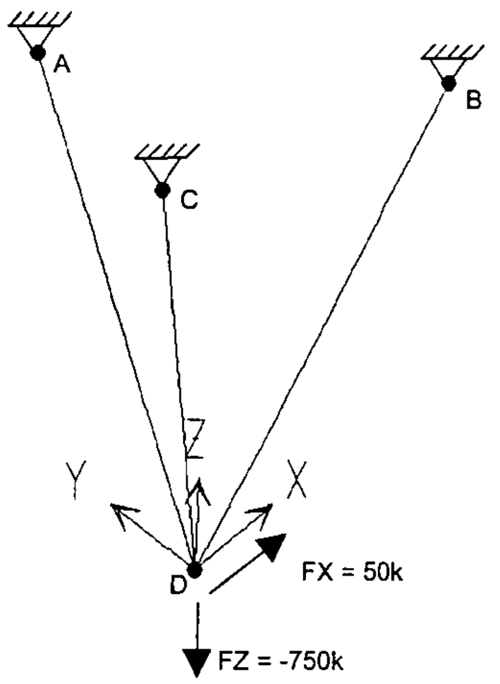
Problem E

Cables In Tension

Steel
E =29000 ksi, Poissons Ratio = 0.3
All members are 1.5" diameter steel cable

Joint Loads At Joint D:
F_X = 50 kips
F_Z = -750 kips

To Do
Determine the X-direction displacements at joint D with and without considering the stiffening affect of tension in the cables. Use P-Delta analysis to consider the stiffening affect.



Joint Coordinates (Feet)			
Joint	X	Y	Z
A	-3	-2	10
B	0	4	10
C	3	-2	10
D	0	0	0

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem E Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model....** This displays the Coordinate System Definition dialog box.

3. In this dialog box

   - Select the Cartesian Tab.
   - In the Number of Grid Spaces area type **2** in the X direction edit box.
   - In the Number of Grid Spaces area type **4** in the Y direction edit box.
   - In the Number of Grid Spaces area type **1** in the Z direction edit box.
   - In the Grid Spacing area type **3** in the X Direction edit box.
   - In the Grid Spacing area type **2** in the Y Direction edit box.
   - In the Grid Spacing area type **10** in the Z Direction edit box.
   - Click the **OK** button.

4. Click in the window titled X-Y Plane @ Z=10 to make sure it is active. The window is highlighted when it is active. The screen appears as shown in Figure E-1.

5. Click the **Draw Special Joint** button on the side toolbar or select **Add Special Joint** from the **Draw** menu.

6. Click on the grid intersection labeled "A" in Figure E-1 to enter a joint at (-3, -2, 10).

7. Click on the grid intersection labeled "B" in Figure E-1 to enter a joint at (0, 4, 10).

8. Click on the grid intersection labeled "C" in Figure E-1 to enter a joint at (3, -2, 10).

9. Click the **Down One Gridline** button to move to the X-Y Plane @ Z=0.

10. Click on the origin to enter a joint at (0, 0, 0).

11. Click in the window titled 3-D View to activate it. The screen now appears as shown in Figure E-2.

12. Click the **Draw Frame Element** button on the side toolbar or select **Draw Frame Element** from the **Draw** menu.

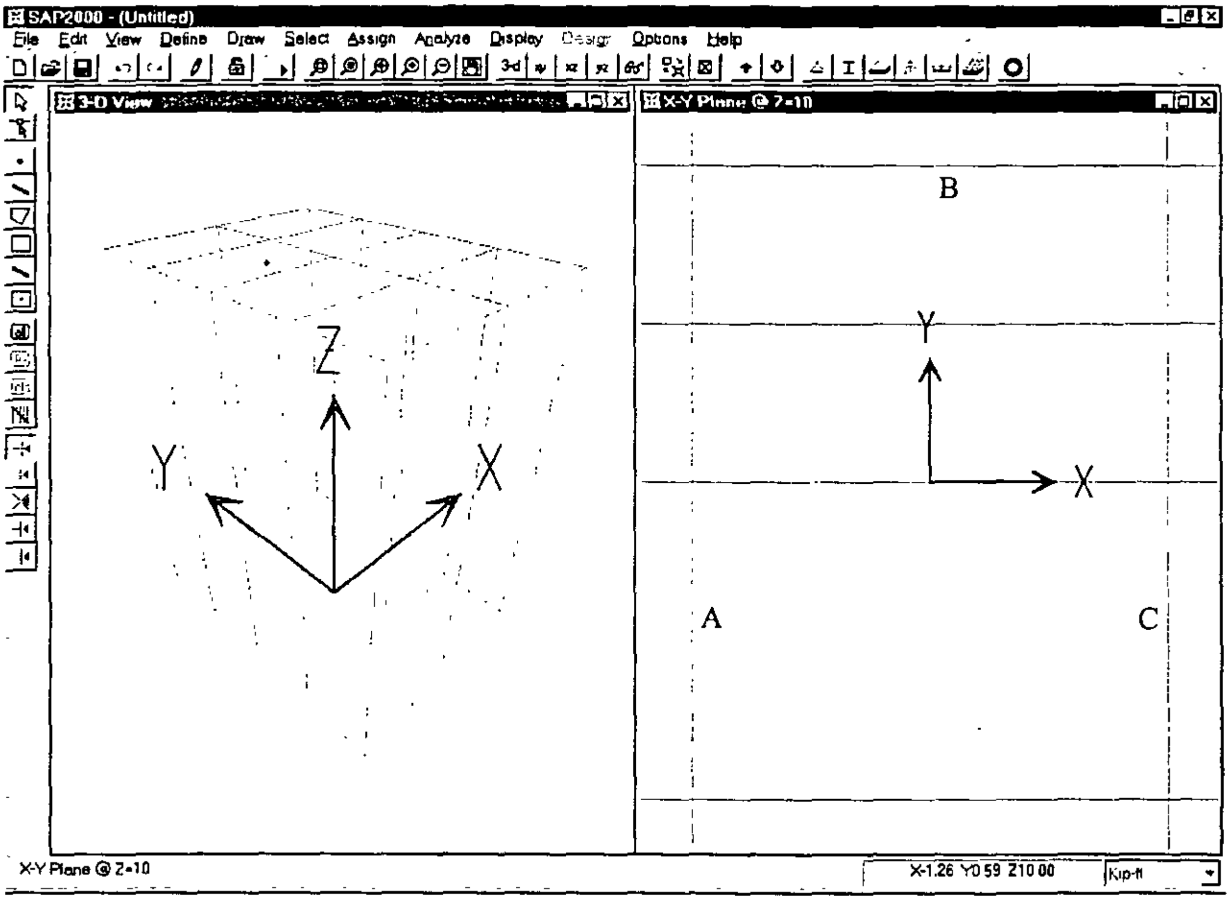


Figure E-1: Screen View Just Before Entering Joints At Elevation Z=10

13. Click on the point labeled "D" and then the point labeled "A" in Figure E-2 and press the Enter key on the keyboard to draw the first frame (cable) element.

14. Click on the point labeled "D" and then the point labeled "B" in Figure E-2 and press the Enter key on the keyboard to draw the next frame (cable) element.

15. Click on the point labeled "D" and then the point labeled "C" in Figure E-2 and press the Enter key on the keyboard to draw the last frame (cable) element.

16. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

17. Click on the joints identified as A, B and C in Figure E-2 to select them.

18. From the **Assign** menu, choose **Joint**, and then **Restraints...**from the submenu. This will display the Joint Restraints dialog box.

19. In this dialog box:

    - Verify that the Translation 1, Translation 2 and Translation 3 boxes are checked.

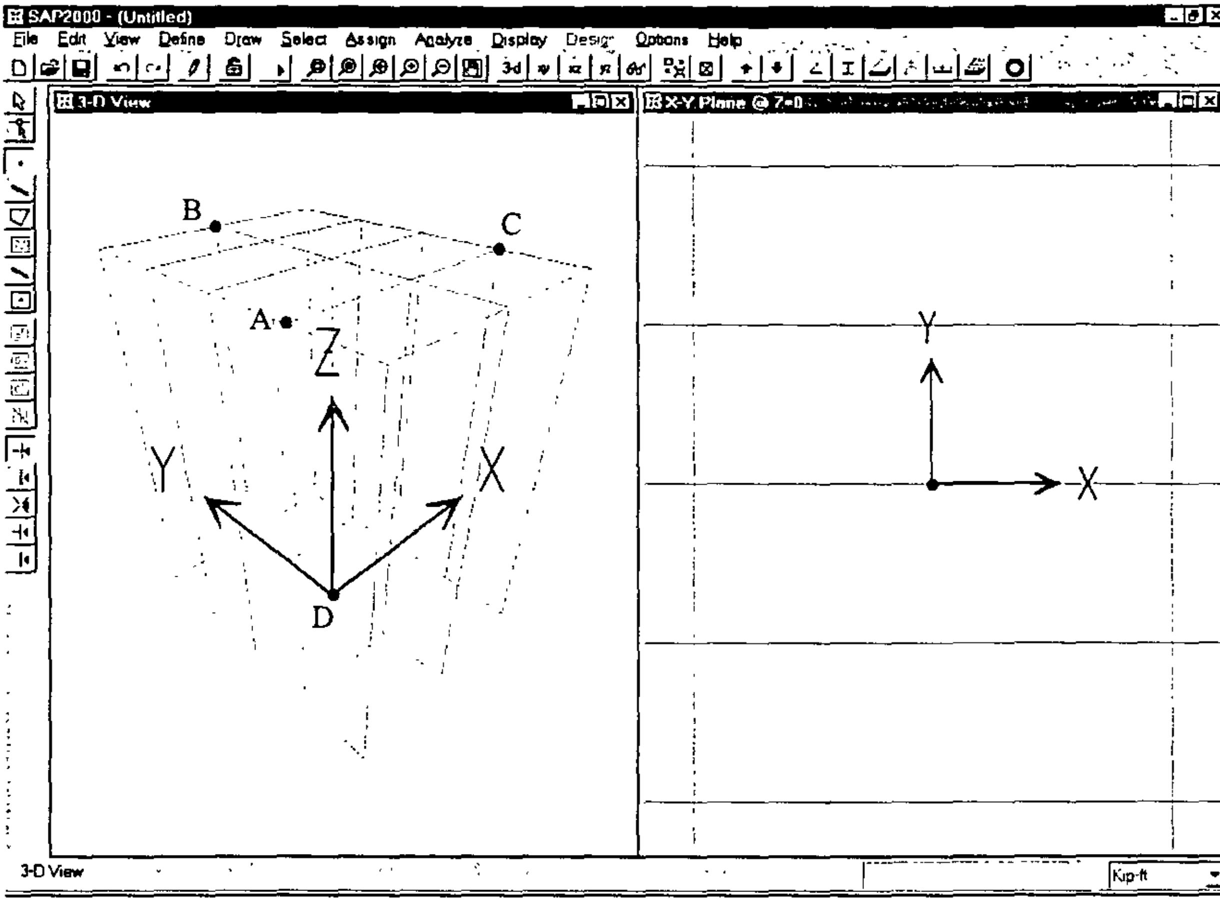


Figure E-2: Screen View After Joints Have Been Entered

- Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.

- Click the **OK** button.

20. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.

21. In this dialog box:

- Type **VERT** in the Load edit box.

- Type **0** in the Self weight Multiplier box.

- Click the **Change Load** button

- Type **LAT** in the Load edit box.

- Select QUAKE from the Type drop-down box.

- Click the **Add New Load** button.
- Click the **OK** button.

22. Select the joint identified as D in Figure E-2 by clicking on it.

23. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

24. In this dialog box:

    - Verify VERT is selected in the Load Case name drop-down box.
    - Type **-750** in the Force Global Z edit box.
    - Click the **OK** button.

25. Select the joint identified as D in Figure E-2 by clicking on it.

26. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

27. In this dialog box:

    - Select LAT from the Load Case name drop-down box.
    - Type **50** in the Force Global X edit box.
    - Type **0** in the Force Global Z edit box.
    - Click the **OK** button.

28. Click the **Show Undeformed Shape** button to remove the display of joint static loads.

29. Click the drop down box in the status bar to change the units to kip-in. Kip-in

30. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

31. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

32. In this dialog box:

    - Verify that the Modulus of Elasticity is 29000.
    - Verify that Poisson's Ratio is 0.3
    - Click the **OK** button twice to exit all dialog boxes.

33. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

34. In this dialog box:

    - Click the drop-down box that says Add I/Wide Flange and select the Add Circle option. This displays the Circle Section dialog box.

    - In this dialog box:

        - Type **CABLE** in the Section Name edit box.

        - Verify that the selected material in the Material drop-down box is STEEL.

        - Type **1.5** in the Diameter (t3) edit box.

        - Click the **OK** button twice to exit all dialog boxes.

35. Select the three frame elements in the 3-D View window by clicking on them.

36. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

37. In this dialog box:

    - Click on CABLE in the Frame Sections area to highlight it..

    - Click the **OK** button.

38. Click the **Show Undeformed Shape** button to remove the display of frame sections.

39. Click the **Run Analysis** button to run the analysis.

40. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

41. Click the drop down box in the status bar to change the units to kip-in. Kip-in

42. Click in the window with the 3-D View to make sure it is active.

43. Click the **Display Static Deformed Shape** button (or select **Show Deformed Shape...** from the **Display** menu). The Deformed Shape dialog box appears.

44. In this dialog box:

    - Select LAT Load Case from the Load drop-down box.

    - Click the **OK** button.

45. Right click on the bottom joint (the one labeled "D" in the problem statement) to see its displacement. Note the X-direction displacement of this joint. This is the displacement without considering the stiffening affect of the tension in the cables.

46. Click the **Lock/Unlock Model** button on the main toolbar to unlock the model. Click the **OK** button when asked if it is OK to delete.

47. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

48. In this dialog box:

    - Check the Include P-Delta check box.
    - Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
    - In this dialog box:
        - In the Iteration Controls area type **5** in the Maximum Iterations edit box.
        - Accept the other default values in the Iteration Controls area.
        - In the P-Delta Load Combination area verify that the Load Case list box says VERT and the Scale Factor list box says 1.
        - Click the **OK** button twice to exit all dialog boxes.

49. Click the **Run Analysis** button to run the analysis.

50. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

51. Click the drop down box in the status bar to change the units to kip-in. Kip-in

52. Click in the window with the 3-D View to make sure it is active.

53. Click the **Display Static Deformed Shape** button (or select **Show Deformed Shape...** from the **Display** menu). The Deformed Shape dialog box appears.

54. In this dialog box:

    - Select LAT Load Case from the Load drop-down box.
    - Click the **OK** button.

55. Right click on the bottom joint (the one labeled "D" in the problem statement) to see its displacement. Note the X-direction displacement of this joint. This is the displacement with the stiffening affect of the tension in the cables considered. Notice the difference between this displacement and that noted in step 45.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "F"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Problem F

Wall Resisting Hydrostatic Pressure

Concrete

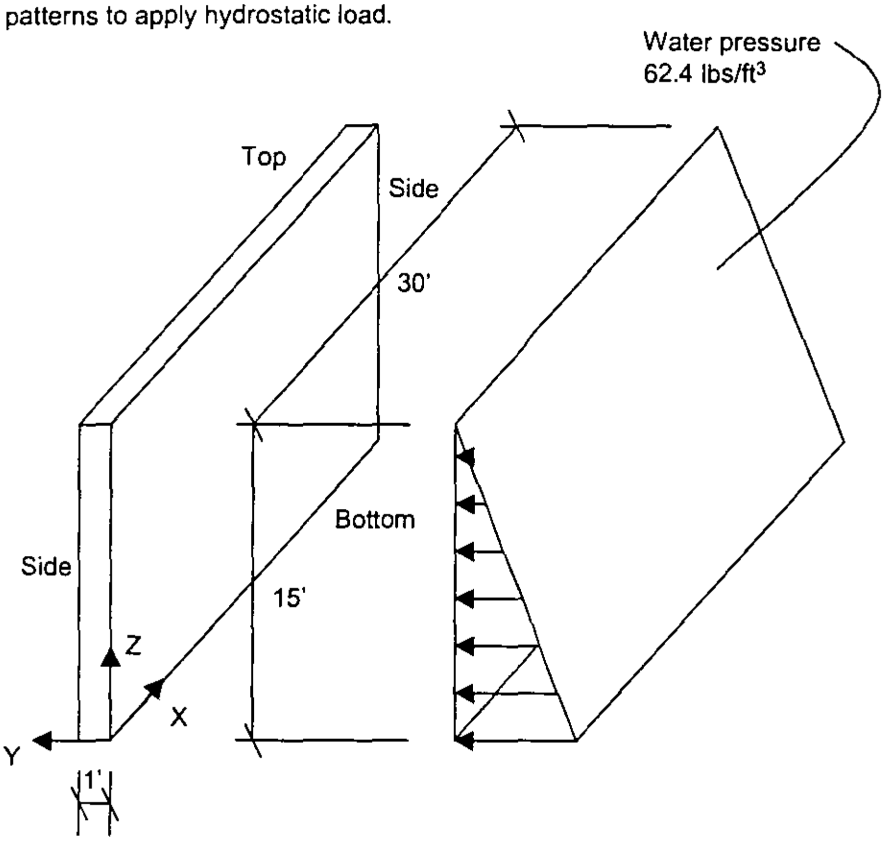
E =3600 ksi, Poissons Ratio = 0.2

Boundary Conditions

Case 1: Wall clamped at bottom only.
Case 2: Wall clamped at bottom and sides.

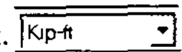
To Do

Determine maximum Y-direction displacements at top of wall for Case 1 and Case 2 support conditions. Use joint patterns to apply hydrostatic load.

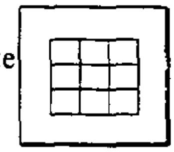


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem F Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

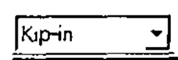
2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Shear Wall** template  button to display the Shear Wall dialog box.

4. In this dialog box

   - Type **30** in the Number of Spaces Along X edit box.
   - Type **15** in the Number of Spaces Along Z edit box.
   - Type **1** Space Width Along X edit box.
   - Type **1** Space Width Along Z edit box.
   - Click the **OK** button.

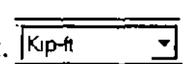
5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the drop down box in the status bar to change the units to kip-in. 

7. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

8. In this dialog box:

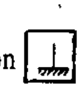
   - Verify that the modulus of elasticity is 3600 and poisson's ratio is 0.2.
   - Click the **OK** button twice to exit the dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-ft. 

10. Select all of the support joints at the bottom of the wall by "windowing".

11. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.

12. In this dialog box:

   - Click the fixed base fast restraint button  to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.

- Click the **OK** button.

13. From the **Define** menu select **Joint Patterns...** to display the Define Pattern Names dialog box.
14. In this dialog box:
15. Type **HYDRO** in the edit box in the Patterns area.
16. Click the **Add New Pattern Name** button
17. Click the **OK** button.
18. Click the **Select All** button on the side tool bar.
19. From the **Assign** menu select **Joint Patterns...** to display the Pattern Data dialog box.
20. In this dialog box:

    - Select HYDRO from the Pattern Name drop-down box.

      Note: Press the F1 key on the keyboard for context sensitive help on the dialog box illustrating the definition of the Constants. When finished reading the help, click the "X" in the top right-hand corner of the Help window to close it.

    - Type **-1** in the Constant C edit box.
    - Type **15** in the Constant D edit box.
    - Click the **OK** button.

21. Click the **Select All** button on the side tool bar.
22. From the **Assign** menu select **Shell Static Loads...** and select **Pressure...** from the submenu to display the Shell Pressure Loads dialog box.
23. In this dialog box:

    - Select the By Joint Pattern option.
    - Select HYDRO from the Pattern drop-down box.
    - Type **.0624** in the Multiplier edit box.
    - Click the **OK** button.

24. Click the **Show Undeformed Shape** button to remove the displayed joint force assignments.

25. Click the **Run Analysis** button to run the analysis.

26. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

27. Right click on the center joint at the top of the wall and note the Y-direction displacement.

28. Click the **Lock/Unlock Model** button and click the resulting **OK** button to unlock the model.

29. Select the joints along the sides of the model by "windowing" each side separately.

30. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.

31. In this dialog box:

    - Click the fixed base fast restraint button to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.

    - Click the **OK** button.

32. Click the **Run Analysis** button to run the analysis.

33. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

34. Right click on the center joint at the top of the wall and note the Y-direction displacement.

-5 [illegible]

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "G"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

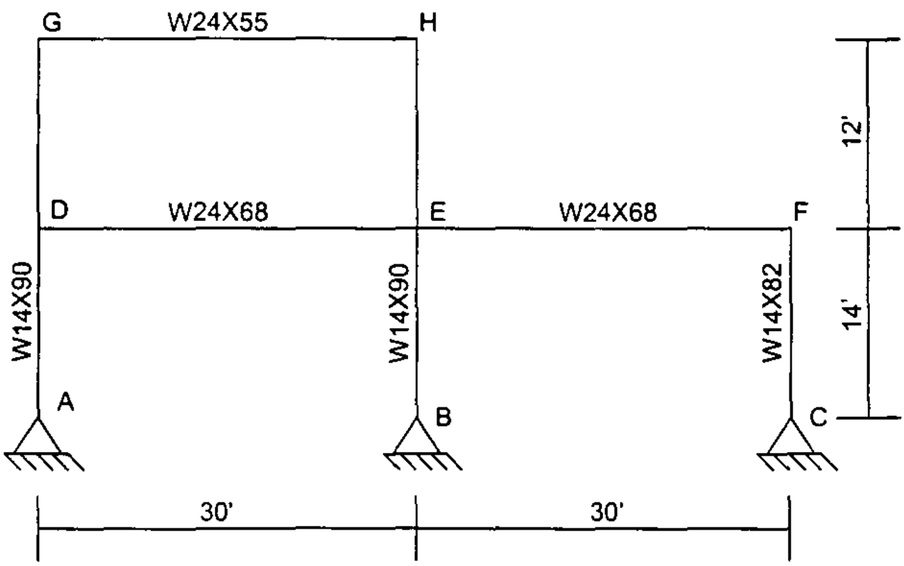
Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels. 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem G

Frame With Support Displacement

Steel
E =29000 ksi, Poissons Ratio = 0.3
Pinned base
All beam-column connections are rigid

To Do
Determine support reactions due to a
1" downward displacement of joint B.

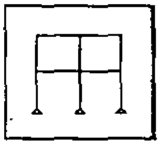


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem G Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box

   - Type **14** in the Story Height edit box.

   - Type **30** in the Bay Width edit box.

   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the drop down box in the status bar to change the units to kip-in. Kip-in

7. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

8. In this dialog box:

   - Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.

   - Click the **OK** button twice to exit the dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

10. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

11. In this dialog box:

    - Check the Labels box in the Frames area.

    - Click the **OK** button.

12. Select frame elements 6 and 10. Press the delete key on the keyboard to delete these elements.

13. Click the **Refresh Window** button to refresh the drawing.

14. From the **Draw** menu choose **Edit Grid...** to display the Modify Grid Lines dialog box.

15. In this dialog box:

    - Verify that the Z option is chosen in the Direction area.

    - Check the Glue Joints To Grid Lines box.

    - Click on the 28 grid line in the Z Location list box to highlight it. Note that the 28 appears in the Z Location edit box.

    - Change the 28 in the Z Location edit box to **26** and click the **Move Grid Line** button.

    - Click the **OK** button.

16. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

17. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

18. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.

19. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X68 section.

    - Scroll down to the W24X55 section, and click on it while holding down the Ctrl key on the keyboard.

    - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.

    - Click the **OK** button three times to exit all dialog boxes.

20. Select frame member 8.

21. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

22. In this dialog box:

    - Click on W24X55 in the Frame Sections area to highlight it.

    - Click the **OK** button.

23. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments so that you can see the frame element labels again.

24. Select frame members 7 and 9.

25. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

26. In this dialog box:

    - Click on W24X68 in the Frame Sections area to highlight it.
    - Click the **OK** button.

27. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments so that you can see the frame element labels again.

28. Select frame members 1 through 5.

29. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

30. In this dialog box:

    - Click on W14X90 in the Frame Sections area to highlight it.
    - Click the **OK** button.

31. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

32. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

33. In this dialog box:

    - Check the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

34. Click the drop down box in the status bar to change the units to kip-in. Kip-in

35. Select joint 4 (labeled B in the problem statement).

36. From the **Assign** menu select **Joint Static Loads...** and then **Displacements...** from the submenu to display the Ground Displacements dialog box.

37. In this dialog box:

    - Type **-1** in the Translation Z edit box.

    - Click the **OK** button.

38. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

39. Click the **Show Undeformed Shape** button to remove the displayed joint displacement assignments.

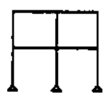
40. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

41. In this dialog box:

    - Uncheck the Labels box in the Joints area.

    - Click the **OK** button.

42. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.

    - Click the **OK** button.

43. Click the **Run Analysis** button to run the analysis.

44. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

45. Click the **Joint Reaction Forces** button **J** to display the Joint Reaction Forces dialog box.

46. In this dialog box:

    - Verify that the Reactions option is selected in the Type area.

    - Click the **OK** button.

47. The reactions are displayed on the screen. You can right click on any joint to see the reactions at that joint or you can just read the reactions on the screen. If the font size for the joint reactions shown on the screen is too small then read the note below.

    *Note: To change the minimum font size select **Preferences** from the **Options** menu and make sure the Dimensions Tab is selected. In the Minimum Graphic font Size edit box input a new size, maybe 5 or 6 points. Click the **OK** button.*

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

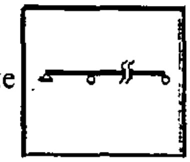
PROBLEM H SOLUTION

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Problem H Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

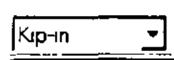
4. In this dialog box:
   - Accept all of the default values.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

7. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

8. In this dialog box:
   - Verify 0.15 is entered in the Weight per Unit Volume edit box.
   - Click the **OK** button twice to exit all dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-in. 

10. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

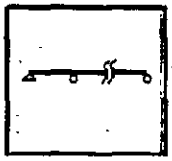
11. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

12. In this dialog box:
    - Verify 3600 is entered in the Modulus of Elasticity edit box.
    - Verify .2 is entered in the Poisson's Ratio edit box.
    - Verify 60 is entered in the Reinforcing Yield Stress, fy edit box.
    - Verify 4 is entered in the Concrete Strength, fc edit box.
    - Type **60** in the Shear steel Yield Stress, fys edit box.

Problem H Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

4. In this dialog box:

   - Accept all of the default values.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

7. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

8. In this dialog box:

   - Verify 0.15 is entered in the Weight per Unit Volume edit box.
   - Click the **OK** button twice to exit all dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-in. Kip-in

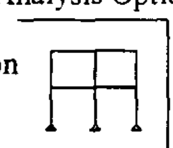
10. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

11. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

12. In this dialog box:

   - Verify 3600 is entered in the Modulus of Elasticity edit box.
   - Verify .2 is entered in the Poisson's Ratio edit box.
   - Verify 60 is entered in the Reinforcing Yield Stress, fy edit box.
   - Verify 4 is entered in the Concrete Strength, fc edit box.
   - Type **60** in the Shear steel Yield Stress, fys edit box.

- Type **0** in the Self weight Multiplier box.
- Click the **Add New Load** button.
- Click the **OK** button.

18. Select the two frame elements.
19. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
20. In this dialog box:
    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-2.2**.
    - Click the **OK** button.
21. Select the two frame elements.
22. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
23. In this dialog box:
    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-1.6**.
    - Click the **OK** button.
24. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
    - Click the **OK** button.
25. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
26. In this dialog box:
    - Select the CONC Tab.

- Type **0** in the Self weight Multiplier box.
- Click the **Add New Load** button.
- Click the **OK** button.

18. Select the two frame elements.

19. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

20. In this dialog box:

    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-2.2**.
    - Click the **OK** button.

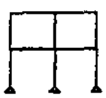
21. Select the two frame elements.

22. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

23. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-1.6**.
    - Click the **OK** button.

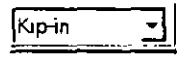
24. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
    - Click the **OK** button.

25. From the **Options** menu select **Preferences...** to display the Preferences dialog box.

26. In this dialog box:

    - Select the CONC Tab.

34. Click the drop down box in the status bar to change the units to kip-in. 

Note that the values for the area of longitudinal reinforcing steel are now in units of square inches.

35. From the **Design** menu select **Display Design Info...** to display the Display Design Results dialog box.

36. In this dialog box:

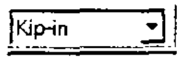
- Verify that the Design Output option is selected.
- Select Shear Reinforcing from the Design Output drop-down box.
- Click the **OK** button. The required shear reinforcing is displayed on the screen.

Note that the values for the shear reinforcing steel are reported as an area per unit length of element. Since the current units are kips and inches, the shear reinforcing reported is in square inches per inch.

37. Right click on the left beam to display the Concrete Design Information dialog box.

38. In this dialog box:

- Note that the required top and bottom longitudinal steel and the required shear steel is reported for each design load combination at each output segment along the beam.
- Click the **Details** button to see design details for the highlighted design load combination and output station location. The Concrete Design Information ACI 318-95 dialog box is displayed.
- When finished viewing the detailed information click the "X" in the upper right-hand corner of the Concrete Design Information ACI 318-95 dialog box to close it.
- Click **OK** to close the Concrete Design Information dialog box.

34. Click the drop down box in the status bar to change the units to kip-in. 

Note that the values for the area of longitudinal reinforcing steel are now in units of square inches.

35. From the **Design** menu select **Display Design Info...** to display the Display Design Results dialog box.

36. In this dialog box:

- Verify that the Design Output option is selected.
- Select Shear Reinforcing from the Design Output drop-down box.
- Click the **OK** button. The required shear reinforcing is displayed on the screen.

Note that the values for the shear reinforcing steel are reported as an area per unit length of element. Since the current units are kips and inches, the shear reinforcing reported is in square inches per inch.

37. Right click on the left beam to display the Concrete Design Information dialog box.

38. In this dialog box:

- Note that the required top and bottom longitudinal steel and the required shear steel is reported for each design load combination at each output segment along the beam.
- Click the **Details** button to see design details for the highlighted design load combination and output station location. The Concrete Design Information ACI 318-95 dialog box is displayed.
- When finished viewing the detailed information click the "X" in the upper right-hand corner of the Concrete Design Information ACI 318-95 dialog box to close it.
- Click **OK** to close the Concrete Design Information dialog box.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "I"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

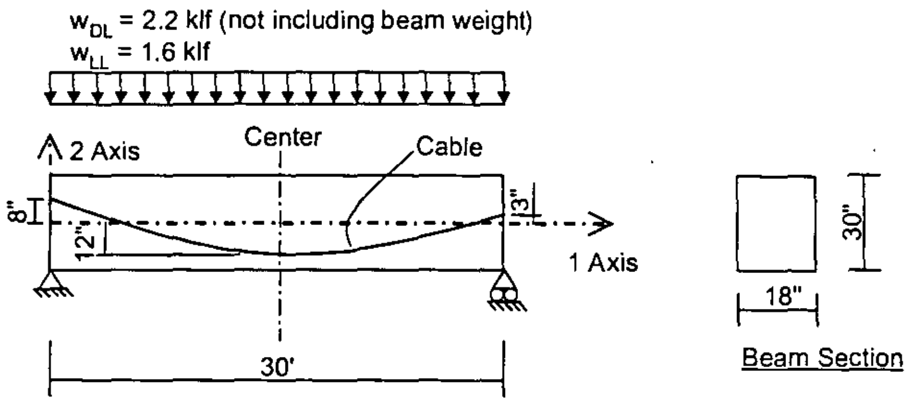
Palacio de Mineria, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels 5521 4021 al 24, 5623 2910 y 5623 2971 • Fax 5510 0573

Problem I

Prestressed Concrete Beam

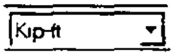
Concrete
E =4400 ksi, Poissons Ratio = 0.2
f'c = 6 ksi
Cable Tension = 200 kips

To Do
Determine the moment diagram for a DL + LL + PRESTRESS loading combination. Compare the results using 4 output segments and using 30 output segments.

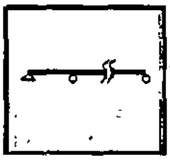


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem I Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

4. In this dialog box:

   - Type **1** in the Number of Spans edit box.
   - Type **30** in the Span Length edit box.
   - Click the **OK** button.

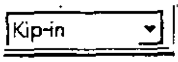
5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

7. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

8. In this dialog box:

   - Verify 0.15 is entered in the Weight per Unit Volume edit box.
   - Click the **OK** button twice to exit all dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-in. 

10. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

11. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

12. In this dialog box:

    - Type **4400** in the Modulus of Elasticity edit box.
    - Verify .2 is entered in the Poisson's Ratio edit box.
    - Verify 60 is entered in the Reinforcing Yield Stress, fy edit box.
    - Type **6** in the Concrete Strength, fc edit box.

- Type **60** in the Shear steel Yield Stress, fys edit box.
- Type **6** in the Concrete Shear Strength, fcs edit box.
- Accept the other default values.
- Click the **OK** button twice to exit all dialog boxes.

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

14. In this dialog box:

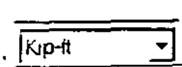
    - With the default section, FSEC1, highlighted, click the **Modify/Show Section** button to display the Rectangular Section dialog box.
    - In this dialog box:
        - Select CONC from the Material drop-down box.
        - Type **30** in the Depth (t3) edit box.
        - Type **18** in the Width (t2) edit box.
        - Click the **OK** button twice to exit all dialog boxes.

15. Select the frame element by clicking on it.

16. From the **Assign** menu select **Frame** and then **Prestress...** from the submenu to display the Frame Prestressing Patterns dialog box.

17. In this dialog box:

    - Type **200** in the Cable Tension edit box.
    - In the Cable Eccentricities area type **8** in the Start edit box.
    - In the Cable Eccentricities area type **12** in the Middle edit box.
    - In the Cable Eccentricities area type **3** in the End edit box.
    - Click the **OK** button.

18. Click the drop down box in the status bar to change the units to kip-ft. 

19. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.

20. In this dialog box:

- Type **DL** in the Load edit box.
- Click the **Change Load** button
- Type **LL** in the Load edit box.
- Select LIVE from the Type drop-down box.
- Type **0** in the Self weight Multiplier box.
- Click the **Add New Load** button.
- Type **PRESTRES** in the Load edit box.
- Select OTHER from the Type drop-down box.
- Click the **Add New Load** button.
- Click the **OK** button.

21. From the **Define** menu select **Load Combinations....** This will display the Define Load Combinations dialog box.

22. In this dialog box:

    - Click the **Add New Combo** button to display the Load Combination Data dialog box.
    - In this dialog box:
        - ➢ Accept the default load combination name, COMB1
        - ➢ Accept the default load combination type, Add.
        - ➢ Type **COMB1: DL + LL + Prestress** in the Title edit box.
        - ➢ Verify the DL Load Case is selected in the Case Name drop-down box.
        - ➢ Verify that 1 is entered in the Scale factor edit box.
        - ➢ Click the **Add** button.
        - ➢ Select LL Load Case from the Case Name drop-down box.
        - ➢ Click the **Add** button.
        - ➢ Select PRESTRES Load Case from the Case Name drop-down box.
        - ➢ Click the **Add** button.

- Click the **OK** button twice to exit all dialog boxes.

23. Select the frame element.

24. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

25. In this dialog box:

    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-2.2**.
    - Click the **OK** button.

26. Select the frame element.

27. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

28. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-1.6**.
    - Click the **OK** button.

29. Select the frame element.

30. From the **Assign** menu select **Frame Static Loads...** and then **Prestress...** from the submenu to display the Frame Prestress Loads dialog box.

31. In this dialog box:

    - Select PRESTRES from the Load Case Name drop-down box.
    - Type **1** in the Scale Factor edit box..
    - Click the **OK** button.

32. Select the frame element.

33. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

34. In this dialog box:

- Type **4** in the Number of Segments edit box.
- Click the **OK** button.

35. Click the **Show Undeformed Shape** button to remove the displayed frame output segment assignments.

36. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- In this dialog box click the **Plane Frame XZ Plane** button to set the available degrees of freedom.
- Click the **OK** button.

37. Click the **Run Analysis** button to run the analysis.

38. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

39. Click the **Member Force Diagram for Frames** button **F**, (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.

40. In this dialog box:

- Select COMB1 Combo from the Load drop-down box.
- Select the Moment 3-3 option in the Component area.
- Uncheck the Fill Diagram check box.
- Check the Show Values on Diagram check box.
- Click the **OK** button to display the moment diagram.

Note: You may want to print this moment diagram for comparison with the one obtained when 30 output segments are specified. To print the moment diagram select ***Print Graphics*** *from the* ***File*** *menu.*

Note: For load combinations, when force diagrams are plotted, exact values are only calculated at the ends of each output segment. These exact values are plotted and then they are connected with straight lines.

41. Click the **Lock/Unlock Model** button on the main toolbar to unlock the model. Click the **OK** button when asked if it is OK to delete.

42. Select the frame element.

43. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

44. In this dialog box:

    - Type **30** in the Number of Segments edit box.

    - Click the **OK** button.

45. Click the **Show Undeformed Shape** button to remove the displayed frame output segment assignments.

46. Click the **Run Analysis** button to run the analysis.

47. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

48. Click the **Member Force Diagram for Frames** button **F**, (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.

49. In this dialog box:

    - Verify that the COMB1 Combo is selected in the Load drop-down box.

    - Verify that the Moment 3-3 option is selected in the Component area.

    - Verify that the Show Values on Diagram check box is checked.

    - Click the **OK** button to display the moment diagram.

- ✓ Click the **OK** button twice to return to the Time History Display Definition dialog box.

- Verify that HIST1 is selected in the Time History Case drop-down box.
- Click on Joint 4 in the List of Functions area to highlight (select) it.
- Click the **Add** button to move Joint 4 to the Plot Functions area.
- Click the **Display** button to display the time history.
- Click the **OK** button to close the time history display and return to the Time History Display Definition dialog box.
- Select HIST2 in the Time History Case drop-down box.
- Click the **Display** button to display the time history.
- Click the **OK** button to close the time history display and return to the Time History Display Definition dialog box.
- Select HIST3 in the Time History Case drop-down box.
- Click the **Display** button to display the time history.
- Click the **OK** button to close the time history display and return to the Time History Display Definition dialog box.
- Click the **Done** button to close the Time History Display Definition dialog box.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "J"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

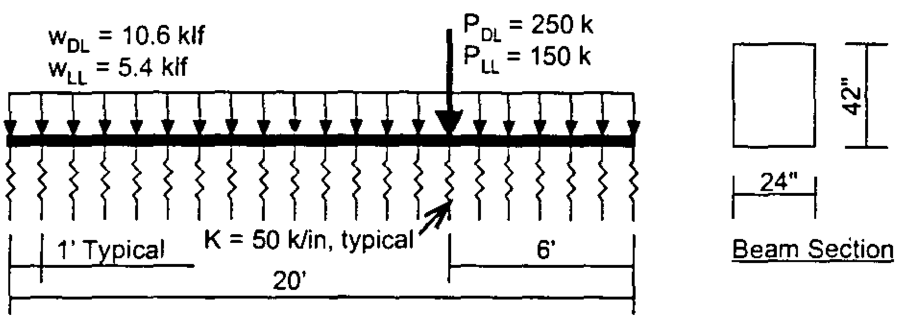
Problem J

Beam On Elastic Foundation

Concrete
E = 3120 ksi
Poissons Ratio = 0.2

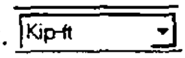
To Do
Determine the moment diagram under combined dead plus live loads and the maximum downward displacement.

Note: Dead load shown does not include beam self weight.

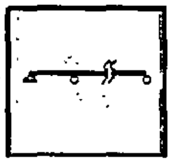


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem J Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

4. In this dialog box:

   - Type **1** in the Number of Spans edit box.
   - Type **20** in the Span Length edit box.
   - Uncheck the Restraints check box
   - Click the **OK** button.

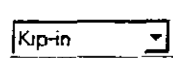
5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

7. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

8. In this dialog box:

   - Verify 0.15 is entered in the Weight per Unit Volume edit box.
   - Click the **OK** button twice to exit all dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-in. 

10. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

11. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

12. In this dialog box:

    - Type **3120** in the Modulus of Elasticity edit box.
    - Verify .2 is entered in the Poisson's Ratio edit box.
    - Accept the other default values.

- Click the **OK** button twice to exit all dialog boxes.

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

14. In this dialog box:

    - With the default section, FSEC1, highlighted, click the **Modify/Show Section** button to display the Rectangular Section dialog box.

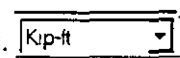
    - In this dialog box:

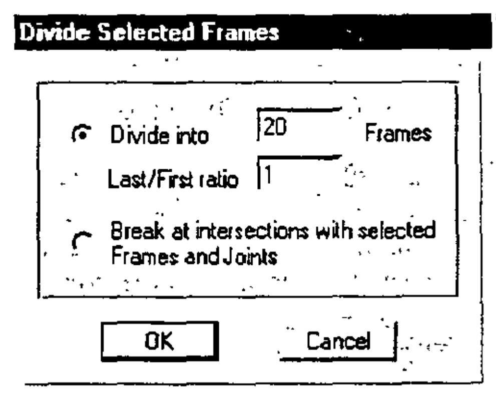
        - Select CONC from the Material drop-down box.

        - Type **42** in the Depth (t3) edit box.

        - Type **24** in the Width (t2) edit box.

        - Click the **OK** button twice to exit all dialog boxes.

15. Click the drop down box in the status bar to change the units to kip-ft. 



16. Select the frame element by clicking on it.

17. From the **Edit** menu select **Divide Frames...** to display the Divide Select frames dialog box.

18. Fill in the dialog box as shown in the adjacent figure and click the **OK** button.

19. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.

20. In this dialog box:

    - Type **DL** in the Load edit box.

    - Click the **Change Load** button

    - Type **LL** in the Load edit box.

    - Select LIVE from the Type drop-down box.

    - Type **0** in the Self weight Multiplier box.

    - Click the **Add New Load** button.

    - Click the **OK** button.

21. From the **Define** menu select **Load Combinations....** This will display the Define Load Combinations dialog box.

22. In this dialog box:

    - Click the **Add New Combo** button to display the Load Combination Data dialog box.
    - In this dialog box:
        - ➢ Accept the default load combination name, COMB1
        - ➢ Accept the default load combination type, Add.
        - ➢ Type **COMB1: DL + LL** in the Title edit box.
        - ➢ Verify the DL Load Case is selected in the Case Name drop-down box.
        - ➢ Verify that 1 is entered in the Scale factor edit box.
        - ➢ Click the **Add** button.
        - ➢ Select LL Load Case from the Case Name drop-down box.
        - ➢ Click the **Add** button.
        - ➢ Click the **OK** button twice to exit all dialog boxes.

23. Select all of the frame elements by "windowing".

24. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

25. In this dialog box:

    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-10.6.**
    - Click the **OK** button.

26. Click the **Restore Previous Selection** button  on the side toolbar (or select Get Previous Selection from the Select menu).

27. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

28. In this dialog box:

- Select LL from the Load Case Name drop-down box.
- In the Uniform Load area type **-5.4**.
- Click the **OK** button.

29. Click the **Show Undeformed Shape** button to remove the displayed frame uniform load assignments.

30. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

31. In this dialog box:

- Check the Labels box in the Joints area.
- Click the **OK** button.

32. Select joint 16 (6 feet from the right end) by clicking on it.

33. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

34. In this dialog box:

- Select DL from the Load Case Name drop-down box.
- Type **-250** in the Force Global Z edit box in the Loads area.
- Click the **OK** button.

35. Click the **Restore Previous Selection** button ps on the side toolbar (or select Get Previous Selection from the Select menu).

36. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

37. In this dialog box:

- Select LL from the Load Case Name drop-down box.
- Type **-150** in the Force Global Z edit box in the Loads area.
- Click the **OK** button.

38. Click the **Show Undeformed Shape** button to remove the displayed joint load assignments.

39. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

40. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Click the **OK** button.

41. Click the drop down box in the status bar to change the units to kip-in. Kip-in

42. Select all of the joint elements by "windowing".

43. From the **Assign** menu select **Joint** and then **Springs...** from the submenu to display the Joint Springs dialog box.

44. In this dialog box:

    - Type **50** in the Translation 3 edit box.
    - Click the **OK** button.

45. Click the **Show Undeformed Shape** button to remove the displayed frame output segment assignments.

46. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

47. In this dialog box:

    - Uncheck the UX, UY, RX and RZ check boxes leaving just the UZ and RY boxes checked.
    - Click the **OK** button.

48. Click the **Run Analysis** button to run the analysis.

49. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

50. Click the **Member Force Diagram for Frames** button **F**, (or select **Show Element Forces/Stresses** from the **Display** menu and then select **Frames...** from the submenu). The Member Force Diagram for Frames dialog box appears.

51. In this dialog box:

    - Select COMB1 Combo from the Load drop-down box.
    - Select the Moment 3-3 option in the Component area.

- Uncheck the Fill Diagram check box.
- Check the Show Values on Diagram check box.
- Click the **OK** button to display the moment diagram.

Note: If the font size is too small for you to read the moment values use the following procedure to increase the font size. From the ***Options*** *menu select* ***Preferences****, click on the Dimensions Tab if it is not already visible, type in a new (larger) font size in the Minimum Graphic Font Size edit box (usually about 6 points is sufficient), click the* ***OK*** *button and then click the* ***Refresh Window*** *button on the main toolbar.*

Note: You can right click on any of the frame elements to see details of the moment diagram for that element.

52. Click the drop down box in the status bar to change the units to kip-in. Kip-in

53. Click the **Display Static Deformed Shape** button (or select **Show Deformed Shape...** from the **Display** menu). The Deformed Shape dialog box appears.

54. In this dialog box:

    - Select COMB1 Combo from the Load drop-down box.
    - Click the **OK** button.

55. Right click on the joint at the far right end of the beam to view its deflection.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "K"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Mineria, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels 5521 4021 al 24, 5623.2910 y 5623.2971 • Fax 5510 0573

Problem K

Steel Moment Frame

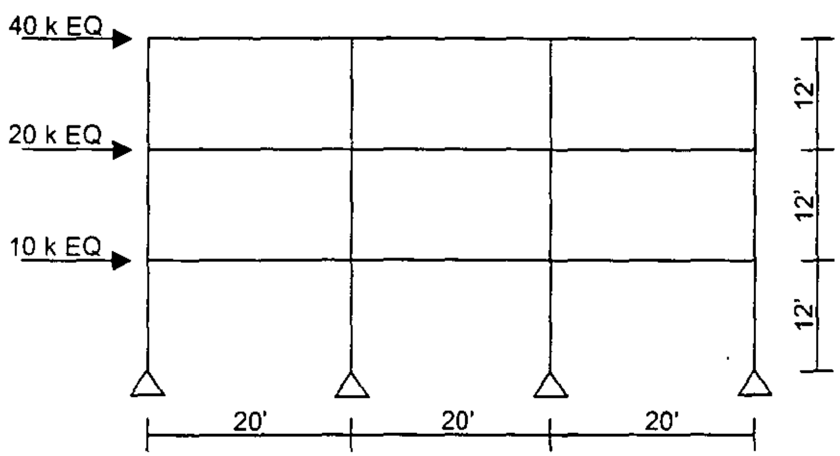
Steel
E =29000 ksi, Poissons Ratio = 0.3
Pinned base
All beam-column connections are rigid
Beams: W24X55, Fy = 36 ksi
Columns: W14X90, Fy = 36 ksi

Beam Span Loading On All Beams
1.0 klf Dead Load (not including steel frame member self weight)
0.5 klf Live Load

Lateral Loading (Earthquake)
As indicated in the figure

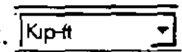
Unbraced Lengths
Assume columns are laterally supported at each floor level
Assume beams are braced at 10 feet on center

To Do
Determine stress ratios using AISC-ASD89 due to DL, LL and EQ loads.

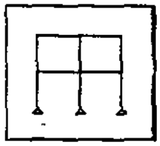


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem K Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

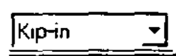
2. From the **File** menu select **New Model From Template...**. This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box

   - Type **3** in the Number of Stories edit box.
   - Type **3** in the Number of Bays edit box.
   - Accept the default value of 12 in the Story Height edit box.
   - Type **20** in the Bay Width edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

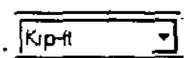
6. Click the drop down box in the status bar to change the units to kip-in. 

7. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

8. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

9. In this dialog box:

   - Verify that the Modulus of Elasticity is 29000.
   - Verify that Poisson's Ratio is 0.3
   - Verify that the Weight per Unit Volume is 2.830E-04.
   - Verify that the steel yield stress is 36.
   - Click the **OK** button twice to exit all dialog boxes.

10. Click the drop down box in the status bar to change the units to kip-ft. 

11. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

12. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

13. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.

14. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X55 section.

    - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.

    - Click the **OK** button three times to exit all dialog boxes.

15. Select all of the column elements by "windowing" each of the four column lines separately.

16. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

17. In this dialog box:

    - Click on W14X90 in the Frame Sections area to highlight it.

    - Click the **OK** button.

18. Select all of the beam elements by using the intersecting line selection method on each of the three beam bays separately.

19. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

20. In this dialog box:

    - Click on W24X55 in the Frame Sections area to highlight it.

    - Click the **OK** button.

21. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

22. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.

23. In this dialog box:

- Type **DL** in the Load edit box.
- Click the **Change Load** button
- Type **LL** in the Load edit box.
- Select LIVE from the Type drop-down box.
- Type **0** in the Self weight Multiplier box.
- Click the **Add New Load** button.
- Type **EQ** in the Load edit box.
- Select QUAKE from the Type drop-down box.
- Click the **Add New Load** button.
- Click the **OK** button.

24. Click the **Restore Previous Selection** button PS on the side toolbar (or select **Get Previous Selection** from the **Select** menu) to reselect the beam elements.

25. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

26. In this dialog box:

    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-1**.
    - Click the **OK** button.

27. Click the **Restore Previous Selection** button PS on the side toolbar (or select Get Previous Selection from the Select menu).

28. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

29. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-.5**.

- Click the **OK** button.

30. Click the **Show Undeformed Shape** button to remove the displayed frame uniform load assignments.

31. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

32. In this dialog box:

    - Check the Labels box in the Joints area.
    - Click the **OK** button.

33. Select joint 4 by clicking on it.

34. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

35. In this dialog box:

    - Select EQ from the Load Case Name drop-down box.
    - Type **40** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

36. Select joint 3 by clicking on it.

37. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

38. In this dialog box:

    - Type **20** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

39. Select joint 2 by clicking on it.

40. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

41. In this dialog box:

    - Type **10** in the Force Global X edit box in the Loads area.
    - Click the **OK** button.

42. Click the **Show Undeformed Shape** button to remove the displayed joint load assignments.

43. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

44. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Click the **OK** button.

45. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button to set the available degrees of freedom.
    - Click the **OK** button.

46. Click the **Run Analysis** button to run the analysis.

47. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

48. Click the **Show Undeformed Shape** button to reset the displayed deformed shape.

49. Select all of the beam elements by using the intersecting line selection method on each of the three beam bays separately.

50. From the **Design** menu select **Redefine Element Design Data** to display the Element Overwrite Assignments dialog box.

51. In this dialog box:

    - Check the Unbraced Length Ratio (Minor, LTB) check box.
    - Type **.5** in the Unbraced Length Ratio (Minor, LTB) edit box.
    - Click the **OK** button.

52. From the **Options** menu select **Preferences...** to display the Preferences dialog box.

53. In this dialog box.

    - Click on the Steel Tab
    - Select AISC-ASD89 from the Steel Design Code drop-down box if it is not already selected.

- Click the **OK** button.

54. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.

55. When the design check completes, the stress ratios are displayed.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "L"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col Centro, México D.F., APDO Postal M-2285 • Tels. 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax. 5510.0573

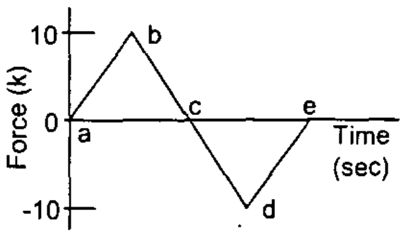
Problem L

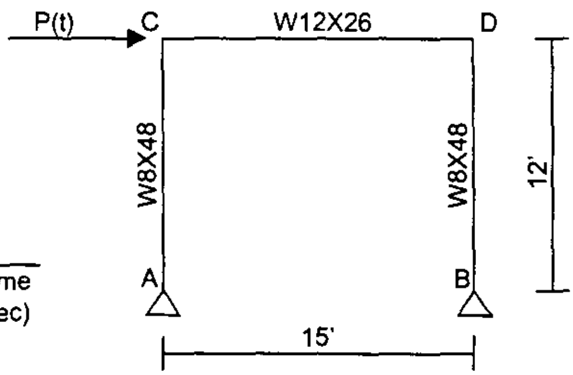
Periodic Loading

Steel
E =29000 ksi, Poissons Ratio = 0.3
Pinned base
All beam-column connections are rigid

Joint Masses
Lumped mass at joints C and D is 0.02 kip-sec^2 / in

Loading
The load P(t), applied to joint C, is a periodic load. Three different loading cases (functions) are defined for P(t). The three loading functions, which have periods of 0.25, 0.50 and 1.00 seconds respectively, are shown in the chart and graph below. Assume 5% damping for all loading.



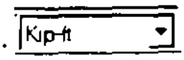


Point	Force (k)	Time Function 1 (sec)	Time Function 2 (sec)	Time Function 3 (sec)	
a	0	0	0	0	Note: The period of time functions 1, 2 and 3 is 0.25, 0.5 and 1 seconds respectively.
b	10	0.0625	0.125	0.25	
c	0	0.125	0.25	0.5	
d	-10	0.1875	0.375	0.75	
e	0	0.25	0.5	1	

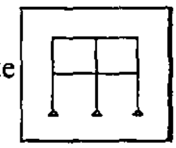
To Do
1. Verify natural period of structure is approximately 0.50 seconds.
2. Determine displacement at joint D for the three periodic functions.

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem L Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

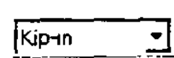
2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box:

   - Type **1** in the Number of Stories edit box.
   - Type **1** in the Number of Bays edit box.
   - Type **15** in the Bay Width edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the drop down box in the status bar to change the units to kip-in. 

7. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

8. In this dialog box:

   - Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.
   - Click the **OK** button twice to exit the dialog boxes.

9. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

10. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

11. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.

12. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

- Scroll down and click on the W12X26 section.
- Scroll down to the W8X48 section, and click on it while holding down the Ctrl key on the keyboard.
- Click the **OK** button three times to exit all dialog boxes.

13. Select the beam element.
14. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
15. In this dialog box:
    - Click on W12X26 in the Frame Sections area to highlight it.
    - Click the **OK** button.
16. Select the two column elements.
17. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.
18. In this dialog box:
    - Click on W8X48 in the Frame Sections area to highlight it.
    - Click the **OK** button.
19. Select the joints labeled C and D in the problem statement.
20. From the **Assign** menu select **Joint** and then **Masses...** from the submenu to display the Joint Masses dialog box.
21. In this dialog box:
    - Type **.02** in the Direction 1 edit box.
    - Type **.02** in the Direction 3 edit box.
    - Click the **OK** button.
22. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft
23. Click the **Show Undeformed Shape** button to remove the displayed joint mass assignments.
24. Select the joint labeled C in the problem statement.

25. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

26. In this dialog box:

    - Type **1** in the Force Global X edit box.
    - Click the **OK** button.

27. Click the **Show Undeformed Shape** button to remove the displayed joint force assignments.

28. From the **Define** menu select **Time History Functions...** to display the Define Time History Functions dialog box.

29. In this dialog box:

    - Click the **Add New Function** button to display the Function Definition dialog box.
    - In this dialog box:
        - ➢ Accept the default FUNC1 function name.
        - ➢ Click the **Add** button.
        - ➢ Type **.0625** in the Time edit box, type **10** in the Value edit box, and click the **Add** button.
        - ➢ Type **.125** in the Time edit box, type **0** in the Value edit box, and click the **Add** button.
        - ➢ Type **.1875** in the Time edit box, type **-10** in the Value edit box, and click the **Add** button.
        - ➢ Type **.25** in the Time edit box, type **0** in the Value edit box, and click the **Add** button.
        - ➢ Click the **OK** button to return to the Define Time History Functions dialog box.
    - Click the **Add New Function** button to display the Function Definition dialog box.
    - In this dialog box:
        - ➢ Accept the default FUNC2 function name.
        - ➢ Click the **Add** button.
        - ➢ Type **.125** in the Time edit box, type **10** in the Value edit box, and click the **Add** button.

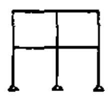
- Type **.25** in the Time edit box, type **0** in the Value edit box, and click the **Add** button.
- Type **.375** in the Time edit box, type **-10** in the Value edit box, and click the **Add** button.
- Type **.5** in the Time edit box, type **0** in the Value edit box, and click the **Add** button.
- Click the **OK** button to return to the Define Time History Functions dialog box.

- Click the **Add New Function** button to display the Function Definition dialog box.
- In this dialog box:
  - Accept the default FUNC3 function name.
  - Click the **Add** button.
  - Type **.25** in the Time edit box, type **10** in the Value edit box, and click the **Add** button.
  - Type **.5** in the Time edit box, type **0** in the Value edit box, and click the **Add** button.
  - Type **.75** in the Time edit box, type **-10** in the Value edit box, and click the **Add** button.
  - Type **1** in the Time edit box, type **0** in the Value edit box, and click the **Add** button.
  - Click the **OK** button twice to exit all of the dialog boxes.

30. From the **Define** menu select **Time History Cases...** to display the Define Time History Cases dialog box.
31. In this dialog box:
    - Click the **Add New History** button to display the Time History Case Data dialog box.
    - In this dialog box:
      - Accept the default History Case Name, HIST1.
      - Select Periodic from the Analysis Type drop-down box.
      - Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.
      - In this dialog box:

    - ✓ Type **.05** in the Damping For All Modes edit box.
    - ✓ Click the **OK** button.
  - ➢ Type **25** in the Number of Output Time Steps edit box.
  - ➢ Type **.01** in the Output Time Step Size edit box.
  - ➢ Check the Envelopes check box.
  - ➢ In the Load drop-down box, select LOAD1.
  - ➢ In the Function drop-down box, select FUNC1.
  - ➢ Click the **Add** button.
  - ➢ Click the **OK** button to return to the Define Time History Cases dialog box.
- Click the **Add New History** button to display the Time History Case Data dialog box.
- In this dialog box:
  - ➢ Accept the default History Case Name, HIST2.
  - ➢ Select Periodic from the Analysis Type drop-down box.
  - ➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.
  - ➢ In this dialog box:
    - ✓ Type **.05** in the Damping For All Modes edit box.
    - ✓ Click the **OK** button.
  - ➢ Type **50** in the Number of Output Time Steps edit box.
  - ➢ Type **.01** in the Output Time Step Size edit box.
  - ➢ Check the Envelopes check box.
  - ➢ In the Load drop-down box, select LOAD1.
  - ➢ In the Function drop-down box, select FUNC2.
  - ➢ Click the **Add** button.
  - ➢ Click the **OK** button to return to the Define Time History Cases dialog box.

- Click the **Add New History** button to display the Time History Case Data dialog box.
- In this dialog box:
  - ➢ Accept the default History Case Name, HIST3.
  - ➢ Select Periodic from the Analysis Type drop-down box.
  - ➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.
  - ➢ In this dialog box:
    - ✓ Type **.05** in the Damping For All Modes edit box.
    - ✓ Click the **OK** button.
  - ➢ Type **100** in the Number of Output Time Steps edit box.
  - ➢ Type **.01** in the Output Time Step Size edit box.
  - ➢ Check the Envelopes check box.
  - ➢ In the Load drop-down box, select LOAD1.
  - ➢ In the Function drop-down box, select FUNC3.
  - ➢ Click the **Add** button.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

32. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
    - Check the Dynamic Analysis check box, if it is not already checked.
    - Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.
    - In this dialog box:
      - ➢ Type **4** in the Number of Modes edit box.
      - ➢ Click the **OK** button twice to exit all dialog boxes.

33. Click the **Run Analysis** button to run the analysis.

34. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Note in the messages that the first mode period is about 0.5 seconds. Click the **OK** button to close the Analysis window.

35. Note again in the window title on the screen that the first mode period is about .5 seconds.

36. From the **Display** menu select **Set Output Table Mode...** to display the Select Output dialog box.

37. In this dialog box:

    - Click on the HIST1 history to highlight it.

    - Hold down the shift key on the keyboard and click on the HIST3 history. The HIST1, HIST2, and HIST3 histories should all be highlighted (selected) now.

    - Click the **OK** button.

38. Right click on the joint labeled D in the problem statement to display a table of envelope values for the displacement at joint D. Note that the maximum displacement occurs for HIST2, as would be expected since the period of Function 2 is close to the first mode period of the structure.

39. Click the "X" in the upper right-hand corner of the table to close it.

40. We have viewed the envelopes of the joint displacement at joint D. Now we will view the time histories of the displacement. Select the joint labeled D in the problem statement.

41. From the **Display** menu select **Show Time History Traces...** to display the Time History Display Definition dialog box.

42. In this dialog box:

    - Click the **Define Functions** button to display the Time History Functions dialog box.

    - In this dialog box:

        - ➢ Highlight Joint 4.

        - ➢ Click the **Modify/Show TH Function** button to display the Time History Joint Function dialog box.

        - ➢ In this dialog box:

            - ✓ Verify that the Displ option is selected in the Vector Type area.

            - ✓ Verify that the UX option is selected in the Component area.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "M"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem M

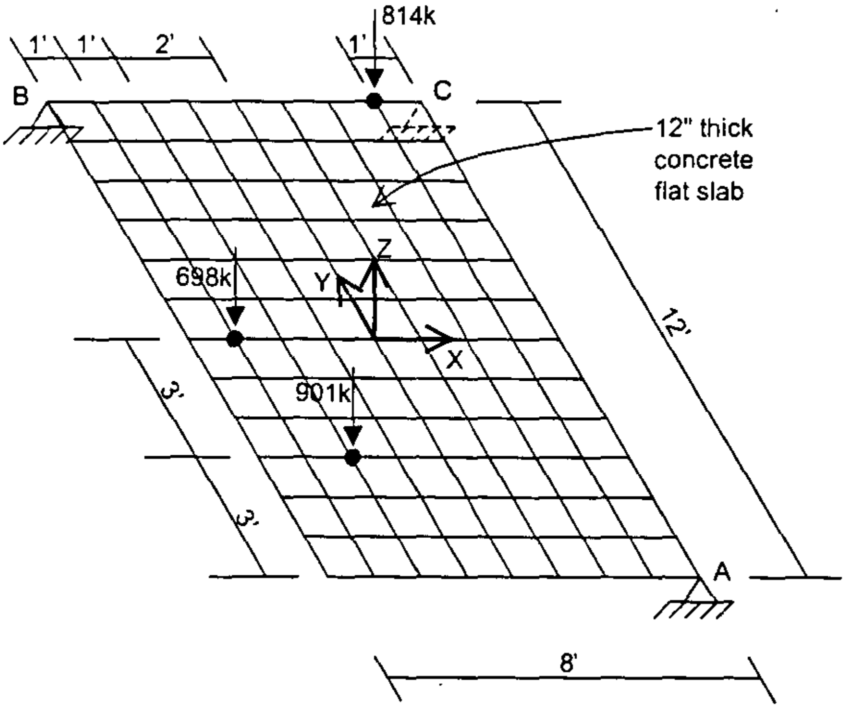
Flat Plate In The X-Y Plane With A Twist

Concrete
E =3600 ksi, Poissons Ratio = 0.2

Available Degrees of Freedom
UZ, RX, RY

Supports
Joints A, B and C have Z-direction supports, as shown.

To Do
Determine support reactions at joints A, B and C. Explain the apparently odd results for the reaction at joint C.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem M Solution

1. Click the drop down box in the status bar to change the units to kip-ft.

2. From the **File** menu select **New Model....** This displays the Coordinate System Definition dialog box.

3. In this dialog box

   - Select the Cartesian Tab.
   - In the Number of Grid Spaces area type **2** in the X direction edit box.
   - In the Number of Grid Spaces area type **2** in the Y direction edit box.
   - In the Number of Grid Spaces area type **0** in the Z direction edit box.
   - In the Grid Spacing area verify the X direction spacing is 4.
   - In the Grid Spacing area type **6** the Y direction edit box.
   - Click the **OK** button.

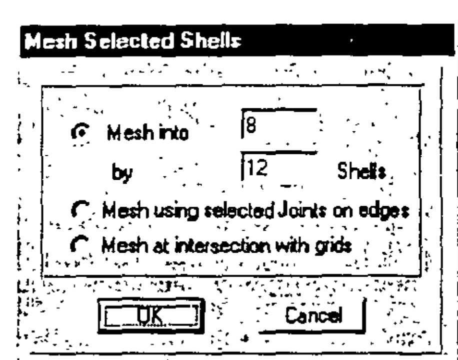
4. Click the "X" in the top right-hand corner of the 3-D View window to close it.

5. Verify that the Snap to Joints and Grid Points button on the side toolbar is depressed.

6. Click the **Draw Rectangular Shell Element** button on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.

7. Click on upper left-hand corner grid intersection (coordinates are (-4, 6, 0)) and then click on the lower right-hand grid intersection (coordinates are (4, -6, 0)) to draw a shell element over the entire structure.

8. Click the **Pointer** button to exit Draw Mode and enter Select Mode.



9. Click on the shell element to select it.

10. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

11. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.

12. Select the joints that are labeled "A", "B" and "C" in the problem statement.

13. From the **Assign** menu select **Joints** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

14. In this dialog box:

    - Uncheck the Translation 1 and Translation 2 check boxes.
    - Verify that the Translation 3 check box is checked.
    - Verify that the Rotation about 1, 2 and 3 check boxes are *not* checked.
    - Click the **OK** button.

15. Click the **Show Undeformed Shape** button to remove the display of joint restraints and reset the window display (and title).

16. Click the drop down box in the status bar to change the units to kip-in. Kip-in

17. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

18. Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

19. In this dialog box:

    - Verify that the Modulus of Elasticity is 3600.
    - Verify that the Poisson's Ratio is 0.2.
    - Click the **OK** button twice to exit the dialog boxes.

20. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

21. In this dialog box:

    - Click the Modify/Show Section button to display the Shell Sections dialog box.
    - In this dialog box:
        - ➢ Verify that the selected material is CONC.
        - ➢ Verify that both the Membrane and the Bending thicknesses are 12.
        - ➢ Verify that the Shell option is selected in the Type area.
        - ➢ Click the **OK** button twice to exit all dialog boxes.

22. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

23. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

24. In this dialog box:

    - Check the Labels box in the Joints area.
    - Click the **OK** button.

    Note: If the font size is too small for you to read the joint labels use the following procedure to increase the font size. From the Options menu select Preferences, click on the Dimensions Tab if it is not already visible, type in a new (larger) font size in the Minimum Graphic Font Size edit box (usually about 6 points is sufficient), click the OK button and then click the Refresh Window button on the main toolbar.

25. Select joint 106 (coordinates (3, 6, 0)) by clicking on it.

26. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

27. In this dialog box:

    - Type **-814** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.

28. Select joint 16 (coordinates (-3, 0, 0)) by clicking on it.

29. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

30. In this dialog box:

    - Type **-698** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.

31. Select joint 32 (coordinates (-2, -3, 0)) by clicking on it.

32. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

33. In this dialog box:

    - Type **-901** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.

34. Click the **Show Undeformed Shape** button to reset the window display.

35. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

36. In this dialog box:

- Uncheck the Labels box in the Joints area.
- Click the **OK** button.

37. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- In this dialog box click the **Plane Grid XY Plane** button to set the available degrees of freedom.
- Click the **OK** button.

38. Click the 3D View button 3-d on the main toolbar to switch to a 3-D View.

39. Click the **Run Analysis** button to run the analysis.

40. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

41. Click the **Joint Reaction Forces** button J on the main toolbar to display the Joint Reaction Forces dialog box.

42. In this dialog box:

- Verify that the Reactions option is selected in the Type area.
- Click the **OK** button and view the support reactions.

Note: The reaction at the joint labeled "C" in the problem statement is zero (0). The reason for this apparently odd result is that the resultant of all the applied loads lies on a line connecting the support points labeled "A" and "B", and thus by simple statics the reaction at support point "C" must be zero. Note that you could move the support point labeled "C" anywhere on the structure (except on the line connecting support points "A" and "B", since this would result in an unstable structure) and the resulting reactions would not change.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "N"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax. 5510.0573

Problem N

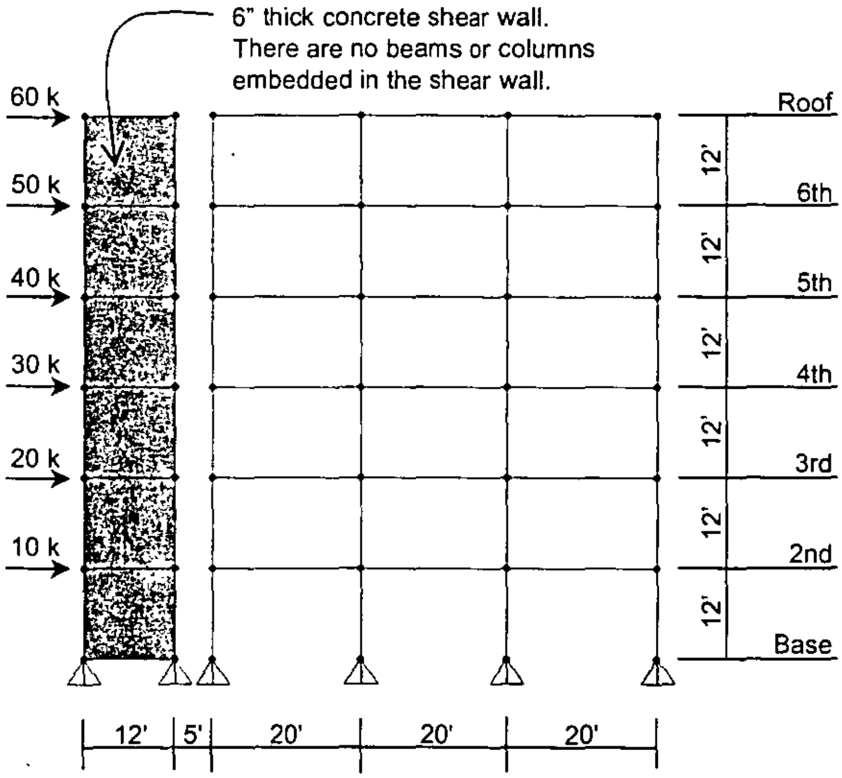
Frame-Shear Wall Interaction

Concrete Material Properties
E =3600 ksi, Poissons Ratio = 0.2

Frame
Beams: 12" wide by 24" deep
Columns: 24" by 24", pinned base

Diaphragm
Provide rigid diaphragm constraint at each level.

To Do
Determine shear carried by wall and by frame at 2nd level and at 6th level.

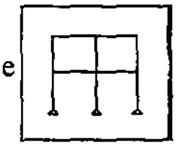


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem N Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template...**. This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box

   - Type **6** in the Number of Stories edit box.
   - Type **3** in the Number of Bays edit box.
   - Accept the default Story Height of 12.
   - Type **20** in the Bay Width edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

7. In this dialog box:

   - Verify that the X option is selected in the Direction area.
   - Type **-35** in the X Location edit box and click the **Add Grid Line** button.
   - Type **-47** in the X Location edit box and click the **Add Grid Line** button.
   - Click the **OK** button. The screen appears as shown in Figure N-1.

8. Click the **Quick Draw Rectangular Shell Element** button on the side toolbar (or select **Quick Draw Rectangular Shell Element** from the **Draw** menu).

9. Click in the area labeled "A" in Figure N-1 to input the first shell element. Note that a quick Shell element is drawn by clicking in a grid space, bounded by four grid lines.

10. Click in the areas labeled "B", "C", "D", "E" and "F", in that order, in Figure N-1 to input the other shell elements.

11. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

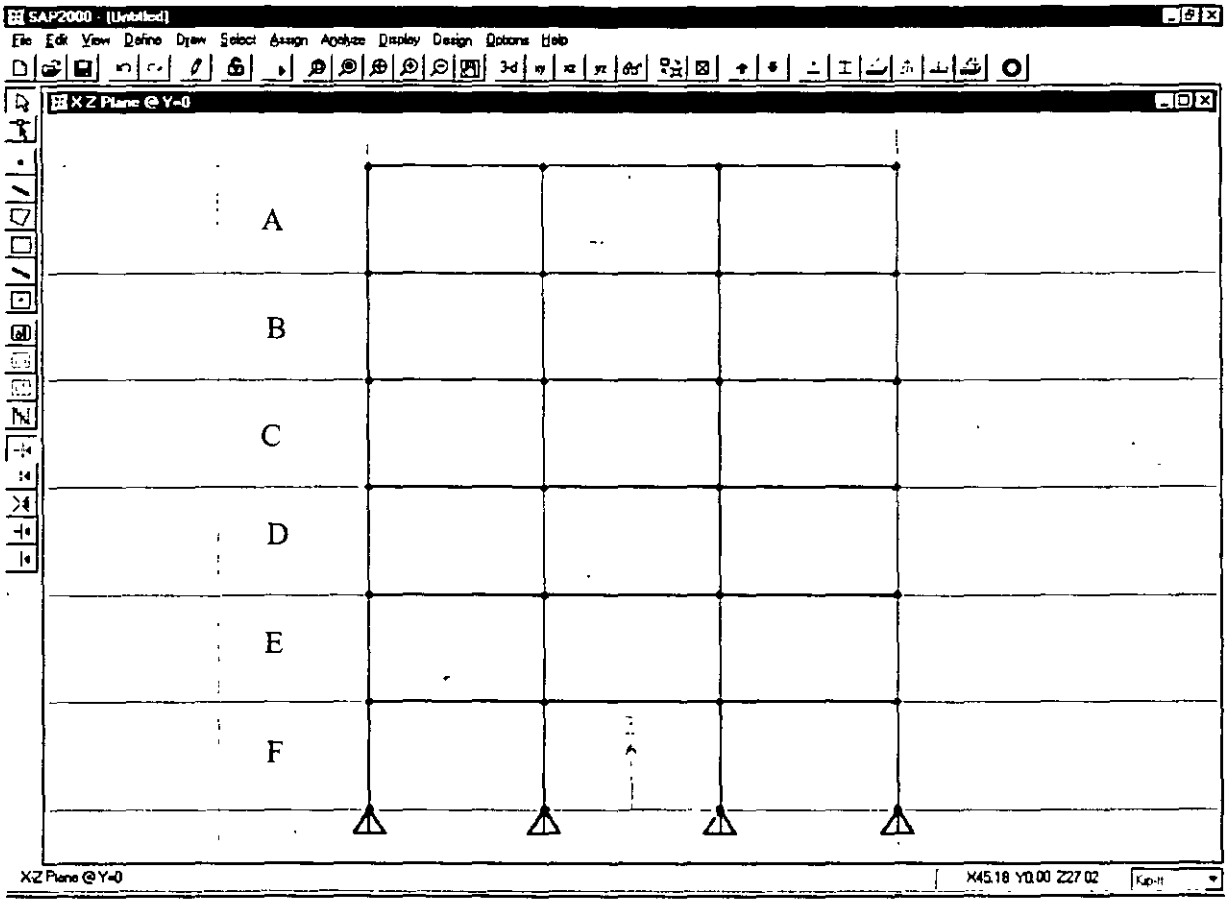


Figure N-1: Screen As It Appears After Step 7

12. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

13. In this dialog box:

    - Check the Labels box in the Joints area.
    - Check the Fill Elements check box.
    - Click the **OK** button.

    Note: If the font size is too small for you to read the joint labels use the following procedure to increase the font size. From the ***Options*** *menu select* ***Preferences****, click on the Dimensions Tab if it is not already visible, type in a new (larger) font size in the Minimum Graphic Font Size edit box (usually about 6 points is sufficient), click the* ***OK*** *button and then click the* ***Refresh Window*** *button on the main toolbar.*

14. Select the joints 41 and 42 at the base of the shear wall.

15. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.

16. In this dialog box:

    - Click the pinned base fast restraint button to set all translational degrees of freedom (U1, U2 and U3) as restrained.

    - Click the **OK** button

17. Click the **Show Undeformed Shape** button to remove the display of joint restraints and reset the window display (title).

18. Click the drop down box in the status bar to change the units to kip-in.

19. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

20. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

21. In this dialog box:

    - Verify that the Modulus of Elasticity is set to 3600.

    - Verify that Poisson's Ratio is set to 0.2.

    - Click the **OK** button twice to exit all dialog boxes.

22. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

23. In this dialog box:

    - In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Rectangular item to display the Rectangular Section dialog box.

    - In this dialog box:

        - Type **BEAM** in the Section name edit box.

        - Select CONC from the Material drop-down box.

        - Type **24** in the Depth (t3) edit box.

        - Type **12** in the Width (t2) edit box.

        - Click the **OK** button to return to the Define Frame Sections dialog box.

- In the Click To area, click the drop-down box that says Add Rectangular and then click on the Add Rectangular item to display the Rectangular Section dialog box.

- In this dialog box:

  - Type **COL** in the Section name edit box.

  - Select CONC from the Material drop-down box.

  - Type **24** in the Depth (t3) edit box.

  - Type **24** in the Width (t2) edit box.

  - Click the **OK** button twice to exit all dialog boxes.

24. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

25. In this dialog box:

    - In the Click To area, click the **Add New Section** button to display the Shell Sections dialog box.

    - In this dialog box:

      - Verify that the selected material is CONC.

      - In the Thickness area type **6** in both the Membrane and the Bending edit boxes.

      - Verify that the Shell option is selected in the Type area.

      - Click the **OK** button twice to close all dialog boxes.

26. Select all of the column elements by "windowing" each of the four column lines separately.

27. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

28. In this dialog box:

    - Click on COL in the Frame Sections area to highlight it.

    - Click the **OK** button.

29. Select all of the beam elements by using the intersecrting line method on each of the three beam bays separately.

30. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

31. In this dialog box:

    - Click on BEAM in the Frame Sections area to highlight it.
    - Click the **OK** button.

32. Select all of the joints at the Roof level by "windowing".

33. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

34. In this dialog box:

    - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
    - In this dialog box:
        - ➢ Type **ROOFDIA** in the Constraint Name edit box.
        - ➢ Select the Z Axis option in the Constraint Axis area.
        - ➢ Click the **OK** button twice to exit all dialog boxes.

35. Select all of the joints at the 6th level by "windowing".

36. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

37. In this dialog box:

    - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
    - In this dialog box:
        - ➢ Type **6THDIA** in the Constraint Name edit box.
        - ➢ Select the Z Axis option in the Constraint Axis area.
        - ➢ Click the **OK** button twice to exit all dialog boxes.

38. Select all of the joints at the 5th level by "windowing".

39. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

40. In this dialog box:

- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

- In this dialog box:

  - Type **5THDIA** in the Constraint Name edit box.

  - Select the Z Axis option in the Constraint Axis area.

  - Click the **OK** button twice to exit all dialog boxes.

41. Select all of the joints at the 4th level by "windowing".

42. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

43. In this dialog box:

- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

- In this dialog box:

  - Type **4THDIA** in the Constraint Name edit box.

  - Select the Z Axis option in the Constraint Axis area.

  - Click the **OK** button twice to exit all dialog boxes.

44. Select all of the joints at the 3rd level by "windowing".

45. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

46. In this dialog box:

- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

- In this dialog box:

  - Type **3RDDIA** in the Constraint Name edit box.

  - Select the Z Axis option in the Constraint Axis area.

  - Click the **OK** button twice to exit all dialog boxes.

47. Select all of the joints at the 2nd level by "windowing".

48. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

49. In this dialog box:

    - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

    - In this dialog box:

        - Type **2NDDIA** in the Constraint Name edit box.

        - Select the Z Axis option in the Constraint Axis area.

        - Click the **OK** button twice to exit all dialog boxes.

50. Select joints 39, 37, 35, 33, 29, and 32 (left-hand side of shear wall from 2nd level to roof level) by "windowing".

51. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

52. In this dialog box:

    - Type **10** in the Force Global X edit box in the Loads area.

    - Click the **OK** button.

53. Select joints 37, 35, 33, 29, and 32 (left-hand side of shear wall from 3rd level to roof level) by "windowing".

54. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

55. In this dialog box click the **OK** button.

56. Select joints 35, 33, 29, and 32 (left-hand side of shear wall from 4th level to roof level) by "windowing".

57. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

58. In this dialog box click the **OK** button.

59. Select joints 33, 29, and 32 (left-hand side of shear wall from 5th level to roof level) by "windowing".

60. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

61. In this dialog box click the **OK** button.

62. Select joints29, and 32 (left-hand side of shear wall from 6th level to roof level) by "windowing".

63. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

64. In this dialog box click the **OK** button.

65. Select joint 32 (left-hand side of shear wall at roof level).

66. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

67. In this dialog box click the **OK** button.

68. Click the **Show Undeformed Shape** button to remove the display of joint forces.

69. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

70. In this dialog box:

    - Check the Labels box in the Frames area.
    - Check the Labels box in the Shells area.
    - Click the **OK** button.

71. Select joints 29 and 30 and select shell element 1.

72. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.

73. In this dialog box:

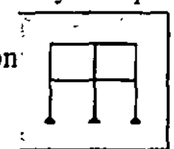
    - Type **6THWALL** in the Groups edit box.
    - Click the **Add New Group Name** button.
    - Click the **OK** button.

74. Select joints 6, 13, 20 and 27 by clicking on each joint individually and select frame (column) elements 6, 12, 18 and 24 using the intersecting line method.

75. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.

76. In this dialog box:

- Type **6THFRAME** in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

77. Select joints 39 and 40 and select shell element 5.
78. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.
79. In this dialog box:
    - Type **2NDWALL** in the Groups edit box.
    - Click the **Add New Group Name** button.
    - Click the **OK** button.
80. Select joints 2, 9, 16 and 23 by clicking on each joint individually and select frame (column) elements 2, 8, 14 and 20 using the intersecting line method.
81. From the **Assign** menu select **Group Names...** to display the Assign Group dialog box.
82. In this dialog box:
    - Type **2NDFRAME** in the Groups edit box.
    - Click the **Add New Group Name** button.
    - Click the **OK** button.
83. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
84. In this dialog box:
    - Uncheck the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Uncheck the Labels box in the Shells area.
    - Click the **OK** button.
85. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.

- Click the **OK** button.

86. Click the **Run Analysis** button ▸ to run the analysis.

87. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

88. From the **Display** menu select **Show Group Joint Force Sums...** to display the Select Groups dialog box.

89. In this dialog box:

    - Click on 2NDFRAME to highlight it.

    - Hold down the shift key on the keyboard and click on 6THWALL. All of the available groups, except the default ALL group should now be highlighted.

    - Click the **OK** button to display the a window with the Group Joint Force Summations tabulated.

    Note: Notice the direction of the shear (F-X force) in the shear wall at the 6th level.

    - When finished viewing the window click the "X" in its upper right-hand corner to close it.

    *Note: If you want to print the group joint force sum tables click on the **File** menu in the Group Joint Force Summations widow and select either **Print Tables** or **Print Tables to File....***

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "O"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels. 5521 4021 al 24, 5623.2910 y 5623.2971 • Fax 5510.0573

Problem O

Isolated Building - Nonlinear Time History Analysis

Steel
E =29000 ksi, Poissons Ratio = 0.3
Beams: W24X55; Columns: W14X90

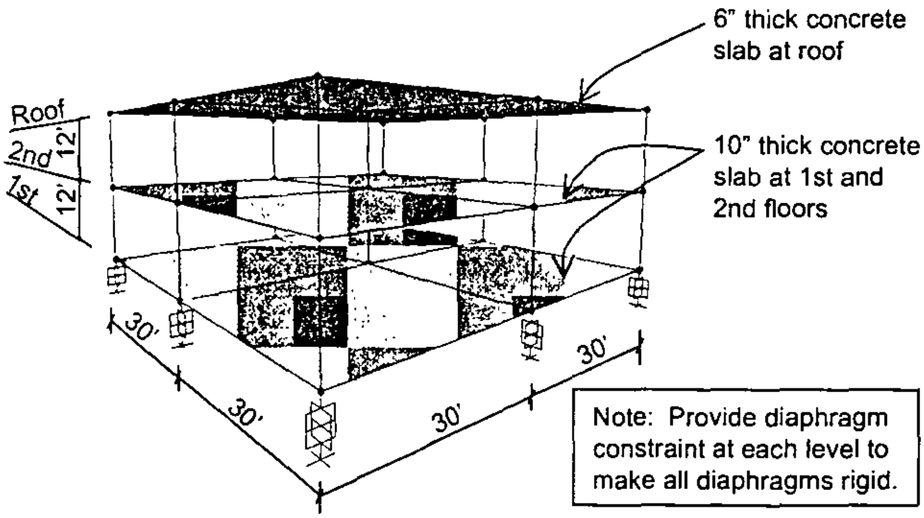
Rubber Isolator Properties
Vertical (axial) stiffness = 10,000 k/in (linear)
Initial shear stiffness in each direction = 10 k/in
Shear yield force in each direction = 5 kips
Ratio of post yield shear stiffness to initial shear stiffness = 0.2

Vertical Loading and Mass

Roof: 75 psf DL	Floor: 125 psf DL
20 psf LL	100 psf LL

Time History
Apply LP-TH0 in the X-direction and LP-TH90 in the Y-direction simultaneously. Each timehistory is given in units of g. There are 2000 timesteps, at an equal spacing of 0.02 sec, for a total of 40 sec. There are 5 accelerations points per line.

To Do
Plot time histories of Y-direction displacement at the 1st level and at the roof level. Plot a time history of the 1st level Y-direction displacement versus the Y-direction base shear.

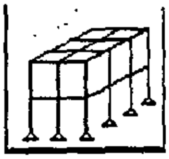


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem O Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template...**. This displays the Model Templates dialog box.

3. In this dialog box click on the **Space Frame** template button  to display the Space Frame dialog box.

4. In this dialog box:

   - Type **2** in the Number of Bays Along X edit box.
   - Type **30** in the Bay Width Along X edit box.
   - Type **30** in the Bay Width Along Y edit box.
   - Uncheck the Restraints check box.
   - Accept the rest of the default values.
   - Click the **OK** button.

5. Click in the window labeled X-Y Plane @ Z=24 to make sure it is active. The window is active when its title is highlighted.

6. Click the **Quick Draw Rectangular Shell Element** button on the side toolbar.

7. Click once in each of the four quadrants in the plan view to input four shell elements.

8. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=12.

9. Click once in each of the four quadrants in the plan view to input four shell elements.

10. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=0.

11. From the **Draw** menu select **Draw NLLink Element.**

12. In the plan view of the X-Y Plane @ Z=0 double click on each of the nine joints to draw nine NLLink elements.

13. Click the **Pointer** button to exit draw mode and enter select mode.

14. Click the drop down box in the status bar to change the units to kip-in. Kip-in

15. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

16. In this dialog box:

    - Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.

    - Click the **OK** button twice to exit the dialog boxes.

17. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

18. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

19. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.

20. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X55 section.

    - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.

    - Click the **OK** button three times to exit all dialog boxes.

21. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

22. In this dialog box:

    - Click the **Add New Section** button to display the Shell Sections dialog box.

    - In this dialog box:

        - Type **ROOF** in the Section Name edit box.

        - Accept the default CONC material

        - Type **6** in the Membrane edit box.

        - Type **6** in the Bending edit box.

        - In the Type area verify that the Shell option is selected.

        - Click the **OK** button to return to the Define Shell Sections dialog box.

- Click the **Add New Section** button to display the Shell Sections dialog box.
- In this dialog box:
  - ➢ Type **FLOOR** in the Section Name edit box.
  - ➢ Accept the default CONC material
  - ➢ Type **10** in the Membrane edit box.
  - ➢ Type **10** in the Bending edit box.
  - ➢ In the Type area verify that the Shell option is selected.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

23. From the **Define** menu select **NLLink Properties...** to display the Define NLLink Properties dialog box.

24. In this dialog box:

- Click the **Modify/Show Property** button to display the NLLink Property Data dialog box.
- In this dialog box:
  - ➢ Select Isolator1 from the Type drop-down box.
  - ➢ Type **.001** in the Mass edit box.
  - ➢ Check the U1 Direction check box.
  - ➢ Click the **Modify/Show For U1** button to display the NLLink Directional Properties dialog box.
  - ➢ In this dialog box:
    - ✓ Type **10000** in the Effective Stiffness edit box.
    - ✓ Click the **OK** button to return to the NLLink Property Data dialog box.
  - ➢ Check the U2 Direction check box.
  - ➢ Check the U2 Nonlinear check box.
  - ➢ Click the **Modify/Show For U2** button to display the NLLink Directional Properties dialog box.
  - ➢ In this dialog box:

    - ✓ In the Linear Properties area type **1.5** in the Effective Stiffness edit box.
    - ✓ In the Nonlinear Properties area type **10** in the Stiffness edit box.
    - ✓ Type **5** in the Yield Strength edit box.
    - ✓ Type **.2** in the Post Yield Stiffness Ratio edit box.
    - ✓ Accept the rest of the default values.
    - ✓ Click the **OK** button to return to the NLLink Property Data dialog box.
  - ➢ Check the U3 Direction check box.
  - ➢ Check the U3 Nonlinear check box.
  - ➢ Click the **Modify/Show For U3** button to display the NLLink Directional Properties dialog box.
  - ➢ In this dialog box:
    - ✓ In the Linear Properties area type **1.5** in the Effective Stiffness edit box.
    - ✓ In the Nonlinear Properties area type **10** in the Stiffness edit box.
    - ✓ Type **5** in the Yield Strength edit box.
    - ✓ Type **.2** in the Post Yield Stiffness Ratio edit box.
    - ✓ Accept the rest of the default values.
    - ✓ Click the **OK** button three times to exit all dialog boxes.

25. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

26. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the CONC material and click the **Modify/Show Section** button to display the Material Property Data dialog box.

27. In this dialog box:
    - Verify that the mass per unit volume is 4.657E-03 and that the weight per unit volume is 0.15.
    - Click the **OK** button twice to exit the dialog boxes.

28. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active.

29. Click the **xz 2D View** button xz on the main toolbar.

30. Click the **Perspective Toggle** button on the main toolbar.

31. Click the Set Intersecting Line select Mode button on the side toolbar and select all of the bottom level columns.

    *Note: To use the Intersecting Line Selection option, click the **Set Intersecting Line Select Mode** button on the side tool bar. Then click the left mouse button to the left of the first level columns, and while holding down the left mouse button drag the mouse to the right of the first level columns. A "rubberband line" will appear and all elements that this "rubberband line" passes through will be selected. Release the left mouse button to make the selection.*

32. Click the Set Intersecting Line select Mode button on the side toolbar and select all of the top level columns.

33. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

34. In this dialog box:

    - Click on W14X90 in the Frame Sections area to highlight it.
    - Click the **OK** button.

35. Click the **Show Undeformed Shape** button to remove the displayed frame element assignments.

36. Click the **xy 2D View** button on the main toolbar. The plan view of the X-Y Plane @ Z=0 appears.

37. Click the **Up One Gridline** button to move the plan display up to the X-Y Plane @ Z=12.

38. Select all of the elements at this level by "windowing".

39. Click the **Up One Gridline** button to move the plan display up to the X-Y Plane @ Z=24.

40. Select all of the elements at this level by "windowing".

41. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

42. In this dialog box:

    - Click on W24X55 in the Frame Sections area to highlight it.
    - Click the **OK** button.

43. Click the **Show Undeformed Shape** button to remove the displayed frame element assignments.

44. Select all of the elements at the Z=24 level by "windowing".

45. From the **Assign** menu select **Shell** and then **Sections...** from the submenu to display the Define Shell Sections dialog box.

46. In this dialog box:

    - Click on ROOF in the Shell Sections area to highlight it.
    - Click the **OK** button.

47. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=12.

48. Select all of the elements at this level by "windowing".

49. From the **Assign** menu select **Shell** and then **Sections...** from the submenu to display the Define Shell Sections dialog box.

50. In this dialog box:

    - Click on FLOOR in the Shell Sections area to highlight it.
    - Click the **OK** button.

51. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

52. In this dialog box:

    - Type **DL** in the Load edit box.
    - Click the **Change Load** button.
    - Type **LL** in the Load edit box.
    - Select LIVE from the Type drop-down box.
    - Type **0** in the Self Weight Multiplier edit box.
    - Click the **Add New Load** button.
    - Click the **OK** button.

53. Click the drop down box in the status bar to change the units to lb-ft. lb-ft

54. Select all of the elements at the Z=12 level by "windowing".

55. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

56. In this dialog box:

    - Select LL from the Load Case Name drop-down box.
    - Type **-100** in the Load edit box.
    - Click the **OK** button.

57. Click the **Up One Gridline** button to move the plan display up to the X-Y Plane @ Z=24.

58. Select all of the elements at the Z=24 level by "windowing".

59. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

60. In this dialog box:

    - Type **-20** in the Load edit box.
    - Click the **OK** button.

61. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

62. Click the **Show Undeformed Shape** button to remove the displayed shell load assignments.

63. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=12.

64. Select all of the elements at the Z=12 level by "windowing".

65. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

66. In this dialog box:

    - Select the Linear Tab.
    - Type **-12** in the Z Distance edit box.
    - Type **1** in the Number edit box.
    - Click the **OK** button.

Note: Prior to defining time history functions, you should locate the time history files named Lp-th0 and Lp-th90 that are in the subdirectory named Examples beneath the directory where you installed SAP2000. Copy these files into the same directory as your SAP2000 input file.

If the Examples subdirectory does not exist you may need to reinstall SAP2000, and select to install the examples.

67. From the **Define** menu select **Time History Functions...** to display the Define Time History Functions dialog box.

68. In this dialog box:

- Click the **Add Function From File** button to display the Time History Function Definition dialog box.
- In this dialog box:
  - ➢ Type **LPTH0** in the Function Name edit box.
  - ➢ Click the **Open File** button to display the Pick Function Data File dialog box.
  - ➢ In this dialog box:
    - ✓ Locate and highlight the file named LP-TH0
    - ✓ Click the **Open** button to return to the Time History Function Definition dialog box.
    - ✓ Type **5** in the Number Of Points Per Line edit box.
    - ✓ Select the Function At Equal Time Step option.
    - ✓ Type **.02** in the Function At Equal Time Step edit box.
    - ✓ Click the **OK** button to return to the Define Time History Functions dialog box.
- Click the **Add Function From File** button to display the Time History Function Definition dialog box.
- In this dialog box:
  - ➢ Type LPTH90 in the Function Name edit box.
  - ➢ Click the **Open File** button to display the Pick Function Data File dialog box.
  - ➢ In this dialog box:
    - ✓ Locate and highlight the file named LP-TH90.

- ✓ Click the **Open** button to return to the Time History Function Definition dialog box.
- ✓ Type **5** in the Number Of Points Per Line edit box.
- ✓ Select the Function At Equal Time Step option.
- ✓ Type **.02** in the Function At Equal Time Step edit box.
- ✓ Click the **OK** button twice to exit all dialog boxes.

69. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - Check the Dynamic Analysis check box, if it is not already checked.
    - Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.
    - In this dialog box:
        - ➢ Type **30** in the Number of Modes edit box.
        - ➢ In the Type Of Analysis area select the Ritz Vectors option.
        - ➢ Verify that ACCEL X, ACCEL Y and ACCEL Z are in the Ritz Load Vectors box in the Starting Ritz Vectors area.
        - ➢ Confirm that the Include NLLink Vectors box is checked.
        - ➢ Click the **OK** button twice to exit all dialog boxes.

70. From the **Define** menu select **Time History Cases...** to display the Define Time History Cases dialog box.

71. In this dialog box:

    - Click the **Add New History** button to display the Time History Case Data dialog box.
    - In this dialog box:
        - ➢ Type **GRAV** in the History Case Name edit box.
        - ➢ Select Nonlinear from the Analysis Type drop-down box.
        - ➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.
        - ➢ In this dialog box:

- ✓ Type **.05** in the Damping For All Modes edit box.
- ✓ Click the **OK** button.

➢ Type **100** in the Number of Output Time Steps edit box.

➢ Type **.1** in the Output Time Step Size edit box.

➢ Check the Envelopes check box.

➢ In the Load drop-down box, select DL.

➢ In the Function drop-down box, select RAMP.

➢ Type **1** in the Scale Factor edit box.

➢ Click the **Add** button.

➢ Click the **OK** button to return to the Define Time History Cases dialog box.

- Click the **Add New History** button to display the Time History Case Data dialog box.
- In this dialog box:

➢ Type **LP** in the History Case Name edit box.

➢ Select Nonlinear from the Analysis Type drop-down box.

➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.

➢ In this dialog box:

- ✓ Type **.05** in the Damping For All Modes edit box.
- ✓ In the Modal Damping Overrides area type Type 1 in the Mode box, type 0.02 in the Damping box and click the **Add** button.
- ✓ In the Modal Damping Overrides area type Type 2 in the Mode box and click the **Add** button.
- ✓ In the Modal Damping Overrides area type Type 3 in the Mode box and click the **Add** button.
- ✓ Click the **OK** button.

➢ Type **2000** in the Number of Output Time Steps edit box.

➢ Type **.02** in the Output Time Step Size edit box.

- In the Start From Previous History drop-down box select GRAV.
- Check the Envelopes check box.
- In the Load drop-down box, select acc dir 1.
- In the Function drop-down box, select LPTH0.
- Type **32.2** in the Scale Factor edit box.
- Click the **Add** button.
- In the Load drop-down box, select acc dir 2.
- In the Function drop-down box, select LPTH90.
- Click the **Add** button.
- Click the **OK** button twice to exit all dialog boxes.

72. Click in the window labeled X-Y Plane @ Z=12 to make sure it is active.

73. Click the **Up One Gridline** button ⇧ to move the plan display up to the X-Y Plane @ Z=24.

74. Select all elements at the Z=24 level by "windowing".

75. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

76. In this dialog box:

    - Click the drop-down box in the Click To area, and click Add Diaphragm to display the Diaphragm Constraint dialog box.
    - In this dialog box:
        - Type **ROOF** in the Constraint Name edit box.
        - Select the Z axis option in the Constraint Axis area if it is not already selected.
        - Click the **OK** button twice to assign the diaphragm constraint.

77. Click the **Down One Gridline** button ⇩ to move the plan display down to the X-Y Plane @ Z=12.

78. Select all elements at the Z=12 level by "windowing".

79. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

80. In this dialog box:

   - Click the drop-down box in the Click To area, and click Add Diaphragm to display the Diaphragm Constraint dialog box.
   - In this dialog box:
     - ➢ Type **2ND** in the Constraint Name edit box.
     - ➢ Select the Z axis option in the Constraint Axis area if it is not already selected.
     - ➢ Click the **OK** button twice to assign the diaphragm constraint.

81. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=0.

82. Select all elements at the Z=0 level by "windowing".

83. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

84. In this dialog box:

   - Click the drop-down box in the Click To area, and click Add Diaphragm to display the Diaphragm Constraint dialog box.
   - In this dialog box:
     - ➢ Type **1ST** in the Constraint Name edit box.
     - ➢ Select the Z axis option in the Constraint Axis area if it is not already selected.
     - ➢ Click the **OK** button twice to assign the diaphragm constraint.

85. Click the **Show Undeformed Shape** button to remove the displayed diaphragm constraint assignments.

86. Click the **Run Analysis** button to run the analysis.

   Note: The analysis would run even quicker if we had not requested envelopes in the time history case data.

87. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.

88. Click in the window labeled X-Y Plane @ Z=0 to make sure it is active.

89. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

90. In this dialog box:

- Check the Labels box in the Joints area.
- Click the **OK** button.

91. Click on the center joint, joint 13, in the plan at Z=0 to select it.

92. Click the **Up One Gridline** button ⇧ twice to move the plan display up to the X-Y Plane @ Z=24.

93. Click on the center joint, joint 15, in the plan at Z=24 to select it.

94. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

95. In this dialog box:

- Uncheck the Labels box in the Joints area.
- Click the **OK** button.

96. From the **Display** menu select **Show Time History Traces...** to display the Time History Display Definition dialog box.

97. In this dialog box:

- Click the **Define Functions** button to display the Time History Functions dialog box.
- In this dialog box:
  - ➢ Highlight Joint 13.
  - ➢ Click the **Modify/Show TH Function** button to display the Time History Joint Function dialog box.
  - ➢ In this dialog box:
    - ✓ Verify that the Displ option is selected in the Vector Type area.
    - ✓ Select the UY option is selected in the Component area.
    - ✓ Click the **OK** button to return to the Time History Functions dialog box.
  - ➢ Highlight Joint 15.

- Click the **Modify/Show TH Function** button to display the Time History Joint Function dialog box.
- In this dialog box:
  - ✓ Verify that the Displ option is selected in the Vector Type area.
  - ✓ Select the UY option is selected in the Component area.
  - ✓ Click the **OK** button to return to the Time History Functions dialog box.
- In the Click To area select Add Base Functions from the drop-down box to display the Base Functions dialog box.
- In this dialog box:
  - ✓ Check the Base Shear Y check box.
  - ✓ Click the **OK** button twice to return to the Time History Display Definition dialog box.

- Select LP from The Time History Case drop-down box.
- Click on Joint 13 in the List of Functions to highlight (select) it.
- Hold down the Ctrl key on the keyboard and click on Joint 15 to add it to the selection.
- Click the **Add** button to move Joints 13 and 15 into the Plot Functions list.
- Click the **Display** button to display the displacement time histories. Note that there is very little difference between the 1st and roof level displacements. The structure is essentially moving as a rigid body on top of the isolators.
- Click the **OK** button to close the time history display and return to the Time History Display Definition dialog box.
- Click the F(t) vs F(t) tab.
- Select Joint 13 from the Horizontal drop-down box.
- Select Base Shear Y from the Vertical drop-down box.
- Click the **Display** button to display the force-displacement plot.
- Click the **OK** button to close the Time History Functions display and return to the Time History Display Definition dialog box.
- Click the **Done** button to close the Time History Display Definition dialog box.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "P"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

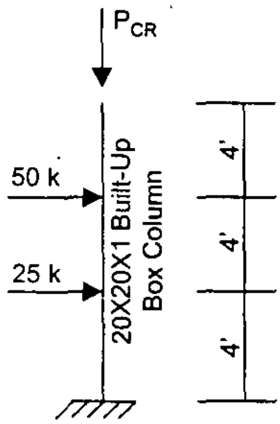
Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem P

Critical Buckling Load

Steel
E =29000 ksi, Poissons Ratio = 0.3

To Do
Use P-Delta option and iteration to determine the critical buckling load for this built-up column. Hint: $P_{CRITICAL}$ is between 15,480 and 15,490 kips.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem P Solution

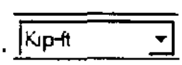
1. Click the drop down box in the status bar to change the units to kip-ft.
2. From the **File** menu select **New Model....** This displays the Coordinate System Definition dialog box.
3. In this dialog box:
   - Select the Cartesian Tab.
   - Type **0** in the X Direction Number of Grid Spaces edit box.
   - Type **0** in the Y Direction Number of Grid Spaces edit box.
   - Type **1** in the Z Direction Number of Grid Spaces edit box.
   - Type **12** in the Z Direction Grid Spacing edit box.
   - Click the **OK** button.
4. Click the **Quick Draw Frame Element** button on the side toolbar.
5. Click once on the grid line in the 3-D View window to draw the frame element.
6. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.
7. Click on the bottom joint in the 3-D View window to select it.
8. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.
9. In this dialog box:
   - Click the **fixed base fast restraint** button to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.
   - Click the **OK** button.
10. Click the drop down box in the status bar to change the units to kip-in.
11. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
12. In this dialog box:

- Verify that the modulus of elasticity is 29000 and poisson's ratio is 0.3.
- Click the **OK** button twice to exit the dialog boxes.

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

14. In the Click To area, click the drop-down box that saysAdd I/Wide Flange and then click on the Add Box/Tube item to display the Box/Tube Section dialog box.

15. In this dialog box:
    - Type **BOX** in the Section Name edit box.
    - Type **20** in the Outside Depth (t3) edit box.
    - Type **20** in the Outside Width (t2) edit box.
    - Type **1** in the Flange Thickness (tf) edit box.
    - Type **1** in the Web Thickness (tw) edit box.
    - Click the **OK** button twice to exit all dialog boxes.

16. Click the drop down box in the status bar to change the units to kip-ft. 

17. Click on the frame element to select it.

18. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

19. In this dialog box:
    - Click on BOX in the Frame Sections area to highlight it.
    - Click the **OK** button.

20. Click the **Show Undeformed Shape** button  to remove the displayed frame element assignments.

21. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

22. In this dialog box:
    - Type **LAT** in the Load edit box.
    - Select OTHER from the Type drop-down box.

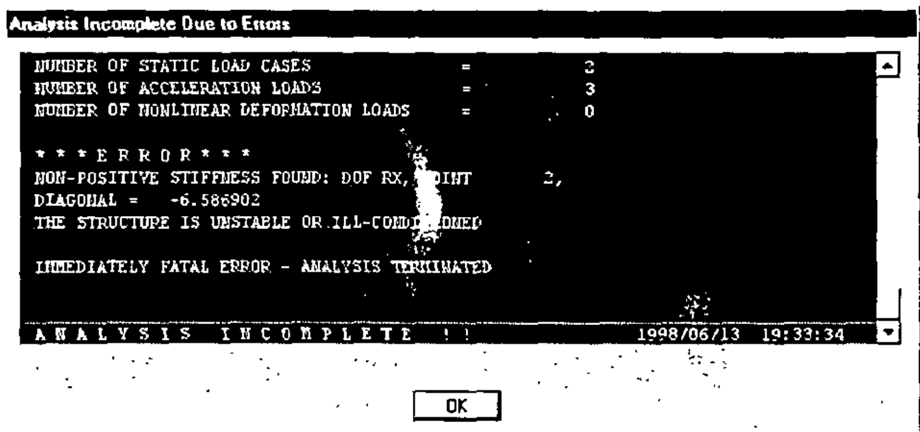
- Type **0** in the Self Weight Multiplier edit box.
- Click the **Change Load** button.
- Type **AXIAL** in the Load edit box.
- Click the **Add New Load** button.
- Click the **OK** button.

23. Click on the frame element to select it.
24. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
25. In this dialog box:
    - Select LAT from the Load Case Name drop-down box.
    - Select Global X from the Direction drop-down box in the Load Type and Direction area.
    - Type **.3333** in the first Distance edit box and type **25** in the first Load edit box.
    - Type **.6667** in the second Distance edit box and type **50** in the second Load edit box.
    - Click the **OK** button.
26. Select the top joint.
27. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.
28. In this dialog box:
    - Select AXIAL from the Load Case Name drop-down box.
    - Type **-1** in the Force Global Z edit box in the Loads area.
    - Click the **OK** button.
29. Click the **Show Undeformed Shape** button to remove the displayed joint force assignments.
30. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
    - Check the Include P-Delta check box, if it is not already checked.

- Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
- In this dialog box:
    - ➢ Type **5** in the Maximum Iterations edit box.
    - ➢ Select AXIAL from the Load Case drop-down box.
    - ➢ Type **15485** in the Scale Factor area.

      Note: Since the hint says $P_{CRITICAL}$ is between 15,480 and 15,490 kips, we will start midway between these two values at 15485 kips.
    - ➢ Click the **Add** button.
    - ➢ Click the **OK** button twice to exit all dialog boxes.

31. Click the **Run Analysis** button to run the analysis.
32. When the analysis is complete check the messages in the Analysis window. There should be an error message similar to that shown below indicating the structure is unstable. This indicates that 15,485 kips is larger than the critical buckling load. Click the **OK** button to close the Analysis window.



33. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
    - Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.
    - In this dialog box:
        - ➢ Highlight the AXIAL/15485 Load Case/Scale Factor..
        - ➢ Type **15484** in the Scale Factor area.
        - ➢ Click the **Modify** button.

- Click the **OK** button twice to exit all dialog boxes.

34. Click the **Run Analysis** button to run the analysis.

35. When the analysis is complete check the messages in the Analysis window. Again there should be an error message indicating the structure is unstable. This indicates that 15,484 kips is larger than the critcal buckling load. Click the **OK** button to close the Analysis window.

36. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

 - Click the **Set P-Delta Parameters** button to display the P-Delta Parameters dialog box.

 - In this dialog box:

   - Highlight the AXIAL/15484 Load Case/Scale Factor..

   - Type **15483** in the Scale Factor area.

   - Click the **Modify** button.

   - Click the **OK** button twice to exit all dialog boxes.

37. Click the **Run Analysis** button to run the analysis.

38. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). The analysis run successfully. Thus the critical buckling load is approximately 15484 kips. Click the **OK** button to close the Analysis window.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "Q"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem Q

Three Frames

Concrete
E =5000 ksi, Poissons Ratio = 0.2
Beams: 24" wide by 36" deep
Columns: 24" by 24"

Damper Properties

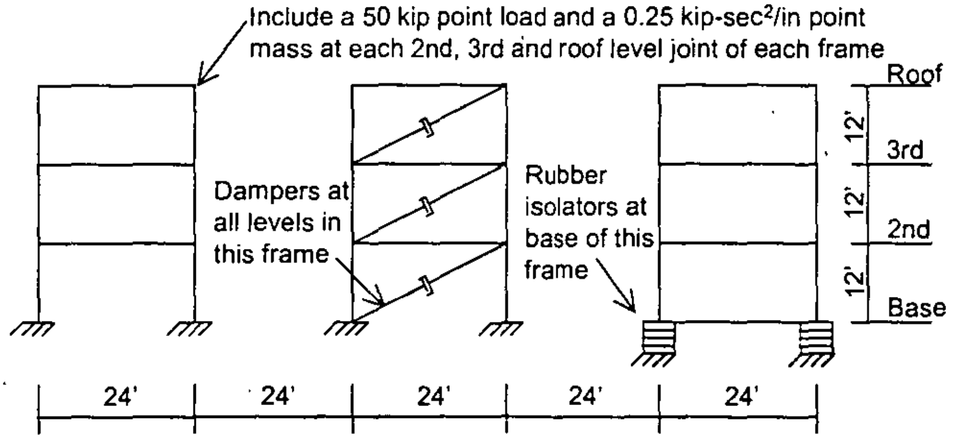
Linear Properties
- Effective stiffness = 0 k/in
- Effective damping = 0 k-sec/in

Nonlinear Properties
- Stiffness = 1000 k/in
- Damping = 30 k-sec/in
- Damping exponent = 0.5

Isolator Properties (Isolator1)
Vertical (axial) stiffness = 10,000 k/in (linear)
Initial shear stiffness = 100 k/in
Shear yield force = 40 kips
Ratio of post yield shear stiffness to initial shear stiffness = 0.1

Time History
Apply the ELCENTRO record. Three time and acceleration values are given on each line of this file. The acceleration value is in units of g. The length of the record is 12.1 seconds.

To Do
Create three frames, one bare, one with dampers, and one with isolators. Create a video (*.avi file) of the nonlinear time history run. Review the mode shapes.

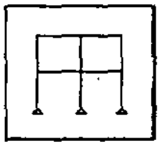


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem Q Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box

   - Type **3** in the Number of Stories edit box.
   - Type **5** in the Number of Bays edit box.
   - Uncheck the Restraints check box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Labels box in the Joints area.
   - Check the Labels box in the Frames area.
   - Click the **OK** button.

8. Select beam elements 22, 23, 24, 28, 29 and 30. Press the delete key on the keyboard to delete these elements.

   *Note: You could select the elements by clicking on each one individually, by using the Intersecting Line Select Mode, or by using the Select By Labels option (**Select** menu > **Select** > **Labels**).*

9. Click the **Refresh Window** button to refresh the drawing.

10. From the **Draw** menu select **QuickDraw Frame Element**.

11. Click on the grid line between joints 17 and 21 to enter a beam at the base of the isolated frame.

12. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

13. Select joints 1, 5, 9 and 13.

14. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.

15. In this dialog box:

    - Click the **Fixed Base Fast Restraint** button to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.

    - Click the **OK** button.

16. Click the drop down box in the status bar to change the units to kip-in. Kip-in

17. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

18. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

19. In this dialog box:

    - Type **0** in the Mass per Unit Volume edit box.

    - Accept the default Weight per Unit Volume.

    - Type **5000** in the Modulus of Elasticity edit box.

    - Type **.2** in the Poisson's Ratio edit box, if it is not already entered.

    - Click the **OK** button twice to exit all dialog boxes.

20. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

21. In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Rectangular item to display the Rectangular Section dialog box.

22. In this dialog box:

    - Type **BEAM** in the Section Name edit box.

    - Select CONC in the Material drop-down box.

    - Type **36** in the Depth (t3) edit box.

    - Type **24** in the Width (t2) edit box.

- Click the **OK** button to return to the Define Frame Sections dialog box.

23. In the Click To area, click the drop-down box that says Add Rectangular and then click on the Add Rectangular item to display the Rectangular Section dialog box.

24. In this dialog box:

    - Type **COL** in the Section Name edit box.
    - Select CONC in the Material drop-down box.
    - Type **24** in the Depth (t3) edit box.
    - Type **24** in the Width (t2) edit box.
    - Click the **OK** button twice to exit all dialog boxes.

25. Select all beam elements (10 total).

26. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

27. In this dialog box:

    - Click on BEAM in the Frame Sections area to highlight it.
    - Click the **OK** button.

28. Select all column sections by "windowing" on each column line separately (18 total).

29. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

30. In this dialog box:

    - Click on COL in the Frame Sections area to highlight it.
    - Click the **OK** button.

31. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments so that you can see the frame element labels again.

32. From the **Define** menu select **NLLink Properties...** to display the Define NLLink Properties dialog box.

33. In this dialog box:

    - Click the **Add New Property** button to display the NLLink Property Data dialog box.

- In this dialog box:
  - Type **ISO** in the Property Name edit box.
  - Select Isolator1 from the Type drop-down box.
  - Type **.001** in the Mass edit box.
  - Check the U1 Direction check box.
  - Click the **Modify/Show For U1** button to display the NLLink Directional Properties dialog box.
  - In this dialog box:
    - ✓ Type **10000** in the Effective Stiffness edit box.
    - ✓ Click the **OK** button to return to the NLLink Property Data dialog box.
  - Check the U2 Direction check box.
  - Check the U2 Nonlinear check box.
  - Click the **Modify/Show For U2** button to display the NLLink Directional Properties dialog box.
  - In this dialog box:
    - ✓ In the Linear Properties area type **10** in the Effective Stiffness edit box.
    - ✓ In the Nonlinear Properties area type **100** in the Stiffness edit box.
    - ✓ Type **40** in the Yield Strength edit box.
    - ✓ Type **.1** in the Post Yield Stiffness Ratio edit box.
    - ✓ Accept the rest of the default values.
    - ✓ Click the **OK** button to return to the NLLink Property Data dialog box.
  - Click the **OK** button to return to the Define NLLink Properties dialog box.
- Click the **Add New Property** button to display the NLLink Property Data dialog box.
- In this dialog box:
  - Type DAMP in the Property Name edit box.
  - Select Damper from the Type drop-down box.

- ➢ Type **.001** in the Mass edit box.
- ➢ Check the U1 Direction check box.
- ➢ Check the U1 Nonlinear check box.
- ➢ Click the **Modify/Show For U1** button to display the NLLink Directional Properties dialog box.
- ➢ In this dialog box:
  - ✓ In the Nonlinear Properties area type **1000** in the Stiffness edit box.
  - ✓ Type **30** in the Damping edit box.
  - ✓ Type **.5** in the Damping Exponent edit box.
  - ✓ Click the **OK** button three times to exit all dialog boxes.

34. From the **Draw** menu select **Draw NLLink Element**.
35. Double click on joints 17 and 21 to draw two NLLink elements.
36. Click on joint 9 and then joint 14 to draw an NLLink element.
37. Click on joint 10 and then joint 15 to draw an NLLink element.
38. Click on joint 11 and then joint 16 to draw an NLLink element.
39. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.
40. Click on the three NLLink element in the center frame to select them.
41. From the **Assign** menu select **NLLink** and then **Properties...** from the submenu to display the Define NLLink Properties dialog box.
42. In this dialog box:
    - Click on DAMP in the NLLink Props area to highlight it.
    - Click the **OK** button.
43. Click the **Show Undeformed Shape** button to remove the displayed NLLink property assignments so that you can see the joint labels again.
44. Hold down the Ctrl key on the keyboard and left click joint 17 to display the Selection List dialog box.

45. In the dialog box click on the NLLink 1 element. The Selection List dialog box closes and the NLLink element is selected.

46. Hold down the Ctrl key on the keyboard and left click joint 21 to display the Selection List dialog box.

47. In the dialog box click on the NLLink 2 element. The Selection List dialog box closes and the NLLink element is selected.

48. From the **Assign** menu select **NLLink** and then **Properties...** from the submenu to display the Define NLLink Properties dialog box.

49. In this dialog box:

    - Click on ISO in the NLLink Props area to highlight it.

    - Click the **OK** button.

50. Click the **Show Undeformed Shape** button to remove the displayed NLLink property assignments.

51. Select all joints at the 2nd, 3rd and Roof levels by "windowing".

52. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

53. In this dialog box:

    - Type **-50** in the Force Global Z edit box.

    - Click the **OK** button.

54. Select all joints at the 2nd, 3rd and Roof levels by "windowing".

55. From the **Assign** menu select **Joint** and then **Masses...** from the submenu to display the Joint Masses dialog box.

56. In this dialog box:

    - Type **.25** in the Direction 1 edit box.

    - Type **.25** in the Direction 3 edit box.

    - Click the **OK** button.

57. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

58. Click the **Show Undeformed Shape** button to remove the displayed joint mass assignments.

59. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

60. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

    Note: Prior to defining time history functions, you should locate the time history file named Elcentro that is in the subdirectory named Examples beneath the directory where you installed SAP2000. Copy this file into the same directory as your SAP2000 input file.

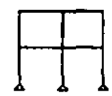
    If the Examples subdirectory does not exist you may need to reinstall SAP2000, and select to install the examples.

61. From the **Define** menu select **Time History Functions...** to display the Define Time History Functions dialog box.

62. In this dialog box:

    - Click the Add Function From File button to display the Time History Function Definition dialog box.
    - In this dialog box:
        - ➢ Type **ELCEN** in the Function Name edit box.
        - ➢ Click the **Open File** button to display the Pick Function Data File dialog box.
        - ➢ In this dialog box:
            - ✓ Locate and highlight the file named Elcentro
            - ✓ Click the **Open** button to return to the Time History Function Definition dialog box.
            - ✓ Type **3** in the Number Of Points Per Line edit box.
            - ✓ Select the Time and Function Values option.
            - ✓ Click the **OK** button twice to exit all dialog boxes.

63. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.

- Check the Dynamic Analysis check box, if it is not already checked.

- Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

- In this dialog box:

  - ➢ Type **30** in the Number of Modes edit box.

  - ➢ In the Type Of Analysis area select the Ritz Vectors option.

  - ➢ Verify that ACCEL X and ACCEL Z are in the Ritz Load Vectors box in the Starting Ritz Vectors area. Move any other vectors out of the Ritz Load Vectors box

  - ➢ Confirm that the Include NLLink Vectors box is checked.

  - ➢ Click the **OK** button twice to exit all dialog boxes.

64. From the **Define** menu select **Time History Cases...** to display the Define Time History Cases dialog box.

65. In this dialog box:

- Click the **Add New History** button to display the Time History Case Data dialog box.

- In this dialog box:

  - ➢ Type **GRAV** in the History Case Name edit box.

  - ➢ Select Nonlinear from the Analysis Type drop-down box.

  - ➢ Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.

  - ➢ In this dialog box:

    - ✓ Type **.02** in the Damping For All Modes edit box.

    - ✓ Click the **OK** button.

  - ➢ Type **100** in the Number of Output Time Steps edit box.

  - ➢ Type **.1** in the Output Time Step Size edit box.

  - ➢ Check the Envelopes check box.

- In the Load drop-down box select GRAV.
- In the Function drop-down box, select RAMP.
- Type **1** in the Scale Factor edit box.
- Click the **Add** button.
- Click the **OK** button to return to the Define Time History Cases dialog box.

- Click the **Add New History** button to display the Time History Case Data dialog box.
- In this dialog box:
  - Type **ELCN** in the History Case Name edit box.
  - Select Nonlinear from the Analysis Type drop-down box.
  - Click the **Modify/Show** button for modal damping to display the Modal Damping dialog box.
  - In this dialog box:
    - ✓ Type **.02** in the Damping For All Modes edit box.
    - ✓ Click the **OK** button.
  - Type **1210** in the Number of Output Time Steps edit box.
  - Type **.01** in the Output Time Step Size edit box.
  - Check the Envelopes check box.
  - In the Load drop-down box select acc dir 1.
  - In the Function drop-down box, select ELCEN.
  - Type **32.2** in the Scale Factor edit box.
  - Click the **Add** button.
  - Click the **OK** button twice to exit all dialog boxes.

66. Click the **Run Analysis** button ▸ to run the analysis.
67. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.

68. Click the **Start Animation** button Start Animation located on the status bar at the bottom of the screen to animate the first mode shape.

69. Click the **Right Arrow** button ⇨ located on the status bar at the bottom of the screen to view the next mode shape.

70. Continuing clicking the **Right Arrow** button ⇨ to step through all of the mode shapes.

71. Click the **Show Undeformed Shape** button to remove the displayed mode shape.

72. From the **File** menu select **Create Video...** and then select **Create Time History Animation Video...** to display the Video File dialog box.

73. In this dialog box select the name and location for the video file (*.avi) and click the **Save** button. The Time History Video File Creation dialog box is displayed.

74. In this dialog box:

    - Type **50** in the Magnification Factor edit box.
    - In the Frame Size (Pixels) edit boxes type **640** by **480**.
    - Click the **OK** button. The *.avi file is created.

*Note: Once the *.avi file has been created it can be played on any multimedia player that supports *.avi files. A program called Media Player is supplied with Windows 95 that will play *.avi files. This program can often be found by clicking the **Start** button, selecting **Programs**, then selecting **Accessories** and finally selecting **Multimedia**. No multimedia player is provided with SAP2000.*

Note: The purpose of this example is to display the capabilities of SAP2000. The structural properties of each frame have not been optimized in this example. Therefore, great care should be taken in drawing any conclusions about the relative performance of different structural systems based on this example.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "R"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Problem R

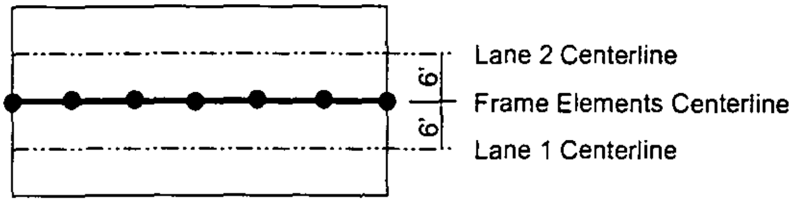
Bridge With Moving Load

Concrete Material Properties
E =5000 ksi, Poissons Ratio = 0.2
Member Properties

Column
- $A = 40\ ft^2$
- $I = 400\ ft^3$
- $AS = 30\ ft^2$

Girder
- $A = 35\ ft^2$
- $I = 500\ ft^3$
- $AS = 12\ ft^2$

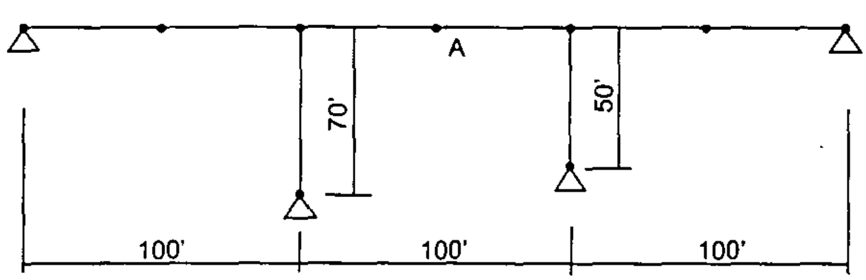
Moving Load
Number of Lanes = 2



Check for worst case of HS20-44 truck load and HS20-44L lane load applied to each lane simultaneously.

Use the Exact method of calculation.

To Do
Set the number of ouput segments for girder elements to 2. Review the moving load influence line for vertical displacement at joint A in Lane 1. Review the moving load girder M33 moments in Lane 1. Set the number of ouput segments for the girder elements to 10. Review the same influence line and moments.

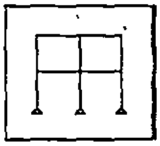


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem R Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box

   - Type **1** in the Number of Stories edit box.
   - Type **3** in the Number of Bays edit box.
   - Type **70** in the Story Height edit box.
   - Type **100** in the Bay Width edit box.
   - Uncheck the Restraints check box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Labels box in the Joints area.
   - Check the Labels box in the Frames area.
   - Click the **OK** button.

8. Select column elements 1 and 4. Press the delete key on the keyboard to delete these elements.

9. Click the **Refresh Window** button to refresh the drawing.

10. Select joints 2, 3, 5 and 8.

11. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

12. In this dialog box:

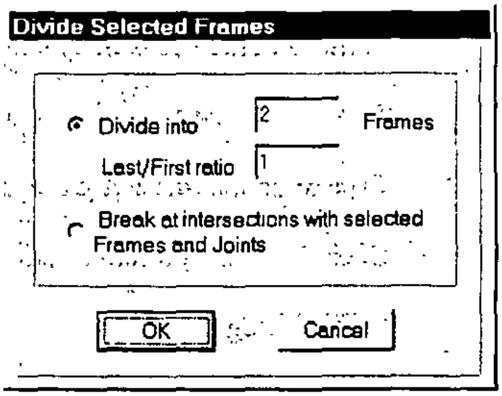
    - Verify that the Translation 1, 2 and 3 check boxes are checked. If they are not checked, check them.
    - Click the **OK** button.

13. Select joint 5.

14. From the **Edit** menu select **Move...** to display the **Move Selected Points** menu.

15. In this dialog box:

    - Type **20** in the Delta Z edit box.
    - Verify that 0 is entered in the Delta X and Delta Y edit boxes.
    - Click the **OK** button.



16. Select frame elements 5, 6 and 7.

17. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.

18. Verify that the dialog box appears as shown in the figure and click the **OK** button.

19. Select frame elements 8 through 13 (i.e., the girder elements).

20. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

21. In this dialog box:

    - Type **2** in the Number of Segments edit box.
    - Click the **OK** button.

22. Click the **Show Undeformed Shape** button to remove the displayed output segment assignments so that you can see the frame element labels again.

23. Click the drop down box in the status bar to change the units to kip-in. Kip-in

24. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

25. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

26. In this dialog box:

   - Type **5000** in the Modulus of Elasticity edit box.
   - Type **.2** in the Poisson's Ratio edit box, if it is not already entered.
   - Click the **OK** button twice to exit all dialog boxes.

27. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

28. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

29. In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add General item to display the Property Data dialog box.

30. In this dialog box:

   - Type **40** in the Cross Sectional (Axial) Area edit box.
   - Type **400** in the Moment Of Inertia About 3 Axis edit box.
   - Type **30** in the Shear Area in 2 Direction edit box.
   - Click the **OK** button to display the General Section dialog box.
   - In this dialog box:
     - Type **COLUMN** in the Section Name edit box.
     - Select CONC from the Material drop-down box.
     - Click the **OK** button to return to the Define Frame Sections dialog box.

31. In the Click To area, click the drop-down box that says Add General and then click on the Add General item to display the Property Data dialog box.

32. In this dialog box:

   - Type **35** in the Cross Sectional (Axial) Area edit box.
   - Type **500** in the Moment Of Inertia About 3 Axis edit box.
   - Type **12** in the Shear Area in 2 Direction edit box.
   - Click the **OK** button to display the General Section dialog box.
   - In this dialog box:

> Type **GIRDER** in the Section Name edit box.

> Select CONC from the Material drop-down box.

> Click the **OK** button twice to exit all dialog boxes.

33. Select frame elements 8 through 13 (girders).

34. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

35. In this dialog box:

- Click on GIRDER in the Frame Sections area to highlight it.
- Click the **OK** button.

36. Select the two column elements.

37. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

38. In this dialog box:

- Click on COLUMN in the Frame Sections area to highlight it.
- Click the **OK** button.

39. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments so that you can see the frame element labels again.

40. From the **Define** menu select **Moving Load Cases** and then select **Lanes...** to display the Define Bridge Lanes dialog box.

41. In this dialog box:

- Click the **Add New Lane** button to display the Lane Data dialog box.
- In this dialog box:

> Accept the default Lane Name, LANE1.

> Type **8** in the Frame edit box.

> Type **-6** in the Eccentricity edit box.

> Click the **Add** button.

> Type **9** in the Frame edit box.

- ➢ Click the **Add** button.
- ➢ Type **10** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Type **11** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Type **12** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Type **13** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Click the OK button to return to the Define Bridge Lanes dialog box. In this dialog box:

- Click the **Add New Lane** button to display the Lane Data dialog box.
- In this dialog box:

- ➢ Accept the default Lane Name, LANE2.
- ➢ Type **8** in the Frame edit box.
- ➢ Type **6** in the Eccentricity edit box.
- ➢ Click the **Add** button.
- ➢ Type **9** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Type **10** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Type **11** in the Frame edit box.
- ➢ Click the **Add** button.
- ➢ Type **12** in the Frame edit box.
- ➢ Click the **Add** button.

- Type **13** in the Frame edit box.
- Click the **Add** button.
- Click the OK button twice to exit all dialog boxes.

42. From the **Define** menu select **Moving Load Cases** and then select **Vehicles...** to display the Define Vehicles dialog box.

43. In this dialog box:

- In the Click To area click the drop-down box and select Add Standard Vehicle to display the Standard Vehicle Data dialog box.
- In this dialog box:
  - In the Data Definition area select HSn-44 in the Vehicle Type drop-down box.
  - Type **20** in the Scale Factor edit box if it is not already there.
  - Click the **OK** button to return to the Define Vehicles dialog box.
- In the Click To area click the drop-down box and select Add Standard Vehicle to display the Standard Vehicle Data dialog box.
- In this dialog box:
  - In the Data Definition area select HSn-44L in the Vehicle Type drop-down box.
  - Type **20** in the Scale Factor edit box if it is not already there.
  - Click the **OK** button twice to exit all dialog boxes.

44. From the **Define** menu select **Moving Load Cases** and then select **Vehicles Classes...** to display the Define Vehicle Classes dialog box.

45. In this dialog box:

- Click the **Add New Class** button to display the Vehicle Class Data dialog box.
- In this dialog box:
  - Accept the default Vehicle Class Name, VECL1
  - Verify that HSn441 is in the Vehicle Name drop-down box.
  - Enter **1** in the Scale Factor edit box if it is not already there.
  - Click the **Add** button.

- Select HSn442 in the Vehicle Name drop-down box.
- Click the **Add** button.
- Click the **OK** button twice to exit all dialog boxes.

46. From the **Define** menu select **Moving Load Cases** and then select **Bridge Responses...** to display the Bridge Response Request dialog box.

47. In this dialog box:

    - In the Type of Results area check all four of the check boxes if they are not already checked.
    - In the Method of Calculation area select the Exact option.
    - Click the **OK** button.

48. From the **Define** menu select **Moving Load Cases** and then select **Moving Load Cases...** to display the Define Moving Load Cases dialog box.

49. In this dialog box:

    - Click the **Add New Load** button to display the Moving Load Case Data dialog box.
    - In this dialog box:
        - Accept the default Moving Load Case Name, MOVE1.
        - Click the **Add New Assign** button to display the Moving Load Case Assignment Data dialog box.
        - In this dialog box:
            - ✓ In the Assignment Lanes area click on LANE1 in the Select Lanes From list box to highlight (select) it.
            - ✓ Hold down the Ctrl key on the keyboard and click on LANE2 to add it to the selection.
            - ✓ Click the **Add** button to transfer these items to the Selected Lanes list box.
            - ✓ Click the **OK** button three times to exit all dialog boxes.

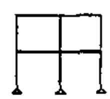
50. Note that the joint labeled A in the problem statement is labeled 10 on the screen.

51. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

52. In this dialog box:

- Uncheck the Labels box in the Joints area.
- Uncheck the Labels box in the Frames area.
- Click the **OK** button.

53. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

- In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
- Click the **OK** button.

54. Click the **Run Analysis** button ▸ to run the analysis.

55. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

56. From the **Display** menu select **Show Influence Lines...** and then **Joints...** to display the Show Joint Influence Line dialog box.

57. In this dialog box:

- Select LANE1 from the Lane drop-down box if it is not already showing.
- Type **10** in the Joint ID edit box.
- In the Vector Type area select the Displacement option if it is not already selected.
- In the Component area select the U3 option (vertical displacement).
- Click the **OK** button to display the influence line.

Note: This influence line is constructed with two output segments specified for the girder elements. Points for the influence line are calculated at the ends of each output segment. These points are then connected by straight lines. You can clearly see the segments in the influence line.

58. Click the **Member Force Diagram For Frames** button **F** to display the Member Force Diagram For Frames dialog box.

59. In this dialog box:

- Select MOVE1 Moving Load from the Load drop-down box.
- In the Component area select the Moment 3-3 option.

- Uncheck the Fill Diagram check box.
- Check the Show Values On Diagram check box.
- Click the **OK** button to display the moment diagram.

Note: This moment diagram is constructed with two output segments specified for the girder elements. Points for the moment diagram are calculated at the ends of each output segment. These points are then connected by straight lines. You can clearly see the segments in the moment diagram.

60. Click the **Lock/Unlock Model** button and click the resulting **OK** button to unlock the model.

61. Select frame elements 8 through 13 (i.e., the girder elements).

62. From the **Assign** menu select **Frame** and then **Output Segments...** from the submenu to display the Frame Output Segments dialog box.

63. In this dialog box:

    - Type **10** in the Number of Segments edit box.
    - Click the **OK** button.

64. Click the **Show Undeformed Shape** button to remove the displayed output segment assignments.

65. Click the **Run Analysis** button to run the analysis.

66. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

67. From the **Display** menu select **Show Influence Lines...** and then **Joints...** to display the Show Joint Influence Line dialog box.

68. In this dialog box click the OK button to display the influence line.

    Note: This influence line is much sharper than the previous one.

69. Click the **Member Force Diagram For Frames** button **F** to display the Member Force Diagram For Frames dialog box.

70. In this dialog box click the **OK** button to display the moment diagram.

    Note: This moment diagram is much sharper than the previous one.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "S"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

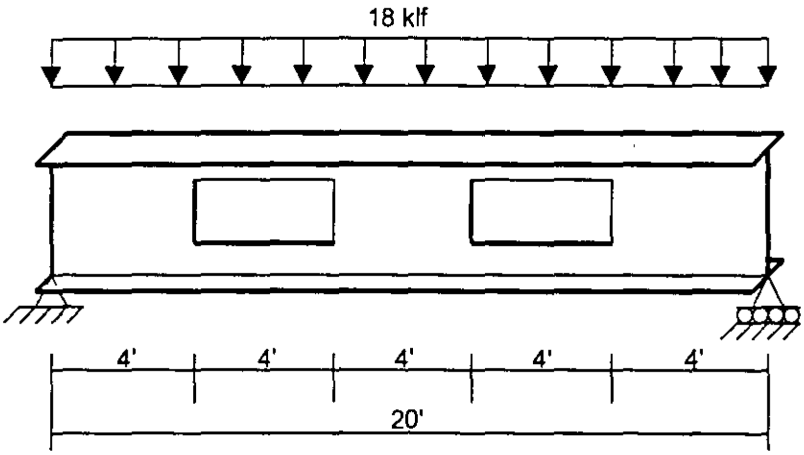
Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem S

Finite Element Model of Steel Beam With Web Openings

Steel
E =29000 ksi
Poissons Ratio = 0.3
Built-Up Beam: d = 40 in
t_w=0.75 in
b_f=16 in
t_f= 2 in
Beam openings are 20" high by 48" wide centered over the depth of the beam

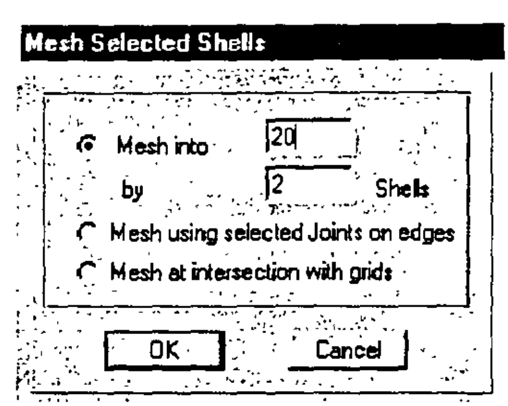
To Do
Construct a finite element model of this simply supported beam using shell elements. Detemine moments and shears at center of beam and at center of left opening. Plot shear stresses (S12) in beam web.



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem S Solution

1. Click the drop down box in the status bar to change the units to kip-in.
2. From the **File** menu select **New Model....** This displays the Coordinate System Definition dialog box.
3. In this dialog box
   - Select the Cartesian Tab.
   - In the Number of Grid Spaces area type **1** in the X direction edit box.
   - In the Number of Grid Spaces area type **2** in the Y direction edit box.
   - In the Number of Grid Spaces area type **1** in the Z direction edit box.
   - In the Grid Spacing area type **240** in the X direction edit box.
   - In the Grid Spacing area type **8** in the Y direction edit box.
   - In the Grid Spacing area type **40** in the Z direction edit box.
   - Click the **OK** button.
4. Click in the window titled X-Y Plane @ Z=40 to make sure it is active. The window is highlighted when it is active. The screen appears as shown in Figure S-1.
5. Click the **Draw Rectangular Shell Element** button on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
6. Click on the point labeled "A" and then the point labeled "B" in Figure S-1 to enter a shell element.
7. Click the **Pointer** button to exit Draw Mode and enter Select Mode.
8. Click on the shell element to select it.
9. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.
10. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.



11. Click the **Select All** button on the side toolbar to select all elements.

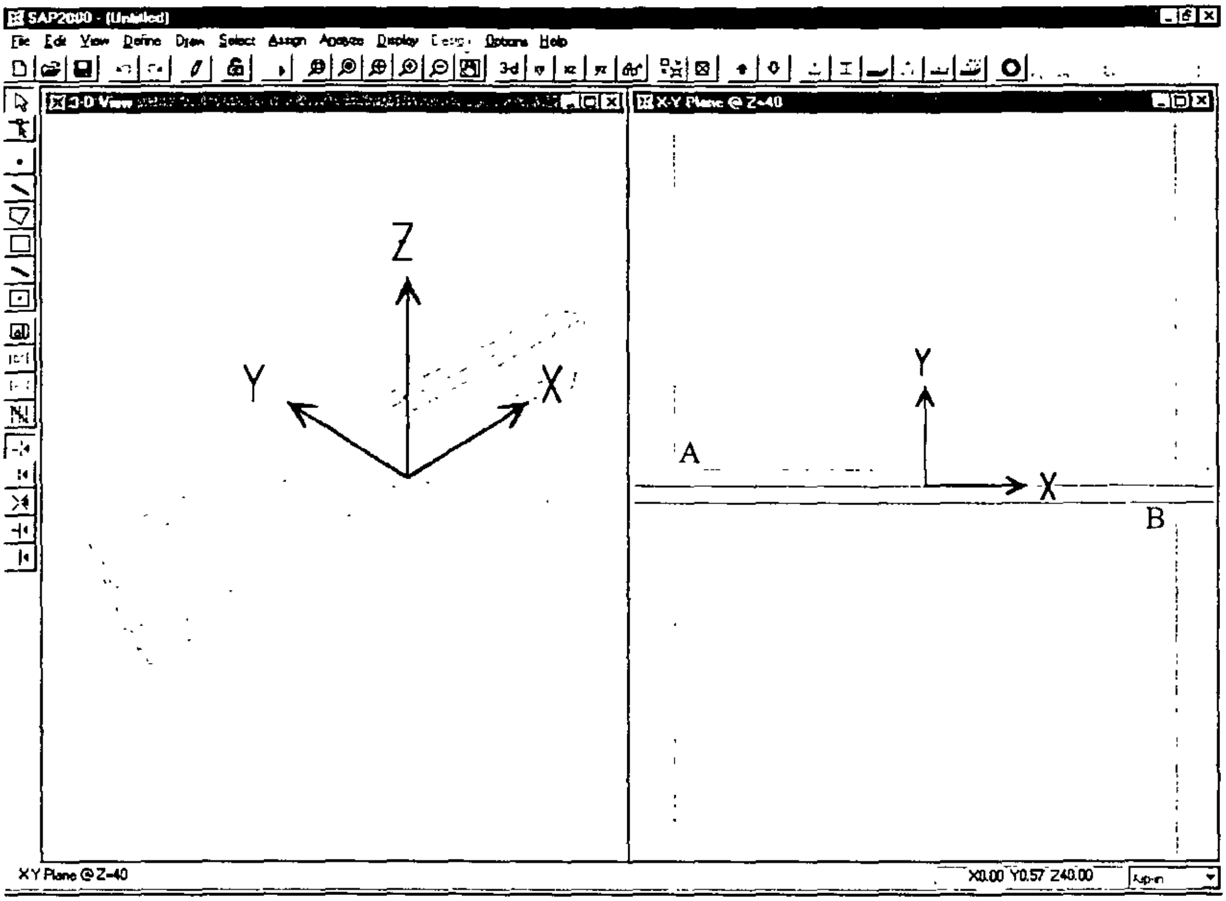


Figure S-1: Screen After Step 4

12. From the **Edit** menu select **Copy**.

13. From the **Edit** menu select **Paste...** to display the Paste Coordinates dialog box.

14. In this dialog box:

    - Type **-40** in the Delta Z edit box.

    - Click the **OK** button.

15. Click in the window titled X-Y Plane @ Z=40 to make sure it is active. Do not accidentally select any elements while doing this.

16. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

17. In this dialog box:

    - Select the X-Z Plane option.

    - Type **0** in the Y= edit box.

- Click the **OK** button. The screen appears as shown in Figure S-2.

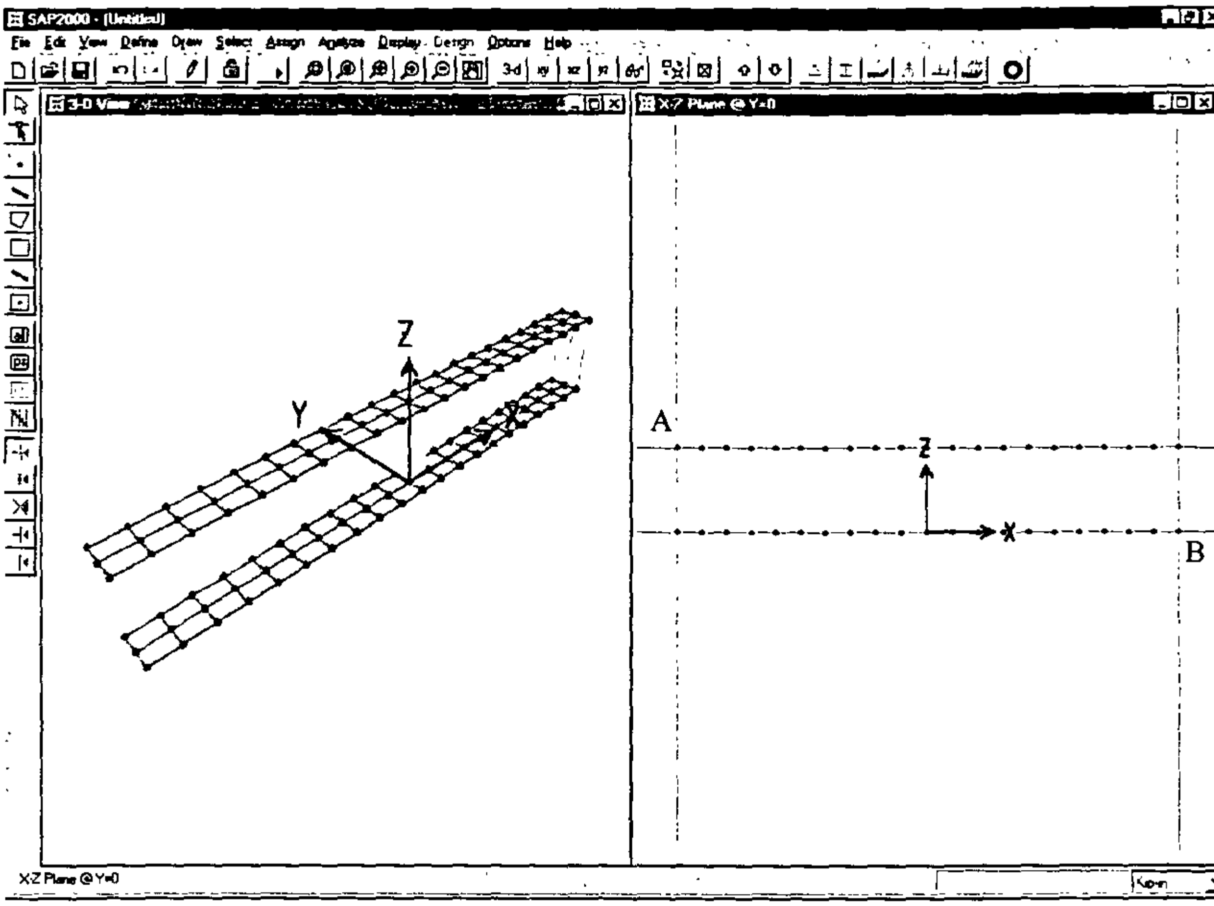


Figure S-2: Screen After Step 17

18. Click the **Draw Rectangular Shell Element** button on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.

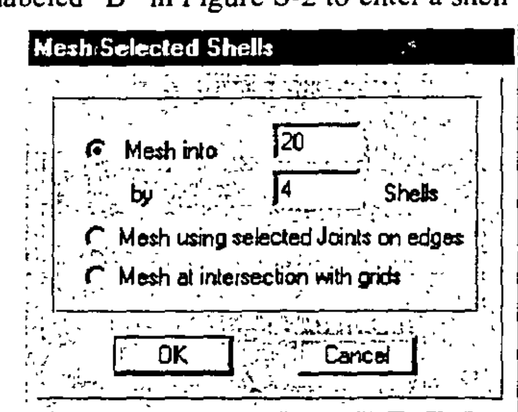
19. Click on the point labeled "A" and then the point labeled "B" in Figure S-2 to enter a shell element.

20. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

21. Click on the shell element to select it.

22. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

23. Fill in this dialog box as shown in the adjacent figure and click the **OK** button.



24. Click in the window titled 3-D View to make sure it is active.

25. Click the **Select All** button on the side toolbar to select all elements.

26. From the **Edit** menu select **Change Labels...** to display the Relabel Selected items dialog box.

27. In this dialog box:

    - Type **1** in the Joint Next Number edit box.
    - Type **1** in the Shell Next Number edit box.
    - Click the **OK** button.

Note: It is not necessary to change relabel the elements. We are doing it here to assure that later on in the solution when we refer to element labels that everybody has the same labeling system.

28. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

29. In this dialog box:

    - Check the Fill Elements box.
    - Click the **OK** button.

30. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.

31. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

32. In this dialog box:

    - Check the Labels box in the Shells area.
    - Check the Fill Elements box.
    - Click the **OK** button.

33. Select shell elements 65, 66, 67, 68, 85, 86, 87 and 88 by "windowing".

34. Press the Delete key on the keyboard to delete these shell elements.

35. Select shell elements 73, 74, 75, 76, 93, 94, 95 and 96 by "windowing".

36. Press the Delete key on the keyboard to delete these shell elements.

37. Click the **Refresh Window** button on the main toolbar to refresh the view.

38. Click in the window titled 3-D View to make sure it is active.

39. Click the **Refresh Window** button on the main toolbar to refresh the view.

40. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.

41. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

42. In this dialog box:

    - Check the Labels box in the Joints area.
    - Uncheck the Labels box in the Shells area.
    - Click the **OK** button.

43. Select joint 22 (lower left-hand corner) by clicking on it.

44. From the **Assign** menu, choose **Joint**, and then **Restraints...**from the submenu. This will display the Joint Restraints dialog box.

45. In this dialog box:

    - Verify that the Translation 1 and Translation 3 boxes are checked.
    - Uncheck the Translation 2 box.
    - Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.
    - Click the **OK** button.

46. Select joint 42 (lower right-hand corner) by clicking on it.

47. From the **Assign** menu, choose **Joint**, and then **Restraints...**from the submenu. This will display the Joint Restraints dialog box.

48. In this dialog box:

    - Uncheck the Translation 1 box.
    - Verify that the Translation 2 box is *not* checked.
    - Verify that the Translation 3 box is checked.
    - Verify that the Rotation About 1, Rotation About 2 and Rotation About 3 boxes are *not* checked.

- Click the **OK** button.

49. Select joints 148 and 168 (upper left-hand and right-hand corners) by clicking on them.

50. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

51. In this dialog box:

    - Type **-9** in the Force Global Z edit box in the Loads area.

    - Click the **OK** button.

52. Select joints 149 through 167 by "windowing".

53. From the **Assign** menu select **Joint Static Loads...** and then **Forces...** from the submenu to display the Joint Forces dialog box.

54. In this dialog box:

    - Type **-18** in the Force Global Z edit box in the Loads area.

    - Click the **OK** button.

55. Click the **Show Undeformed Shape** button to remove the display of joint force assignments.

56. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

57. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

58. In this dialog box:

    - Verify that the Modulus of Elasticity is 29000.

    - Verify that Poisson's ratio is 0.3.

    - Click the **OK** button twice to exit the dialog boxes.

59. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

60. In this dialog box:

    - Click the **Add New Section** button to display the Shell Sections dialog box.

    - In this dialog box:

- Type **WEB** in the Section Name edit box.
- Select STEEL from the Material drop-down box.
- In the Thickness area type **.75** in both the Membrane and Bending edit boxes.
- Verify that the Shell option is chosen in the Type area.
- Click the **OK** button to return to the Define Shell Sections dialog box.

- Click the **Add New Section** button to display the Shell Sections dialog box.
- In this dialog box:
  - Type **FLANGE** in the Section Name edit box.
  - Select STEEL from the Material drop-down box.
  - In the Thickness area type **2** in both the Membrane and Bending edit boxes.
  - Verify that the Shell option is chosen in the Type area.
  - Click the **OK** button twice to exit all dialog boxes.

61. Click the **Select All** button on the side toolbar to select all elements.
62. From the **Assign** menu select **Shell** and then **Sections...** to display the Define Shell Sections dialog box.
63. In this dialog box:
    - Highlight FLANGE in the Shell Sections area by clicking on it.
    - Click the **OK** button.
64. Select all of the elements in the X-Z Plane @ Y=0 window by "windowing".
65. From the **Assign** menu select **Shell** and then **Sections...** to display the Define Shell Sections dialog box.
66. In this dialog box:
    - Highlight WEB in the Shell Sections area by clicking on it.
    - Click the **OK** button.
67. Click the **Show Undeformed Shape** button to remove the display of shell section assignments.

68. Click in the window titled X-Z Plane @ Y=0 to make sure it is active.

69. From the **View** menu select **Set Limits...** to display the Set Limits dialog box.

70. In this dialog box:

- In the Set X Axis Limits area type **-12** in the Min edit box.
- In the Set X Axis Limits area type **0** in the Max edit box.
- Click the **OK** button.

71. Click the **Perspective Toggle** button on the main toolbar. A perspective view of the X-Z elevation is displayed.

72. Click the **Rubber Band Zoom** button on the main toolbar and zoom in on the view by "drawing" a window tightly around it.

73. Select all of the joints on the right side of the section by "windowing".

74. Click the **Set Intersecting Line Select Mode** button and select all of the shell elements by "drawing" a line through them. There should now be 9 joints and 8 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

75. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

76. In this dialog box:

- Type **CENTER** in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

77. Click the **Show Undeformed Shape** button to reset the limits of the display.

78. From the **View** menu select **Refresh View** to rescale the view.

79. From the **View** menu select **Set Limits...** to display the Set Limits dialog box.

80. In this dialog box:

- In the Set X Axis Limits area type **-60** in the Min edit box.
- In the Set X Axis Limits area type **-48** in the Max edit box.
- Click the **OK** button.

81. Click the **Rubber Band Zoom** button on the main toolbar and zoom in on the view by "drawing" a window tightly around it.

82. Select all of the joints on the right side of the section by "windowing".

83. Click the **Set Intersecting Line Select Mode** button and select all of the shell elements by "drawing" a line through them. There should now be 8 joints and 6 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

84. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

85. In this dialog box:

    - Type **LEFT** in the Groups edit box.
    - Click the **Add New Group Name** button.
    - Click the **OK** button.

86. Select all of the joints on the right side of the top half of the section by "windowing".

87. Click the **Set Intersecting Line Select Mode** button and select all of the shell elements in the top half by "drawing" a line through them. There should now be 4 joints and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

88. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

89. In this dialog box:

    - Type **LEFTTOP** in the Groups edit box.
    - Click the **Add New Group Name** button.
    - Click the **OK** button.

90. Select all of the joints on the right side of the bottom half of the section by "windowing".

91. Click the **Set Intersecting Line Select Mode** button and select all of the shell elements in the bottom half by "drawing" a line through them. There should now be 4 joints and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

92. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

93. In this dialog box:

    - Type **LEFTBOT** in the Groups edit box.

- Click the **Add New Group Name** button.

- Click the **OK** button.

94. Click the **Show Undeformed Shape** button to reset the limits of the display.

95. From the **View** menu select **Refresh View** to rescale the view.

96. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

97. In this dialog box:

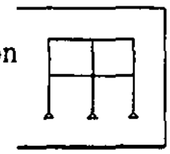
    - Uncheck the Labels box in the Joints area.

    - Click the **OK** button.

98. Click on the **xz 2D View** button on the main toolbar to return from the perspective view to a 2-D view. Note the title of the window is X-Z Plane @ Y=0.

99. Click the "X" in the upper right-hand corner of the 3-D View window to close it.

100. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.

    - Click the **OK** button.

101. Click the **Run Analysis** button to run the analysis.

102. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

103. From the **Display** menu select **Show Group Joint Force Sums...** to display the Select Groups dialog box.

104. In this dialog box:

    - Click on CENTER to highlight it.

    - Holding down the Shift key on the keyboard click on LEFTTOP to highlight it. All of the groups except the default ALL should now be highlighted.

    - Click the **OK** button to display the Group Joint Force Summation window and view the group joint force sums. When finished viewing the group joint force sums, click the "X" in the top right-hand corner of the Group Joint Force Summation window to close it.

105. From the **Display** menu select **Show Element Forces/Stresses** and then **Shells...** from the submenu to display the Element Force/Stress Contours For Shells dialog box.

106. In this dialog box:

- Click the Stresses option button to select it.
- In the Component area click the S12 option button to select it.
- In the Stress Averaging area verify that the At All Joints option is selected.
- Check the Display on Deformed Shape check box.
- Click the **OK** button to display the shell stresses. The screen should look similar to that in Figure S-3.

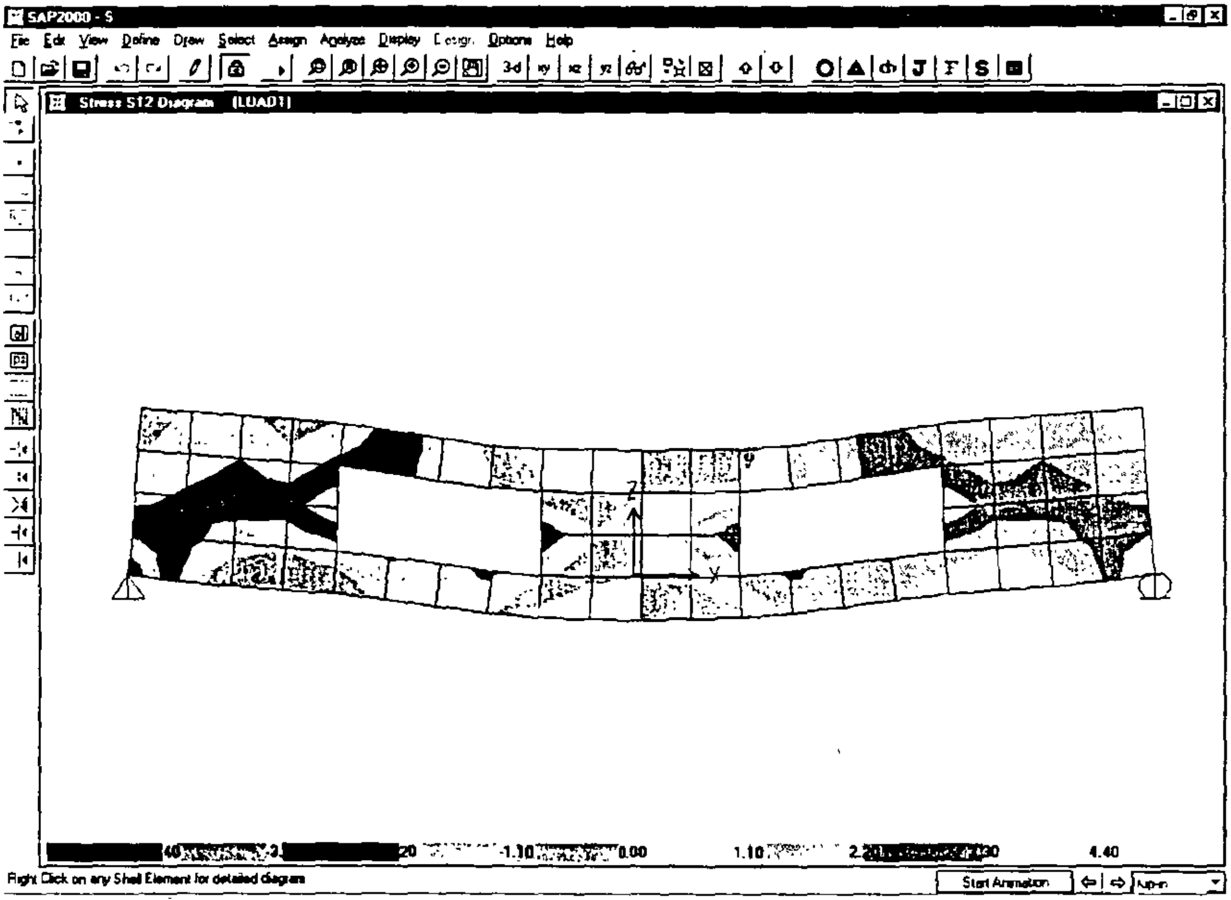


Figure S-3: Shell Stresses

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "T"

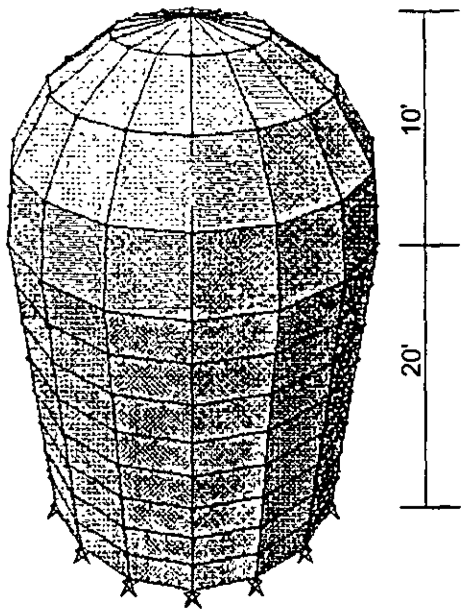
EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels. 5521 4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

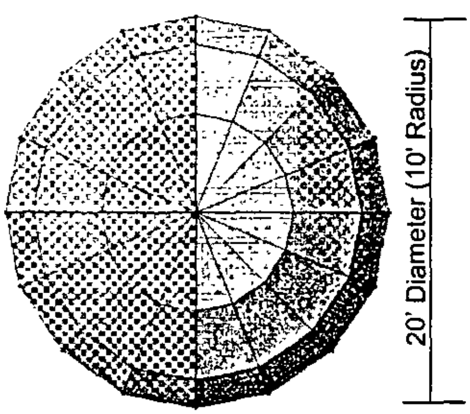
Problem T

Domed Cylindrical Structure

To Do
Create the model of this cylinder topped by a circular dome. Can you create this model in one minute or less?



Three Dimensional Perspective View



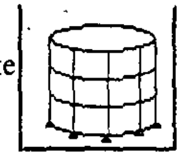
Top View

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem T Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Cylinder** template  button to display the Cylinder dialog box.

4. In this dialog box:

   - Accept the default Number of Circumferential Spaces, 16.
   - Type **8** in the Number of Height Spaces edit box.
   - Type **20** in the Cylinder Height edit box.
   - Type **10** in the Radius edit box.
   - Click the **OK** button.

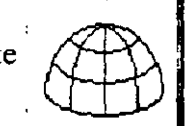
5. Click in the "X" in the upper right-hand corner of the r-theta Plane @ Z=20 window to close it.

6. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements**... from the **View** menu) to display the Set Elements Dialog box.

7. In this dialog box:

   - Check the Fill Elements box.
   - Click the **OK** button.

8. From the **Edit** menu select **Add To Model From Template...** to display the Model Templates dialog box.

9. In this dialog box click on the **Dome** template  button to display the Dome dialog box.

10. In this dialog box:

    - Accept the default Number of Circumferential Spaces, 16.
    - Type **4** in the Number of Segments edit box.

- Type **10** in the Radius edit box.
- Accept the default Roll Down Angle, 90.
- Uncheck the Restraints check box if it is not already unchecked.
- Click the **Advanced** button to display the Location and Orientation dialog box.

  Note: In this dialog box you are defining a location of the origin of a new coordinate system and its rotation with respect to the global coordinate system. The origin point of the template will be inserted at the origin of this newly defined coordinate system.

- In this dialog box:

  - In the Translations area type **20** in the Z edit box.
  - Click the **OK** button twice to exit all dialog boxes.

11. Press the F7 key on the keyboard to toggle the grid lines off.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "U"

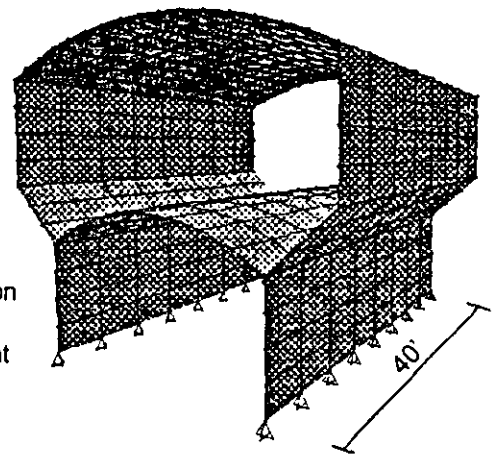
EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

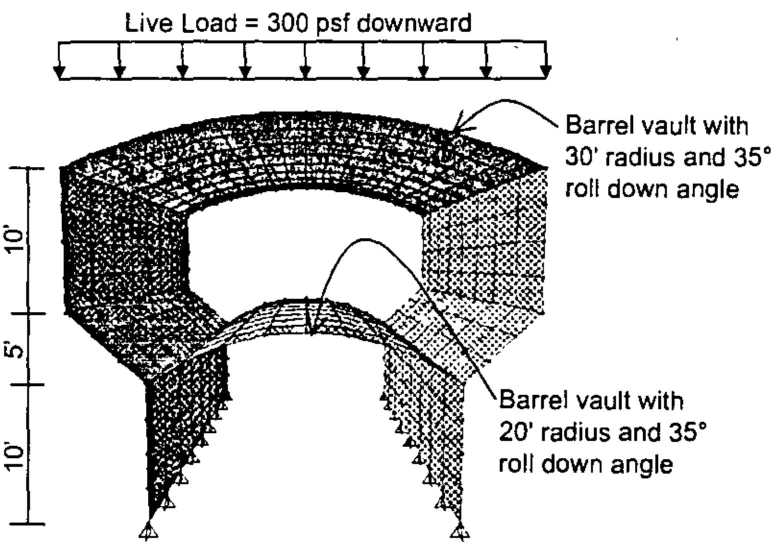
Problem U

Barrel Vaulted Structure

Concrete
E =3600 ksi
Poissons Ratio = 0.2
12" thick concrete walls and slabs

To Do
Determine the maximum deflection at the center of upper and lower barrel vaults due to the self weight of the structure. Also determine the maximum deflection at center of upper and lower barrel vaults due to the self weight plus the prescribed live load applied to the top barrel vault.



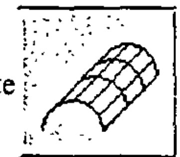


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem U Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Barrel** template  button to display the Barrel dialog box.

4. In this dialog box:

   - Accept the default Number of Circumferential Spaces, 8.
   - Accept the default Number of Span Spaces, 8.
   - Type **40** in the Span edit box.
   - Type **20** in the Radius edit box.
   - Type **35** in the Roll Down Angle edit box.
   - Uncheck the Restraints check box.
   - Click the **OK** button.

5. Click in the 3-D View window to make sure it is active. The window is highlighted when it is active.

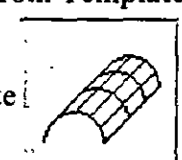
6. Click the **Select All** button all located on the side toolbar to select the entire structure.

7. From the **Edit** menu select **Move...** to display the Move Selected Points dialog box.

8. In this dialog box:

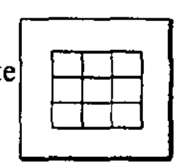
   - Type **10** in the Delta Z edit box.
   - Click the **OK** button.

9. From the **Edit** menu select **Add To Model From Template...** to display the Model Templates dialog box.

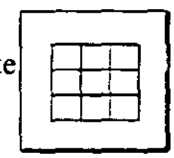
10. In this dialog box click on the **Barrel** template  button to display the Barrel dialog box.

11. In this dialog box:

- Accept the default Number of Circumferential Spaces, 8.
- Accept the default Number of Span Spaces, 8.
- Type **40** in the Span edit box.
- Type **30** in the Radius edit box.
- Type **35** in the Roll Down Angle edit box.
- Uncheck the Restraints check box if it is not already unchecked.
- Click the **Advanced** button.
  - ➢ Type **25** in the Z edit box in the Translations area.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

12. Click in the Y-Z Plane @ X=20 window to make sure it is active.
13. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements**... from the **View** menu) to display the Set Elements Dialog box.
14. In this dialog box:
    - Check the Labels box in the Joints area.
    - Click the **OK** button.
15. Right click on joint 154 in window with the Y-Z Plane @ X=20. The Joint Information dialog box is displayed.
16. Highlight the Y-coordinate (-17.2073) and press the Ctrl and the C keys on the keyboard at the same time to copy the value to the clipboard.
17. Click the **Cancel** button to close the Joint Information dialog box.
18. From the **Edit** menu select **Add To Model From Template...** to display the Model Templates dialog box.
19. In this dialog box click on the **Shear Wall** template  button to display the Shear Wall dialog box.
20. In this dialog box
    - Type **8** in the Number of Spaces Along X edit box.
    - Type **4** in the Number of Spaces Along Z edit box.

- Type **5** Space Width Along X edit box.
- Type **2.5** Space Width Along Z edit box.
- Uncheck the Restraints check box if it is not already unchecked.
- Click the **Advanced** button.
    - ➢ Highlight the Y edit box and press the Ctrl and the V keys on the keyboard at the same time to paste the -17.2073 value from the clipboard.
    - ➢ Type **15** in the Z edit box in the Translations area.
    - ➢ Click the **OK** button twice to exit all dialog boxes.

21. Right click on joint 73 in window with the Y-Z Plane @ X=20. The Joint Information dialog box is displayed.
22. Highlight the Y-coordinate (-11.4715) and press the Ctrl and the C keys on the keyboard at the same time to copy the value to the clipboard.
23. Click the **Cancel** button to close the Joint Information dialog box.
24. From the **Edit** menu select **Add To Model From Template...** to display the Model Templates dialog box.
25. In this dialog box click on the **Shear Wall** template  button to display the Shear Wall dialog box.
26. In this dialog box
    - Type **8** in the Number of Spaces Along X edit box.
    - Type **4** in the Number of Spaces Along Z edit box.
    - Type **5** Space Width Along X edit box.
    - Type **2.5** Space Width Along Z edit box.
    - Uncheck the Restraints check box if it is not already unchecked.
    - Click the **Advanced** button.
        - ➢ Highlight the Y edit box and press the Ctrl and the V keys on the keyboard at the same time to paste the -11.4715 value from the clipboard.
        - ➢ Click the OK button twice to exit all dialog boxes.
27. Click the **Pointer** button on the side tool bar to enter select mode.

28. Click in the Y-Z Plane @ X=20 window to make sure it is active.

29. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

30. In this dialog box:

- Select the X-Z Plane option.
- Type **-11.4715** in the Y= edit box.
- Click the **OK** button.

31. Select all of the elements in this view by "windowing"

32. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

33. In this dialog box:

- Select the X-Z Plane option.
- Type **-17.2073** in the Y= edit box.
- Click the **OK** button.

34. Select all of the elements in this view by "windowing"

35. From the **View** menu select **Show Selection Only**.

36. Click the **Perspective Toggle** button on the main toolbar.

37. From the **View** menu select **Show Grid** to toggle the grids off.

38. From the **View** menu select **Show Axes** to toggle the axes off.

39. From the **Draw** menu select **Draw Quad Shell Element.**

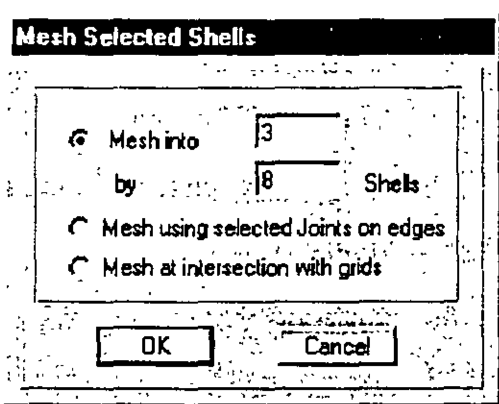
40. In the X-Z perspective view click on joints 163, 195, 73 and 1, in that order, to draw a shell element.

41. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

42. Click on the just drawn shell element to select it.

43. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

44. Fill in this dialog box as shown in the figure and



click the **OK** button.

Note: It is difficult to figure out whether to mesh shells 3 by 8 or 8 by 3. It is often easiest to just go ahead and try one way, and if it isn't right, simply click the Undo button on the main toolbar and mesh the shells other way.

45. Select all of the elements in the X-Z perspective view by "windowing".

46. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

47. In this dialog box:

    - Select the Mirror Tab.
    - In the Mirror About area select the XZ Plane option.
    - In the Ordinate area type **0** in the Y edit box if it is not already there.
    - Click the **OK** button.

48. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

49. In this dialog box:

    - Select the X-Y Plane option.
    - Type **0** in the Z= edit box.
    - Click the **OK** button.

50. Select all of the joints at this level by "windowing".

51. From the Assign menu select Joint and then Restraints... from the submenu to display the Joint Restraints dialog box.

52. In this dialog box:

    - Verify that the Translation 1, 2 and 3 boxes are all checked.
    - Click the **OK** button.

53. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

54. In this dialog box:

    - Select the Y-Z Plane option.
    - Type **20** in the X= edit box.

- Click the **OK** button.

55. Click the **Show Undeformed Shape** button to reset the window title.

56. From the **View** menu select **Show All**.

57. Click the drop down box in the status bar to change the units to kip-in. Kip-in

58. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

59. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

60. In this dialog box:

    - Type **3600** in the Modulus of Elasticity edit box.
    - Verify **.2** is entered in the Poisson's Ratio edit box.
    - Click the **OK** button twice to exit all dialog boxes.

61. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

62. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

63. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

64. In this dialog box:

    - Verify that 4.657E-03 is entered in the Mass Per Unit Volume edit box.
    - Verify that .15 is entered in the Weight Per Unit Volume edit box.
    - Click the **OK** button twice to exit all dialog boxes.

65. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

66. In this dialog box:

    - Click the **Modify/Show Section** button to display the Shell Sections dialog box.
    - In this dialog box:
        - ➢ Verify that the Material is CONC.
        - ➢ Verify that both the Membrane and Bending thicknesses are 1.

- ➢ Verify that the Shell option is chosen in the Type area.
- ➢ Click the **OK** button twice to exit all dialog boxes.

67. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

68. In this dialog box:
    - Type **DL** in the Load edit box.
    - Click the **Change Load** button.
    - Type **LL** in the Load edit box.
    - Select Live from the Type drop-down box.
    - Type **0** in the Self Weight Multiplier edit box.
    - Click the **Add New Load** button.
    - Click the **OK** button.

69. From the **Define** menu select **Load Combinations...** to display the Define Load Case Combinations dialog box.

70. In this dialog box:
    - Click the **Add New Combo** button to display the Load Combination Data dialog box..
    - In this dialog box:
        - ➢ Accept the default Load Combination Name COMB1.
        - ➢ Accept the default Load Combination Type, Add.
        - ➢ Type **COMB1: DL + LL** in the Title edit box.
        - ➢ Verify that DL Load Case appears in the Case Name drop-down box, and that the Scale Factor is 1.
        - ➢ Click the **Add** button.
        - ➢ Select LL Load Case from the Case Name drop-down box
        - ➢ Click the **Add** button.
        - ➢ Click **OK** twice to exit all dialog boxes.

71. Click in the 3-D View window to activate it. Be careful not to accidentally select any members when you do this.

72. From the View menu select Set Limits... to display the Set Limits dialog box.

73. In this dialog box:

  - Type **25** in the Min edit box in the Set Z Axis limits area.
  - Click the **OK** button.

74. Select all of the displayed elements in the 3D View window (i.e., all elements in the roof barrel) by "windowing".

75. From the **Assign** menu choose **Shell Static Loads...** and then choose **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

76. In this dialog box:

  - Select LL from the Load Case Name drop-down box.
  - Type **-.3** in the Load edit box.
  - Verify that Global Z is selected in the Dir drop-down box.
  - Click the **OK** button.

77. Click the **Show Undeformed Shape** button to remove the display of shell uniform loads and to remove the limits that were set.

78. Click the **Run Analysis** button to run the analysis.

79. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

80. Click in the window titled Y-Z Plane @ X=20 to activate it.

81. Click the **Display Static Deformed Shape** button on the main toolbar to display the Deformed Shape dialog box.

82. In this dialog box:

  - Select the DL Load Case from the Load drop-down box.
  - Select the Auto Scaling option.
  - Click the **OK** button.

83. Click the drop down box in the status bar to change the units to kip-in. Kip-in

84. Right click on the center joint of the upper barrel vault to see its self weight displacement in inches.

85. Right click on the center joint of the lower barrel vault to see its self weight displacement in inches.

86. Click the **Display Static Deformed Shape** button on the main toolbar to display the Deformed Shape dialog box.

87. In this dialog box:

    - Select the COMB1 Combo from the Load drop-down box.

    - Click the **OK** button.

88. Right click on the center joint of the upper barrel vault to see its self weight plus live load displacement in inches.

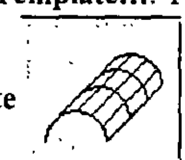
89. Right click on the center joint of the lower barrel vault to see its self weight plus live load displacement in inches.

Alternative Problem U Solution

Now an alternative method of creating the model is shown. This alternative method does not use the Advanced button feature of the **Add To Model From Template** feature. Instead it makes use of groups and the move command to accomplish essentially the same thing. It is a good example of one of the uses of groups. This alternative method takes more steps than the initial method shown. The initial method is recommended; this method is just shown for example purposes.

A1. Click the drop down box in the status bar to change the units to kip-ft.

A2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

A3. In this dialog box click on the **Barrel Template**  button to display the Barrel dialog box.

A4. In this dialog box

- Accept the default Number of Circumferential Spaces, 8.

- Accept the default Number of Span Spaces, 8.

- Type **40** in the Span edit box.
- Type **20** in the Radius edit box.
- Type **35** in the Roll Down Angle edit box.
- Uncheck the Restraints check box.
- Click the **OK** button.

A5. Click in the 3-D View window to make sure it is active. The window is highlighted when it is active.

A6. Click the **Select All** button 

located on the side toolbar to select the entire structure.

A7. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

A8. In this dialog box:

- Type **LOBARREL** in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

A9. Click the **Restore Previous Selection** button 

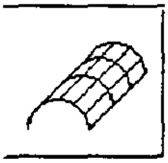
on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

A10. From the **Edit** menu select **Move...** to display the Move Selected Points dialog box.

A11. In this dialog box:

- Type **10** in the Delta Z edit box.
- Click the **OK** button.

A12. From the **Edit** menu select **Add To Model From Template...** to display the Model Templates dialog box.

A13. In this dialog box click on the **Barrel** template  button to display the Barrel dialog box.

A14. In this dialog box

- Accept the default Number of Circumferential Spaces, 8.
- Accept the default Number of Span Spaces, 8.
- Type **40** in the Span edit box.

- Type **30** in the Radius edit box.
- Type **35** in the Roll Down Angle edit box.
- Uncheck the Restraints check box if it is not already unchecked.
- Click the **OK** button.

A15. Click the **Pointer** button on the side tool bar to enter select mode.

A16. Click the **Select All** button all located on the side toolbar to select the entire structure.

A17. From the **Select** menu choose **Deselect** and then choose **Groups...** from the submenu to display the Select Groups dialog box.

A18. In this dialog box:

- Click on the LOBARREL group to highlight it.
- Click the **OK** button.

A19. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

A20. In this dialog box:

- Type **HIBARREL** in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

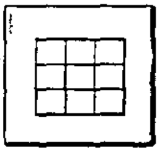
A21. Click the **Restore Previous Selection** button ps on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

A22. From the **Edit** menu select **Move...** to display the Move Selected Points dialog box.

A23. In this dialog box:

- Type **25** in the Delta Z edit box.
- Click the **OK** button.

A24. From the **Edit** menu select **Add To Model From Template...** to display the Model Templates dialog box.

A25. In this dialog box click on the **Shear Wall** template  button to display the Shear Wall dialog box.

A26. In this dialog box

- Type **8** in the Number of Spaces Along X edit box.
- Type **4** in the Number of Spaces Along Z edit box.
- Type **5** Space Width Along X edit box.
- Type **2.5** Space Width Along Z edit box.
- Uncheck the Restraints check box if it is not already unchecked.
- Click the **OK** button.

A27. Click the **Pointer** button on the side tool bar to enter select mode.

A28. Click the **Select All** button located on the side toolbar to select the entire structure.

A29. From the **Select** menu choose **Deselect** and then choose **Groups...** from the submenu to display the Select Groups dialog box.

A30. In this dialog box:

- Click on the HIBARREL group to highlight it.
- Hold down on the Ctrl key on the keyboard and click on the LOBARREL group to add it to the selection
- Click the **OK** button.

A31. From the **Assign** menu select **Group Name...** to display the Assign Group dialog box.

A32. In this dialog box:

- Type **WALL1** in the Groups edit box.
- Click the **Add New Group Name** button.
- Click the **OK** button.

A33. Click in the Window titled Y-Z Plane @ X=20 to activate it.

A34. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

A35. In this dialog box:

- Check the Labels box in the Joints area.
- Click the **OK** button.

A36. Right click on joint 154. The Joint Information dialog box is displayed.

A37. In this dialog box:

- Note that the Y-coordinate of this joint is -17.2073.
- Click the **OK** button to close the dialog box.

A38. From the **Select** menu choose **Select** and then choose **Groups...** from the submenu to display the Select Groups dialog box.

A39. In this dialog box:

- Click on the WALL1 group to highlight it.
- Click the **OK** button.

A40. From the **Edit** menu select **Move...** to display the Move Selected Points dialog box.

A41. In this dialog box:

- Type **-17.2073** in the Delta Y edit box.
- Type **15** in the Delta Z edit box.
- Click the **OK** button.

A42. Right click on joint 73. The Joint Information dialog box is displayed.

A43. In this dialog box:

- Note that the Y-coordinate of this joint is -11.4715.
- Note that the Y-direction distance between joints 154 and 73 can be calculated as 17.2073 - 11.4715 = 5.7358.
- Click the **OK** button to close the dialog box.

A44. From the **Select** menu choose **Select** and then choose **Groups...** from the submenu to display the Select Groups dialog box.

A45. In this dialog box:

- Click on the WALL1 group to highlight it.
- Click the **OK** button.

A46. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

A47. In this dialog box:

- Select the Linear Tab.
- Type **5.7358** in the Y Distance edit box.
- Type **-15** in the Z Distance edit box.
- Type **1** in the Number edit box.
- Click the **OK** button.

A48. Click the **Select All** button located on the side toolbar to select the entire structure.

A49. From the **Select** menu choose **Deselect** and then choose **Groups...** from the submenu to display the Select Groups dialog box.

A50. In this dialog box:

- Click on the HIBARREL group to highlight it.
- Hold down on the Ctrl key on the keyboard and click on the LOBARREL group to add it to the selection
- Click the **OK** button.

A51. From the **View** menu select **Show Selection Only**.

A52. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

A53. In this dialog box:

- Select the X-Z Plane option.
- Type **-17.2073** in the Y= edit box.
- Click the **OK** button.

A54. Click the **Perspective Toggle** button on the main toolbar.

A55. From the **View** menu select Show Grid to toggle the grids off.

A56. From the **Draw** menu select **Draw Quad Shell Element.**

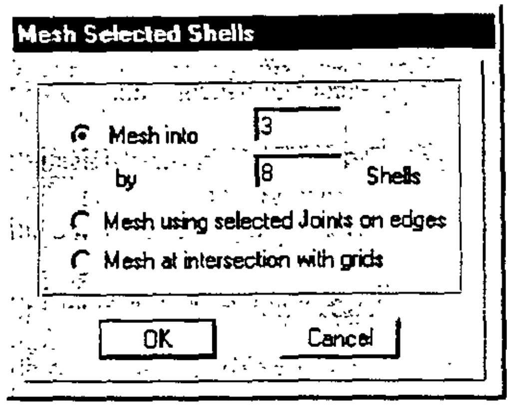
A57. In the X-Z perspective view click on joints 163, 203, 73 and 1, in that order, to draw a shell element.

A58. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

A59. Click on the just drawn shell element to select it.

A60. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

A61. Fill in this dialog box as shown in the figure and click the **OK** button.



Note: It is difficult to figure out whether to mesh shells 3 by 8 or 8 by 3. It is often easiest to just go ahead and try one way, and if it isn't right, simply click the Undo button on the main toolbar and mesh the shells the other way.

A62. Select all of the elements in the X-Z perspective view by "windowing".

A63. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

A64. In this dialog box:

- Select the Mirror Tab.
- In the Mirror About area select the XZ Plane option.
- In the Ordinate area type **0** in the Y edit box if it is not already there.
- Click the **OK** button.

A65. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

A66. In this dialog box:

- Select the X-Y Plane option.
- Type **0** in the Z= edit box.
- Click the **OK** button.

A67. Select all of the joints at this level by "windowing".

A68. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

A69. In this dialog box:

- Verify that the Translation 1, 2 and 3 boxes are all checked.

- Click the OK button.

A70. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

A71. In this dialog box:

- Select the Y-Z Plane option.
- Type **20** in the X= edit box.
- Click the **OK** button.

A72. Click the **Show Undeformed Shape** button to reset the window title.

A73. From the **View** menu select **Show All**.

A74. Click the drop down box in the status bar to change the units to kip-in. Kip-in

A75. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

A76. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

A77. In this dialog box:

- Type **3600** in the Modulus of Elasticity edit box.
- Verify .2 is entered in the Poisson's Ratio edit box.
- Click the **OK** button twice to exit all dialog boxes.

A78. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

A79. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

A80. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

A81. In this dialog box:

- Verify that 4.657E-03 is entered in the Mass Per Unit Volume edit box.
- Verify that **.15** is entered in the Weight Per Unit Volume edit box.
- Click the **OK** button twice to exit all dialog boxes.

A82. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

A83. In this dialog box:

- Click the **Modify/Show Section** button to display the Shell Sections dialog box.
- In this dialog box:
    - ➢ Verify that the Material is CONC.
    - ➢ Verify that both the Membrane and Bending thicknesses are 1.
    - ➢ Verify that the Shell option is chosen in the Type area.
    - ➢ Click the **OK** button twice to exit all dialog boxes.

A84. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

A85. In this dialog box:

- Type **DL** in the Load edit box.
- Click the **Change Load** button.
- Type **LL** in the Load edit box:
- Select Live from the Type drop-down box.
- Type **0** in the Self Weight Multiplier edit box.
- Click the **Add New Load** button.
- Click the **OK** button.

A86. From the **Define** menu select **Load Combinations...** to display the Define Load Case Combinations dialog box.

A87. In this dialog box:

- Click the Add New Combo button to display the Load Combination Data dialog box..
- In this dialog box:
    - ➢ Accept the default Load Combination Name COMB1.
    - ➢ Accept the default Load Combination Type, Add.
    - ➢ Type **COMB1: DL + LL** in the Title edit box.

- Verify that DL Load Case appears in the Case Name drop-down box, and that the Scale Factor is 1.
- Click the **Add** button.
- Select LL Load Case from the Case Name drop-down box
- Click the **Add** button.
- Click **OK** twice to exit all dialog boxes.

A88. Click in the 3-D View window to activate it. Be careful not to accidentally select any members when you do this.

A89. From the **Select** menu choose **Select** and then choose **Groups...** from the submenu to display the Select Groups dialog box.

A90. In this dialog box:

- Click on the HIBARREL group to highlight it.
- Click the **OK** button.

A91. From the **Assign** menu choose **Shell Static Loads...** and then choose **Uniform...** from the submenu to display the Shell Uniform Loads dialog box.

A92. In this dialog box:

- Select LL from the Load Case Name drop-down box.
- Type **-.3** in the Load edit box.
- Verify that Global Z is selected in the Dir drop-down box.
- Click the **OK** button.

A93. Click the **Show Undeformed Shape** button to remove the display of shell uniform loads.

A94. Click the **Run Analysis** button to run the analysis.

A95. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

A96. Click in the window titled Y-Z Plane @ X=20 to activate it.

A97. Click the **Display Static Deformed Shape** button on the main toolbar to display the Deformed Shape dialog box.

A98. In this dialog box:

- Select the DL Load Case from the Load drop-down box.
- Select the Auto Scaling option.
- Click the **OK** button.

A99. Click the drop down box in the status bar to change the units to kip-in. Kip-in

A100. Right click on the center joint of the upper barrel vault to see its self weight displacement in inches.

A101. Right click on the center joint of the lower barrel vault to see its self weight displacement in inches.

A102. Click the **Display Static Deformed Shape** button on the main toolbar to display the Deformed Shape dialog box.

A103. In this dialog box:

- Select the COMB1 Combo from the Load drop-down box.
- Click the **OK** button.

A104. Right click on the center joint of the upper barrel vault to see its self weight plus live load displacement in inches.

A105. Right click on the center joint of the lower barrel vault to see its self weight plus live load displacement in inches.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "V"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

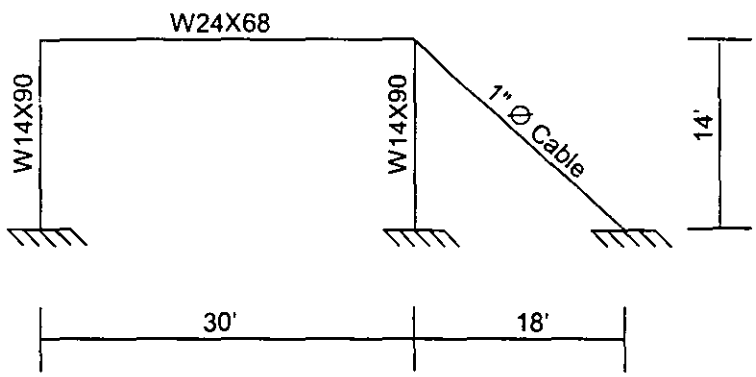
Palacio de Mineria, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem V

Temperature Loading

Steel
E =29000 ksi
Poissons Ratio = 0.3
Coefficient of thermal expansion = 0.0000065 (degrees Fahrenheit)
Beam-column connections are rigid
Cable is pinned at each end

To Do
Determine support reactions and frame displacement due to a 100° Fahrenheit temperature drop in the cable only.

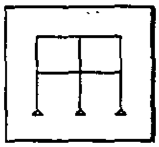


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem V Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Portal Frame** template  button to display the Portal Frame dialog box.

4. In this dialog box:

   - Type **1** in the Number of Stories edit box.
   - Type **1** in the Number of Bays edit box.
   - Type **14** in the Story Height edit box.
   - Type **30** in the Bay Width edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines edit box.

7. In this dialog box:

   - Verify that the X Direction option is selected.
   - Type **33** in the X Location edit box.
   - Click the **Add Grid Line** button.
   - Click the **OK** button.

8. Click the **xz 2D View** button xz on the main toolbar to reset the view.

9. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

10. In this dialog box:

    - Check the Labels box in the Joints area.
    - Check the Labels box in the Frames area.

- Click the **OK** button.

11. Verify that the **Snap to Joints and Grid Points** button on the side tool bar is depressed.

12. Click the **Draw Frame Element** button on the side toolbar, or select **Draw Frame Element** from the **Draw** menu.

13. Draw the cable element as follows:

    - Place the mouse pointer on joint 4. When the text box saying "Grid Intersection" appears click the left mouse button once.

    - Move the mouse pointer to to the grid intersection at the bottom of the cable (lower right-hand grid intersection). When the text box saying "Grid Intersection" appears click the left mouse button once.

    - Press the Enter key on the keyboard.

14. Click the **Pointer** button to exit draw mode and enter select mode.

15. Select joints 1, 3 and 5.

16. From the **Assign** menu, choose **Joint**, and then **Restraints...** from the submenu. This will display the Joint Restraints dialog box.

17. In this dialog box:

    - Click the **fixed base fast restraint** button to set all degrees of freedom (U1, U2, U3, R1, R2 and R3) as restrained.

    - Click the **OK** button.

18. Click the drop down box in the status bar to change the units to kip-in. Kip-in

19. From the **Define** menu select **Materials...** to display the Define Materials dialog box. Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

20. In this dialog box:

    - Verify that the modulus of elasticity is 29000, poisson's ratio is 0.3 and the coefficient of thermal expansion is 0.0000065.

    - Click the **OK** button twice to exit the dialog boxes.

21. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

22. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

23. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.

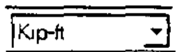
24. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W24X68 section.
    - Scroll down to the W14X90 section, and click on it while holding down the Ctrl key on the keyboard.
    - Click the **OK** button twice to return to the Define Frame Sections dialog box.

25. In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Circle item.

26. The Circle Section dialog box appears. In this dialog box:

    - Type **CABLE** in the Section Name edit box.
    - Type **1** in the Diameter (t3) edit box.
    - Click the **OK** button twice to exit all dialog boxes.

27. Click the drop down box in the status bar to change the units to kip-ft. 

28. Select the beam element (frame element 3).

29. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

30. In this dialog box:

    - Click on W24X68 in the Frame Sections area to highlight it.
    - Click the **OK** button.

31. Select the two column elements (frame elements 1 and 2).

32. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

33. In this dialog box:

    - Click on W14X90 in the Frame Sections area to highlight it.

- Click the **OK** button.

34. Select the cable element (frame element 4).

35. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

36. In this dialog box:

    - Click on CABLE in the Frame Sections area to highlight it.

    - Click the **OK** button.

37. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

38. Select the cable element (frame element 4).

39. From the **Assign** menu select **Frame** and then **Releases...** from the submenu to display the Frame Releases dialog box.

40. In this dialog box check both the Start and the End boxes for Moment 33 (Major) and then click the **OK** button.

41. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

42. In this dialog box:

    - Type **0** in the Self Weight Multiplier edit box.

    - Click the **Change Load** button.

    - Click the **OK** button.

43. Select the cable element (frame element 4).

44. From the **Assign** menu select **Frame Static Loads...** and then **Temperature...** from the submenu to display the Frame Temperature Loading dialog box.

45. In this dialog box:

    - Verify that the Temperature option is selected in the Type area.

    - Verify that the By Element option is selected in the Temperature area.

    - Type **-100** in the Temperature edit box.

    - Click the **OK** button.

46. Click the **Show Undeformed Shape** button to remove the displayed temperature assignments.

47. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

48. In this dialog box:

    - Uncheck the Labels box in the Joints area.
    - Uncheck the Labels box in the Frames area.
    - Click the **OK** button.

49. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button to set the available degrees of freedom.
    - Click the **OK** button.

50. Click the **Run Analysis** button to run the analysis.

51. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.

52. Click the **Joint Reaction.Forces** button to display the Joint Reaction Forces dialog box.

53. In this dialog box:

    - Verify that the Reactions option is selected in the Type area.
    - Click the **OK** button.

54. The reactions are displayed on the screen. You can right click on any joint to see the reactions at that joint or you can just read the reactions on the screen. If the text goes off of the screen, you can use click the **Pan** button on the main toolbar to move the display, or you can click the **Zoom Out One Step** button on the main toolbar as many times as required to resize the display. If the text is too small to read, you can zoom in on the joint, or you can change the minimum font size as described in the note below.

    *Note: To change the minimum font size select **Preferences** from the **Options** menu and make sure the Dimensions Tab is selected. In the Minimum Graphic font Size edit box input a new size, maybe 5 or 6 points. Click the **OK** button.*

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "W"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem W

Simple Beam With Trapezoidal Loads

Steel
E =29000 ksi
Poissons Ratio = 0.3
Beam = W21X50

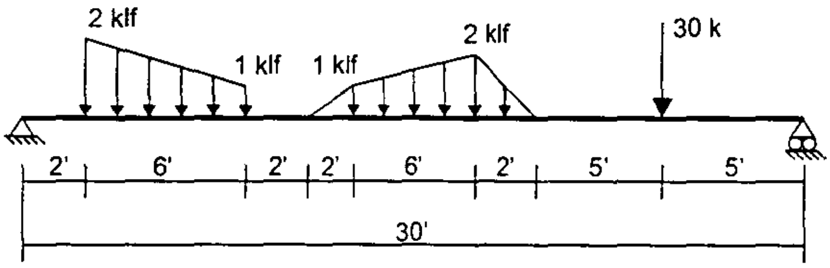
To Do
Determine midspan deflection of beam. Ignore the self weight of the beam. Create the model as follows:

1. From the File menu, choose New Model From Template. Select the Beam template in the upper left hand corner. Set the number of spans to one.

2. Define the frame section properties.

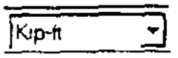
3. Apply the loads to the beam.

4. Use the Divide Frames option in the Edit Menu to break the beam into two elements with a joint at the center.

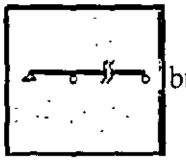


Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem W Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

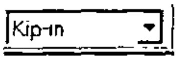
2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

4. In this dialog box:

   - Type **1** in the Number of Spans edit box.
   - Type **30** in the Span Length edit box.
   - Click the **OK** button.

5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

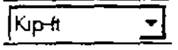
6. Click the drop down box in the status bar to change the units to kip-in. 

7. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

8. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

9. In this dialog box:

   - Verify 29000 is entered in the Modulus of Elasticity edit box.
   - Verify .3 is entered in the Poisson's Ratio edit box.
   - Accept the other default values.
   - Click the **OK** button twice to exit all dialog boxes.

10. Click the drop down box in the status bar to change the units to kip-ft. 

11. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

12. In this dialog box:

    - Type **0** in the Self Weight Multiplier edit box.
    - Click the **Change Load** button.

- Click the **OK** button.

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

14. In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

15. If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.

16. A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

    - Scroll down and click on the W21X50 section.

    - Click the **OK** button three times.

17. Select the frame element.

18. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

19. Highlight W21X50 in the Frame Sections area and click the **OK** button.

20. Click the **Show Undeformed Shape** button to remove the displayed frame section assignment.

21. Select the frame element.

22. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.

23. In this dialog box:

    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.

    - In the Point Loads area select the Absolute Distance From End I option.

    - In the Point Loads area type **25** in the first Distance edit box and type **-30** in the first Load edit box

    - Click the **OK** button.

24. Select the frame element.

25. From the **Assign** menu select **Frame Static Loads...** and then **Trapezoidal...** from the submenu to display the Trapezoidal Span Loads dialog box.

26. In this dialog box:

    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.

    - In the Trapezoidal Loads area select the Absolute Distance From End I option.

    - In the Trapezoidal Loads area type **2** in the first Distance edit box and type **-2** in the first Load edit box

    - Type **2** in the second Distance edit box and type **-2** in the second Load edit box

    - Type **8** in the third Distance edit box and type **-1** in the third Load edit box

    - Type **8** in the fourth Distance edit box and type **-1** in the fourth Load edit box

    - Click the **OK** button.

27. Select the frame element.

28. From the **Assign** menu select **Frame Static Loads...** and then **Trapezoidal...** from the submenu to display the Trapezoidal Span Loads dialog box.

29. In this dialog box:

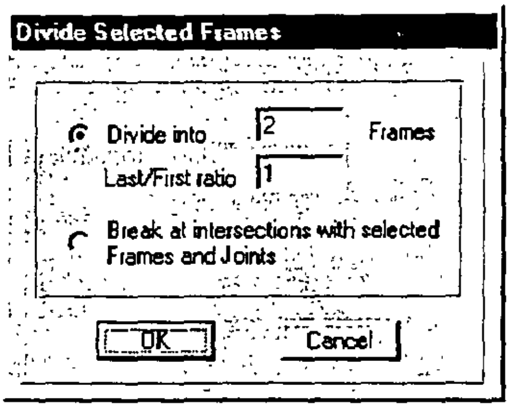
    - In the Trapezoidal Loads area type **10** in the first Distance edit box and type **0** in the first Load edit box

    - Type **12** in the second Distance edit box and type **-1** in the second Load edit box

    - Type **18** in the third Distance edit box and type **-2** in the third Load edit box

    - Type **20** in the fourth Distance edit box and type **0** in the fourth Load edit box

    - Click the **OK** button.

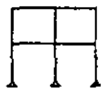


30. Select the frame element.

31. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box. Verify that the dialog box appears as shown in the figure and click the **OK** button. The frame element is broken into two elements with a joint at the center. Note that the frame loading did not change.

32. Click the **Show Undeformed Shape** button to remove the displayed frame static load assignments.

33. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
    - Click the **OK** button.

34. Click the **Run Analysis** button to run the analysis.

35. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

36. Right click on the center joint to see its displacement in feet.

37. Click the drop down box in the status bar to change the units to kip-in. 

38. Right click on the center joint to see its displacement in inches.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

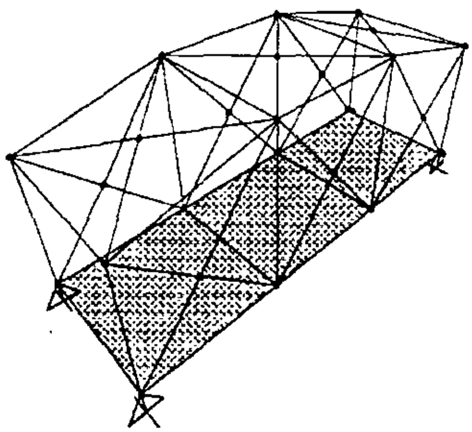
PROBLEM "X"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Mineria, Calle de Tacuba No 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem X

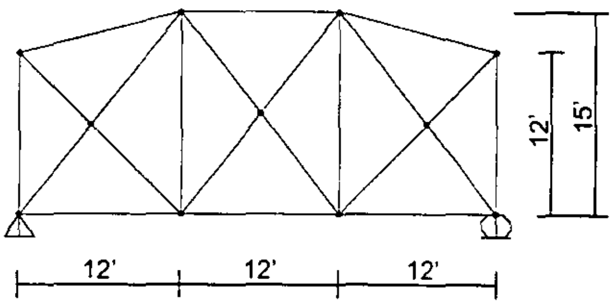
Through Truss Bridge

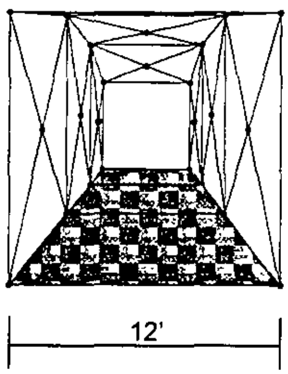


Steel
E =29000 ksi
Poissons Ratio = 0.3
All members are W6X12
Fy = 36 ksi

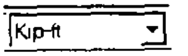
Concrete Bridge Deck
E =3600 ksi
Poissons Ratio = 0.2
12 inches thick
Live Load = 250 psf

To Do
Review steel member stresses due to self weight plus live load.
Use AISC-ASD89.



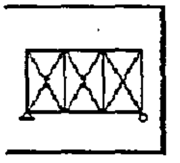


<u>Problem X Solution</u>



File **New Model From Template...**

Vertical Truss



-
-
-
- **OK**

Select All

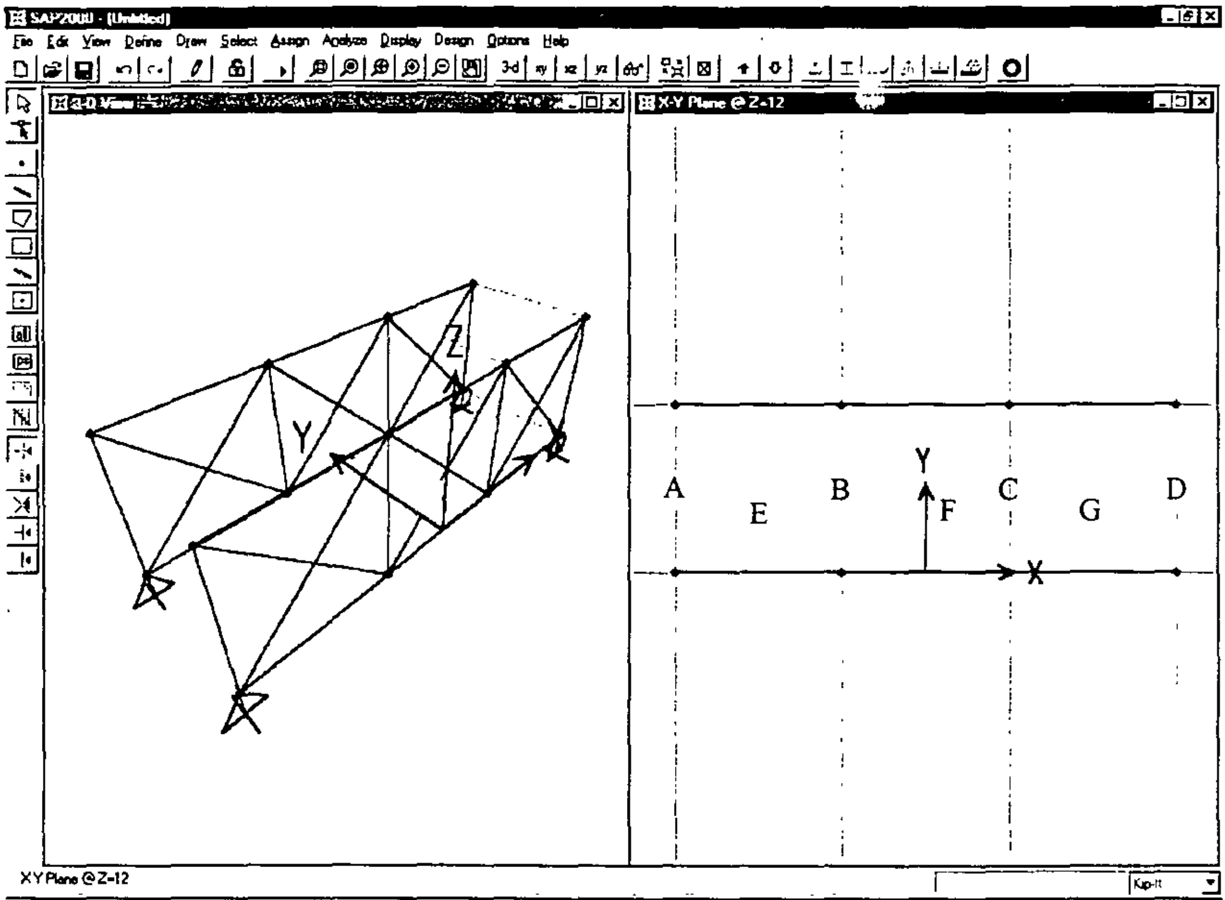
Edit **Replicate...**

-
- **12**
-
-
- **OK**

Draw **Edit Grid...**

-
- **12** **Add Grid Line**
- **OK**

xy 2D View



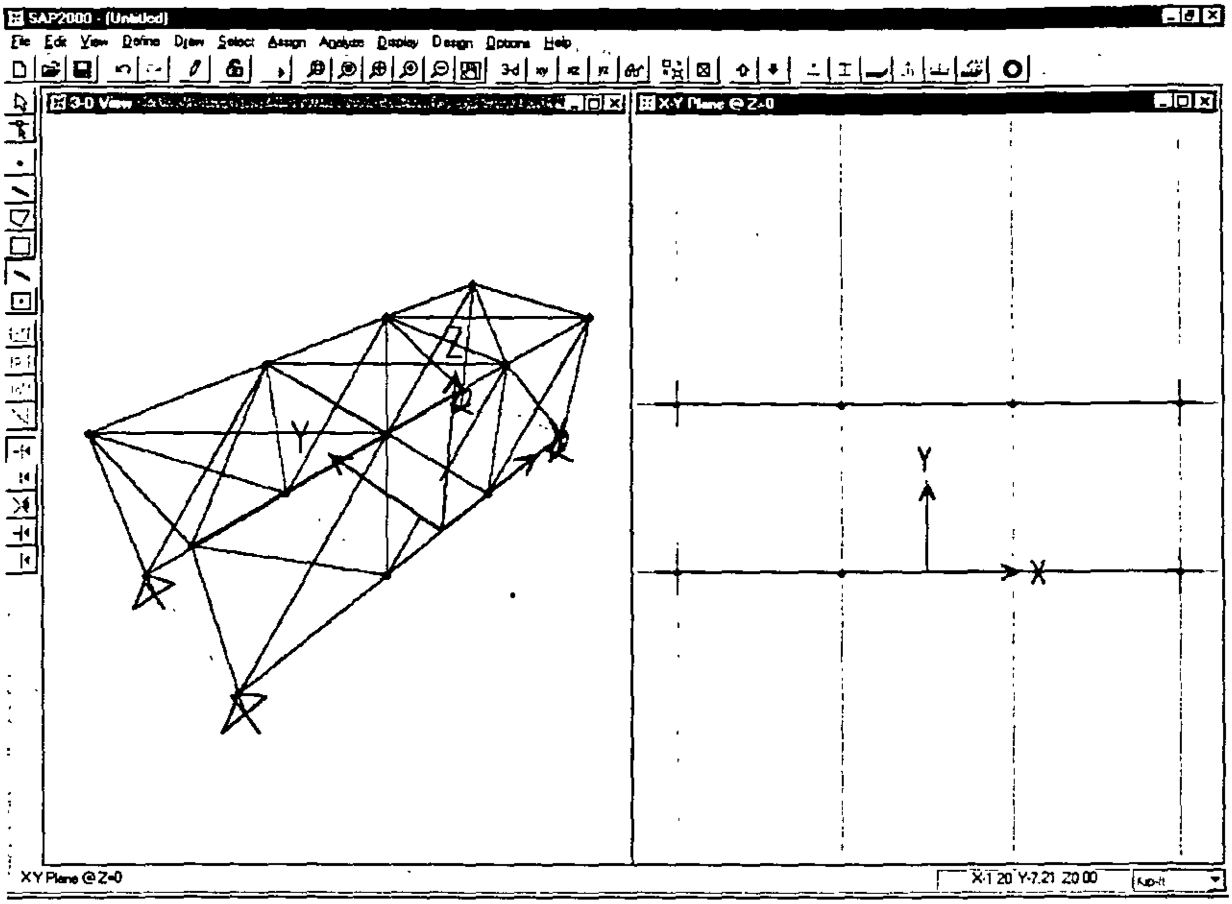
Quick Draw Frame Element **Quick Draw Frame Element** **Draw**

Click on the grid lines at the points labeled "A", "B", "C" and "D" in Figure X-1 to enter four frame elements spanning between the two vertical frames.

14. Click on the grid lines at the points labeled "E", "F" and "G" in Figure X-1 to enter three sets of diagonal frame elements spanning between the two vertical frames.

15. Click the **Down One Gridline** button on the main toolbar to display the X-Y Plane @ Z=0. The screen appears as shown in Figure X-2.

16. Click the **Quick Draw Rectangular Shell Element** button on the side toolbar (or select **Quick Draw Rectangular Shell Element** from the **Draw** menu).



Click on the points labeled "A", "B" and "C" in Figure X-2 to enter three shell elements spanning between the two vertical frames.

18. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

19. Click in the window titled X-Y Plane @ Z=0 to make sure it is active.

20. Click the **Up One Gridline** button on the main toolbar to display the plan view at Z=12.

21. Select the center four joints by clicking on them.

22. From the **Edit** menu select **Move...**

- **3**

- **OK**

Perspective Toggle

Set Intersecting Line Select Mode
elements by "drawing" a line through them. There should now be 10 frames and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

26. From the **Edit** menu select **Divide Frames...**

-
- **OK**

xz 2D View

Perspective Toggle

Set Intersecting Line Select Mode
diagonal elements by "drawing" a line through them. There should now be 20 frames selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

31. From the **Edit** menu select **Divide Frames...**

-
- **OK**

xz 2D View

Kip-in

Define **Materials...**

Modify/Show Material

-

- Verify that Poisson's Ratio is 0.3.

- Verify that the steel stress, Fy is 36.

- Click the **OK** button.

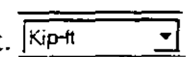
38. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

39. In this dialog box:

    - Verify that the Modulus of Elasticity is 3600.

    - Verify that Poisson's Ratio is 0.2.

    - Click the **OK** button twice to exit all dialog boxes.

40. Click the drop down box in the status bar to change the units to kip-ft. 

41. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

42. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

43. In this dialog box:

    - Verify that the Weight per Unit Volume is 0.15.

    - Click the **OK** button twice to exit all dialog boxes.

44. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

45. In this dialog box:

    - In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.

    - If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.

    - A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

➢ Scroll down and click on the W6X12 section.

➢ Click the **OK** button three times to exit all dialog boxes.

46. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

47. In the Click To area, click the **Modify/Show Section** button to display the Shell Sections dialog box.

48. In this dialog box:

-
-
-
- **OK**

Select All

Assign **Frame** **Sections...**

-
- **OK**

Show Undeformed Shape

Define **Static Load Cases...**

- **DL**
- **Change Load**
- **LL**
-
- **0**

- **Add New Load**
- **OK**

Select All

Assign **Shell Static Loads...** **Uniform...**

-
- **-.25**
-
- **OK**

Show Undeformed Shape

Click the "X" in the upper right-hand corner of the window labeled X-Z Plane @ Y=0 to close it.

60. Click the **Run Analysis** button to run the analysis.
61. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.
62. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
63. In this dialog box.
    - Click on the Steel Tab
    - Select AISC-ASD89 from the Steel Design Code drop-down box if it is not already selected.
    - Click the **OK** button.
64. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.
65. When the design check completes, the stress ratios are displayed.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "Y"

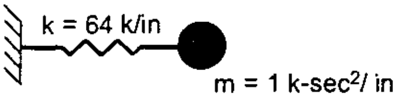
EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem Y

Response Spectrum Analysis For Single Degree of Freedom System

System



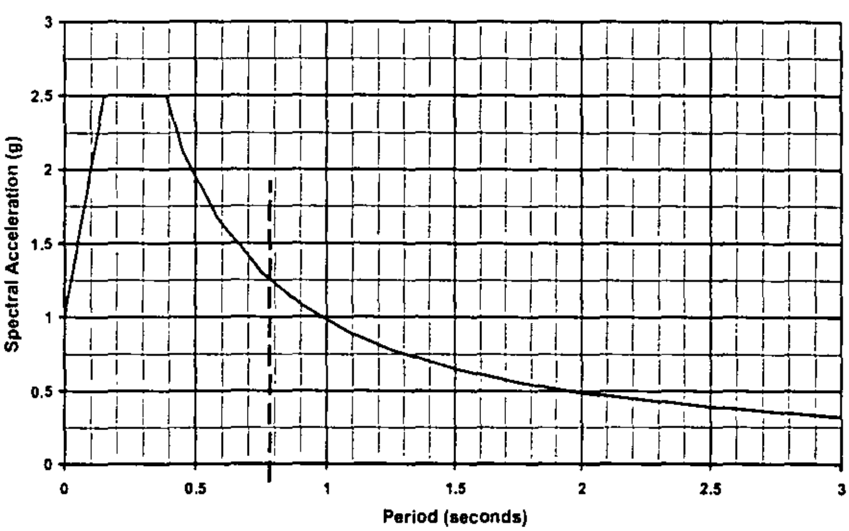
To Do

Perform a response spectrum analysis of this single degree of freedom system using the built-in 1994 UBC S1 spectrum. Compare the period with the calculated period below. Compare the spring force with the response spectrum below.

$$T = 2\pi\sqrt{\frac{m}{k}} = 2\pi\sqrt{\frac{1}{64}} = 0.7854 \text{ seconds}$$

Note: $1.25\ g * \dfrac{386.4\ in/sec^2}{1\ g} = 483\ in/sec^2$

1994 UBC S1 Spectrum



Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem Y Solution

1. Click the drop down box in the status bar to change the units to kip-in.

2. From the **File** menu select **New Model...**. This displays the Coordinate System Definition dialog box.

3. In this dialog box

   - Select the Cartesian Tab.
   - In the Number of Grid Spaces area type **0** in the X direction edit box.
   - In the Number of Grid Spaces area type **0** in the Y direction edit box.
   - In the Number of Grid Spaces area type **0** in the Z direction edit box.
   - Click the **OK** button.

4. Click the "X" in the upper right-hand corner of the window labeled 3-D View to close it.

5. Click the **Draw Special Joint** button on the side toolbar or select **Add Special Joint** from the **Draw** menu.

6. Click on the grid intersection at the origin to enter a joint.

7. Click the **Pointer** button to exit Draw Mode and enter Select Mode.

8. Select the joint by clicking on it.

9. From the **Assign** menu select **Joint** and then **Springs...** from the submenu to display the Joint Springs dialog box.

10. In this dialog box:

    - Type **64** in the Translation 1 edit box.
    - Click the **OK** button.

11. Select the joint by clicking on it.

12. From the **Assign** menu select **Joint** and then **Masses...** from the submenu to display the Joint Masses dialog box.

13. In this dialog box:

    - In the Masses in Local Directions area type **1** in the Direction 1 edit box.

18. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window. Note that the 3-D window now shows the first mode shape.

19. Note that the period is reported in the window title. It should be 0.7854 seconds.

20. From the Display menu select Show Element Forces/Stresses and then Joints... from the submenu to display the Joint Reaction Forces dialog box.

21. In this dialog box:

    - Select SPEC1 Spectra from the load drop-down box.
    - In the Type area select the Spring Forces option.
    - Click the **OK** button.

22. If the axes make it difficult to read the spring force, then from the **View** menu select **Show Axes** to toggle the axes display off.

- Click the **OK** button.

14. From the **Define** menu select **Response Spectrum Cases...** to display the Define Response Spectra dialog box.

15. In this dialog box:

    - Click the Add New Spectra button to display the Response Spectrum Case Data dialog box.

    - In this dialog box:

        - Accept the default Spectrum case name, SPEC1.

        - Accept the default Excitation Angle, 0.

        - Accept the default Modal Combination option, CQC.

        - Type **.05** in the Damping edit box.

        - Accept the default Directional Combination option, SRSS. Note that this option is irrelevant in this example since the response spectrum is run in only one direction.

        - In the Input Response Spectra area select UBC94S1 from the U1 Function drop-down box.

        - In the Input Response Spectra area type 386.4 in the U1 Scale factor edit box.

        - Click the **OK** button twice to exit all dialog boxes.

16. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - Uncheck all of the Available DOFs check boxes except for UX.

    - Verify the the Dynamic Analysis check box is checked.

    - Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

    - In this dialog box:

        - Verify that the Number of Modes is 1.

        - Verify that the Eigenvectors option is selected in the Type of Analysis area.

        - Click the **OK** button twice to exit all dialog boxes.

17. Click the **Run Analysis** button ▸ to run the analysis.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 THREE DIMENSIONAL STATIC AND DYNAMIC FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

PROBLEM "Z"

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

Problem Z

Response Spectrum Analysis

Building Description
The building is a four-story concrete shear wall building with concrete flat slabs supported by concrete columns. There is a 30 foot high steel flagpole on the roof at one corner of the building. A 250 pound man sits on top of the flagpole.

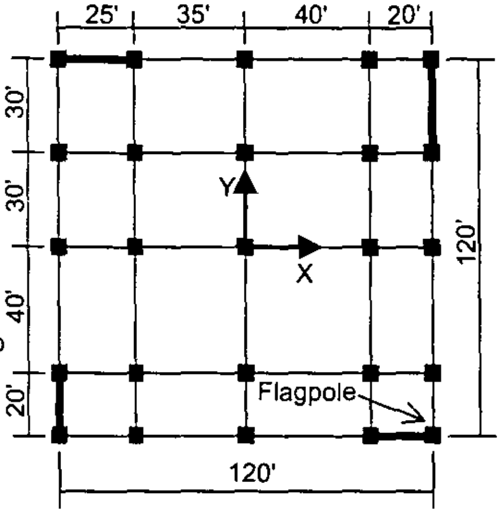
Steel
E =29000 ksi
Poissons Ratio = 0.3
Flagpole is 3" ∅ standard pipe

Concrete
E =3600 ksi
Poissons Ratio = 0.2
Walls are 12" thick
Columns & beams are 20" x 20"
Floors & roof are 10" thick flat slab

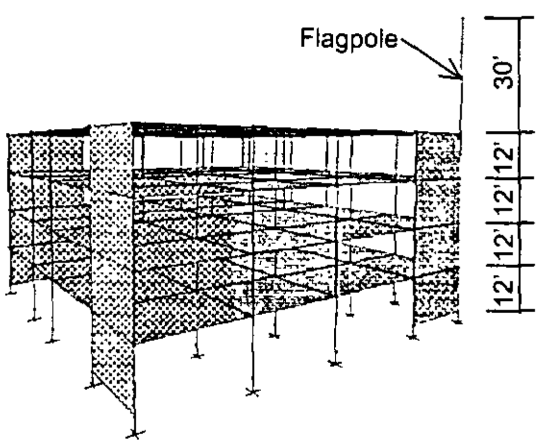
Response Spectrum Loading
X-Dir (U1): 1994 UBC S2
Y-Dir (U2): 30% of 1994 UBC S2

Assumptions
- Diaphragms are rigid in plane.
- Columns are fixed base.
- Consider the mass of the 250 pound man which is 0.00065 kip-sec2/in.

To Do
Determine maximum X-direction (U1) displacement at top and bottom of the flagpole for the specified response spectrum loading.



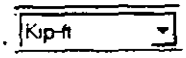
Typical Floor and Roof Plan



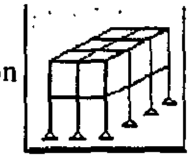
Three Dimensional Perspective View

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem Z Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Space Frame** template button  to display the Space Frame dialog box.

4. In this dialog box:

   - Type **4** in the Number of Stories edit box.
   - Type **4** in the Number of Bays Along X edit box.
   - Type **4** in the Number of Bays Along Y edit box.
   - Accept the default Story Height, 12.
   - Type **30** in the Bay Width Along X edit box.
   - Type **30** in the Bay Width Along Y edit box.
   - Verify that the Restraints and Gridlines check boxes are checked.
   - Click the **OK** button.

5. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

6. In this dialog box:

   - Check the Glue Joints To Grid Lines check box.
   - Verify that the X option is selected in the Direction area.
   - Click on the -30 grid line in the X Location list box to highlight it. The -30 value appears in the X Location edit box.
   - Type **-35** in the X Location edit box and click the **Move Grid Line** button.
   - Click on the 30 grid line in the X Location list box to highlight it. The 30 value appears in the X Location edit box.
   - Type **40** in the X Location edit box and click the **Move Grid Line** button.

- Select the Y option in the Direction area.
- Click on the -30 grid line in the Y Location list box to highlight it. The -30 value appears in the Y Location edit box.
- Type **-40** in the Y Location edit box and click the **Move Grid Line** button.
- Click the **OK** button.

7. Verify that the 3-D View window is active. The window is active when its title is highlighted.
8. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
9. In this dialog box:
   - Check the Fill Elements check box.
   - Click the **OK** button.
10. From the **View** menu select **Refresh View** to update the 3-D view.
11. Click in the window labeled X-Y Plane @ Z=48 to activate it.
12. Click the **Set Elements** button ⊠ on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
13. In this dialog box:
   - Check the Fill Elements check box.
   - Click the **OK** button.
14. Click the **xz 2D View** button xz on the main toolbar. The view switches to the X-Z plane @ Z=60 and appears as shown in Figure Z-1.
15. Click the **Quick Draw Rectangular Shell Element** button ⊡ on the side toolbar.
16. Click once in each of the area labeled "A", "B", "C" and "D" in Figure Z-1 to enter four shell elements.
17. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.
18. In this dialog box:
   - Verify that the X-Z Plane option is selected.
   - Type **-60** in the Y= edit box.

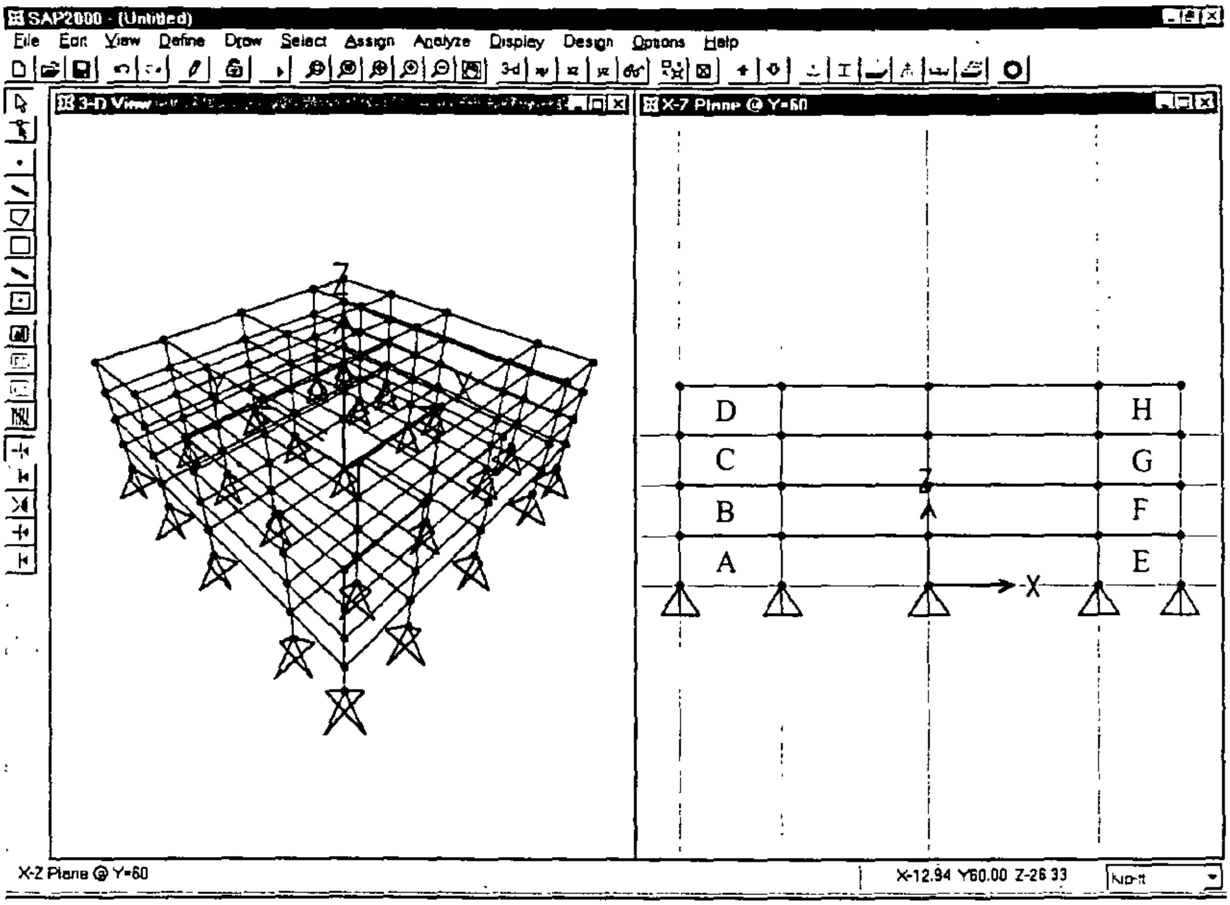


Figure Z-1: Screen After Step 14

- Click the **OK** button. The screen appears similar to that shown in Figure Z-1 (except that the location is now at Y=-60).

19. Click once in each of the area labeled "E", "F", "G" and "H" in Figure Z-1 to enter four shell elements.

20. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

21. In this dialog box:

    - Select the Y-Z Plane option.

    - Verify that 60 is entered in the X= edit box.

    - Click the **OK** button. The screen appears as shown in Figure Z-2.

22. Click once in each of the area labeled "A", "B", "C" and "D" in Figure Z-2 to enter four shell elements.

23. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

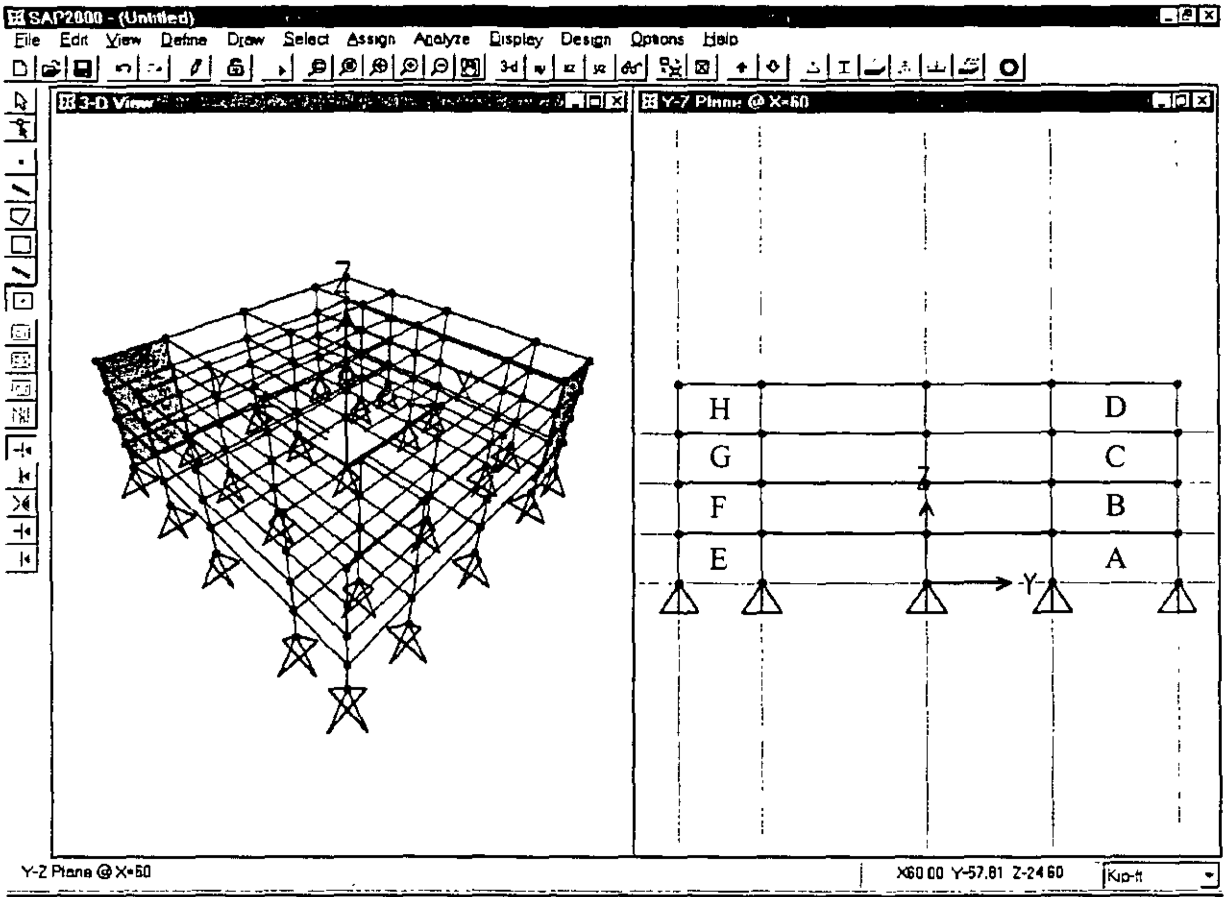


Figure Z-2: Screen After Step 21

24. In this dialog box:

    - Verify that the Y-Z Plane option is selected.

    - Type **-60** in the X= edit box.

    - Click the **OK** button. The screen appears similar to that shown in Figure Z-2 (except that the location is now at X=-60).

25. Click once in each of the area labeled "E", "F", "G" and "H" in Figure Z-2 to enter four shell elements. This completes the drawing of the shear walls.

26. Click the **Pointer** button to exit draw mode and enter select mode.

27. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

28. In this dialog box:

    - Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.

- In this dialog box:
  - ➢ Verify that the Mass per Unit Volume is 4.658E-03.
  - ➢ Verify that the Weight per Unit Volume is 0.15.
  - ➢ Click the **OK** button to return to the Define Materials dialog box.
- Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
- In this dialog box:
  - ➢ Verify that the Mass per Unit Volume is 0.0152.
  - ➢ Verify that the Weight per Unit Volume is 0.489.
  - ➢ Click the **OK** button twice to exit all dialog boxes.

29. Click the drop down box in the status bar to change the units to kip-in. 

30. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

31. In this dialog box:
    - Highlight the CONC material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
    - In this dialog box:
      - ➢ Verify that the Modulus of Elasticity is 3600.
      - ➢ Verify that Poisson's ratio is 0.2.
      - ➢ Click the **OK** button to return to the Define Materials dialog box.
    - Highlight the STEEL material and click the **Modify/Show Material** button to display the Material Property Data dialog box.
    - In this dialog box:
      - ➢ Verify that the Modulus of Elasticity is 29000.
      - ➢ Verify that Poisson's ratio is 0.3.
      - ➢ Click the **OK** button twice to exit all dialog boxes.

32. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

33. In this dialog box:

- In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import Pipe item.

- If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files.

- A dialog box appears with a list of all pipe sections in the database. In this dialog box:

  - ➢ Scroll down and click on the P3 (3" diameter standard pipe section) item.

  - ➢ Click the OK button twice to return to the Define Frame Sections dialog box.

- In the Click To area, click the drop-down box that says Add I/Wide Flange and then click on the Add Rectangular item. The Rectangular Section dialog box is displayed.

- In this dialog box:

  - ➢ Type **BMCOL** in the Section Name edit box.

  - ➢ Select CONC from the Material drop-down box.

  - ➢ Type **20** in the Depth (t3) edit box.

  - ➢ Type **20** in the Width (t2) edit box.

  - ➢ Click the OK button twice to exit all dialog boxes.

34. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

35. In this dialog box:

- Click the **Modify/Show Section** button to display the Shell Sections dialog box.

- In this dialog box:

  - ➢ Type **WALL** in the Section Name edit box.

  - ➢ Accept the default CONC material

  - ➢ In the Thickness area verify that both the Membrane and the Bending thicknesses are 12.

  - ➢ In the Type area verify that the Shell option is selected.

  - ➢ Click the **OK** button to return to the Define Shell Sections dialog box.

- Click the **Add New Section** button to display the Shell Sections dialog box.
- In this dialog box:
  - Type **FLOOR** in the Section Name edit box.
  - Accept the default CONC material.
  - Type **10** in the Membrane edit box.
  - Type **10** in the Bending edit box.
  - In the Type area verify that the Shell option is selected.
  - Click the **OK** button twice to exit all dialog boxes.

36. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft
37. Click in the window labeled Y-Z Plane @ X=-60 to verify it is active.
38. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.
39. In this dialog box:
    - Select the X-Y Plane option.
    - Verify that 48 is entered in the Z= edit box.
    - Click the **OK** button.
40. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.
41. In this dialog box:
    - Check the Labels box in the Joints area.
    - Click the **OK** button.
42. Click the **Draw Rectangular Shell Element** button on the side toolbar or select **Draw Rectangular Shell Element** from the **Draw** menu.
43. Click on joint 25 and then joint 105 to enter a single shell element for the entire floor.
44. Click the **Pointer** button to exit draw mode and enter select mode.
45. Select all elements in the X-Y Plane @ Z=48 by "windowing".
46. From the **Edit** menu select **Mesh Shells...** to display the Mesh Selected Shells dialog box.

47. In this dialog box:

- Select the mesh Using Selected Joints on Edges option.

  Note: The Mesh At Intersections With Grids option would work equally well.

- Click the **OK** button.

48. Select all elements in the X-Y Plane @ Z=48 by "windowing".

49. From the **Assign** menu select **Shell** and then **Sections...** from the submenu to display the Define Shell Sections dialog box.

50. In this dialog box:

- Highlight the FLOOR section by clicking on it.

- Click the **OK** button.

51. Click the **Show Undeformed Shape** button to remove the displayed shell section assignments.

52. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

53. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

54. In this dialog box:

- Verify the Linear Tab is selected.

- In the Distance area type **-12** in the Z edit box.

- Verify that 0 is entered in the X and Y edit boxes.

- Type **3** in the Number edit box.

- Click the **OK** button to proceed with the replication and create the other floor diaphragms.

55. Click the **Restore Previous Selection** button on the side toolbar (or select **Get Previous Selection** from the **Select** menu).

56. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

57. In this dialog box:

- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

- In this dialog box:

  - Type **ROOFDIA** in the Constraint Name edit box.

  - Select the Z Axis option in the Constraint Axis area.

  - Click the **OK** button twice to exit all dialog boxes.

58. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=36. You can confirm the elevation by looking on the right-hand side of the staus bar at the bottom of the SAP2000 window.

59. Select all elements at this level by "windowing".

60. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

61. In this dialog box:

    - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

    - In this dialog box:

      - Type **4THDIA** in the Constraint Name edit box.

      - Select the Z Axis option in the Constraint Axis area.

      - Click the **OK** button twice to exit all dialog boxes.

62. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=24.

63. Select all elements at this level by "windowing".

64. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

65. In this dialog box:

    - In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.

    - In this dialog box:

      - Type **3RDDIA** in the Constraint Name edit box.

- Select the Z Axis option in the Constraint Axis area.
- Click the **OK** button twice to exit all dialog boxes.

66. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=12.

67. Select all elements at this level by "windowing".

68. From the **Assign** menu select **Joint** and then **Constraints...** from the submenu to display the Constraints dialog box.

69. In this dialog box:

- In the Click To area click the drop-down box and select Add Diaphragm to display the Diaphragm Constraint dialog box.
- In this dialog box:
  - Type **2NDDIA** in the Constraint Name edit box.
  - Select the Z Axis option in the Constraint Axis area.
  - Click the **OK** button twice to exit all dialog boxes.

70. Click the **Down One Gridline** button to move the plan display down to the X-Y Plane @ Z=0.

71. Select all elements at this level by "windowing".

72. From the **Assign** menu select **Joint** and then **Restraints...** from the submenu to display the Joint Restraints dialog box.

73. In this dialog box:

- Click the **Fixed Support** button in the Fast Restraints area.
- Click the **OK** button.

74. Click the **Select All** button on the side toolbar to select all elements.

75. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

76. In this dialog box:

- Click on BMCOL in the Frame Sections area to highlight it.
- Click the **OK** button.

77. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

78. From the **View** menu select **Set 2D View...** to display the Set 2D View dialog box.

79. In this dialog box:

    - Select the X-Z Plane option.
    - Verify -60 is entered in the Y= edit box.
    - Click the **OK** button.

80. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

81. In this dialog box:

    - Select the Z option in the Direction area.
    - Type **78** in the Z Location edit box and click the **Add Grid Line** button.
    - Click the **OK** button. The screen appears as shown in Figure Z-3.

82. Click the **Quick Draw Frame Element** button on the side toolbar or select **Quick Draw Frame Element** from the **Draw** menu.

83. Click on the grid line at the point labeled "A" in Figure Z-3 to enter the flagpole frame element.

84. Click the **Pointer** button to exit draw mode and enter select mode.

85. Click on the frame element to select it.

86. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

87. In this dialog box:

    - Click on P3 in the Frame Sections area to highlight it.
    - Click the **OK** button.

88. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

89. Click the drop down box in the status bar to change the units to kip-in. Kip-in

90. Click on the joint at the top of the flagpole to select it.

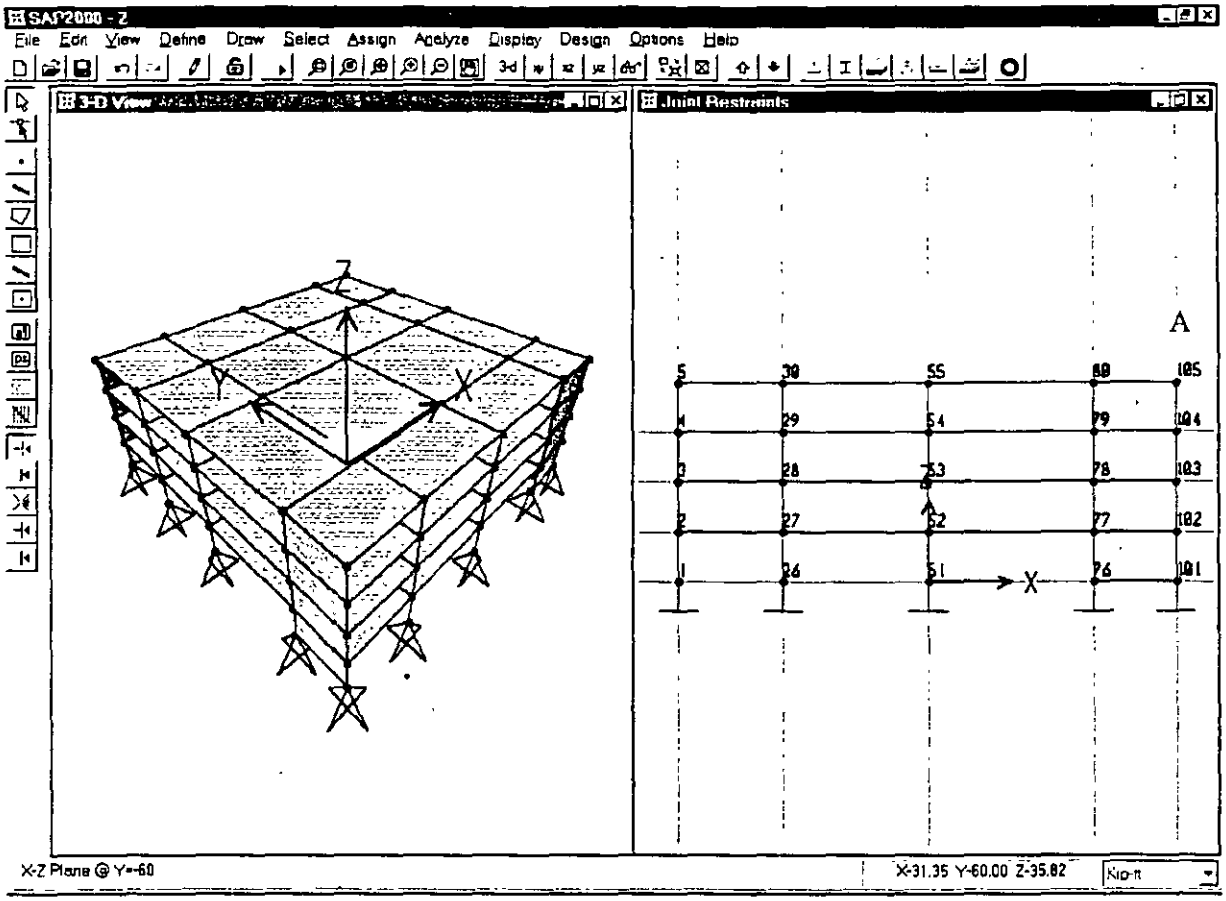


Figure Z-3: Screen After Step 81

91. From the **Assign** menu select **Joint** and then **Masses...** from the submenu to display the Joint Masses dialog box.

92. In this dialog box:

    - Type **.00065** in the Direction 1, Direction 2 and Direction 3 edit boxes.

    - Click the **OK** button.

93. Click the **Show Undeformed Shape** button to remove the displayed joint mass assignments.

94. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

95. Click the **Set Elements** button on the main toolbar (or select **Set Elements...** from the **View** menu) to display the Set Elements Dialog box.

96. In this dialog box:

    - Uncheck the Labels box in the Joints area.

- Click the **OK** button.

97. From the **Define** menu select **Response Spectrum Cases...** to display the Define Response Spectra dialog box.

98. In this dialog box:

    - Click the Add New Spectra button to display the Response Spectrum Case Data dialog box.

    - In this dialog box:

        ➢ Accept the default Spectrum case name, SPEC1.

        ➢ Accept the default Excitation Angle, 0.

        ➢ Accept the default Modal Combination option, CQC.

        ➢ Type **.05** in the Damping edit box.

        ➢ Accept the default Directional Combination option, SRSS.

        ➢ In the Input Response Spectra area select UBC94S2 from the U1 Function drop-down box.

        ➢ In the Input Response Spectra area type **32.2** in the U1 Scale factor edit box.

        ➢ In the Input Response Spectra area select UBC94S2 from the U2 Function drop-down box.

        ➢ In the Input Response Spectra area type **9.66** (0.3 * 32.2 = 9.66) in the U2 Scale factor edit box.

        ➢ Click the **OK** button twice to exit all dialog boxes.

99. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.

    - Verify the the Dynamic Analysis check box is checked.

    - Click the **Set Dynamic Parameters** button to display the Dynamic Analysis Parameters dialog box.

    - In this dialog box:

        ➢ Type **20** in the Number of Modes edit box.

        ➢ In the Type of Analysis area select Ritz Vectors.

        ➢ Click on ACCEL Z in the Ritz Load Vectors list box to highlight it.

- Click the **Remove** button to remove ACCEL Z from the Ritz Load Vectors list box. ACCEL X and ACCEL Y should remain in the Ritz Load Vectors list box.

- Click the **OK** button twice to exit all dialog boxes.

100. Click the **Run Analysis** button to run the analysis.

101. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors). Click the **OK** button to close the Analysis window.

102. Click in the window labeled X-Z Plane @ Y=-60 to make sure it is active.

103. Click the **Display Static Deformed Shape** button (or select **Show Deformed Shape...** from the **Display** menu). The Deformed Shape dialog box appears.

104. In this dialog box:

    - Select SPEC1 Spectra from the Load drop-down box.

    - Click the **OK** button.

105. Right click the joints at the top and bottom of the flagpole to see their displacements.

    *Note: If the top of the flagpole goes off of the screen when the deformed shape is displayed, then click the **Zoom Out One Step** button on the main toolbar as many times as required to bring the joint back on to the screen.*

CURSOS ABIERTOS

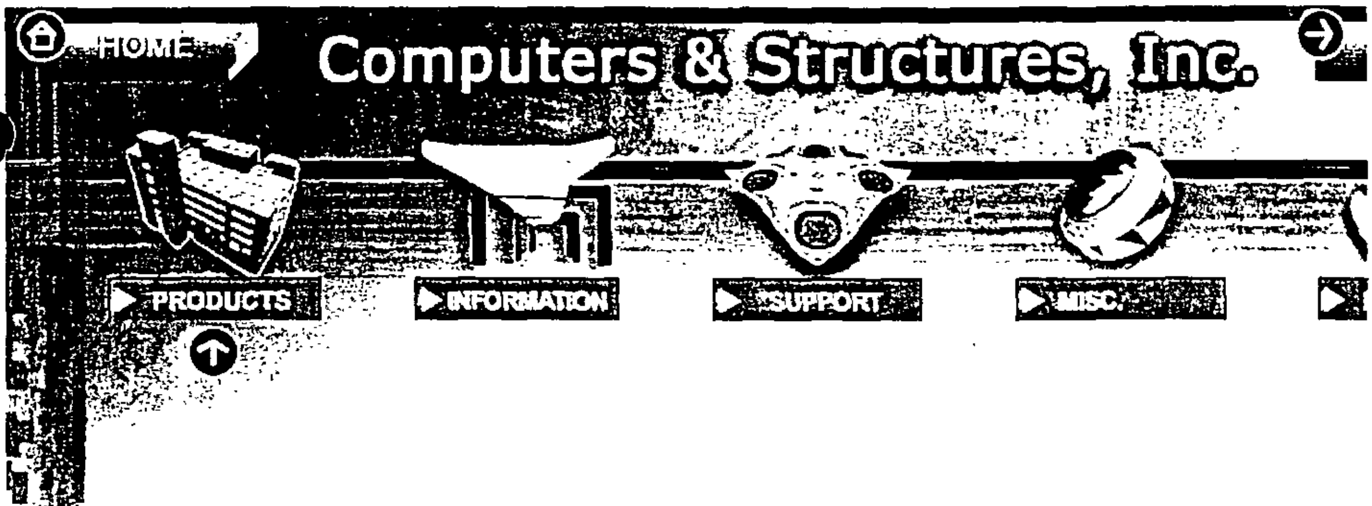
SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 1
HOW SAFE CALCULATES PUNCHING SHEAR FOR THE ACI 318-95 CODE

1. TERMINOLOGY FOR SAFE METTHOD OF CALCULATING PUCHING SHEAR

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001



Products - SAP2000

SAP2000

SAP2000 represents the state-of-the-art in three dimensional finite element technology for structural engineering. This SAP is completely integrated within Windows 95/98/NT/2000, and offers an easy-to-use graphical interface along with the latest developments in analytical techniques. **Click here for a graphical animation and user interface for SAP2000.**

Key Benefits

- Easy to use windows interface
- Powerful model generation capabilities
- US and International design codes

Modelling Options

SAP2000 provides powerful capabilities for modeling a wide range of structures, including bridges, dams, tanks and buildings. The Windows based graphical interface allows for quick model creation using templates The creation and modification of models, execution of the analysis, viewing of the results, and **design optimization** are all performed interactively within the same interface

Analysis Options

Element types range from **Frame/Truss** to **Shell/Plate** to Solid to **Nonlinear Link** elements. The Frame element can be non-prismatic with rigid end offsets, and supports a wide range of load types, including **prestress**. The Shell element is three dimensional and includes in-plane rotational stiffness components, and can be used either as a 3 or 4 node element. The Solid element is a three dimensional, 8 node brick with anisotropic material properties The Nonlinear Link element may be used for dynamic analysis involving base isolators, dampers, and hook and gap elements.

Loading Options

Static loading options allow for **gravity, pressure, thermal** and prestress conditions in addition to nodal loading with specified forces or displacements Dynamic loading can be in the form of **multiple base response spectrums**, or **multiple time varying loads and base excitations**. The programs support both **Eigen** and **Ritz** analysis, as well as modal combinations by the **SRSS**, the **CQC** or the **GMC** method. Vehicle live load generation for **truck, lane and train loading** is also available.

Design Options

Integrated design capabilities include steel member (**AISC-ASD** and **LFRD**) selection and optimization, as well as reinforcing calculations for concrete members (**ACI-89**).

Nonlinear Pushover

New computational techniques have been integrated into SAP2000 Nonlinear Version 7 to allow a Static Pushover analysis to be performed in a simple and practical manner. Nonlinear hinges may be defined anywhere in the Frame elements, and properties may be user defined or calculated automatically by the program. Analyses may be force or displacement controlled, and results are available in both graphical and table form. The following are some of the highlights of SAP2000/NL-PUSH The Only 3 Dimensional Design Tool for Performance Based Design of Steel and Concrete Structures Nonlinear pushover analysis based on ATC-40 and FEMA –273 is fully integrated into the SAP2000 Windows Analysis and Design Graphical User Interface. Same model can also be used for Static, Dynamic and Nonlinear Time History analysis. Nonlinear Pushover Analysis results can be used with the integrated steel and concrete design postprocessor options. Default and user defined Moment, Shear, Axial and PMM hinges at any location along a frame member. Pushover Output is displayed and available Step by Step in both graphical and text formats. Highly interactive capacity spectrum curve display form allows study of the effects of parameter changes instantaneously.

Three Available Versions Windows 95/98/NT/2000

• SAP2000 Standard - Program offers Static and Dynamic Response Spectrum Analysis for Frame and Shell Elements, along with Steel and Concrete Frame Design.
Capacity: 1500 Nodes

• SAP2000 PLUS - Offers all of the same features as the Standard version Plus Dynamic Time History Analysis, Plane, Solid and Asolid Elements, and Bridge Analysis
Capacity: No practical limit

• SAP2000 Nonlinear - Expands the PLUS options with Dynamic Nonlinear Time History Analysis, an external Damping Element, Base Isolators, and Gap and Hook Elements
Capacity: No practical limit

PDF Brochure
Download the full SAP2000 brochure. Adobe Acrobat is required

SAFE Technical Note 1

How SAFE Calculates Punching Shear
For the
ACI 318-95 Code

(CSA A23.3-94, NZS 3101-95 and IS 456-1978 R1996 Codes Similar)

The information in this document is consistent with SAFE version 6.14 and later

ttp: // www.csiberkeley.com/

Initial Release Date: November 16, 1998
Revision Number: 4
Revision Date: June 16, 2000

Table Of Contents

Item	Page
1. Terminology For SAFE Method of Calculating Punching Shear	1
2. Basic Equations For SAFE Method of Calculating Punching Shear	1
3. Limitations of Punching Shear Calculations in SAFE	3
4. Other Comments	4
5. References	4
6. Numerical Example	5
a. Problem Statement	5
b. SAFE Computer Model	6
c. Hand Calculation For Interior Column Using SAFE Method	8
d. Hand Calculation For Edge Column With Edge Parallel To X-Axis Using SAFE Method	11
e. Hand Calculation For Edge Column With Edge Parallel To Y-Axis Using SAFE Method	14
f. Hand Calculation For Corner Column Using SAFE Method	17
g. Hand Calculation For Corner Column Using SAFE Method Except That I_{XY} Is Set To Zero	20
h. Terminology For PCA Publication Method of Calculating Punching Shear	21
i. Basic Equation For PCA Publication Method	21
j. Hand Calculation For Interior Column Using PCA Publication Method	22
k. Hand Calculation For Edge Column With Edge Parallel To X-Axis Using PCA Publication Method	24
l. Hand Calculation For Edge Column With Edge Parallel To Y-Axis Using PCA Publication Method	26
m. Hand Calculation For Corner Column Using PCA Publication Method	29
n. Comparison Of Punching Shear Stress Results	31

1. Terminology For SAFE Method of Calculating Punching Shear

The following terms are used in describing the SAFE method of calculating punching shear:

b_0	=	perimeter of critical section for punching shear
d	=	effective depth at critical section for punching shear based on average of d for X direction and d for Y direction
I_{XX}	=	Moment of inertia of critical section for punching shear about an axis that is parallel to the global X-axis
I_{YY}	=	Moment of inertia of critical section for punching shear about an axis that is parallel to the global Y-axis
I_{XY}	=	Product of inertia of critical section for punching shear with respect to the X and Y planes
L	=	Length of side of critical section for punching shear currently being considered
M_{UX}	=	Moment about line parallel to X-axis at center of column (positive per right-hand rule)
M_{UY}	=	Moment about line parallel to Y-axis at center of column (positive per right-hand rule)
v_U	=	Punching shear stress
V_U	=	Shear at center of column (positive upward)
x_1, y_1	=	Coordinates of column centroid
x_2, y_2	=	Coordinates of center of one side of critical section for punching shear
x_3, y_3	=	Coordinates of centroid of critical section for punching shear
x_4, y_4	=	Coordinates of location where you are calculating stress
γ_{VX}	=	Percent of M_{UX} resisted by shear per ACI 318-95 equations 11-41 and 13-1
γ_{VY}	=	Percent of M_{UY} resisted by shear per ACI 318-95 equations 11-41 and 13-1

2. Basic Equations For SAFE Method of Calculating Punching Shear

$$v_U = \frac{V_U}{b_0 d} + \frac{\gamma_{VX}[M_{UX} - V_U(y_3 - y_1)][I_{YY}(y_4 - y_3) - I_{XY}(x_4 - x_3)]}{I_{XX}I_{YY} - I_{XY}^2} - \frac{\gamma_{VY}[M_{UY} + V_U(x_3 - x_1)][I_{XX}(x_4 - x_3) - I_{XY}(y_4 - y_3)]}{I_{XX}I_{YY} - I_{XY}^2} \qquad \text{Eq. 1}$$

$I_{XX} = \sum_{sides=1}^{n} \bar{I}_{XX}$, where sides refers to the sides of the critical section for punching shear Eq. 2

$I_{YY} = \sum_{sides=1}^{n} \bar{I}_{YY}$, where sides refers to the sides of the critical section for punching shear Eq. 3

$I_{XY} = \sum_{sides=1}^{n} \bar{I}_{XY}$, where sides refers to the sides of the critical section for punching shear Eq. 4

The equations for $\bar{I}_{XX}$, $\bar{I}_{YY}$, and $\bar{I}_{XY}$ are different depending on whether the side of the critical section for punching shear being considered is parallel to the X-axis or parallel to the Y-axis. Refer to Figures 1a and 1b.

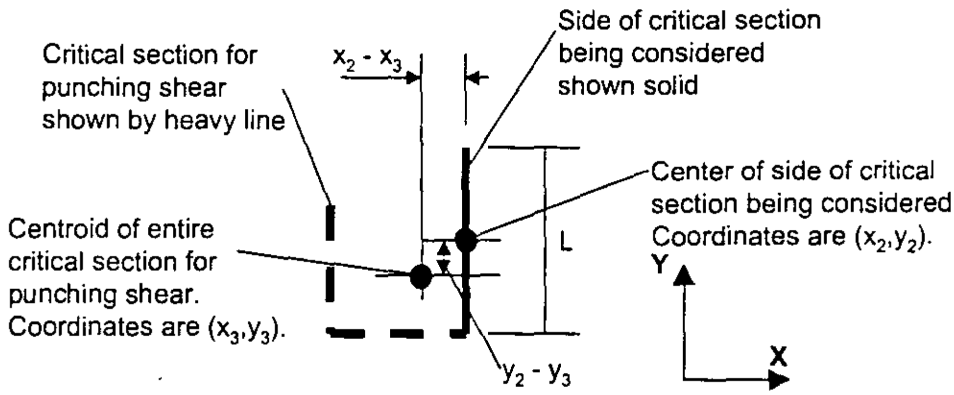


Figure 1b: Plan View For Side of Critical Section Parallel to Y-Axis

Work This Sketch With Equations 5b, 6b and 7

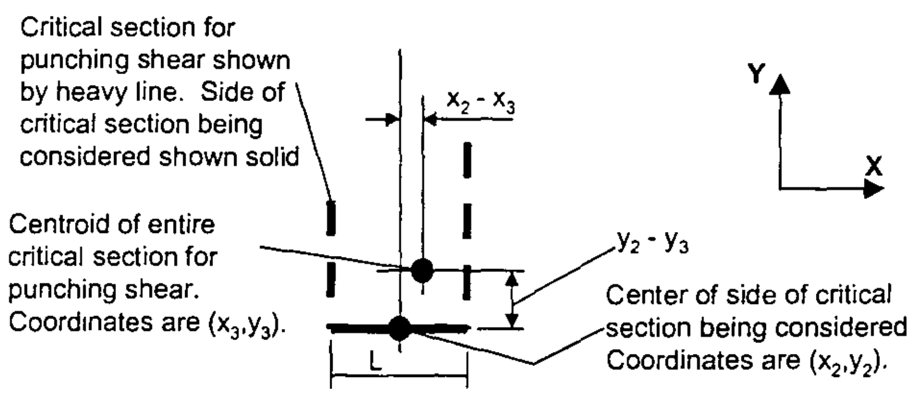


Figure 1a: Plan View For Side of Critical Section Parallel to X-Axis

Work This Sketch With Equations 5a, 6a and 7

$\bar{I}_{XX} = Ld(y_2 - y_3)^2$, for side of critical section parallel to X-axis — Eq. 5a

$\bar{I}_{XX} = \dfrac{Ld^3}{12} + \dfrac{dL^3}{12} + Ld(y_2 - y_3)^2$, for side of critical section parallel to Y-axis — Eq. 5b

$\bar{I}_{YY} = \dfrac{Ld^3}{12} + \dfrac{dL^3}{12} + Ld(x_2 - x_3)^2$, for side of critical section parallel to X-axis — Eq. 6a

$\bar{I}_{YY} = Ld(x_2 - x_3)^2$, for side of critical section parallel to Y-axis — Eq. 6b

$\bar{I}_{XY} = Ld(x_2 - x_3)(y_2 - y_3)$, for side of critical section parallel to X-axis or Y-axis — Eq. 7

3. Limitations of Punching Shear Calculations in SAFE

- The shear and moment values used in the punching shear check have not been reduced by the load (or reaction) that is included within the boundaries of the punching shear critical section. Typically the effect of this simplification is small except in some cases for deep slabs (e.g., mat foundations) and slabs with closely spaced columns.

- Punching shear is calculated for columns punching through a slab or a drop panel. It is not calculated for a drop panel punching through a slab. The effect of column capitals is included in the punching shear calculation.

- The program checks that each slab element in the area enclosed between the face of the column and the critical section for punching shear has identically the same slab property label. If so, the punching shear check is performed, if not, punching shear is not calculated and N/C is displayed.

- When line objects (beams, walls or releases) frame into a column, punching shear is not calculated and N/C is displayed.

- If a point load or column falls (call it load/column A) within the critical section for punching shear for another point load or column (call it load/column B), then it is ignored in the punching shear calculation (that is, the effect of load/column A is ignored when doing punching shear calculations for load/column B.)

- The program only considers critical sections for punching shear that have sides parallel to the X and Y axes. Thus when the edge of a slab is not parallel to the X or Y axis, the program may not pick up the worst case critical section for punching shear. For example, in Figure 2, the critical sections for punching shear identified in Cases A and B are considered by SAFE, but CASE C is not considered.

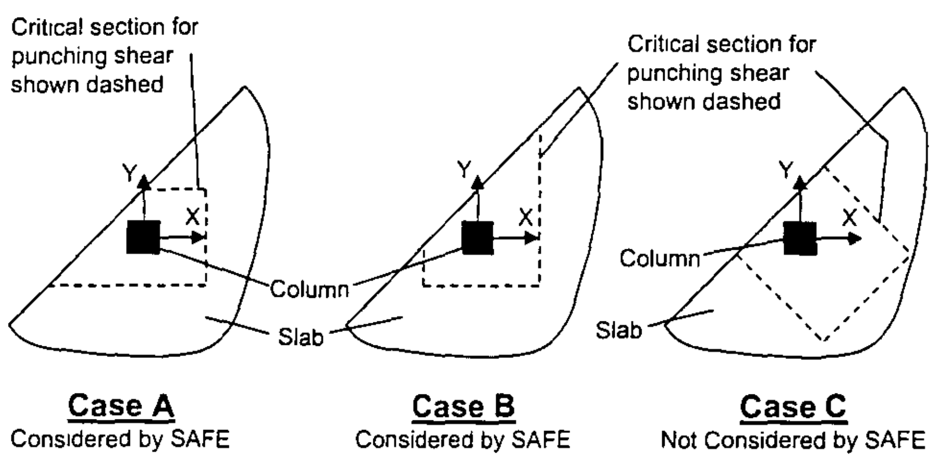


Figure 2: Critical Sections For Punching Shear Considered By SAFE

4. Other Comments

Equation 1 is based on Chapter 8, Section 50 of S. Timoshenko's book titled "Strength of Materials" (Ref. 1) and on the requirements of codes such as ACI 318-95.

For interior columns and edge columns the punching shear formulation used in SAFE yields the same results as those that are obtained using the equations given in Figure 18-16 of the PCA Publication (Ref 2). The results are different for corner columns. The reason for the difference is the I_{XY} term that is included in the SAFE equations but not in the PCA equations. For interior columns and edge columns the I_{XY} term included in the SAFE formulation reduces to zero. The I_{XY} term is nonzero for corner columns because the X and Y axes are not the principal axes for this section. If the I_{XY} term in the SAFE equations is set to zero, then the SAFE equations yield the same results for corner columns as the PCA equations. (Note that there is no way to actually set the I_{XY} term to zero in SAFE.) It could be argued that the SAFE formulation is perhaps more theoretically sound than the equations given in the PCA Publication. The net effect of the I_{XY} term is to make the SAFE punching shear calculation for corner columns a little more conservative than that given in the PCA Publication.

5. References

1. S. Timoshenko, 1958
   Strength of Materials, Part I, 3rd Edition, D. Van Nostrand Company, New York, New York.

2. PCA, 1996
   Notes On ACI 318-95 Building Code Requirements For Structural Concrete With Design Applications, Portland Cement Association, Skokie, Illinois.

6. Numerical Example

6a. Problem Statement

The numerical example is a flat slab with drop panels that has three 24' spans in each direction, as shown in Figure 3. The slab overhangs beyond the face of the column by 6" along each side of the structure. The columns are typically 12" x 36" with the long side parallel to the Y-axis. The slab is typically 10" thick. At the drop panels there is an additional 5" of slab thickness, thus the total slab thickness at the drop panels is 15". The plan dimensions of the interior drop panels are 8' x 8', the edge drop panels where the edge is parallel to the X-axis are 8' x 6', the edge drop panels where the edge is parallel to the Y-axis are 5' x 8', and the corner drop panels are 5' x 6'.

The concrete has a unit weight of 150 pcf and a f'c of 4000 psi. The dead load consists of the self weight of the structure plus an additional 20 psf. The live load is 80 psf.

Thick plate properties are used for the slab.

Each column in Figure 3 is referenced with a number. For example, the column at the intersection of grid lines A and 1 is number 5. These numbers refer to the point object ID's in the associated SAFE model

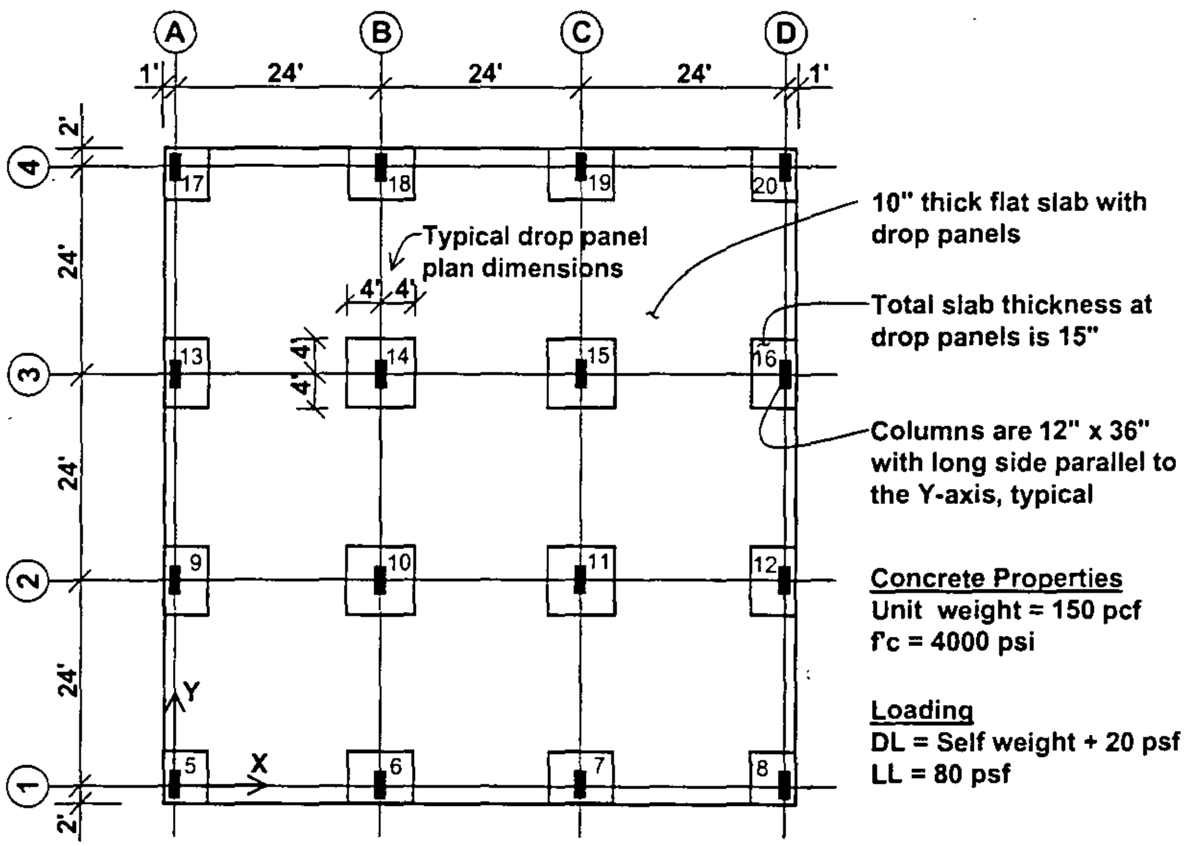


Figure 3: Flat Slab With Drop Panels For Numerical Example

6b. SAFE Computer Model

In SAFE it is easy to create a computer model for this example, analyze it, design it, and print out the punching shear results. The following steps are required:

1. Set the units to kips and feet.

2. From the File menu select New Model From Template, and then click on the Flat Slab button to display the Flat Slab dialog box.

3. In this dialog box:

    - Change the Top Edge Distance and Bottom Edge Distance to 2.

    - Check the Create Live Load Patterns check box.

    - Accept the rest of the default values and click the OK button.

4. From the Define menu select Slab Properties to display the Support Properties dialog box. Highlight the property named COL, click the Modify\Show Property button to display the Slab Property Data dialog box, check the Thick Plate check box and click OK. Highlight the property named DROP, click the Modify\Show Property button to display the Slab Property Data dialog box, check the Thick Plate check box and click OK. Highlight the property named SLAB, click the Modify\Show Property button to display the Slab Property Data dialog box, check the Thick Plate check box and click OK twice to exit all dialog boxes.

5. From the Define menu select Column Supports to display the Support Properties dialog box. With the support property named COLUMN highlighted click the Modify\Show Property button to display the Column Support Property Data dialog box. In this dialog box change the Y Dimension to 3. Accept the rest of the default values and click the OK button twice to exit all dialog boxes.

6. The SAFE model is now created and ready to run. From the Analyze menu select Run Analysis. Select a directory and provide a name for the input file and then click the Save button to proceed with the analysis.

7. When the analysis is completed review the messages in the Analysis window and click the OK button. The model is now ready for design.

8. From the Options menu select Preferences and click the Concrete tab. Verify that the Concrete Design Code selected is ACI 318-95. If necessary, change the selection to this code. Click the OK button to exit this dialog box.

9. From the Design menu select Start Design.

10. When the design is complete click Display Punching Shear Ratios on the display menu to show the punching shear results graphically.

11. Change the units to kips and inches.

12. From the File menu select Print Design Tables to display the Design Tables dialog box.

13 In this dialog box:

- In the Design Output area uncheck the Slab Strip Reinforcing check box and check the Punching Shear check box.

- In the Design Forces area uncheck the Slab Strip check box.

- Check the Print To File check box. Note that the default name for this file is the name of your model with a .txt extension. Accept this default name.

- Click the OK button to create the punching shear output.

The punching shear output is now in a text file whose name is the same as your input file with the extension .txt. You can open this text file in any text editor or word processor, or you can use the Display Input/Output Text Files feature on the File menu in SAFE to open it. This feature opens the file in Wordpad, a text editor that is supplied with Windows. If you open this text file in a word processor or a spreadsheet it should look similar to that shown in Figure 4.

SAFE v6.14 File: PUNCHEX Kip-in Units PAGE 1
November 14,1998 7.59

P U N C H I N G S H E A R S T R E S S C H E C K

POINT	X	Y	RATIO	COMBO	VMAX	VCAP	V	MX	MY	DEPTH	PERIM	LOC
5	0.00	0.00	1.000	DCON2	0.179	0.179	-54.696	-1962	1145.680	13.500	73.500	C
6	288.00	0.00	0.802	DCON2	0.144	0.179	-119.738	-3778	-348.558	13.500	123.000	E
7	576.00	0.00	0.802	DCON2	0.144	0.179	-119.738	-3778	348.558	13.500	123.000	E
8	864.00	0.00	1.000	DCON2	0.179	0.179	-54.696	-1962	-1146	13.500	73.500	C
9	0.00	288.00	0.515	DCON2	0.092	0.179	-94.860	174.953	1463.801	13.500	99.000	E
10	288.00	288.00	0.702	DCON2	0.126	0.179	-225 707	464.658	-419.712	13.500	150.000	I
11	576.00	288.00	0.702	DCON2	0.126	0.179	-225.707	464.658	419.712	13.500	150.000	I
12	864.00	288.00	0.515	DCON2	0 092	0.179	-94.860	174.953	-1464	13.500	99.000	E
13	0 00	576.00	0.515	DCON2	0.092	0.179	-94.860	-174.953	1463.801	13.500	99.000	E
14	288.00	576.00	0.702	DCON2	0.126	0.179	-225.707	-464.658	-419.712	13.500	150.000	I
15	576.00	576.00	0.702	DCON2	0.126	0.179	-225.707	-464.658	419.712	13.500	150.000	I
16	864.00	576.00	0.515	DCON2	0.092	0.179	-94.860	-174.953	-1464	13.500	99.000	E
17	0.00	864.00	1.000	DCON2	0.179	0.179	-54.696	1962.206	1145.680	13.500	73.500	C
18	288.00	864.00	0.802	DCON2	0.144	0.179	-119.738	3777.943	-348.558	13.500	123.000	E
19	576.00	864.00	0.802	DCON2	0.144	0.179	-119.738	3777.943	348.558	13.500	123.000	E
20	864.00	864.00	1.000	DCON2	0.179	0.179	-54.696	1962.206	-1146	13.500	73.500	C

Figure 4: SAFE Output For Punching Shear

Following are explanations of each of the column headings in the output shown in Figure 4:

POINT = SAFE point object ID at location where punching shear stress is reported
X = X-coordinate of POINT

Y	=	Y-coordinate of POINT
RATIO	=	Punching shear stress divided by punching shear capacity
COMBO	=	Load combination that produces maximum punching shear stress
VMAX	=	Punching shear stress with the maximum absolute value
VCAP	=	Punching shear stress capacity with a phi factor included, i.e., ϕv_C
V	=	Shear used in punching shear stress calculation
MX	=	Moment about a line through the column centroid and parallel to the X-axis used in punching shear stress calculation
MY	=	Moment about a line through the column centroid and parallel to the Y-axis used in punching shear stress calculation
DEPTH	=	Effective depth for punching shear calculated as the average of the effective depths in the X and Y directions
PERIM	=	Perimeter length of critical section for punching shear
LOC	=	Identifier for column location: I is an interior column, E is an edge column, and C is a corner column

We will now derive some of the results shown in Figure 4 using hand calculations. The results we will calculate are the shear stress, shear capacity and the shear ratio for an interior column, an edge column with the edge parallel to the X-axis, an edge column with the edge parallel to the Y-axis, and a corner column. We will calculate these values using the SAFE formulation and then calculate them again using the formulation in the PCA Publication (Ref 2). Finally, we will compare the results obtained from SAFE, our hand calculations using the SAFE formulation, and our hand calculations using the PCA Publication formulation.

6c. Hand Calculation For Interior Column Using SAFE Method

$d = [(15 - 1) + (15 - 2)] / 2 = 13.5"$

Refer to Figure 5.

$b_0 = 49.5 + 25.5 + 49.5 + 25.5 = 150"$

$$\gamma_{fX} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{49.5}{25.5}}} = 0.482$$

$$\gamma_{fY} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{25.5}{49.5}}} = 0.324$$

The coordinates of the center of the column (x_1, y_1) are taken as (0, 0).

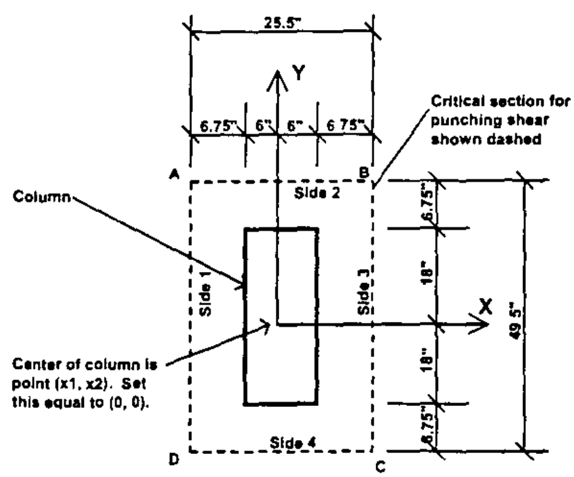


Figure 5: Interior Column, Point 10 in SAFE Model

The following table is used for calculating the centroid of the critical section for punching shear. Side 1 , Side 2, Side 3 and Side 4 refer to the sides of the critical section for punching shear as identified in Figure 5.

Item	Side 1	Side 2	Side 3	Side 4	Sum
x_2	-12.75	0	12.75	0	N.A.
y_2	0	24.75	0	-24.75	N.A.
L	49.5	25.5	49.5	25.5	$b_0 = 150$
d	13.5	13.5	13.5	13.5	N.A.
Ld	668.25	344.25	668.25	344.25	2025
Ldx_2	-8520.19	0	8520.19	0	0
Ldy_2	0	8520.19	0	-8520.19	0

$$x_3 = \frac{\sum Ldx_2}{Ld} = \frac{0}{2025} = 0"$$

$$y_3 = \frac{\sum Ldy_2}{Ld} = \frac{0}{2025} = 0"$$

The following table is used to calculate I_{XX}, I_{YY} and I_{XY}. The values for I_{XX}, I_{YY} and I_{XY} are given in the column titled "Sum".

Item	Side 1	Side 2	Side 3	Side 4	Sum
L	49.5	25.5	49.5	25.5	N.A.
d	13.5	13.5	13.5	13.5	N.A.
$x_2 - x_3$	-12.75	0	12.75	0	N.A.
$y_2 - y_3$	0	24.75	0	24.75	N.A.
Parallel to	Y-Axis	X-axis	Y-Axis	X-axis	N.A.
Equations	5b, 6b, 7	5a, 6a, 7	5b, 6b, 7	5a, 6a, 7	N.A.
I_{XX}	146597	210875	146597	210875	714944
I_{YY}	108633	23882	108633	23882	265030
I_{XY}	0	0	0	0	0

From the SAFE output at column point 10:

$V_U = -225.707$ k
$M_{UX} = 464.658$ k-in
$M_{UY} = -419.712$ k-in

At the point labeled A in Figure 5, $x_4 = -12.75$ and $y_4 = 24.75$, thus:

$$v_U = \frac{-225.707}{150*13.5} + \frac{0.482[464.658-(-225.707)(0-0)][265030(24.75-0)-(0)(-12.75-0)]}{(714944)(265030)-(0)^2} - \frac{0.324[-419.712-(-225.707)(0-0)][714944(-12.75-0)-(0)(24.75-0)]}{(714944)(265030)-(0)^2}$$

v_U = -0.111 + 0.008 - 0.007 = -0.110 ksi at point A

At the point labeled B in Figure 5, x_4 = 12.75 and y_4 = 24.75, thus:

$$v_U = \frac{-225.707}{150*13.5} + \frac{0.482[464.658-(-225.707)(0-0)][265030(24.75-0)-(0)(12.75-0)]}{(714944)(265030)-(0)^2} - \frac{0.324[-419.712-(-225.707)(0-0)][714944(12.75-0)-(0)(24.75-0)]}{(714944)(265030)-(0)^2}$$

v_U = -0.111 + 0.008 + 0.007 = -0.097 ksi at point B

At the point labeled C in Figure 5, x_4 = 12.75 and y_4 = -24.75, thus:

$$v_U = \frac{-225.707}{150*13.5} + \frac{0.482[464.658-(-225.707)(0-0)][265030(-24.75-0)-(0)(12.75-0)]}{(714944)(265030)-(0)^2} - \frac{0.324[-419.712-(-225.707)(0-0)][714944(12.75-0)-(0)(-24.75-0)]}{(714944)(265030)-(0)^2}$$

v_U = -0.111 - 0.008 + 0.007 = -0.113 ksi at point C

At the point labeled D in Figure 5, x_4 = -12.75 and y_4 = -24.75, thus:

$$v_U = \frac{-225.707}{150*13.5} + \frac{0.482[464.658-(-225.707)(0-0)][265030(24.75-0)-(0)(-12.75-0)]}{(714944)(265030)-(0)^2} - \frac{0.324[-419.712-(-225.707)(0-0)][714944(-12.75-0)-(0)(-24.75-0)]}{(714944)(265030)-(0)^2}$$

v_U = -0.111 - 0.008 - 0.007 = -0.126 ksi at point D

Point D has the largest absolute value of v_u, thus v_{max} = 0.126 ksi

The shear capacity is calculated based on the smallest of ACI 318-95 equations 11-35, 11-36 and 11-37 with the b_0 and d terms removed to convert force to stress.

$$\varphi v_C = \frac{0.85\left(2+\frac{4}{36/12}\right)\sqrt{4000}}{1000} = 0.179 \text{ ksi per equation 11-35}$$

$$\varphi vc = \frac{0.85\left(\frac{40*13.5}{150}+2\right)\sqrt{4000}}{1000} = 0.301 \text{ ksi per equation 11-36}$$

$$\varphi vc = \frac{0.85*4*\sqrt{4000}}{1000} = 0.215 \text{ ksi per equation 11-37}$$

Equation 11-35 yields the smallest value of $\boxed{\phi v_C = 0.179 \text{ ksi}}$, and thus this is the shear capacity.

$$\boxed{Shear\ Ratio = \frac{vu}{\varphi vc} = \frac{0.126}{0.179} = 0.702}$$

6d. Hand Calculation For Edge Column With Edge Parallel To X-Axis Using SAFE Method

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

Refer to Figure 6.

$b_0 = 48.75 + 25.5 + 48.75 = 123"$

$$\gamma_{VX} = 1 - \frac{1}{1+\left(\frac{2}{3}\right)\sqrt{\frac{48.75}{25.5}}} = 0.480$$

$$\gamma_{VY} = 1 - \frac{1}{1+\left(\frac{2}{3}\right)\sqrt{\frac{25.5}{48.75}}} = 0.325$$

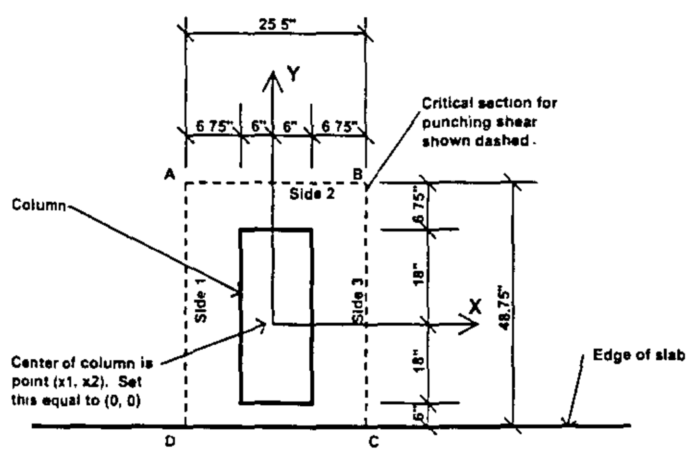


Figure 6: Edge Column With Edge Parallel To X-Axis, Point 6 in SAFE Model

The coordinates of the center of the column (x_1, y_1) are taken as (0, 0).

The following table is used for calculating the centroid of the critical section for punching shear. Side 1, Side 2 and Side 3 refer to the sides of the critical section for punching shear as identified in Figure 6.

Item	Side 1	Side 2	Side 3	Sum
x_2	-12.75	0	12.75	N.A.
y_2	0.375	24.75	0.375	N.A.
L	48.75	25.5	48.75	$b_0 = 123$
d	13.5	13.5	13.5	N.A.
Ld	658.125	344.25	658.125	1660.5
Ldx_2	-8391.09	0	8391.09	0
Ldy_2	246.80	8520.19	246.80	9013.78

$$x_3 = \frac{\sum Ldx_2}{Ld} = \frac{0}{1660.5} = 0''$$

$$y_3 = \frac{\sum Ldy_2}{Ld} = \frac{9013.78}{1660.5} = 5.43''$$

The following table is used to calculate I_{XX}, I_{YY} and I_{XY}. The values for I_{XX}, I_{YY} and I_{XY} are given in the column titled "Sum".

Item	Side 1	Side 2	Side 3	Sum
L	48.75	25.5	48.75	N.A.
d	13.5	13.5	13.5	N.A.
$x_2 - x_3$	-12.75	0	12.75	N.A.
$y_2 - y_3$	-5.05	19.32	-5.05	N.A.
Parallel to	Y-Axis	X-axis	Y-Axis	N.A.
Equations	5b, 6b, 7	5a, 6a, 7	5b, 6b, 7	N.A.
I_{XX}	157141	128518	157141	442800
I_{YY}	106986	23883	106986	237855
I_{XY}	42403	0	-42403	0

From the SAFE output at column point 6:

$V_U = -119.738$ k
$M_{UX} = 3778$ k-in
$M_{UY} = -348.588$ k-in

At the point labeled A in Figure 6, $x_4 = -12.75$ and $y_4 = 24.75$, thus:

$$v_U = \frac{-119.738}{123 * 13.5} + \frac{0.480[3778 - (-119.738)(5.43 - 0)][237855(24.75 - 5.43) - (0)(-12.75 - 0)]}{(442800)(237855) - (0)^2} - \frac{0.325[-348.588 - (-119.738)(0 - 0)][442800(-12.75 - 0) - (0)(24.75 - 5.43)]}{(442800)(237855) - (0)^2}$$

v_U = -0.0721 - 0.0655 - 0.0061 = -0.144 ksi at point A

At the point labeled B in Figure 6, x_4 = 12.75 and y_4 = 24.75, thus:

$$v_U = \frac{-119.738}{123 * 13.5} + \frac{0.480[3778 - (-119.738)(5.43 - 0)][237855(24.75 - 5.43) - (0)(12.75 - 0)]}{(442800)(237855) - (0)^2} - \frac{0.325[-348.588 - (-119.738)(0 - 0)][442800(12.75 - 0) - (0)(24.75 - 5.43)]}{(442800)(237855) - (0)^2}$$

v_U = -0.072 - 0.065 + 0.006 = -0.131 ksi at point B

At the point labeled C in Figure 6, x_4 = 12.75 and y_4 = -24, thus:

$$v_U = \frac{-119.738}{123 * 13.5} + \frac{0.480[3778 - (-119.738)(5.43 - 0)][237855(-24 - 5.43) - (0)(12.75 - 0)]}{(442800)(237855) - (0)^2} - \frac{0.325[-348.588 - (-119.738)(0 - 0)][442800(12.75 - 0) - (0)(-24 - 5.43)]}{(442800)(237855) - (0)^2}$$

v_U = -0.072 + 0.100 + 0.006 = 0.034 ksi at point C

At the point labeled D in Figure 6, x_4 = -12.75 and y_4 = -24, thus:

$$v_U = \frac{-119.738}{123 * 13.5} + \frac{0.480[3778 - (-119.738)(5.43 - 0)][237855(-24 - 5.43) - (0)(-12.75 - 0)]}{(442800)(237855) - (0)^2} - \frac{0.325[-348.588 - (-119.738)(0 - 0)][442800(-12.75 - 0) - (0)(-24 - 5.43)]}{(442800)(237855) - (0)^2}$$

v_U = -0.072 + 0.100 - 0.006 = 0.022 ksi at point D

$\boxed{\text{Point A has the largest absolute value of } v_u \text{, thus } v_{max} = 0.144 \text{ ksi}}$

The shear capacity is calculated based on the smallest of ACI 318-95 equations 11-35, 11-36 and 11-37 with the b_0 and d terms removed to convert force to stress.

$$\varphi v_C = \frac{0.85\left(2 + \frac{4}{36/12}\right)\sqrt{4000}}{1000} = 0.179 \text{ ksi per equation 11-35}$$

$$\varphi vc = \frac{0.85\left(\dfrac{30*13.5}{123}+2\right)\sqrt{4000}}{1000} = 0.285 \text{ ksi per equation 11-36}$$

$$\varphi vc = \frac{0.85*4*\sqrt{4000}}{1000} = 0.215 \text{ ksi per equation 11-37}$$

Equation 11-35 yields the smallest value of $\boxed{\phi v_C = 0.179 \text{ ksi}}$, and thus this is the shear capacity.

$$\boxed{Shear\ Ratio = \frac{vu}{\varphi vc} = \frac{0.144}{0.179} = 0.802}$$

6e. Hand Calculation For Edge Column With Edge Parallel To Y-Axis Using SAFE Method

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

Refer to Figure 7.

b_0 = 24.75 + 49.5 + 24.75 = 99"

$$\gamma_{VX} = 1 - \frac{1}{1+\left(\dfrac{2}{3}\right)\sqrt{\dfrac{49.5}{24.75}}} = 0.485$$

$$\gamma_{VY} = 1 - \frac{1}{1+\left(\dfrac{2}{3}\right)\sqrt{\dfrac{24.75}{49.5}}} = 0.320$$

The coordinates of the center of the column (x_1, y_1) are taken as (0, 0).

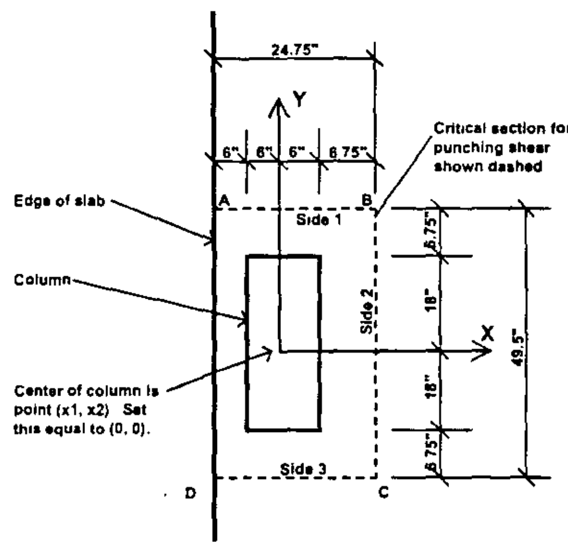


Figure 7: Edge Column With Edge Parallel To Y-Axis, Point 9 in SAFE Model

The following table is used for calculating the centroid of the critical section for punching shear. Side 1 , Side 2 and Side 3 refer to the sides of the critical section for punching shear as identified in Figure 7.

Item	Side 1	Side 2	Side 3	Sum
x_2	0.375	12.75	0.375	N.A.
y_2	24.75	0	-24.75	N.A.
L	24.75	49.5	24.75	$b_0 = 99$
d	13.5	13.5	13.5	N.A.
Ld	344.125	668.25	344.125	1336.5
Ldx_2	125.30	8520.19	125.30	8770.78
Ldy_2	8269.59	0	-8269.59	0

$$x_3 = \frac{\sum Ldx_2}{Ld} = \frac{8770.78}{1336.5} = 6.56"$$

$$y_3 = \frac{\sum Ldy_2}{Ld} = \frac{0}{1336.5} = 0"$$

The following table is used to calculate I_{XX}, I_{YY} and I_{XY}. The values for I_{XX}, I_{YY} and I_{XY} are given in the column titled "Sum".

Item	Side 1	Side 2	Side 3	Sum
L	24.75	49.5	24.75	N.A.
d	13.5	13.5	13.5	N.A.
$x_2 - x_3$	-6.19	6.19	-6.19	N.A.
$y_2 - y_3$	24.75	0	-24.75	N.A.
Parallel to	X-axis	Y-Axis	X-axis	N.A.
Equations	5a, 6a, 7	5b, 6b, 7	5a, 6a, 7	N.A.
I_{XX}	204672.4	146597.2	204672.4	555942
I_{YY}	34922.6	25584.1	34922.6	95429
I_{XY}	-51168	0	51168	0

From the SAFE output at column point 6:

V_U = -94.86 k
M_{UX} = 174.953 k-in
M_{UY} = 1463.801 k-in

At the point labeled A in Figure 7, x_4 = -12 and y_4 = 24.75, thus:

$$v_U = \frac{-94.86}{123 * 13.5} + \frac{0.485[174.953 - (-94.86)(0-0)][95429(24.75-0) - (0)(-12-6.56)]}{(555942)(95429) - (0)^2} - \frac{0.320[1463.801 - (-94.86)(6.56-0)][555942(-12-6.56) - (0)(24.75-0)]}{(555942)(95429) - (0)^2}$$

v_U = -0.071 + 0.004 + 0.052 = -0.015 ksi at point A

At the point labeled B in Figure 7, x_4 = 12.75 and y_4 = 24.75, thus:

$$v_U = \frac{-94.86}{123 * 13.5} + \frac{0.485[174.953 - (-94.86)(0-0)][95429(24.75-0) - (0)(12.75-6.56)]}{(555942)(95429) - (0)^2} - \frac{0.320[1463.801 - (-94.86)(6.56-0)][555942(12.75-6.56) - (0)(24.75-0)]}{(555942)(95429) - (0)^2}$$

v_U = -0.071 + 0.004 - 0.017 = -0.084 ksi at point B

At the point labeled C in Figure 7, x_4 = 12.75 and y_4 = -24.75, thus:

$$v_U = \frac{-94.86}{123 * 13.5} + \frac{0.485[174.953 - (-94.86)(0-0)][95429(-24.75-0) - (0)(12.75-6.56)]}{(555942)(95429) - (0)^2} - \frac{0.320[1463.801 - (-94.86)(6.56-0)][555942(12.75-6.56) - (0)(-24.75-0)]}{(555942)(95429) - (0)^2}$$

v_U = -0.071 - 0.004 - 0.017 = 0.092 ksi at point C

At the point labeled D in Figure 7, x_4 = -12 and y_4 = -24.75, thus:

$$v_U = \frac{-94.86}{123 * 13.5} + \frac{0.485[174.953 - (-94.86)(0-0)][95429(-24.75-0) - (0)(-12-6.56)]}{(555942)(95429) - (0)^2} - \frac{0.320[1463.801 - (-94.86)(6.56-0)][555942(-12-6.56) - (0)(-24.75-0)]}{(555942)(95429) - (0)^2}$$

v_U = -0.071 - 0.004 + 0.052 = -0.023 ksi at point D

Point C has the largest absolute value of v_u, thus v_{max} = 0.092 ksi

The shear capacity is calculated based on the smallest of ACI 318-95 equations 11-35, 11-36 and 11-37 with the b_0 and d terms removed to convert force to stress.

$$\varphi v_C = \frac{0.85\left(2 + \frac{4}{36/12}\right)\sqrt{4000}}{1000} = 0.179 \text{ ksi per equation 11-35}$$

$$\varphi vc = \frac{0.85\left(\frac{30 * 13.5}{99} + 2\right)\sqrt{4000}}{1000} = 0.327 \text{ ksi per equation 11-36}$$

$$\varphi vc = \frac{0.85 * 4 * \sqrt{4000}}{1000} = 0.215 \text{ ksi per equation 11-37}$$

Equation 11-35 yields the smallest value of $\boxed{\phi v_C = 0.179 \text{ ksi}}$, and thus this is the shear capacity.

$$\boxed{Shear\ Ratio = \frac{vu}{\varphi vc} = \frac{0.092}{0.179} = 0.515}$$

6f. Hand Calculation For Corner Column Using SAFE Method

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

Refer to Figure 8.

$b_0 = 24.75 + 48.75 = 73.5"$.

$$\gamma_{vX} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{48.75}{24.75}}} = 0.483$$

$$\gamma_{vY} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{24.75}{48.75}}} = 0.322$$

The coordinates of the center of the column (x_1, y_1) are taken as (0, 0).

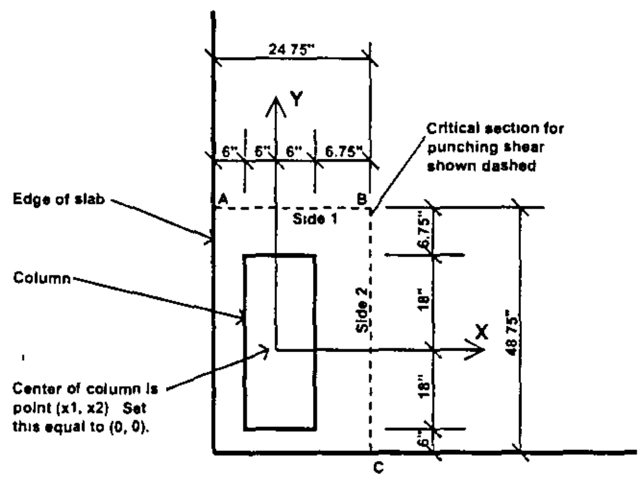


Figure 8: Corner Column, Point 5 in SAFE Model

The following table is used for calculating the centroid of the critical section for punching shear. Side 1 and Side 2 refer to the sides of the critical section for punching shear as identified in Figure 8.

Item	Side 1	Side 2	Sum
x_2	0.375	12.75	N.A.
y_2	24.75	0.375	N.A.
L	24.75	48.75	$b_0 = 73.5$
d	13.5	13.5	N.A.
Ld	334.125	658.125	992.25
Ldx_2	125.30	8391.09	8516.39
Ldy_2	8269.59	246.80	8516.39

$$x_3 = \frac{\sum Ldx_2}{Ld} = \frac{8516.39}{992.25} = 8.58''$$

$$y_3 = \frac{\sum Ldy_2}{Ld} = \frac{8516.39}{992.25} = 8.58''$$

The following table is used to calculate I_{XX}, I_{YY} and I_{XY}. The values for I_{XX}, I_{YY} and I_{XY} are given in the column titled "Sum".

Item	Side 1	Side 2	Sum
L	24.75	48.75	N.A.
d	13.5	13.5	N.A.
$x_2 - x_3$	-8.21	4.17	N.A.
$y_2 - y_3$	16.17	-8.21	N.A.
Parallel to	X-axis	Y-Axis	N.A.
Equations	5a, 6a, 7	5b, 6b, 7	N.A.
I_{XX}	87332	184673	272005
I_{YY}	44640	11428	56069
I_{XY}	-44338	-22510	-66848

From the SAFE output at column point 5:

$V_U = -54.696$ k
$M_{UX} = -1962$ k-in
$M_{UY} = 1145.68$ k-in

At the point labeled A in Figure 8, $x_4 = -12$ and $y_4 = 24.75$, thus:

$$v_U = \frac{-54.696}{73.5 * 13.5} + \frac{0.483[1962 - (-54.696)(8.58 - 0)][56069(24.75 - 8.58) - (-66848)(-12 - 8.58)]}{(272005)(56069) - (-66848)^2} - \frac{0.322[1145.68 - (-54.696)(8.58 - 0)][272005(-12 - 8.58) - (-66848)(24.75 - 8.58)]}{(272005)(56069) - (-66848)^2}$$

v_U = -0.055 + 0.031 + 0.091 = 0.067 ksi at point A

At the point labeled B in Figure 8, x_4 = 12.75 and y_4 = 24.75, thus:

$$v_U = \frac{-54.696}{73.5 * 13.5} + \frac{0.483[1962 - (-54.696)(8.58 - 0)][56069(24.75 - 8.58) - (-66848)(12.75 - 8.58)]}{(272005)(56069) - (-66848)^2} - \frac{0.322[1145.68 - (-54.696)(8.58 - 0)][272005(12.75 - 8.58) - (-66848)(24.75 - 8.58)]}{(272005)(56069) - (-66848)^2}$$

v_U = -0.055 - 0.079 - 0.045 = -0.179 ksi at point B

At the point labeled C in Figure 8, x_4 = 12.75 and y_4 = -24, thus:

$$v_U = \frac{-54.696}{73.5 * 13.5} + \frac{0.483[1962 - (-54.696)(8.58 - 0)][56069(-24 - 8.58) - (-66848)(12.75 - 8.58)]}{(272005)(56069) - (-66848)^2} - \frac{0.322[1145.68 - (-54.696)(8.58 - 0)][272005(12.75 - 8.58) - (-66848)(-24 - 8.58)]}{(272005)(56069) - (-66848)^2}$$

v_U = -0.055 + 0.104 + 0.021 = 0.070 ksi at point C

Point B has the largest absolute value of v_u, thus v_{max} = 0.179 ksi

The shear capacity is calculated based on the smallest of ACI 318-95 equations 11-35, 11-36 and 11-37 with the b_0 and d terms removed to convert force to stress.

$$\varphi v_C = \frac{0.85\left(2 + \frac{4}{36/12}\right)\sqrt{4000}}{1000} = 0.179 \text{ ksi per equation 11-35}$$

$$\varphi v_C = \frac{0.85\left(\frac{20 * 13.5}{73.5} + 2\right)\sqrt{4000}}{1000} = 0.404 \text{ ksi per equation 11-36}$$

$$\varphi v_C = \frac{0.85 * 4 * \sqrt{4000}}{1000} = 0.215 \text{ ksi per equation 11-37}$$

Equation 11-35 yields the smallest value of $\boxed{\phi v_C = 0.179 \text{ ksi}}$, and thus this is the shear capacity.

$$\text{Shear Ratio} = \frac{v_U}{\varphi v_C} = \frac{0.179}{0.179} = 1.000$$

6g. Hand Calculation For Corner Column Using SAFE Method Except That I_{XY} Is Set To Zero

This calculation is done for comparison with the corner calculation using the PCA Publication method (Ref. 2). Note that the computer program SAFE does not do the calculation with I_{XY} set to zero.

The calculations in this case are exactly the same as in Section 6f, except that IXY is set to zero. Thus, using the intermediate results in Section 6f we can go right to calculating the stresses.

At the point labeled A in Figure 8, $x_4 = -12$ and $y_4 = 24.75$, thus:

$$v_U = \frac{-54.696}{73.5 * 13.5} + \frac{0.483[1962 - (-54.696)(8.58 - 0)][56069(24.75 - 8.58) - (0)(-12 - 8.58)]}{(272005)(56069) - (0)^2} - \frac{0.322[1145.68 - (-54.696)(8.58 - 0)][272005(-12 - 8.58) - (0)(24.75 - 8.58)]}{(272005)(56069) - (0)^2}$$

v_U = -0.055 - 0.043 + 0.080 = -0.018 ksi at point A

At the point labeled B in Figure 8, $x_4 = 12.75$ and $y_4 = 24.75$, thus:

$$v_U = \frac{-54.696}{73.5 * 13.5} + \frac{0.483[1962 - (-54.696)(8.58 - 0)][56069(24.75 - 8.58) - (0)(12.75 - 8.58)]}{(272005)(56069) - (0)^2} - \frac{0.322[1145.68 - (-54.696)(8.58 - 0)][272005(12.75 - 8.58) - (0)(24.75 - 8.58)]}{(272005)(56069) - (0)^2}$$

v_U = -0.055 - 0.043 - 0.016 = -0.114 ksi at point B

At the point labeled C in Figure 8, $x_4 = 12.75$ and $y_4 = -24$, thus:

$$v_U = \frac{-54.696}{73.5 * 13.5} + \frac{0.483[1962 - (-54.696)(8.58 - 0)][56069(-24 - 8.58) - (0)(12.75 - 8.58)]}{(272005)(56069) - (0)^2} - \frac{0.322[1145.68 - (-54.696)(8.58 - 0)][272005(12.75 - 8.58) - (0)(-24 - 8.58)]}{(272005)(56069) - (0)^2}$$

v_U = -0.055 + 0.086 - 0.016 = 0.015 ksi at point c

Point B has the largest absolute value of v_u, thus v_{max} = 0.114 ksi

6h. Terminology For PCA Publication Method of Calculating Punching Shear

a = Distance from center of column to centroid of critical section for punching shear

A_C = Area of critical section for punching shear

b_1 = Length of side of critical section for punching shear parallel to Y-axis for bending about X-axis, and parallel to the X-axis for bending about Y-axis

b_2 = Length of side of critical section for punching shear parallel to X-axis for bending about X-axis, and parallel to the Y-axis for bending about Y-axis

c = For bending about the X-axis, shorter Y distance from centroid of critical section for punching shear to an edge of the critical section for punching shear that is parallel to the Y-axis. For bending about the Y-axis, shorter X distance from centroid of critical section for punching shear to an edge of the critical section for punching shear that is parallel to the X-axis.

c' = For bending about the X-axis, longer Y distance from centroid of critical section for punching shear to an edge of the critical section for punching shear that is parallel to the Y-axis. For bending about the Y-axis, longer X distance from centroid of critical section for punching shear to an edge of the critical section for punching shear that is parallel to the X-axis.

J/c = Section modulus associated with critical section for punching shear

J/c' = Section modulus associated with critical section for punching shear

M_{UX} = Moment about line parallel to X-axis at center of column or load

M_{UY} = Moment about line parallel to Y-axis at center of column or load

$M_{UXtransformed}$ = Moment about line parallel to X-axis at centroid of critical section for punching shear

$M_{UYtransformed}$ = Moment about line parallel to Y-axis at centroid of critical section for punching shear

γ = Percent of M_{UX} resisted by shear per ACI 318-95 equations 11-41 and 13-1

6i. Basic Equation For PCA Publication Method

The basic equation for the PCA Publication method is:

$$v_U = \frac{V_U}{A_C} + \frac{\gamma_{VX} M_{UXtransformed} C_X}{J_X} + \frac{\gamma_{VY} M_{UYtransformed} C_Y}{J_Y} \qquad \text{Eq. 8}$$

Depending on the corner of the critical section for punching shear where you are calculating stress this equation may be in one of the following forms:

$$v_U = \frac{V_U}{A_C} - \frac{\gamma_{VX} M_{UXtransformed} C'_X}{J_X} - \frac{\gamma_{VY} M_{UYtransformed} C'_Y}{J_Y} \qquad \text{Eq. 9}$$

$$v_U = \frac{V_U}{A_C} - \frac{\gamma_{VX} M_{UXtransformed} C'_X}{J_X} + \frac{\gamma_{VY} M_{UYtransformed} C_Y}{J_Y} \qquad \text{Eq. 10}$$

$$v_U = \frac{V_U}{A_C} + \frac{\gamma_{VX} M_{UXtransformed} C_X}{J_X} - \frac{\gamma_{VY} M_{UYtransformed} C'_Y}{J_Y}$$ Eq. 11

6j. Hand Calculation For Interior Column Using PCA Publication Method

From the SAFE computer output given in Figure 4 for Point:

V_U = --225.707 k (upward positive)
M_{UX} = 464.658 k-in (right-hand rule)
M_{UY} = -419.712 k-in (right-hand rule)

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

To calculate data for bending about X-axis refer to Figure 9. Note that the direction and sign of M_{UX} shown in Figure 15 is consistent with that given in the SAFE output.

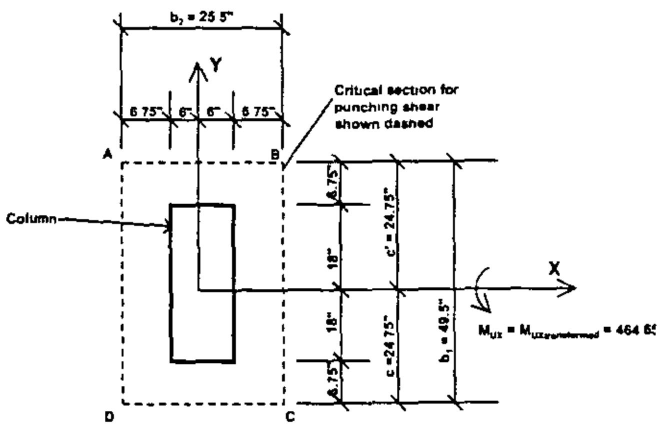


Figure 9: Interior Column, Point 10 in SAFE Model Bending About X-Axis

b_1 = 49.5"
b_2 = 25.5"

$$\gamma_{VX} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{49.5}{25.5}}} = 0.482$$

$A_C = 2(b_1 + b_2)d = 2 * (49.5 + 25.5) * 13.5 = 2025\ in^2$

$$\frac{J_X}{c_X} = \frac{b_1 d(b_1 + 3b_2) + d^3}{3} = \frac{49.5 * 13.5(49.5 + (3 * 25.5)) + 13.5^3}{3} = 28886.6\ in^3$$

$$\frac{J_X}{c'_X} = \frac{b_1 d(b_1 + 3b_2) + d^3}{3} = \frac{49.5 * 13.5(49.5 + (3 * 25.5)) + 13.5^3}{3} = 28886.6\ in^3$$

$$c_X = \frac{b_1}{2} = \frac{49.5}{2} = 24.75"$$

$$c'_x = \frac{b_1}{2} = \frac{49.5}{2} = 24.75"$$

$M_{UXtransformed} = M_{UX}$ = 464.658 k-in (direction of moment shown in Figure 9)

To calculate data for bending about Y-axis refer to Figure 10. Note that the direction and sign of M_{UY} shown in Figure 10 is consistent with that given in the SAFE output.

$b_1 = 25.5"$
$b_2 = 49.5"$

$$\gamma_{fY} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{25.5}{49.5}}} = 0.324$$

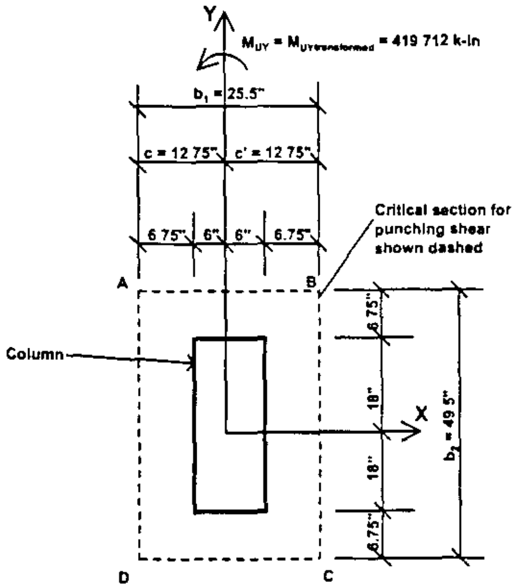


Figure 10: Interior Column, Point 10 in SAFE Model Bending About Y-Axis

$$\frac{J_Y}{c_Y} = \frac{b_1 d(b_1 + 3b_2) + d^3}{3} = \frac{25.5 * 13.5(25.5 + (3 * 49.5)) + 13.5^3}{3} = 20786.6\, in^3$$

$$\frac{J_y}{c'_y} = \frac{b_1 d(b_1 + 3b_2) + d^3}{3} = \frac{25.5 * 13.5(25.5 + (3 * 49.5)) + 13.5^3}{3} = 20786.6\, in^3$$

$$c_Y = \frac{b_1}{2} = \frac{25.5}{2} = 12.75"$$

$$c'_x = \frac{b_1}{2} = \frac{25.5}{2} = 12.75"$$

$M_{UYtransformed} = M_{UY} = 419.712$ k-in (direction of moment shown in Figure 10)

The punching shear stress at point A in Figures 9 and 10 is calculated using equation 10:

$$v_U = \frac{225.707}{2025} - \frac{0.482 * 464.658}{28886.6} + \frac{0.324 * 419.712}{20786.6} = 0.111 - .008 + .007 = 0.110\, ksi$$

The punching shear stress at point B in Figures 9 and 10 is calculated using equation 9:

$$v_U = \frac{225.707}{2025} - \frac{0.482 * 464.658}{28886.6} - \frac{0.324 * 419.712}{20786.6} = 0.111 - .008 - .007 = 0.096\, ksi$$

The punching shear stress at point C in Figures 9 and 10 is calculated using equation 11:

$$v_U = \frac{225.707}{2025} + \frac{0.482 * 464.658}{28886.6} - \frac{0.324 * 419.712}{20786.6} = 0.111 + .008 - .007 = 0.112\ ksi$$

The punching shear stress at point D in Figures 9 and 10 is calculated using equation 8:

$$v_U = \frac{225.707}{2025} + \frac{0.482 * 464.658}{28886.6} + \frac{0.324 * 419.712}{20786.6} = 0.111 + .008 + .007 = 0.126\ ksi$$

Point D has the largest absolute value of v_u, thus v_{max} = 0.126 ksi

6k. Hand Calculation For Edge Column With Edge Parallel To X-Axis Using PCA Publication Method

From the SAFE computer output given in Figure 4 for Point 6:

V_U = -119.738 k (upward positive)
M_{UX} = -3778 k-in (right-hand rule)
M_{UY} = -348.588 k-in (right-hand rule)

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

To calculate data for bending about X-axis refer to Figure 11. Note that the direction and sign of M_{UX} shown in Figure 11 is consistent with that given in the SAFE output. Also note that X-direction bending is bending perpendicular to the edge.

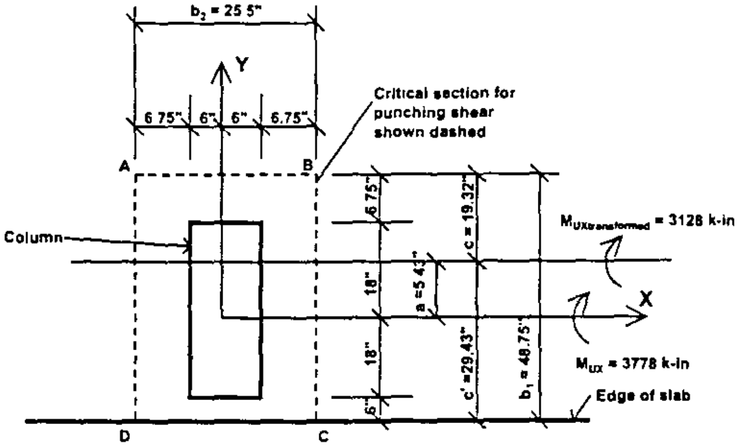


Figure 11: Edge Column With Edge Parallel To X-Axis Point 6 in SAFE Model, Bending About X-Axis

b_1 = 48.75"
b_2 = 25.5"

$$\gamma_{vX} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{48.75}{25.5}}} = 0.480$$

$A_C = (2b_1 + b_2)d$ = ((2 * 48.75) + 25.5) * 13.5 = 1660.5 in^2

$$\frac{J_X}{c_X} = \frac{2b_1^2 d(b_1 + 2b_2) + d^3(2b_1 + b_2)}{6b_1} =$$

$$\frac{2 * 48.75^2 * 13.5(48.75 + (2 * 25.5)) + 13.5^3((2 * 48.75) + 25.5)}{6 * 48.75} = 22917.3\ in^3$$

$$\frac{J_X}{c'_X} = \frac{2b_1^2 d(b_1 + 2b_2) + d^3(2b_1 + b_2)}{6(b_1 + b_2)} =$$

$$\frac{2 * 48.75^2 * 13.5(48.75 + (2 * 25.5)) + 13.5^3((2 * 48.75) + 25.5)}{6 * (48.75 + 25.5)} = 15046.7\ in^3$$

$$c_X = \frac{b_1^2}{2b_1 + b_2} = \frac{48.75^2}{(2 * 48.75) + 25.5} = 19.32''$$

$$c'_X = \frac{b_1(b_1 + b_2)}{2b_1 + b_2)} = \frac{48.75 * (48.75 + 25.5)}{(2 * 48.75) + 25.5} = 29.43''$$

$a_X = 18 + 6.75 - 19.32 = 5.43''$

$M_{UXtransformed} = 3778 - 119.738 * 5.43 = 3128$ k-in (direction of moment shown in Figure 11)

To calculate data for bending about Y-axis refer to Figure 12. Note that the direction and sign of M_{UY} shown in Figure 12 is consistent with that given in the SAFE output. Also note that Y-direction bending is bending parallel to the edge.

$b_1 = 25.5''$
$b_2 = 48.75''$

$$\gamma_{vY} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{25.5}{48.75}}} = 0.325$$

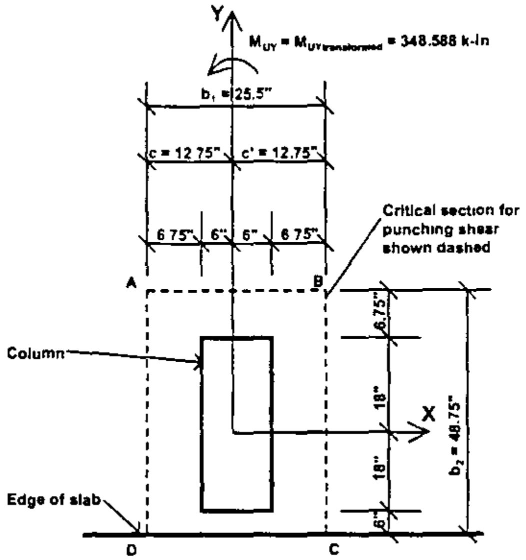


Figure 12: Edge Column With Edge Parallel To X-Axis
Point 6 in SAFE Model, Bending About Y-Axis

$$\frac{J_Y}{c_Y} = \frac{b_1 d(b_1 + 6b_2) + d^3}{6} = \frac{25.5 * 13.5(25.5 + (6 * 48.75)) + 13.5^3}{6} = 18655.3\ in^3$$

$$\frac{J_Y}{c'_Y} = \frac{b_1 d(b_1 + 6b_2) + d^3}{6} = \frac{25.5 * 13.5(25.5 + (6 * 48.75)) + 13.5^3}{6} = 18655.3\ in^3$$

$$c_Y = \frac{b_1}{2} = \frac{25.5}{2} = 12.75"$$

$$c'_Y = \frac{b_1}{2} = \frac{25.5}{2} = 12.75"$$

$M_{UYtransformed} = M_{UY} = 348.588$ k-in (direction of moment shown in Figure 12)

The punching shear stress at point A in Figures 11 and 12 is calculated using equation 8:

$$v_U = \frac{119.738}{1660.5} + \frac{0.480 * 3128}{22917.3} + \frac{0.325 * 348.588}{18655.3} = 0.072 + .066 + .006 = 0.144\ ksi$$

The punching shear stress at point B in Figures 11 and 12 is calculated using equation 11:

$$v_U = \frac{119.738}{1660.5} + \frac{0.480 * 3128}{22917.3} - \frac{0.325 * 348.588}{18655.3} = 0.072 + .066 - .006 = 0.132\ ksi$$

The punching shear stress at point C in Figures 11 and 12 is calculated using equation 9:

$$v_U = \frac{119.738}{1660.5} - \frac{0.480 * 3128}{15046.7} - \frac{0.325 * 348.588}{18655.3} = 0.072 - .100 - .006 = -0.034\ ksi$$

The punching shear stress at point D in Figures 11 and 12 is calculated using equation 10:

$$v_U = \frac{119.738}{1660.5} - \frac{0.480 * 3128}{15046.7} + \frac{0.325 * 348.588}{18655.3} = 0.072 - .100 + .006 = -0.022\ ksi$$

Point A has the largest absolute value of v_u, thus $v_{max} = 0.144$ ksi

6l. Hand Calculation For Edge Column With Edge Parallel To Y-Axis Using PCA Publication Method

From the SAFE computer output given in Figure 4 for Point 9:

$V_U = -94.86$ k (upward positive)
$M_{UX} = 174.953$ k-in (right-hand rule)
$M_{UY} = 1463.801$ k-in (right-hand rule)

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

To calculate data for bending about X-axis refer to Figure 13. Note that the direction and sign of M_{UX} shown in Figure 13 is consistent with that given in the SAFE output. Also note that X-direction bending is bending parallel to the edge.

$b_1 = 49.5''$
$b_2 = 24.75''$

$$\gamma_{fX} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{49.5}{24.75}}} = 0.485$$

$$\begin{aligned} A_C &= (b_1 + 2b_2)d \\ &= (49.5 + (2 * 24.75)) * 13.5 \\ &= 1336.5 \text{ in}^2 \end{aligned}$$

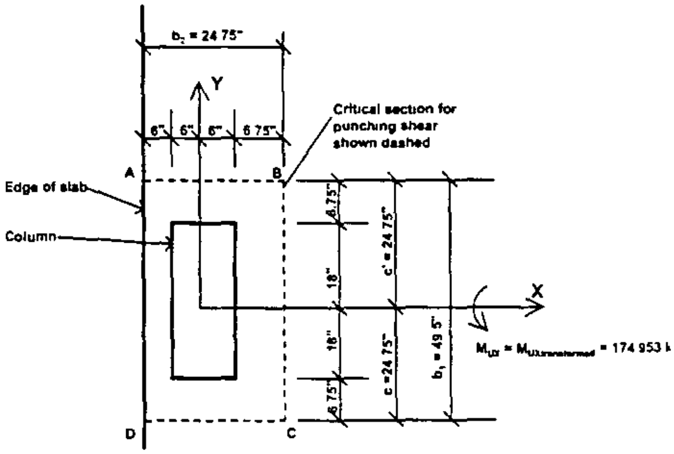


Figure 13: Edge Column With Edge Parallel To Y-Axis
Point 9 in SAFE Model, Bending About X-Axis

$$\frac{J_X}{c_X} = \frac{b_1 d(b_1 + 6b_2) + d^3}{6} = \frac{49.5 * 13.5(49.5 + (6 * 24.75)) + 13.5^3}{6} = 22462.3\ in^3$$

$$\frac{J_X}{c'_X} = \frac{b_1 d(b_1 + 6b_2) + d^3}{6} = \frac{49.5 * 13.5(49.5 + (6 * 24.75)) + 13.5^3}{6} = 22462.3\ in^3$$

$$c_X = \frac{b_1}{2} = \frac{49.5}{2} = 24.75''$$

$$c'_X = \frac{b_1}{2} = \frac{49.5}{2} = 24.75''$$

$M_{UXtransformed} = M_{UX} = 174.953$ k-in (direction of moment shown in Figure 13)

To calculate data for bending about Y-axis refer to Figure 14. Note that the direction and sign of M_{UY} shown in Figure 14 is consistent with that given in the SAFE output. Also note that Y-direction bending is bending perpendicular to the edge.

$b_1 = 24.75''$
$b_2 = 49.5''$

$$\gamma_{fY} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{24.75}{49.5}}} = 0.320$$

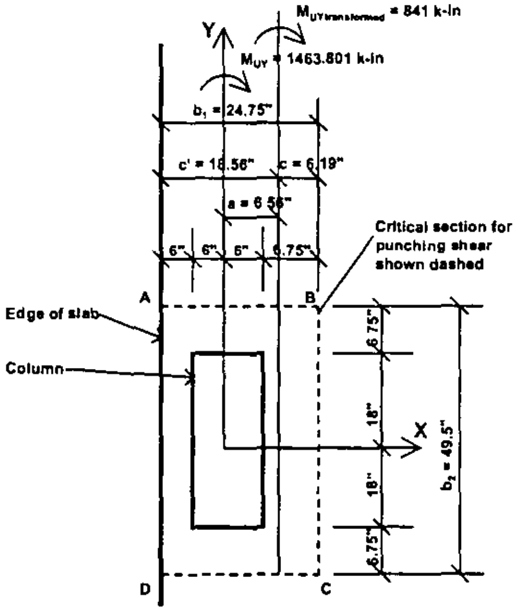


Figure 14: Edge Column With Edge Parallel To Y-Axis
Point 6 in SAFE Model, Bending About Y-Axis

$$\frac{J_Y}{c_Y} = \frac{2b_1^2 d(b_1 + 2b_2) + d^3(2b_1 + b_2)}{6b_1} =$$

$$\frac{2 * 24.75^2 * 13.5(24.75 + (2 * 49.5)) + 13.5^3((2 * 24.75) + 49.5)}{6 * 24.75} = 15422.9\ in^3$$

$$\frac{J_Y}{c'_Y} = \frac{2b_1^2 d(b_1 + 2b_2) + d^3(2b_1 + b_2)}{6(b_1 + b_2)} =$$

$$\frac{2 * 24.75^2 * 13.5(24.75 + (2 * 49.5)) + 13.5^3((2 * 24.75) + 49.5)}{6 * (24.75 + 49.5)} = 5141.0\ in^3$$

$$c_Y = \frac{b_1^2}{2b_1 + b_2} = \frac{24.75^2}{(2 * 24.75) + 49.5} = 6.19"$$

$$c'_Y = \frac{b_1(b_1 + b_2)}{2b_1 + b_2)} = \frac{24.75 * (24.75 + 49.5)}{(2 * 24.75) + 49.5} = 18.56"$$

$a_Y = 6 + 6.75 - 6.19 = 6.56"$

$M_{UYtransformed} = 1463.801 - 94.86 * 6.56 = 841$ k-in (direction of moment shown in Figure 14)

The punching shear stress at point A in Figures 13 and 14 is calculated using equation 9:

$$v_U = \frac{94.86}{1336.5} - \frac{0.485 * 174.953}{22462.3} - \frac{0.320 * 841}{5141.0} = 0.071 - .004 - .052 = 0.015\ ksi$$

The punching shear stress at point B in Figures 13 and 14 is calculated using equation 10:

$$v_U = \frac{94.86}{1336.5} - \frac{0.485 * 174.953}{22462.3} + \frac{0.320 * 841}{15422.9} = 0.071 - .004 + .017 = 0.084\ ksi$$

The punching shear stress at point C in Figures 13 and 14 is calculated using equation 8:

$$v_U = \frac{94.86}{1336.5} + \frac{0.485 * 174.953}{22462.3} + \frac{0.320 * 841}{15422.9} = 0.071 + .004 + .017 = 0.092\ ksi$$

The punching shear stress at point D in Figures 13 and 14 is calculated using equation 11:

$$v_U = \frac{94.86}{1336.5} + \frac{0.485 * 174.953}{22462.3} - \frac{0.320 * 841}{5141.0} = 0.071 + .004 - .052 = 0.023\ ksi$$

Point C has the largest absolute value of v_u, thus $v_{max} = 0.092$ ksi

6m. Hand Calculation For Corner Column Using PCA Publication Method

From the SAFE computer output given in Figure 4 for Point 5:

V_U = -54.696 k (upward positive)
M_{UX} = -1962 k-in (right-hand rule)
M_{UY} = 1145.68 k-in (right-hand rule)

d = [(15 - 1) + (15 - 2)] / 2 = 13.5"

To calculate data for bending about X-axis refer to Figure 15. Note that the direction and sign of M_{UX} shown in Figure 15 is consistent with that given in the SAFE output.

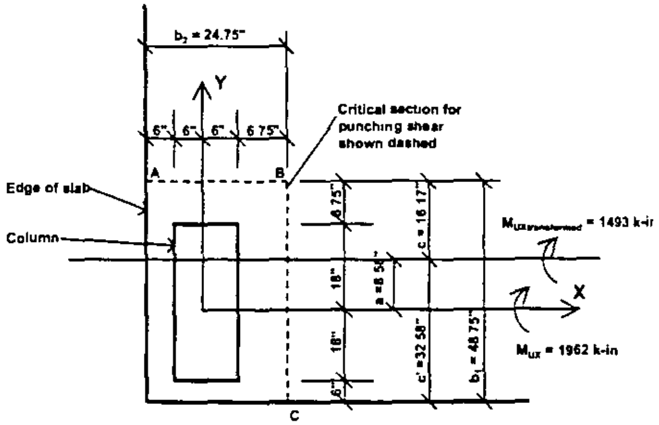


Figure 15: Corner Column, Point 5 in SAFE Model Bending About X-Axis

b_1 = 48.75"
b_2 = 24.75"

$$\gamma_{vx} = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{48.75}{24.75}}} = 0.483$$

$A_C = (b_1 + b_2)d = (48.75 + 24.75) * 13.5 = 992.25 \text{ in}^2$

$$\frac{J_x}{c_x} = \frac{b_1^2 d(b_1 + 4b_2) + d^3(b_1 + b_2)}{6b_1} =$$

$$\frac{48.75^2 * 13.5(48.75 + (4 * 24.75)) + 13.5^3(48.75 + 24.75)}{6 * 48.75} = 16824.6\, in^3$$

$$\frac{J_x}{c'_x} = \frac{b_1^2 d(b_1 + 4b_2) + d^3(b_1 + b_2)}{6(b_1 + 2b_2)} =$$

$$\frac{48.75^2 * 13.5(48.75 + (4 * 24.75)) + 13.5^3(48.75 + 24.75)}{6 * (48.75 + (2 * 24.75))} = 8348.1\, in^3$$

$$c_x = \frac{b_1^2}{2(b_1 + b_2)} = \frac{48.75^2}{2 * (48.75 + 24.75)} = 16.17"$$

$$c'_x = \frac{b_1(b_1 + 2b_2)}{2(b_1 + b_2)} = \frac{48.75 * (48.75 + (2 * 24.75))}{2 * (48.75 + 24.75)} = 32.58"$$

a_X = 18 + 6.75 - 16.17 = 8.58"

$M_{UXtransformed}$ = 1962 - 54.696 * 8.58 = 1493 k-in (direction of moment shown in Figure 15)

To calculate data for bending about Y-axis refer to Figure 16. Note that the direction and sign of M_{UY} shown in Figure 16 is consistent with that given in the SAFE output.

b_1 = 24.75"
b_2 = 48.75"

$$\gamma_{fY} = 1 - \frac{1}{1+\left(\frac{2}{3}\right)\sqrt{\frac{24.75}{48.75}}} = 0.322$$

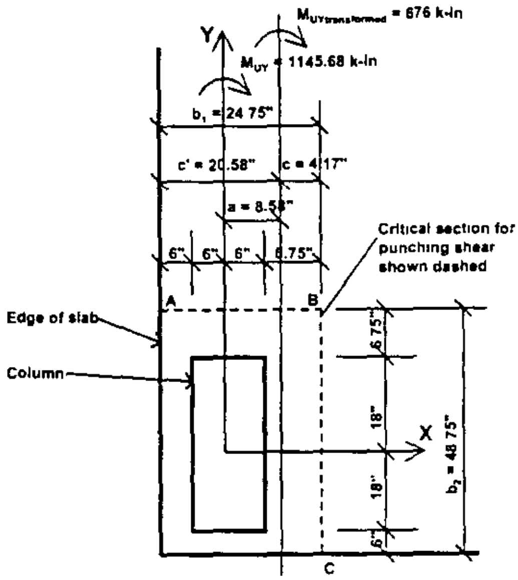


Figure 16: Corner Column, Point 5 in SAFE Model Bending About Y-Axis

$$\frac{J_Y}{c_Y} = \frac{b_1^2 d(b_1+4b_2)+d^3(b_1+b_2)}{6b_1} =$$

$$\frac{24.75^2 * 13.5(24.75+(4*48.75))+13.5^3(24.75+48.75)}{6*24.75} = 13455.1\,in^3$$

$$\frac{J_Y}{c'_Y} = \frac{b_1^2 d(b_1+4b_2)+d^3(b_1+b_2)}{6(b_1+2b_2)} =$$

$$\frac{24.75^2 * 13.5(24.75+(4*48.75))+13.5^3(24.75+48.75)}{6*(24.75+(2*48.75))} = 2724.0\,in^3$$

$$c_Y = \frac{b_1^2}{2(b_1+b_2)} = \frac{24.75^2}{2*(24.75+48.75)} = 4.17"$$

$$c'_x = \frac{b_1(b_1+2b_2)}{2(b_1+b_2)} = \frac{24.75*(24.75+(2*48.75))}{2*(48.75+24.75)} = 20.58"$$

a_Y = 6 + 6.75 - 4.17 = 8.58"

$M_{UYtransformed}$ = 1145.68 - 54.696 * 8.58 = 676 k-in (direction of moment shown in Figure 16)

The punching shear stress at point A in Figures 15 and 16 is calculated using equation 11:

$$v_U = \frac{54.696}{992.25} + \frac{0.483 * 1493}{16824.6} - \frac{0.322 * 676}{2724.0} = 0.055 + .043 - .080 = 0.018\ ksi$$

The punching shear stress at point B in Figures 15 and 16 is calculated using equation 8:

$$v_U = \frac{54.696}{992.25} + \frac{0.483 * 1493}{16824.6} + \frac{0.322 * 676}{13455.1} = 0.055 + .043 + .016 = 0.114\ ksi$$

The punching shear stress at point C in Figures 15 and 16 is calculated using equation 10:

$$v_U = \frac{54.696}{992.25} - \frac{0.483 * 1493}{8348.1} + \frac{0.322 * 676}{2724.0} = 0.055 - .086 + .016 = -0.018\ ksi$$

Point B has the largest absolute value of v_u, thus $v_{max} = 0.114$ ksi

6n. Comparison Of Punching Shear Stress Results

A comparison of the punching shear stress results is shown in the table below. The hand calculations using the SAFE method yield results the same as the SAFE computer model for all cases, that is for the interior column, the edge column with the edge parallel to the X-axis, the edge column with the edge parallel to the Y-axis and the corner column. The hand calculations using the PCA Publication method (Ref. 2) yield results the same as the SAFE computer model (and SAFE hand calculations) for all cases except the corner column. For the corner column if the I_{XY} term in the hand calculations using the SAFE method is set to zero, then this calculation yields the same result as the hand calculation using the PCA method.

Punching Shear Stress Results, ksi			
Column Type	**SAFE Computer Model**	**Hand Calculation Using SAFE Method**	**Hand Calculation Using PCA Publication Method**
Interior	0.126	0.126	0.126
Edge parallel to X-axis	0.144	0.144	0.144
Edge parallel to Y-axis	0.144	0.144	0.144
Corner	0.179	0.179	0.114
Corner with $I_{XY} = 0$	N. A.	0.114	N. A.

15.9.3 Design Checks for Steel and Concrete Beams

Unfortunately, most design check equations for steel structures are written in terms of "design strength ratios" which are a nonlinear function of the axial force in the member; therefore, the ratios cannot be calculated in each mode. A new approximate method, to replace the current state of the art approach of calculating strength ratios based on maximum peak values of member forces, is proposed by the author. This would involve first calculating the maximum axial force. The design ratios would then be evaluated mode by mode, assuming the maximum axial force reduction factor remains constant for all modes. The design ratio for the member would then be estimated by a double-sum modal combination method such as the CQC3 method. This approach would improve accuracy and still be conservative.

For concrete structures additional development work is required in order to develop a completely rational method for the use of maximum spectral forces in a design check equation because of the nonlinear behavior of concrete members. A time history analysis may be the only approach that will produce rational design forces.

15.9.4 Calculation of Shear Force in Bolts

With respect to the interesting problem of calculating the maximum shear force in a bolt, it is not correct to estimate the maximum shear force from a vector summation since the x and y shears do not obtain their peak values at the same time. A correct method of estimating the maximum shear in a bolt is to check the maximum bolt shear at several different angles about the bolt axis. This would be a tedious approach using hand calculations; however, if the approach is built into a post processor computer program, the computational time to calculate the maximum bolt force is trivial.

The same problem exists if principal stresses are to be calculated from a response spectrum analysis. One must check at several angles in order to estimate the maximum and minimum value of the stress at each point in the structure.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

USING THE ALING VERTICAL/HORIZONTAL COMMAND ONTHE EDIT MENU

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623 2971 • Fax: 5510.0573

SAFE Technical Note 2

Using the Align Vertical/Horizontal Command on the Edit Menu

The information in this document is consistent with SAFE version 6.20 and later

Initial Release Date: December 28, 1998
Revision Number: 1
Revision Date: January 11, 1999

Table Of Contents

Item	Page
1. Background	1
2. How to Implement the Align Vertical/Horizontal Command	1
3. How the Align Vertical/Horizontal Command Works	1
4. Tips on Using the Align Vertical/Horizontal Command	3
5. Final Comments	4

1. Background

SAFE creates a rectangular mesh for analysis. There is an X and Y direction mesh line passing through each point in the model. The points these mesh lines pass through include point objects, the end points of line objects and the corner points of area objects. As you can imagine, if your points do not align somewhat in an X-Y grid system you can very easily have a large number of closely spaced grid lines. This may not be desirable since it creates a bad aspect ratio for the shell elements, greatly increase the running time for your model and also increase the display time for results.

The Align Vertical/Horizontal command was added to the Edit menu in SAFE version 6.16. The purpose of this command is to assist you in aligning points, lines and area objects in your model.

2. How to Implement the Align Vertical/Horizontal Command

To use the Align Vertical/Horizontal command do the following:

1. Select the point, line and area objects that you want to align.

2. On the **Edit** menu click **Align Vertical/Horizontal** to display the Align Selected Lines/Edges/Points dialog box.

3. In this dialog box either type in the maximum move allowed or accept the default and click the OK button. The value you type in is a distance in the current units. It is the maximum distance any point will move to align with another point.

SAFE will automatically align all of the selected objects as described below. You may want to run through this process three times to cover the structual layer, X-strip layer and Y-strip layer.

3. How the Align Vertical/Horizontal Command Works

When you implement the Align Vertical/Horizontal command SAFE goes through the following sequence of steps in the order specified:

1. SAFE changes the selection from point line and area objects to a group of points. This group of points consists of all of the selected point objects, the end points of all of the selected line objects and the corner points of all of the selected area objects.

2. SAFE steps through the grid lines one-by-one. If one of the selected points is within the specified maximum tolerance of the grid line, then that point is moved onto the grid line. The program first steps through all of the horizontal

grid lines from bottom to top and then through all the vertical grid lines from left to right.

Note that once a point is moved horizontally it will not be moved horizontally again because of another criterion further down the list. Similarly, once a point is moved vertically it will not be moved vertically again because of another criterion further down the list. However, a point that has been moved horizontally can still be moved vertically and vice versa.

3. SAFE goes over all of the column supports in the order in which they were input into the model. If one of the selected points is within the specified maximum tolerance of a vertical line passing through the column support then the point is moved to that vertical line. Similarly, if one of the selected points is within the specified maximum tolerance of a horizontal line passing through the column support then the point is moved to that horizontal line.

   Note that if the point is within the maximum specified tolerance in both the vertical and horizontal directions then it will move right on top of the column support.

4. SAFE steps through each of the unselected points on the structural layer in the order that they were input into the model. If one of the selected points is within the specified maximum tolerance of a vertical line passing through the point currently being considered, then the other point is moved to that vertical line. Similarly, if one of the selected points is within the specified maximum tolerance of a horizontal line passing through the point currently being considered, then the other point is moved to that horizontal line

5. SAFE steps through each of the selected points that can still be moved in the order in which they were input into the model. If one of the other selected points is within the specified maximum tolerance of a vertical line passing through the point currently being considered, then the other point is moved to that vertical line. Similarly, if one of the other selected points is within the specified maximum tolerance of a horizontal line passing through the point currently being considered, then the other point is moved to that horizontal line.

   Note that if a point is moved in one direction to align with the considered point, then the considered point is assumed to have been moved once already in that direction and will not be moved in that direction again.

   Also note that at the end of this sequence there may be some selected points which were not moved at all.

4. Tips on Using the Align Vertical/Horizontal Command

1. From the moment you start your model you should try to align objects as much as possible/practical.

2. We recommend that you align points first for the structural layer, then for the X-strip layer and finally for the Y-strip layer.

3. The best way for you to control where a point moves to is to create a temporary grid line at the location where you want the point to move.

4. You can use the Align Vertical/Horizontal command on local portions of your model or globally on the entire model. Our intent is for you to use it on local areas of your model. If you use the command globally then you should typically use a small value for the maximum move allowed and you should check the model carefully to make sure you did not get any unanticipated aligning.

5. The Align Vertical/Horizontal command may cause some objects to be deleted. If the two end points of a line object move on top of each other then the line object will be deleted. If an area object ends up with no area then the area object will be deleted.

   If two point objects end up one on top of the other then the point object that was moved merges into the other point object. The loads on the moved point object are added to the other point object. If the moved point object has support properties and the other point object does not, then the support properties are transferred to the other point object. If both point objects have support properties then the support properties for the moved point are lost and the support properties for the other point are maintained.

6. The Undo command on the Edit menu does work for undoing the effects of the Align Vertical/Horizontal command.

7. We recommend that when you are learning how the Align Vertical/Horizontal command works you try it on a small test model, not on a large model that you have spent many hours developing. Once you are comfortable with the command on the test model then you can move on to using it in local areas of your real model.

8. It is probably a good idea to make a backup copy of your model right before you use the Align Vertical/Horizontal command in case you later (perhaps the next day) decide you want to go back to the old model.

5. Final Comments

We will be adding additional capability to the Align Vertical/Horizontal command in a future release. Specifically, we will add a feature that allows you to align to any specified line that can be drawn in any orientation. We will also add the capability to align vertically and horizontally separately.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

GEOMETRIC STIFFNESS AND P – DELTA EFFECTS

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

GEOMETRIC STIFFNESS AND P- DELTA EFFECTS

P-Delta Effects, Due To Dead Load, Can Be Considered Without Iteration For Both Static And Dynamic Analysis

11.1. DEFINITION OF GEOMETRIC STIFFNESS

We are all aware that a cable, when subjected to a large tension force, has an increased lateral stiffness. If a long rod is subjected to a large compressive force, and is on the verge of buckling, we know that the lateral stiffness of the rod has been reduced significantly and a small lateral load may cause the rod to buckle. This general type of behavior is caused by a change in the "geometric stiffness" of the structure. It is apparent that this stiffness is a function of the load in the structural member and can be either positive or negative.

The fundamental equations for the geometric stiffness for a rod or a cable are very simple to derive. Consider the horizontal cable, shown in Figure 11.1, of length L with an initial tension T. If the cable is subjected to lateral displacements, v_i and v_j , at both ends, as shown, then additional forces, F_i and F_j , must be developed for the cable element to be in equilibrium in its displaced position. Note that we have assumed all forces and displacements are positive in the up direction. We have also made the assumption that the displacements are small and do not change the tension in the cable.

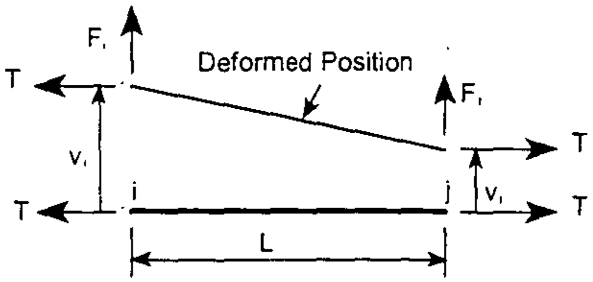


Figure 11.1. Forces Acting on a Cable Element

Taking moments about point j in the deformed position, the following equilibrium equation can be written:

$$F_i = \frac{T}{L}(v_i - v_j) \tag{11.1}$$

And from vertical equilibrium the following equation is apparent:

$$F_j = -F_i \tag{11.2}$$

Combining Equation 11.1 and 11.2 the lateral forces can be expressed in terms of the lateral displacements by the following matrix equation:

$$\begin{bmatrix} F_i \\ F_j \end{bmatrix} = \frac{T}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} v_i \\ v_j \end{bmatrix} \quad \text{or symbolically,} \quad \mathbf{F_g} = \mathbf{k_g v} \tag{11.3}$$

Note that the 2 by 2 geometric stiffness, $\mathbf{k}_g$, matrix is not a function of the mechanical properties of the cable and is only a function of the element's length and the force in the element. Hence, the term "geometric" or "stress" stiffness matrix is introduced in order that the matrix has a different name than the "mechanical" stiffness matrix which is based on the physical properties of the element. The geometric stiffness exists in all structures; however, it only becomes important if it is large compared to the mechanical stiffness of the structural system.

In the case of a beam element with bending properties in which the deformed shape is assumed to be a cubic function due to the rotations ϕ_i and ϕ_j at the ends, additional moments M_i and M_j are developed. From Reference [1] the force-displacement relationship is given by the following equation:

$$\begin{bmatrix} F_i \\ M_i \\ F_j \\ M_j \end{bmatrix} = \frac{T}{30L}\begin{bmatrix} 36 & 3L & -36 & 3L \\ 3L & 4L^2 & -3L & -L^2 \\ -36 & -3L & 36 & -3L \\ 3L & -L^2 & -3L & 4L^2 \end{bmatrix}\begin{bmatrix} v_i \\ \phi_i \\ v_j \\ \phi_j \end{bmatrix} \quad \text{or, } \mathbf{F}_G = \mathbf{k}_G\mathbf{v} \tag{11.4}$$

The well-known elastic force deformation relationship, for a prismatic beam without shearing deformations, is

$$\begin{bmatrix} F_i \\ M_i \\ F_j \\ M_j \end{bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & -2L^2 \\ -12 & -6L & 12 & -6L \\ -6L & -2L^2 & -6L & 4L^2 \end{bmatrix}\begin{bmatrix} v_i \\ \phi_i \\ v_j \\ \phi_j \end{bmatrix} \quad \text{or, } \mathbf{F}_E = \mathbf{k}_E\mathbf{v} \tag{11.5}$$

Therefore, the total forces acting on the beam element will be

$$\mathbf{F}_T = \mathbf{F}_E + \mathbf{F}_G = [\mathbf{k}_E + \mathbf{k}_G]\mathbf{v} = \mathbf{k}_T\mathbf{v} \tag{11.6}$$

Hence, if the large axial force in the member remains constant, it is only necessary to form the total stiffness matrix, $\mathbf{k}_T$, in order to account for this stress stiffening or softening effect.

11.2. APPROXIMATE BUCKLING ANALYSIS

In the case when the axial compressive force is large, $T = -P$, the total stiffness matrix of the beam can become singular. In order to illustrate this instability, consider the beam shown in Figure 11.2 with the displacements at point j set to zero.

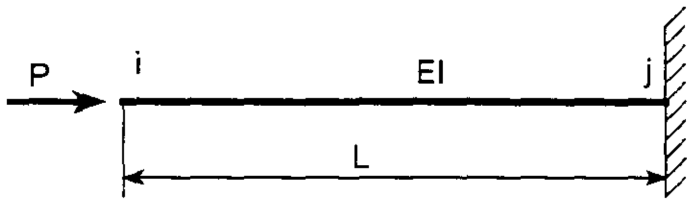


Figure 11.2 Cantilever Beam Subjected to Buckling Load

From Equation (11.6) the equilibrium equations for the beam, shown in Figure 11.2, are in matrix form

$$\begin{bmatrix} 12+36\lambda & 6L+3L\lambda \\ 6L+3L\lambda & 4L^2+4L^2\lambda \end{bmatrix} \begin{bmatrix} v_i \\ \phi_i \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix} \tag{11.7}$$

Where $\lambda = -\dfrac{PL^2}{30EI}$. This eigenvalue problem can be solved for the lowest root which is

$$\lambda_1 = -0.0858 \qquad \text{or} \qquad P_{cr} = 2.57\frac{EI}{L^2} \tag{11.8}$$

The well-known exact Euler buckling load for the cantilever beam is given by

$$P_{cr} = \frac{\pi^2 EI}{4L^2} = 2.47\frac{EI}{L^2} \tag{11.9}$$

Therefore, the approximate solution Equation (11.8), which is based on a cubic shape, is within five percent of the exact solution.

If the straight line approximation is used, given by Equation (11.3), an approximate buckling load of $3.0\dfrac{EI}{L^2}$ is obtained. This is still a reasonable approximation.

11.3. P-DELTA ANALYSIS OF BUILDINGS

The use of the geometric stiffness matrix is a general approach to include secondary effects in the static and dynamic analysis of all types of structural systems. However, in Civil Structural Engineering it is commonly referred to as P-Delta Analysis that is based on a more physical approach. For example, in building analysis the lateral movement of a story mass to a deformed position generates second-order overturning moments. This second-order behavior has been termed the P-Delta effect since the additional overturning moments on the building are equal to the sum of story weights "P" times the lateral displacements "Delta".

Many techniques have been proposed for evaluating this second-order behavior. Rutenberg [2] summarized the publications on this topic and presents a simplified method to include these second-order effects. Some methods consider the problem as one of geometric non-linearity and propose iterative solution techniques that can be numerically inefficient. Also, these iterative methods are not appropriate for dynamic analysis where the P-Delta effect causes lengthening of the periods of vibration. The equations presented in this section are not new. However, the simple approach used in their derivation should add physical insight to the understanding of P-Delta behavior in buildings [3].

The P-Delta problem can be linearized and the solution to the problem obtained ***directly*** and ***exactly***, without iteration, in building type structures where the weight of the structure is constant during lateral motions and the overall structural displacements can be assumed to be small compared to the structural dimensions. Furthermore, the additional numerical effort required is negligible.

The method does not require iteration since the total axial force at a story level is equal to the weight of the building above that level and does not change during the application of lateral loads. Therefore, the sum of the column of geometric stiffness terms associated with the lateral loads cancels and only the axial forces due to the weight of the structure need be included in the evaluation of the geometric stiffness terms for the complete building.

The effects of P-Delta are implemented in the basic analytical formulation that causes the effects to be consistently included in both static and dynamic analyses. The structural displacements and the mode shapes and frequencies thus obtained indicate the structural softening automatically. Member forces satisfy both static and dynamic equilibrium and reflect the additional P-Delta moments consistent with the calculated displacements directly.

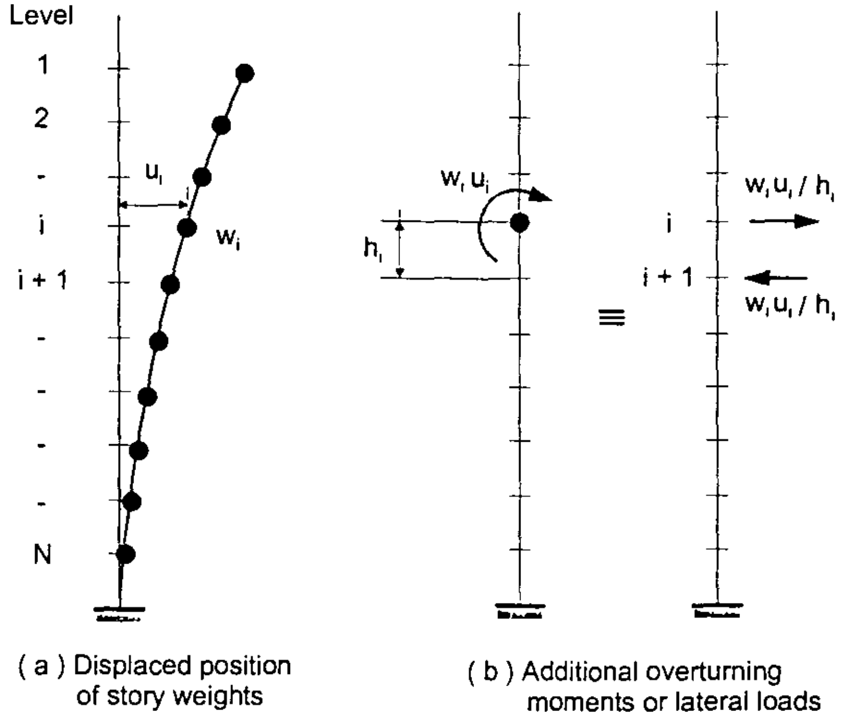


Figure 11.3 Overturning Loads Due to Translation of Story Weights

The vertical "cantilever type" structure shown in Figure 11.3 (a) is considered to illustrate the basic problem. Under lateral displacements let us consider the additional overturning moments due to one mass, or story weight, at level "i". The total overturning effects will be the sum of all story weight contributions. Figure 11.3 (b) indicates statically equivalent force systems which produce the same overturning moments. Or, in terms of matrix notation

$$\begin{bmatrix} f_i \\ f_{i+1} \end{bmatrix} = \frac{w_i}{h_i} \begin{bmatrix} 1.0 \\ -1.0 \end{bmatrix} \begin{bmatrix} u_i \end{bmatrix} \tag{11.10}$$

The lateral forces shown in Figure 11.3 (b) can be evaluated for all stories and added to the external loads on the structure. The resulting lateral equilibrium equation of the structure is

$$\mathbf{K}\mathbf{u} = \mathbf{F} + \mathbf{L}\mathbf{u} \tag{11.11}$$

where $\mathbf{K}$ is the lateral stiffness matrix with respect to the lateral story displacements $\mathbf{u}$. The vector $\mathbf{F}$ represents the known lateral loads and $\mathbf{L}$ is a matrix that contains w_i / h_i factors. Equation (11.11) can be rewritten in the form

$$\mathbf{K}^*\mathbf{u} = \mathbf{F} \tag{11.12}$$

where $\mathbf{K}^* = \mathbf{K} - \mathbf{L}$

Equation (11.12) can be solved directly for the lateral displacements. If internal member forces are evaluated from these displacements, consistent with the linear theory used, it will be found that equilibrium with respect to the deformed position has been obtained. One minor problem exists with the solution of Equation (11.12) - the matrix $\mathbf{K}^*$ is not symmetric. However, it can be made symmetric by replacing the lateral loads shown in Figure 11.3 (b) with another statically equivalent load system.

From simple statics the total contribution to overturning, due to the relative story displacement " u_i - u_{i+1} ", can be written as

$$\begin{bmatrix} f_i \\ f_{i+1} \end{bmatrix} = \frac{W_i}{h_i} \begin{bmatrix} 1.0 & -1.0 \\ -1.0 & 1.0 \end{bmatrix} \begin{bmatrix} u_i \\ u_{i+1} \end{bmatrix} \tag{11.13}$$

where W_i is the total dead load weight above story "i". The $\mathbf{L}$ matrix is now symmetrical and no special non-symmetric equation solver is required.

It is of significant interest to note that Equation (11.13) is the exact form of the "geometric stiffness", Equation (11.3), for a column including axial force effects only. Therefore, the physical development given here is completely equivalent to the more theoretical approach normally used to formulate the incremental stiffness in nonlinear structural analysis.

The equilibrium of a complete building can be formulated in terms of the lateral displacement of the floor level. Then, one can evaluate the contribution to the total geometric stiffness for each column at a particular story level in which the effects of the external lateral loads **F** are included in the evaluation of the axial forces in all columns. If this approach is used, the total geometric stiffness at the lateral equilibrium level is identical to Equation (11.13) since the lateral axial forces **F** do not produce a net increase in the total of all axial forces which exist in the columns at any level. Such a refined analysis must be iterative in nature; however, it does not produce more exact results.

It is clear that the beam-column stiffness effects, as defined by Equation (11.4), have been neglected. The errors associated with these cubic shape effects can be estimated at the time member forces are calculated. However, the method presented here does include the overall large displacement side-sway behavior of the complete structure that is associated with the global stability of the building.

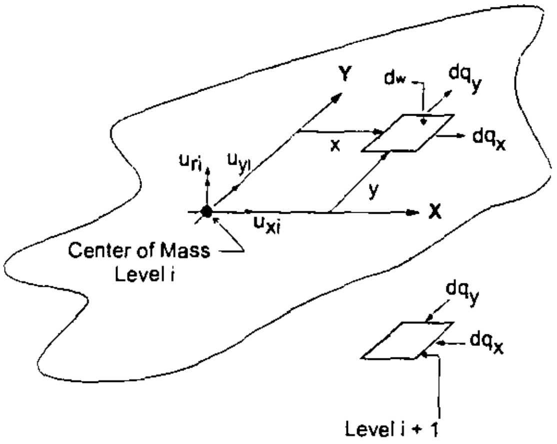


Figure 11.4 Mass Distribution at Typical Floor Level

11.4. EQUATIONS FOR THREE-DIMENSIONAL BUILDINGS

Equation (11.13) can be applied directly in both directions for buildings in which the centroids are the same for all story levels. However, for the more general building the equations for the story couples are more complicated. A general three-dimensional building system is shown schematically in Figure 11.4. It is assumed that the three dimensional building stiffness of the system has been formulated with respect to the two lateral displacements, u_{xi} , u_{yi}, and rotation, u_{ri}, at the center of mass at each story level. In addition to the overturning forces given by Equation 11.13, secondary forces exist due to the distribution of the story mass over a finite floor size.

The first step, prior to the development of the 6 by 6 geometric stiffness matrix for each story, is to calculate the location of the center of mass and the rotational moment of inertia for all story levels. For a typical story "i" it is then necessary to calculate the total weight and centroid of the structure above that level. Due to the relative displacements between story "i" and story "i + 1", from Equation 11.13, forces must be developed to maintain equilibrium. These forces and displacements must then be transformed to the center of mass at both level "i" and "i + 1".

11.5. THE MAGNITUDE OF P-DELTA EFFECTS

The comparison of the results of two analyses with and without P-Delta will illustrate the magnitude of the P-Delta effects. A well-designed building usually has well-conditioned level by level stiffness/weight ratios. For such structures, P-Delta effects are usually not very significant. The changes in displacements and member forces are less than 10%.

However, if the weight of the structure is high in proportion to the lateral stiffness of the structure, the contributions from the P-Delta effects are highly amplified and, under certain circumstances, can change the displacements and member forces by 25 percent or more. Excessive P-Delta effects will eventually introduce singularities into the solution, indicating physical structure instability. Such behavior is clearly indicative of a poorly designed structure that is in need of additional stiffness.

An analysis of a 41-story steel building was conducted with and without P-Delta effects. The basic construction was braced frame and welded steel shear wall. The building was constructed in a region where the principal lateral loading is wind. The results are summarized in Table 11.1.

Table 11.1. P-Delta Effects on Typical Building

	Without P-Delta	With P-Delta
First Mode Period (seconds)	5.33	5.52
Second Mode Period (seconds)	4.21	4.30
Third Mode Period (seconds)	4.01	4.10
Fourth Mode Period (seconds)	1.71	1.75
Wind Displacement (inches)	7.99	8.33

Since the building is relatively stiff, the P-Delta effects are minimal. Also, it is apparent that P-Delta effects are less important for higher frequencies.

11.6. P-DELTA ANALYSIS WITHOUT COMPUTER PROGRAM MODIFICATION

Many engineers are using general purpose, structural analysis programs for buildings that cannot be easily modified to include the equations presented here. Equation 11.4 presents the form of the lateral force-displacement equations for story "i". We note that the form of this 2 x 2 geometric stiffness matrix is the same as the stiffness matrix for a prismatic column that has zero rotations at the top and bottom. Therefore, it is possible to add "dummy columns" between story levels of the building and assign appropriate properties in order to achieve the same effects as the use of geometric stiffness [2]. The force-displacement equations of the "dummy column" are

$$\begin{bmatrix} f_i \\ f_{i+1} \end{bmatrix} = \frac{12EI}{h_i^3} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} u_i \\ u_{i+1} \end{bmatrix} \tag{11.14}$$

Therefore, if the moment of inertia of the column is selected as

$$I = \frac{W_i h_i^2}{12E} \tag{11.15}$$

The dummy column will have the same negative stiffness values as the linear geometric stiffness

11.7. EFFECTIVE LENGTH - K FACTORS

The solution procedure for the P-Delta effects described in this chapter has been implemented and verified in the ETABS program. The application of the method of analysis presented in this chapter should lead to the elimination of the column effective length (K-) factors, since the P-Delta effects automatically produce the required design moment amplifications. Also, the K-factors are approximate, complicated, and time-consuming to calculate. Building codes for concrete [4] and steel [5] now allow explicit accounting of P-Delta effects as an alternative to the more involved and approximate methods of calculating moment magnification factors for most column designs.

11.8. GENERAL FORMULATION OF GEOMETRY STIFFNESS

It is relatively simple to develop the geometric stiffness matrix for any type of displacement based finite element [1]. It is only necessary to add to the linear strain-displacement equations, Equation (2.3), the higher order nonlinear terms. These large strain equations, in a local x-y-z reference system, are

$$
\begin{aligned}
\varepsilon_x &= \frac{\partial u_x}{\partial x} + \frac{1}{2}\overline{\mathbf{u}}_{,x}^T \overline{\mathbf{u}}_{,x} \\
\varepsilon_y &= \frac{\partial u_y}{\partial y} + \frac{1}{2}\overline{\mathbf{u}}_{,y}^T \overline{\mathbf{u}}_{,y} \\
\varepsilon_z &= \frac{\partial u_z}{\partial z} + \frac{1}{2}\overline{\mathbf{u}}_{,z}^T \overline{\mathbf{u}}_{,z} \\
\gamma_{xy} &= \frac{\partial u_x}{\partial y} + \frac{\partial u_y}{\partial x} + \frac{1}{2}\overline{\mathbf{u}}_{,x}^T \overline{\mathbf{u}}_{,y} + \frac{1}{2}\overline{\mathbf{u}}_{,y}^T \overline{\mathbf{u}}_{,x} \\
\gamma_{xz} &= \frac{\partial u_x}{\partial z} + \frac{\partial u_z}{\partial x} + \frac{1}{2}\overline{\mathbf{u}}_{,x}^T \overline{\mathbf{u}}_{,z} + \frac{1}{2}\overline{\mathbf{u}}_{,z}^T \overline{\mathbf{u}}_{,x} \\
\gamma_{yz} &= \frac{\partial u_y}{\partial z} + \frac{\partial u_z}{\partial x} + \frac{1}{2}\overline{\mathbf{u}}_{,x}^T \overline{\mathbf{u}}_{,y} + \frac{1}{2}\overline{\mathbf{u}}_{,y}^T \overline{\mathbf{u}}_{,x}
\end{aligned}
\tag{11.16}
$$

The nonlinear terms are the product of matrices that are defined as

$$\bar{\mathbf{u}}_{,x} = \begin{bmatrix} u_{x,x} \\ u_{y,x} \\ u_{z,x} \end{bmatrix}, \quad \bar{\mathbf{u}}_{,y} = \begin{bmatrix} u_{x,y} \\ u_{y,y} \\ u_{z,y} \end{bmatrix}, \quad \bar{\mathbf{u}}_{,z} = \begin{bmatrix} u_{x,z} \\ u_{y,z} \\ u_{z,z} \end{bmatrix} \tag{11.17}$$

Equation (11.16) can be expressed in terms of the following sum of linear and nonlinear components:

$$\mathbf{d} = \mathbf{d}_L + \mathbf{d}_N \tag{11.18}$$

These strain-displacement equations, written in terms of engineering strains and in matrix notation, are identical to the classical Green-Lagrange strains. This is often referred to as the total Lagrangian approach in which the strains are computed with respect to the original reference system and the large rigid-body rotation is exact.

If the initial stresses are large, the potential energy of the structure must be modified by the addition of the following term:

The initial stresses are defined as

$$\mathbf{s}_0^T = \begin{bmatrix} \sigma_{xx} & \sigma_{yy} & \sigma_{zz} & \sigma_{xy} & \sigma_{xz} & \sigma_{yz} \end{bmatrix}_0 \tag{11.19}$$

Equation (11.19) can be written in the following form:

$$\Omega_\sigma = \frac{1}{2}\int \begin{bmatrix} \bar{\mathbf{u}}_{,x}^T & \bar{\mathbf{u}}_{,y}^T & \bar{\mathbf{u}}_{,z}^T \end{bmatrix} \begin{bmatrix} \mathbf{s}_{xx} & \mathbf{s}_{xy} & \mathbf{s}_{xz} \\ \mathbf{s}_{yx} & \mathbf{s}_{yy} & \mathbf{s}_{yz} \\ \mathbf{s}_{zx} & \mathbf{s}_{zy} & \mathbf{s}_{zz} \end{bmatrix} \begin{bmatrix} \bar{\mathbf{u}}_{,x} \\ \bar{\mathbf{u}}_{,y} \\ \bar{\mathbf{u}}_{,z} \end{bmatrix} dV = \frac{1}{2}\int \mathbf{g}^T \mathbf{S}\,\mathbf{g}\,dV \tag{11.20}$$

The 3 by 3 initial stress matrices are of the following form:

$$\mathbf{s}_{ij} = \begin{bmatrix} \sigma_{ij} & 0 & 0 \\ & \sigma_{ij} & 0 \\ 0 & 0 & \sigma_{ij} \end{bmatrix}_0 \tag{11.21}$$

Using the same shape functions as used to form the element stiffness matrix, the derivatives of the displacements can be written as

$$\mathbf{g} = \mathbf{G}\mathbf{u} \tag{11.22}$$

Therefore, the geometric stiffness for any element can be calculated from

$$\mathbf{k}_g = \int \mathbf{G}^T \mathbf{S}\mathbf{G} dV \tag{11.24}$$

For most finite elements the geometric stiffness is evaluated by numerical integration.

11.9. SUMMARY

The SAP2000 program has the option to add a three-dimensional geometric stiffness matrix to each frame element. Therefore, guyed towers, cable stay and suspension bridges can be modeled if the tension in the cable is not modified by the application of the load. If the initial axial forces in the elements are significantly changed by the addition of loads, iteration may be required. However, in the case of dynamic analysis the evaluation of the eigen or LDR vectors must be based on one set of axial forces.

Most traditional methods for incorporating P-Delta effects in analysis of buildings are based on iterative techniques. These techniques are time-consuming and are, in general, used for static analysis only. For building structures, the mass, which causes the P-Delta effect, is constant irrespective of the lateral loads and displacements. This information is used to linearize the P-Delta effect for buildings and solve the problem "exactly", satisfying equilibrium in the deformed position without iterations. An algorithm is developed that incorporates P-Delta effects into the basic formulation of the structural stiffness matrix as a geometric stiffness correction. This procedure can be used for static and dynamic analysis and will account for the lengthening of the periods and changes in mode shapes due to P-Delta effects.

A well designed building should not have significant P-Delta effects. Analyses ***with and without the P-Delta effects*** will yield the magnitude of the P-Delta effects separately. If these lateral displacements differ by more than 5%, ***for the same lateral load***, the basic design may be too flexible and a redesign should be considered.

The current SEAOC Blue Book states "the drift ratio of $0.02/R_W$ serves to define the threshold of deformation beyond which there may be significant P-Delta effects". Clearly, if one includes P-Delta effects in all analyses one can disregard this statement. If the loads acting on the structure have been reduced by a ductility factor R_W, however, the P-Delta effects should be amplified by R_W in order to reflect ultimate load behavior. This can be automatically included in a computer program by using a multiplication factor for the geometric stiffness terms.

It is possible to calculate geometric stiffness matrices for all types of finite elements. The same shape functions used in the development of elastic stiffness matrices are used in the calculation of the geometric stiffness matrix.

11.10. REFERENCES

1. R. D. Cook., D. S. Malkus and M. E. Plesha, *Concepts and Applications of Finite Element Analysis*, Third Edition, John Wiley & Sons, Inc, ISBN 0-471-84788-7, 1989.

2. A. Rutenberg, "Simplified P-Delta Analysis for Asymmetric Structures," *ASCE Journal of the Structural Division*, Vol. 108, No. 9, Sept. 1982.

3. E. L. Wilson and A. Habibullah, "Static and Dynamic Analysis of Multi-Story Buildings Including P-Delta Effects," *Earthquake Spectra,* Earthquake Engineering Research Institute, Vol. 3, No.3, May 1987.

4. *Building Code Requirements for Reinforced Concrete (ACI 318-95) and Commentary (ACI 318R-95)*, American Concrete Institute, Farmington Hills, Michigan, 1995.

5. *Load and Resistance Factor Design Specification for Structural Steel Buildings*, American Institute of Steel Construction, Inc., Chicago, Illinois, December, 1993.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

DYNAMIC ANALYSIS

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

CURSOS ABIERTOS

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

DYNAMIC ANALYSIS

Force Equilibrium Is Fundamental In The Dynamic Analysis Of Structures

12.1 INTRODUCTION

All real physical structures, when subjected to loads or displacements, behave dynamically. The additional inertia forces, ***from Newton's second law***, are equal to the mass times the acceleration. If the loads or displacements are applied very slowly then the inertia forces can be neglected and a static load analysis can be justified. Hence, dynamic analysis is a simple extension of static analysis.

In addition, all real structures potentially have an infinite number of displacements. Therefore, the most critical phase of a structural analysis is to create a computer model, with a finite number of massless members and a finite number of node (joint) displacements, that will simulate the behavior of the real structure. The mass of a structural system, which can be accurately estimated, is lumped at the nodes. Also, for linear elastic structures the stiffness properties of the members, with the aid of experimental data, can be approximated with a high degree of confidence. However, the dynamic loading, energy dissipation properties and boundary (foundation) conditions for many structures are difficult to estimate. This is always true for the cases of seismic input or wind loads.

To reduce the errors that may be caused by the approximations summarized in the previous paragraph, it is necessary to conduct many different dynamic analyses using different computer models, loading and boundary conditions. It is not unrealistic to conduct 20 or more computer runs to design a new structure or to investigate retrofit options for an existing structure.

Because of the large number of computer runs required for a typical dynamic analysis, it is very important that accurate and numerically efficient methods be used within computer programs. Some of these methods have been developed by the author and are relatively new. Therefore, one of the purposes of this book is to summarize these numerical algorithms, their advantages and limitations.

12.2 DYNAMIC EQUILIBRIUM

The force equilibrium of a multi-degree-of-freedom lumped mass system as a function of time can be expressed by the following relationship:

$$\mathbf{F}(t)_I + \mathbf{F}(t)_D + \mathbf{F}(t)_S = \mathbf{F}(t) \tag{12.1}$$

in which the force vectors at time t are

$\mathbf{F}(t)_I$ is a vector of inertia forces acting on the node masses

$\mathbf{F}(t)_D$ is a vector of viscous damping, or energy dissipation, forces

$\mathbf{F}(t)_S$ is a vector of internal forces carried by the structure

$\mathbf{F}(t)$ is a vector of externally applied loads

Equation (12.1) is based on physical laws and is valid for both linear and nonlinear systems if equilibrium is formulated with respect to the deformed geometry of the structure.

For many structural systems, the approximation of linear structural behavior is made in order to convert the physical equilibrium statement, Equation (12.1), to the following set of second-order, linear, differential equations:

$$\mathbf{M}\,\ddot{\mathbf{u}}(t)_a + \mathbf{C}\,\dot{\mathbf{u}}(t)_a + \mathbf{K}\,\mathbf{u}(t)_a = \mathbf{F}(t) \tag{12.2}$$

in which $\mathbf{M}$ is the mass matrix (lumped or consistent), $\mathbf{C}$ is a viscous damping matrix (which is normally selected to approximate energy dissipation in the real structure) and $\mathbf{K}$ is the static stiffness matrix for the system of structural elements. The time-dependent vectors $\mathbf{u}(t)_a$, $\dot{\mathbf{u}}(t)_a$ and $\ddot{\mathbf{u}}(t)_a$ are the absolute node displacements, velocities and accelerations, respectively.

Many books on structural dynamics present several different methods of applied mathematics to obtain the exact solution of Equation (12.2). Within the past several years, however, with the general availability of inexpensive, high-speed personal computers (see Appendix Z) the exact solution of Equation (12.2) can be obtained without the use of complex mathematical techniques. Therefore, the modern structural engineer, with a physical understanding of dynamic equilibrium and energy dissipation, can perform dynamic analysis of complex structural systems. A strong engineering mathematical background is desirable; however, in my opinion, it is no longer mandatory.

For seismic loading, the external loading $\mathbf{F}(t)$ is equal to zero. The basic seismic motions are the three components of free-field ground displacements $u(t)_{ig}$ that are known at some point below the foundation level of the structure. Therefore, we can write Equation (12.2) in terms of the displacements $\mathbf{u}(t)$, velocities $\dot{\mathbf{u}}(t)$ and accelerations $\ddot{\mathbf{u}}(t)$ that are relative to the three components of free-field ground displacements.

Therefore, the absolute displacements, velocities and accelerations can be eliminated from Equation (12.2) by writing the following simple equations:

$$\begin{aligned}
\mathbf{u}(t)_a &= \mathbf{u}(t) + \mathbf{I}_x\, u(t)_{xg} + \mathbf{I}_y\, u(t)_{yg} + \mathbf{I}_z\, u(t)_{zg} \\
\dot{\mathbf{u}}(t)_a &= \dot{\mathbf{u}}(t) + \mathbf{I}_x\, \dot{u}(t)_{xg} + \mathbf{I}_y\, \dot{u}(t)_{yg} + \mathbf{I}_z\, \dot{u}(t)_{zg} \\
\ddot{\mathbf{u}}(t)_a &= \ddot{\mathbf{u}}(t) + \mathbf{I}_x\, \ddot{u}(t)_{xg} + \mathbf{I}_y\, \ddot{u}(t)_{yg} + \mathbf{I}_z\, \ddot{u}(t)_{zg}
\end{aligned} \tag{12.3}$$

where $\mathbf{I}_i$ is a vector with ones in the "i" directional degrees-of-freedom and zero in all other positions. The substitution of Equation (12.3) into Equation (12.2) allows the node point equilibrium equations to be rewritten as

$$\mathbf{M}\ddot{\mathbf{u}}(t) + \mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t) = -\mathbf{M}_x\ddot{u}(t)_{xg} - \mathbf{M}_y\ddot{u}(t)_{yg} - \mathbf{M}_z\ddot{u}(t)_{zg} \tag{12.4}$$

where $\mathbf{M}_i = \mathbf{M}\mathbf{I}_i$.

The simplified form of Equation (12.4) is possible since the rigid body velocities and displacements associated with the base motions cause no additional damping or structural forces to be developed.

It is important for engineers to realize that the displacements, which are normally printed by a computer program, are relative displacements and that the fundamental loading on the structure is foundation displacements and not externally applied loads at the joints of the structure. For example, the static pushover analysis of a structure is a poor approximation of the dynamic behavior of a three dimensional structure subjected to complex time-dependent base motions. Also, one must calculate absolute displacements to properly evaluate base isolation systems.

There are several different classical methods that can be used for the solution of Equation (12.4). Each method has advantages and disadvantages that depend on the type of structure and loading. To provide a general background for the various topics presented in this book, the different numerical solution methods are summarized below.

12.3 STEP BY STEP SOLUTION METHOD

The most general solution method for dynamic analysis is an incremental method in which the equilibrium equations are solved at times Δt, $2\Delta t$, $3\Delta t$, etc. There are a large number of different incremental solution methods. In general, they involve a solution of the complete set of equilibrium equations at each time increment. In the case of nonlinear analysis, it may be necessary to reform the stiffness matrix for the complete structural system for each time step. Also, iteration may be required within each time increment to satisfy equilibrium. As a result of the large computational requirements it can take a significant amount of time to solve structural systems with just a few hundred degrees-of-freedom.

In addition, artificial or numerical damping must be added to most incremental solution methods in order to obtain stable solutions. For this reason, engineers must be very careful in the interpretation of the results. For some nonlinear structures, subjected to seismic motions, incremental solution methods are necessary.

For very large structural systems, a combination of mode superposition and incremental methods has been found to be efficient for systems with a small number of nonlinear members. This method has been incorporated in the new versions of SAP and ETABS and will be presented in detail later in this book.

12.4 MODE SUPERPOSITION METHOD

The most common and effective approach for seismic analysis of linear structural systems is the mode superposition method. This method, after a set of orthogonal vectors are evaluated, reduces the large set of global equilibrium equations to a relatively small number of uncoupled second order differential equations. The numerical solution of these equations involves greatly reduced computational time.

It has been shown that seismic motions excite only the lower frequencies of the structure. Typically, earthquake ground accelerations are recorded at increments of 200 points per second. Therefore, the basic loading data does not contain information over 50 cycles per second. Hence, neglecting the higher frequencies and mode shapes of the system normally does not introduce errors.

12.5 RESPONSE SPECTRA ANALYSIS

The basic mode superposition method, which is restricted to linearly elastic analysis, produces the complete time history response of joint displacements and member forces due to a specific ground motion loading [1,2]. There are two major disadvantages of using this approach. First, the method produces a large amount of output information that can require an enormous amount of computational effort to conduct all possible design checks as a function of time. Second, the analysis must be repeated for several different earthquake motions in order to assure that all the significant modes are excited, since a response spectrum for one earthquake, in a specified direction, is not a smooth function.

There are significant computational advantages in using the response spectra method of seismic analysis for prediction of displacements and member forces in structural systems. The method involves the calculation of only the maximum values of the displacements and member forces in each mode using smooth design spectra that are the average of several earthquake motions. In this book, we will recommend the CQC method to combine these maximum modal response values to obtain the most probable peak value of displacement or force. In addition, it will be shown that the

SRSS and CQC3 methods of combining results from orthogonal earthquake motions will allow one dynamic analysis to produce design forces for all members in the structure.

12.6 SOLUTION IN THE FREQUENCY DOMAIN

The basic approach, used to solve the dynamic equilibrium equations in the frequency domain, is to expand the external loads $\mathbf{F}(t)$ in terms of Fourier series or Fourier integrals. The solution is in terms of complex numbers that cover the time span from $-\infty$ to ∞. Therefore, it is very effective for periodic types of loads such as mechanical vibrations, acoustics, sea-waves and wind [1]. However, the use of the frequency domain solution method for solving structures subjected to earthquake motions has the following disadvantages:

1. The mathematics, for most structural engineers including myself, is difficult to understand. Also, the solutions are difficult to verify.

2. Earthquake loading is not periodic; therefore, it is necessary to select a long time period in order that the solution from a finite length earthquake is completely damped out prior to the application of the same earthquake at the start of the next period of loading.

3. For seismic type loading the method is not numerically efficient. The transformation of the result from the frequency domain to the time domain, even with the use of Fast Fourier Transformation methods, requires a significant amount of computational effort.

4. The method is restricted to the solution of linear structural systems.

5. The method has been used, without sufficient theoretical justification, for the approximate nonlinear solution of site response problems and soil/structure interaction problems. Typically, it is used in an iterative manner to create linear equations. The linear damping terms are changed after each iteration in order to approximate the energy dissipation in the soil. Hence, dynamic equilibrium, within the soil, is not satisfied.

12.7 SOLUTION OF LINEAR EQUATIONS

The step-by-step solution of the dynamic equilibrium equations, the solution in the frequency domain, and the evaluation of eigenvectors and Ritz vectors all require the solution of linear equations of the following form:

$$\mathbf{AX} = \mathbf{B} \tag{12.5}$$

Where **A** is an 'N by N' symmetric matrix which contains a large number of zero terms. The 'N by M' **X** displacement and **B** load matrices indicate that more than one load condition can be solved at the same time.

The method used in many computer programs, including SAP2000 [5] and ETABS [6], is based on the profile or active column method of compact storage. Because the matrix is symmetric, it is only necessary to form and store the first nonzero term in each column down to the diagonal term in that column. Therefore, the sparse square matrix can be stored as a one dimensional array along with a *N by 1* integer array that indicates the location of each diagonal term. If the stiffness matrix exceeds the high-speed memory capacity of the computer a block storage form of the algorithm exists. Therefore, the capacity of the solution method is governed by the low speed disk capacity of the computer. This solution method is presented in detail in Appendix C of this book.

12.8 UNDAMPED HARMONIC RESPONSE

The most common and very simple type of dynamic loading is the application of steady-state harmonic loads of the following form:

$$\mathbf{F}(t) = \mathbf{f}\sin(\overline{\omega}\,t) \tag{12.5}$$

The node point distribution of all static load patterns, $\mathbf{f}$, which are not a function of time, and the frequency of the applied loading, $\overline{\omega}$, are user specified. Therefore, for the case of zero damping, the exact node point equilibrium equations for the structural system are

$$\mathbf{M}\ddot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t) = \mathbf{f}\ \sin(\overline{\omega}\,t) \tag{12.6}$$

The exact steady-state solution of this equation requires that the node point displacements and accelerations are given by

$$\mathbf{u}(t) = \mathbf{v}\ \sin(\overline{\omega}\, t), \qquad \ddot{\mathbf{u}}(t) = -\mathbf{v}\ \overline{\omega}^2 \sin(\overline{\omega}\, t) \tag{12.7}$$

Therefore, the harmonic node point response amplitude is given by the solution of the following set of linear equations:

$$[\mathbf{K} - \overline{\omega}^2 \mathbf{M}]\mathbf{v} = \mathbf{f} \quad \text{or} \quad \overline{\mathbf{K}}\mathbf{v} = \mathbf{f} \tag{12.8}$$

It is of interest to note that the normal solution for static loads is nothing more than a solution of this equation for zero frequency for all loads. It is apparent that the computational effort required for the calculation of undamped steady-state response is almost identical to that required by a static load analysis. Note that it is not necessary to evaluate mode shapes or frequencies to solve for this very common type of loading. The resulting node point displacements and member forces vary as $\sin(\overline{\omega}\, t)$. However, other types of loads that do not vary with time, such as dead loads, must be evaluated in a separate computer run.

12.9 UNDAMPED FREE VIBRATIONS

Most structures are in a continuous state of dynamic motion because of random loading such as wind, vibrating equipment, or human loads. These small ambient vibrations are normally near the natural frequencies of the structure and are terminated by energy dissipation in the real structure. However, special instruments attached to the structure can easily measure the motion. Ambient vibration field tests are often used to calibrate computer models of structures and their foundations.

After all external loads are removed from the structure, the equilibrium equation, which governs the undamped free vibration of a typical displaced shape $\mathbf{v}$, is

$$\mathbf{M}\ddot{\mathbf{v}} + \mathbf{K}\mathbf{v} = \mathbf{0} \tag{12.9}$$

At any time the displaced shape $\mathbf{v}$ may be a natural mode shape of the system, or any combination of the natural mode shapes. However, it is apparent the total energy within an undamped free vibrating system is a constant with respect to time. The sum of the kinetic energy and strain energy, at all points in time, is a constant and is defined as the ***mechanical energy*** of the dynamic system and can be calculated from:

$$E_M = \frac{1}{2}\dot{\mathbf{v}}^T\mathbf{M}\dot{\mathbf{v}} + \frac{1}{2}\mathbf{v}^T\mathbf{K}\mathbf{v} \tag{12.10}$$

12.10 SUMMARY

Dynamic analysis of three dimensional structural systems is a direct extension of static analysis. The elastic stiffness matrices are the same for both dynamic and static analysis. It is only necessary to lump the mass of the structure at the joints. The addition of inertia forces and energy dissipation forces will satisfy dynamic equilibrium. The dynamic solution for steady state harmonic loading, without damping, involves the same numerical effort as a static solution. Classically, there are many different mathematical methods to solve the dynamic equilibrium equations. However, it will later be shown in this book that the majority of both linear and nonlinear systems can be solved with one numerical method.

Energy is fundamental in dynamic analysis. At any point in time the external work supplied to the system must be equal to the sum of the kinetic and strain energy plus the energy dissipated in the system.

It is my opinion, with respect to earthquake resistant design, that we should try to minimize the mechanical energy in the structure. It is apparent that a rigid structure will have only kinetic energy and zero strain energy. On the other hand, a completely base isolated structure will have zero kinetic energy and zero strain energy. A structure cannot fail if it has zero strain energy.

12.11 REFERENCES

1. R. Clough, and J. Penzien, *Dynamics of Structures*, Second Edition, McGraw-Hill, Inc., ISBN 0-07-011394-7, 1993.

2. A. Chopra, *Dynamics of Structures*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, ISBN 0-13-855214-2, 1995.

3. K. Bathe, *Finite Element Procedures in Engineering Analysis*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, ISBN 0-13-317305-4, 1982.

4. E. L. Wilson and K. Bathe, "Stability and Accuracy Analysis of Direct Integration Methods," *Earthquake Engineering and Structural Dynamics*, Vol. 1, pp. 283-291, 1973.

5. *SAP2000 - Integrated Structural Analysis & Design Software*, Computers and Structures, Inc., Berkeley, California, 1997.

6. A. Habibullah, *ETABS - Three Dimensional Analysis of Building Systems, Users Manual*, Computers and Structures Inc., Berkeley, California, 1997.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

DYNAMIC ANALYSIS USING RESPONSE SPECTRUM SEISMIC LOADING

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

DYNAMIC ANALYSIS USING RESPONSE SPECTRUM SEISMIC LOADING

Prior To The Existence Of Inexpensive Personal Computers The Response Spectrum Method Was The Standard Approach For Linear Seismic Analysis

15.1 INTRODUCTION

The basic mode superposition method, which is restricted to linearly elastic analysis, produces the complete time history response of joint displacements and member forces. In the past there have been two major disadvantages in the use of this approach. First, the method produces a large amount of output information that can require a significant amount of computational effort to conduct all possible design checks as a function of time. Second, the analysis must be repeated for several different earthquake motions in order to assure that all frequencies are excited, since a response spectrum for one earthquake in a specified direction is not a smooth function.

There are computational advantages in using the response spectrum method of seismic analysis for prediction of displacements and member forces in structural systems. The method involves the calculation of only the maximum values of the displacements and member forces in each mode using smooth design spectra that are the average of several earthquake motions.

The purpose of this chapter is to summarize the fundamental equations used in the response spectrum method and to point out the many approximations and limitations of the method. For, example it cannot be used to approximate the nonlinear response of a complex three-dimensional structural system.

The recent increase in the speed of computers has made it practical to run many time history analyses in a short period of time. In addition, it is now possible to run design checks as a function of time, which produces superior results, since each member is not designed for maximum peak values as required by the response spectrum method.

15.2 DEFINITION OF A RESPONSE SPECTRUM

For three dimensional seismic motion, the typical modal Equation (13.6) is rewritten as

$$\begin{aligned}\ddot{y}(t)_n + 2\zeta_n\omega_n\dot{y}(t)_n + \omega_n^2 y(t)_n &= \\ p_{nx}\ddot{u}(t)_{gx} + p_{ny}\ddot{u}(t)_{gy} &+ p_{nz}\ddot{u}(t)_{gz}\end{aligned} \tag{15.1}$$

where the three *Mode Participation Factors* are defined by $p_{ni} = -\phi_n^T \mathbf{M}_i$ in which i is equal to x, y or z. Two major problems must be solved in order to obtain an approximate response spectrum solution to this equation. First, for each direction of ground motion maximum peak forces and displacements must be estimated. Second, after the response for the three orthogonal directions is solved it is necessary to estimate the maximum response due to the three components of earthquake motion acting at the same time. This section will address the modal combination problem due to one component of motion only. The separate problem of combining the results from motion in three orthogonal directions will be discussed later in this chapter.

For input in one direction only, Equation (15.1) is written as

$$\ddot{y}(t)_n + 2\zeta_n\omega_n\dot{y}(t)_n + \omega_n^2 y(t)_n = p_{ni}\ddot{u}(t)_g \tag{15.2}$$

Given a specified ground motion $\ddot{u}(t)_g$, damping value and assuming $p_{ni} = -1.0$ it is possible to solve Equation (15.2) at various values of ω and plot a curve of the maximum peak response $y(\omega)_{MAX}$. For this acceleration input, the curve is by definition the ***displacement response spectrum*** for the earthquake motion. A different curve will exist for each different value of damping.

A plot of $\omega\, y(\omega)_{MAX}$ is defined as the ***pseudo-velocity spectrum*** and a plot of $\omega^2 y(\omega)_{MAX}$ is defined as the ***pseudo-acceleration spectrum***. These three curves are normally plotted as one curve on special log paper. However, these pseudo-values have minimum physical significance and are not an essential part of a response spectrum analysis. The true values for maximum velocity and acceleration must be calculated from the solution of Equation (15.2).

There is a mathematical relationship, however, between the pseudo-acceleration spectrum and the total acceleration spectrum. The total acceleration of the unit mass, single degree-of-freedom system, governed by Equation (15.2), is given by

$$\ddot{u}(t)_T = \ddot{y}(t) + \ddot{u}(t)_g \tag{15.3}$$

Equation (15.2) can be solved for $\ddot{y}(t)$ and substituted into Equation (15.3) which yields

$$\ddot{u}(t)_T = -\omega^2 y(t) - 2\xi\omega\dot{y}(t) \tag{15.4}$$

Therefore, for the special case of zero damping, the total acceleration of the system is equal to $\omega^2 y(t)$. For this reason, the ***displacement response spectrum*** curve is normally not plotted as modal displacement $y(\omega)_{MAX}$ vs ω. It is standard to present the curve in terms of $S(\omega)$ vs. a period T in seconds. where

$$S(\omega)_a = \omega^2 y(\omega)_{MAX} \quad \text{and} \quad T = \frac{2\pi}{\omega} \qquad \text{(15.5a) and (15.5b)}$$

The pseudo-acceleration spectrum, $S(\omega)_a$, curve has the units of acceleration vs. period which has some physical significance for zero damping only. It is apparent that all response spectrum curves represent the properties of the earthquake at a specific site and are not a function of the properties of the structural system. After an estimation is made of the linear viscous damping properties of the structure, a specific response spectrum curve is selected.

15.3 CALCULATION OF MODAL RESPONSE

The maximum modal displacement, for a structural model, can now be calculated for a typical mode n with period T_n and corresponding spectrum response value $S(\omega_n)$. The maximum modal response associated with period T_n is given by

$$y(T_n)_{MAX} = \frac{S(\omega_n)}{\omega_n^2} \tag{15.6}$$

The maximum modal displacement response of the structural model is calculated from

$$\mathbf{u}_n = y(T_n)_{MAX}\,\phi_n \tag{15.7}$$

The corresponding internal modal forces, f_{kn}, are calculated from standard matrix structural analysis using the same equations as required in static analysis.

15.4 TYPICAL RESPONSE SPECTRUM CURVES

A ten second segment of the Loma Prieta earthquake motions, recorded on a soft site in the San Francisco Bay Area, is shown in Figure 15.1. The record has been corrected, by use of an iterative algorithm, for zero displacement, velocity and acceleration at the beginning and end of the ten second record. For the earthquake motions given in Figure 15.1a, the response spectrum curves for displacement and pseudo-acceleration are summarized in Figure 15.2a and 15.2b

The velocity curves have been intentionally omitted since they are not an essential part of the response spectrum method. Furthermore, it would require considerable space to clearly define terms such as peak ground velocity, pseudo velocity spectrum, relative velocity spectrum and absolute velocity spectrum.

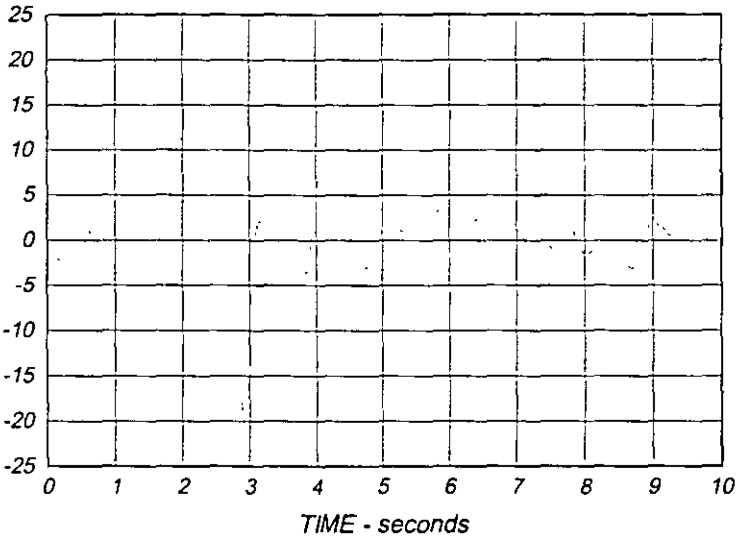


Figure 15.1a. Typical Earthquake Ground Acceleration - Percent of Gravity

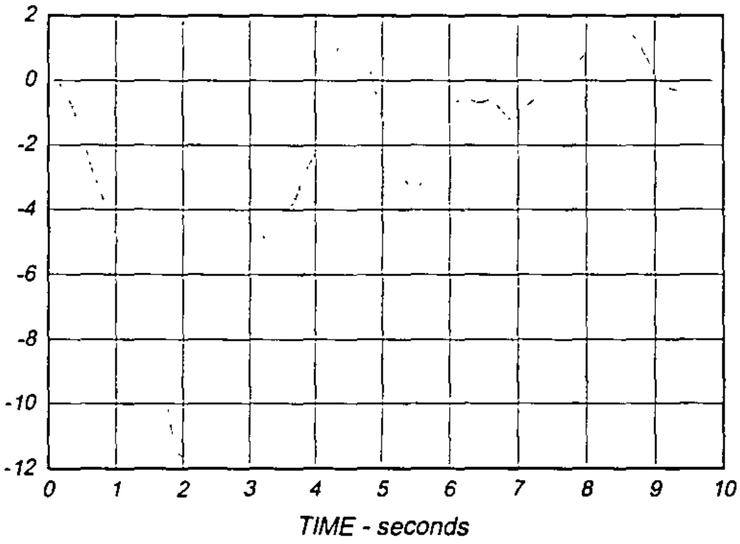


Figure 15.1b. Typical Earthquake Ground Displacements - Inches

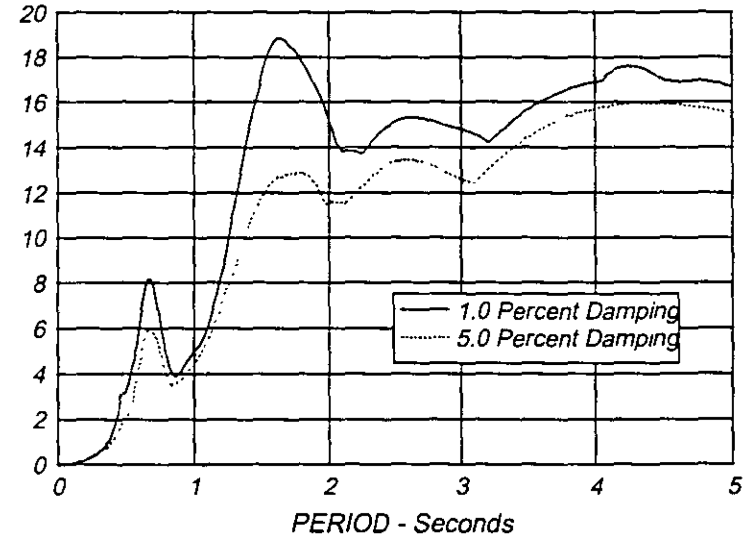


Figure 15.2a. Relative Displacement Spectrum $y(\omega)_{MAX}$ ***- Inches***

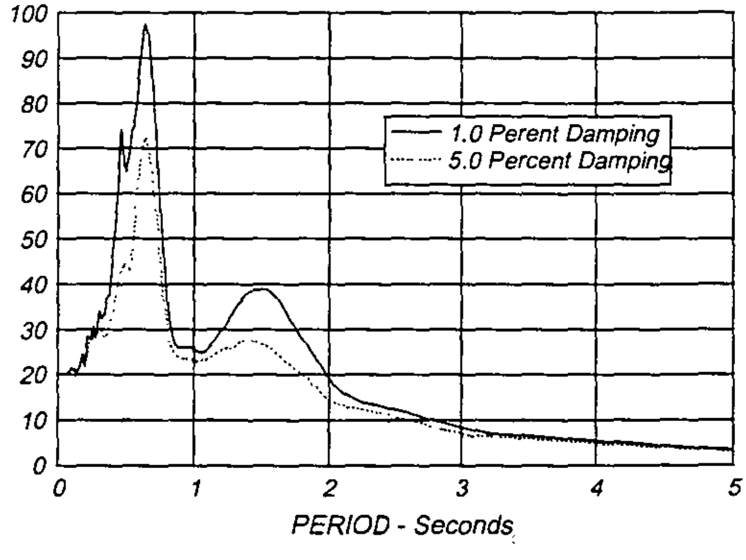


Figure 15.2b. Pseudo Acceleration Spectrum, $S_a = \omega^2 y(\omega)_{MAX}$ ***- Percent of Gravity***

The maximum ground acceleration, for the earthquake defined by Figure 15.1a, is 20.01 percent of gravity at 2.92 seconds. It is important to note that the pseudo acceleration spectrum, shown in Figure 15.2b, has the same value for a very short period system. This is due to the physical fact that a very rigid structure moves as a rigid body and the relative displacements within the structure are equal to zero as indicated by Figure 15.2a. Also, the behavior of a rigid structure is not a function of the viscous damping value.

The maximum ground displacement shown in Figure 15.1b is -11.62 inches at 1.97 seconds. For long period systems, the mass of the one-degree-of-freedom structure does not move significantly and has approximately zero absolute displacement. Therefore, the relative displacement spectrum curves, shown in Figure 15.2a, will converge to 11.62 inches for long periods and all values of damping. This type of real physical behavior is fundamental to the design of base isolated structures.

The relative displacement spectrum, Figure 15.2a, and the absolute acceleration spectrum, Figure 15.2b, have physical significance. However, the maximum relative displacement is directly proportional to the maximum forces developed in the structure. For this earthquake the maximum relative displacement is 18.9 inches at a period of 1.6 seconds for one percent damping and 16.0 inches at a period of four seconds for five percent damping. It is important to note the significant difference between one and five percent damping for this typical soft site record.

Figure 15.2b, the absolute acceleration spectrum, indicates maximum values at a period of 0.64 seconds for both values of damping. Also, the multiplication by ω^2 tends to completely eliminate the information contained in the long period range. Since most structural failures, during recent earthquakes, have been associated with soft sites, perhaps we should consider using the relative displacement spectrum as the fundamental form for selecting a design earthquake. The high frequency, short period, part of the curve should always be defined by

$$y(\omega)_{MAX} = \ddot{u}_{g\,MAX} / \omega^2 \quad \text{or} \quad y(T)_{MAX} = \ddot{u}_{g\,MAX} \frac{T^2}{4\pi^2} \tag{15.8}$$

where $\ddot{u}_{g\,MAX}$ is the peak ground acceleration.

15.5 THE CQC METHOD OF MODAL COMBINATION

The most conservative method that is used to estimate a peak value of displacement or force within a structure is to use the sum of the absolute of the modal response values. This approach assumes that the maximum modal values, for all modes, occur at the same point in time.

Another very common approach is to use the Square Root of the Sum of the Squares, SRSS, on the maximum modal values in order to estimate the values of displacement or forces. The SRSS method assumes that all of the maximum modal values are statistically independent. For three dimensional structures, in which a large number of frequencies are almost identical, this assumption is not justified.

The relatively new method of modal combination is the Complete Quadratic Combination, CQC, method [2] that was first published in 1981. It is based on random vibration theories and has found wide acceptance by most engineers and has been incorporated as an option in most modern computer programs for seismic analysis. Because many engineers and building codes are not requiring the use of the CQC method, one purpose of this chapter is to explain by example the advantages of using the CQC method and illustrate the potential problems in the use of the SRSS method of modal combination.

The peak value of a typical force can now be estimated, from the maximum modal values, by the CQC method with the application of the following double summation equation:

$$F = \sqrt{\sum_n \sum_m f_n \rho_{nm} f_m} \tag{15.9}$$

where f_n is the modal force associated with mode n. The double summation is conducted over all modes. Similar equations can be applied to node displacements, relative displacements and base shears and overturning moments.

The cross-modal coefficients, ρ_{nm}, for the CQC method with constant damping are

$$\rho_{nm} = \frac{8\zeta^2(1+r)\,r^{3/2}}{(1-r^2)^2 + 4\zeta^2 r(1+r)^2} \tag{15.10}$$

where $r = \omega_n / \omega_m$ and must be equal to or less than 1.0. It is important to note that the cross-modal coefficient array is symmetric and all terms are positive.

15.6 NUMERICAL EXAMPLE OF MODAL COMBINATION

The problems associated with the use of the absolute sum and the SRSS of modal combination can be illustrated by their application to the four story building shown in Figure 15.3. The building is symmetrical; however, the center of mass, of all floors, is located 25 inches from the geometric center of the building.

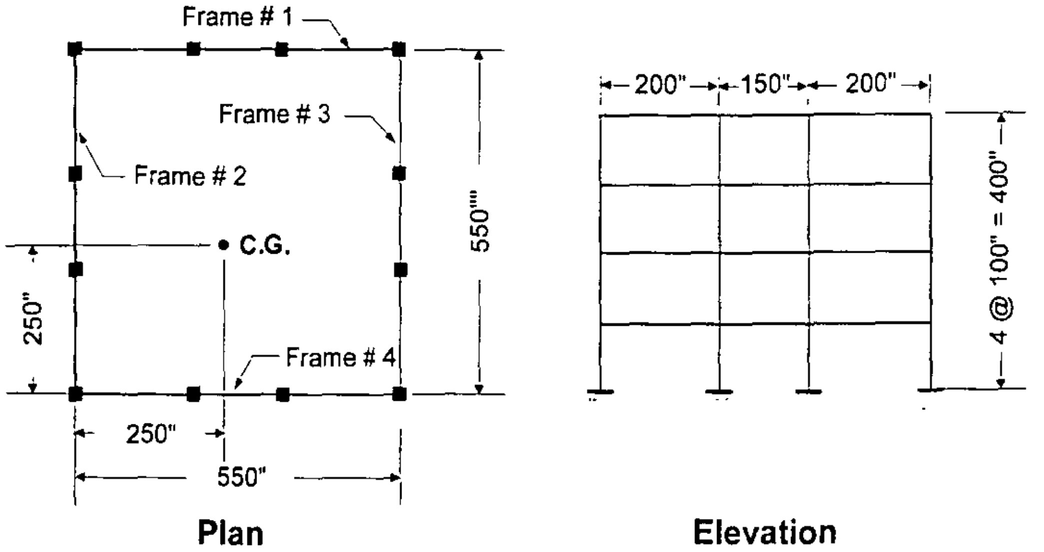
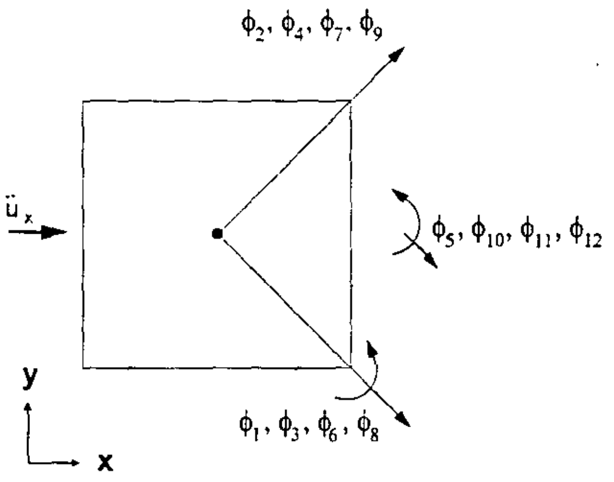


Figure 15.3. A Simple Three Dimensional Building Example



Mode	Frequencies (Radians/sec.)
1	13.869
2	13.931
3	43.995
4	44.189
5	54.418
6	77.686
7	78.029
8	108.32
9	108.80
10	172.613
11	304.80
12	425.00

Figure 15.4 Frequencies and Approximate Directions of Mode Shapes

The direction of the applied earthquake motion, a table of natural frequencies and the principal direction of the mode shape are summarized in Figure 15.4. One notes the closeness of the frequencies which is typical of most three dimensional building structures that are designed to equally resist earthquakes from both directions. Because of the small mass eccentricity, which is normal in real structures, the fundamental mode shape has x, y, as well as torsion components. Therefore, the model represents a very common three dimensional building system. Also, note that there is not ***a mode shape in a particular given direction*** as implied in many building codes and some text books on elementary dynamics.

The building was subjected to one component of the Taft, 1952, earthquake. An exact time history analysis, using all 12 modes, and a response spectrum analysis were conducted. The maximum modal base shears in the four frames for the first five modes are shown in Figure 15.5.

Figure 15.6 summarizes the maximum base shears, in each of the four frames, using different methods. The time history base shears, Figure 15.6a, are exact. The SRSS method, Figure 15.6b, produces base shears which under-estimate the exact values in the direction of the loads by approximately 30 percent and over-estimate the base shears normal to the loads by a factor of ten. The sum of the absolute values, Figure 15.6c, grossly over-estimates all results. The CQC method, Figure 15.6d, produces very realistic values that are close to the exact time history solution.

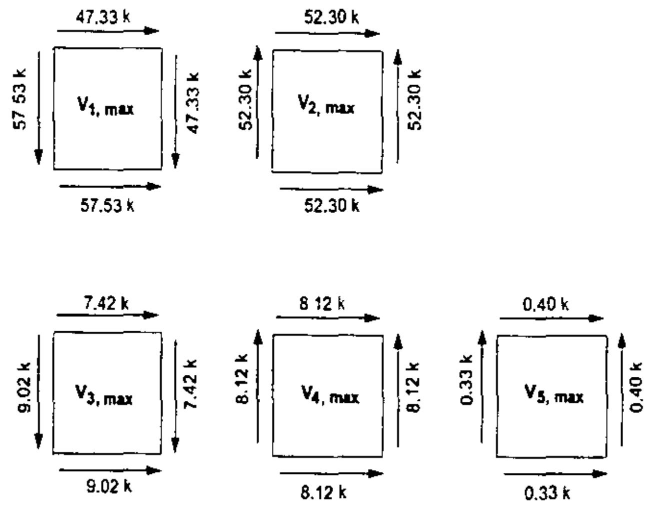


Figure 15.5. Base Shears in Each Frame for First Five Modes

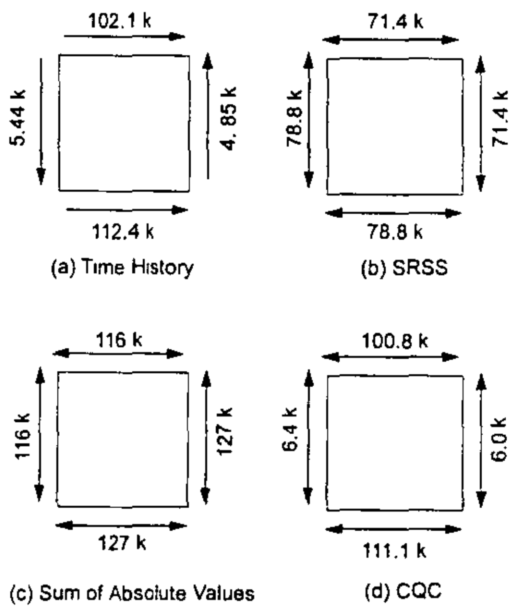


Figure 15.6. Comparison of Modal Combination Methods

The modal cross-correlation coefficients for this building are summarized in Table 15.1. It is of importance to note the existence of the relatively large off-diagonal terms that indicate which modes are coupled.

Table 15.1. Modal Cross-Correlation Coefficients - $\zeta = 0.05$

Mode	1	2	3	4	5	ω_n rad/sec
1	1.000	0.998	0.006	0.006	0.004	13.87
2	0.998	1.000	0.006	0.006	0.004	13.93
3	0.006	0.006	1.000	0.998	0.180	43.99
4	0.006	0.006	0.998	1.000	0.186	44.19
5	0.004	0.004	0.180	0.186	1.000	54.42

If one notes the signs of the modal base shears, shown in Figure 15.3, it is apparent how the application of the CQC method allows the sum of the base shears in the direction of the external motion to be added directly. In addition, the sum of the base shears, normal to the external motion, tend to cancel. The ability of the CQC method to recognize the relative sign of the terms in the modal response is the key to the elimination of errors in the SRSS method.

15.7 DESIGN SPECTRA

Design spectra are not uneven curves as shown in Figure 15.2 since they are intended to be the average of many earthquakes. At the present time, many building codes specify design spectra in the form shown in Figure 15.7.

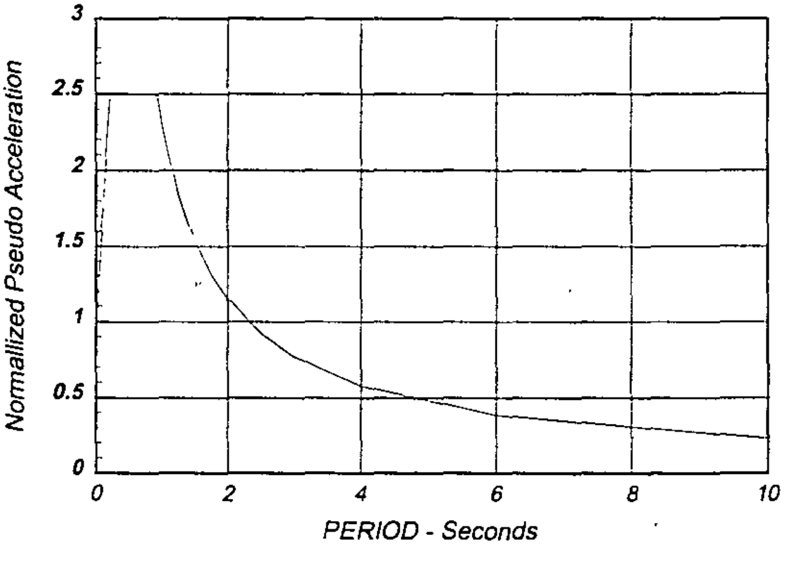


Figure 15.7 Typical Design Spectrum

The Uniform Building Code has defined specific equations for each range of the spectrum curve for four different soil types. For major structures it is now common practice to develop a site-dependent design spectrum which includes the effect of local soil conditions and distance to the nearest faults.

15.8 ORTHOGONAL EFFECTS IN SPECTRAL ANALYSIS

A well-designed structure should be capable of equally resisting earthquake motions from all possible directions. One option in existing design codes for buildings and bridges requires that members be designed for "100 percent of the prescribed seismic forces in one direction plus 30 percent of the prescribed forces in the perpendicular direction". Other codes and organizations require the use of 40 percent rather than 30 percent. However, they give no indication on how the directions are to be determined for complex structures. For structures that are rectangular and have clearly defined principal directions, these "percentage" rules yield approximately the same results as the SRSS method.

For complex three dimensional structures such as non-rectangular buildings, curved bridges, arch dams or piping systems, the direction of the earthquake which produces the maximum stresses, in a particular member or at a specified point, is not apparent. For time history input, it is possible to perform a large number of dynamic analyses at various angles of input in order to check all points for the critical earthquake directions. Such an elaborate study could conceivably produce a different critical input direction for each stress evaluated. However, the cost of such a study would be prohibitive.

It is reasonable to assume that motions that take place during an earthquake have one principal direction [1]. Or, during a finite period of time, when maximum ground acceleration occurs, a principal direction exists. For most structures this direction is not known and, for most geographical locations, cannot be estimated. Therefore, the only rational earthquake design criterion is that the structure must resist an earthquake of a given magnitude from any possible direction. In addition to the motion in the principal direction, a probability exists that motions normal to that direction will occur simultaneously. In addition, because of the complex nature of three dimensional wave propagation, it is valid to assume that these normal motions are statistically independent.

Based on these assumptions, a statement of the design criterion is "a structure must resist a major earthquake motion of magnitude S_1 for all possible angles θ and, at the same point in time, resist earthquake motions of magnitude S_2 at 90° to the angle θ". These motions are shown schematically in Figure 15.1.

15.8.1 Basic Equations For Calculation Of Spectral Forces

The stated design criterion implies that a large number of different analyses must be conducted in order to determine the maximum design forces and stresses. It will be shown, in this section, that maximum values for all members can be exactly evaluated from one computer run in which two global dynamic motions are applied. Furthermore, the maximum member forces calculated are invariant with respect to the selection system.

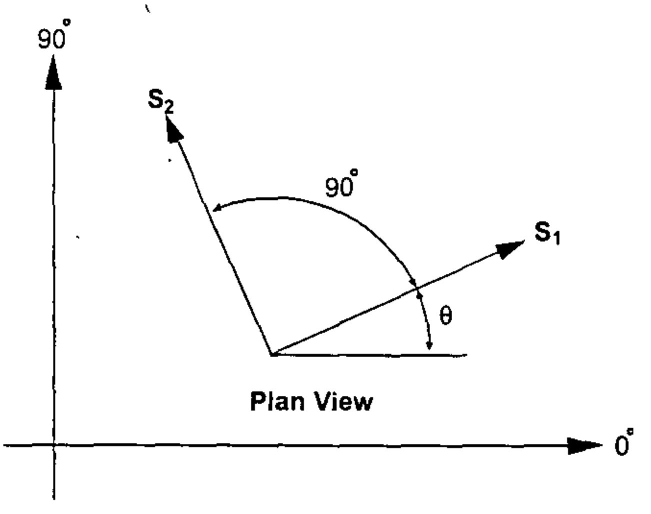


Figure 15.7. Definition of Earthquake Spectra Input

Figure 15.7 indicates that the basic input spectra S_1 and S_2 are applied at an arbitrary angle θ. At some typical point within the structure, a force, stress or displacement F is produced by this input. In order to simplify the analysis, it will be assumed that the minor input spectrum is some fraction of the major input spectrum. Or,

$$S_2 = a S_1 \tag{15.11}$$

where a is a number between 0 and 1.0.

Recently, Menun and Der Kiureghian [3] presented the CQC3 method for the combination of the effects of orthogonal spectrum.

The fundamental CQC3 equation for the estimation of a peak value is

$$
\begin{aligned}
F = [F_0^2 + a^2 F_{90}^2 - (1-a^2)(F_0^2 - F_{90}^2)\sin^2\theta \\
+ 2(1-a^2)F_{0-90}\sin\theta\cos\theta + F_z^2]^{\frac{1}{2}}
\end{aligned}
\tag{15.12}
$$

where

$$F_0^2 = \sum_n \sum_m f_{0n} \rho_{nm} f_{0m} \tag{15.13}$$

$$F_{90}^2 = \sum_n \sum_m f_{90n} \rho_{nm} f_{90m} \tag{15.14}$$

$$F_{0-90} = \sum_n \sum_m f_{0n} \rho_{nm} f_{90m} \tag{15.15}$$

$$F_Z^2 = \sum_n \sum_m f_{zn} \rho_{nm} f_{zm} \tag{15.16}$$

in which f_{0n} and f_{90n} are the modal values produced by 100 percent of the lateral spectrum applied at 0 and 90 degrees respectively and f_{zn} is the modal response from the vertical spectrum which can be different from the lateral spectrum.

It is important to note that for equal spectra $a = 1$, the value F is not a function of θ and the selection of the analysis reference system is arbitrary. Or,

$$F_{MAX} = \sqrt{F_0^2 + F_{90}^2 + F_z^2} \tag{15.17}$$

This indicates that it is possible to conduct only one analysis, with any reference system, and the resulting structure will have all members that are designed to equally resist earthquake motions from all possible directions. This method is acceptable by most building codes.

15.8.2 The General CQC3 Method

For a=1 the CQC3 method reduces to the SRSS method. However, this can be over conservative since real ground motions of equal value in all directions have not been recorded. Normally, the value of θ in Equation (15.12) is not known; therefore, it is necessary to calculate the critical angle that produces the maximum response. Differentiation of Equation (15.12) and setting the results to zero yields

$$\theta_{cr} = \frac{1}{2}\tan^{-1}[\frac{2F_{0-90}}{F_0^2 - F_{90}^2}] \tag{15.17}$$

Two roots exist for Equation (15.17) that must be checked in order that the following equation is maximum:

$$\begin{aligned} F_{MAX} = [&F_0^2 + a^2 F_{90}^2 - (1-a^2)(F_0^2 - F_{90}^2)\sin^2\theta_{cr} \\ &- 2(1-a^2)F_{0-90}\sin\theta_{cr}\cos\theta_{cr} + F_z^2]^{\frac{1}{2}} \end{aligned} \tag{15.18}$$

At the present time no specific guidelines have been suggested for the value of a. Reference [3] presented an example with values a between 0.50 and 0.85.

15.8.3 Examples Of Three Dimensional Spectra Analyses

The previously presented theory clearly indicates that the CQC3 combination rule, with a equal 1.0, is identical to the SRSS method and produces results, for all structural systems, which are not a function of the reference system used by the engineer. One example will be presented in order to show the advantages of the method. A very simple one-story structure, shown in Figure 15.8, was selected to compare the results of the 100/30 and 100/40 percentage rules with the SRSS rule. Note that the masses are not at the geometric center of the structure. The structure has two translations and one rotational degrees-of-freedom located at the center of mass. The columns, which are subjected to bending about the local 2 and 3 axes, are pinned at the top where they are connected to an in-plane rigid diaphragm.

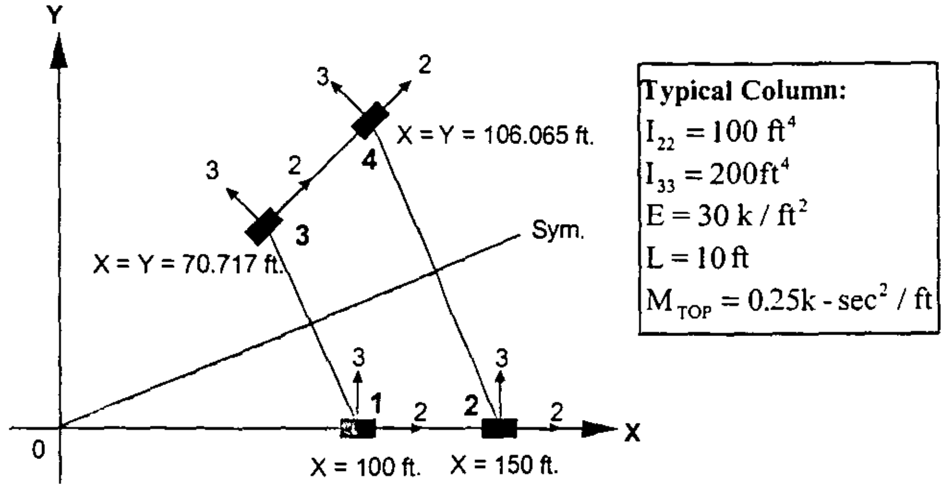


Figure 15.8. Three Dimensional Structure

The periods and normalized base shear forces associated with the mode shapes are summarized in Table 15.2. Since the structure has a plane of symmetry at 22.5 degrees, the second mode has no torsion and has a normalized base shear at 22.5 degrees with the x-axis. Due to this symmetry, it is apparent that columns 1 and 3 (or columns 2 and 4) should be designed for the same forces.

Table 15.2. Periods and Base Reaction Forces

Mode	Period Seconds	X-Force	Y-Force	X-Moment	Y-Moment	Torsion
1	1.01029	.383	-.924	9.24	3.83	-115.5
2	0.76918	-.924	-.383	3.83	-9.24	0.0
3	0.43102	.383	-.924	9.24	3.83	-115.5

The definition of the mean displacement response spectrum used in the spectra analysis is given in Table 15.3.

Table 15.3. Participating Masses and Response Spectrum Used

Mode	Period Seconds	X-MASS	Y-MASS	Spectral Displacement Used For Analysis
1	1.01029	14.43	84.081	1.00
2	0.76918	85.36	14.65	1.00
3	0.43102	0.22	1.27	1.00

The moments about the local 2 and 3 axes at the base of each of the four columns for the spectrum applied separately at 0.0 and 90 degrees are summarized in Tables 15.4 and 15.5 and are compared to the 100/30 rule.

Table 15.4. Moments About 2-Axes - 100/30 Rule

Member	M_0	M_{90}	$M_{SRSS} = \sqrt{M_0^2 + M_{90}^2}$	$M_{100/30}$	Error%
1	0.742	1.750	1.901	1.973	3.8
2	1.113	2.463	2.703	2.797	3.5
3	0.940	1.652	1.901	1.934	1.8
4	1.131	2.455	2.703	2.794	3.4

Table 15.5. Moments About 3-Axes - 100/30 Rule

Member	M_0	M_{90}	$M_{SRSS} = \sqrt{M_0^2 + M_{90}^2}$	$M_{100/30}$	Error%
1	2.702	0.137	2.705	2.743	1.4
2	2.702	0.137	2.705	2.743	1.4
3	1.904	1.922	2.705	2.493	-7.8
4	1.904	1.922	2.705	2.493	-7.8

For this example, the maximum forces do not vary significantly between the two methods. However, it does illustrate that the 100/30 combination method produces moments which are not symmetric, whereas the SRSS combination method produces logical and symmetric moments. For example. member 4 would be over-designed by 3.4 percent about the local 2-axis and under-designed by 7.8 percent about the local 3-axis using the 100/30 combination rule.

The SRSS and 100/40 design moments about the local 2 and 3 axes at the base of each of the four columns are summarized in Tables 15.6 and 15.7

Table 15.6. Moments About 2-Axes - 100/40 Rule

Member	M_0	M_{90}	$M_{SRSS} = \sqrt{M_0^2 + M_{90}^2}$	$M_{100/40}$	Error%
1	0.742	1.750	1.901	2.047	7.7
2	1.113	2.463	2.703	2.908	7.6
3	0.940	1.652	1.901	2.028	1.2
4	1.131	2.455	2.703	2.907	7.5

Table 15.7. Moments About 3-Axes - 100/40 Rule

Member	M_0	M_{90}	$M_{SRSS} = \sqrt{M_0^2 + M_{90}^2}$	$M_{100/40}$	Error%
1	2.702	0.137	2.705	2.757	1.9
2	2.702	0.137	2.705	2.757	1.9
3	1.904	1.922	2.705	2.684	-0.8
4	1.904	1.922	2.705	2.684	-0.8

The results presented in Tables 15.6 and 15.7 also illustrate that the 100/40 combination method produces results which are not reasonable. Because of symmetry, members 1 and 3 and members 2 and 4 should be designed for the same moments. Both the 100/30 and 100/40 rules fail this simple test.

If a structural engineer wants to be conservative, the results of the SRSS directional combination rule or the input spectra can be multiplied by an additional factor greater than one. One should not try to justify the use of the 100/40 percentage rule because it is conservative in "most cases". For complex three dimensional structures the use of the 100/40 or 100/30 percentage rule will produce member designs which are not equally resistant to earthquake motions from all possible directions.

15.8.4 Recommendations On Orthogonal Effects

For three dimensional response spectra analyses, it has been shown that the "design of elements for 100 percent of the prescribed seismic forces in one direction plus 30 or 40 percent of the prescribed forces applied in the perpendicular direction" is dependent on the user's selection of the reference system. These commonly used "percentage combination rules" are empirical and can underestimate the design forces in certain members and produce a member design which is relatively weak in one direction. It has been shown that the alternate building code approved method, in which an SRSS combination of two 100 percent spectra analyses with respect to any user defined orthogonal axes, will produce design forces that are not a function of the reference system. Therefore, the resulting structural design has equal resistance to seismic motions from all directions.

The use of the CQC3 method should be used if a value of a less than 1.0 can be justified. It will produce realistic results that are not a function of the user selected reference system.

15.9 LIMITATIONS OF THE RESPONSE SPECTRUM METHOD

It is apparent that use of the response spectrum method has limitations, some of which can be removed by additional development. However, it will never be accurate for nonlinear analysis of multi-degree of freedom structures. The author believes that in the future more time history dynamic response analyses will be conducted and the many approximations associated with the use of the response spectrum method will be avoided. Some of these additional limitations will be discussed in this section.

15.9.1 Story Drift Calculations

All displacements produced by the response spectrum method are positive numbers. Therefore, a plot of a dynamic displaced shape has very little meaning since each displacement is an estimation of the maximum value. Inter-story displacements are used to estimate damage to nonstructural elements and cannot be calculated directly from the probable peak values of displacement. A simple method to obtain a probable peak value of shear strain is to place a very thin panel element, with a shear modulus of unity, in the area where the deformation is to be calculated. The peak value of shear stress will be a good estimation of the damage index. The current code suggests a maximum value of 0.005 horizontal drift ratio, which is the same as panel shear strain if the vertical displacements are neglected.

15.9.2 Estimation of Spectra Stresses in Beams

The fundamental equation for the calculation of the stresses within the cross section of a beam is

$$\sigma = \frac{P}{A} + \frac{M_y x}{I_y} + \frac{M_x y}{I_x} \tag{15.19}$$

This equation can be evaluated for a specified x and y point in the cross section and for the calculated maximum spectral axial force and moments which are all positive values. It is apparent that the resulting stress may be conservative since all forces will probably not obtain their peak values at the same time.

For response spectrum analysis, the correct and accurate approach for the evaluation of equation (15.19) is to evaluate the equation for each mode of vibration. This will take into consideration the relative signs of axial forces and moments in each mode. An accurate value of the maximum stress can then be calculated from the modal stresses using the CQC double sum method. It has been the author's experience, with large three dimensional structures, that stresses calculated from modal stresses can be less than 50 percent of the value calculated using maximum peak values of moments and axial force.

15.10 SUMMARY

In this chapter it has been illustrated that the response spectrum method of dynamic analysis must be used carefully. The CQC method should be used to combine modal maxima in order to minimize the introduction of avoidable errors. The increase in computational effort, as compared to the SRSS method, is small compared to the total computer time for a seismic analysis. The CQC method has a sound theoretical basis and has been accepted by most experts in earthquake engineering. The use of the absolute sum or the SRSS method for modal combination cannot be justified.

In order for a structure to have equal resistance to earthquake motions from all directions, the CQC3 method should be used to combine the effects of earthquake spectra applied in three dimensions. The percentage rule methods have no theoretical basis and are not invariant with respect to the reference system.

Engineers, however, should clearly understand that the response spectrum method is an approximate method used to estimate maximum peak values of displacements and forces and that it has significant limitations. It is restricted to linear elastic analysis in which the damping properties can only be estimated with a low degree of confidence. The use of nonlinear spectra, which are commonly used, has very little theoretical background and should not be used for the analysis of complex three dimensional structures. For such structures, true nonlinear time-history response should be used as indicated in Chapter 19.

15.11 REFERENCES

1. J. Penzien and M. Watabe, "Characteristics of 3-D Earthquake Ground Motions," *Earthquake Engineering and Structural Dynamics*, Vol. 3, pp. 365-373, 1975.

2. E. L. Wilson, A. Der Kiureghian and E. R. Bayo, "A Replacement for the SRSS Method in Seismic Analysis," *Earthquake Engineering and Structural Dynamics*, Vol. 9, pp. 187-192, 1981.

3. C. Menun and A. Der Kiureghian, "A Replacement for the 30 % Rule for Muticomponent Excitation", *Earthquake Spectra, Vol. 13,* Number 1, February 1998.

15.9.3 Design Checks for Steel and Concrete Beams

Unfortunately, most design check equations for steel structures are written in terms of "design strength ratios" which are a nonlinear function of the axial force in the member; therefore, the ratios cannot be calculated in each mode. A new approximate method, to replace the current state of the art approach of calculating strength ratios based on maximum peak values of member forces, is proposed by the author. This would involve first calculating the maximum axial force. The design ratios would then be evaluated mode by mode, assuming the maximum axial force reduction factor remains constant for all modes. The design ratio for the member would then be estimated by a double-sum modal combination method such as the CQC3 method. This approach would improve accuracy and still be conservative.

For concrete structures additional development work is required in order to develop a completely rational method for the use of maximum spectral forces in a design check equation because of the nonlinear behavior of concrete members. A time history analysis may be the only approach that will produce rational design forces.

15.9.4 Calculation of Shear Force in Bolts

With respect to the interesting problem of calculating the maximum shear force in a bolt, it is not correct to estimate the maximum shear force from a vector summation since the x and y shears do not obtain their peak values at the same time. A correct method of estimating the maximum shear in a bolt is to check the maximum bolt shear at several different angles about the bolt axis. This would be a tedious approach using hand calculations; however, if the approach is built into a post processor computer program, the computational time to calculate the maximum bolt force is trivial.

The same problem exists if principal stresses are to be calculated from a response spectrum analysis. One must check at several angles in order to estimate the maximum and minimum value of the stress at each point in the structure.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

DYNAMIC ANALYSIS USING MODE SUPEROSITION

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

DYNAMIC ANALYSIS USING MODE SUPERPOSITION

The Mode Shapes Used To Uncouple The Dynamic Equilibrium Equations Need Not Be The Exact Free-Vibration Mode Shapes

13.1 EQUATIONS TO BE SOLVED

The dynamic force equilibrium Equation (12.4) can be rewritten in the following form as a set of N second order differential equations:

$$\mathbf{M}\ddot{\mathbf{u}}(t) + \mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t) = \mathbf{F}(t) = \sum_{j=1}^{J} \mathbf{f}_j \mathbf{g}(t)_j \tag{13.1}$$

All possible types of time-dependent loading, including wind, wave and seismic, can be represented by a sum of "J" space vectors $\mathbf{f}_j$, which are not a function of time, and J time functions $\mathbf{g}(t)_j$, where J cannot be greater than the number of displacements N.

The number of dynamic degrees-of-freedom is equal to the number of lumped masses in the system. Many publications advocate the elimination of all massless displacements by static condensation prior to the solution of Equation (13.1). The static condensation method reduces the number of dynamic equilibrium equations to solve; however, it can significantly increase the density and the bandwidth of the condensed stiffness matrix. In building type structures, in which each diaphragm has only three lumped masses, this approach is effective and is automatically used in building analysis programs.

For the dynamic solution of arbitrary structural systems, however, the elimination of the massless displacement is, in general, not numerically efficient. Therefore, the modern versions of the SAP program do not use static condensation in order to retain the sparseness of the stiffness matrix.

13.2 TRANSFORMATION TO MODAL EQUATIONS

The fundamental mathematical method that is used to solve Equation (13.1) is the separation of variables. This approach assumes the solution can be expressed in the following form:

$$\mathbf{u}(t) = \mathbf{\Phi Y}(t) \tag{13.2a}$$

Where $\mathbf{\Phi}$ is an "N by L" matrix containing L spatial vectors which are not a function of time, and $\mathbf{Y}(t)$ is a vector containing L functions of time.

From Equation (13.2a) it follows that

$$\dot{\mathbf{u}}(t) = \mathbf{\Phi}\dot{\mathbf{Y}}(t) \quad \text{and} \quad \ddot{\mathbf{u}}(t) = \mathbf{\Phi}\ddot{\mathbf{Y}}(t) \qquad \text{(13.2b) and (13.2c)}$$

Prior to solution, we require that the space functions satisfy the following mass and stiffness orthogonality conditions:

$$\mathbf{\Phi}^T \mathbf{M\Phi} = \mathbf{I} \quad \text{and} \quad \mathbf{\Phi}^T \mathbf{K\Phi} = \mathbf{\Omega}^2 \tag{13.3}$$

where $\mathbf{I}$ is a diagonal unit matrix and $\mathbf{\Omega}^2$ is a diagonal matrix which may or may not contain the free vibration frequencies. It should be noted that the fundamentals of mathematics place no restrictions on these vectors, other than the orthogonality properties. In this book all space function vectors are normalized so that the *Generalized Mass* $\phi_n^T M\phi_n = 1$.

After substitution of Equations (13.2) into Equation (13.1) and the pre-multiplication by $\mathbf{\Phi}^T$, the following matrix of L equations are produced:

$$\mathbf{I}\ddot{\mathbf{Y}}(t) + \mathbf{d}\dot{\mathbf{Y}}(t) + \Omega^2 = \sum_{j=1}^{J} \mathbf{p}_j \, \mathbf{g}(t)_j \tag{13.4}$$

where $\mathbf{p}_j = \Phi^T \mathbf{f}_j$ and are defined as the modal participation factors for time function j. The term p_{nj} is associated with the n th mode.

For all real structures the "L x L" matrix **d** is not diagonal; however, in order to uncouple the modal equations it is necessary to assume that there is no coupling between the modes. Therefore, it is assumed to be diagonal with the modal damping terms defined by

$$d_{nn} = 2\zeta_n \omega_n \tag{13.5}$$

where ζ_n is defined as the ratio of the damping in mode n to the critical damping of the mode [1].

A typical uncoupled modal equation, for linear structural systems, is of the following form:

$$\ddot{y}(t)_n + 2\zeta_n \omega_n \dot{y}(t)_n + \omega_n^2 y(t)_n = \sum_{j=1}^{J} p_{nj} \, g(t)_j \tag{13.6}$$

For three dimensional seismic motion, this equation can be written as

$$\ddot{y}(t)_n + 2\zeta_n \omega_n \dot{y}(t)_n + \omega_n^2 y(t)_n = p_{nx} \ddot{u}(t)_{gx} + p_{ny} \ddot{u}(t)_{gy} + p_{nz} \ddot{u}(t)_{gz} \tag{13.7}$$

where the three directional *Mass Participation Factors* are defined by $p_{ni} = -\phi_n^T \mathbf{M}_i$ in which i is equal to x, y or z and n is the mode number.

Prior to presenting the solution of Equation (13.6) for various types of loading it is convenient to define additional constants and functions which are summarized in Table 13.1. This will allow many of the equations presented in other parts of this book to be written in a compact form. Also, the notation reduces the tedium involved in the algebraic derivation and verification of various equations. In addition, it will allow the equations to be in a form that can be easily programmed and verified.

13.3 RESPONSE DUE TO INITIAL CONDITIONS ONLY

If the "n" subscript is dropped, Equation (13.6) can be written for a typical mode as

$$\ddot{y}(t) + 2\xi\omega\dot{y}(t) + \omega^2 y(t) = 0 \tag{13.8}$$

in which the initial modal displacement y_0 and velocity $\dot{y}_0$ are specified due to previous loading acting on the structure. Note that the functions $S(t)$ and $C(t)$, given in Table 13.1, are solutions to Equation (13.8).

Table 13.1. Summary of Notation used in Dynamic Response Equations

CONSTANTS

$$\omega_D = \omega\sqrt{1-\xi^2} \qquad \overline{\omega} = \omega\xi \qquad \overline{\xi} = \frac{\xi}{\sqrt{1-\xi^2}} \qquad a_0 = \frac{2\xi}{\omega\Delta t}$$

$$a_1 = 1 + a_0 \qquad a_2 = -\frac{1}{\Delta t} \qquad a_3 = -\overline{\xi}a_1 - a_2/\omega_D \qquad a_4 = -a_1$$

$$a_5 = -a_0 \qquad a_6 = -a_2 \qquad a_7 = -\overline{\xi}a_5 - a_6/\omega_D \qquad a_8 = -a_5$$

$$a_9 = \omega_D^2 - \overline{\omega}^2 \qquad a_{10} = 2\overline{\omega}\omega_D$$

FUNCTIONS

$$S(t) = e^{-\xi\omega t}\sin(\omega_D t) \qquad \dot{S}(t) = -\overline{\omega}S(t) + \omega_D C(t)$$

$$C(t) = e^{-\xi\omega t}\cos(\omega_D t) \qquad \dot{C}(t) = -\overline{\omega}C(t) - \omega_D S(t)$$

$$A_1(t) = C(t) + \overline{\xi}S(t) \qquad \ddot{S}(t) = -a_9 S(t) - a_{10}C(t)$$

$$A_2(t) = \frac{1}{\omega_D}S(t) \qquad \ddot{C}(t) = -a_9 C(t) + a_{10}S(t)$$

$$A_3(t) = \frac{1}{\omega^2}[a_1 + a_2 t + a_3 S(t) + a_4 C(t)]$$

$$A_4(t) = \frac{1}{\omega^2}[a_5 + a_6 t + a_7 S(t) + a_8 C(t)]$$

The solution of Equation (13.8) can now be written in the following compact form:

$$y(t) = A_1(t)y_0 + A_2(t)\dot{y}_0 \tag{13.9}$$

This solution can be easily verified since it satisfies Equation (13.8) and the initial conditions.

13.4 GENERAL SOLUTION DUE TO ARBITRARY LOADING

There are many different methods available to solve the typical modal equations. However, the use of the exact solution for a linear load over a small time increment has been found to be the most economical and accurate method to numerically solve this equation within computer programs. It does not have problems with stability and it does not introduce numerical damping. Since most seismic ground motions are defined as linear within 0.005 second intervals, the method is exact for this type of loading for all frequencies.

In order to simplify the notation, all loads are added together to form a typical modal equation of the following form:

$$\ddot{y}(t) + 2\zeta\,\omega\,\dot{y}(t) + \omega^2 y(t) = R(t) \tag{13.10}$$

where the modal loading $R(t)$ is a piece-wise linear function as shown in Figure 13.1.

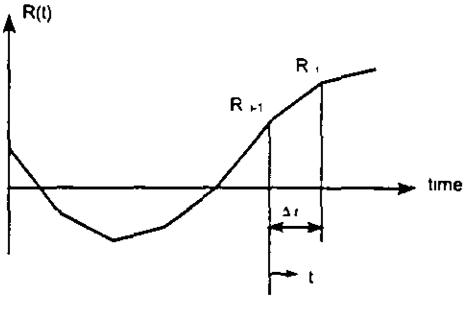


Figure 13.1 Typical Modal Load Function

The equation for the linear load function within the time step is by definition

$$R(t) = (1 - \frac{t}{\Delta t})R_{i-1} + \frac{t}{\Delta t}R_i \tag{13.11}$$

where the time t is in reference to the start of the time step. Now the exact solution within the time step can be written as

$$y(t) = A_1(t)y_{i-1} + A_2(t)\dot{y}_{i-1} + A_3(t)R_{i-1} + A_4(t)R_i \tag{13.12a}$$

where all functions are defined in Table 13.1. Again, the solution can be easily verified by substitution of Equation (13.12a) into Equation (13.10). It is apparent that the exact modal velocity and acceleration within the time step are given by

$$\dot{y}(t) = \dot{A}_1(t)y_{i-1} + \dot{A}_2(t)\dot{y}_{i-1} + \dot{A}_3(t)R_{i-1} + \dot{A}_4(t)R_i \tag{13.12b}$$

$$\ddot{y}(t) = \ddot{A}_1(t)y_{i-1} + \ddot{A}_2(t)\dot{y}_{i-1} + \ddot{A}_3(t)R_{i-1} + \ddot{A}_4(t)R_i \tag{13.12c}$$

Equations (13.12a, b and c) are evaluated at the end of the time increment Δt and the following modal displacement, velocity and acceleration at the end of the i th time step are given by the following set of recurrence equations:

$$y_i = A_1 y_{i-1} + A_2 \dot{y}_{i-1} + A_3 R_{i-1} + A_4 R_i \tag{13.13a}$$

$$\dot{y}_i = A_5 y_{i-1} + A_6 \dot{y}_{i-1} + A_7 R_{i-1} + A_8 R_i \tag{13.13b}$$

$$\ddot{y}_i = A_9 y_{i-1} + A_{10} \dot{y}_{i-1} + A_{11} R_{i-1} + A_{12} R_i \tag{13.13c}$$

The constants A_1 to A_{12}, which are summarized in Table 13.2, need to be computed only once for each mode. Therefore, for each time increment only 12 multiplications and 9 additions are required. Modern, inexpensive personal computers can complete one multiplication and one addition in approximately 10^{-6} seconds. Hence, the computer time required to solve for 200 steps per second for a 50 second duration earthquake is approximately 0.01 seconds. Or, 100 modal equations can be solved in one second of computer time. Therefore, there is no need to consider other numerical methods, such as the approximate Fast Fourier Transformation method or the numerical evaluation of the Duhamel integral, to solve these equations. Because of the speed of this exact piece-wise linear technique, it can also be used to develop accurate earthquake response spectra using a very small amount of computer time.

Table 13.2. Constants Used in Recurrence Equations (13.13)

$$A_1 = A_1(\Delta t) = C(\Delta t) + \overline{\xi} S(\Delta t)$$

$$A_2 = A_2(\Delta t) = \frac{1}{\omega_D} S(\Delta t)$$

$$A_3 = A_3(\Delta t) = \frac{1}{\omega^2}[a_1 + a_2\Delta t + a_3 S(\Delta t) + a_4 C(\Delta t)]$$

$$A_4 = A_4(\Delta t) = \frac{1}{\omega^2}[a_5 + a_6\Delta t + a_7 S(\Delta t) + a_8 C(\Delta t)]$$

$$A_5 = \dot{A}_1(\Delta t) = \dot{C}(\Delta t) + \overline{\xi} \dot{S}(\Delta t)$$

$$A_6 = \dot{A}_2(\Delta t) = \frac{1}{\omega_D} \dot{S}(\Delta t)$$

$$A_7 = \dot{A}_3(\Delta t) = \frac{1}{\omega^2}[a_2 + a_3\dot{S}(\Delta t) + a_4\dot{C}(\Delta t)]$$

$$A_8 = \dot{A}_4(\Delta t) = \frac{1}{\omega^2}[a_6 + a_7\dot{S}(\Delta t) + a_8\dot{C}(\Delta t)]$$

$$A_9 = \ddot{A}_1(\Delta t) = \ddot{C}(\Delta t) + \overline{\xi} \ddot{S}(\Delta t)$$

$$A_{10} = \ddot{A}_2(\Delta t) = \frac{1}{\omega_D} \ddot{S}(\Delta t)$$

$$A_{11} = \ddot{A}_3(\Delta t) = \frac{1}{\omega^2}[a_3\ddot{S}(\Delta t) + a_4\ddot{C}(\Delta t)]$$

$$A_{12} = \ddot{A}_4(\Delta t) = \frac{1}{\omega^2}[a_7\ddot{S}(\Delta t) + a_8\ddot{C}(\Delta t)]$$

13.5 SOLUTION FOR PERIODIC LOADING

The recurrence solution algorithm, summarized by Equation 13.13, is a very efficient computational method for arbitrary, transient, dynamic loads with initial conditions. It is possible to use this same simple solution method for arbitrary periodic loading as shown in Figure 13.2. Note that the total duration of the loading is from $-\infty$ to $+\infty$ and the loading function has the same amplitude and shape for each typical period T_p. Wind, sea wave and acoustic forces can produce this type of periodic loading. Also, dynamic live loads on bridges may also be of periodic form.

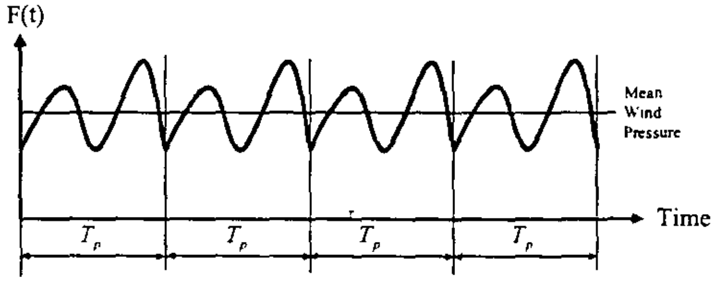


Figure 13.2. Example of Periodic Loading

For a typical duration T_p of loading, a numerical solution, for each mode, can be evaluated by the application of Equation (13.13) without initial conditions. This solution is incorrect since it does not have the correct initial conditions. Therefore, it is necessary for this solution $y(t)$ to be corrected in order that the exact solution $z(t)$ has the same displacement and velocity at the beginning and end of each loading period. In order to satisfy the basic dynamic equilibrium equation the corrective solution $x(t)$ must have the following form:

$$x(t) = x_0 A_1(t) + \dot{x}_0 A_2(t) \tag{13.14}$$

where the functions are defined in Table 13.1.

The total exact solution for displacement and velocity for each mode can now be written as

$$z(t) = y(t) + x(t) \tag{13.15a}$$

$$\dot{z}(t) = \dot{y}(t) + \dot{x}(t) \tag{13.15b}$$

In order that the exact solution is periodic the following conditions must be satisfied:

$$z(T_p) = z(0) \tag{13.16a}$$

$$\dot{z}(T_p) = \dot{z}(0) \tag{13.16b}$$

The numerical evaluation of Equation (13.14) produces the following matrix equation which must be solved for the unknown initial conditions:

$$\begin{bmatrix} 1 - A_1(T_p) & -A_2(T_p) \\ -\dot{A}_1(T_p) & 1 - \dot{A}_2(T_p) \end{bmatrix} \begin{bmatrix} x_0 \\ \dot{x}_0 \end{bmatrix} = \begin{bmatrix} -y(T_p) \\ -\dot{y}(T_p) \end{bmatrix} \tag{13.17}$$

The exact periodic solution for modal displacements and velocities can now be calculated from Equations (13.15a and 13.15b).

13.6 PARTICIPATING MASS RATIOS

Several Building Codes require that at least 90 percent of the participating mass is included in the calculation of response for each principal horizontal direction. This requirement is based on a unit base acceleration in a particular direction and calculating the base shear due to that load. The steady state solution for this case involves no damping or elastic forces; therefore, the modal response equations, for a unit base acceleration in the x-direction, can be written as

$$\ddot{y}_n = p_{nx} \tag{13.18}$$

The node point inertia forces, in the x-direction, for that mode are by definition

$$\mathrm{f}_{xn} = \mathrm{M}\ddot{\mathrm{u}}(t) = \mathrm{M}\,\phi_n \ddot{\mathrm{y}}_n = p_{nx}\mathrm{M}\,\phi_n \tag{13.19}$$

The total resisting base shear in the x-direction for mode n is the sum of all node point x forces. Or,

$$V_{nx} = -p_{nx}\,\mathrm{I}_x^T \mathrm{M}\,\phi_n = p_{nx}^2 \tag{13.20}$$

The total base shear in the x-direction, including L modes, will be

$$V_x = \sum_{n=1}^{L} p_{nx}^2 \tag{13.21}$$

We can now define the participating mass in all three directions as a ratio of the total mass in that direction by

$$X_{mass} = \frac{\sum_{n=1}^{L} p_{nx}^2}{\sum m_x} \tag{13.22a}$$

$$Y_{mass} = \frac{\sum_{n=1}^{L} p_{ny}^2}{\sum m_y} \tag{13.22b}$$

$$Z_{mass} = \frac{\sum_{n=1}^{L} p_{nz}^2}{\sum m_z} \tag{13.22c}$$

If all modes are used, these ratios will all be equal to 1.0. It is clear that the 90 percent participation rule is intended to estimate the accuracy of a solution for base motion only. ***It can not be used as an error estimator for other types of loading such as point loads acting on the structure.*** The SAP and ETABS programs produce the contribution of each mode to these ratios. In addition, an examination of these factors gives the engineer an indication of the direction of the base shear associated with each mode.

13.7 STATIC LOAD PARTICIPATION RATIOS

For arbitrary loading it is useful to determine if the number of vectors used is adequate to approximate the true response of the structural system. One method, which the author has proposed, is to evaluate the static displacements using a truncated set of vectors to solve for the response due to static load patterns. As indicated by Equation (13.1) the loads can be written as

$$\mathbf{F}(t) = \sum_{j=1}^{J} \mathrm{f}_j \mathrm{g}(\mathrm{t})_j \tag{13.23}$$

If one solves the statics problem for the exact displacement u_j due to the load pattern f_j the total strain energy associated with load condition j is

$$E_j = \frac{1}{2} f_j^T u_j \tag{13.24}$$

From the fundamental definition of the mode superposition method, a truncated set of vectors defines the approximate static displacement $\overline{u}_j$ as

$$\overline{u}_j = \sum_{n=1}^{L} y_n \phi_n \tag{13.25}$$

where, from Equation 13.6, the static modal response, neglecting inertia and damping forces, is given by

$$y_n = \frac{1}{\omega_n^2} \phi_n^T f_j \tag{13.26}$$

The total strain energy associated with the truncated mode shape solution is

$$\overline{E}_j = \frac{1}{2} f_j^T \overline{u}_j = \frac{1}{2} \sum_{n=1}^{L} \left(\frac{\phi_n^T f_j}{\omega_n} \right)^2 \tag{13.27}$$

A ***load participation ratio*** r_j can now be defined for load condition j as

$$r_j = \frac{\overline{E}_j}{E_j} \tag{13.28}$$

If this ratio is close to 1.0 the errors introduced by vector truncation will be very small. However, if this ratio is less than 90 percent additional vectors should be used in the analysis in order to capture the ***static load response***. Additional experience with this factor is required in order to use it as an accurate error estimator for all problems.

It has been the experience of the author that the use of exact eigenvectors is not an accurate vector basis for the dynamic analysis of structures subjected to point loads. Whereas, load-dependent vectors, which are defined in the following chapter, always produce a load participation ratio of 1.0.

13.8 SUMMARY

The mode superposition method is a very powerful method used to reduce the number of unknowns in a dynamic response analysis. All types of loading can be accurately approximated by piece-wise linear functions within a small time increment. An exact solution exists for this type of loading and this solution can be computed with a trivial amount of computer time for equal time increments. Therefore, there is no need to present other methods for the numerical evaluation of modal equations when a computer program is used.

To solve for the linear dynamic response of structures subjected to periodic loading it is only necessary to add a corrective solution to the transient solution for a typical time period of loading. Hence, only one numerical algorithm is required to solve a large number of different dynamic response problems in structural engineering.

Participating mass factors can be used to estimate the number of vectors required in an elastic seismic analysis. The use of mass participation factors to estimate the accuracy of a nonlinear seismic analysis can introduce significant errors; because, internal nonlinear forces, that are in equal and opposite directions, do not produce a base shear. A dynamic load participation ratio is defined which can be used to estimate the number of vectors required for other types of loading.

Practical Three Dimensional Nonlinear Static Pushover Analysis

By Ashraf Habibullah, S.E.[1], and Stephen Pyle, S.E.[2]

(Published in *Structure* Magazine, Winter, 1998)

The recent advent of performance based design has brought the nonlinear static pushover analysis procedure to the forefront. Pushover analysis is a static, nonlinear procedure in which the magnitude of the structural loading is incrementally increased in accordance with a certain predefined pattern. With the increase in the magnitude of the loading, weak links and failure modes of the structure are found. The loading is monotonic with the effects of the cyclic behavior and load reversals being estimated by using a modified monotonic force-deformation criteria and with damping approximations. Static pushover analysis is an attempt by the structural engineering profession to evaluate the real strength of the structure and it promises to be a useful and effective tool for performance based design.

The ATC-40 and FEMA-273 documents have developed modeling procedures, acceptance criteria and analysis procedures for pushover analysis. These documents define force-deformation criteria for hinges used in pushover analysis. As shown in Figure 1, five points labeled A, B, C, D, and E are used to define the force deflection behavior of the hinge and three points labeled IO, LS and CP are used to define the acceptance criteria for the hinge. (IO, LS and CP stand for Immediate Occupancy, Life Safety and Collapse Prevention respectively.) The values assigned to each of these points vary depending on the type of member as well as many other parameters defined in the ATC-40 and FEMA-273 documents.

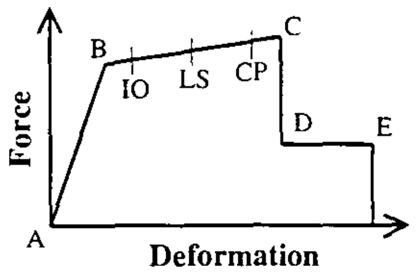


Figure 1: Force-Deformation For Pushover Hinge

This article presents the steps used in performing a pushover analysis of a simple three-dimensional building. SAP2000, a state-of-the-art, general-purpose, three-dimensional structural analysis program, is used as a tool for performing the pushover. The SAP2000 static pushover analysis capabilities, which are fully integrated into the program, allow quick and easy implementation of the pushover procedures prescribed in the ATC-40 and FEMA-273 documents for both two and three-dimensional buildings.

The following steps are included in the pushover analysis. Steps 1 through 4 discuss creating the computer model, step 5 runs the analysis, and steps 6 through 10 review the pushover analysis results.

1. Create the basic computer model (without the pushover data) in the usual manner as shown in Figure 2. The graphical interface of SAP2000 makes this a quick and easy task.

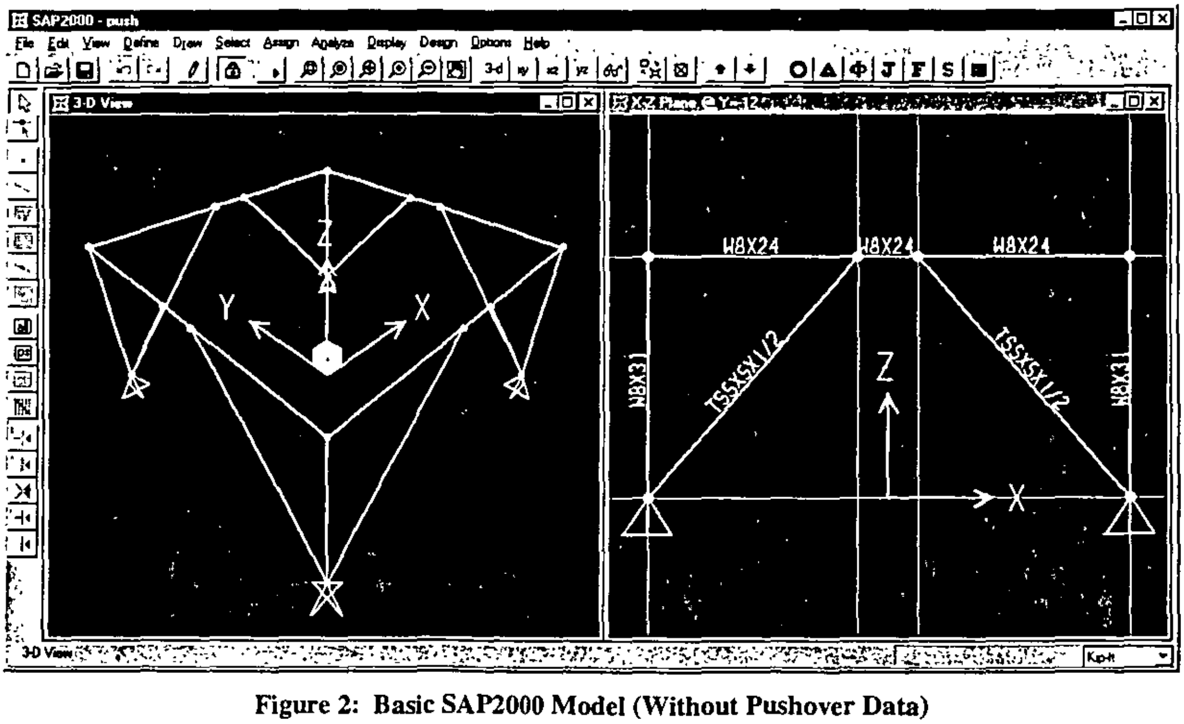


Figure 2: Basic SAP2000 Model (Without Pushover Data)

2. Define properties and acceptance criteria for the pushover hinges as shown in Figure 3. The program includes several built-in default hinge properties that are based on average values from ATC-40 for concrete members and average values from FEMA-273 for steel members. These built in properties can be useful for preliminary analyses, but user-defined properties are recommended for final analyses. This example uses default properties.

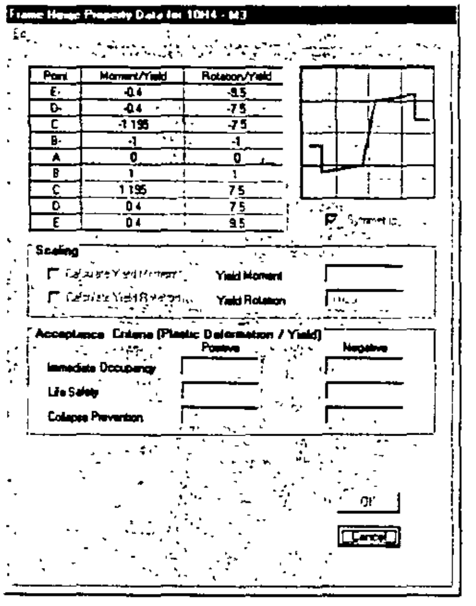


Figure 3: Frame Hinge Property

3. Locate the pushover hinges on the model by selecting one or more frame members and assigning them one or more hinge properties and hinge locations as shown in Figure 4.

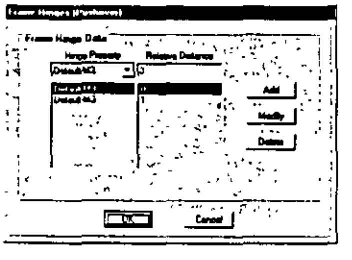


Figure 4: Assign Pushover Hinges

4. Define the pushover load cases. In SAP2000 more than one pushover load case can be run in the same analysis. Also a pushover load case can start from the final conditions of another pushover load case that was previously run in the same analysis. Typically the first pushover load case is used to apply gravity load and then subsequent lateral pushover load cases are specified to start from the final conditions of the gravity pushover. Pushover load cases can be force controlled, that is, pushed to a certain defined force level, or they can be displacement controlled, that is, pushed to a specified displacement. Typically a gravity load pushover is force controlled and lateral pushovers are displacement controlled. SAP2000 allows the distribution of lateral force used in the pushover to be based on a uniform acceleration in a specified direction, a specified mode shape, or a user-defined static load case. The dialog box shown in Figure 5 shows how the displacement controlled lateral pushover case that is based on a user-defined static lateral load pattern (named Push) is defined for this example.

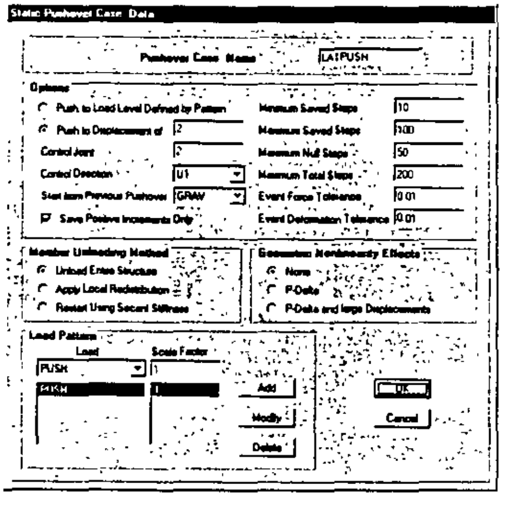


Figure 5: Pushover Load Case Data

5. Run the basic static analysis and, if desired, dynamic analysis. Then run the static nonlinear pushover analysis.

6. Display the pushover curve as shown in Figure 6. The File menu shown in this display window allows you to view and if desired, print to either a printer or an ASCII file, a table which gives the coordinates of each step of the pushover curve and summarizes the number of hinges in each state as defined in Figure 1 (for example, between IO and LS, or between D and E). This table is shown in Figure 7.

7. Display the capacity spectrum curve as shown in Figure 8. Note that you can interactively modify the magnitude of the earthquake and the damping information on this form and immediately see the new capacity spectrum plot. The performance point for a given set of values is defined by the intersection of the capacity curve (green) and the single demand spectrum curve (yellow). Also, the file menu in this display allows you to print the coordinates of the capacity curve and the demand curve as well as other information used to convert the pushover curve to Acceleration-Displacement Response Spectrum format (also known as ADRS format, see page 8-12 in ATC-40).

8. Review the pushover displaced shape and sequence of hinge formation on a step-by-step basis as shown in the left-hand side of Figure 9. The arrows in the bottom right-hand corner of the screen allow you to move through the pushover step-by-step. Hinges appear when they yield and are color coded based on their state (see legend at bottom of screen).

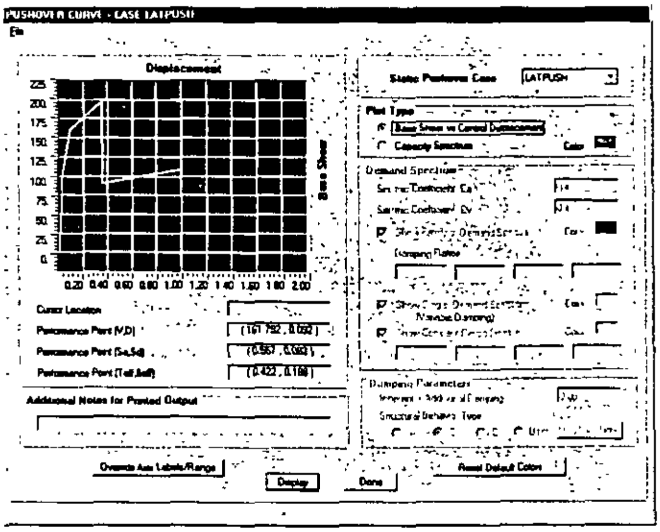


Figure 6: Pushover Curve

PUSHOVER CURVE

File

Step	Displacement	Base Shear	A-B	B-IO	IO-LS	LS-CP	CP-C	C-D	D-E	>E	TOTAL
0	0.0000	0.0000	60	0	0	0	0	0	0	0	60
1	0.0274	100.4879	57	3	0	0	0	0	0	0	60
2	0.0821	157.4939	51	8	1	0	0	0	0	0	60
3	0.0878	161.1481	48	11	1	0	0	0	0	0	60
4	0.3481	198.9614	46	6	3	2	1	2	0	0	60
5	0.3481	160.7877	46	6	3	2	1	0	2	0	60
6	0.3513	161.2326	46	6	3	2	1	0	0	2	60
7	0.3513	142.7613	46	6	3	2	1	0	0	2	60
8	0.3530	144.4586	46	6	3	2	1	0	0	2	60
9	0.3564	144.9255	46	6	3	0	1	2	0	2	60
10	0.3565	108.0511	46	6	3	0	1	0	2	2	60
11	0.3596	100.2019	46	6	3	0	1	0	0	4	60
12	0.3596	89.6470	46	6	3	0	1	0	0	4	60
13	0.4530	93.9815	40	12	3	0	1	0	0	4	60
14	0.6530	99.5164	40	0	15	0	1	0	0	4	60
15	0.8530	105.1322	40	0	10	5	1	0	0	4	60
16	1.0031	109.3216	40	0	0	13	1	2	0	4	60
17	1.0031	-0.4043	40	0	0	11	1	0	0	8	60
18	1.2031	-0.4404	40	0	0	11	1	0	0	8	60
19	1.4031	-0.3965	40	0	0	11	1	0	0	8	60
20	1.6031	-0.3526	40	0	0	11	1	0	0	8	60
21	1.8031	-0.3086	40	0	0	11	1	0	0	8	60
22	2.0000	-0.2654	40	0	0	11	1	0	0	8	60

Figure 7: Tabular Data For Pushover Curve

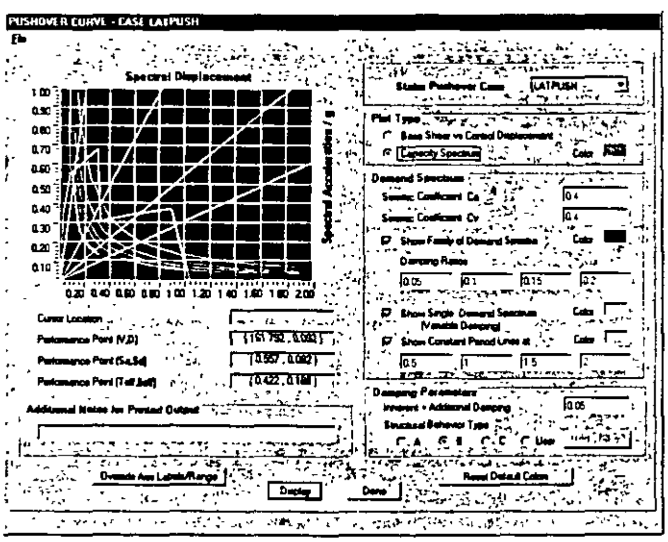


Figure 8: Capacity Spectrum Curve

9. Review member forces on a step-by-step basis as shown in the right-hand side of Figure 9. Often it is useful to view the model in two side-by-side windows with the step-by-step displaced shape in one window and the step-by-step member forces in the other, as shown in Figure 9. These windows can be synchronized to the same step, and can thus greatly enhance the understanding of the pushover results.

10. Output for the pushover analysis can be printed in a tabular form for the entire model or for selected elements of the model. The types of output available in this form include joint displacements at each step of the pushover, frame member forces at each step of the pushover, and hinge force, displacement and state at each step of the pushover.

For buildings that are being rehabilitated it is easy to investigate the effect of different strengthening schemes. The effect of added damping can be immediately seen on the capacity spectrum form (step 7, Figure 8). You can easily stiffen or strengthen the building by changing member properties and rerunning the analysis. Finally you can easily change the assumed detailing of the building by modifying the hinge acceptance criteria (step 2, Figures 1 and 3) and rerunning the analysis.

References

ATC, 1996
Seismic Evaluation and Retrofit of Concrete Buildings, Volume 1, ATC-40 Report, Applied Technology Council, Redwood City, California.

FEMA, 1997
NEHRP Guidelines for the Seismic Rehabilitation of Buildings, Developed by the Building Seismic Safety Council for the Federal Emergency Management Agency (Report No. FEMA 273), Washington, D.C.

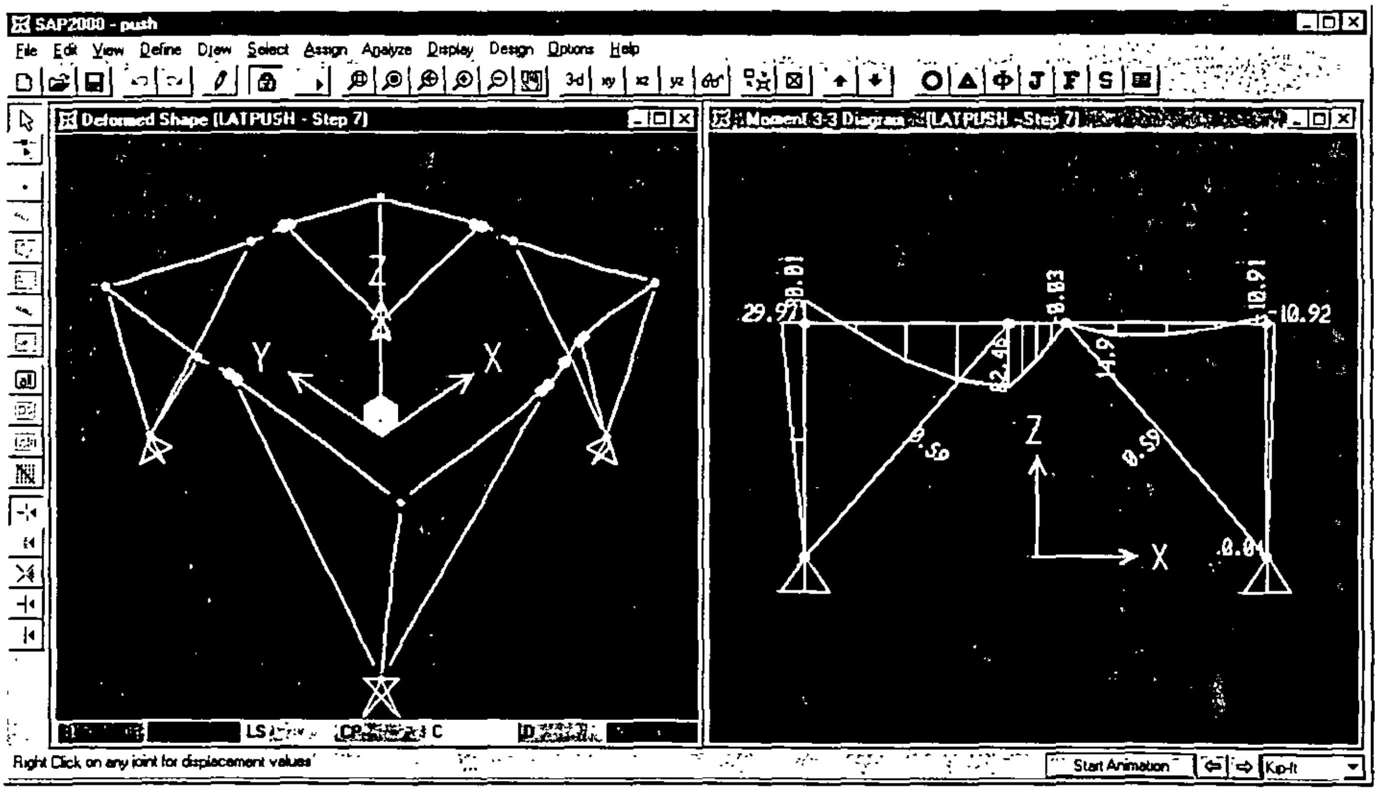


Figure 9: Step-By-Step Deformations and Member Forces For Pushover

1. President, Computers and Structures, Inc., Berkeley, CA.
2. Senior Structural Engineer, Computers and Structures, Inc., Berkeley, CA.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

SOIL – STRUCTURE INTERACTION

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

SOIL-STRUCTURE INTERACTION

At A Finite Distance from A Structure the Absolute Displacements Must Approach the Free-Field Displacements

16.1. INTRODUCTION

The estimation of earthquake motions at the site of a structure is the most important phase of the design or retrofit of a structure. Because of the large number of assumptions required, experts in the field often disagree by over a factor of two as to the magnitude of motions expected at the site without the structure present. This lack of accuracy of the basic input motions, however, does not justify the introduction of additional unnecessary approximations in the dynamic analysis of the structure and its interaction with the material under the structure. Therefore, it will be assumed that the free-field motions at the location of the structure, without the structure present, can be estimated and are specified in the form of earthquake acceleration records in three directions. It is now common practice, on major engineering projects, to investigate several different sets of ground motions in order to consider both near fault and far fault events.

If a lightweight flexible structure is built on a very stiff rock foundation, a valid assumption is that the input motion at the base of the structure is the same as the free-field earthquake motion. This assumption is valid for a large number of building systems since most building type structures are approximately 90 percent voids, and, it is not unusual that the weight of the structure is excavated before the structure is built. However, if the structure is very massive and stiff, such as a concrete gravity dam, and the foundation is relatively soft, the motion at the base of the structure may be significantly different than the free-field surface motion. Even for this extreme case, however, it is apparent that the most significant interaction effects will be near the structure. and, at some finite distance from the base of the structure, the displacements will converge back to the free-field earthquake motion.

16.2. SITE RESPONSE ANALYSIS

The 1985 Mexico City and many recent earthquakes clearly illustrate the importance of local soil properties on the earthquake response of structures. These earthquakes demonstrated that the rock motions could be amplified at the base of a structure by over a factor of five. Therefore, there is a strong engineering motivation for a site-dependent dynamic response analysis for many foundations in order to determine the free-field earthquake motions. The determination of a realistic site-dependent free-field surface motion at the base of a structure can be the most important step in the earthquake resistant design of any structure.

For most horizontally layered sites a one dimensional pure shear model can be used to calculate the free-field surface displacements given the earthquake motion at the base of a soil deposit. Many special purpose computer programs exist for this purpose. SHAKE [1] is a well-known program, based on the frequency domain solution method, which iterates to estimate effective linear stiffness and damping properties in order to approximate the nonlinear behavior of the site. WAVES [2] is a new nonlinear program in which the nonlinear equations of motion are solved by a direct step-by-step integration method. If the soil material can be considered linear then the SAP2000 program, using the SOLID element, can be used to calculate either the one, two or three dimensional free-field motions at the base of a structure. In addition, a one dimensional nonlinear site analysis can be accurately conducted using the FNA option in the SAP2000 program.

16.3. KINEMATIC OR SOIL-STRUCTURE INTERACTION

The most common soil-structure interaction SSI approach, used for three dimensional soil-structure systems, is based on the "added motion" formulation [3]. This formulation is mathematically simple, theoretically correct, and is easy to automate and use within a general linear structural analysis program. In addition, the formulation is valid for free-field motions caused by earthquake waves generated from all sources. The method requires that the free-field motions at the base of the structure be calculated prior to the soil-structure interactive analysis.

In order to develop the fundamental SSI dynamic equilibrium equations consider the three dimensional soil-structure system shown in Figure 16.1.

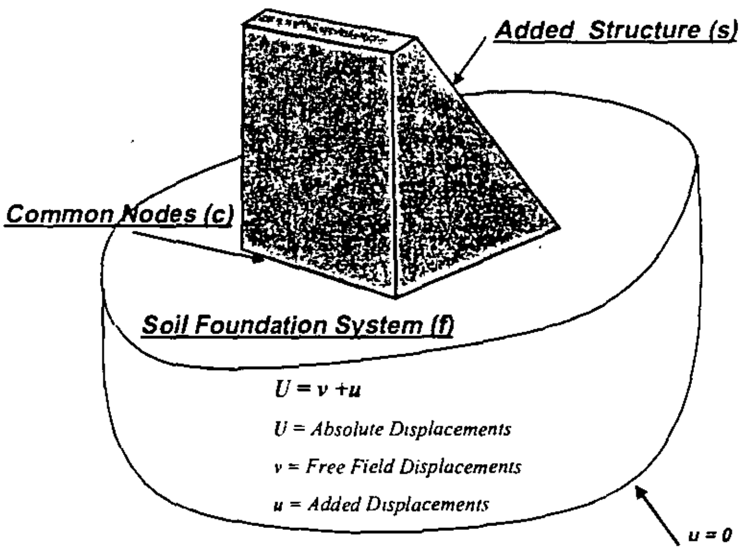


Figure 16.1. Soil-Structure Interaction Model

Consider the case where the SSI model is divided into three sets of node points. The common nodes at the interface of the structure and foundation are identified with "c"; the other nodes within the structure are "s" nodes; and the other nodes within the foundation are "f" nodes. From the direct stiffness approach in structural analysis, the dynamic force equilibrium of the system is given in terms of the ***absolute displacements***, $\mathbf{U}$, by the following sub-matrix equation:

$$\begin{bmatrix} \mathbf{M}_{ss} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{M}_{ff} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{U}}_s \\ \ddot{\mathbf{U}}_c \\ \ddot{\mathbf{U}}_f \end{bmatrix} + \begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sf} & \mathbf{0} \\ \mathbf{K}_{cf} & \mathbf{K}_{cc} & \mathbf{K}_{cf} \\ \mathbf{0} & \mathbf{K}_{fc} & \mathbf{K}_{ff} \end{bmatrix} \begin{bmatrix} \mathbf{U}_s \\ \mathbf{U}_c \\ \mathbf{U}_f \end{bmatrix} = \begin{bmatrix} \mathbf{0} \\ \mathbf{0} \\ \mathbf{0} \end{bmatrix} \tag{16.1}$$

where the mass and the stiffness at the contact nodes are the sum of the contribution from the structure *(s)* and foundation *(f)*, and are given by

$$\mathbf{M}_{cc} = \mathbf{M}_{cc}^{(s)} + \mathbf{M}_{cc}^{(f)} \quad \text{and} \quad \mathbf{K}_{cc} = \mathbf{K}_{cc}^{(s)} + \mathbf{K}_{cc}^{(f)} \tag{16.2}$$

In terms of absolute motion, there are no external forces acting on the system. However, the displacements at the boundary of the foundation must be known. In order to avoid solving this SSI problem directly, the dynamic response of the foundation without the structure is calculated. In many cases, this ***free-field*** solution can be obtained from a simple one-dimensional site model. The three dimensional free-field solution is designated by the absolute displacements $\mathbf{v}$ and absolute accelerations $\ddot{\mathbf{v}}$. By a simple change of variables it is now possible to express the absolute displacements $\mathbf{U}$ and accelerations $\ddot{\mathbf{U}}$ in terms of displacements $\mathbf{u}$ relative to the free-field displacements $\mathbf{v}$. Or,

$$\begin{bmatrix} \mathbf{U}_s \\ \mathbf{U}_c \\ \mathbf{U}_f \end{bmatrix} \equiv \begin{bmatrix} \mathbf{u}_s \\ \mathbf{u}_c \\ \mathbf{u}_f \end{bmatrix} + \begin{bmatrix} \mathbf{v}_s \\ \mathbf{v}_c \\ \mathbf{v}_f \end{bmatrix} \quad \text{and} \quad \begin{bmatrix} \ddot{\mathbf{U}}_s \\ \ddot{\mathbf{U}}_c \\ \ddot{\mathbf{U}}_f \end{bmatrix} \equiv \begin{bmatrix} \ddot{\mathbf{u}}_s \\ \ddot{\mathbf{u}}_c \\ \ddot{\mathbf{u}}_f \end{bmatrix} + \begin{bmatrix} \ddot{\mathbf{v}}_s \\ \ddot{\mathbf{v}}_c \\ \ddot{\mathbf{v}}_f \end{bmatrix} \tag{16.3}$$

Equation (16.1) can now be written as

$$\begin{gathered} \begin{bmatrix} \mathbf{M}_{ss} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{M}_{ff} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{u}}_s \\ \ddot{\mathbf{u}}_c \\ \ddot{\mathbf{u}}_f \end{bmatrix} + \begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sc} & \mathbf{0} \\ \mathbf{K}_{cs} & \mathbf{K}_{cc} & \mathbf{K}_{cf} \\ \mathbf{0} & \mathbf{K}_{fc} & \mathbf{K}_{ff} \end{bmatrix} \begin{bmatrix} \mathbf{u}_s \\ \mathbf{u}_c \\ \mathbf{u}_f \end{bmatrix} = \\ - \begin{bmatrix} \mathbf{M}_{ss} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{M}_{ff} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{v}}_s \\ \ddot{\mathbf{v}}_c \\ \ddot{\mathbf{v}}_f \end{bmatrix} - \begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sc} & \mathbf{0} \\ \mathbf{K}_{cs} & \mathbf{K}_{cc} & \mathbf{K}_{cf} \\ \mathbf{0} & \mathbf{K}_{fc} & \mathbf{K}_{ff} \end{bmatrix} \begin{bmatrix} \mathbf{v}_s \\ \mathbf{v}_c \\ \mathbf{v}_f \end{bmatrix} = \mathbf{R} \end{gathered} \tag{16.4}$$

If the free-field displacement $\mathbf{v}_c$ is constant over the base of the structure, the term $\mathbf{v}_s$ is the rigid body motion of the structure. Therefore, Equation (16.4) can be further simplified by the fact that the static rigid body motion of the structure is

$$\begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sc} \\ \mathbf{K}_{cs} & \mathbf{K}_{cc}^{(s)} \end{bmatrix} \begin{bmatrix} \mathbf{v}_s \\ \mathbf{v}_c \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix} \qquad (16.5)$$

Also, the dynamic free-field motion of the foundation requires that

$$\begin{bmatrix} \mathbf{M}_{cc}^{(f)} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{ff} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{v}}_c \\ \ddot{\mathbf{v}}_f \end{bmatrix} + \begin{bmatrix} \mathbf{K}_{cc}^{(f)} & \mathbf{K}_{cf} \\ \mathbf{K}_{cf} & \mathbf{K}_{ff} \end{bmatrix} \begin{bmatrix} \mathbf{v}_c \\ \mathbf{v}_f \end{bmatrix} = \begin{bmatrix} \mathbf{0} \\ \mathbf{0} \end{bmatrix} \qquad (16.6)$$

Therefore, the right-hand side of Equation (16.4) can be written as

$$\mathbf{R} = \begin{bmatrix} \mathbf{M}_{ss} & 0 & 0 \\ 0 & \mathbf{M}_{cc}^{(s)} & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{v}}_s \\ \ddot{\mathbf{v}}_c \\ 0 \end{bmatrix} \qquad (16.7)$$

Hence, the right-hand side of the Equation (16.4) does not contain the mass of the foundation. Therefore, three dimensional dynamic equilibrium equations, for the complete soil-structure system with damping added, are of the following form for a lumped mass system:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = -\mathbf{m}_x \ddot{\mathbf{v}}_x(t) - \mathbf{m}_y \ddot{\mathbf{v}}_y(t) - \mathbf{m}_z \ddot{\mathbf{v}}_z(t) \qquad (16.8)$$

where **M**, **C** and **K** are the mass, damping and stiffness matrices, respectively, of the soil-structure model. The added, relative displacements, $\underline{\mathbf{u}}$, exist for the soil-structure system and must be set to zero at the sides and bottom of the foundation. The terms $\ddot{\mathbf{v}}_x(t)$, $\ddot{\mathbf{v}}_y(t)$ and $\ddot{\mathbf{v}}_z(t)$ are the free-field components of the acceleration if the structure is not present. The column matrices, $\mathbf{m}_i$, are the directional masses for the added structure only.

Most structural analysis computer programs automatically apply the seismic loading to all mass degrees-of-freedom within the computer model and cannot solve the SSI problem. This lack of capability has motivated the development of the massless foundation model. This allows the correct seismic forces to be applied to the structure; however, the inertia forces within the foundation material are neglected. The results from a massless foundation analysis converge as the size of the foundation model is increased. However, the converged solutions may have avoidable errors in the mode shapes, frequencies and response of the system.

To activate the soil-structure interaction within a computer program it is only necessary to identify the foundation mass in order that the loading is not applied to that part of the structure. The program then has the required information to form both the total mass and the mass of the added structure. The SAP2000 program has this option and is capable of solving the SSI problem correctly.

16.4. RESPONSE DUE TO MULTI-SUPPORT INPUT MOTIONS

The previous SSI analysis assumes that the free-field motion at the base of the structure is constant. For large structures such as bridges and arch dams the free-field motion, at all points where the structure is in contact with the foundation, is not constant.

The approach normally used to solve this problem is to define a ***quasi-static displacement*** v_c that is calculated from the following equation:

$$\mathbf{K}_{ss}\mathbf{v}_s + \mathbf{K}_{sc}\mathbf{v}_c = \mathbf{0} \quad \text{or,} \quad \mathbf{v}_s = -\mathbf{K}_{ss}^{-1}\mathbf{K}_{sc}\mathbf{v}_c = \mathbf{T}_{sc}\mathbf{v}_c \tag{16.9a}$$

The transformation matrix $\mathbf{T}_{sc}$ allows the corresponding quasi-static acceleration in the structure to be calculated from

$$\ddot{\mathbf{v}}_s = \mathbf{T}_{sc}\ddot{\mathbf{v}}_c \tag{16.9b}$$

Equation (16.4) can be written as

$$\mathbf{R} = -\begin{bmatrix} \mathbf{M}_{ss} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{M}_{ff} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{v}}_s \\ \ddot{\mathbf{v}}_c \\ \ddot{\mathbf{v}}_f \end{bmatrix} - \begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sc} & \mathbf{0} \\ \mathbf{K}_{cs} & \mathbf{K}_{cc} & \mathbf{K}_{cf} \\ \mathbf{0} & \mathbf{K}_{fc} & \mathbf{K}_{ff} \end{bmatrix} \begin{bmatrix} \mathbf{v}_s \\ \mathbf{v}_c \\ \mathbf{v}_f \end{bmatrix} \tag{16.10}$$

After substitution of Equations (16.6) and (16.9), Equation (16.10) can be written as

$$\mathbf{R} = -\begin{bmatrix} \mathbf{0} & \mathbf{M}_{ss}\mathbf{T}_{sc} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{v}}_s \\ \ddot{\mathbf{v}}_c \\ \ddot{\mathbf{v}}_f \end{bmatrix} - \begin{bmatrix} \mathbf{0} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \overline{\mathbf{K}}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{v}_s \\ \mathbf{v}_c \\ \mathbf{v}_f \end{bmatrix} \tag{16.11}$$

The reduced structural stiffness at the contact surface $\overline{\mathbf{K}}_{cc}$ is given by

$$\overline{\mathbf{K}}_{cc} = \mathbf{K}_{cc} + \mathbf{K}_{cs}\mathbf{T}_{sc} \tag{16.12}$$

Therefore, this approach requires a special program option to calculate the mass and stiffness matrices to be used on the right-hand side of the dynamic equilibrium equations. Note that the loads are a function of both the free-field displacements and accelerations at the soil-structure contact. Also, in order to obtain the total stresses and displacements within the structure the quasi-static solution must be added to the solution. At the present time, there is not a general-purpose structural analysis computer program that is based on this "numerically cumbersome" approach.

An alternative approach is to formulate the solution directly in terms of the absolute displacements of the structure. This involves the introduction of the following change of variables:

$$\begin{bmatrix} \mathbf{U}_s \\ \mathbf{U}_c \\ \mathbf{U}_f \end{bmatrix} \equiv \begin{bmatrix} \mathbf{u}_s \\ \mathbf{u}_c \\ \mathbf{u}_f \end{bmatrix} + \begin{bmatrix} \mathbf{0} \\ \mathbf{v}_c \\ \mathbf{v}_f \end{bmatrix} \quad \text{and} \quad \begin{bmatrix} \ddot{\mathbf{U}}_s \\ \ddot{\mathbf{U}}_c \\ \ddot{\mathbf{U}}_f \end{bmatrix} \equiv \begin{bmatrix} \ddot{\mathbf{u}}_s \\ \ddot{\mathbf{u}}_c \\ \ddot{\mathbf{u}}_f \end{bmatrix} + \begin{bmatrix} \mathbf{0} \\ \ddot{\mathbf{v}}_c \\ \ddot{\mathbf{v}}_f \end{bmatrix} \tag{16.13}$$

Substitution of this change of variables into Equation (16.1) yields the following dynamic equilibrium equations in terms of the absolute displacement, $\mathbf{u}_s$, of the structure:

$$\begin{bmatrix} \mathbf{M}_{ss} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{M}_{ff} \end{bmatrix} \begin{bmatrix} \ddot{\mathbf{u}}_s \\ \ddot{\mathbf{u}}_c \\ \ddot{\mathbf{u}}_f \end{bmatrix} + \begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sf} & \mathbf{0} \\ \mathbf{K}_{cf} & \mathbf{K}_{cc} & \mathbf{K}_{cf} \\ \mathbf{0} & \mathbf{K}_{fc} & \mathbf{K}_{ff} \end{bmatrix} \begin{bmatrix} \mathbf{u}_s \\ \mathbf{u}_c \\ \mathbf{u}_f \end{bmatrix} = \mathbf{R} \tag{16.14}$$

After the free-field response, Equation (16.6), is removed the dynamic loading is calculated from the following equation:

$$\mathbf{R} = -\begin{bmatrix} \mathbf{K}_{ss} & \mathbf{K}_{sc} & \mathbf{0} \\ \mathbf{K}_{cs} & \mathbf{K}_{cc}^{(s)} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{0} \\ \mathbf{v}_c \\ \mathbf{0} \end{bmatrix} - \begin{bmatrix} \mathbf{M}_{ss} & \mathbf{0} & \mathbf{0} \\ \mathbf{0} & \mathbf{M}_{cc}^{(s)} & \mathbf{0} \\ \mathbf{0} & \mathbf{0} & \mathbf{0} \end{bmatrix} \begin{bmatrix} \mathbf{0} \\ \ddot{\mathbf{v}}_c \\ \mathbf{0} \end{bmatrix} \tag{16.15a}$$

This equation can be further simplified by connecting the structure to the foundation with stiff massless springs that are considered as part of the structure. Therefore, the mass of the structure at the contact nodes is eliminated and Equation (16.15a) is reduced to

$$\mathbf{R} = -\begin{bmatrix} \mathbf{K}_{sc} \\ \mathbf{K}_{cc}^{(s)} \\ \mathbf{0} \end{bmatrix} \left[\mathbf{v}_c\right] \tag{16.15b}$$

It is apparent that the stiffness terms in Equation (16.15b) represent the stiffness of the contact springs only. Therefore, for a typical displacement component (n = x, y or z), the forces acting at point "i" on the structure and point "j" on the foundation are given by

$$\begin{bmatrix} R_i \\ R_j \end{bmatrix}_n = -k_n \begin{bmatrix} 1.0 & -1.0 \\ -1.0 & 1.0 \end{bmatrix} \begin{bmatrix} 0 \\ v_n \end{bmatrix} \tag{16.16}$$

where k_n is the massless spring stiffness in the nth direction and v_n is the free-field displacement. Hence, points "i" and "j" can be at the same location in space and the only loads acting are a series of time-dependent, concentrated, point loads that are equal and opposite forces between the structure and foundation. The spring stiffness must be selected approximately three orders-of-magnitude greater than the stiffness of the structure at the connecting nodes. The spring stiffness should be large enough so the fundamental periods of the system are not changed. and small enough not to cause numerical problems.

The dynamic equilibrium equations, with damping added, can be written in the following form:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{R} \tag{16.17}$$

It should be pointed out that concentrated dynamic loads generally require a large number of eigenvectors in order to capture the correct response of the system. However, if LDR vectors are used, in a mode superposition analysis, the required number of vectors is reduced significantly. The SAP2000 program has the ability to solve the multi-support, soil-structure interaction problems using this approach. At the same time, selective nonlinear behavior of the structure can be considered.

16.5. ANALYSIS OF GRAVITY DAM AND FOUNDATION

In order to illustrate the use of the soil-structure interaction option several earthquake response analyses of the Pine Flat Dam were conducted with different foundation models. The foundation properties were assumed to be the same properties as the dam. Damping was set at five percent. Ten Ritz vectors, generated from loads on the dam only, were used. However, the resulting approximate mode shapes, used in the standard mode superposition analysis, included the mass inertia effects of the foundation. The horizontal dynamic loading was the typical segment of the Loma Prieta earthquake defined in Figure 15.1a. A finite element model of the dam on a rigid foundation is shown in Figure 16.2.

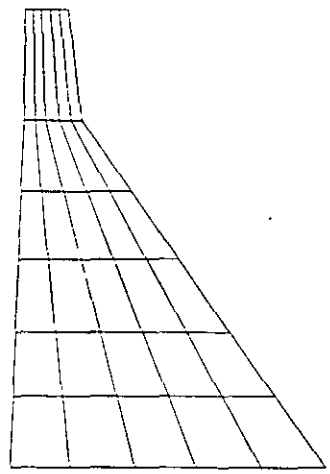


Figure 16.2. Finite Element Model of Dam only

The two different foundation models used are shown in Figure 16.3.

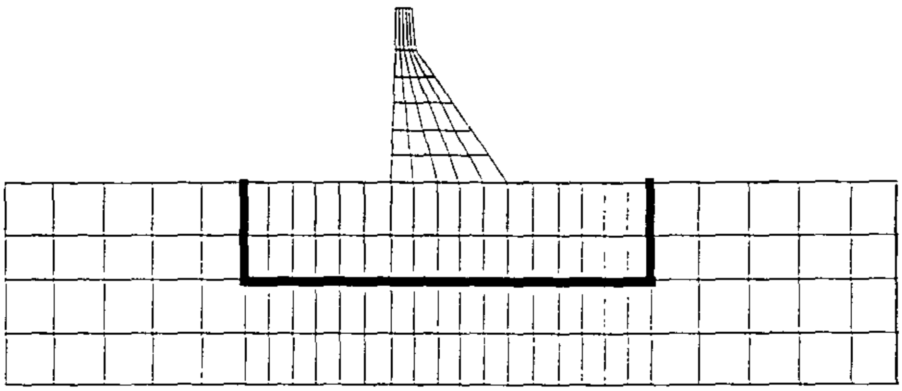


Figure 16.3. Models of Dam with Small and Large Foundation

Selective results are summarized in Table 16.1. For the purpose of comparison, it will be assumed that Ritz vector results, for the large foundation mesh, are the referenced values.

Table 16.1. Selective Results Of Dam-Foundation Analyses

	DAM WITH NO Foundation	SMALL Foundation	LARGE Foundation
TOTAL MASS lb-sec^2/in	1,870	13,250	77,360
PERIODS seconds	0.335 0.158	0.404 0.210	0.455 0.371
Max. Displacement inches	0.65	1.28	1.31
Max & Min Stress ksi	-37 to +383	-490 to +289	-512 to +297

The differences between the results of the small and large foundation models are very close which indicates that the solution of the large foundation model may be nearly converged. It is true that the radiation damping effects in a finite foundation model are neglected. However, as the foundation model becomes larger, the energy dissipation due to normal modal damping within the massive foundation is significantly larger than the effects of radiation damping for transient earthquake type of loading.

16.6. THE MASSLESS FOUNDATION APPROXIMATION

Most general purpose programs for the earthquake analysis of structures do not have the option of identifying the foundation mass as a separate type of mass on which the earthquake forces do not act. Therefore, an approximation that has commonly been used is to neglect the mass of the foundation completely in the analysis. Table 16.2 summarizes the results for an analysis of the same dam-foundation systems using a massless foundation. As expected, these results are similar. For this case the results are conservative; however, one cannot be assured of this for all cases.

Table 16.2. Selective Results Of Dam With Massless Foundation Analyses

	DAM WITH NO Foundation	SMALL Foundation	LARGE Foundation
TOTAL MASS lb-sec^2/in	1,870	1,870	1,870
PERIODS seconds	0.335 0.158	0.400 0.195	0.415 0.207
Max. Displacement inches	0.65	1.27	1.43
Max & Min Stress ksi	-37 to +383	-480 to +289	-550 to +330

16.7. APPROXIMATE RADIATION BOUNDARY CONDITIONS

If the foundation volume is large and the modal damping exists, it was demonstrated in the previous section that a finite foundation with fixed boundaries can produce converged results. However, the use of energy absorbing boundaries can further reduce the size of the foundation required to produce a converged solution.

In order to calculate the properties of this boundary condition consider a plane wave propagating in the x-direction. The forces, which cause wave propagation, are shown acting on a unit cube in Figure 16.4.

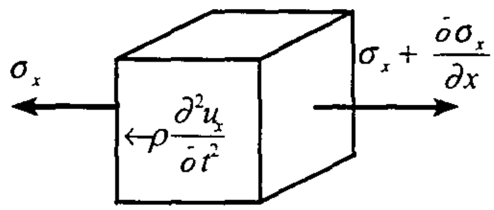


Figure 16.4. Forces Acting on Unit Cube

From Figure 16.4 the one dimensional equilibrium equation in the x-direction is

$$\rho \frac{\partial^2 u}{\partial t^2} - \frac{\partial \sigma_x}{\partial x} = 0 \tag{16.18}$$

Since $\sigma_x = \lambda \varepsilon_x = \lambda \frac{\partial u}{\partial x}$ the one dimensional partial differential equation is written in the following classical wave propagation form:

$$\frac{\partial^2 u}{\partial t^2} - V_p^2 \frac{\partial^2 u_x}{\partial x^2} = 0 \tag{16.19}$$

where V_p is the wave propagation velocity of the material and is given by

$$V_p = \sqrt{\frac{\lambda}{\rho}} \tag{16.20}$$

in which ρ is the mass density and λ is the bulk modulus given by

$$\lambda = \frac{1-\nu}{(1+\nu)(1-2\nu)} E \tag{16.21}$$

The solution of Equation (16.13), for harmonic wave propagation in the positive x-direction, is a displacement of the following form:

$$u(t,x) = U[\sin(\omega t - \frac{\omega x}{V_p}) + \cos(\omega t - \frac{\omega x}{V_p})] \tag{16.22}$$

This equation can be easily verified by substitution into Equation (16.18). The arbitrary frequency of the harmonic motion is ω. The velocity, $\frac{\partial u}{\partial t}$, of a particle at location x is

$$\dot{u}(t,x) = U\omega\,[\cos(\omega t - \frac{\omega x}{V_p}) - \sin(\omega t - \frac{\omega x}{V_p})] \tag{16.23}$$

The strain in the x-direction is

$$\varepsilon(x,t) = \frac{\partial u}{\partial x} = -\frac{\dot{u}(x,t)}{V_p} \tag{16.24}$$

The corresponding stress can now be expressed in the following simplified form:

$$\sigma(x,t) = \lambda\varepsilon(x,t) = -V_p\,\rho\,\dot{u}(x,t) \tag{16.25}$$

The compression stress is identical to the force in a simple viscous damper with constant damping value equal to $V_p\,\rho$ per unit area of the boundary. Therefore, a boundary condition can be created, at a cut boundary, which will allow the wave to pass without reflection and allow the strain energy to "radiate" away from the foundation.

Also, it can be easily shown that the shear wave "radiation" boundary condition, parallel to a free boundary, is satisfied if damping values are assigned to be $V_S\rho$ per unit of boundary area. The shear wave propagation velocity is given by

$$V_s = \sqrt{\frac{G}{\rho}} \tag{16.26}$$

where G is the shear modulus.

The FNA method can be used to solve structures, in the time domain, with these types of boundary conditions. In later editions of this book, the accuracy of these boundary conditions approximation will be illustrated with numerical examples. Also, it will be used with a fluid boundary where only compression waves exist.

16.8. USE OF SPRINGS AT THE BASE OF A STRUCTURE

Another important structural modeling problem, which must be solved, is at the interface of the major structural elements within a structure and the foundation material. For example, the deformations at the base of a major shear wall in a building structure will significantly affect the displacement and force distribution in the upper stories of a building for both static and dynamic loads. Realistic spring stiffness can be selected from separate finite element studies or by using the classical half-space equations that are given in Table 16.3.

It is the opinion of the author that the use of appropriate site-dependent free-field earthquake motions and selection of realistic massless springs at the base of the structure are the only modeling assumptions required to include site and foundation properties in the earthquake analysis of most structural systems.

Table 16.3 also contains effective mass and damping factors that include the approximate effects of radiation damping. These values can be used directly in a computer model without any difficulty. However, considerable care should be taken in using these equations at the base of a complete structure. For example, the effective earthquake forces must not be applied to the foundation mass.

Table 16.3. Properties Of Rigid Circular Plate On Surface Of Half-Space

DIRECTION	STIFFNESS	DAMPING	MASS
Vertical	$K = \dfrac{4Gr}{1-\nu}$	$1.79\sqrt{K\rho r^3}$	$1.50\rho r^3$
Horizontal	$18.2Gr\dfrac{(1-\nu^2)}{(2-\nu)^2}$	$1.08\sqrt{K\rho r^3}$	$0.28\rho r^3$
Rotation	$2.7Gr^3$	$0.47\sqrt{K\rho r^3}$	$0.49\rho r^5$
Torsion	$5.3Gr^3$	$1.11\sqrt{K\rho r^5}$	$0.70\rho r^5$

r = plate radius; G = shear modulus; ν = Poisson's ratio; ρ = mass density

Source: Adapted from "Fundamentals of Earthquake Engineering, by Newmark and Rosenblueth, Prentice-Hall, 1971

16.9. SUMMARY

A large number of research papers and several books have been written on structure-foundation-soil analysis and site response due to earthquake loading. However, the majority of these publications have been restricted to the linear behavior of soil-structure systems. It is possible, with the use of the numerical methods presented in this book, to conduct accurate earthquake analysis of real soil-structure systems in the time domain, including many realistic nonlinear properties. Also, it can be demonstrated that the solution obtained is converged to the correct soil-structure interactive solution.

For major structures on soft soil one dimensional site response analyses should be conducted. Under major structural elements, such as the base of a shear wall, massless elastic springs should be used to estimate the foundation stiffness. For massive structures, such as gravity dams, a part of the foundation should be modeled by three dimensional SOLID elements in which SSI effects are included.

16.10. REFERENCES

1. "SHAKE - A Computer Program for the Earthquake Response for Horizontally Layered Sites", by P. Schnabel, J. Lysmer and H. Seed, EERC Report No. 72-2, University of California, Berkeley, February 1970.

2. J. Hart. and E. Wilson "WAVES - An Efficient Microcomputer Program for Nonlinear Site Response Analysis", National Information Center for Earthquake Engineering, Davis Hall, University of California, Berkeley, Tel. # (415) 642-5113.

3. R. Clough, and J. Penzien, *Dynamics of Structures*, Second Edition, McGraw-Hill, Inc., ISBN 0-07-011394-7, 1993.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

SEISMIC ANALYSIS MODELING TO SATISFY BUILDING CODES

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

SEISMIC ANALYSIS MODELING TO SATISFY BUILDING CODES

The Current Building Codes Use the Terminology Principal Direction without A Unique Definition

17.1. INTRODUCTION

Currently a three-dimensional dynamic analysis is required for a large number of different types of structural systems that are constructed in Seismic Zones 2, 3 and 4 [1]. The lateral force requirements suggest several methods that can be used to determine the distribution of seismic forces within a structure. However, these guidelines are not unique and need further interpretations.

The major advantage of using the forces obtained from a dynamic analysis as the basis for a structural design is that the vertical distribution of forces may be significantly different from the forces obtained from an equivalent static load analysis. Consequently, the use of dynamic analysis will produce structural designs that are more earthquake resistant than structures designed using static loads.

For many years, approximate two-dimensional static load was acceptable as the basis for seismic design in many geographical areas and for most types of structural systems. During the past twenty years, due to the increasing availability of modern digital computers, most engineers have had experience with the static load analysis of three dimensional structures. However, few engineers, and the writers of the current building code, have had experience with the three dimensional dynamic response analysis. Therefore, the interpretation of the dynamic analysis requirement of the current code represents a new challenge to most structural engineers.

The current code allows the results obtained from a dynamic analysis to be normalized so that the maximum dynamic base shear is equal to the base shear obtained from a simple two-dimensional static load analysis. Most members of the profession realize that there is no theoretical foundation for this approach. However, for the purpose of selecting the magnitude of the dynamic loading that will satisfy the code requirements, this approach can be accepted, in a modified form, until a more rational method is adopted.

The calculation of the "design base shears" is simple and the variables are defined in the code. It is of interest to note, however, that the basic magnitude of the seismic loads has not changed significantly from previous codes. The major change is that "dynamic methods of analysis" must be used in the "principal directions" of the structure. The present code does not state how to define the principal directions for a three dimensional structure of arbitrary geometric shape. Since the design base shear can be different in each direction, this "scaled spectra" approach can produce a different input motion for each direction, for both regular and irregular structures. Therefore, ***the current code dynamic analysis approach can result in a structural design which is relatively "weak" in one direction***. The method of dynamic analysis proposed in this chapter results in a structural design that has equal resistance in all directions.

In addition, the maximum possible design base shear, which is defined by the present code, is approximately 35 percent of the weight of the structure. For many structures, it is less than 10 percent. It is generally recognized that this force level is small when compared to measured earthquake forces. Therefore, the use of this design base shear requires that substantial ductility be designed into the structure.

The definition of an irregular structure, the scaling of the dynamic base shears to the static base shears for each direction, the application of accidental torsional loads and the treatment of orthogonal loading effects are areas which are not clearly defined in the current building code. The purpose of this section is to present one method of three dimensional seismic analysis that will satisfy the Lateral Force Requirements of the code. The method is based on the response spectral shapes defined in the code and previously published and accepted computational procedures.

17.2. THREE DIMENSIONAL COMPUTER MODEL

Real and accidental torsional effects must be considered for all structures. Therefore, all structures must be treated as three dimensional systems. Structures with irregular plans, vertical setbacks or soft stories will cause no additional problems if a realistic three dimensional computer model is created. This model should be developed in the very early stages of design since it can be used for static wind and vertical loads, as well as dynamic seismic loads.

Only structural elements with significant stiffness and ductility should be modeled. Non-structural brittle components can be neglected. However, shearing, axial deformations and non-center line dimensions can be considered in all members without a significant increase in computational effort by most modern computer programs. The rigid, in-plane approximation of floor systems has been shown to be acceptable for most buildings. For the purpose of elastic dynamic analysis, gross concrete sections, neglecting the stiffness of the steel, are normally used. A cracked section mode should be used to check the final design.

The P-Delta effects should be included in all structural models. It has been shown in Chapter 11 that these second order effects can be considered, without iteration, for both static and dynamic loads. The effect of including P-Delta displacements in a dynamic analysis results in a small increase in the period of all modes. In addition to being more accurate, an additional advantage of automatically including P-Delta effects is that the moment magnification factor for all members can be taken as unity in all subsequent stress checks.

The mass of the structure can be estimated with a high degree of accuracy. The major assumption required is to estimate the amount of live load to be included as added mass. For certain types of structures it may be necessary to conduct several analyses with different values of mass. The lumped mass approximation has proven to be accurate. In the case of the rigid diaphragm approximation, the rotational mass moment of inertia must be calculated.

The stiffness of the foundation region of most structures can be modeled by massless structural elements. It is particularly important to model the stiffness of piles and the rotational stiffness at the base of shear walls.

The computer model for static loads only should be executed prior to conducting a dynamic analysis. Equilibrium can be checked and various modeling approximations can be verified with simple static load patterns. The results of a dynamic analysis are generally very complex and the forces obtained from a response spectra analysis are always positive. Therefore, dynamic equilibrium is almost impossible to check. However, it is relatively simple to check energy balances in both linear and nonlinear analysis.

17.3. THREE DIMENSIONAL MODE SHAPES AND FREQUENCIES

The first step in the dynamic analysis of a structural model is the calculation of the three dimensional mode shapes and natural frequencies of vibration. Within the past several years, very efficient computational methods have been developed which have greatly decreased the computational requirements associated with the calculation of orthogonal shape functions as presented in Chapter 14. It has been demonstrated that load-dependent Ritz vectors, which can be generated with a minimum of numerical effort, produce more accurate results when used for a seismic dynamic analysis than if the exact free-vibration mode shapes are used.

Therefore, a dynamic response spectra analysis can be conducted with approximately twice the computer time requirements of a static load analysis. Since systems with over 60,000 dynamic degrees-of-freedom can be solved within a few hours on personal computers, there is not a significant increase in cost between a static and a dynamic analysis. The major cost is the "man hours" required to produce the three dimensional computer model that is necessary for a static or a dynamic analysis.

In order to illustrate the dynamic properties of the three dimensional structure, the mode shapes and frequencies are calculated for the irregular, eight story, 80 foot tall building shown in Figure 17.1. This building is a concrete structure with several hundred degrees-of-freedom. However, the three components of mass are lumped at each of the eight floor levels. Therefore, only 24 three dimensional mode shapes are possible.

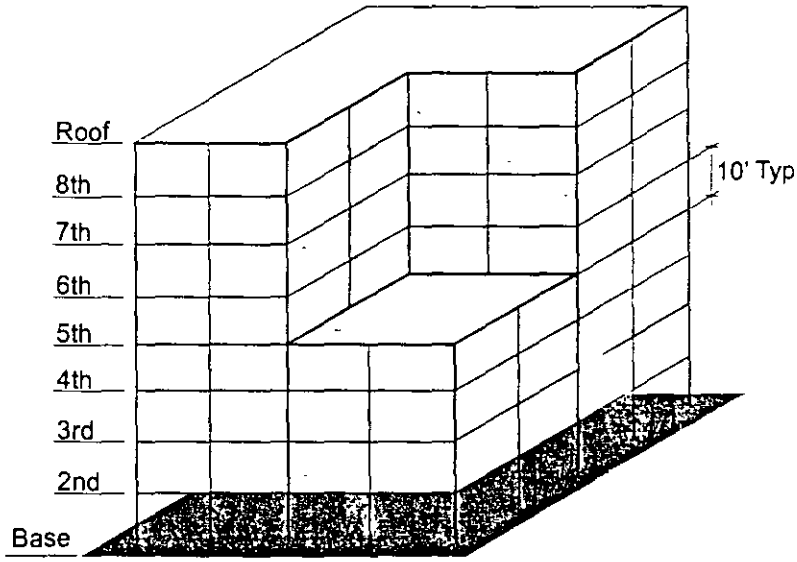


Figure 17.1. Example of Eight Story Irregular Building

Each three-dimensional mode shape of a structure may have displacement components in all directions. For the special case of a symmetrical structure, the mode shapes are uncoupled and will have displacement in one direction only. Since each mode can be considered to be a deflection due to a set of static loads, six base reaction forces can be calculated for each mode shape. For the structure shown in Figure 17.1, Table 17.1 summarizes the two base reactions and three overturning moments associated with each mode shape. Since vertical mass has been neglected there is no vertical reaction. The magnitudes of the forces and moments have no meaning since the amplitude of a mode shape can be normalized to any value. However, the relative values of the different components of the shears and moments associated with each mode are of considerable value. The modes with a large torsional component are highlighted in **bold**.

Table 17.1. Three Dimensional Base Forces and Moments

MODE	PERIOD	MODAL BASE SHEAR REACTIONS			MODAL OVERTURNING MOMENTS		
	Seconds	X-DIR	Y-DIR	Angle Deg.	X-AXIS	Y-AXIS	Z-AXIS
1	.6315	.781	.624	38.64	-37.3	46.6	-18.9
2	.6034	-.624	.781	-51.37	-46.3	-37.0	38.3
3	**.3501**	**.785**	**.620**	**38.30**	**-31.9**	**40.2**	**85.6**
4	.1144	-.753	-.658	41.12	12.0	-13.7	7.2
5	.1135	.657	-.754	-48.89	13.6	11.9	-38.7
6	**.0706**	**.989**	**.147**	**8.43**	**-33.5**	**51.9**	**2438.3**
7	.0394	-.191	.982	-79.01	-10.4	-2.0	29.4
8	.0394	-.983	-.185	10.67	1.9	-10.4	26.9
9	**.0242**	**.848**	**.530**	**32.01**	**-5.6**	**8.5**	**277.9**
10	.0210	.739	.673	42.32	-5.3	5.8	-3.8
11	.0209	.672	-.740	-47.76	5.8	5.2	-39.0
12	**.0130**	**-.579**	**.815**	**-54.63**	**-.8**	**-8.8**	**-1391.9**
13	.0122	.683	.730	46.89	-4.4	4.1	-6.1
14	.0122	.730	-.683	-43.10	4.1	4.4	-40.2
15	.0087	-.132	-.991	82.40	5.2	-.7	-22.8
16	.0087	-.991	.135	-7.76	-.7	-5.2	30.8
17	**.0074**	**-.724**	**-.690**	**43.64**	**4.0**	**-4.2**	**-252.4**
18	.0063	-.745	-.667	41.86	3.1	-3.5	7.8
19	.0062	-.667	.745	-48.14	-3.5	-3.1	38.5
20	.0056	-.776	-.630	39.09	2.8	-3.4	54.1
21	.0055	-.630	.777	-50.96	-3.4	-2.8	38.6
22	.0052	.776	.631	39.15	-2.9	3.5	66.9
23	**.0038**	**-.766**	**-.643**	**40.02**	**3.0**	**-3.6**	**-323.4**
24	**.0034**	**-.771**	**-.637**	**39.58**	**2.9**	**-3.5**	**-436.7**

A careful examination of the directional properties of the three dimensional mode shapes at the early stages of a preliminary design can give a structural engineer additional information which can be used to improve the earthquake resistant design of a structure. The current code defines an "irregular structure" as one which has a certain geometric shape or in which stiffness and mass discontinuities exist. A far more rational definition is that a "regular structure" is one in which there is a minimum coupling between the lateral displacements and the torsional rotations for the mode shapes associated with the lower frequencies of the system. Therefore, if the model is modified and "tuned" by studying the three dimensional mode shapes during the preliminary design phase, it may be possible to convert a "geometrically irregular" structure to a "dynamically regular" structure from an earthquake-resistant design standpoint.

Table 17.2. Three Dimensional Participating Mass - (percent)

MODE	X-DIR	Y-DIR	Z-DIR	X-SUM	Y-SUM	Z-SUM
1	34.224	21.875	.000	34.224	21.875	.000
2	23.126	36.212	.000	57.350	58.087	.000
3	2.003	1.249	.000	59.354	59.336	.000
4	13.106	9.987	.000	72.460	69.323	.000
5	9.974	13.102	.000	82.434	82.425	.000
6	.002	.000	.000	82.436	82.425	.000
7	.293	17.770	.000	82.729	90.194	.000
8	7.726	.274	.000	90.455	90.469	.000
9	.039	.015	.000	90.494	90.484	.000
10	2.382	1.974	.000	92.876	92.458	.000
11	1.955	2.370	.000	94.831	94.828	.000
12	.000	.001	.000	94.831	94.829	.000
13	1.113	1.271	.000	95.945	96.100	.000
14	1.276	1.117	.000	97.220	97.217	.000
15	.028	1.556	.000	97.248	98.773	.000
16	1.555	.029	.000	98.803	98.802	.000
17	.011	.010	.000	98.814	98.812	.000
18	.503	.403	.000	99.316	99.215	.000
19	.405	.505	.000	99.722	99.720	.000
20	.102	.067	.000	99.824	99.787	.000
21	.111	.169	.000	99.935	99.957	.000
22	.062	.041	.000	99.997	99.998	.000
23	.003	.002	.000	100.000	100.000	.000
24	.001	.000	.000	100.000	100.000	.000

For this building, it is of interest to note that the mode shapes, which tend to have directions that are 90 degrees apart, have almost the same value for their period. This is typical of three dimensional mode shapes for both regular and irregular buildings. For regular symmetric structures, which have equal stiffness in all directions, the periods associated with the lateral displacements will result in pairs of identical periods. However, the directions associated with the pair of three dimensional mode shapes are not mathematically unique. For identical periods, most computer programs allow round-off errors to produce two mode shapes with directions which differ by 90 degrees. Therefore, the SRSS method should not be used to combine modal maximums in three dimensional dynamic analysis. The CQC method eliminates problems associated with closely spaced periods.

For a response spectrum analysis, the current code states that "at least 90 percent of the participating mass of the structure must be included in the calculation of response for each principal direction." Therefore, the number of modes to be evaluated must satisfy this requirement. Most computer programs automatically calculate the participating mass in all directions using the equations presented in Chapter 13. This requirement can be easily satisfied using LDR vectors. For the structure shown in Figure 17.1, the participating mass for each mode and for each direction is shown in Table 17.2. For this building, only eight modes are required to satisfy the 90 percent specification in both the x and y directions.

17.4. THREE DIMENSIONAL DYNAMIC ANALYSIS

It is possible to conduct a dynamic, time-history, response analysis by either the mode superposition or step-by-step methods of analysis. However, a standard time-history ground motion, for the purpose of design, has not been defined. Therefore, most engineers use the response spectrum method of analysis as the basic approach. The first step in a response spectrum analysis is the calculation of the three dimensional mode shapes and frequencies as indicated in the previous section.

17.4.1. Dynamic Design Base Shear

For dynamic analysis, the 1994 UBC requires that the "design base shear", **V**, is to be evaluated from the following formula:

$$\mathbf{V} = [\, \mathbf{Z\,I\,C} / \mathbf{R_W} \,]\, \mathbf{W} \tag{17.1}$$

Where

$\mathbf{Z}$ = Seismic zone factor given in Table 16-I.

$\mathbf{I}$ = Importance factor given in Table 16-K.

$\mathbf{R_W}$ = Numerical coefficient given in Table 16-N or 16-P.

$\mathbf{W}$ = The total seismic weight of the structure.

$\mathbf{C}$ = Numerical coefficient (2.75 maximum value) determined from:

$$\mathbf{C = 1.25\,S/\,T^{2/3}} \tag{1-2}$$

Where

$\mathbf{S}$ = Site coefficient for soil characteristics given in Table 16-J.

$\mathbf{T}$ = Fundamental period of vibration (seconds).

The period, **T**, determined from the three dimensional computer model, can be used for most cases. This is essentially Method B of the code.

Since the computer model often neglects nonstructural stiffness, the code requires that Method A be used under certain conditions. Method A defines the period, **T**, as follows:

$$\mathbf{T = C_t\,h^{3/4}} \tag{1-3}$$

where **h** is the height of the structure in feet and $\mathbf{C_t}$ is defined by the code for various types of structural systems.

The Period calculated by Method B cannot be taken as more than 30% longer than that computed using Method A in Seismic Zone 4 and more than 40% longer in Seismic Zones 1, 2 and 3.

For a structure that is defined by the code as "regular", the design base shear may be reduced by an additional 10 percent. However, it must not be less than 80 percent of the shear calculated using Method A. For an "irregular" structure this reduction is not allowed.

17.4.2. Definition of Principal Directions

A weakness in the current code is the lack of definition of the "principal horizontal directions" for a general three dimensional structure. If each engineer is allowed to select an arbitrary reference system, the "dynamic base shear" will not be unique and each reference system could result in a different design. One solution to this problem, that will result in a unique design base shear, is to use the direction of the base shear associated with the fundamental mode of vibration as the definition of the "major principal direction" for the structure. The "minor principal direction" will be, by definition, ninety degrees from the major axis. This approach has some rational basis since it is valid for regular structures. Therefore, this definition of the principal directions will be used for the method of analysis presented in this chapter.

17.4.3. Directional and Orthogonal Effects

The required design seismic forces may come from any horizontal direction and, for the purpose of design, they may be assumed to act non-concurrently in the direction of each principal axis of the structure. In addition, for the purpose of member design, the effects of seismic loading in two orthogonal directions may be combined on a square-root-of-the-sum-of-the-squares (SRSS) basis. (Also, it is allowable to design members for 100 percent of the seismic forces in one direction plus 30 percent of the forces produced by the loading in the other direction. We will not use this approach in the procedure suggested here for reasons presented in Chapter 15.)

17.4.4. Basic Method of Seismic Analysis

In order to satisfy the current requirements, it is necessary to conduct two separate spectrum analyses in the major and minor principal directions (as defined above). Within each of these analyses, the Complete Quadratic Combination (CQC) method is used to accurately account for modal interaction effects in the estimation of the maximum response values. The spectra used in both of these analyses can be obtained directly from the Normalized Response Spectra Shapes given by the Uniform Building Code.

17.4.5. Scaling of Results

Each of these analyses will produce a base shear in the major principal direction. A single value for the "dynamic base shear" is calculated by the SRSS method. Also, a "dynamic base shear" can be calculated in the minor principal direction. The next step is to scale the previously used spectra shapes by the ratio of "design base shear" to the minimum value of the "dynamic base shear". This approach is more conservative than proposed by the current requirements, since only the scaling factor that produces the largest response is used. However, this approach is far more rational since it results in the same design earthquake in all directions.

17.4.6. Dynamic Displacements and Member Forces

The displacement and force distribution are calculated using the basic SRSS method to combine the results from 100 percent of the scaled spectra applied in each direction. If two analyses are conducted in any two orthogonal directions, in which the CQC method is used to combine the modal maximums for each analysis, and the results are combined by the SRSS method, exactly the same results will be obtained regardless of the orientation of the orthogonal reference system. Therefore, the direction of the base shear of the first mode defines a reference system for the building.

If site-specific spectra are given, for which scaling is not required, any orthogonal reference system can be used. In either case, only one computer run is necessary to calculate all member forces to be used for design.

17.4.7. Torsional Effects

Possible torsional ground motion, the unpredictable distribution of live load mass and the variations of structural properties are three reasons why both regular and irregular structures must be designed for accidental torsional loads. Also, for a regular structure lateral loads do not excite torsional modes. One method suggested in the Code is to conduct several different dynamic analyses with the mass at different locations. This approach is not practical since the basic dynamic properties of the structure (and the dynamic base shears) would be different for each analysis. In addition, the selection of the maximum member design forces would be a monumental post-processing problem.

The current Code allows the use of pure static torsional loads to predict the additional design forces caused by accidental torsion. The basic vertical distribution of lateral static loads is given by the Code equations. The static torsional moment at any level is calculated by the multiplication of the static load at that level by 5 percent of the maximum dimension at that level. In this book it is recommended that these pure torsional static loads, applied at the center of mass at each level, be used as the basic approach to account for accidental torsional loads. This static torsional load is treated as a separate load condition so that it can be appropriately combined with the other static and dynamic loads.

17.5. NUMERICAL EXAMPLE

To illustrate the base-shear scaling method recommended here, a static seismic analysis is conducted on the building shown in Figure 17.1. The eight-story building has 10 feet story heights. The seismic dead load is 238.3 kips for the top four stories and 363.9 kips for the lower four stories. For I = 1, Z = 0.4, S = 1.0, and R_W = 6.0, the evaluation of Equation 17.1 yields the design base forces given in Table 17.3.Table 17.3. Static Design Base Forces Using The Uniform Building Code

Period (sec)	Angle (deg)	Base Shear	Overturning Moment
0.631	38.64	279.9	14,533
0.603	-51.36	281.2	14,979

The normalized response spectra shape for soil type 1, which is defined in the Uniform Building Code, is used as the basic loading for the three dimensional dynamic analyses. Using eight modes only and the SRSS method of combining modal maxima, the base shears and overturning moments are summarized in Table 17.4 for various directions of loading.

Table 17.4. Dynamic Base Forces Using The SRSS Method

	BASE SHEARS		OVERTURNING MOMENTS	
Angle -deg	V_1	V_2	M_1	M_2
0	58.0	55.9	2982	3073
90	59.8	55.9	2983	3185
38.64	70.1	5.4	66	4135
-51.36	83.9	5.4	66	4500

The 1-axis is in the direction of the seismic input and the 2-axis is normal to the direction of the loading. This example clearly illustrates the major weakness of the SRSS method of modal combination. Unless the input is in the direction of the fundamental mode shapes, a large base shear is developed normal to the direction of the input and the dynamic base shear in the direction of the input is significantly underestimated as illustrated in Chapter 15.

As indicated by Table 17.5, the CQC method of modal combination eliminates problems associated with the SRSS method. Also, it clearly illustrates that the directions of 38.64 and -51.36 degrees are a good definition of the principal directions for this structure. Note that the directions of the base shears of the first two modes differ by 90.00 degrees.

Table 17.5. Dynamic Base Forces Using The CQC Method

	BASE SHEARS		OVERTURNING MOMENTS	
Angle -deg	V_1	V_2	M_1	M_2
0	78.1	20.4	1202	4116
90	79.4	20.4	1202	4199
38.64	78.5	0.2	3.4	4145
-51.36	84.2	0.2	3.4	4503

Table 17.6 summarizes the scaled dynamic base forces to be used as the basis for design by two different methods.

Table 17.6 Normalized Base Forces In Principal Directions

	38.64 Degrees		-51.36 Degrees	
	V (kips)	M(ft-kips)	V (kips)	M(ft-kips)
Static Code Forces	279.9	14,533	281.2	14,979
Dynamic Design Forces Scaled by Base Shear 279.9/78.5 = 3.57	279.9	14,732	299.2	16,004

For this case, the input spectra scale factor of 3.57 should be used for all directions and is based on the fact that both the dynamic base shears and the dynamic overturning moments must not be less than the static code forces. This approach is clearly more conservative than the approach suggested by the current Uniform Building Code. It is apparent that the use of different scale factors for a design spectra in the two different directions, as allowed by the code, results in a design that has a weak direction relative to the other principle direction.

17.6. DYNAMIC ANALYSIS METHOD SUMMARY

In this section, a dynamic analysis method is summarized that produces unique design displacements and member forces which will satisfy the current Uniform Building Code. It can be used for both regular and irregular structures. The major steps in the approach are as follows:

1. A three dimensional computer model must be created in which all significant structural elements are modeled. This model should be used in the early phases of design since it can be used for both static and dynamic loads.

2. The three dimensional mode shapes should be repeatedly evaluated during the design of the structure. The directional and torsional properties of the mode shapes can be used to improve the design. A well-designed structure should have a minimum amount of torsion in the mode shapes associated with the lower frequencies of the structure.

3. The direction of the base reaction of the mode shape associated with the fundamental frequency of the system is used to define the principal directions of the three dimensional structure.

4. The "design base shear" is based on the longest period obtained from the computer model, except when limited to 1.3 or 1.4 times the Method A calculated period.

5. Using the CQC method, the "dynamic base shears" are calculated in each principal direction due to 100 percent of the Normalized Spectra Shapes. Use the minimum value of the base shear in the principal directions to produce one "scaled design spectra".

6. The dynamic displacements and member forces are calculated using the SRSS value of 100 percent of the scaled design spectra applied non-concurrently in any two orthogonal directions as presented in Chapter 15.

7. A pure torsion static load condition is produced using the suggested vertical lateral load distribution defined in the code.

8. The member design forces are calculated using the following load combination rule:

$$F_{DESIGN} = F_{DEAD\ LOAD} \pm [\, F_{DYNAMIC} + |\, F_{TORSION} \,| \,] + F_{OTHER}$$

The dynamic forces are always positive and the accidental torsional forces must always increase the value of force. If vertical dynamic loads are to be considered, a dead load factor can be applied.

One can justify many other methods of analyses that will satisfy the current code. The approach presented in this chapter can be used directly with the computer programs ETABS and SAP2000 with their steel and concrete post-processors. Since these programs have very large capacities and operate on personal computers, it is possible for a structural engineer to investigate a large number of different designs very rapidly with a minimum expenditure of manpower and computer time.

17.7. SUMMARY

After being associated with the three dimensional dynamic analysis and design of a large number of structures during the past 40 years, the author would like to take this opportunity to offer some constructive comments on the lateral load requirements of the current code.

First: *the use of the "dynamic base shear" as a significant indication of the response of a structure may not be conservative.* An examination of the modal base shears and overturning moments in Tables 17.1 and 17.2 clearly indicates that base shears associated with the shorter periods produce relatively small overturning moments. Therefore, a dynamic analysis, which will contain higher mode response, will always produce a larger dynamic base shear relative to the dynamic overturning moment. Since the code allows all results to be scaled by the ratio of dynamic base shear to the static design base shear, the dynamic overturning moments can be significantly less than the results of a simple static code analysis. A scale factor based on the ratio of the "static design overturning moment" to the "dynamic overturning moment" would be far more logical. The static overturning moment can be calculated by using the static vertical distribution of the design base shear which is currently suggested in the code.

Second: *for irregular structures, the use of the terminology "period (or mode shape) in the direction under consideration" must be discontinued.* The stiffness and mass properties of the structure define the directions of all three dimensional mode shapes. The term "principal direction" should not be used unless it is clearly and uniquely defined.

Third: *the scaling of the results of a dynamic analysis should be re-examined.* The use of site-dependent spectra is encouraged.

Finally: *it is not necessary to distinguish between regular and irregular structures when a three dimensional dynamic analysis is conducted.* If an accurate three dimensional computer model is created, the vertical and horizontal irregularities and known eccentricities of stiffness and mass will cause the displacement and rotational components of the mode shapes to be coupled. A three dimensional dynamic analysis, based on these coupled mode shapes, will produce a far more complex response with larger forces than the response of a regular structure. It is possible to predict the dynamic force distribution in a very irregular structure with the same degree of accuracy and reliability as the evaluation of the force distribution in a very regular structure. Consequently, if the design of an irregular structure is based on a realistic dynamic force distribution, there is no logical reason to expect that it will be any less earthquake resistant than a regular structure which was designed using the same dynamic loading. A reason why many irregular structures have a documented record of poor performance during earthquakes is that their designs were often based on approximate two dimensional static analyses.

One major advantage of the modeling method presented in this chapter is that one set of dynamic design forces, including the effects of accidental torsion, is produced with one computer run. Of greater significance, however, is the resulting structural design has equal resistance to seismic motions from all possible directions.

17.8. REFERENCES

1. *Recommended Lateral Force Requirements and Commentary, 1996 Sixth Edition*, Seismology Committee, Structural Engineers Association of California, Tel. 916-427-3647.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

DAMPING AND ENERGY DISSIPATION

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax. 5510.0573

DAMPING AND ENERGY DISSIPATION

Linear Viscous Damping
Is A Property Of The Computer Model
And Is Not A Property Of A Real Structure

19.1. INTRODUCTION

In structural engineering, viscous, velocity-dependent damping is very difficult to visualize for most real structural systems. Only a small number of structures have a finite number of damping elements where real viscous dynamic properties can be measured. In most cases modal damping ratios are used in the computer model to approximate unknown nonlinear energy dissipation within the structure.

Another form of damping, that is often used in the mathematical model for the simulation of the dynamic response of a structure, is proportional to the stiffness and mass of the structure. This is referred to as Rayleigh damping. Both modal and Rayleigh damping are used in order to avoid the need to form a damping matrix based on the physical properties of the real structure.

In recent years, the addition of energy dissipation devices to the structure has forced the structural engineer to treat the energy dissipation in a more exact manner. In this chapter, the limitations of modal and Rayleigh damping will be discussed. In addition, detailed algorithms will be presented for numerical solution, by iteration, for several different types of nonlinear energy dissipation devices.

19.2. DAMPING IN REAL STRUCTURES

It is possible to estimate an "effective" viscous damping ratio directly from laboratory or field tests of structures. One method is to apply a static displacement by attaching a cable to the structure and then suddenly removing the load by cutting the cable. If the structure can be approximated by a single degree of freedom, the displacement response will be of the form shown in Figure 19.1. For multi-degree-of-freedom structural systems, the response will involve the response of more modes and the test and the analysis method required to predict the damping ratios will be more complex.

It should be pointed out that the decay of the typical displacement response only indicates that energy dissipation is taking place. The cause of the energy dissipation may be due to many different effects such as material damping, joint friction and radiation damping at the supports. However, if it is assumed that all energy dissipation is due to linear viscous damping, the free vibration response is given by the following equation:

$$u(t) = u(0)\ e^{-\xi\omega t}\ \cos(\omega_D t) \tag{19.1}$$

where $\omega_D = \omega\sqrt{1-\xi^2}$

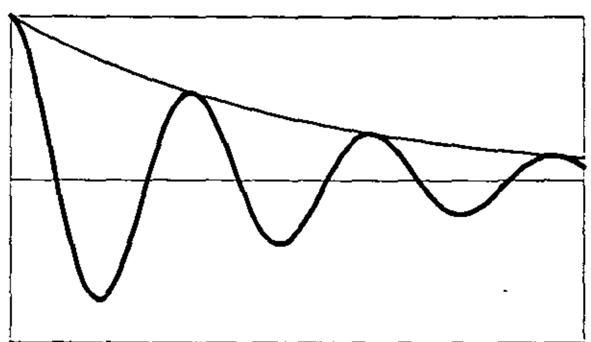


Figure 19.1. Free Vibration Test of Real Structures, Response vs. Time

Equation (19.1) can be evaluated at any two maximum points m cycles apart and the following two equations are produced:

$$u(2\pi n) = u_n = u(0)\, e^{-\xi\omega 2\pi n/\omega_D} \tag{19.2}$$

$$u(2\pi(n+m)) = u_{n+1} = u(0)\, e^{-\xi\omega 2\pi(n+m)/\omega_D} \tag{19.3}$$

The ratio of these two equations is

$$\frac{u_n}{u_{n+1}} = e^{\frac{2\pi m\xi}{\sqrt{1-\xi^2}}} \tag{19.4}$$

Taking the natural logarithm of this ratio and rewriting produces the following equation:

$$\xi = \frac{\ln(u_n / u_{n+m})}{2\pi m}\sqrt{1-\xi^2} \quad \text{or,} \quad \xi_{(i)} = \xi_0\sqrt{1-\xi_{(i-1)}^2} \tag{19.5}$$

This equation can easily be solved for the damping ratio ξ by iteration. For example, if the decay ratio u_{n+m} is equal to 1.25 between two adjacent maximums, three iterations yield

$$\xi \approx 0.07023 \approx 0.07005 = 0.07005$$

Field testing of real structures, subjected to small displacements, indicates typical damping ratios are less than two percent. Also, for most structures, the damping is not linear and is not proportional to the velocity.

19.3. USE OF VISCOUS DAMPING IN ANALYSIS

In the elastic dynamic analysis of most structures subjected to earthquake motions it is very common to use five percent damping for all modes. However, this value, in most cases, has very little experimental or theoretical justification. Also, for multi degree-of-freedom systems, the use of modal damping violates dynamic equilibrium and the fundamental laws of physics. For example, it is possible to calculate the reactions, as a function of time, at the base of a structure by the following two methods:

First, the inertia forces, at each mass point, can be calculated in a specific direction by the multiplication of the absolute acceleration in that direction times the mass at the point. In the case of earthquake loading, the sum of all these forces must be equal to the sum of the base reaction forces in that direction since no other forces act on the structure.

Second, the member forces, at the ends of all members attached to reaction points, can be calculated as a function of time. The sum of the components of the member forces in the direction of the load is the base reaction force experienced by the structure.

In the case of zero modal damping these reaction forces, as a function of time, are identical. However, for nonzero modal damping, these reaction forces are significantly different. These differences indicate that linear modal damping introduces external loads, acting on the structure above the base, which are physically impossible. This is clearly an area where the standard "state of the art" assumption of modal damping needs to be re-examined and an alternative approach must be developed.

Energy dissipation exists in real structures. However, it must be in the form of equal and opposite forces between points within the structure. Therefore, a viscous damper, or any other type of energy dissipating device, connected between two points within the structure is physically possible and will not cause an error in the reaction forces. There must be zero base shear for all internal energy dissipation forces.

Another type of energy dissipation that exists in real structures is radiation damping at the supports of the structure. The vibration of the structure strains the foundation material near the supports and causes stress waves to radiate into the infinite foundation. This can be significant if the foundation material is soft relative to the stiffness of the structure. A spring, damper and mass at each support often approximate this type of damping.

19.4. NUMERICAL EXAMPLE

In order to illustrate the errors involved in the use of modal damping a simple seven-story building was subjected to a typical earthquake motion. Table 19.1 indicates the values of base shear calculated from the external inertia forces, which satisfy dynamic equilibrium, and the base shear calculated from the exact summation of the shears at the base of the three columns at each time increment.

It is of interest to note that the maximum values of base shear, calculated from two different methods, are significantly different for the same computer run. The only logical explanation is the existence of external damping forces that exist only in the mathematical model of the structure. Since this is physically impossible, the use of standard modal damping can produce a small error in the analysis.

TABLE 19.1. Comparison of Base Shear for Seven Story Building

Damping Percent	Dynamic Equilibrium BASE SHEAR (kips)	Sum of Column SHEARS (kips)	ERROR Percent
0	370.7 @ 5.355 Sec.	370.7 @ 5.355 Sec.	0.0
2	314.7 @ 4.690 Sec	318.6 @ 4.695 Sec	+1.2
5	253.7 @ 4.675 Sec	259.6 @ 4.690 Sec	+2.3
10	214.9 @ 3.745 Sec	195.4 @ 4.035 Sec	-9.1
20	182.3 @ 3.055 Sec	148.7 @ 3.365 Sec	-18.4

It is of interest to note that the use of only five percent damping reduces the base shear from 371 kips to 254 kips for this example. Since the measurement of damping in most real structures has been found to be less than two percent the selection of five percent reduces the results significantly.

19.5. STRUCTURES WITH LINEAR VISCOUS DAMPERS

It is possible to model structural systems with linear viscous dampers at arbitrary locations within a structural system. The exact solution involves the calculation of complex eigenvalues and eigenvectors and a large amount of computational effort. Since the basic nature of energy dissipation is not clearly defined in real structures and viscous damping is often used to approximate nonlinear behavior, this increase in computational effort is not justified since we are not solving the real problem. A more efficient method to solve this problem is to move the damping force to the right hand side of the dynamic equilibrium equation and solve the problem as a nonlinear problem using the FNA method.

19.6. STIFFNESS AND MASS PROPORTIONAL DAMPING

A very common type of damping used in the nonlinear incremental analysis of structures is to assume that the damping matrix is proportional to the mass and stiffness matrices. Or,

$$\mathbf{C} = \eta\,\mathbf{M} + \delta\,\mathbf{K} \tag{19.6}$$

This type of damping is normally referred to as Rayleigh damping. In mode superposition analysis the damping matrix must have the following properties in order for the modal equations to be uncoupled:

$$2\omega_n\zeta_n = \phi_n^T\mathbf{C}\phi_n \tag{19.7}$$

Due to the orthogonality properties of the mass and stiffness matrices, this equation can be rewritten as

$$2\omega_n\zeta_n = \eta + \delta\,\omega_n^2 \qquad \text{or,}\ \zeta_n = \frac{1}{2\omega_n}\eta + \frac{\omega_n}{2}\delta \tag{19.8}$$

It is apparent that modal damping can be specified exactly at only two frequencies in order to solve for η and δ in the above equation. In addition, the assumption of mass proportional damping implies the existence of external supported dampers that are physically impossible for a base supported structure. The use of stiffness proportional damping has the effect of increasing the damping in the higher modes of the structure for which there is no physical justification and can result in significant errors for impact type of problems. Therefore, the use of Rayleigh type of damping is difficult to justify for most structures. However, it continues to be used by many computer programs in order to obtain results for numerically sensitive structural systems.

19.7. NONLINEAR ENERGY DISSIPATION

Most physical energy dissipation in real structures is in phase with the displacements and is a nonlinear function of the magnitude of the displacements. Nevertheless, it is common practice to approximate the nonlinear behavior with an "equivalent linear damping" and not conduct a nonlinear analysis. The major reason for this approximation is that all linear programs for mode superposition or response spectrum analysis can consider linear viscous damping in an exact manner. This approximation is no longer necessary if the structural engineer can identify where and how the energy is dissipated within the structural system. The FNA method provides an alternative to the use of equivalent linear viscous damping. In this section various nonlinear devices will be discussed and their iterative solution algorithm will be summarized.

Base isolators are one of the most common types of predefined nonlinear elements used in earthquake resistant designs. Mechanical dampers, friction devices and plastic hinges are other type of common nonlinear elements. In addition, gap elements are required to model contact between structural components and uplifting of structures. A special type of gap element, with the ability to crush and dissipate energy, is useful to model concrete and soil types of materials. Cables, that can take tension only and dissipate energy in yielding, are necessary to capture the behavior of many bridge type structures.

19.8. BILINEAR PLASTICITY ELEMENT

The general plasticity element can be used to model many different types of nonlinear material properties. The fundamental properties and behavior of the element are illustrated in the figure shown below:

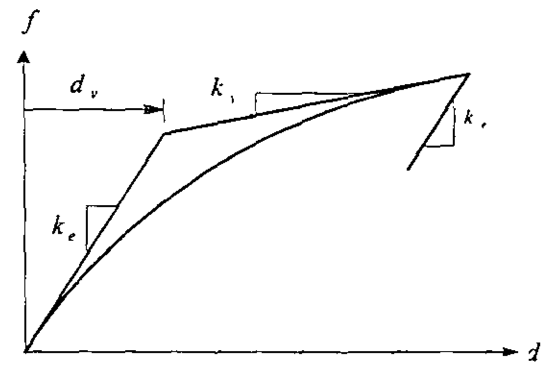


Figure 19.2. Fundamental Behavior of Bilinear Plasticity Element

Where k_e = initial linear stiffness

k_y = Yield stiffness

d_y = Yield deformation

The force-deformation relationship is calculated from

$$f = k_y\, d + (k_e - k_y)\, e \tag{19.9}$$

Where d is the total deformation and e is an elastic deformation term and has a range $\pm d_y$. It is calculated at each time step by the numerical integration of one of the following differential equations:

$$\text{If } \dot{d}e \geq 0 \quad \dot{e} = (1 - |\frac{e}{d_y}|^n)\dot{d} \tag{19.10}$$

$$\text{If } \dot{d}e < 0 \quad \dot{e} = \dot{d} \tag{19.11}$$

The following finite difference approximations, for each time step, can be made:

$$\dot{d} = \frac{d_t - d_{t-\Delta t}}{\Delta t} \qquad \text{And} \qquad \dot{e} = \frac{e_t - e_{t-\Delta t}}{\Delta t}$$

The numerical solution algorithm (six program statements) can be summarized at the end of each time increment Δt, at time "t" for iteration "i", in Table 19.2.

Table 19.2. Iterative Algorithm For Bilinear Plasticity Element

1. Change in deformation for time step Δt at time t for iteration i

$$v = d_t^{(i)} - d_{t-\Delta t}$$

2. Calculate elastic deformation for iteration i

$$\text{if } v e_t^{(i-1)} \leq 0 \qquad e_t^{(i)} = e_{t-\Delta t} + v$$

$$\text{if } v e_t^{(i-1)} > 0 \qquad e_t^{(i)} = e_{t-\Delta t} + \left(1 - \left|\frac{e_{t-\Delta t}}{d_y}\right|^n\right) v$$

$$\text{if } e_t^{(i)} > d_y \qquad e_t^{(i)} = d_y$$

$$\text{if } e_t^{(i)} < -d_y \qquad e_t^{(i)} = -d_y$$

3. Calculate iterative force:

$$f_t^{(i)} = k_y d_t^{(i)} + (k_e - k_y) e_t^{(i)}$$

Note that the approximate term $\frac{e_{t-\Delta t}}{d_y}$ is used from the end of the last time increment rather than the iterative term $\frac{e_t^{(i)}}{d_y}$. This approximation eliminates all problems associated with convergence for large values of n. However, the approximation has insignificant effects on the numerical results for all values of n. For all practical purposes, a value of n equal to 20 produces true bilinear behavior.

19.9. DIFFERENT POSITIVE AND NEGATIVE PROPERTIES

The previously presented plasticity element can be generalized to have different positive, d_p, and negative, d_n, yield properties. This will allow the same element to model many different types of energy dissipation devices such as the double diagonal Pall friction element.

For constant friction the double diagonal Pall element has $k_e = 0$ and $n \approx 20$. For small forces both diagonals remain elastic, one in tension and one in compression. At some deformation, d_x, the compressive element may reach a maximum possible value. Friction slipping will start at the deformation u_p after which both the tension and compression forces will remain constant until the maximum displacement for the load cycle is obtained.

This element can be used to model bending hinges in beams or columns with non-symmetric sections. The numerical solution algorithm for the general bilinear plasticity element is given in Table 19.3.

Table 19.3. Iterative Algorithm For Non-Symmetric Bilinear Element

1. Change in deformation for time step Δt at time t for iteration i

$$v = d_t^{(i)} - d_{t-\Delta t}$$

2. Calculate elastic deformation for iteration i

if $v e_t^{(i-1)} \leq 0$ $\quad e_t^{(i)} = e_{t-\Delta t} + v$

if $v e_t^{(i-1)} > 0$ and $e_{t-\Delta t} > 0$

$$e_t^{(i)} = e_{t-\Delta t} + \left(1 - \left|\frac{e_{t-\Delta t}}{d_y}\right|^n\right) v$$

if $v e_t^{(i-1)} > 0$ and $e_{t-\Delta t} < 0$

$$e_t^{(i)} = e_{t-\Delta t} + \left(1 - \left|\frac{e_{t-\Delta t}}{d_n}\right|^n\right) v$$

if $e_t^{(i)} > d_y$ $\quad e_t^{(i)} = d_y$

if $e_t^{(i)} < -d_y$ $\quad e_t^{(i)} = -d_y$

3. Calculate iterative force at time t :

$$f_t^{(i)} = k_y d_t^{(i)} + (k_e - k_y) e_t^{(i)}$$

19.10. THE BILINEAR TENSION-GAP-YIELD ELEMENT

The bilinear tension only element can be used to model cables connected to different parts of the structure. In the retrofit of bridges this type of element is often used at expansion joints to limit the relative movement during earthquake motions. The fundamental behavior of the element is summarized in Figure 19.3. The positive number d_0 is the initial pre-stress deformation. A negative number specifies the axial deformation associated with initial cable sag. The permanent element yield deformation is u_p.

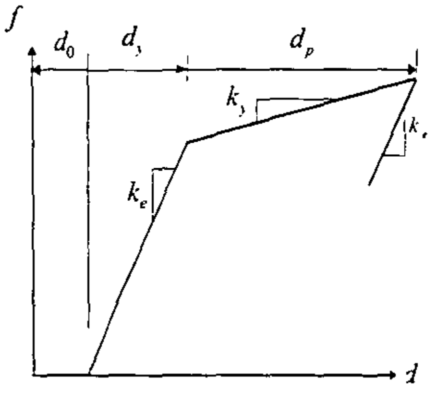


Figure 19.3. Tension-Gap-Yield Element

The numerical solution algorithm for this element is summarized in Table 19.4. Note that the permanent deformation calculation is based on the converged deformation at the end of the last time step. This avoids numerical solution problems.

Table 19.4. Iterative Algorithm -Tension-Gap-Yield Element

1. Update Tension Yield Deformation From Previous Converged Time Step

   $y = d_{t-\Delta t} + d_0 - d_y$

   if $y < d_p$ then $d_p = y$

2. Calculate Elastic Deformation for Iteration (i)

   $d = d_t^{(i)} + d_0$

   $e_t^{(i)} = d - d_p$

   if $e_t^{(i)} > d_y$ then $e_t^{(i)} = d_y$

3. Calculate Iterative Force:

   $f_t^{(i)} = k_y(d_t^{(i)} - d_0) + (k_e - k_y)e_t^{(i)}$

   if $f_t^{(i)} > 0$ then $f_t^{(i)} = 0$

19.11. NONLINEAR GAP-CRUSH ELEMENT

Perhaps the most common type of nonlinear behavior that occurs in real structural systems is the closing of a gap between different parts of the structure; or, the uplifting of the structure at its foundation. The element can be used at abutment-soil interfaces and for modeling soil-pile contact. The gap/crush element has the following physical properties:

1. The element cannot develop a force until the opening d_0 gap is closed
2. The element can only develop a compression force
3. The crush deformation d_p is always a monatomic decreasing negative number

The numerical algorithm for the gap-crush element is summarized in Table 19.5.

Table 19.5. Iterative Algorithm To Model Gap-Crush Element

1. Update Crush Deformation From Previous Converged Time Step

$$y = d_{t-\Delta t} + d_0 + d_y$$

$$\text{if } y > d_n \text{ then } d_n = y$$

2. Calculate Elastic Deformation:

$$e_t^{(i)} = d_t^{(i)} + d_n - d_y$$

$$\text{if } e_t^{(i)} < -d_y \text{ then } e_t^{(i)} = -d_y$$

3. Calculate Iterative Force:

$$f_t^{(i)} = k_y(d_t^{(i)} - d_0) + (k_e - k_y)e_t^{(i)}$$

$$\text{if } f_t^{(i)} > 0 \text{ then } f_t^{(i)} = 0$$

The numerical convergence of the gap element can be very slow if a large elastic stiffness term k_e is used. The user must take great care in selecting a physically realistic number. In order to minimize numerical problems, the stiffness k_e should not be over 100 times the stiffness of the elements adjacent to the gap. The dynamic contact problem between real structural components often does not have a unique solution. Therefore, it is the responsibility of the design engineer to select materials at contact points and surfaces to have realistic material properties that can be predicted accurately.

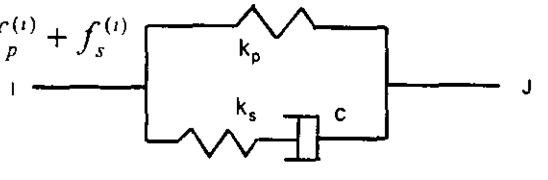
19.12. VISCOUS DAMPING ELEMENTS

Linear velocity-dependent energy-dissipation forces exist in only a few special materials subjected to small displacements. In terms of equivalent model damping, experiments indicate that they are a small fraction of one percent. Manufactured mechanical dampers cannot be made with linear viscous properties since all fluids have finite compressibility and nonlinear behavior is present in all manmade devices. In the past it has been common practice to approximate the behavior of these viscous nonlinear elements by a simple linear viscous force. More recently, vendors of these devices have claimed that the damping forces are proportional to a power of the velocity. Experimental examination of a mechanical device indicates a far more complex behavior that cannot be represented by a simple one-element model.

The FNA method does not require that these damping devices be linearized or simplified in order to obtain a numerical solution. If the physical behavior is understood it is possible for an iterative solution algorithm to be developed which will accurately simulate the behavior of almost any type of damping device. In order to illustrate the procedure let us consider the device shown in Figure 19.4.

$$f_p^{(i)} = k_p d_t^{(i)}$$

$$f^{(i)} = f_p^{(i)} + f_s^{(i)}$$



$$f_s^{(i)} = k_s(d_t^{(i)} - e^{(i)}) = sign(\dot{e}^{(i)})\dot{e}^{(i)^N} c$$

Figure 19.4. General Damping Element Connected Between Points I and J

It is apparent that the total deformation, $e_t^{(i)}$, across the damper must be accurately calculated in order to evaluate the equilibrium within the element at each time step. The finite difference equation used to estimate the damper deformation at time t is

$$e_t^{(i)} = e_{t-\Delta t} + \int_{t-\Delta t}^{t} \dot{e}_\tau^{(i)} d\tau = e_{t-\Delta t} + \frac{\Delta t}{2}(\dot{e}_{t-\Delta t} + \dot{e}_t^{(i)}) \tag{19.12}$$

A summary of the numerical algorithm is summarized in Table 19.6.

Table 19.6. Iterative Algorithm For Nonlinear Viscous Element

1. Estimate damper force from last iteration:
$$f_c^{(i)} = k(d_t^{(i)} - e_t^{(i-1)})$$
2. Estimate damper velocity:
$$\dot{e}_t^{(i)} = \left(\frac{f_c^{(i)}}{c}\right)^{\frac{1}{N}} sign(f_c^{(i)})$$
3. Estimate damper deformation:
$$e_t^{(i)} = e_{t-\Delta t} + \frac{1}{2\Delta t}(\dot{e}_{t-\Delta t} + \dot{e}_t^{(i)})$$
4. Calculate total iterative force:
$$f_t^{(i)} = k_p d_t^{(i)} + k_s(d_t^{(i)} - e_t^{(i)})$$

19.13. SUMMARY

The use of linear modal damping, as a percentage of critical damping, has been used to approximate the nonlinear behavior of structures. The energy dissipation in real structures is far more complicated and tends to be proportional to displacements rather than proportional to the velocity. The use of approximate "equivalent viscous damping" has little theoretical or experimental justification.

It is now possible to accurately simulate the behavior of structures with a finite number of discrete energy dissipation devices installed. The experimental determined properties of the devices can be directly incorporated into the computer model.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

DYNAMIC ANALYSIS BY NUMERICAL INTEGRATION

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

DYNAMIC ANALYSIS BY NUMERICAL INTEGRATION

Normally, For Earthquake Loading Direct.Numerical Integration Is Very Slow

20.1 INTRODUCTION

The most general approach for the solution of the dynamic response of structural systems is the direct numerical integration of the dynamic equilibrium equations. This involves, after the solution is defined at time zero, the attempt to satisfy dynamic equilibrium at discrete points in time. Most methods use equal time intervals at Δt, $2\Delta t$, $3\Delta t$........$N\Delta t$. Many different numerical techniques have previously been presented; however, all approaches can fundamentally be classified as either *explicit* or *implicit* integration methods.

Explicit methods do not involve the solution of a set of linear equations at each step. Basically, these methods use the differential equation at time "t" to predict a solution at time "$t + \Delta t$". For most real structures, which contain stiff elements, a very small time step is required in order to obtain a stable solution. Therefore, all explicit methods are *conditionally stable* with respect to the size of the time step.

Implicit methods attempt to satisfy the differential equation at time "t" after the solution at time "$t - \Delta t$" is found. These methods require the solution of a set of linear equations at each time step; however, larger time steps may be used. Implicit methods can be *conditionally or unconditionally* stable.

There exist a large number of accurate, higher-order, multi-step methods that have been developed for the numerical solution of differential equations. These multi-step methods assume that the solution is a smooth function in which the higher derivatives are continuous. The exact solution of many nonlinear structures requires that the accelerations, the second derivative of the displacements, are not smooth functions. This discontinuity of the acceleration is caused by the nonlinear hysteresis of most structural materials, contact between parts of the structure, and buckling of elements. Therefore, only single-step methods will be presented in this chapter. Based on a significant amount of experience, it is the conclusion of the author that only single-step, implicit, unconditional stable methods be used for the step-by-step seismic analysis of practical structures.

20.2 NEWMARK FAMILY OF METHODS

In 1959 Newmark [1] presented a family of single-step integration methods for the solution of structural dynamic problems for both blast and seismic loading. During the past 40 years Newmark's method has been applied to the dynamic analysis of many practical engineering structures. In addition, it has been modified and improved by many other researchers. In order to illustrate the use of this family of numerical integration methods consider the solution of the linear dynamic equilibrium equations written in the following form:

$$\mathbf{M}\ddot{\mathbf{u}}_t + \mathbf{C}\dot{\mathbf{u}}_t + \mathbf{K}\mathbf{u}_t = \mathbf{F}_t \tag{20.1}$$

The direct use of Taylor's series provides a rigorous approach to obtain the following two additional equations:

$$\mathbf{u}_t = \mathbf{u}_{t-\Delta t} + \Delta t\,\dot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^2}{2}\ddot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^3}{6}\dddot{\mathbf{u}}_{t-\Delta t} + \ldots\ldots \tag{20.2a}$$

$$\dot{\mathbf{u}}_t = \dot{\mathbf{u}}_{t-\Delta t} + \Delta t\,\ddot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^2}{2}\dddot{\mathbf{u}}_{t-\Delta t} + \ldots\ldots \tag{20.2b}$$

Newmark truncated these equations and expressed them in the following form:

$$\mathbf{u}_t = \mathbf{u}_{t-\Delta t} + \Delta t\,\dot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^2}{2}\ddot{\mathbf{u}}_{t-\Delta t} + \beta\,\Delta t^3\dddot{\mathbf{u}} \tag{20.2a}$$

$$\dot{\mathbf{u}}_t = \dot{\mathbf{u}}_{t-\Delta t} + \Delta t\,\ddot{\mathbf{u}}_{t-\Delta t} + \gamma\,\Delta t^2\dddot{\mathbf{u}} \tag{20.2b}$$

If the acceleration is assumed to be linear within the time step, the following equation can be written:

$$\dddot{\mathbf{u}} = \frac{(\ddot{\mathbf{u}}_t - \ddot{\mathbf{u}}_{t-\Delta t})}{\Delta t} \tag{20.3}$$

The substitution of Equation (20.3) into Equations (20.2a and b) produces Newmark's equations in standard form

$$\mathbf{u}_t = \mathbf{u}_{t-\Delta t} + \Delta t\,\dot{\mathbf{u}}_{t-\Delta t} + (\frac{1}{2} - \beta)\Delta t^2\ddot{\mathbf{u}}_{t-\Delta t} + \beta\,\Delta t^2\ddot{\mathbf{u}}_t \tag{20.4a}$$

$$\dot{\mathbf{u}}_t = \dot{\mathbf{u}}_{t-\Delta t} + (1-\gamma)\,\Delta t\,\ddot{\mathbf{u}}_{t-\Delta t} + \gamma\,\Delta t\,\ddot{\mathbf{u}}_t \tag{20.4b}$$

Newmark used Equations (20.4a, 20.4b and 20.1) iteratively, for each time step, for each displacement DOF of the structural system. The term $\ddot{u}_t$ was obtained from Equation (20.1) by dividing the equation by the mass associated with the DOF.

In 1962 Wilson [2] formulated Newmark's method in matrix notation, added stiffness and mass proportional damping, and eliminated the need for iteration by introducing the direct solution of equations at each time step. This requires that Equations (20.4a and 20.4b) be rewritten in the following form:

$$\ddot{\mathbf{u}}_t = b_1(\mathbf{u}_t - \mathbf{u}_{t-\Delta t}) + b_2\dot{\mathbf{u}}_{t-\Delta t} + b_3\ddot{\mathbf{u}}_{t-\Delta t} \tag{20.5a}$$

$$\dot{\mathbf{u}}_t = b_4(\mathbf{u}_t - \mathbf{u}_{t-\Delta t}) + b_5\dot{\mathbf{u}}_{t-\Delta t} + b_6\ddot{\mathbf{u}}_{t-\Delta t} \tag{20.5b}$$

where the constants b_1 to b_6 are defined in Table 20.1. The substitution of Equations (20.5a and20.5b) into Equation (20.1) allows the dynamic equilibrium of the system at time " t "to be written in terms of the unknown node displacements $\mathbf{u}_t$. Or,

$$\begin{aligned}(b_1\mathbf{M} + b_4\mathbf{C} + \mathbf{K})\mathbf{u}_t &= \mathbf{F}_t + \mathbf{M}(b_1\mathbf{u}_{t-\Delta t} - b_2\dot{\mathbf{u}}_{t-\Delta t} - b_3\ddot{\mathbf{u}}_{t-\Delta t}) \\ &\quad + \mathbf{C}(b_4\mathbf{u}_{t-\Delta t} - b_5\dot{\mathbf{u}}_{t-\Delta t} - b_6\ddot{\mathbf{u}}_{t-\Delta t})\end{aligned} \tag{20.6}$$

The Newmark direct integration algorithm is summarized in Table 20.1. Note that the constants b_i need be calculated only once. Also, for linear systems, the effective dynamic stiffness matrix $\overline{\mathbf{K}}$ is formed and triangularized only once.

20.3 STABILITY OF NEWMARK'S METHOD

For zero damping Newmark's method is conditionally stable if

$$\gamma \geq \frac{1}{2}, \quad \beta \leq \frac{1}{2} \quad \text{and} \quad \Delta t \leq \frac{1}{\omega_{MAX}\sqrt{\gamma/2 - \beta}} \tag{20.7}$$

where ω_{MAX} is the maximum frequency in the structural system [1]. Newmark's method is unconditionally stable if

$$2\beta \geq \gamma \geq \frac{1}{2} \tag{20.8}$$

However, if γ is greater than ½, errors are introduced. These errors are associated with "numerical damping" and "period elongation".

For large multi degree-of-freedom structural systems the time step limit, given by Equation (20.7), can be written in a more useable form as

$$\frac{\Delta t}{T_{MIN}} \leq \frac{1}{2\pi\sqrt{\gamma/2 - \beta}} \tag{20.9}$$

Computer models of large real structures normally contain a large number of periods which are smaller than the integration time step; therefore, it is essential that one select a numerical integration method that is unconditional for all time steps.

Table 20.1. Summary of the Newmark Method for Direct Integration

I. INITIAL CALCULATION

A. Form static stiffness matrix $\mathbf{K}$, mass matrix $\mathbf{M}$ and damping matrix $\mathbf{C}$

B. Specify integration parameters β and γ

C. Calculate integration constants

$$b_1 = \frac{1}{\beta \Delta t^2} \qquad b_2 = \frac{1}{\beta \Delta t} \qquad b_3 = \beta - \frac{1}{2} \qquad b_4 = \gamma \, \Delta t \, b_1$$

$$b_5 = 1 + \gamma \, \Delta t \, b_2 \quad b_6 = \Delta t \, (1 + \gamma \, b_3 - \gamma)$$

D. Form effective stiffness matrix $\overline{\mathbf{K}} = \mathbf{K} + b_1 \mathbf{M} + b_4 \mathbf{C}$

E. Triangularize effective stiffness matrix $\overline{\mathbf{K}} = \mathbf{LDL}^T$

F. Specify initial conditions $\mathbf{u}_0$, $\dot{\mathbf{u}}_0$, $\ddot{\mathbf{u}}_0$

II. FOR EACH TIME STEP $t = \Delta t,\ 2\Delta t,\ 3\Delta t$ ------

A. Calculate effective load vector

$$\overline{\mathbf{F}}_t = \mathbf{F}_t + \mathbf{M}(b_1 \mathbf{u}_{t-\Delta t} - b_2 \dot{\mathbf{u}}_{t-\Delta t} - b_3 \ddot{\mathbf{u}}_{t-\Delta t}) + \mathbf{C}(b_4 \mathbf{u}_{t-\Delta t} - b_5 \dot{\mathbf{u}}_{t-\Delta t} - b_6 \ddot{\mathbf{u}}_{t-\Delta t})$$

B. Solve for node displacement vector at time **t**

$$\mathbf{LDL}^T \mathbf{u}_t = \overline{\mathbf{F}}_t \qquad \text{forward and back-substitution only}$$

C. Calculate node velocities and accelerations at time **t**

$$\dot{\mathbf{u}}_t = b_4 (\mathbf{u}_t - \mathbf{u}_{t-\Delta t}) + b_5 \dot{\mathbf{u}}_{t-\Delta t} + b_6 \ddot{\mathbf{u}}_{t-\Delta t}$$

$$\ddot{\mathbf{u}}_t = b_1 (\mathbf{u}_t - \mathbf{u}_{t-\Delta t}) + b_2 \dot{\mathbf{u}}_{t-\Delta t} + b_3 \ddot{\mathbf{u}}_{t-\Delta t}$$

D. Go to Step II.A with $t = t + \Delta t$

20.4 THE AVERAGE ACCELERATION METHOD

The average acceleration method is identical to the trapezoidal rule that has been used to numerically evaluate second order differential equations for approximately 100 years. It can easily be derived from the following truncated Taylor's series expansion:

$$\begin{aligned}\mathbf{u}_\tau &= \mathbf{u}_{t-\Delta t} + \tau\,\dot{\mathbf{u}}_{t-\Delta t} + \frac{\tau^2}{2}\ddot{\mathbf{u}}_{t-\Delta t} + \frac{\tau^3}{6}\dddot{\mathbf{u}}_{t-\Delta t} + \ldots\ldots \\ &\approx \mathbf{u}_{t-\Delta t} + \tau\,\dot{\mathbf{u}}_{t-\Delta t} + \frac{\tau^2}{2}\left(\frac{\ddot{\mathbf{u}}_{t-\Delta t} + \ddot{\mathbf{u}}_t}{2}\right)\end{aligned} \tag{20.10}$$

where τ is a variable point within the time step. The consistent velocity can be obtained by differentiation of Equation (20.10). Or,

$$\dot{\mathbf{u}}_\tau = \dot{\mathbf{u}}_{t-\Delta t} + \tau\left(\frac{\ddot{\mathbf{u}}_{t-\Delta t} + \ddot{\mathbf{u}}_t}{2}\right) \tag{20.11}$$

If $\tau = \Delta t$

$$\mathbf{u}_t = \mathbf{u}_{t-\Delta t} + \Delta t\,\dot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^2}{4}\ddot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^2}{4}\ddot{\mathbf{u}}_t \tag{20.12a}$$

$$\dot{\mathbf{u}}_t = \dot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t}{2}\ddot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t}{2}\ddot{\mathbf{u}}_t \tag{20.12b}$$

These equations are identical to Newmark's Equations (20.4a and b) with $\gamma = 1/2$ and $\beta = 1/4$.

It can easily be shown that the average acceleration method conserves energy for the free vibration problem, $\mathbf{M}\ddot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{0}$, for all possible time steps [4]. . Therefore, the sum of the kinetic and strain energy is constant. Or,

$$2E = \dot{\mathbf{u}}_t^T\mathbf{M}\dot{\mathbf{u}}_t + \mathbf{u}_t^T\mathbf{K}\mathbf{u}_t = \dot{\mathbf{u}}_{t-\Delta t}^T\mathbf{M}\dot{\mathbf{u}}_{t-\Delta t} + \mathbf{u}_{t-\Delta t}^T\mathbf{K}\mathbf{u}_{t-\Delta t} \tag{20.13}$$

20.5 WILSON'S θ FACTOR

In 1973, the general Newmark method was made unconditionally stable by the introduction of a θ factor [3]. The introduction of the θ factor is motivated by the observation that an unstable solution tends to oscillate about the true solution. Therefore, if the numerical solution is evaluated within the time increment the spurious oscillations are minimized. This can be accomplished by a simple modification to the Newmark method by using a time step defined by

$$\Delta t' = \theta\ \Delta t \tag{20.14a}$$

and a load defined by

$$\mathbf{R}_{t'} = \mathbf{R}_{t-\Delta t} + \theta\ (\mathbf{R}_t - \mathbf{R}_{t-\Delta t}) \tag{20.14b}$$

where $\theta \geq 1.0$. After the acceleration $\ddot{\mathbf{u}}_{t'}$ vector is evaluated by Newmark's method at the integration time step $\theta\ \Delta t$, values of node accelerations, velocities and displacements are calculated from the following fundamental equations:

$$\ddot{\mathbf{u}}_t = \ddot{\mathbf{u}}_{t-\Delta t} + \frac{1}{\theta}(\ddot{\mathbf{u}}_{t'} - \ddot{\mathbf{u}}_{t-\Delta t}) \tag{20.15a}$$

$$\dot{\mathbf{u}}_t = \dot{\mathbf{u}}_{t-\Delta t} + (1-\gamma)\Delta t\,\ddot{\mathbf{u}}_{t-\Delta t} + \gamma\ \Delta t\,\ddot{\mathbf{u}}_t \tag{20.15b}$$

$$\mathbf{u}_t = \mathbf{u}_{t-\Delta t} + \Delta t\,\dot{\mathbf{u}}_{t-\Delta t} + \frac{\Delta t^2(1-2\beta)}{2}\ddot{\mathbf{u}}_{t-\Delta t} + \beta\,\Delta t^2\ddot{\mathbf{u}}_t \tag{20.15c}$$

The use of the θ factor tends to numerically damp out the high modes of the system. If θ equals 1.0 Newmark's method is not modified. However, for problems where the higher mode response is important, the errors that are introduced can be large. In addition, the dynamic equilibrium equations are not exactly satisfied at time t. Therefore, the author no longer recommends the use of the θ factor. At the time of the introduction of the method, it solved all problems associated with stability of the Newmark family of methods. However, during the past twenty years new and more accurate numerical methods have been developed.

20.6 THE USE OF STIFFNESS PROPORTIONAL DAMPING

Because of the unconditional stability of the average acceleration method, it is the most robust method to be used for the step-by-step dynamic analysis of large complex structural systems in which a large number of high frequencies, short periods, are present. The only problem with the method is that the short periods, which are smaller than the time step, oscillate indefinitely after they are excited. The higher mode oscillation can be reduced by the addition of stiffness proportional damping. The additional damping that is added to the system is of the form

$$C_D = \delta \ K \tag{20.16}$$

where the modal damping ratio, given by Equation (8.8) is defined by

$$\xi_n = \frac{1}{2}\delta \ \omega_n = \frac{\pi}{T_n}\delta \tag{20.17}$$

One notes that the damping is large for short periods and small for the long periods or low frequencies. It is apparent that periods which are greater than the time step cannot be integrated accurately by any direct integration method. Therefore, it is logical to damp these short periods to prevent them from oscillating during the solution procedure. For a time step equal to the period, Equation (20.17) can be rewritten as

$$\delta = \xi_n \frac{\Delta T}{\pi} \tag{20.18}$$

Hence, if the integration time step is 0.02 seconds and we wish to assign a minimum of 1.0 to all periods shorter than the time step, a value of $\delta = 0.0064$ should be used. The damping ratio in all modes is now predictable for this example from Equation (20.17). Therefore, the damping ratio for a 1.0 second period is 0.02 and for a 0.10 second period is 0.2.

20.7 THE HILBER, HUGHES AND TAYLOR α METHOD

The α method [4] uses the Newmark method to solve the following modified equations of motion:

$$\begin{aligned} \mathbf{M}\ddot{\mathbf{u}}_t + (1+\alpha)\,\mathbf{C}\dot{\mathbf{u}}_t + (1+\alpha)\,\mathbf{K}\mathbf{u}_t &= (1+\alpha)\mathbf{F}_t \\ &\quad - \alpha\mathbf{F}_t + \alpha\,\mathbf{C}\dot{\mathbf{u}}_{t-\Delta t} + \alpha\mathbf{K}\mathbf{u}_{t-\Delta t} \end{aligned} \tag{20.19}$$

With α equals zero the method reduces to the constant acceleration method. It produces numerical energy dissipation in the higher modes; however, it cannot be predicted as a damping ratio as in the use of stiffness proportional damping. Also, it does not solve the fundamental equilibrium equation at time t. However, it is currently being used in many computer programs. The performance of the method appears to be very similar to the use of stiffness proportional damping.

20.8 SELECTION OF A DIRECT INTEGRATION METHOD

It is apparent that a large number of different direct numerical integration methods are possible by specifying different integration parameters. A few of the most commonly used methods are summarized in 20.2.

Table 20.2. Summary of Newmark Methods Modified by the δ Factor

METHOD	γ	β	δ	$\frac{\Delta t}{T_{MIN}}$	ACCURACY
Central Difference	1/2	0	0	0.3183	Excellent for small Δt Unstable for large Δt
Linear Acceleration	1/2	1/6	0	0.5513	Very good for small Δt Unstable for large Δt
Average Acceleration	1/2	1/4	0	∞	Good for small Δt No energy dissipation
Modified Average Acceleration	1/2	1/4	$\frac{\Delta T}{\pi}$	∞	Good for small Δt Energy dissipation for large Δt

For single degree-of-freedom systems the central difference method is most accurate; and the linear acceleration method is more accurate than the average acceleration method. However, if only single degree-of-freedom systems are to be integrated the piece-wise exact method, previously presented, should be used since there is no need to use an approximate method.

It appears that the modified average acceleration method, with a minimum addition of stiffness proportional damping, is a general procedure that can be used for the dynamic analysis of all structural systems. Using $\delta = \Delta T / \pi$ will damp out periods shorter than the time step and introduces a minimum error in the long period response.

20.9 NONLINEAR ANALYSIS

The basic Newmark Constant acceleration method can be extended to nonlinear dynamic analysis. This requires that iteration must be performed at each time step in order to satisfy equilibrium. Also, the incremental stiffness matrix must be formed and triangularized at each iteration or at selective points in time. Many different numerical tricks, including element by element methods, have been developed in order to minimize the computational requirements. Also, the triangularization of the effective incremental stiffness matrix may be avoided by the introduction of iterative solution methods.

20.10 SUMMARY

For earthquake analysis of linear structures, it should be noted that the direct integration of the dynamic equilibrium equations is normally not numerically efficient as compared to the mode superposition method using LDR vectors. If the triangularized stiffness and mass matrices and other vectors cannot be stored in high-speed storage, the computer execution time can be large.

After using direct integration methods for approximately forty years, the author can no longer recommend the Wilson method for the direct integration of the dynamic equilibrium equations. The Newmark constant acceleration method, with the addition of very small amount of stiffness proportional damping, is recommended for dynamic analysis nonlinear structural systems. For all methods of direct integration great care should be taken to make certain that the stiffness proportional damping does not eliminate important high-frequency response. Mass proportional damping cannot be justified because it causes external forces to be applied to the structure that reduce the base shear for seismic loading.

In the area of nonlinear dynamic analysis one cannot prove that any one method will always converge. One should always check the error in the conservation of energy for every solution obtained. In future editions of this book it is hoped that numerical examples will be presented in order that the appropriate method can be recommended for different classes of problems in structural analysis.

20.11 REFERENCES

1. Newmark, N. M., "A Method of Computation for Structural Dynamics", ASCE Journal of the Engineering Mechanics Division, Vol. 85 No. EM3, (1959).

2. Wilson, E. L., "Dynamic Response by Step-By-Step Matrix Analysis", Proceedings, Symposium On The Use of Computers in Civil Engineering, Labortorio Nacional de Engenharia Civil, Lisbon, Portugal, October 1-5, (1962).

3. Wilson, E. L., I. Farhoomand and K. J. Bathe, "Nonlinear Dynamic Analysis of Complex Structures", *Earthquake Engineering and Structural Dynamics*, 1, 241-252, (1973).

4. Hughes, Thomas, "The Finite Element Method - Linear Static and Dynamic Finite Element Analysis", Prentice Hall, Inc., (1987).

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

THE EIGENVALUE PROBLEM

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

THE EIGENVALUE PROBLEM

Eigenvalues And Eigenvectors Are Properties Of The Equations That Simulate The Behavior Of A Real Structure

D.1 INTRODUCTION

The classical mathematical eigenvalue problem is defined as the solution of the following equation:

$$\mathbf{A}\mathbf{v}_n = \lambda_n \mathbf{v}_n \quad n = 1......N \tag{D.1}$$

The N by N $\mathbf{A}$ matrix is real and symmetric; however, it may be singular and have zero eigenvalues λ_n. A typical eigenvector $\mathbf{v}_n$ has the following orthogonality properties:

$$\begin{aligned} &\mathbf{v}_n^T \mathbf{v}_n = 1 \quad \text{and} \quad \mathbf{v}_n^T \mathbf{v}_m = 0 \quad \text{if } n \neq m, \text{ therefore} \\ &\mathbf{v}_n^T \mathbf{A} \mathbf{v}_n = \lambda_n \text{ and } \quad \mathbf{v}_n^T \mathbf{A} \mathbf{v}_m = 0 \quad \text{if } n \neq m \end{aligned} \tag{D.2}$$

If all eigenvectors $\mathbf{V}$ are considered the problem can be written as

$$\mathbf{AV} = \mathbf{V}\Omega \quad \text{or} \quad \mathbf{V}^T\mathbf{AV} = \Omega \tag{D.3}$$

There are many different numerical methods to solve Equation (D.3) for eigenvectors $\mathbf{V}$ and the diagonal matrix of eigenvalues Ω. In structural analysis, in general, it is only necessary to solve for the exact eigenvalues of small systems. Therefore, the most reliable and robust will be selected since the computational time will always be relatively small. For the determination of the dynamic mode shapes and frequencies of large structural systems subspace iteration or Load Dependent Ritz, LDR, vectors are the most efficient approaches.

D.2 THE JACOBI METHOD

One of the oldest and most general approaches for the solution of the classical eigenvalue problem is the Jacobi method that was first presented in 1846. This is a simple iterative algorithm in which the eigenvectors are calculated from the following series of matrix multiplications:

$$\mathbf{V} = \mathbf{T}^{(0)}\mathbf{T}^{(1)}.....\mathbf{T}^{(k)}.........\mathbf{T}^{(n-1)}\mathbf{T}^{(n)} \tag{D.4}$$

The starting transformation matrix $\mathbf{T}^{(0)}$ is set to a unit matrix. The iterative orthogonal transformation matrix $\mathbf{T}^{(k)}$, with four nonzero terms in the i and j rows and columns, is of the following orthogonal form:

$$\mathbf{T}^{(k)} = \begin{bmatrix} - & - & - & - & - & - & - & - \\ - & - & - & - & - & - & - & - \\ - & - & T_{ii} & - & - & T_{ij} & - & - \\ - & - & - & - & - & - & - & - \\ - & - & - & - & - & - & - & - \\ - & - & T_{ji} & - & - & T_{jj} & - & - \\ - & - & - & - & - & - & - & - \\ - & - & - & - & - & - & - & - \end{bmatrix} \tag{D.5}$$

The four nonzero terms are functions of an unknown rotation angle θ and are defined by

$$T_{ii} = T_{jj} = \cos\theta \quad \text{and} \quad T_{ji} = -T_{ij} = \sin\theta \tag{D.6}$$

Therefore, $\mathbf{T}^{(k)T}\mathbf{T}^{(k)} = \mathbf{I}$ which is independent of the angle θ. The typical iteration involves the following matrix operation:

$$\mathbf{A}^{(k)} = \mathbf{T}^{(k)T}\mathbf{A}^{(k-1)}\mathbf{T}^{(k)} \tag{D.7}$$

The angle is selected to force the terms *i,j* and *j,i* in the matrix $\mathbf{A}^{(k)}$ to be zero. This is satisfied if the angle is calculated from

$$\tan 2\theta = \frac{2A_{ij}^{(k-1)}}{A_{ii}^{(k-1)} - A_{jj}^{(k-1)}} \tag{D.8}$$

The classical Jacobi eigenvalue algorithm is summarized within the computer subroutine given in Table D.1.

Table D.1 Subroutine to Solve the Symmetric Eigenvalue Problem

```
      SUBROUTINE JACOBI(A,V,NEQ,TL)
      IMPLICIT REAL*8 (A-H,O-Z)
      DIMENSION A(NEQ,NEQ),V(NEQ,NEQ)
C     EIGENVALUE SOLUTION BY JACOBI METHOD -
C     WRITTEN BY ED WILSON DEC. 25, 1990
C     A - MATRIX (ANY RANK) TO BE SOLVED ---
C         EIGENVALUES ON DIAGONAL
C     V - MATRIX OF EIGENVECTORS PRODUCED
C     TL- NUMBER OF SIGNIFICANT FIGURES
C---- INITIALIZATION ---------------------
      ZERO = 0.0D0
      SUM = ZERO
      TOL = DABS(TL)
C---- SET INITIAL EIGENVECTORS ------------
      DO 200 I=1,NEQ
      DO 190 J=1,NEQ
      IF (TL.GT.ZERO) V(I,J) = ZERO
  190 SUM = SUM + DABS(A(I,J))
      IF (TL.GT.ZERO) V(I,I) = 1.0
  200 CONTINUE
C---- CHECK FOR TRIVIAL PROBLEM -----------
      IF (NEQ.EQ.1)    RETURN
      IF (SUM.LE.ZERO) RETURN
      SUM = SUM/DFLOAT(NEQ*NEQ)
C----------------------------------------
C---- REDUCE MATRIX TO DIAGONAL -----------
C----------------------------------------
  400 SSUM = ZERO
      AMAX = ZERO
      DO 700 J=2,NEQ
      IH = J - 1
      DO 700 I=1,IH
C---- CHECK IF A(I,J) IS TO BE REDUCED -----
      AA = DABS(A(I,J))
      IF (AA.GT.AMAX) AMAX = AA
      SSUM = SSUM + AA
      IF (AA.LT.0.1*AMAX) GO TO 700
C---- CALCULATE ROTATION ANGLE ----------
      AA=ATAN2(2.0*A(I,J),A(I,I)-(J,J))/2.0
      SI = DSIN(AA)
      CO = DCOS(AA)
C---- MODIFY "I" AND "J" COLUMNS --------
      DO 500 K=1,NEQ
      TT = A(K,I)
      A(K,I) =  CO*TT + SI*A(K,J)
      A(K,J) = -SI*TT + CO*A(K,J)
      TT = V(K,I)
      V(K,I) =  CO*TT + SI*V(K,J)
 500  V(K,J) = -SI*TT + CO*V(K,J)
C---- MODIFY DIAGONAL TERMS -------------
      A(I,I) = CO*A(I,I) + SI*A(J,I)
      A(J,J) =-SI*A(I,J) + CO*A(J,J)
      A(I,J) = ZERO
C---- MAKE "A" MATRIX SYMMETRICAL -------
      DO 600 K=1,NEQ
      A(I,K) =  A(K,I)
      A(J,K) =  A(K,J)
  600 CONTINUE
C---- A(I,J) MADE ZERO BY ROTATION ------
  700 CONTINUE
C---- CHECK FOR CONVERGENCE -------------
      IF(DABS(SSUM)/SUM .GT.TOL)GO TO 400
      RETURN
      END
```

One notes that the subroutine for the solution of the symmetric eigenvalue problem by the classical Jacobi method does not contain a division by any number. Also, it can be proved that, after each iteration cycle, the absolute sum of the off-diagonal terms is always reduced. Hence, the method will always converge and yield an accurate solution for positive, zero or negative eigenvalues.

The Jacobi algorithm can be directly applied to all off-diagonal terms, in sequence, until all terms are reduced to a small number compared to the absolute value of all terms in the matrix. However, the subroutine presented uses a "threshold" approach in which it skips the relatively small off-diagonal terms and operates only on the large off-diagonal terms.

To reduce one off-diagonal term to zero requires approximately 8N numerical operations. Clearly, one cannot precisely predict the total number of numerical operation because it is an iterative method; however, experience has indicated that the total number of numerical operations to obtain convergence is the order of $10N^3$. Assuming a modern (1998) personal computer can perform over 6,000,000 operations a second, it would require approximately one second of computer time to calculate the eigenvalues and eigenvectors of a full 100 by 100 matrix.

D.3 CALCULATION OF 3D PRINCIPAL STRESSES

The calculation of the principal stresses for a three dimensional solid can be numerically evaluated from the stresses in the x-y-z system by the solution of a cubic equation. However, the definition of the directions of the principal stresses is not a simple procedure. An alternative approach to this problem is to write the basic stress transformation equation in terms of the unknown directions of the principal stresses in the 1-2-3 reference system. Or,

$$\begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix} = \begin{bmatrix} V_{x1} & V_{y1} & V_{z1} \\ V_{x2} & V_{y2} & V_{z2} \\ V_{x3} & V_{y3} & V_{z3} \end{bmatrix} \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \begin{bmatrix} V_{x1} & V_{x2} & V_{x3} \\ V_{y1} & V_{y2} & V_{y3} \\ V_{z1} & V_{z2} & V_{z3} \end{bmatrix} \tag{D.9}$$

Or, in symbolic form

$$\Omega = \mathbf{V}^T \mathbf{S} \mathbf{V} \tag{D.10}$$

in which $\mathbf{V}$ is the standard direction cosine matrix. Since $\mathbf{V}\,\mathbf{V}^T$ is a unit matrix Equation (D.3) can be written as the following eigenvalue problem:

$$\mathbf{SV} = \mathbf{V\Omega} \tag{D.11}$$

where Ω is an unknown diagonal matrix of the principal stresses (eigenvalues) and $\mathbf{V}$ is the unknown direction cosine matrix (eigenvectors) which uniquely define the directions of the principal stresses. In order to illustrate the practical application of the classical Jacobi method consider the following state of stress:

$$\mathbf{S} = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} = \begin{bmatrix} 120 & -55 & -75 \\ -55 & -55 & 33 \\ -75 & 33 & -85 \end{bmatrix} \tag{D.12}$$

The eigenvalues, principal stresses, and eigenvectors (direction cosines) are

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \end{bmatrix} = \begin{bmatrix} 162.54 \\ -68.40 \\ -114.14 \end{bmatrix} \quad \text{and} \quad \mathbf{V} = \begin{bmatrix} .224 & .352 & .909 \\ -.308 & .910 & -.277 \\ .925 & .217 & -.312 \end{bmatrix} \tag{D.13}$$

The solution of a 3 by 3 eigenvalue problem can be considered as a trivial numerical problem. Several hundred of these problems can be solved in one second of computer time.

D.4 SOLUTION OF THE GENERAL EIGENVALUE PROBLEM

The general eigenvalue problem is written as

$$\mathbf{AV} = \mathbf{BV\Omega} \tag{D.14}$$

where both $\mathbf{A}$ and $\mathbf{B}$ are symmetrical matrices. The first step is to calculate the eigenvectors $\mathbf{V}_B$ of the $\mathbf{B}$ matrix. We can now let the eigenvectors $\mathbf{V}$ be a linear combination or the eigenvectors of the $\mathbf{B}$ matrix. Or,

$$\mathbf{V} = \mathbf{V}_B \overline{\mathbf{V}} \tag{D.15}$$

Substitution of Equation (D.15) into Equation (D.14) and the pre multiplication of both sides by $\mathbf{V}_B^T$ yields

$$\mathbf{V}_B^T \mathbf{A} \mathbf{V}_B \overline{\mathbf{V}} = \mathbf{V}_B^T \mathbf{B} \mathbf{V}_B \overline{\mathbf{V}} \Omega \tag{D.16}$$

If all eigenvalues of the **B** matrix are nonzero the eigenvectors can be normalized so that $\mathbf{V}_B^T \mathbf{B} \mathbf{V}_B = \mathbf{I}$. Hence, Equation (D.16) can be written in the following classical form:

$$\overline{\mathbf{A}}\, \overline{\mathbf{V}} = \overline{\mathbf{V}} \Omega \tag{D.17}$$

where $\overline{\mathbf{A}} = \mathbf{V}_B^T \mathbf{A} \mathbf{V}_B$. Therefore, the general eigenvalue problem can be solved by the application of the Jacobi algorithm to both matrices. If the **B** matrix is diagonal the eigenvectors $\mathbf{V}_B$ matrix will be diagonal with the diagonal terms equal to $1/\sqrt{B_{nn}}$. This is the case for a lumped mass matrix. Also, mass must be associated with all degrees-of-freedom and all eigenvectors and values must be calculated.

D.5 SUMMARY

Only the Jacobi method has been presented in detail in this section. It is restricted to small full matrices in which all eigenvalues are required. For this problem the method is very robust and simple to program. For the dynamic modal analysis of large structural systems; or, for the stability analysis of structural systems, other more numerically efficient methods are recommended.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

SPEED OF COMPUTER SYSTEMS

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

SPEED OF COMPUTER SYSTEMS

The Current Speed Of A \$2,000 Personal Computer Is Faster Than The \$10,000,000 Cray Computer Of 1975

H.1 INTRODUCTION

The calculation of element stiffness matrices, solution of equations and evaluation of mode shapes and frequencies are all computationally intensive. Furthermore, it is necessary to use double-precision floating-point arithmetic to avoid numerical errors. Therefore, all numbers must occupy 64 bits of computer storage. The author started developing structural analysis and design programs on the IBM-701 in 1957 and since that time has been exposed to a large number of different computer systems. In this section the approximate double-precision floating-point performances of some of these computer systems are summarized. Since different FORTRAN compilers and operating systems were used the speeds presented can only be considered accurate to within 50 percent.

H.2 DEFINITION OF ONE NUMERICAL OPERATION

For the purpose of comparing floating-point speeds the evaluation of the following equation is defined as one operation:

$$A = B + C * D \qquad \text{Definition of one numerical operation}$$

Using double precision arithmetic, the definition involves the sum of one multiplication, one addition, extracting three numbers from high-speed storage, and transferring the results to storage. In most cases, this type of operation is within the inner DO LOOP for the solution of linear equations and the evaluation of mode shapes and frequencies.

H.3 SPEED OF DIFFERENT COMPUTER SYSTEMS

Table H.1 indicates the speed of different computers used by the author.

Table H.1. Floating-Point Speeds of Computer Systems

YEAR	COMPUTER Or CPU	OPERATIONS PER SECOND	RELATIVE SPEED
1963	CDC-6400	50,000	1.0
1967	CDC-6600	100,000	2.0
1974	CRAY-1	3,000,000	60.0
1980	VAX-780	60,000	1.2
1981	IBM-3090	20,000,000	400.
1981	CRAY-XMP	40,000,000	800.
1990	DEC-5000	3,500,000	70.
1994	Pentium-90	3,500,000	70.
1995	Pentium-133	5,200,000	104.
1995	DEC-5000 upgrade	14,000,000	280.
1998	Pentium II - 333	16,500,000	330.

If one considers the initial cost and maintenance of the various computer systems, it is apparent that the overall cost of engineering calculations has reduced significantly during the past 20 years. The most cost effective computer system, at the present time, is the INTEL Pentium type of personal computer system. At the present time, a very powerful personal computer system, that is five times faster than the first CRAY computer, can be purchased for approximately $2,500.

H.4 SPEED OF PERSONAL COMPUTER SYSTEMS

Many engineers do not realize the computational power of the present day inexpensive personal computer. Table H.2 indicates the increased speed of personal computers that has occurred during the past 18 years.

Table H.2. Floating-Point Speeds of Personal Computer Systems

YEAR	INTEL CPU	Speed MHz	Operations Per Second	Relative Speed	COST
1980	8080	4	200	1	$6,000
1984	8087	10	13,000	65	$2,500
1988	80387	20	93,000	465	$8,000
1991	80486	33	605,000	3,025	$10,000
1994	80486	66	1,210,000	6,050	$5,000
1995	Pentium	90	4,000,000	26,000	$5,000
1996	Pentium	233	10,300,000	52,000	$4,000
1997	Pentium II	233	11,500,000	58,000	$3,000
1998	Pentium II	333	16,500,000	82,500	$2,500

One notes that the floating-point speed of the Pentium II is not significantly different than the basic Pentium chip. The increase in clock speed, from 90 to 333 MHz, has accounted for the increase in speed during the last three years.

H.5 PAGING OPERATING SYSTEMS

The above computer speeds assume all numbers are in high-speed memory. For the analysis of large structural systems it is not possible to store all information within high-speed storage. If data needs to be obtained from low-speed disk storage, the effective speed of a computer can be reduced significantly. Within the SAP and ETABS programs the transfer of data to and from disk storage is conducted in large blocks in order to minimize disk access time. This programming philosophy was used prior to the introduction of the paging option used in the modern Windows operating systems.

In a paging operating system, if the data requested is not stored in high-speed memory, the computer automatically reads the data from disk storage in relatively small blocks of information. Therefore, the modern programmer need not be concerned with data management. However, there is a danger in the application of this approach. The classical example, that illustrates the problem with paging, is the following example of adding two large matrices together. The FORTRAN statement can be one of the following forms:

```
    DO 100 J=1,NCOL                   DO 100 I=1,NROW
    DO 100 I=1,NROW                   DO 100 J=1,NCOL
100 A(I,J)=B(I,J)+C(I,J)          100 A(I,J)=B(I,J)+C(I,J)
```

Since all arrays are stored row-wise, the data will be paged to and from disk storage in the same order as needed by the program statements on the left. However, if the program statements on the right are used the computer may be required to read and write blocks of data to the disk for each term in the matrix. Hence, the computer time required for this simple operation can be very large if paging is automatically used.

H.6 SUMMARY

Personal computers will continue to increase in speed and decrease in price. Intel's Merced 64-bit CPU chip will be released in 1999. It is the opinion of many experts in the field that the only way significant increases in speed will occur is by the addition of multi-processors to the personal computer systems. The NT operating system supports the use of multi-processors.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

PROBLEM H

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

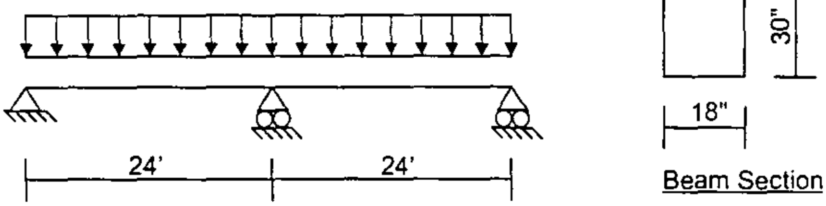
Prob!em H

Reinforced Concrete Beam

Concrete
E =3600 ksi, Poissons Ratio = 0.2
f'c = 4 ksi
fy = 60 ksi
Concrete cover to longitudinal rebar center at top of beam = 3.5 in
Concrete cover to longitudinal rebar center at bottom of beam = 2.5 in

To Do
Determine required longitudinal reinforcing steel and required shear stirrups based on ACI 318-95

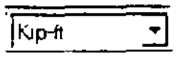
w_{DL} = 2.2 klf (unfactored, doesn't include beam weight)
w_{LL} = 1.6 klf (unfactored)



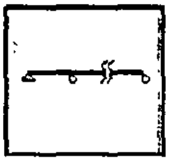
Beam Section

Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem H Solution

1. Click the drop down box in the status bar to change the units to kip-ft. 

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Beam** template  button to display the Beam dialog box.

4. In this dialog box:

   - Accept all of the default values.
   - Click the **OK** button.

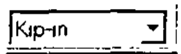
5. Click the "X" in the top right-hand corner of the 3-D View window to close it.

6. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

7. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

8. In this dialog box:

   - Verify 0.15 is entered in the Weight per Unit Volume edit box.
   - Click the **OK** button twice to exit all dialog boxes.

9. Click the drop down box in the status bar to change the units to kip-in. 

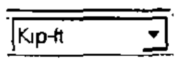
10. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

11. Click on CONC in the Materials area to highlight (select) it. and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

12. In this dialog box:

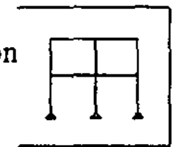
    - Verify 3600 is entered in the Modulus of Elasticity edit box.
    - Verify .2 is entered in the Poisson's Ratio edit box.
    - Verify 60 is entered in the Reinforcing Yield Stress, fy edit box.
    - Verify 4 is entered in the Concrete Strength, fc edit box.
    - Type **60** in the Shear steel Yield Stress, fys edit box.

- Verify 4 is entered in the Concrete Shear Strength. fcs edit box.
- Accept the other default values.
- Click the **OK** button twice to exit all dialog boxes.

13. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.
14. In this dialog box:
    - With the default section, FSEC1, highlighted, click the **Modify/Show Section** button to display the Rectangular Section dialog box.
    - In this dialog box:
        - ➢ Select CONC from the Material drop-down box.
        - ➢ Type **30** in the Depth (t3) edit box.
        - ➢ Type **18** in the Width (t2) edit box.
        - ➢ Click the **Reinforcement** button to display the Reinforcement Data dialog box.
        - ➢ In this dialog box:
            - ✓ In the Element Class area select the Beam option.
            - ✓ In the Concrete Cover To Rebar Center area type **3.5** in the Top edit box.
            - ✓ In the Concrete Cover To Rebar Center area type **2.5** in the Bottom edit box.
            - ✓ Click the **OK** button three times to exit all dialog boxes.
15. Click the drop down box in the status bar to change the units to kip-ft. 

16. From the **Define** menu select **Static Load Cases....** This will display the Define Static Load Case Names dialog box.
17. In this dialog box:
    - Type **DL** in the Load edit box.
    - Click the **Change Load** button
    - Type **LL** in the Load edit box.
    - Select LIVE from the Type drop-down box.

- Type **0** in the Self weight Multiplier box.
- Click the **Add New Load** button.
- Click the **OK** button.

18. Select the two frame elements.
19. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
20. In this dialog box:
    - Verify that the Load Case Name is DL.
    - In the Load Type and Direction area verify that the Forces option is selected and that the Global Z direction is selected.
    - In the Uniform Load area type **-2.2**.
    - Click the **OK** button.
21. Select the two frame elements.
22. From the **Assign** menu select **Frame Static Loads...** and then **Point and Uniform...** from the submenu to display the Point and Uniform Span Loads dialog box.
23. In this dialog box:
    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-1.6**.
    - Click the **OK** button.
24. From the **Analyze** menu select **Set Options...** to display the Analysis Options dialog box.
    - In this dialog box click the **Plane Frame XZ Plane** button  to set the available degrees of freedom.
    - Click the **OK** button.
25. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
26. In this dialog box:
    - Select the CONC Tab.

- Select ACI 318-95 from the Concrete Design Code drop-down box.

- Verify that the Strength Reduction Factors are 0.9, 0.85, 0.7 and 0.75 for Bending/Tension, Shear, Compression (T) and Compression respectively.

- Click the **OK** button.

27. Click the **Run Analysis** button to run the analysis.

28. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.

29. On the **Design** menu verify that the **Concrete Design** feature is active. There should be a check mark next **Concrete Design** if it is active.

30. From the **Design** menu click Select Design Combos... to display the Design Load Combinations Selection dialog box.

31. In this dialog box:

    - Verify that the default combinations for concrete design, DCON1 and DCON2 are included in the Design Combos list box.

    - Highlight DCON1 and click the **Show** button to display the Load Combination Data dialog box.

    - In this dialog box:

        - Note the definition of the load combination in the Define Combination area. It should be 1.4DL.

        - Click the **OK** button to return to the Design Load Combinations Selection dialog box.

    - Highlight DCON2 and click the **Show** button to display the Load Combination Data dialog box.

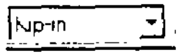
    - In this dialog box:

        - Note the definition of the load combination in the Define Combination area. It should be 1.4DL + 1.7LL.

        - Click the **OK** button twice to exit all dialog boxes.

32. From the **Design** menu select **Start Design/Check of Structure** to run the design check.

33. When the design is done the area of longitudinal bar required is displayed on the screen. Note that the current units are kips and feet.

34. Click the drop down box in the status bar to change the units to kip-in. 

Note that the values for the area of longitudinal reinforcing steel are now in units of square inches.

35. From the **Design** menu select **Display Design Info...** to display the Display Design Results dialog box.

36. In this dialog box:

    - Verify that the Design Output option is selected.
    - Select Shear Reinforcing from the Design Output drop-down box.
    - Click the **OK** button. The required shear reinforcing is displayed on the screen.

Note that the values for the shear reinforcing steel are reported as an area per unit length of element. Since the current units are kips and inches, the shear reinforcing reported is in square inches per inch.

37. Right click on the left beam to display the Concrete Design Information dialog box.

38. In this dialog box:

    - Note that the required top and bottom longitudinal steel and the required shear steel is reported for each design load combination at each output segment along the beam.
    - Click the **Details** button to see design details for the highlighted design load combination and output station location. The Concrete Design Information ACI 318-95 dialog box is displayed.
    - When finished viewing the detailed information click the "X" in the upper right-hand corner of the Concrete Design Information ACI 318-95 dialog box to close it.
    - Click **OK** to close the Concrete Design Information dialog box.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAFE TECHNICAL NOTE 2

PROBLEM X

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

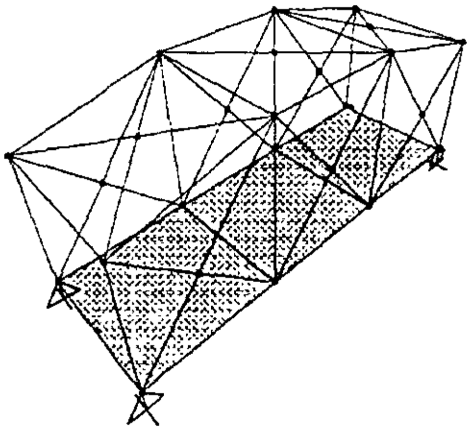
Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 ■ Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 ■ Fax: 5510.0573

Problem X

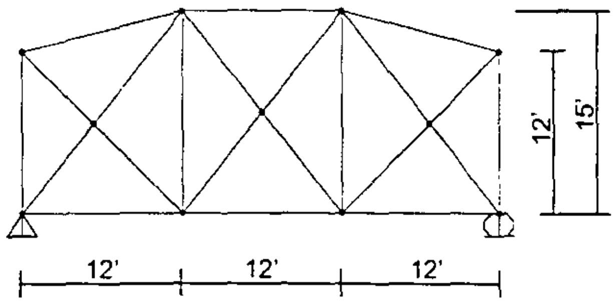
Through Truss Bridge

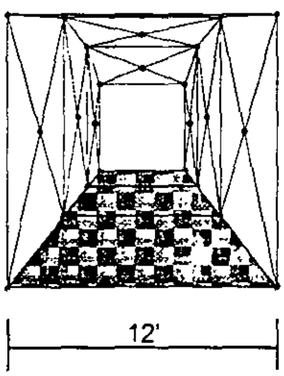
Steel
E =29000 ksi
Poissons Ratio = 0.3
All members are W6X12
Fy = 36 ksi

Concrete Bridge Deck
E =3600 ksi
Poissons Ratio = 0.2
12 inches thick
Live Load = 250 psf



To Do
Review steel member stresses due to self weight plus live load.
Use AISC-ASD89.





Note: Our intent is that you try this problem on your own first. After you have solved it on your own, you can step through our solution if desired. If you have problems trying to create the model, then follow the steps in our solution.

Problem X Solution

1. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

2. From the **File** menu select **New Model From Template....** This displays the Model Templates dialog box.

3. In this dialog box click on the **Vertical Truss** template  button to display the Vertical Truss dialog box.

4. In this dialog box:

    - Accept the default number of bays, 3.
    - Accept the default Height of Truss, 12.
    - Accept the default Truss Bay Length, 12.
    - Click the **OK** button.

5. Click the **Select All** button on the side toolbar to select all elements.

6. From the **Edit** menu select **Replicate...** to display the Replicate dialog box.

7. In this dialog box:

    - Verify the Linear Tab is selected.
    - In the Distance area type **12** in the Y edit box.
    - Verify that 0 is entered in the X and Z edit boxes.
    - Verify that 1 is entered in the Number edit box.
    - Click the **OK** button to proceed with the replication.

8. From the **Draw** menu select **Edit Grid...** to display the Modify Grid Lines dialog box.

9. In this dialog box:

    - Select the Y option in the Direction area.
    - Type **12** in the Y Location edit box and click the **Add Grid Line** button.
    - Click the **OK** button.

10. Click in the window titled X-Z Plane @ Y=0 to activate it. The window is activated when its title bar is highlighted.

11. Click the **xy 2D View** button to change the view to an X-Y plan. Note that the title of the window changes to X-Y Plane @ Z=12. The screen appears as shown in Figure X-1.

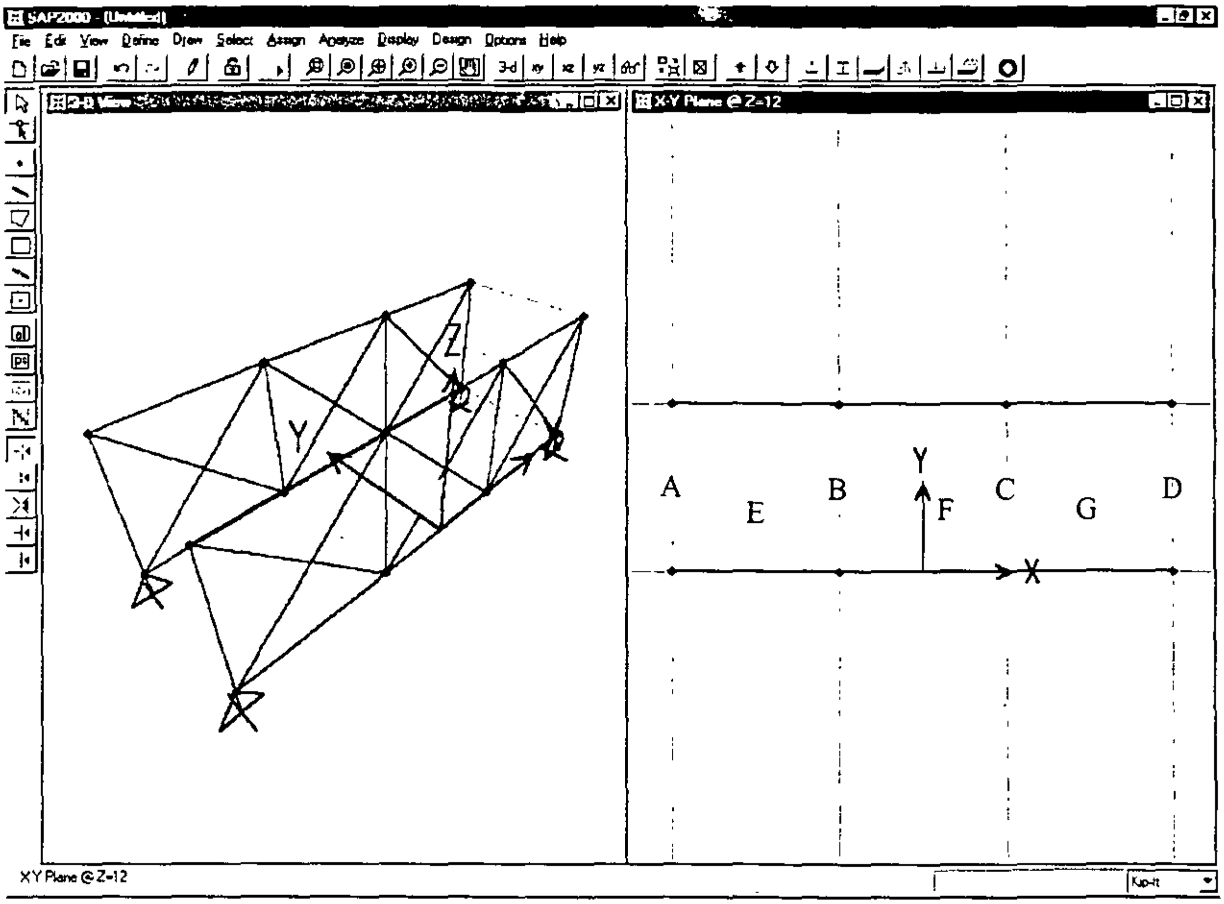


Figure X-1: View of Screen After Step 11

12. Click the **Quick Draw Frame Element** button on the side toolbar or select **Quick Draw Frame Element** from the **Draw** menu.

13. Click on the grid lines at the points labeled "A", "B", "C" and "D" in Figure X-1 to enter four frame elements spanning between the two vertical frames.

14. Click on the grid lines at the points labeled "E", "F" and "G" in Figure X-1 to enter three sets of diagonal frame elements spanning between the two vertical frames.

15. Click the **Down One Gridline** button on the main toolbar to display the X-Y Plane @ Z=0. The screen appears as shown in Figure X-2.

16. Click the **Quick Draw Rectangular Shell Element** button on the side toolbar (or select **Quick Draw Rectangular Shell Element** from the **Draw** menu).

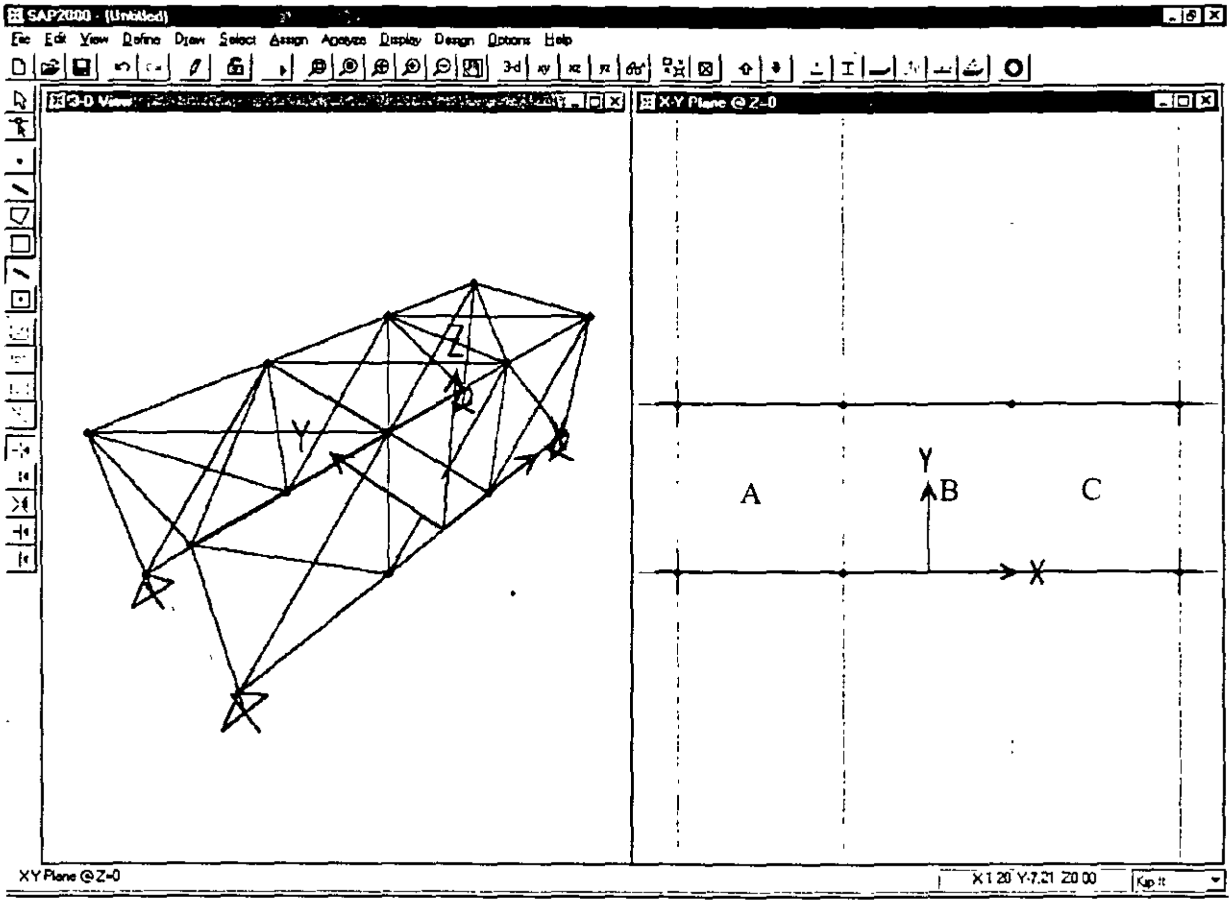


Figure X-2: View of Screen After Step 15

17. Click on the points labeled "A", "B" and "C" in Figure X-2 to enter three shell elements spanning between the two vertical frames.

18. Click the **Pointer** button on the side tool bar to exit draw mode and enter select mode.

19. Click in the window titled X-Y Plane @ Z=0 to make sure it is active.

20. Click the **Up One Gridline** button on the main toolbar to display the plan view at Z=12.

21. Select the center four joints by clicking on them.

22. From the **Edit** menu select **Move...** to display the Move Selected Points dialog box.

23. In this dialog box:

    - Type **3** in the Delta Z edit box.
    - Click the **OK** button.

24. Click the **Perspective Toggle** button on the main toolbar. A perspective birds-eye view of the structure is displayed.

25. Click the **Set Intersecting Line Select Mode** button and select all of the roof and floor elements by "drawing" a line through them. There should now be 10 frames and 3 shells selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

26. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.

27. In this dialog box:

    - Click on the Break At Intersections With Selected Frames and Joints option to select it.

    - Click the **OK** button to add a center joint at each set of cross braces in the roof.

28. Click on the **xz 2D View** button on the main toolbar to view an elevation in the X-Z plane. Note the title of the window is X-Z Plane @ Y=0.

29. Click the **Perspective Toggle** button on the main toolbar. A perspective view of the structure is displayed.

30. Click the **Set Intersecting Line Select Mode** button and select all of the vertical and diagonal elements by "drawing" a line through them. There should now be 20 frames selected. You can verify this by looking at the left-hand side of the status bar at the bottom of the SAP2000 window.

31. From the **Edit** menu select **Divide Frames...** to display the Divide Selected Frames dialog box.

32. In this dialog box:

    - Verify that the Break At Intersections With Selected Frames and Joints option is selected.

    - Click the **OK** button to add a center joint at each of the six sets of vertical cross braces.

33. Click on the **xz 2D View** button on the main toolbar to view an elevation in the X-Z plane.

34. Click the drop down box in the status bar to change the units to kip-in. Kip-in

35. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

36. Click on STEEL in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

37. In this dialog box:

    - Verify that the Modulus of Elasticity is 29000.
    - Verify that Poisson's Ratio is 0.3.
    - Verify that the steel stress, Fy is 36.
    - Click the **OK** button.

38. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

39. In this dialog box:

    - Verify that the Modulus of Elasticity is 3600.
    - Verify that Poisson's Ratio is 0.2.
    - Click the **OK** button twice to exit all dialog boxes.

40. Click the drop down box in the status bar to change the units to kip-ft. Kip-ft

41. From the **Define** menu select **Materials...** to display the Define Materials dialog box.

42. Click on CONC in the Materials area to highlight (select) it, and then click the **Modify/Show Material** button. The Material Property Data dialog box is displayed.

43. In this dialog box:

    - Verify that the Weight per Unit Volume is 0.15.
    - Click the **OK** button twice to exit all dialog boxes.

44. From the **Define** menu select **Frame Sections...** to display the Define Frame Sections dialog box.

45. In this dialog box:

    - In the Click To area, click the drop-down box that says Import I/Wide Flange and then click on the Import I/Wide Flange item.
    - If the Section Property File dialog box appears then locate the Sections.pro file which should be located in the same directory as the SAP2000 program files. Highlight Sections.pro and click the **Open** button.
    - A dialog box appears with a list of all wide flange sections in the database. In this dialog box:

- ➢ Scroll down and click on the W6X12 section.
- ➢ Click the **OK** button three times to exit all dialog boxes.

46. From the **Define** menu select **Shell Sections...** to display the Define Shell Sections dialog box.

47. In the Click To area, click the **Modify/Show Section** button to display the Shell Sections dialog box.

48. In this dialog box:
    - Verify that the Material chosen is CONC.
    - Verify that both the Membrane and the Bending thicknesses are 1.
    - Verify that the Shell option is selected in the Type area.
    - Click the **OK** button twice to exit all dialog boxes.

49. Click the **Select All** button on the side toolbar to select all elements.

50. From the **Assign** menu select **Frame** and then **Sections...** from the submenu to display the Define Frame Sections dialog box.

51. In this dialog box:
    - Click on W6X12 in the Frame Sections area to highlight it.
    - Click the **OK** button.

52. Click the **Show Undeformed Shape** button to remove the displayed frame section assignments.

53. From the **Define** menu select **Static Load Cases...** to display the Define Static Load Case Names dialog box.

54. In this dialog box:
    - Type **DL** in the Load Edit box.
    - Click the **Change Load** button.
    - Type **LL** in the Load Edit box.
    - Select Live from the Type drop-down box.
    - Type **0** in the Self Weight Multiplier edit box.

- Click the **Add New Load** button.
- Click the **OK** button.

55. Click the **Select All** button on the side toolbar to select all elements.
56. From the **Assign** menu select **Shell Static Loads...** and then **Uniform...** from the submenu. This will display the Shell Uniform Loads dialog box.
57. In this dialog box:
    - Select LL from the Load Case Name drop-down box.
    - In the Uniform Load area type **-.25** in the Load box and select Global Z from the drop-down Dir box.
    - In the Options area verify that the Add To Existing Loads option is selected.
    - Click the **OK** button to apply the load.
58. Click the **Show Undeformed Shape** button to reset the window display.
59. Click the "X" in the upper right-hand corner of the window labeled X-Z Plane @ Y=0 to close it.
60. Click the **Run Analysis** button to run the analysis.
61. When the analysis is complete check the messages in the Analysis window (there should be no warnings or errors) and then click the **OK** button to close the Analysis window.
62. From the **Options** menu select **Preferences...** to display the Preferences dialog box.
63. In this dialog box.
    - Click on the Steel Tab
    - Select AISC-ASD89 from the Steel Design Code drop-down box if it is not already selected.
    - Click the **OK** button.
64. From the **Design** menu click **Start Design/Check Of Structure** to run the design check of the steel frame elements.
65. When the design check completes, the stress ratios are displayed.

LINEAR AND NONLINEAR STATIC AND DYNAMIC ANALYSIS AND DESIGN OF BUILDING SYSTEMS

THIS IS ETABS

USER FRIENDLY GRAPHICAL INTERFACE

- Fully integrated interface within Windows 95/98/NT/2000
- Optimized for modeling of multistory buildings
- CAD drawing/editing for fast, intuitive framing layout
- 3D model generation using plans and elevations
- Fast generation of model using the concept of similar stories
- Automated templates for typical structures
- Easy editing with move, merge, mirror and replicate
- Convenient dividing and meshing of design objects
- Accurate dimensioning with guidelines and snapping
- Quick-draw options to create objects with one mouse click
- Multiple views in 3D perspective with zooming and panning
- Onscreen assignment of properties, loading and supports
- Powerful grouping, selection and display options
- Cut, copy and paste options
- Unlimited levels of undo and redo
- 3D perspective, plan, elevation, developed elevation, and custom views
- Graphical custom section designer
- Cut/Paste geometry to and from spreadsheets
- Import and export of .DXF file for model geometry
- Multiple simultaneous rectangular and cylindrical grid systems
- Detailed context-sensitive online help
- Analysis integrated with post-processing and design
- Right button click for element or design information

For nearly thirty years, the TABS and ETABS series of computer programs have defined the standard for building analysis and design software, and the tradition continues with this latest release of ETABS.

These programs were the first to take into account the unique properties inherent in a mathematical model of a building, allowing a computer representation to be constructed in the same fashion as a real building, floor by floor, story by story. ETABS uses terminology familiar to the building designer such as columns, beams, braces and walls rather than nodes and finite elements.

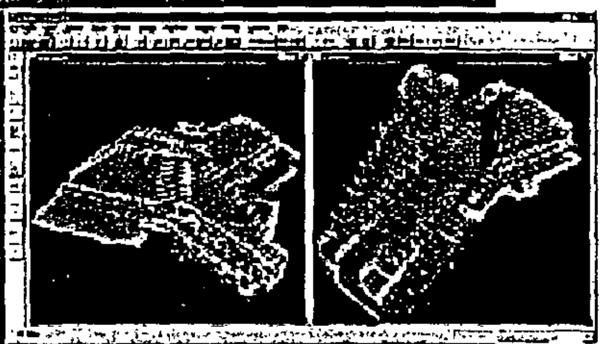
In any endeavor, a tool tailored to a specific task is the most efficient. For buildings, ETABS provides the automation and specialized options needed to make the process of model creation, analysis and design fast and convenient. Tools for laying out floor framing, columns, frames and walls, in either concrete or steel, as well as techniques for quickly generating gravity and lateral loads offer many advantages not available from most general purpose finite element programs. Seismic and wind loads are generated automatically according to the requirements of the selected building code. All of these modeling and analysis options are completely integrated with a wide range of steel and concrete design features.

While ETABS is familiar and straightforward to use for the building designer, it also offers many sophisticated analytical and design capabilities not found in other commercial programs. Full dynamic analysis, including nonlinear time-history capabilities for seismic base isolation and viscous dampers, along with static nonlinear pushover features offer state-of-the-art technology to the engineer doing performance design. Powerful features for the selection and optimization of vertical framing members as well as the identification of key elements for lateral drift control provide significant time savings in the design cycle. In addition, because ETABS includes complete and detailed steel and concrete design calculations for beams and columns, braces, walls and slabs, the time typically associated with the transfer of data between analysis and design programs has been eliminated. This design integration, in combination with the fact that ETABS generates CAD output files, means that production drawings can be generated faster and with greater accuracy.

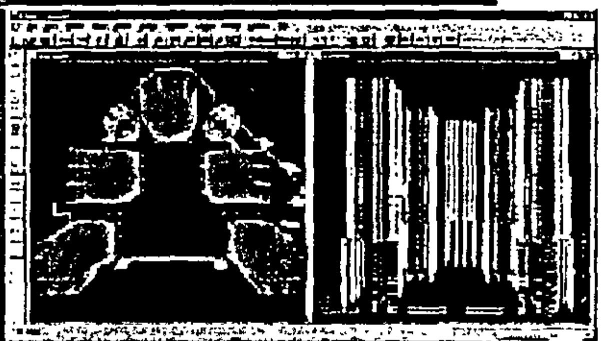
ETABS has long been a favorite for the analysis and design of buildings, and whether the project is a one story shopping center or the tallest building in the world, this latest release offers the comprehensive tools needed to produce timely, efficient and elegant engineering solutions

FULL 3D BUILDING MODEL
LINEAR STATIC AND DYNAMIC ANALYSIS
STEEL AND CONCRETE FRAME DESIGN
STEEL COMPOSITE BEAM DESIGN
CONCRETE SHEAR WALL DESIGN AND SLAB DESIGN

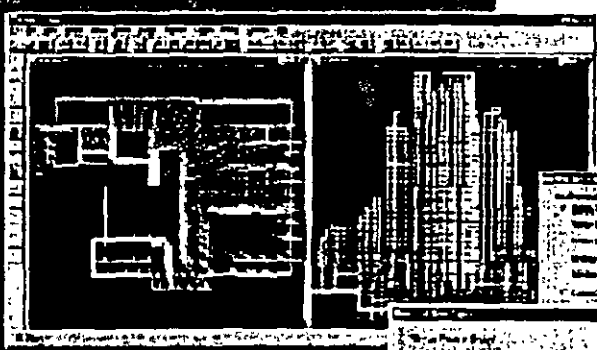
ETABS® PLUS



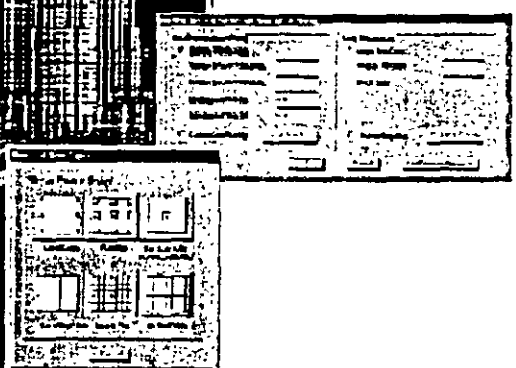
CONCRETE SHEARWALL BUILDINGS



MULTIPLE TOWERS



STEEL HIGHRISE BUILDINGS



ETABS PLUS FEATURES

Building Model

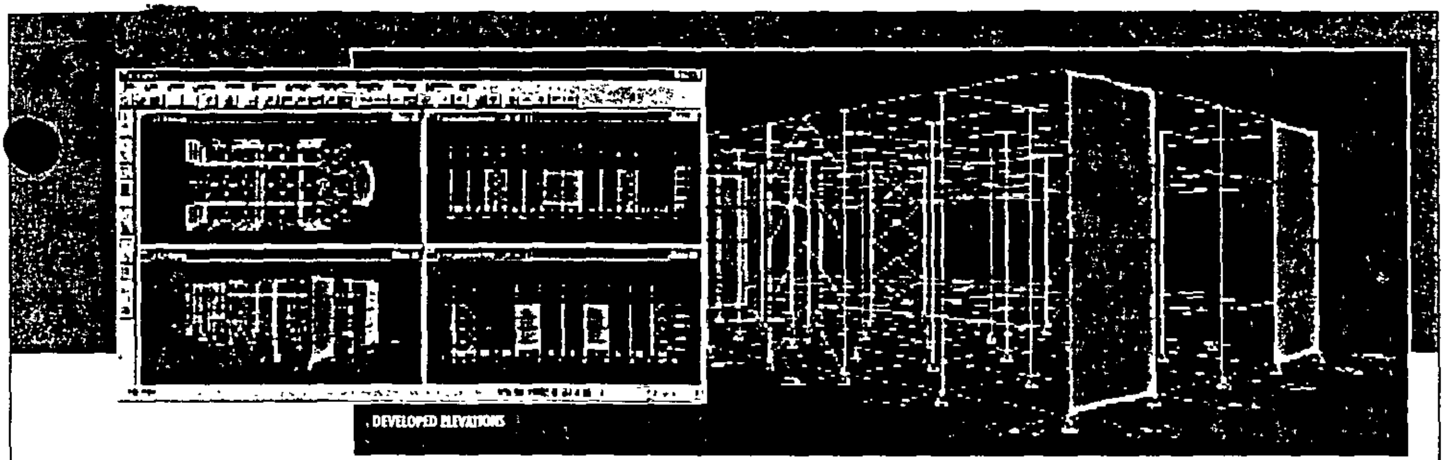
- Multiple simultaneous rectangular and cylindrical grid systems
- Story definitions using the concept of similar Stories
- Building modeled as Area, Line and Point objects
- Common labeling of Objects between similar Stories
- Area objects for: Walls, Slabs/Decks, Openings, Springs, Mass, Loads
- Line objects for: Columns, Beams, Braces, Links, Springs, Mass, Loads
- Point objects for Supports, Springs, Mass, Loads
- Rigid Diaphragm definitions
- Built-in database of steel sections
- Graphical Section Designer for defining custom sections

Building Loads

- No limit on number of independent load cases
- Gravity loads specified as point, line or area loads
- Automatic wind load generation: UBC, BOCA, ASCE, NBCC
- Automatic seismic load generation: UBC, BOCA, NBCC
- Built-in response spectrum and time history input
- Temperature and thermal-gradient loads
- Algebraic, absolute, SRSS, and enveloping load combinations
- Mass directly specified or calculated from gravity loads

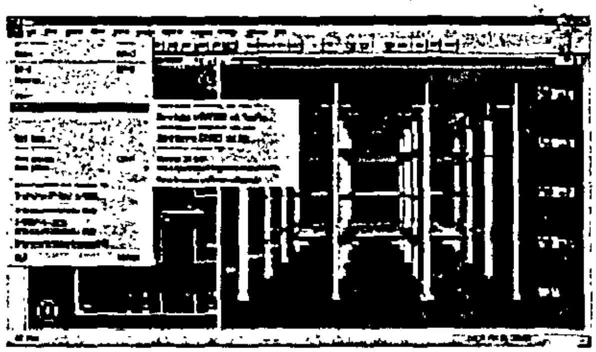
Analytical Options

- Static and dynamic analysis
- Automatic meshing of frame members into analysis elements
- Automatic transfer of loads on decks/slabs to beams and walls
- Automatic meshing of decks/slabs for flexible diaphragm analysis
- P-delta analysis with either static or dynamic analysis
- Automated center-of-rigidity calculations
- Integrated output forces for walls/slabs/decks for all loads
- Explicit Panel-zone deformations
- Automatic tributary-area calculations for Live-Load reduction factors
- Construction sequence loading analysis
- Eigen and load-dependent Ritz vector determination
- Multiple Response Spectrum cases
- Modal combination by SRSS, CQC or GMC (Gupta) method
- Combination of three directions by ABS or SRSS method
- Static and dynamic response combinations and envelopes
- Multiple Time History cases
- Sequential Time History cases
- Seismic acceleration or displacement excitation
- Wind-load forcing functions
- Transient or steady-state excitation
- Envelope or step-by-step design for Time-History loads



DEVELOPED ELEVATIONS

Analysis Output Options
- Deformed and Undeformed geometry in 3D perspective
- Loading diagrams
- Bending-Moment and Shear-Force diagrams for Frames
- Stress contours for Shells
- Integrated-force diagrams for Wall Piers and Spandrels
- Interactive Section-force results using Groups
- Animation of deformed shapes
- Time-History deformed shapes as real time AVI files
- Displays of nodal and element time-history records
- Time History displays of function vs time or function vs. function
- Response spectrum curves for any joint from Time History response
- Instantaneous on-screen results output with right-button click on element
- Selective or complete tabulated output for all output quantities
- Graphics output to screen, printer, DXF file, or Windows Metafile
- Tabulated output to screen, printer, or Access Database

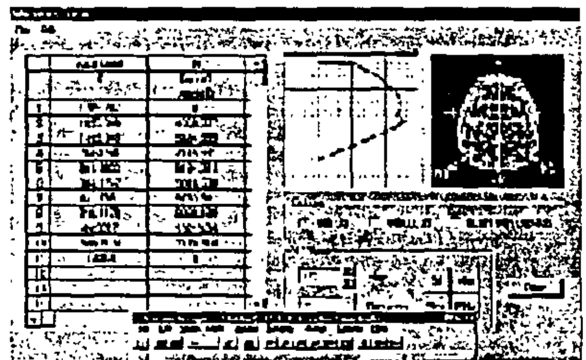


SAFE™ FLOOR IMPORTS

THE ELEMENT LIBRARY

Underlying the ETABS object-based building models is a comprehensive analysis engine comprised of the following element types.

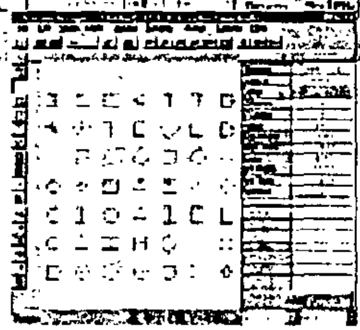
The 3D Beam/Column/Brace (Frame) Element
- Axial, bending, torsional and shear deformations
- Multiple non-prismatic segments over element length
- Ends offset from reference nodes in any direction
- Automated evaluation of offsets for joint size
- Moment and shear releases and partial-fixity
- Point, uniform and trapezoidal loading in any direction
- Temperature and thermal-gradient loading



CONCRETE DESIGN INTERACTION CURVES

The 3D Wall/Slab/Deck (Shell) Element
- Shell, plate or membrane action
- Thick-shell option
- General quadrilateral or triangular element
- Orthotropic materials
- Six degrees of freedom per joint
- Uniform load in any direction
- Temperature and thermal-gradient loading

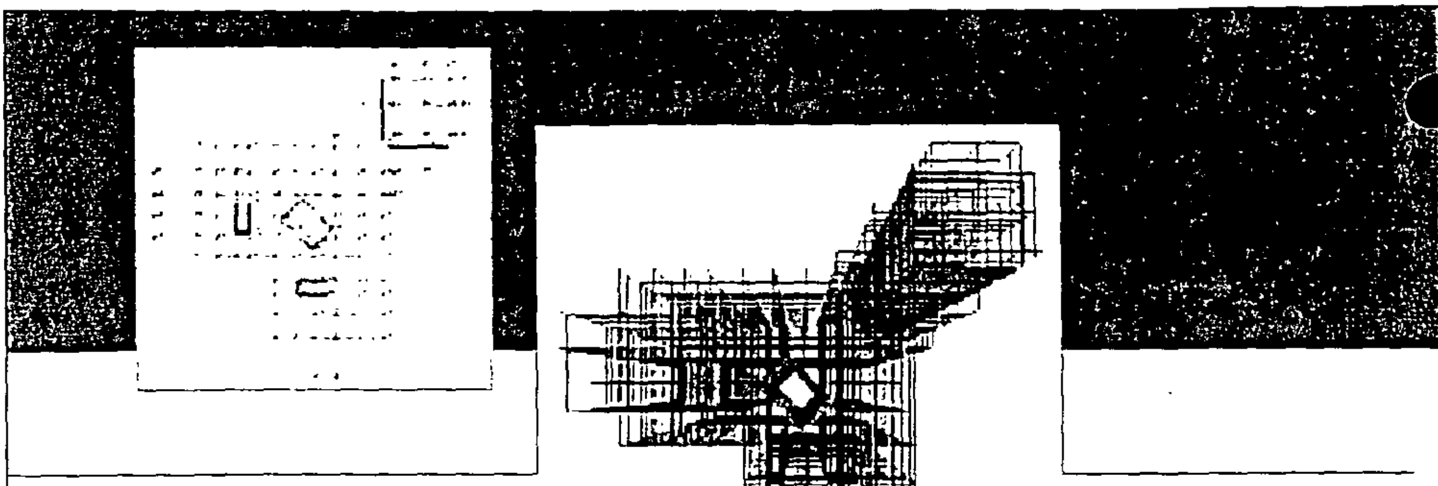
The Joint Element
- Support
- Coupled or uncoupled grounded springs
- Force loads
- Ground-displacement loads



SECTION DESIGNER

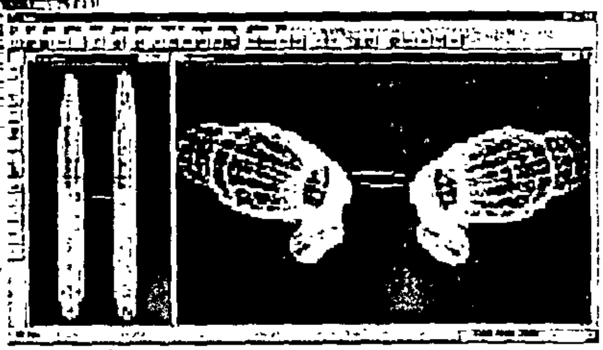
The Link Element
- Two node linear spring with 6 degrees of freedom
- Can be used to model Panel-zone deformations

A COMMITMENT TO SOFTWARE INNOVATION AND SUPPORT

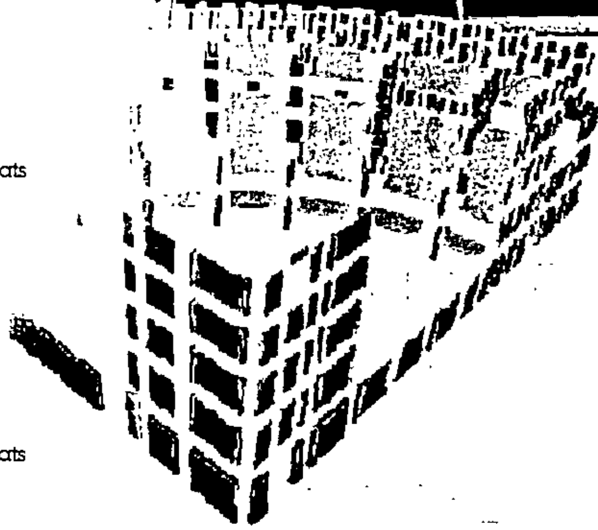


DESIGN OPTIONS

The following design options are fully integrated with analysis in the ETABS® graphical user interface.



TALLEST BUILDING (1999)



Steel Frame Design

- Fully integrated steel frame design
- AISC-ASD, AISC-LRFD, UBC, Canadian and Euro Codes
- Design for static and dynamic loads
- Grouping for design envelopes
- Optimization for strength and lateral drift
- Seismic design of special moment-resisting frames
- Seismic design of concentric and eccentric braced frames
- Check of panel zones for doubler and continuity plates
- Graphical display of stress ratios
- Interactive design and review
- Summary and detailed reports including database formats

Concrete Frame Design

- Fully integrated concrete frame design
- ACI, UBC, Canadian and Euro Codes
- Design for static and dynamic loads
- Seismic design of intermediate/special moment-resisting frames
- Seismic design of beam/column joints
- Seismic check for strong-column/weak-beam design
- Graphical Section Designer for concrete rebar location
- Biaxial-moment/axial-load interaction diagrams
- Graphical display of reinforcement and stress ratios
- Interactive design and review
- Summary and detailed reports including database formats

Composite Beam Design

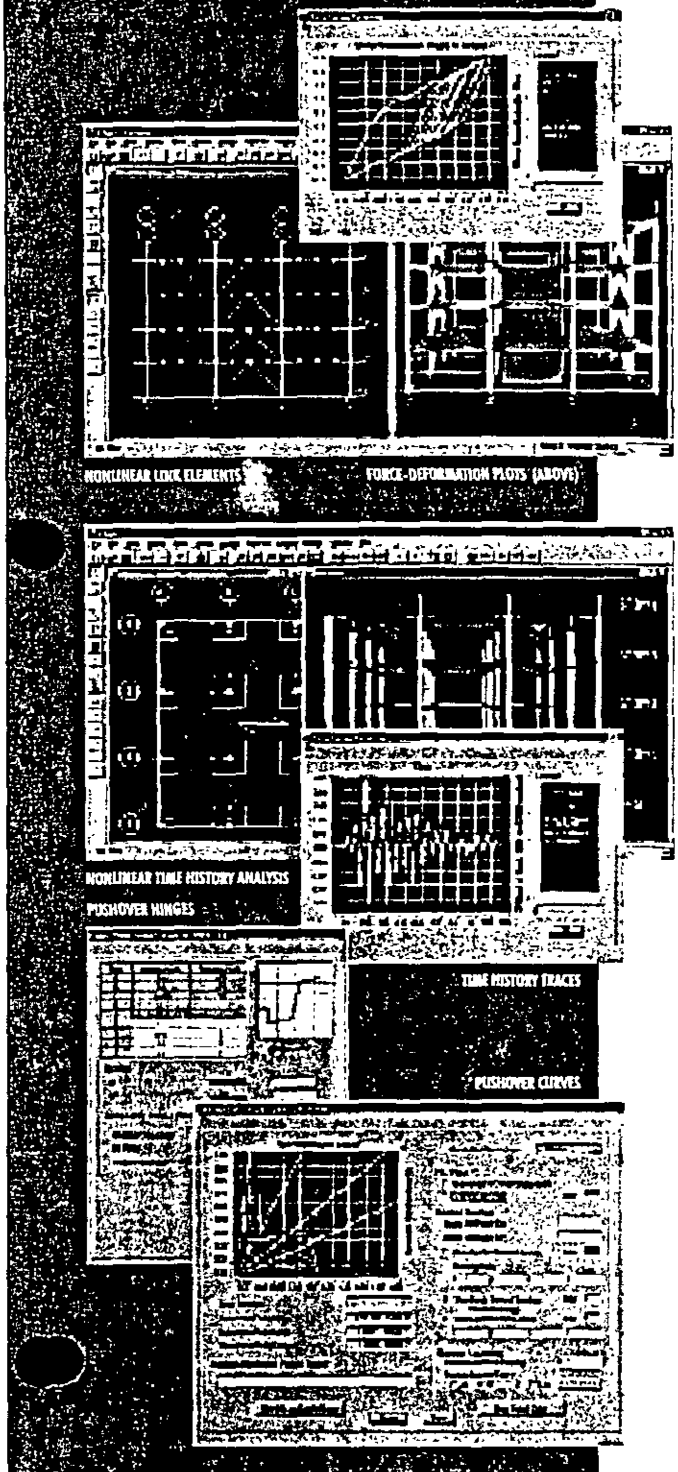
- Fully integrated composite beam design
- AISC-ASD and AISC-LRFD Specifications
- Automatic calculation of effective slab widths
- Numerous user-specified constraints
- Shored and unshored design
- Optimal design for strength and deflections
- Camber calculations
- Floor vibration analysis
- Graphical display of all design quantities
- Interactive design and review
- Summary and detailed reports including database formats

Concrete Shear Wall Design

- Fully integrated wall pier and spandrel design
- ACI, UBC and Canadian Codes
- Design for static and dynamic loads
- Automatic integration of forces for piers and spandrels
- 2D wall pier design and boundary-member checks
- 2D wall spandrel design
- 3D wall pier check for provided reinforcement
- Graphical Section Designer for concrete rebar location
- Graphical display of reinforcement and stress ratios
- Interactive design and review
- Summary and detailed reports including database formats

STATIC PUSHOVER ANALYSIS
NONLINEAR TIME HISTORY ANALYSIS
BASE ISOLATORS
VISCOUS DAMPERS
STRUCTURAL POUNDING

ETABS® NONLINEAR



ETABS NONLINEAR FEATURES

ETABS Nonlinear extends the capabilities of the PLUS version to include the following static and dynamic nonlinear analysis options

Static Nonlinear Analysis Options
Large displacement option
Sequential loading option

Plastic Hinge Element
- Used as Spring, Link, Panel zone or inside Frame Elements
- Axial, flexural, shear and torsional behavior
- Axial-load/biaxial-moment interaction
- Multilinear behavior including softening
- Tabulated and Graphical display of hinge status

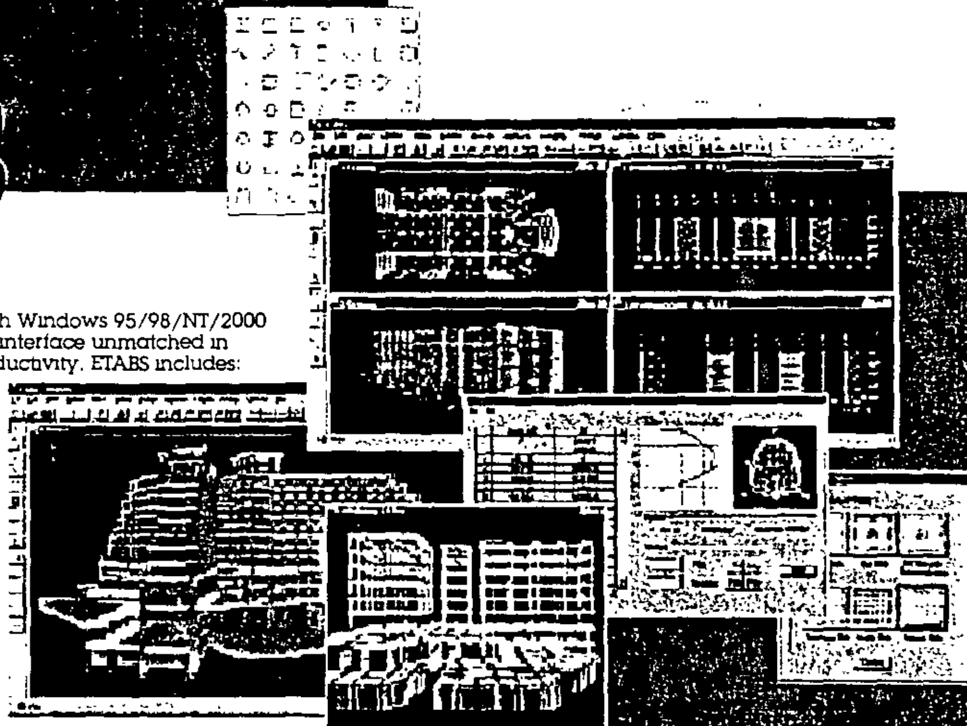
Specialization for Static Pushover Analysis
- FEMA 273, ATC-40
- Automated force-deformation relations for steel and concrete hinges
- Modal, uniform, or user-defined lateral load patterns
- Start from applied gravity load
- Capacity Spectrum conversions
- Effective damping calculation
- Demand Spectrum comparisons
- Performance point calculation
- Summary reports including plastic-hinge deformations

Dynamic Nonlinear Analysis Options
The nonlinear dynamic analysis option extends the capabilities of the Linear Time History option of the ETABS Plus by allowing for nonlinearity in predefined nonlinear elements

Nonlinear Link Element
- Used with the Dynamic Nonlinear Analysis option
- Used as Link, Spring or as Panel zone
- Viscous damper with nonlinear exponent on velocity term
- Gap (compression only) and Hook (tension only)
- Uniaxial plasticity (all 6 degrees of freedom)
- Base isolator with biaxial-plasticity behavior
- Base isolator with friction and/or pendulum behavior
- Force or displacement vs. time plots
- Force vs. deformation plots

The Wilson FNA Method
The ETABS nonlinear time history analysis uses the new numerical integration technique known as the Wilson FNA (Fast Nonlinear Analysis) Method. The procedure uses an iterative vector superposition algorithm that is extremely efficient for analyzing structures with predefined, localized nonlinearity. The method has demonstrated significant reductions in processing times when compared with other nonlinear analysis methods.

ETABS



ETABS PLUS is fully integrated with Windows 95/98/NT/2000 and features a powerful graphical interface unmatched in ease of use, sophistication and productivity. ETABS includes:

Full 3D Building Model
Building Terminology
Automated Gravity Load Tracing
Automated Wind Loads
Automated Seismic Loads

3D Finite Element Analysis
Frame, Shell, Joint, Link Elements
P-Delta Option
Linear Static Analysis
Modal Analysis
Response Spectrum Analysis
Linear Time History Analysis

Steel Frame Design
Concrete Frame Design
Composite Beam Design
Concrete Shear Wall Design

ETABS NONLINEAR extends the capabilities of the PLUS version to include nonlinear analysis options:

Static Nonlinear Analysis Options
Large Displacement Option
Sequential Loading Option
Plastic Hinge Element
Static Pusnover Analysis - FEMA 273 and ATC-40

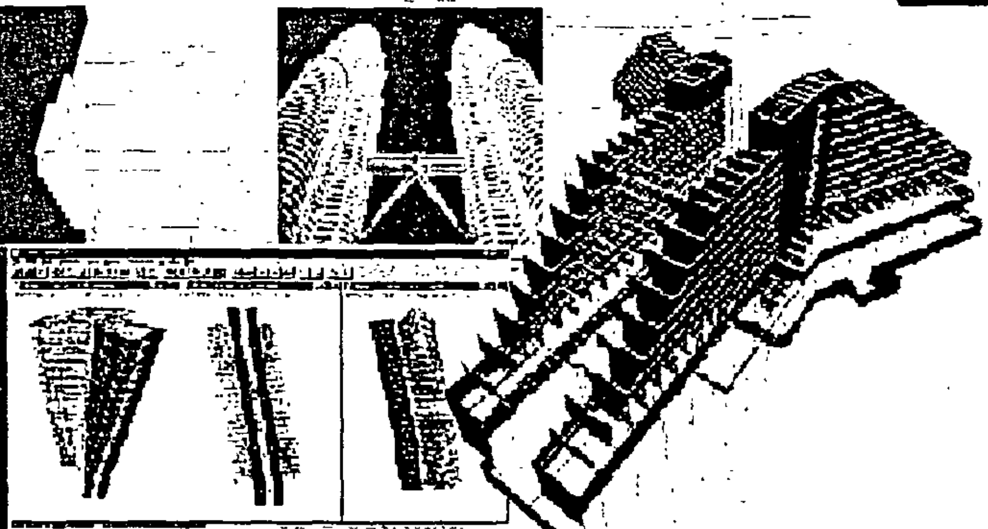
Dynamic Nonlinear Analysis Options
Gap/Hook Element
Damper Element
Plasticity Element
Base Isolator with Plasticity Behavior
Base Isolator with Friction/Pendulum Behavior

The ETABS Package comes with a comprehensive set of printed and online documentation including User Manuals, Design Manuals, Tutorials, and the latest edition of the book *Three Dimensional Static and Dynamic Analysis of Structures – A Physical Approach with Emphasis on Earthquake Engineering*, a CSI publication authored by Professor Edward L. Wilson, Professor Emeritus, University of California, Berkeley

COMPUTERS AND STRUCTURES, INC.
1995 UNIVERSITY AVENUE
BERKELEY, CALIFORNIA 94704
510 845 2177 PHONE   FAX 510 845 4096
info@csiberkeley.com e-mail
www.csiberkeley.com web

INTEGRATED DESIGN AND ANALYSIS SOFTWARE FOR BUILDING SYSTEMS

FOR WINDOWS 95/98/NT/2000

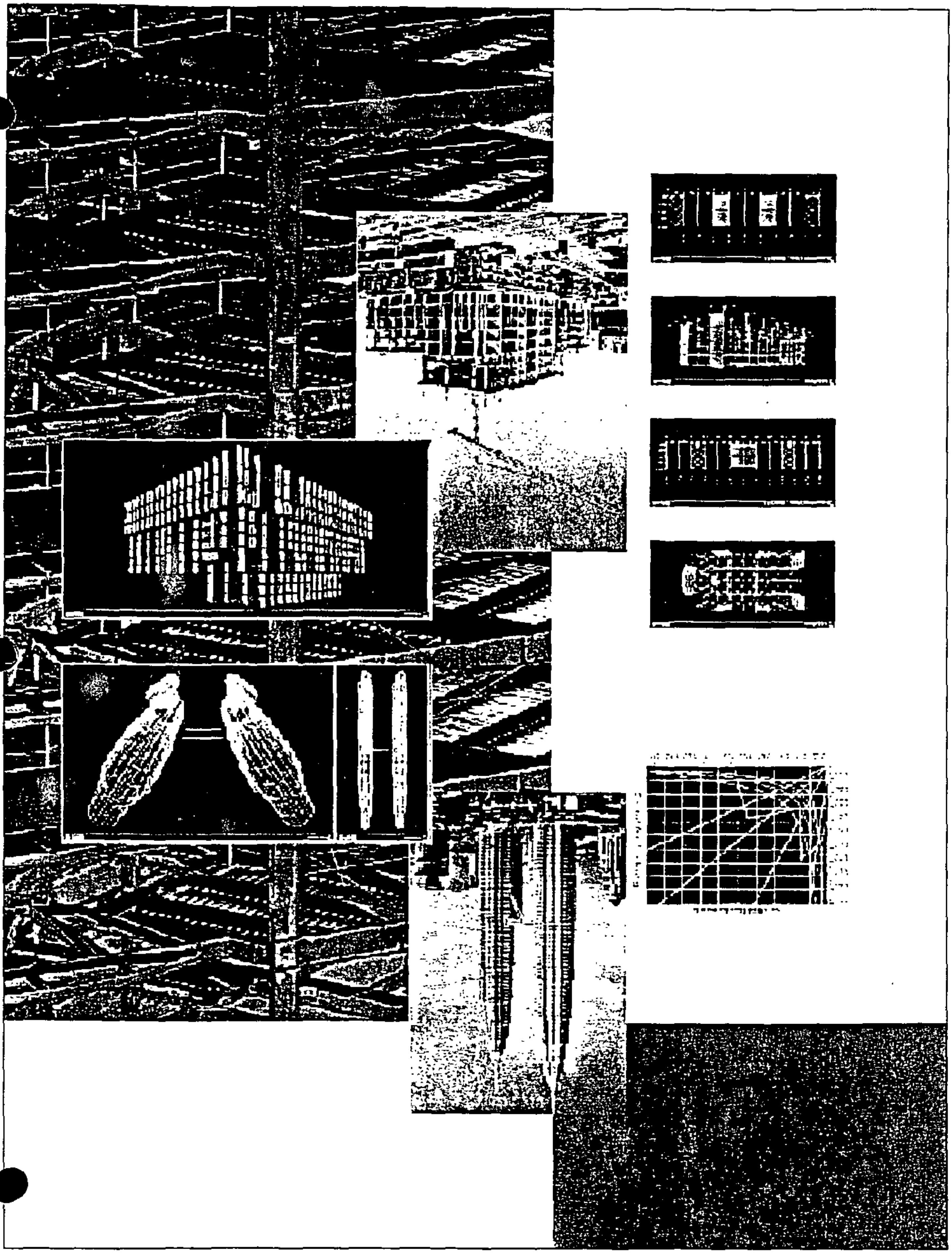


ETABS®

LINEAR AND NONLINEAR STATIC AND DYNAMIC ANALYSIS AND DESIGN OF BUILDING SYSTEMS

COMPUTERS & STRUCTURES INC ®

STRUCTURAL AND EARTHQUAKE ENGINEERING SOFTWARE



FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

INTEGRATED FINITE ELEMENTS ANALYSIS AND DESING OF STRUCTURES

TUTORIAL MANUAL

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F., APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax. 5510.0573

SAP2000®

Integrated
Finite Elements Analysis
and
Design of Structures

TUTORIAL MANUAL

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

Version 6 1
September 1997

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *support@csiberkeley.com*
web: *www.csiberkeley.com*

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMP-TIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table Of Contents

Tutorial 1	2-D Frame with Static loading	1
Tutorial 2	2-D Frame with Response Spectrum Loading	11
Tutorial 3	2-D Frame with Time History Loading	17
Tutorial 4	2-D Frame Steel Design	27
Appendix A	Toolbar Icon Descriptions	A1

TUTORIAL 1

2-D Frame with Static Loading

Description

This tutorial describes the modeling and analysis of a seven-story two-dimensional frame structure, subjected to static earthquake loads. The framing and the static loads are shown in Figure 1-1.

Significant Features of Model and SAP2000

- Using Templates
- Editing the model graphically
- Two-dimensional frame analysis
- Diaphragm constraints
- Lateral joint loads
- Vertical span loads
- Graphical output of results

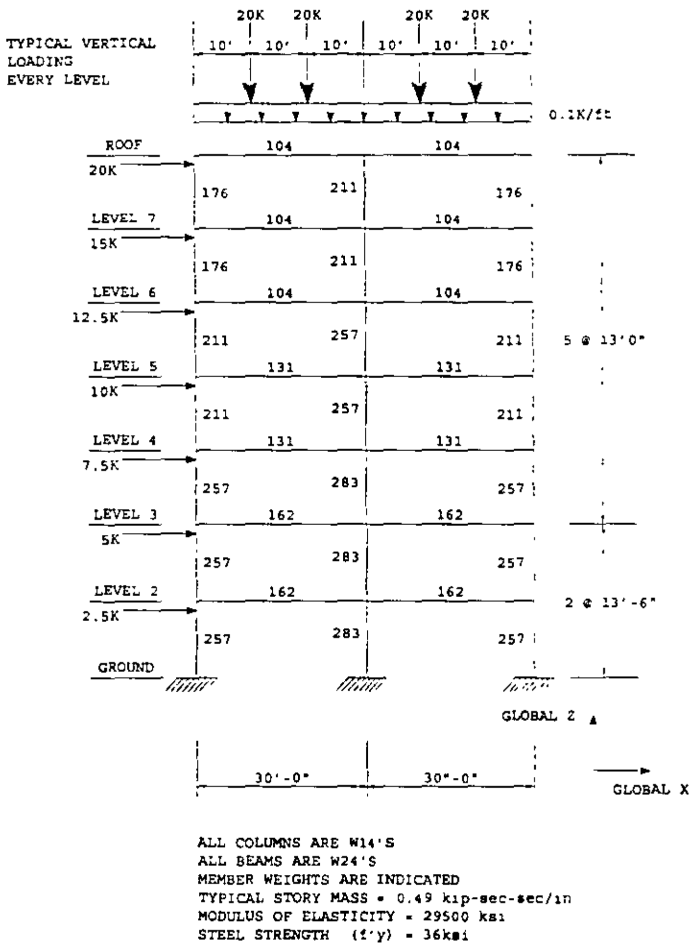
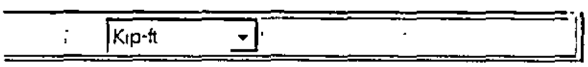


Figure 1-1 Two-dimensional Frame Tutorial

Building the Model in SAP2000

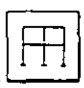
Using Templates

1. Select the units you want to work with from the status bar at the bottom of the SAP2000 window. In this case let's start with Kip-ft.



   Note: ***You can change the units you are working with at any time and SAP2000 will handle the conversion.***

2. From the **File** menu select **New Model from Template**.

3. Select 2D Frame. 

4. Fill in story and bay information. For now use 13 ft for story height and we will edit the first two floors and their grids next.

Editing Model

We are going to edit the framing and the grid at the same time.

1. From the **Draw** menu select **Edit Grid**.

   Hint: ***Always try to locate your joints at grid intersections. By providing grids you can make inputting, modifying and viewing of your model much easier and faster.***

2. Select the **Z** direction button to change the grids defining the story heights.

3. Select the **Glue Joints to Grid Lines** option. This will let you edit the joint locations and framing by simply editing the grid.

4. Edit the grid locations and press the OK button to close the form.

   Hint: ***When you have finished editing the grid you can right click with the mouse on the columns to find out if the columns are the proper length. This is a very practical way to inquire about any joint or frame member.***

Editing Supports

The next step is to change the supports on the structure from the default of pinned to fixed.

1. Select the **Pointer Tool** icon from the floating tool bar.

2. Draw a box completely around the three joints at the base of the structure

   Hint: ***You can always look at the status bar to see what type of elements and how many of them have been selected.***

3. Select **Assign Joint Restraints** icon from the floating tool bar to set the fixed supports. You can also define other Joint properties including the restraint through the **Assign** menu.

Assigning Member Sections

1. Load all the frame sections that you will need for your model first. From the **Define** menu select **Frame Sections** option. Then import the wide flanges shown in Figure1-1.

   Note: ***You can select more than one section at a time in the Section Selection list by using the ctrl key while you make the selections.***

2. You will find under the **Select** menu item many ways to select joints and elements. For this problem you may find the **Pointer/Window** and the **Intersecting Line** select mode to be the most helpful.

3. Once you have selected the frame members you want, you can assign steel sections to them through the **Assign Frame Sections** button on the floating tool bar.

Assigning Loads

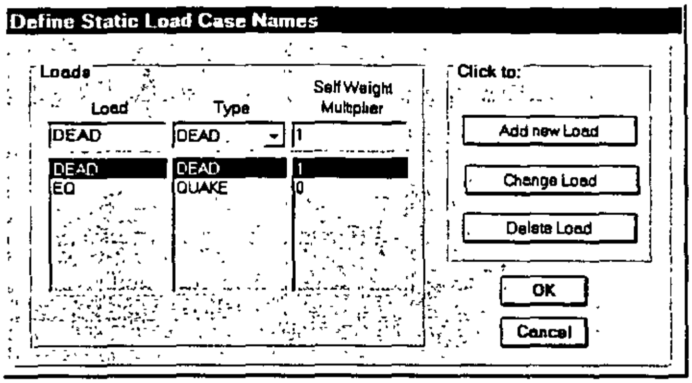


Figure 1-2 Static Load Case Names Input Form

1. The first step in entering the loads is to define static load cases. From the **Define** menu select **Static Load Cases.**
   - DEAD can be used for the beam vertical loads and keep the **Self Weight Multiplier** as 1.
2. Define a static lateral load case called EQ for the earthquake load. Assign the lateral load as a **QUAKE** type load. This will allow the load combinations for the design features of the program to be calculated automatically. Also set the **Self Weight Multiplier** to zero.
3. The vertical loads shown in Figure 1-1 can be assigned to the beams by selecting all the beams and using the **Assign Frame Span Loads** button on the floating tool bar.
4. The static lateral loads need to be entered by selecting each node individually and using the **Assign Joint Loads** button.

   Reminder: ***Make sure that you are adding the loads to the proper load case.***

Setting Up Floor Diaphragms

Setting up floor diaphragms and specifying the floor mass in only the X direction are some common techniques used to reduce the size of the problem solved by SAP2000. In addition, setting diaphragms makes the structure behave more like a building with a rigid diaphragm.

1. Repeat these Steps for each floor:
   - Select all the joints on a floor.

- From the **Assign** menu select **Joint ... Constraints.**
- Select **Add Diaphragm** from the drop down list box.
    - In the **Diaphragm Constraint** form give the diaphragm a name like DIA1 for the first floor.
    - Select Z-axis constraint. This defines a diaphragm perpendicular to the Z-axis.
    - Press the OK button.
- Press the OK buttons to finish the operation.
- Repeat the steps for the other floors with a different diaphragm name for each one.

2. The Story masses are the same on all floors so select one node from each floor.
3. Change the input units to Kip-in since the Typical Story Mass provided in Figure 1-1 is given in those units.
4. From the **Assign** menu select **Joint ... Masses**
    - Enter the story mass in the local direction 1 (which in this case is the global X) direction.
    - All the other values are zero.
5. Change the units back to Kip-ft.

Material Properties

The final thing to check before running the analysis is the material properties.

1. From the **Define** menu select **Materials**.
2. In the **Materials** form select STEEL and press the MODIFY/SHOW MATERIAL button.
    - In the **Material Property Data** form check that the material properties are correct. Remember that the values are reported in the current units.

Running Analysis

Once data has been entered, it is time to run the model and take a look at our results.

1. Save your model.
2. Set the parameters for the analysis run by selecting from the **Analyze** menu **Set Options**.
    - In the **Analysis Options** form select a **Plane Frame** analysis to reduce the size of the solution and thus reduce the time needed for the analysis.
    - Press the OK button to accept your changes.

3. Select **Run** from the **Analyze** menu to analyze the structure.

   Note: ***Once the analysis has finished, you may want to look at the entire analysis results screen before pressing the OK button. This is your first check to see if there are any problems with the model.***

Using the Results

Checking Results

Once you have run the model successfully you need to check and see that the output is correct and close to what you expected.

Model checks:

1. Check that the base shear adds up to the total lateral load in the EQ load case.
   - Select the bottom set of frame members and the nodes at the base of the structure.
   - From the **Assign** menu select **Group Names.**
   - Give the group a name like BASE SHEAR.
   - Select the ADD NEW GROUP NAME button and press the OK button.
   - From the **Display** menu select **Show Group Joint Force Sums** and select the group name that you just assigned.
2. Look at the deflection and animate it for both lateral and vertical loads to make sure that the model is behaving as expected.
   - From the **Display** menu select **Show Deformed Shape** and select the loading case in which you are interested. Also select the **Wire Shadow** option, so you can see the undeformed structural shape. See Figure 1-3 and Figure 1-4 for the deformed shapes of the structure. Right click on any joint to see its displacement and rotation.
   - Animate the displayed load by pressing the START ANIMATION button at the bottom of the status bar (A window with the deformed shape will need to be active for the button to show). You can also save the animation as an *.AVI file for later viewing from the **File** menu. (See online help under "Export an AVI file".)

     Try This: ***Press the + and – buttons next to the Animate button and see what happens to the form with the deformed shape.***
   - Press the STOP ANIMATION button when you are finished looking at it.

If these checks show that the input information appears to be correct then we can move to the more advanced checks.

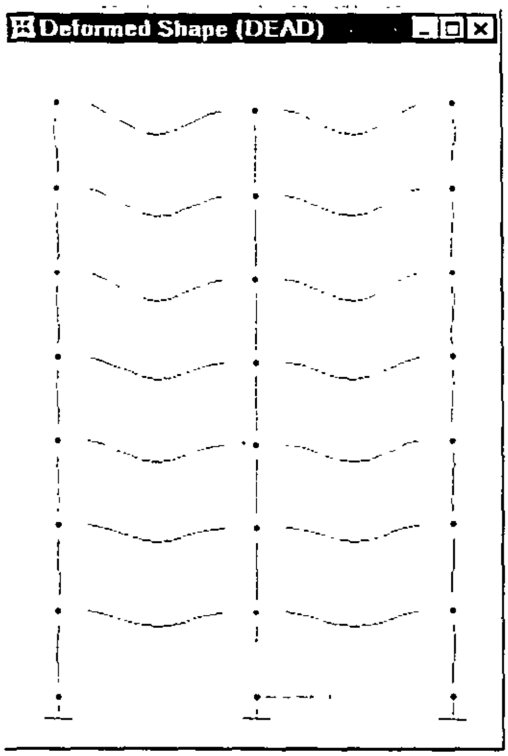


Figure 1-3 Static Deformed Shape

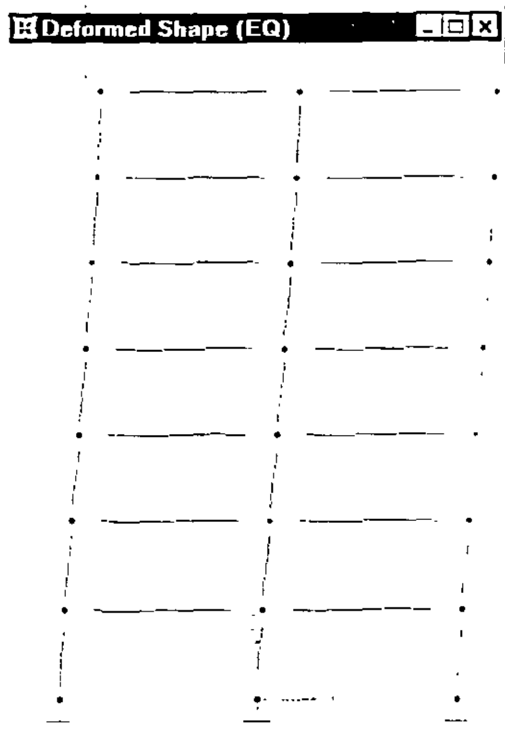


Figure 1-4 Lateral Deformed Shape

Structural Performance

You will often want to make certain that the structure is within the performance limits, such as stresse ratio, set by the code being used. SAP2000 will do the stress checks automatically when a member is designed. (The design features of SAP2000 will be discussed in more detail in subsequent tutorials.)

1. The structural elements can be designed by simply selecting from the **Design** menu the **Start Design/Check of Structure** option.

   - The frame elements will now display colors showing their stress level with the stress ratio level value indicated below the beam. A value of 1 is 100% stressed.

   - Other design and input information can also be displayed by selecting from the **Design** menu the **Display Design Info** option.

2. There is also the ability to see the design information on individual members and to assign alternate sections to them by right clicking on the element.

   - From the screen that appears you can select the DETAILS button to view detailed information on the section for each of the load combinations used in the design.

   - You can also redesign the element after changing its design parameters, effective length, K factor and section properties by pressing the REDESIGN button.

3. If you have selected a new section and want to use that for your final design you can select from the **Design** menu the **Update Analysis Sections** option so the structure can be reanalyzed using the new sections that have been selected.

   Note: You may need to use the Refresh Window button on the tool bar to update the design information in the active window after changing any of the design parameters.

Looking At and Printing Analysis Results

You will often want to look at and have a copy of the results that you get from a SAP2000 analysis run. There are different ways to do this.

1. You can choose to output the results during the analysis run by selecting **Generate Output** on the **Analysis Options** form. The **Select Output Options** button that appears will allow you to select how much of the results to output. The results are written to a file with the name of the data file and an extension of *.OUT .

2. The input and most of the output information can be viewed from the **Display** menu.

3. From the **File** menu you can choose to print Graphics, Input Tables, Output Tables and Design Tables of the model's data and results.

   Hint: If members or joints are selected when printing the tables, you will have the output to only print the selected members. To do this, check the Selection Only option.

4. The SAP2000 analysis run also provides two output files. The *filename*.EKO file includes all the input information used for the analysis. The *filename*.OUT file contains analysis result data as well as any output requested from the **Analyze ...Set Options** menu.

   Remember: ***You may want to make a habit of printing the output to a file first. This will give you a fast way to look at your output using a text editor without wasting large piles of paper.***

Final Comments

As you can see, SAP2000 is a powerful structural analysis tool and can be used for a variety of problems. However it is very important to understand the engineering principles on which it is based.

Most projects start as feasibility exercises and mature into a full analysis/design project. This makes it very important to decide early on what the proper tool is so there will be no need to change programs during the middle of a project. SAP2000 tries to address as many of the needs that a designer may have during the life of the project.

Features that help in the design process include:

- The ability to design small and large projects without having to learn a new program.
- The ability to design steel and concrete members within the same program.
- Fast analysis algorithms allowing time for developing the model and optimizing the design of structural elements.
- The ability to easily modify and improve the design.

There are probably as many ways to model a structure as there are engineers. However, you may find some of these ideas helpful:

- Start with a basic model of your structure and try and understand it before adding more detail. It is always easier to fix structural system related problems while the model is still simple.
- Ensure the structure can be constructed and will behave the way you have modeled it. If it can not be built in such a way, you may need to understand what effect that will have on the structure.
- Thoroughly document your design including assumptions, areas that need to be reviewed and information that is still required. You can use the **User Comments and Session Log** text editor under the **File** menu. This basic text editor is built into the program allowing your notes to be with your model.
- Experiment with alternative structural systems. SAP2000 was designed to be fast; use the extra time to improve your design.
- If there is time to do it properly later there should be time to do it properly from the start.

TUTORIAL 2

2-D Frame with Response Spectrum Loading

Description

This tutorial is a continuation of Tutorial 1. In this tutorial we will demonstrate how to add a Response Spectrum analysis to the 2-d frame. The basis for the Response Spectrum will be the UBC94S2 spectrum, which is included in SAP2000.

Significant Features of Model and SAP2000

- Using Help to get instructions on SAP2000 features
- Adding a Response Spectrum load case
- Scaling the Response Spectrum for use in design

Defining Response Spectrum

A Response Spectrum is the maximum response of a system excited at its base by a time acceleration function. It is expressed in terms of the natural frequency and the amount of damping of the system. The UBC94S2 Response Spectrum function that we are going to use is provided with SAP2000 and so it will not need to be defined separately. If there were a need to define your own Response Spectrum Function you could use the online help to get step-by-step instructions.

Online Help

Remember: ***You can use the following methods to get information on any SAP2000's functions.***

1. From the **Help** menu select **Search for Help on**.
2. With the **Index** tab selected:
   - In Area 1 type 'Define'. You will then see in the Area 2 a list of all the help topics that start with define. One of those topics is Define Response Spectrum Functions, which is the topic we need help on. Double click on the line with the words 'Define Response Spectrum Functions' to show the help information on that topic.
3. Alternately, select the **Find** tab to search for key words in any of the online help topics.
   - If this is the first time that you use Find for the SAP2000 online help, a **Find Setup Wizard** form will appear.
     - Press the NEXT button to accept the criteria for building the find database.
     - Press the FINISH button to have the database built.
   - In Area 1 type 'Response Spectrum'
   - In Area 3 you will again find 'Define Response Spectrum Functions' which you can select to get to the help information.

   Note: ***You can find more information on using online help, in your windows documentation. You can also run the WINHELP32.HLP file in your C:\WINDOWS\HELP folder.***

Defining Response Spectrum Case

1. If the model is locked then use the **Lock/Unlock Model** button to unlock it so you can make changes to the model.
2. Set the units to Kip-ft.
3. From the **Define** menu select **Response Spectrum Case.**
4. Press the ADD NEW SPECTRA button on the **Response Spectra** form.
5. In the **Response Spectrum Case Data** form:

- Input a **Damping** of 0.05 (5 %)
- Select the **UBC94S2** for the **U1** direction and a **Scale Factor** of 32.2 ft/sec^2. The scale factor is used for the Response Spectrum because UBC94S2 is normalized by the acceleration due to gravity g.
- The remainder of the default values are acceptable.
- Press the OK buttons to accept the changes you have made to both forms.

Running Analysis

Once you have made the modifications, it is time to run the model and take a look at the Response Spectrum results.

1. Save your model.
2. Set the parameters for the analysis by selecting from the **Analyze** menu **Set Options**.
   - Select the **Dynamic Analysis** check box.
   - Press the SET DYNAMIC PARAMETERS button and change the **Number of Modes** used in the solution to 7. The remaining default values are acceptable.
   - Press the OK button on both forms to accept your changes.

   Note: ***You have to decide how many modes you will need to consider in your analysis to make the results meaningful. There are many criteria to take into account, but for a structure as simple as ours you can consider the same number of modes as there are floors.***
3. Select **Run Minimized** from the **Analyze** menu to analyze the structure.

   Note: ***The Run Minimized option is extremely helpful when running large models that may take a lot of time to analyze. This option allows SAP2000 to run in the background so you can continue working on other programs. The other advantage to this option is that you get a cancel button that allows you to cancel a run if you need to.***

Checking Results

1. Check the modal shapes and periods to see that they are as expected.
   - From the **Display** menu select **Show Mode Shape** and select the mode you are interested in. You may also want to select the **Wire Shadow** option so you can see what the undeformed shape looks like. See Figures 2-1 through 2-4 and note that the mode number and period is shown as the title of the window.

Note: ***You can look at subsequent mode shapes by pressing the + and – buttons that are next to the START ANIMATION button.***

2. It is helpful to see the base shear produced due to the Response Spectrum analysis.
   - Using the BASE SHEAR group that was set up in Tutorial 1, look at the base shear for the structure due to the Response Spectrum. You will see that it is substantially larger than the static load case.
3. You can also check the displacement of a joint due to the Response Spectrum.
   - From the **Display** menu select **Show Deformed Shape**.
     - In the **Deformed Shape** form select the load case for the spectral analysis.
     - Press the OK button.
   - Right click on a joint at the top level of the structure to see the displacement of the joint in the global X direction.
4. Check the mass participation to see if enough modes were included in the solution. This will need to be done outside of SAP2000 by looking at the *filename*.OUT file using a text editor like WordPad.
   - Minimize the SAP2000 program.
   - Start WordPad or another text editor.
   - In WordPad open the file *filename*.OUT. Where *filename* is the name of the file you used when saving this tutorial.
     - Find the section titled MODAL PARTICIPATING MASS RATIOS as shown in Figure 2-5.
     - You will find under the CUMULATIVE SUM column that Mode 1 through Mode 7 includes 100% of the mass participation. Which means that the 7 modes included in the analysis were enough.

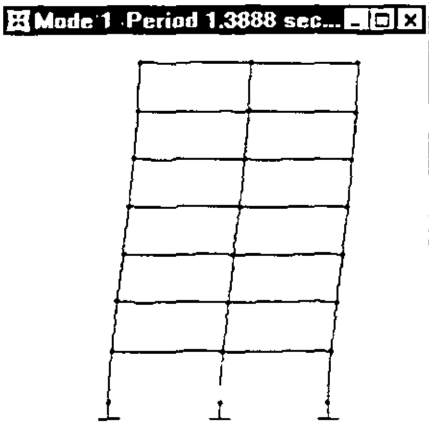


Figure 2-1 Mode 1 Shape and Period

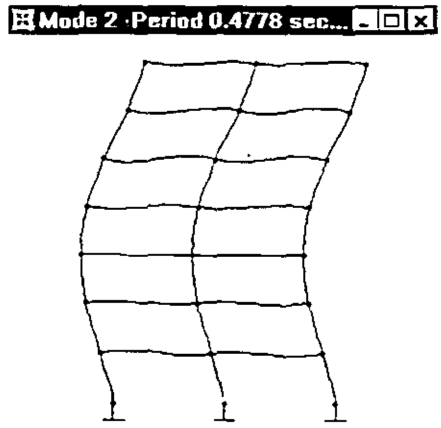


Figure 2-2 Mode 2 Shape and Period

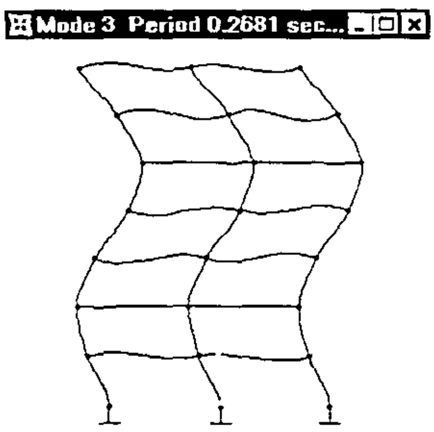


Figure 2-3 Mode 3 Shape and Period

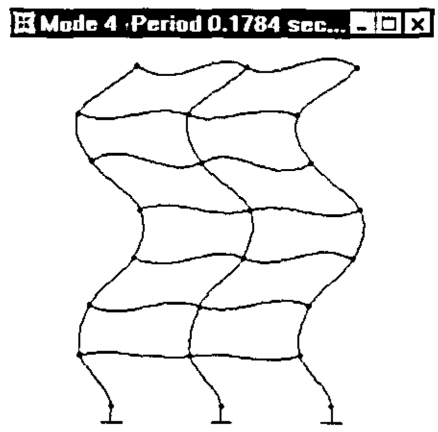


Figure 2-4 Mode 4 Shape and Period

```
M O D A L   P A R T I C I P A T I N G   M A S S   R A T I O S

 MODE     PERIOD     INDIVIDUAL MODE (PERCENT)     CUMULATIVE SUM (PERCENT)
                        UX        UY        UZ        UX        UY        UZ

   1    1.388750    79.6359   0.0000   0.0000    79.6359   0.0000   0.0000
   2    0.477833    11.5761   0.0000   0.0000    91.2120   0.0000   0.0000
   3    0.268126     4.3023   0.0000   0.0000    95.5144   0.0000   0.0000
   4    0.178439     2.1229   0.0000   0.0000    97.6373   0.0000   0.0000
   5    0.133678     1.4077   0.0000   0.0000    99.0450   0.0000   0.0000
   6    0.107637     0.6592   0.0000   0.0000    99.7043   0.0000   0.0000
   7    0.090778     0.2957   0.0000   0.0000   100.0000   0.0000   0.0000
```

Figure 2-5 Output File Block for Mass Participation

Scaling Response Spectra

Some design codes allow you to scale the spectral analysis base shear to the static base shear. So in this case to get the new scale factor for the Response Spectra you would:

1. Divide the Static Base Shear by the Spectral Base Shear and multiply that number by 32.2 ft/sec^2 to get a new scale factor for the Response Spectrum.
2. Substitute the new scale factor for the Response Spectra Case.
3. Rerun the analysis to get the new member forces due to the scaled Response Spectrum.

Final Comments

A Response Spectra analysis introduces another level of complexity, which requires the engineer to further check the analysis results and the assumptions used in the modeling. Things to keep in mind during a Response Spectrum analysis:

- Completely understand the static behavior of the model before running a dynamic analysis.
- You will need to completely understand the rationale and applicability of scaling dynamic analysis results to equivalent static base shear before scaling results for any given model.
- The speed advantages of running a Response Spectrum analysis over a full Time History analysis can be substantial. In design, the Response Spectrum analysis can provide an even greater speed advantage, due to the fact that the design check does not need to be done at each time segment. However, one needs to be aware of the limitations of this method over a full Time History analysis.

TUTORIAL 3

2-D Frame with Time History Loading

Description

This tutorial builds on Tutorials 1 and 2 by adding an earthquake load specified as a base acceleration time history. The earthquake excitation is shown in Figure 3-1. It is the N-S component of the 1940 El Centro earthquake. The results of the Time History are used to generate a Response Spectrum which in turn is used to reanalyze the structure as a comparison.

Significant Features of Model and SAP2000

- Time History response to base excitation
- Plotting time history results
- Plotting a Response Spectrum from a Time History
- Importing a Response Spectrum to use for analysis

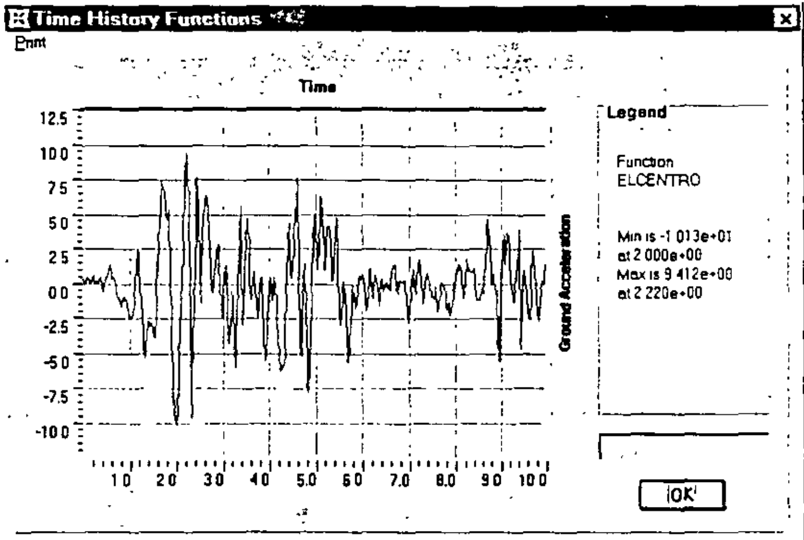


Figure 3-1 El Centro Ground Acceleration Input - ft/sec[2]

Time History Definition

Time History is a record of the ground acceleration at defined time segments for a specific earthquake in a certain direction. The record is usually normalized and therefore needs to be multiplied by the acceleration due to gravity or a factor thereof.

1. From the **Define** menu select **Time History Functions**.
2. Select the ADD FUNCTION FROM FILE button.
   - Press the **Open File** button and then find and select the ELCENTRO file from the EXAMPLES directory in SAP2000.
   - Change the name of the function to ELCENTRO to make it easier to recognize.
   - The format for this file is three pairs of data columns per row. The first column of the pair is the time and the second is the acceleration.
     - Enter 3 for Points Per Line.
     - Select the **Time and Function Values** (in data file) option.
     - Press the OK button.
   - Press the OK button to accept the additions that you have made.

3. From the **Define** menu select **Time History Cases** to define the specifics of the Time History for your model.
   - Select the ADD NEW HISTORY button.
   - Press the MODIFY/SHOW MODAL DAMPING button and enter 0.05 (5%) for all modes and hit the OK button.
   - Enter 500 in **Number of Output Time Steps.**
   - Enter 0.02 (sec) in **Output Time Step Sizes.** This will give us 10 seconds of the earthquake time history.
   - Select **Linear** from the **Analysis Type** drop down list box.
   - In the **Load Assignment** area:
     - select ACC DIR1 under **Load**
     - select ELCENTRO under **Function**
     - Set the **Scale Function** to the acceleration due to gravity which is 386.4 in/sec^2 if your units are in Kip-in and 32.2 ft/sec^2 if your units are in Kip-ft.
     - Set the **Arrival Time** and **Angle** to Zero.
     - Press the **ADD** button to add the load assignments and the OK button to accept the data you just entered.
   - Press the OK button on both forms to accept your additions.

We have now entered all the information we need for a Time History analysis.

> Note: ***It is usually a good idea to run your model every time you make a major change or addition. This will give you a way to catch your mistakes early and save you time in your final design.***

Running Analysis

1. Save your model.
2. Set the parameters for the design run by selecting from the **Analyze** menu **Set Options.**
   - Check that the **Dynamic Analysis** settings are the same as for Tutorial 2.
3. Select **Run** from the **Analyze** menu to analyze the structure.

Using the Results

Checking Results

Once you have run the model successfully you need to check and see that the output is correct and close to what you expected.

1. Check the base shear produced due to the time history analysis.
   - From the **Display** menu select the **Show Time History Traces** option.
     - ♦ From the **Time History Display Definition** form press the DEFINE FUNCTIONS button.
     - ♦ In the **Time History Functions** form select Add Base Functions and check only the Base Shear X option.
     - ♦ Press the OK buttons to get back to the **Time History Display Definition** form.
     - ♦ Add the Base Shear X function to the **Plot Functions** list box.
     - ♦ Then press the DISPLAY button to see the plot of the base shear in the global X direction as a function of time. See Figure 3-2.

   Note: ***You can also view the Time History plot of the base shear by selecting Add Group Summation Forces instead of Add Base Functions and selecting the base shear group that was defined in Tutorial 1.***

2. You can also check the displacement of a joint due to the time history.
   - Select a joint and from the **Display** menu choose the **Show Time History Traces** option.
     - ♦ Press the DEFINE FUNCTIONS button and in the **Time History Functions** form select the joint name from the list and press the MODIFY/SHOW TH FUNCTION button.
     - ♦ In the Time History Joint Function form select the DISPL **Vector Type** and UX **Vector Direction**.
     - ♦ Press the OK button to accept the changes.
     - ♦ Press the OK buttons to go back to the **Time History Display Definition** form.
     - ♦ Add the joint from the **List of Functions** to the **Plot Functions** list box and remove the Base Shear X function.
     - ♦ Press the DISPLAY button to see the joint displacement with respect to time. See Figure 3-3.

- You can also define a joint function directly in the **Time History Display Definition** form without selecting it first.
  - ♦ In the **Time History Display Definition** form press the DEFINE FUNCTIONS button and in the **Time History Functions** form select Add Joint Disps/Forces.
  - ♦ In the Time History Joint Function form enter the Joint ID.
  - ♦ Select the **Vector Type** and **Vector Direction.**
  - ♦ Press the OK buttons to go back to the **Time History Display Definition** form where you will find the new joint function in the **List of Functions** list box.

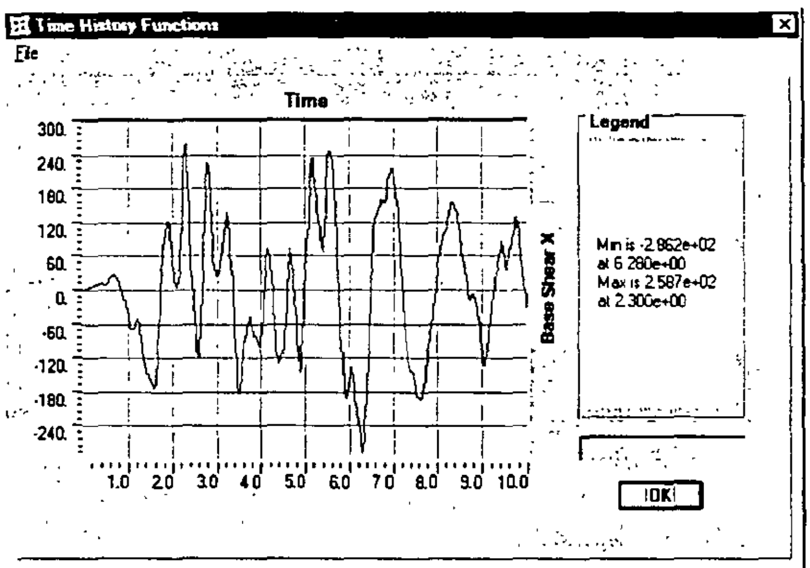


Figure 3-2 Base Shear - Kips

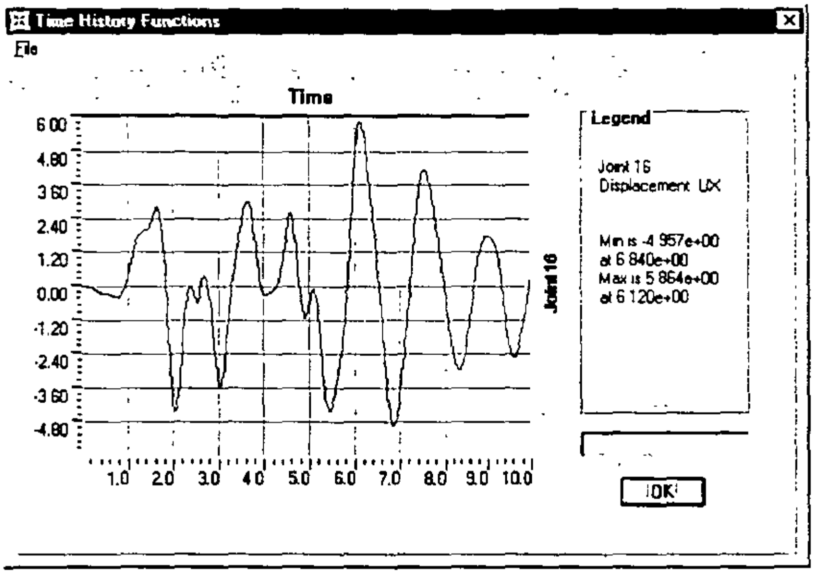


Figure 3-3 Roof Time History Displacement - in

Building Response Spectrum Data

The first thing to do is plot a Response Spectrum from the time history data. Then the data will need to be printed to a file and the file edited into a format that can be read by SAP2000.

Plotting the Response Spectrum

1. Select one joint at the base of the structure.
2. From the **Display** menu select **Show Response Spectrum Curves**. This option only appears when a joint is selected.
3. In the **Response Spectrum Generation** form you will find the name of the joint that was selected.
   - Under the **Define** tab select the X **Vector Direction**.
   - Under the **Axes** tab select **Period** for the **Abscissa** and **PSA** (Pseudo Spectral Acceleration) for the **Ordinate**.
   - Under the **Options** tab select **Arithmetic** for both the **Abscissa** and **Ordinate**. For the **Ordinate** set the scale factor to 1/g (g=32.2 ft/sec^2) which will be .03106 sec^2/ft if the units are set to kip-ft.

Note: ***The scale factor is used to normalize the Response Spectra. The Time History that is being used to produce the Response Spectra has been scaled by g so we need to divide by the same number to get back to normalized values.***

- Under the **Period** tab select the **Default** and **Structural** frequencies, which are used in the generation of Response Spectrum. **Default** frequencies are a set of built-in frequencies, typically of interest in structures. **Structural** frequencies are the structure's natural frequencies.
- Under the **Damping** tab keep only the .05 **Damping Value**. As the structure is assumed to have 5% damping, we will not need any other values.
- Press the DISPLAY button when you finish.

4. You will see a plot of a response spectrum for the El Centro earthquake at 5% damping. See Figure 3-4.

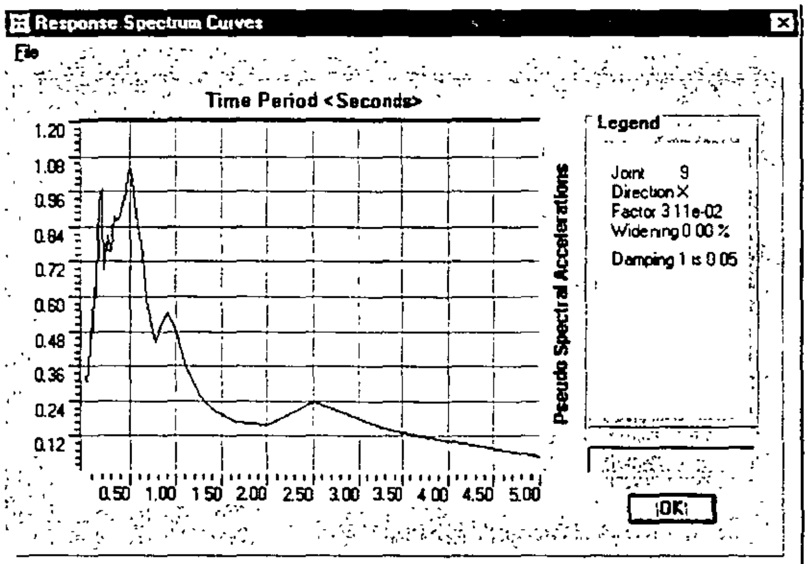


Figure 3-4 Response Spectrum from Time History

5. In the **Response Spectrum Curves** form select from the **File** menu the **Print Tables to File** option. This will make a file that has two columns. The first one is the Period and the second is the PSA for each period.
   - Save the file under the name RS-ELCEN.TXT

Editing Table

The next step is to do some minor editing on the RS-ELCEN.TXT text file so it is in a format that SAP2000 can read. When the file is made, there is some extraneous information that is added to help you understand the content of the file. This extra information needs to be deleted.

1. Open RS-ELCEN.TXT in a text editor like WORDPAD or NOTEPAD.
    - Select all the text that is shown highlighted in Figure 3-5 and delete it.
    - Save RS-ELCEN.TXT as a text file using the same name.
2. Now that the file only has the Period and the PSA columns, it is in a format that SAP2000 can read.

```
SAP2000 v6.06 File: TUTORIAL1  Kip-ft Units  PAGE 1
May 19, 1997 17:26

S P E C T R U M   D A T A

Joint        9
Direction    XT
Factor       0.03
Widening     0 %

Pseudo Spectral Accelerations vs Time Period <Seconds>

              DAMPING
               0.0500
    0.0303  3.0866E-01
    0.0357  3.2657E-01
    0.0400  3.1139E-01
    0.0455  3.2217E-01
    0.0500  3.2205E-01
    0.0556  3.3098E-01
    0.0606  3.4265E-01
    0.0667  3.7431E-01
```

Figure 3-5 Text File of Response Spectrum Table

Reading Spectral Data

Now that we have the data in a format that can be read by SAP2000 we simply need to tell the program where the file is and how it is set up.

1. If the model is locked press the **Lock/ Unlock Model** button on the toolbar. This will unlock the model and allow you to make modifications to it.
2. From the **Define** menu select **Response Spectrum Functions**.

3. In the **Response Spectrum Functions** form press the **Add Function from File** button.
   - Name the spectra RSELCEN
   - Press the **Open File** button and pick the RS-ELCEN.TXT file from the **Pick File** form.
   - Keep the **Number Of Points Per Line** to 1 as there is only one set of response data on each line.
   - Select the **Period and Acceleration Values** option.
   - Press the OK buttons to close the forms.
4. From the **Define** menu select **Response Spectrum Cases.**
5. In the **Response Spectrum** form press the ADD NEW SPECTRA button.
   - Set the **Modal Damping** to 0.05.
   - In the **Input Response Spectra** area select RSELCEN for the U1 direction and give it a scale factor of 32.2 ft/sec^2.
   - The remaining default values are acceptable.
   - Press the OK buttons to close the forms.

Running Analysis

Once you have made the modifications, it is time to run the model and take a look at our results.

1. Save your model.
2. Select **Run Minimized** from the **Analyze** menu to analyze the structure.

Checking Results

The first thing to do is check the maximum deflection at the top of the structure and the base shear due to the Time History and the Response Spectrum. This will show how well the outlined method works. At the end of this section you will find the results of the static lateral load, Response Spectrum and Time History analysis.

Response Spectrum Deflection

1. From the **Display** menu select **Display Deformed Shape**.
   - In the **Deformed Shape** form select the load case for the spectral analysis.
   - Press the OK button.
2. Right click on a joint at the top level of the structure to see the displacement of the joint in the global X direction.

Response Spectrum Base Shear

Using the BASE SHEAR group that was set up in Tutorial 1, look at the base shear for the structure due to the Response Spectrum.

Time History Deflection and Base Shear

1. Using the method outlined in the first part of this tutorial, plot the deflection at the top of the structure.
2. Now remove the joint from the **Plot Functions** list and plot the **Base Shear X** direction by selecting it from the **List of Functions** in the Time **History Display Definitions** form.

	Static Lateral	**Response Spectrum**	**Time History**
Max Deflection	1.6 in	5.5 in	5.9 in
Max Base Shear	72.5 Kips	306 Kips	286 Kips

Table 3-1 Comparison of Lateral Load Results

Final Comments

As you have seen the Time History analysis is much more time consuming than a Response Spectrum analysis. The Response Spectrum and Time History analysis results can give similar results. However it is very important for the engineer to understand the strengths and limitations of each method so they can be effectively used.

TUTORIAL 4

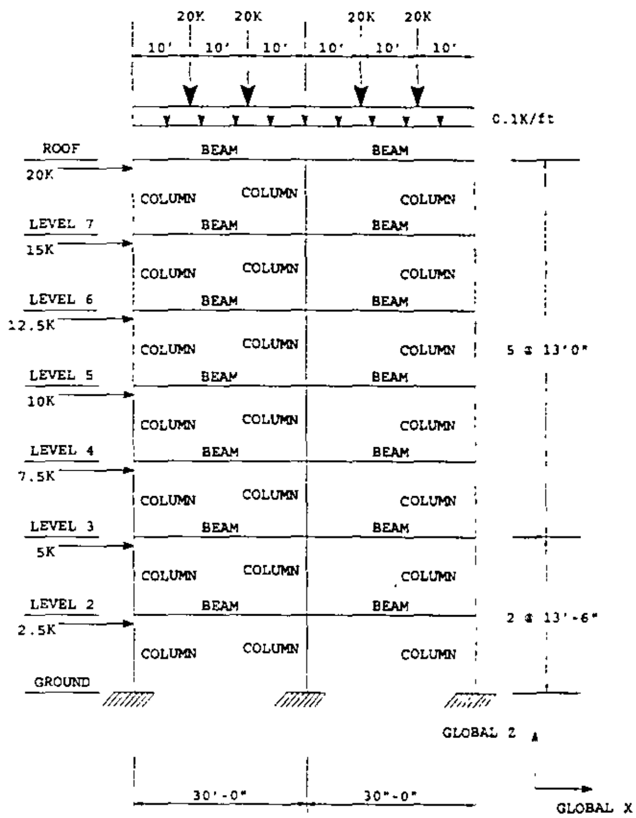
2-D Frame Steel Design

Description

This tutorial is provided to highlight the powerful tools in SAP2000 for designing the structure after it has been analyzed. The steel design features will be looked at in this tutorial to design the structure that was modeled in Tutorial 1.

Significant Features of Model and SAP2000

- Assigning member rigid End Offsets
- Automatic selection groups
- Changing member properties
- Designing members by groups
- Including P-Delta effects in analysis
- Viewing design results
- Overriding Auto Selection sections



ALL COLUMNS ARE W14'S
ALL BEAMS ARE W24'S
TYPICAL STORY MASS = 0.49 kip-sec-sec/in
MODULUS OF ELASTICITY = 29500 ksi
STEEL STRENGTH (f'y) = 36ksi

Figure 4-1 Two-dimensional Frame Tutorial

Building the SAP2000 Model

You will be able to use the file you developed in Tutorial 1 with a few modifications.

Materials

First thing to do is set the material properties.

1. Check that the units are set to Kip-in.
2. From the **Define** menu select **Materials.**
3. Select the STEEL **Material** type and press the MODIFY/SHOW MATERIAL button.
4. Set the **Steel Yield Stress** fy to 36 Ksi.
5. Set **Modulus of Elasticity** E to 29,500 Ksi.
6. Press the OK buttons to accept the changes and close the forms.

Loads

1. In Tutorial 1 you assigned a set of point and uniform loads as DEAD load and included the member self weight. (See Figure 4-2 for the new **Static Loads Case** list.) In this tutorial we will assign a load case for the live load and one for the self weight of the members. Including a load case for the self weight of the structure is a good way to keep track of the structural weight for design optimization purposes. The loads are separated into dead, live and lateral earthquake load cases so the design part of SAP2(can automatically generate load combinations.
   - For the DEAD load case set the .lf **Weight Multiplier** to 0.
   - Add a load case SELF of **Type** DEAD for the member self weight and set **Self Weight Multiplier** to 1.
   - Add another static load case named LIVE and assign it as **Type** LIVE.
2. Add the same loads that are in the DEAD load case to the LIVE load case. This will mean that each beam in the structure has identical dead and live loads. (See Tutorial 1 for instructions on entering loads.)

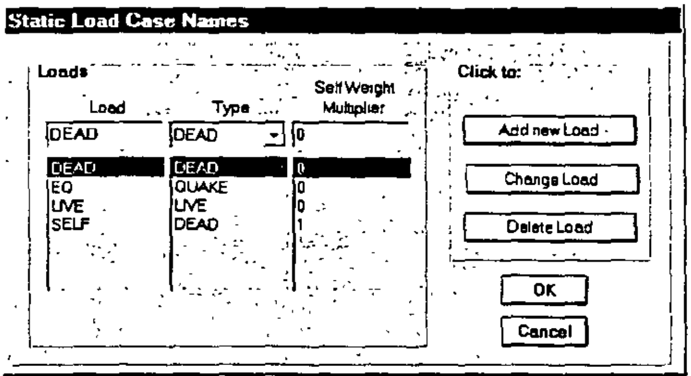


Figure 4-2 Static Load Cases

Defining an Auto Selection Group

The Auto Selection feature in SAP2000 is a very effective way to design structures. By defining a set of steel sections as an Auto Selection section group, the program can design each frame element using only the sections in that group. For example we can define an Auto Selection group called COLUMN with only W14 sections and a BEAM group with only W24 sections. In this way frame members assigned a COLUMN Auto Selection section will only be designed using W14 sections.

The first thing to do is define an Auto Selection group, which only includes column sections. Essentially what we are doing is giving the program a list of sections which it can choose from when designing the frame members. In turn the program will select the most efficient section out of that group.

Once the preliminary design is done and it is time to start fine tuning the design, the BEAM and COLUMN groups can be replaced by the optimum sections from those Auto Selection groups. This will then assign a specific section for both the analysis and design, which will make it much easier to change the few sections that may need to be modified.

Note: ***Auto Selection only works for steel frame sections.***

1. From the **Define** menu select **Frame Sections.**
2. In the **Frame Sections** form import all the steel sections between W14x61 and W14x283.
   - Select IMPORT I/WIDE FLANGE from the Import drop down list box.
   - In the sections form scroll down and select W14x283.
   - With the SHIFT button pressed left click on W14x61 and press the OK button. This will select these sections and all the sections in between.
3. In the **Frame Sections** form delete any duplicate sections.

Remember*: You will not be able to delete a section that is in use. This feature is a way for the program to insure that all the members are assigned to existing sections.*

4. From the **Frame Section** form add an **Auto Select** section which will be at the bottom of the **Add** drop down list box.

   - Change the **Auto Section Name** to COLUMN.

   - In the **Auto Selections** list box select and remove using the **Remove** button all the sections except the W14's. This will mean that any frame member assigned a COLUMN section will be designed from the list of W14 sections in the **Auto Selections** list box. (See Figure 4-3.)

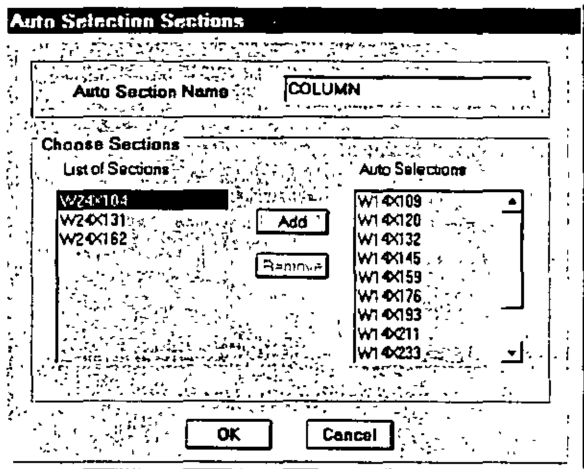


Figure 4-3 Defining the COLUMN Auto Selection group

5. Using the instructions outlined in steps 2 through 4:

   - Import all the sections between W24x55 and W24x162.

   - Assign an Auto Selection group called BEAM with only the W24 sections in it.

6. Finally, select all the vertical frame members and assign the COLUMN section to them. Then select all the horizontal frame members and assign the BEAM section to them. (See Tutorial 1 for instructions on assigning sections to frame members.)

   Note*: You can of course select a specific section for both the analysis and design instead of using an Auto Select section. You simply need to assign the frame member a steel section and design it as outlined in Tutorial 1. The steel section can either be a user-defined section or a section from the Section Property file.*

Running Analysis

Once data has been entered, it is time to run the model and take a look at our results.

1. Save your model.
2. Set the parameters for the design run by selecting from the **Analyze** menu **Set Options**.
   - In the **Analysis Options** form select a **Plane Frame** analysis to reduce the size of the solution and thus reduce the time needed for the analysis.
   - Select the **Include P-Delta** check box.
   - Press the SET P-DELTA PARAMETERS button to set the analysis parameters.
     - Set **Maximum Iterations** to 10.
     - Include the two dead load cases DEAD and SELF in the P-Delta load combination with a factor of 1.
     - Include the LIVE load case with a factor of 1.

     Note: ***The load factors that you use need to be the load factor for the design load combo that includes lateral loads and has the largest vertical load on the structure.***

     - The remaining default values are acceptable.
     - Press the OK buttons to accept the changes and close the forms.
3. Select from the **Analyze** menu the **Run** option to analyze the structure.

   Note: ***Because we have assigned groups of sections and not specific sections for our design, SAP2000 will take suitable section properties to calculate the stiffness matrix and any other properties it may need. Once the first analysis and design has been performed, the program can be instructed to use the designed sections for analysis.***

Designing Sections

Once you have run the analysis and checked the analysis results, you can set the parameters needed for steel design.

Selecting Code

The information in the analysis run is used to do a code check on the frame elements.

1. From the **Options** menu select **Preferences**.

2. In the **Preferences** form under the **Steel** tab, select the steel code you want to use. In this case let's use AISC-ASD-89.

    - Use the same **Section Properties file** that was used for importing the steel sections.

Load Combinations and Design

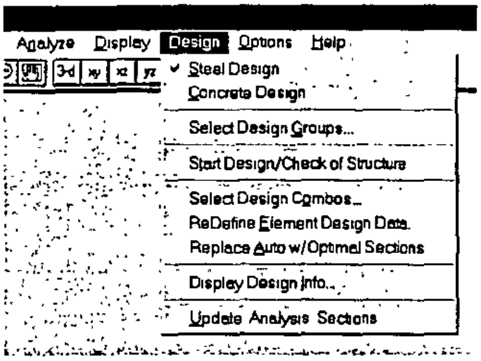


Figure 4-4 Design Menu Options

Once you have selected the code you will be using to design the members, you need to check the load combinations that will be used for the design.

1. The first thing to do is make sure that under the **Design** menu the **Steel Design** item has a check mark on it. This will tell SAP2000 to design the steel sections.

2. From the **Design** menu choose **Select Design Combos.**

    - Look at the automatically created combos in the **Design Combos** list box by selecting the combo and pressing the SHOW button.

    - If you find that there are other load combos that you want to use in your design, you can add them. Simply define the combo from the **Define** menu and then in the **Select Design Combos** form add the combo to the **Design Combos** list.

3. Run the Design/Check from the **Design** menu by selecting **Start Design/Check of Structure.**

    - Each of the elements will be designed with the most efficient section from its Auto Selection section group.

    - SAP2000 will automatically display on the active window the percent of maximum stress below each of the elements.

    - The elements will also be color coded for convenience with a stress ratio color key located at the bottom of the window.

Note: ***If you just want to check the design for a limited number of frame members, you can select those members and then choose Start Design/Check of Structure.***

Viewing Results and Redesigning

The first thing to do when a Design/Check is performed, is to see that the results are correct. SAP2000 gives the user some tools to check the results.

1. Right click on any member to view its design results. The element you have selected will be flashing to identify itself.
2. In the **Steel Stress Check Information** form, you will find a list of the load combinations used to check the section at various locations along the members. (See Figure 4-5)
   - One of the combos will be highlighted when you first see the form. That is the controlling combo that was used to design the section.
   - Next to each combo there is the location along the member where the check was performed followed by the stress ratio for moment interaction and shear.

     Hint: ***You can change the number of locations at which the forces are reported. Select the frame elements and choose from the Assign menu Frame ...Output Segments and change the number of segments. You will then need to rerun the analysis to get the result for the new segments.***
3. Selecting any of the combo checks and pressing the **Details** button will show the analysis results for that member and governing equation from the code. (See Figure 4-6)
4. Pressing the **ReDesign** button will bring up the **Element Overwrite Assignments** form. In this form you can:

   Note: ***If you make any changes in the Element Overwrite Assignments form through the ReDesign button, you will need to press the Refresh Window button on the tool bar to see the updated design results on the current window.***

   - Select another section to see the effects of the change on the stress ratios.

     Note: In the Auto Selection mode, this section can be used in developing a new stiffness matrix if you "Update Analysis Sections". However, Auto Selection will still take place the next time you run the design check.
   - Set the member as a **Moment Resisting Element** or a **Brace.**
   - Overwrite design factors like the effective length, unbraced length ratio and so on.

- **Overwrite Allowable Stresses** used for the design. This is a value in the current units for the allowable stress of the section.
- When you have finished modifying the design parameters press the OK button.

  Note: ***Changing any of the information in the ReDesign form will make SAP2000 automatically recalculate the design stress ratios for that new information and update the Steel Stress Check Information form. See the Re-Analyzing section for instructions on updating the analysis sections.***

5. To use the ReDesign section in the next analysis you will need to select from the **Design** menu the **Update Analysis Sections** option. This will replace the sections used to build the stiffness matrix so that a more accurate design check can be made.

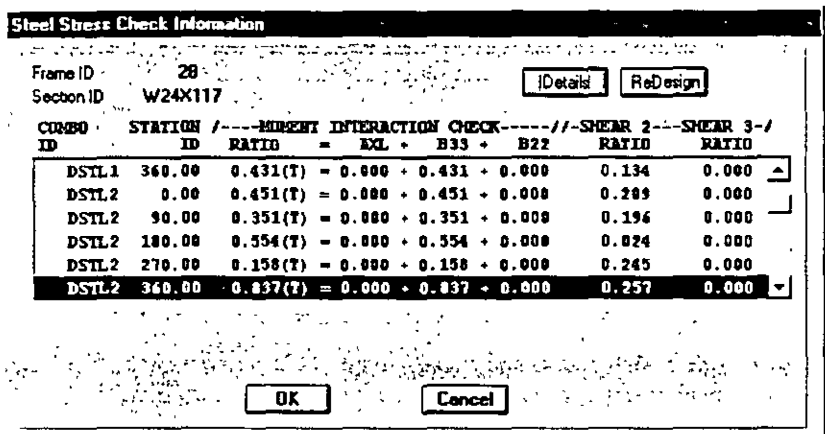


Figure 4-5 Steel Stress Check for Specified Load Combinations

Note: ***If you just want to check the design for a limited number of frame members, you can select those members and then choose Start Design/Check of Structure.***

Viewing Results and Redesigning

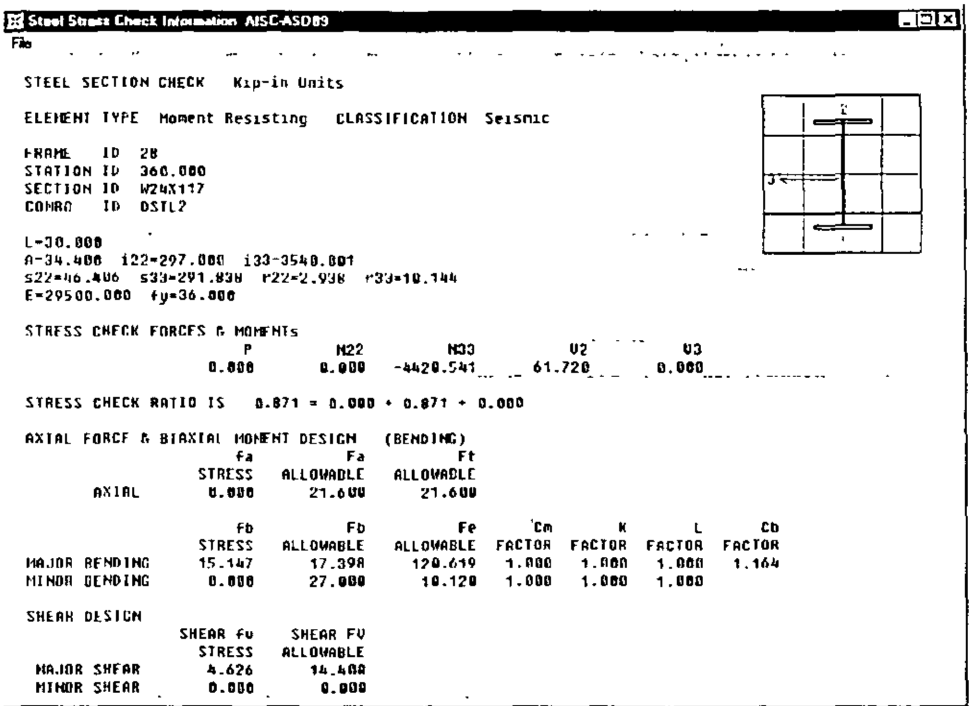
The first thing to do when a Design/Check is performed, is to see that the results are correct. SAP2000 gives the user some tools to check the results.

1. Right click on any member to view its design results. The element you have selected will be flashing to identify itself.
2. In the **Steel Stress Check Information** form, you will find a list of the load combinations used to check the section at various locations along the members. (See Figure 4-5)
   - One of the combos will be highlighted when you first see the form. That is the controlling combo that was used to design the section.
   - Next to each combo there is the location along the member where the check was performed followed by the stress ratio for moment interaction and shear.

     Hint: ***You can change the number of locations at which the forces are reported. Select the frame elements and choose from the Assign menu Frame ...Output Segments and change the number of segments. You will then need to rerun the analysis to get the result for the new segments.***
3. Selecting any of the combo checks and pressing the **Details** button will show the analysis results for that member and governing equation from the code. (See Figure 4-6)
4. Pressing the **ReDesign** button will bring up the **Element Overwrite Assignments** form. In this form you can:

   Note: ***If you make any changes in the Element Overwrite Assignments form through the ReDesign button, you will need to press the Refresh Window button on the tool bar to see the updated design results on the current window.***
   - Select another section to see the effects of the change on the stress ratios.

     Note: In the Auto Selection mode, this section can be used in developing a new stiffness matrix if you "Update Analysis Sections". However, Auto Selection will still take place the next time you run the design check.
   - Set the member as a **Moment Resisting Element** or a **Brace.**
   - Overwrite design factors like the effective length, unbraced length ratio and so on.



```
Steel Stress Check Information  AISC-ASD89
File

STEEL SECTION CHECK   Kip-in Units

ELEMENT TYPE  Moment Resisting   CLASSIFICATION  Seismic

FRAME   ID  28
STATION ID  360.000
SECTION ID  W24X117
COMBO   ID  DSTL2

L=30.000
A=34.400  i22=297.000  i33=3540.001
s22=46.406  s33=291.838  r22=2.938  r33=10.144
E=29500.000  fy=36.000

STRESS CHECK FORCES & MOMENTS
                     P          M22          M33          V2          V3
                 0.000        0.000    -4420.541      61.720       0.000

STRESS CHECK RATIO IS   0.871 = 0.000 + 0.871 + 0.000

AXIAL FORCE & BIAXIAL MOMENT DESIGN   (BENDING)
                    fa          Fa           Ft
                STRESS   ALLOWABLE    ALLOWABLE
       AXIAL     0.000      21.600       21.600

                    fb          Fb           Fe        Cm         K         L        Cb
                STRESS   ALLOWABLE    ALLOWABLE    FACTOR    FACTOR    FACTOR    FACTOR
MAJOR BENDING   15.147      17.398      120.619     1.000     1.000     1.000     1.164
MINOR BENDING    0.000      27.000       10.120     1.000     1.000     1.000

SHEAR DESIGN
              SHEAR fv    SHEAR FV
                STRESS   ALLOWABLE
 MAJOR SHEAR     4.626      14.400
 MINOR SHEAR     0.000       0.000
```

Figure 4-6 Detailed Steel Design Info of a Beam Element

6. You can also look at various design results on the graphics screen by selecting from the **Design** menu the **Display Design Info** option. These results will be plotted on the bottom and to the right of frame members.

   Note: ***The sections used for the analysis are shown at the top and to the left of the frame members. All the design information is shown at the bottom and to the right of the frame members.***

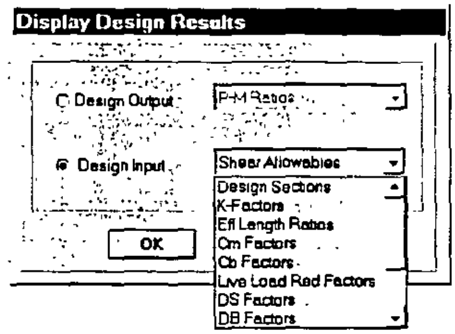


Figure 4-7 Display Design Input and Results Form

7. You can also Print out the design results by selecting from the **File** menu **Print Design Tables.** To print results on a limited number of members, select the members and then selecting from the **File** menu **Print Design Tables.**

Editing Section Properties

From Figure 4-1 you will see that the beams have point loads coming in at their third points. Let us assume these point loads are coming from other members and so the beams are supported laterally at their third points. The model as it is defined does not take that into consideration, so the beams are over designed. To design the beam more accurately we will need to edit the beam design properties.

1. Select all the beams in the structure. You may want to use the **Intersecting Line Select Mode**.
2. Select from the **Design** menu the **ReDefine Element Design Data.**
   - In the **Element Overwrite Assignments** form select the **Unbraced Length Ratio, L22** check box and set the value to 0.33. This will brace that member at 1/3 rd points (against buckling in the local 1-2 plane) instead of the default of braced at the ends only. (See Figure 4-8)
   - Press the OK button when you have entered the new value. SAP2000 will then automatically rerun the Design/Check and update the model.
3. You will now see that the beam members are smaller.

   Hint: ***You can get a group load joint sum report on the Base Shear group to find what the self-weight of the structure is. This is a quick way to see if your changes have produced a more efficient structure. You can also get the total weight of the structure from the filename.EKO file.***

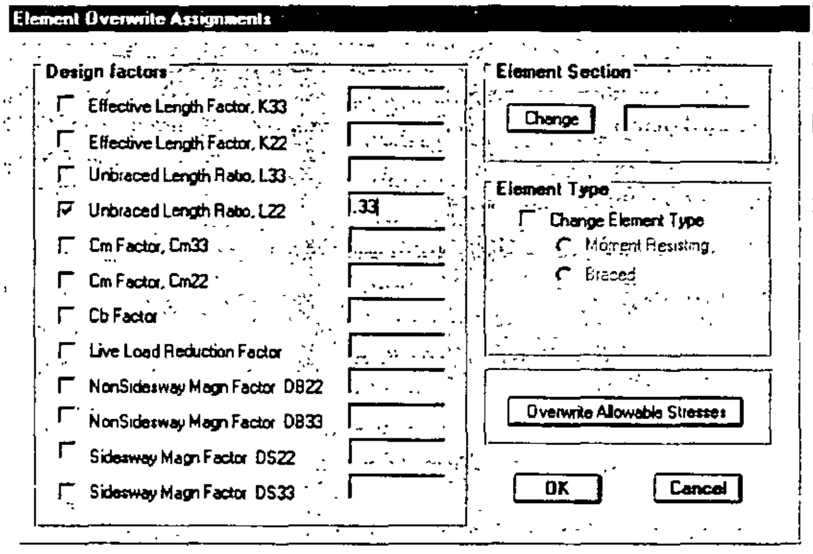


Figure 4-8 Element Overwrite Assignments for Beams

Re-Analyzing

The first analysis run used approximate section properties to develop the stiffness matrix. So the model will need to be run again as an iterative process to make sure that the analysis is done using the selected sections.

1. Once you have finished modifying the structural sections you will need to use, from the **Design** menu select the **Update Analysis Sections** option. Then rerun the analysis to use the selected member properties for the stiffness matrix.
2. Then rerun the design to see which members have changed.
3. Once you are satisfied with the sections selected, from the **Design** menu choose the **Replace Auto w/ Optimal Sections** option. This will permanently replace the auto sections with the current design sections. This effectively makes the design section the analysis section and therefore replaces the BEAM and COLUMN analysis sections with the optimal or user selected sections.

LRFD Design

The method used for LRFD design is essentially the same as that used for ASD. The combos and checks made on the frame members will be performed using the LRFD code so the results and information reported are different. To run an LRFD Design/Check you will need to change some of your input parameters.

1. Enter new load factors for the P-Delta analysis.
2. Select from the **Preferences** form the AISC-LRFD93 steel code.
3. Redesign the steel sections.

Advanced Features

Defining Frame Element Groups

Sometimes you may find it helpful to design using frame member groups. This will force all the members in the group to be designed with the same section. The advantage of this method is that it will reduce the various number of sections used in the design. For example you can group two or three floors as one group when designing them. This will then provide you with one section that works for all the columns and one section that works for all the beams in that group.

1. Reassign the Auto Selection sections to the frame members.
2. Assign all the frame sections at level 3 and below to a group name BOTTOM.
3. Assign all the frame elements between Level 5 and Level 3 to a group name MIDDLE.
4. Assign the remaining elements to a group name TOP.
5. Rerun the analysis on the model.
6. From the **Design** menu choose the **Select Design Group** option..This is where one can have the program design a group of members and assign all of them with the lightest section that satisfies the stresses in all of them.
   - Include in the **Design Groups** list box only the member groups TOP, MIDDLE and BOTTOM. This will mean that each frame element group is designed with the most efficient section from the Auto Selection group.

     Note: ***If there are no groups in the Design Group list box then all the members will be designed individually.***

   - When you press the OK button, SAP2000 will automatically design the steel sections and display the results in the active window.
3. Compare the results from the first design run and the group design run to see how that will affect the selected sections.

End Offset

This structure is designed to be a moment frame system with members that have no cross sectional dimensions for analysis purposes. However, even though this is not a bad assumption, SAP2000 allows a more accurate way of modeling this problem. By setting Member End Offsets, you can define a region about the beam column connection where the members can not bend. This essentially produces a rigid zone at that connection. The area can be as large as the user wants, but it is usually taken as the depth of the member that is framing into it, at that joint (or a fraction thereof).

1. Select all the frame elements.
2. From the **Assign** menu select **Frame... End Offsets.**
   - In the **End Offset** form select the **Update Lengths From Current Connectivity** option. This will make the program automatically calculate the end offset from the sections coming into each joint.
   - Enter 1 for **Rigid Zone Factor**. This means that 100% of the potential End Offset length should be taken as rigid in the analysis run.
   - Press the OK button.
3. If you set the **Element Shrink** option on from the **Set Elements** form and look at the active graphics window, you will see that there are lines at each joint that show the assigned End Offsets.

   Remember: ***You will need to reset End Offsets every time the frame sections are changed.***

   Note: ***The moment and shear values on the beams and columns will be slightly different than with no rigid End Offsets. This is because the rigid End Offset assignment reduces the flexible length of the members.***

Final Comments

The steel design tools in SAP2000 are very helpful in designing the frame members. However there are a few things to keep in mind:

1. Make sure that all the design data for the sections are correct. The default values that the program uses for design may not be correct due to various methods you may have used to model your structure, e.g. K and Unbraced Length Ratios. You can use the **Display Design Results** form to view this information on the framing. For convenience, you can also view the analysis sections at the same time as the design information.
2. Check that the design combos the program has provided are correct and sufficient for your particular structure. If they are not, add your own to the list used in design.
3. Check the final design results at key locations to make sure they are the results you expect.

4. Check that the load factors used for the P-Delta analysis are correct.
5. Redesign the structure whenever you make changes to the model. This will give you a way to see if the members are still acceptable.
6. Use a group to help find the total weight of the structure. (See Tutorial 1 for instructions on how to do this.)
7. Use groups for your design to reduce the different number of sections that are in your model.
8. The *filename*.EKO file contains the total weights for each section used. This information can be used for initial cost estimations.

Appendix A –Tool Bar Icon Descriptions

Icon	Control Name	Allows You to
	New Model	Start a new model.
	Open *.SDB file	Open existing SAP2000 file.
	Save Model	Save the current model.
	Undo	Undo last change.
	Redo	Reverse last Undo.
	Refresh Window	Refresh current window using latest data.
	Lock/Unlock Model	Lock model against changes to analysis data.
	Run Analysis	Run analysis.
	Zoom	Zoom in on structure by defining area with mouse.
	Restore Full View	Restore full view of model.
	Restore Previous View	Restore previous view of model
	Zoom In	Zoom in to model.
	Zoom Out	Zoom out of model.
	Pan	Dynamically move structure in any direction.
3-d	Show 3-d view	Show 3-d view of model.
xy	Show 2-d View of X-Y/r-Θ Plane	Show 2-d View of model parallel to X-Y/r-Θ plane.
xz	Show 2-d View of X-Z/r-Z Plane	Show 2-d View of model parallel to X-Z/r-Z plane.
yz	Show 2-d View of Y-Z/Θ-Z Plane	Show 2-d View of model parallel to X-Z plain or a developed view of the r-Z plane.
66'	Perspective Toggle	Show view in 3-d perspective.
	Shrink Elements	Shrink all elements to help view element connectivity.
	Set Element	Set visibility of elements and their properties.
	Up One Gridline	Move up one grid line in 2-d viewing plane.
	Down One Gridline	Move down one grid line in 2-d viewing plane.

Appendix B – Floating Toolbar Icon Descriptions

Icon	Control Name	Allows You to
	Pointer Tool	Select elements individually or within a box.
	Select All	Select all elements in drawing.
	Restore Previous Selection	Restore previously selected elements.
	Clear Selection	Clear all selected elements.
	Set Intersecting Line Select Mode	Select all elements intersected by drawn line.
	Reshape Element	Move elements by selecting them at their center and to reshape them by selecting them at their ends.
	Add Special Joint	Add a joint that is not automatically added when an element is entered.
	Draw Frame Element	Draw a frame element by locating its end locations.
	Draw Shell Element	Draw a shell element by locating its corners.
	Quick Draw Frame Element	Draw a frame element between grid lines.
	Quick Draw Shell Element	Draw a shell element between grid lines.
	Assign Joint Restraints	Assign translational and rotational restraints.
	Assign Frame Sections	Assign frame section and material properties.
	Assign Shell Sections	Assign shell section and material properties.
	Assign Joint Load	Assign joint loads.
	Assign Frame Span Loading	Assign frame loads.
	Assign Shell Uniform Loading	Assign shell loads.
	Show Undeformed Shape	Show original structural shape.
	Display Static Deformed Shape	Display deformed shape due to static loads.
	Display Mode Shapes	Display Mode shapes and periods.
	Display Element Force/Stress Diagram	Display Element analysis data on the structure and individually.
	Set Output Table Mode	Display Joint or Element Text Output on Screen.

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 INTEGRATED FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

GRAPHIC USER INTERFACE MANUAL

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

SAP2000®

Integrated
Finite Elements Analysis
and
Design of Structures

GRAPHIC USER INTERFACE MANUAL

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.

Berkeley, California, USA

Version 6.1

September 1997

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *support@csiberkeley.com*
web: *www.csiberkeley.com*

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMP-TIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table of Contents

New Interface 1
Designer's Convention 1
Noun-Verb 1
2 Modes of SAP2000 1
Select Mode 2
Setup Coordinate System 2
Frame Element 3
Shell Element 4
Asolid Element 5
Solid Element 5
Nonlinear Analysis 5
Getting a Head Start With Templates 7
Import/Export Capabilities 8
Import a SAP90 text input file 8
Import a SAP2000 text input file 8
Export a SAP2000 text input file 8
Import a DXF file 9
Export a DXF file 9
Import a SAP2000 Job Run in DOS mode 9
Export an AVI file 9
View Real Time Animation for Time History Results 10
Print Setup 10
Print Selected Graphical Output to a Printer or a File 10
Text Output 11
Print Input Tables to a Printer or a File 11
Print Output to a Printer or a File 11
Print Design Results to a Printer or a File 11
User Comments and Session Log 12
Undo and Redo Capabilities 13
Cut, Copy and Paste 13
Deleting Members 13
Move 14
Replicate 14
Replicate in a Linear Array 14
Replicate in a Radial Array 14
Replicate in a Radial Array by Shifting the Origin 15
Replicate by Using the Mirroring Option 15
Merge Joints 16
Automatic Meshing 16
Divide or Break Frames 16
Mesh Shells 17
Join Frames 17
Disconnect 17
Connect 18
Show Duplicates 18
Joint and Element Labels 18
Re-Label Previously Assigned Labels 18
Select 3-D Views 21
Select 2-D Views 21
Set Elements 22
Set Limits 22
Define Materials 23
Add a New Steel Material Type 23
Add a New Concrete Material Type 23

ADD A NEW OTHER MATERIAL TYPE....24
DEFINE SECTION PROPERTIES....24
DEFINE FRAME SECTION PROPERTIES....24
IMPORT A FRAME SECTION FROM THE SECTION PROPERTY FILE (E.G. SECTIONS.PRO)....25
ADD A FRAME SECTION BY DEFINING THE DIMENSIONS AND OR PROPERTIES MANUALLY....25
ADD A NONPRISMATIC FRAME SECTION....26
ADD AN AUTO SELECT FRAME GROUP....27
ADD A SHELL SECTION BY DEFINING THE DIMENSIONS AND / OR PROPERTIES....27
ADD AN NLLINK PROPERTY....27
NL DAMPER, GAP AND HOOK PROPERTIES....28
NL PLASTIC 1 PROPERTIES....28
NL ISOLATOR 1 PROPERTIES....28
NL ISOLATOR 2 PROPERTIES....29
DEFINE GROUP NAMES....29
DEFINE STATIC LOAD CASES....30
BRIDGE ANALYSIS....30
DEFINE LANES....30
DEFINE VEHICLES....31
STANDARD VEHICLES....32
GENERAL VEHICLE....35
DEFINE VEHICLE CLASSES....35
DEFINE BRIDGE RESPONSE....36
DEFINE MOVING LOADS....36
DEFINE JOINT PATTERNS....37
ASSIGN DYNAMIC LOADS....37
DEFINE TIME HISTORY FUNCTIONS....37
DEFINE TIME HISTORY CASES....38
DEFINE RESPONSE SPECTRUM FUNCTIONS....38
DEFINE RESPONSE SPECTRUM CASES....39
DEFINE LOAD COMBINATIONS....39
DRAW MODE....41
DRAW MEMBERS....41
RESHAPE ELEMENT....42
ADD SPECIAL JOINTS....42
DRAW A FRAME ELEMENT FROM JOINT TO JOINT....43
DRAW A QUICK FRAME ELEMENT....43
DRAW A SHELL ELEMENT BETWEEN 4 JOINTS....43
DRAW A QUICK SHELL ELEMENT....44
DRAW AN NLLINK ELEMENT....44
EDIT GRIDS....44
SNAP TO GRID....45
SNAP TO JOINTS....45
SNAP TO FRAME/EDGE....46
INITIALIZE A NEW LABELING SCHEME....46
SELECT MENU....46
SELECTION PROCEDURES....46
SELECT OBJECTS BY WINDOW....47
SELECT OBJECTS BY INTERSECTING LINE....47
SELECT OBJECTS BY 2D PLANES....47
SELECT OBJECTS BY GROUPS....47
SELECT OBJECTS BY FRAME SECTIONS....47
SELECT OBJECTS BY SHELL SECTIONS....47
SELECT OBJECTS BY NLLINK PROPERTIES....48
SELECT OBJECTS BY CONSTRAINTS....48
SELECT OBJECTS BY LABELS....48
SELECT ALL OBJECTS....48

ASSIGN OPTIONS 49
ASSIGN SECTION PROPERTIES 49
ASSIGN JOINT CONSTRAINTS 50
ADD JOINTS TO AN EXISTING CONSTRAINT 50
DELETE OR REMOVE JOINTS FROM AN EXISTING CONSTRAINT 50
GENERALIZED CONSTRAINTS 50
BODY CONSTRAINT 51
DIAPHRAGM CONSTRAINT 51
PLATE CONSTRAINT 52
ROD CONSTRAINT 52
BEAM CONSTRAINT 53
EQUAL CONSTRAINT 53
PARTIAL CONNECTION 54
LOCAL CONSTRAINT 54
SYMMETRY ABOUT A PLANE 55
ANTI-SYMMETRY ABOUT A PLANE 55
AXISYMMETRY 55
CYCLIC SYMMETRY 56
SYMMETRY ABOUT A POINT 56
ASSIGN JOINT SPRINGS 56
ASSIGN JOINT MASSES 57
ASSIGN JOINT RESTRAINTS 57
ASSIGN JOINT PATTERNS 57
ASSIGN LOCAL AXES 58
ASSIGN LOCAL AXES FOR JOINTS 58
ASSIGN LOCAL AXES FOR FRAME ELEMENTS 58
ASSIGN FRAME RELEASES 59
ASSIGN FRAME END OFFSETS 59
ASSIGN FRAME OUTPUT SEGMENTS 60
ASSIGN FRAME PRESTRESS 60
ASSIGN FRAME INITIAL P-DELTA FORCE 61
ASSIGN FRAME LANES 61
ASSIGN PRESTRESS TO FRAME STATIC LOAD 61
ASSIGN LOCAL AXES FOR SHELL ELEMENTS 61
ASSIGN LOCAL AXIS FOR NLLINKS 62
ASSIGN STATIC LOADS 62
ASSIGN LOADS OR DISPLACEMENTS TO JOINTS 62
ASSIGN GRAVITY LOADS TO FRAMES 63
ASSIGN POINT AND UNIFORM LOADS TO FRAMES 63
ASSIGN TRAPEZOIDAL LOADS TO FRAMES 63
ASSIGN TEMPERATURE LOADS TO FRAMES 64
ASSIGN GRAVITY LOADS TO SHELLS 64
ASSIGN UNIFORM LOADS TO SHELLS 64
ASSIGN PRESSURE LOADS TO SHELLS 64
ASSIGN TEMPERATURE LOADS TO SHELLS 65
ASSIGN GRAVITY LOADS TO NLLINKS 65
ASSIGN GROUP NAMES 65
ANALYZING A MODEL 67
DISPLAY OPTIONS 69
GRAPHICAL OUTPUT 69
DISPLAYING UNDEFORMED GEOMETRY 70
DISPLAY STATIC LOADS 70
DISPLAY JOINT PATTERNS 70
DISPLAY BRIDGE LANES 70
DISPLAY INPUT IN TABULAR FORMAT 71
DISPLAY STATIC DEFORMED SHAPE 71

DISPLAY MODE SHAPE....71
ANIMATE DEFORMED SHAPE....72
DISPLAY MEMBER FORCE OR STRESS DIAGRAM....72
VIEW TIME HISTORY RESULTS....73
DISPLAY INPUT TIME HISTORY FUNCTIONS....73
DISPLAY JOINT OUTPUT TIME HISTORY TRACE....73
DISPLAY FRAME ELEMENT FORCE OUTPUT TIME HISTORY TRACE....74
DISPLAY SHELL ELEMENT STRESS OUTPUT TIME HISTORY TRACE....75
DISPLAY STRUCTURAL ENERGY TIME HISTORY TRACE....76
DISPLAY BASE FUNCTION TIME HISTORY TRACE....77
DISPLAY GROUP SUMMATION FORCES TIME HISTORY....77
F(T) VS. T....78
F(T) VS. F(T)....78
SCALE FACTOR, LINE TYPE AND COLOR....79
ZOOM INTO THE PLOT....79
PRINT TIME HISTORY OR RESPONSE SPECTRUM PLOTS OR TABLES....79
VIEW GENERATED RESPONSE SPECTRA CURVES....79
SPECTRUM GENERATION DEFINE TAB....80
SPECTRUM GENERATION AXES TAB....80
SPECTRUM GENERATION OPTIONS TAB....80
SPECTRUM GENERATION FREQUENCY TAB....80
SPECTRUM GENERATION DAMPING TAB....81
DISPLAY GROUP JOINT FORCE SUMS....81
DISPLAY JOINT INFLUENCE LINES....81
DISPLAY FRAME INFLUENCE LINES....82
DISPLAY JOINT OR MEMBER TEXT OUTPUT ON SCREEN....82
STEEL DESIGN....83
CONCRETE DESIGN....83
GROUP ELEMENTS FOR DESIGN....84
START DESIGN/CHECK OF STRUCTURE....84
SELECTING DESIGN LOAD COMBINATIONS....84
OVERWRITE ELEMENT DESIGN DATA....85
DEFINING ELEMENTS AS A SINGLE MEMBER FOR BENDING....85
REPLACE AUTO WITH OPTIMAL SECTIONS....86
DISPLAY DESIGN INPUT/RESULTS ON SCREEN....86
DISPLAY STEEL STRESS CHECK/DESIGN RATIOS ON SCREEN....86
DISPLAY CONCRETE DESIGN/CHECK OUTPUT ON SCREEN....86
DISPLAY STEEL STRESS CHECK/DESIGN INPUT ON SCREEN....87
DISPLAY CONCRETE DESIGN/CHECK INPUT ON SCREEN....87
REVIEW STRESS CHECK/DESIGN DETAILS....87
REVIEW STEEL STRESS CHECK/DESIGN DETAIL....87
REVIEW CONCRETE DESIGN/CHECK DETAIL....87
VIEW INTERACTION DIAGRAM FOR A CONCRETE COLUMN....88
REDESIGN A STEEL MEMBER....88
REDESIGN A CONCRETE MEMBER....88
UPDATE ANALYSIS SECTIONS....89
RESET DESIGN SECTIONS....89
PREFERENCES....91
SETTING STEEL DESIGN PARAMETERS....91
SETTING CONCRETE DESIGN PARAMETERS....91
SETTING COLORS....92
EDITING TOOLBARS....92
USER DEFINED COORDINATE SYSTEM....92
GENERAL TIPS....93
SAP2000 VERSIONS AND LIMITATIONS....94
ERROR MESSAGES....94

PHONE AND FAX SUPPORT 94
ONLINE SUPPORT 95
FREQUENTLY ASKED QUESTIONS 95
PROGRAM MANUALS 95
TEXT BOOKS 96
REFERENCES 96
END USER LICENSE AGREEMENT FOR CSI SOFTWARE 98
QUESTIONS AND ANSWERS ABOUT THE END USER LICENSE AGREEMENT 100
CAN I SELL MY END USER LICENSE AGREEMENT FOR SAP2000 TO SOMEONE ELSE? 101
CAN I SELL OR GIVE AWAY OLD VERSIONS OF SAP WHEN I ACQUIRE AN UPGRADE? 101
CAN I MAKE A SECOND COPY FOR MY PERSONAL USE? 101
CAN I RENT OR LEASE THE PROGRAM TO SOMEONE ELSE? 101
WHEN I UPGRADE FROM SAP90, DO MY LICENSE RIGHTS FOR SAP2000 CHANGE? 101
IN WHAT WAYS CAN I USE THE SOFTWARE OVER A NETWORK? 101
EIGENVECTORS 103
RITZ 103
RIGID-BODY BEHAVIOR 103
EQUAL-DISPLACEMENT BEHAVIOR 103
SYMMETRY CONDITIONS 103
ANTI-SYMMETRY CONDITIONS 104
VON MISES STRESS 104
ANALYSIS CASES 104
LOAD COMBINATION TYPE 104
LINEAR TIME HISTORY ANALYSIS 105
PERIODIC TIME HISTORY ANALYSIS 105
NONLINEAR TIME HISTORY ANALYSIS 105
GROUND ACCELERATION 105
UBC94 RESPONSE SPECTRUM 105
ENERGY FUNCTIONS 105
BASE FUNCTIONS 106
DXF 106
JOINT STRESS AVERAGING 106
PASTE COORDINATES 106
INSERTION POINT 107
COORDINATE SYSTEM 107
SET COORDINATE SYSTEM 108
COORDINATE SYSTEM (ADVANCED) 108
SHELL ELEMENT (4 JOINTS) 108
SHELL ELEMENT (4 JOINTS 3-D) 109
SHELL ELEMENT (QUICK) 109
FRAME ELEMENT (QUICK) 109
FRAME ELEMENT (CROSS BRACE) 109
FRAME ELEMENT (2 JOINT) 109
FRAME ELEMENT (2 JOINT 3-D) 110
FLOATING TOOLBAR 110
SAP2000 SCREEN 110
DIMENSION TOLERANCES 111
STEEL DESIGN PARAMETERS 111
CONCRETE DESIGN PARAMETERS 112

GENERAL

New Interface

1 SAP2000 is now fully integrated into Windows.
2 Model building, Analysis, Design and Display of results may be carried out in the same window.
3 The model may be viewed in multiple windows (up to 4).
4 Zooming is possible either in steps or with a mouse defined window
5 Members may be extruded on their centerlines.
6 The model may be viewed in Perspective
7 Context sensitive help is available in the "forms" with a right mouse button click.
8 Detailed information about joints and members in the model is also available with a right mouse button click (e.g. bending moment diagrams, joint displacements or connectivity etc.)

Designer's Convention

The designer's sign convention is quite different from the typical structural analysis sign convention described in most text books. The moment which produces compression in the positive face/fiber is called a positive moment and the axial force which produces tension in the member is called a positive axial force

Noun-Verb

All commands in SAP2000 work in the "Noun-Verb" mode i.e. you make a selection first and then perform an action on it. As an example, if you want to delete a member from the model you would first select that member and then press the delete key.

2 Modes of SAP2000

SAP2000 has two very distinct modes i.e. DRAW and SELECT. SELECT mode is the default mode. DISPLAY and ASSIGN functions are carried out in the SELECT mode.

Select Mode

When SAP2000 is not in the DRAW mode, SELECT is the default mode. It is possible to make multiple selections and then perform a task on the selected entities. Display, Assign, Design, Output or even Delete functions are carried out in this mode.

Tell me more about

» Selection Procedures

» Display Options

» Assign Options

Setup Coordinate System

To set up a Coordinate System for creating a new model

1 On the File menu, click New Model .
2 Choose either the Cartesian or the Cylindrical tab.
3 Type in the number of grids and their spacing and then click OK.

To add another Coordinate System

1 On the Options menu, click Set Coordinate System.
2 Click Add new System.
3 Choose either the Cartesian or the Cylindrical tab.
4 Type in System Name, to define the new Coordinate System
5 Type in the number of grids and their spacing.
6 Click on Advanced to specify Location and Orientation.
7 Click Ok, Ok, and OK.

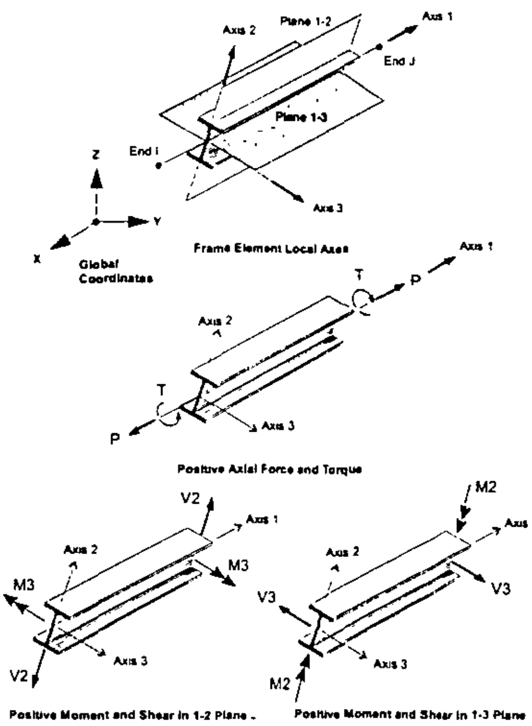
Tip: You can add a new grid by pressing the CTRL key and dragging an existing grid to a new location, or by using the Edit Grid command from the Draw menu Double clicking on an existing grid opens the Edit Grid box where you may add or modify the location of a grid.

Also See

» Edit Grids

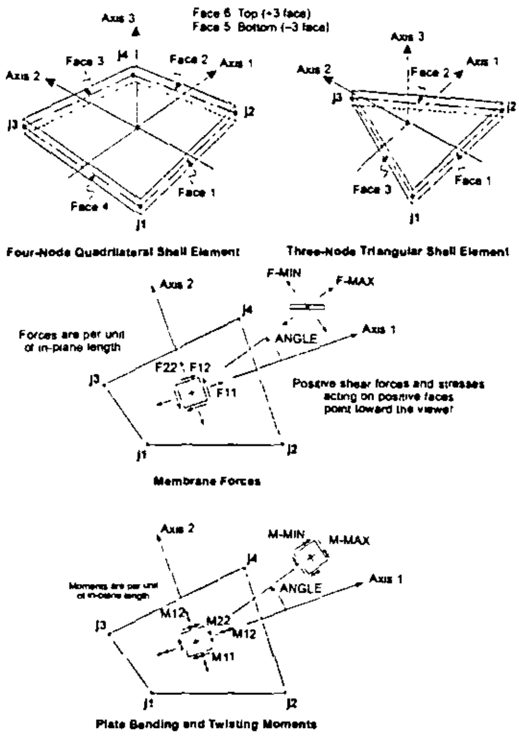
Frame Element

FRAME element follows the designer's convention.



Frame Element Internal Forces

Shell Element



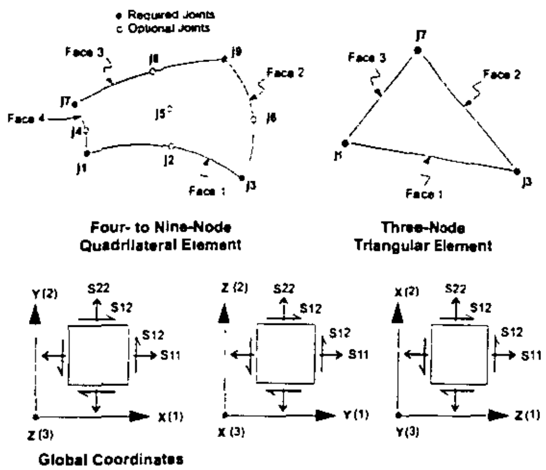
Shell Internal Forces

FINT = F intermediate

FVM =√{1/2[(FMAX - FINT)^2+(FMAX - FMIN)^2+(FINT - FMIN)^2]}.

Also See: Von Mises Stress

Asolid Element



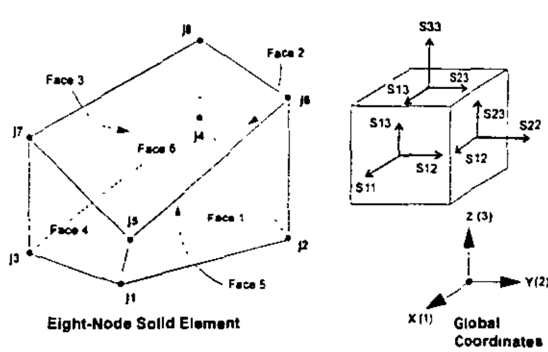
Plane and Asolid Element Stresses

SINT = F intermediate

SVM =√{1/2[(SMAX - SINT)^2+(SMAX - SMIN)^2+(SINT - SMIN)^2]}.

Also See: Von Mises Stress.

Solid Element



Solid Element Stresses

Nonlinear Analysis

Nonlinear analysis capabilities are available through a nonlinear link element (NLLink). This link element is used to model local structural nonlinearities such as gaps, dampers, isolators, and the like. Nonlinear behavior is exhibited only during nonlinear time-history analyses. For all other analyses, this link element behaves linearly.

Each nonlinear link element may be either:

- a one-joint grounded spring, or
- a two-joint link

Properties for either type of element are defined in the same way.

Each element is assumed to be composed of six separate nonlinear springs, one for each of the six deformational degrees-of freedom. The force-deformation relationships of these springs may be coupled or independent of each other.

The types of nonlinear behavior that can be modeled with the link element include.

1. Viscoelastic damping
2. Gap (compression only) and hook (tension only)
3. Uniaxial plasticity
4. Biaxial-plasticity base isolator
5. Friction-pendulum base isolator

Each element has its own local coordinate system for defining the force-deformation properties and for interpreting output.

Each NLLink element may be loaded by gravity (in any direction).

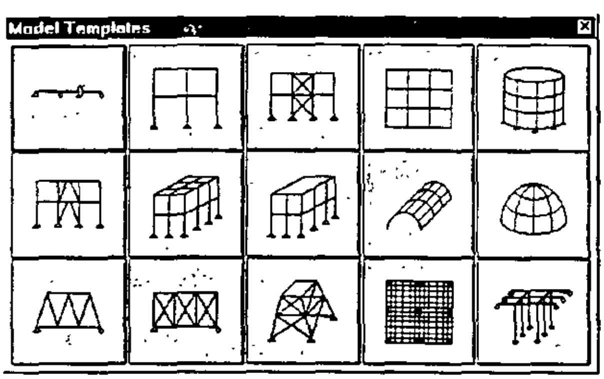
Available output includes the deformation across the element, and the internal forces at the joints of the element.

FILE MENU

Getting a Head Start With Templates

New models may be created with very little effort using pre-programmed templates.

Click on any of the template buttons to learn more about them.



To Base a new model on a template:

1. On the File menu, click New Model from Template. This will display the possible templates shown above.
2. Click on a template most appropriate for your model.
3. Choose or change the default values in the edit boxes.
4. Click OK.

To add one or more templates to the existing model:

1. On the Edit menu, click Add to Model From Template... This will display the template options similar to the one shown above
2. Click on the template you want.
3. Fill in the parameters in the edit boxes and then click on Advanced to specify an insertion point.
4. Click OK.

Note: Remember to save your model periodically. Click on .

Import/Export Capabilities

Import and Export are available through the File pull down menu. SAP2000 now works around a central database created for the model The database is maintained in a binary format and does not operate with text input files. However, text input files may be read or written using the Import Export facility.

What do you want to do?

» Import a SAP90 text input file
» Import a SAP2000 text input file
» Export a SAP2000 text input file
» Import a DXF file
» Export a DXF file
» Import a SAP2000 Job Run in DOS mode
» Export an AVI file

Import a SAP90 text input file

1. On the File menu, click Import and then SAP90. This will display the Open SAP90 Data File dialog box
2. Choose the file from the list box (extension not allowed).
3. Click Open

Import a SAP2000 text input file

1. On the File menu, click Import and then SAP2000.S2K. This will display the Open SAP2000 Data File dialog box.
2. Choose the file from the list box (must have either .S2K or .$2K extension).
3. Click Open.

Caution: The text file with an extension . $2k is automatically written out whenever the model is saved This file is good for analysis purposes only or recovering the model in emergency crash situations and is not a substitute for the database file (.SDB). It does not contain all the information about the model.

Note: The analysis and design information provided in the SDB file is also saved in the S2K file. This gives the user the ability to modify the model using a text editor like WordPad. To find the format used in the S2K file for the design data. Create a frame element in SAP2000 with the design information that you desire. Export the model and open it in a text editor to see the format used for the defined design information in the S2K file.

Export a SAP2000 text input file

1. On the File menu, click Export and then SAP2000.S2K. This will display the Save Model File As dialog box.
2. Type in the file name in the File name edit box.
3. Click Save This will save the text input file with an extension *filename*.S2K.

Note: A text file with an extension *filename* $2k is also saved at the same time as the model database is saved (as a file with .SDB extension). This text file is good for analysis purposes only or for recovering the model in emergency crash situations and is not a substitute for the database file. It does not contain all the information about the model.

Import a DXF file

1 On the File menu, click Import and then .DXF. This will display the Import DXF File dialog box.
2 Choose the file from the list box. Frame elements must be *Lines* on a layer named SAP_FRAMES and shell elements must be *3D Faces* on a layer named SAP_SHELLS. Nodes will be entered automatically by SAP2000.
3 Click Open.

Note: The DXF file import is intended to facilitate importing geometry from AutoCAD. Point, Line and Area (Joint, Frame and Shell elements) are the only entities imported currently. DXF for AutoCAD r12, r13 and r14 are supported.

Export a DXF file

1 If you want to export a limited number of elements then select the elements otherwise go directly to step 2.
2 On the File menu, click Export and then .DXF. This will display the Save AutoCAD DXF File As dialog box.
3 Type in the file name in the File name edit box.
4 Click Save. This will save the DXF file, which may be imported in AutoCAD. Frame elements will be *Lines* on a layer named SAP_FRAMES and shell elements will be *3D faces* on a layer named SAP_SHELLS.

Note: The DXF file export is intended to facilitate exporting geometry from SAP2000. Point, Line and Area (Joint, Frame and Shell elements) are the only entities exported currently.

Hint: If you want to only export a 2D plain of the model then set the 2D view that you are interested in Select all the elements that are visible, and then select Export... DXF.

Import a SAP2000 Job Run in DOS mode

1 On the File menu, click Import and then SAP2000.JOB. This will display the Open SAP2000 File dialog box.
2 Choose the file from the list box (with extension .JOB).
3 Click Open.

Export an AVI file

1 On the File menu, click Create History Video... This will display the Time History Video File Creation dialog box.
2 Click on AVI File Name button to change the AVI file name or accept the default name.
   - In the Time History Data area:
     - Select the History Name from the drop down list box.
     - Specify the Start and End Time and the Time Increment in the edit boxes.
   - In the Display Options area·
     - Click to check if Wire Shadow is to be retained in the Video file capture.
     - Click to check if Cubic Curve is to be used.
     - Type in the Magnification Factor in the edit box.
   - In the AVI Options area:
     - Type in the animation speed in the Frames per Second edit box or accept the default value.
     - Specify the Frame size in Pixels in the edit boxes or accept the default values.

4 Click OK

Note: The AVI file may be played back using Windows Media Player.

Tip: To play the AVI file in **real time animation** make sure that the number of frames per second is equal to 1/Time Increment. The value of number of frames per second may be adjusted to speed up or down the animation

View Real Time Animation for Time History Results

1 On the File menu, click Create History Video... This will display the Video File dialog box.
2 Type in a file name or accept the default name.
3 Click Save. This will display the Time History Video File Creation dialog box.
4 Select the Time History case from the History Name drop down box.
5 Enter the Start Time, End Time and Time Increment in the edit boxes.
6 Checking the Wire Shadow box will also display the undeformed shape as a reference.
7 Checking the Cubic Curve box will display the deformed elements with cubic curve fit.
8 Changing the Magnification Factor will scale the vector components. The default factor is 10.
9 Enter Frame Speed in frames per second for animation and the Frame Size in Pixels. The default for Frame Speed is 10 and Frame Size is 320x240.
10 Click OK. Creating AVI File window will appear and the animation will be saved as an AVI file which may be played back through the Windows Media Player.

Note: If you want to view real time animation then the value for frames per second should be the same as 1/Time Increment. A larger value will speed up the animation and a smaller value will slow it down.

Print Setup

Print Setup is where you can set number of lines of text to be printed per page, project information and notes to be printed with text and graphics output. You can also do regular printer setup.

1 From the File menu select Print Setup.
2 In the Print Page Setup dialog box:
   - Check the No Page Eject if you do not want SAP2000 to break the printed text into pages and place page headings.
   - If you want to have SAP2000 break the printed text into pages and place page headings then:
     - Select Default to use the default number of lines per page
     - Or select User Defined and enter in the provided text input box the number of lines per page.
   - Enter under the Titles area, Project and other comments in the provided text input boxes.
   - Click the Setup button to set printer options. (See windows documentation for instructions on using this dialog box.)
   - Click OK on the Print Page Setup dialog box when finished.

Print Selected Graphical Output to a Printer or a File

1 Plot the desired deformed or undeformed shape on the Screen.
2 On the File menu, click Print Graphics...
3 The plot will be printed to the default printer. If the default Printer has been set to print to a file then the file dialog box will be displayed.

Note: If more than one Window is open the highlighted window will be printed.

Text Output

Text output for a Joint or a member for the chosen Load Case(s) and/or Load Combination(s) may be either viewed on the screen or printed to a printer or a file.

What do you want to do?

»| Display Joint or Member Text Output on Screen

»| Print Output to a Printer or a File

Print Input Tables to a Printer or a File

1 Select the members or Joints for which the input data is desired. If no members are selected the input data is generated for all the members in the model.
2 On the File menu, click Print Input Tables... This will display the Print Input Tables dialog box.
3 In the Print Input Tables dialog box:
   - Check the information you want printed
   - If you want to generated output for all the members in the model even though some members were selected then remove the check for Selection Only check box.
   - If Output to a File is desired check the Print to File check box.
      - If you want the output appended to an existing file check the Append check box.
   - Click OK.

Print Output to a Printer or a File

1 Select the members or Joints for which the output is desired. If no members are selected the output is generated for all the members in the model.
2 On the File menu, click Print Output Tables... This will display the Print Output Tables dialog box.
3 In the Print Output Tables dialog box:
   - Click on Select Loads to select Load Cases or Combos.
   - Select the Load Cases and/or the Load Combinations labels for which the output is to be displayed in a tabulated form. Loads may be selected by clicking on the Load labels in the list box: For selecting a range of loads, click and drag, or for multiple loads hold down the Ctrl Key and click on different load labels.
   - Click OK to close the Select Output dialog box.
   - Click on the check boxes to select the Type of Analysis Results desired, e.g. Displacements, Reactions, Spring Forces, Frame Forces, NLLink Forces, Shell Resultants, Shell Stresses, Plane Stresses, ASOLID Stresses and Solid Stresses, Element Nodal Forces, Group Forces.
      - If Group Forces is checked, you can use the Select Group button that appears to select predefined Groups for which you want forces reported.
   - If Output to a File is desired check the Print to File check box.
      - If Spreadsheet output format is desired check the Spreadsheet check box.
      - If you want the output appended to an existing file check the Append check box.
   - If only Envelopes of Load Combos are desired check the Envelopes Only check box.
   - If you want to generated output for all the members in the model even though some members are selected then remove the check for Selection Only check box.
   - Click OK.

Note: You can not select Envelope results in a Spreadsheet format.

Print Design Results to a Printer or a File

1 Select the members or Joints for which the design results are desired. If no members are selected the design results are generated for all the designed members in the model.
2 On the File menu, click Print Design Tables... This will display the Print Design Tables dialog box.
3 In the Print Design Tables dialog box:
   - Check the information you want printed for steel and concrete members.

- If you want to generated output for all the members in the model even though some members were selected then remove the check for Selection Only check box.
- If Output to a File is desired check the Print to File check box.
    - If you want the output appended to an existing file check the Append check box.
- Click OK.

User Comments and Session Log

SAP2000 provides the user with a way to keep notes with the model file.

1. This is done by selecting from the File menu, User Comments and Session Log.
    - This will bring up a text editor that works just like Notepad.

Uses for the User Comments and Session Log Text Editor:

- Keeping track of what has been done on the model to date.
- Keeping a To Do list for the model
- Retaining key results to see the effect of changes on the model
- Paste a Cut or Copied model into the text editor. Modifying the information, Cut/Copying it and then Pasting the modified model back into SAP2000.
- The Log file automatically provides a log of all file saves by the user.

EDIT MENU

Undo and Redo Capabilities

SAP2000 allows you to go back one step at a time in an editing session. Therefore, it is possible to Undo a series of actions previously performed. If you go too far in the Undo process you may Redo those actions.

1 Click to **Undo** (in steps) the most recent actions performed. **Undo** is also available in the Edit menu.

If you later decide you didn't want to undo an action

1 Click to **Redo** the action(s). **Redo** is also available in the Edit menu.

Note: Undo only works on Objects which have been moved, deleted or added. Once a file is saved Undo and Redo actions are no longer available.

Cut, Copy and Paste

1 SAP2000 now supports standard Windows Cut, Copy and Paste commands.
2 The entire structure or any selected part may either be cut or copied and then pasted back into the model area at any user specified location.
3 It is also possible to Paste the Cut or Copied model into Microsoft Excel, modify, Cut/Copy in Excel and then Paste the modified model back in the SAP2000 window.

Also see:

Replicate

Deleting Members

1 You can delete members by selecting them and pressing the Delete button on your keyboard.
2 You can also delete members by selecting them and clicking on Delete From the Edit menu.

Move

The move feature is a way to select parts of the structure and relocate them in the model.

1 Select the members and joints you want to move.
2 On the Edit menu, click Move. This will display the Move dialog box.
   - Enter the relative movement of the selected members in the global X, Y and Z direction.
   - Click OK to move the selected elements.

Replicate

Replicate is another very powerful way of generating a large model from a small model when the elements and/or joints form a linear or radial pattern or are symmetrical about a plane. When joints or elements are replicated the assignments on those joints and elements are also replicated, e.g. member section assignments, member loads, joint loads and joint restraints This is a major benefit of using Replicate over Cut, Copy and Paste, which only operates on lines and joints.

What do you want to do?

» Replicate in a Linear Array

» Replicate in a Radial Array

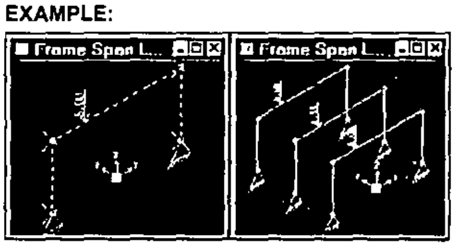
» Replicate in a Radial Array by Shifting the Origin

» Replicate by Using the Mirroring Option

Replicate in a Linear Array

1 Select the members and joints you want to replicate.
2 On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
3 Click on the Linear Tab.
4 Fill in the X, Y and Z offset distances in the Distance edit boxes.
5 Type in the number of times you want the selected entities replicated in the Number edit box.
6 Click OK.

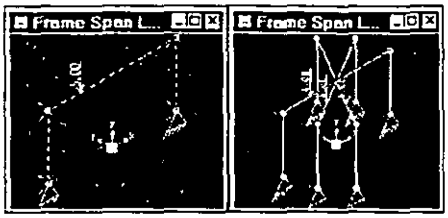
EXAMPLE:



Replicate in a Radial Array

1 Select the members and joints you want to replicate.
2 On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
3 Click on the Radial Tab.
4 Check the axis about which the selected entities are to be rotated, i.e. X, Y or Z in the Rotate About section.
5 Type in the increment angle and the number of times you want the selected entities replicated in the Increment Data box.
6 Click OK.

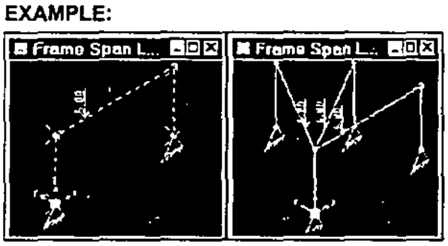
EXAMPLE:



Replicate in a Radial Array by Shifting the Origin

1 On the Options menu, click Set Coordinate System... This will display the Coordinate Systems dialog box.
2 Select the coordinate system from the Systems list box.
3 Click on Modify/Show System This will display the Location and Orientation dialog box.
   - In the Location and Orientation dialog box
      - Type in the values by which you need to shift the coordinate system in Translations edit boxes and/or Rotations in Degrees edit boxes.
      - Click OK
4 Click OK
5 Select the members and joints you want to replicate.
6 On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
7 Click on the Radial Tab.
8 Check the axis about which the selected entities are to be rotated, i.e. X, Y or Z in the Rotate About section.
9 Type in the increment angle and the number of times you want the selected entities replicated in the Increment Data box.
10 Click OK.

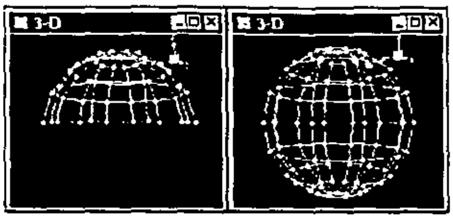
EXAMPLE:



Replicate by Using the Mirroring Option

1 Select the members and joints you want to replicate.
2 On the Edit menu, click Replicate. This will display the Replicate dialog box with the Linear, Radial and Mirror Tabs.
3 Click on the Mirror Tab.
4 Check the plane about which the selected entities are to be rotated, i.e. XY, YZ or XZ in the Mirror About section.
5 Type in the ordinate by which you want to shift the mirrored replication in the Ordinate section.
6 Click OK.

EXAMPLE:



Merge Joints

Joints within the default Auto Merge Tolerance are automatically merged. Those Joints which fall outside this tolerance may be merged as follows:

1. Select the Joints you want merged.
2. On the Edit menu click Merge Joints... This will display the Merge Selected Joints dialog box.
3. Accept or change the Merge Tolerance.
4. Click Ok

Hints: This is a very useful command for merging two separate models together.

If elements are drawn with Snaps turned off there may be extraneous joints at common locations. These extraneous Joints may be easily eliminated by this command.

Models developed in CAD programs may have beam joints drawn 6 inches or so away from column joints. This is a powerful option to merge beam joints to column joints in such models imported from CAD.

Automatic Meshing

It is not necessary to define Joints prior to defining elements. Joints are automatically created when elements are drawn Typically, draw the boundary of your structure and then use the following meshing techniques to create a detailed model.

What do you want to do?

- Divide or Break Frames
- Mesh Shells
- Join Frames
- Merge Joints
- Disconnect
- Connect
- Show Duplicates

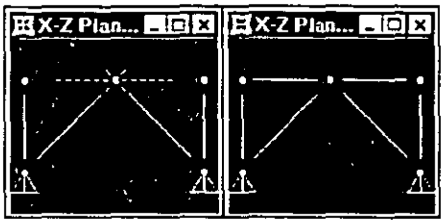
Divide or Break Frames

To Divide

1. Select the Frame Elements you want divided.
2. On the Edit menu click Divide Frames... This will display the Divide Selected Frames dialog box.
3. Select Divide into option.
4. Type in the number of elements the Frame Elements should be divided into.
5. Type in the ratio of the last element to the first, if the division is not to be in equal lengths.
6. Click OK.

To Break

1 Select the Frame Elements you want to break into multiple elements and also select the intersecting Elements or Joints.
2 On the Edit menu click Divide Frames... This will display the Divide Selected Frames dialog box.
3 Select Break at intersections with selected Frames and Joints option.
4 Click Ok

EXAMPLE: 

Mesh Shells

To Mesh

1 Select the Shell Elements you want meshed.
2 On the Edit menu click Mesh Shells. This will display the Mesh Selected Shells dialog box.
3 Select Mesh into option.
4 Type in the number of elements (in both directions) the Shell Elements should be meshed into.
5 Click OK.

To Mesh using selected Joints on edges

1 Select the Shell Elements you want meshed into multiple elements and also select the Joints on the edges of the Shell Elements.
2 On the Edit menu click Mesh Shells This will display the Mesh Selected Shells dialog box.
3 Select Mesh using selected Joints on edges option.
4 Click Ok

To Mesh using grid intersections

1 Select the Shell Elements you want meshed into multiple elements.
2 On the Edit menu click Mesh Shells This will display the Mesh Selected Shells dialog box.
3 Select Mesh at intersection with grids.
4 Click Ok

Note: In order for the mesh using selected Joints on edges to work you have to have an equal number of joints on the opposite edges. Also, if both sides are not divided this option will not work.

Join Frames

1 Select the Frame Elements you want joined.
2 On the Edit menu click Join Frames.
3 This will join the selected Frame Elements into a single element and remove unused joints left over from the joining process.

Disconnect

All elements connected to each other normally share a common Joint. Disconnect will break off the Elements from the Joint and will add duplicate Joints to each of those Elements. To use Disconnect

1 Select the Joints you want disconnected from the Elements.
2 On the Edit menu click Disconnect.

Hints: This is a very useful command for some special modeling conditions.

This feature adds separate joints to Elements sharing common Joints. After the Joint is disconnected special Constraints may be added to those Joints. e.g. You could use this feature to specify moment release Constraints between two adjacent edges of Shell elements or if you need to add a zero length non-linear link you would need separate Joints at the same location.

Connect

Connect facilitates re-connecting all selected elements which had been disconnected from each other and were connected to their own independent Joints. When Elements are connected back to each other the independent Joints are collapsed to a common joint which is shared by all the combined Elements. To use Connect

1 Select the Elements you want connected to each other.
2 On the Edit menu click Connect.

Show Duplicates

This is a useful command to select duplicate Joints, Frames, Shells, ASOLID and Solids from the entire structure. If the duplicates are unnecessary they may be deleted or merged.

1 Select the Joints and or Elements.
2 On the Edit menu click Show Duplicates.
3 Duplicate Joints and Elements will be re-drawn with a different color.

Joint and Element Labels

SAP2000 automatically labels JOINTS, FRAME and SHELL elements. However, it is possible to initialize a labeling scheme or to change labels after they have been assigned.

What do you want to do?

Initialize a New Labeling Scheme

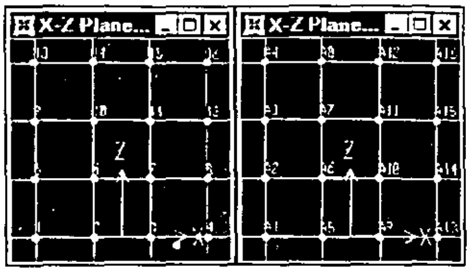
Re-Label Previously Assigned Labels

Re-Label Previously Assigned Labels

The labels in SAP2000 are alphanumeric. It is possible to select some Joints and Elements and change their labels by assigning them another labeling scheme.

To Change Labels:

1 Select the Joints and Elements (FRAME and SHELL) for which you want to change the labels.
2 On the Edit menu, click Change Labels.. This will display the Re-Label Selected Items dialog box.
3 In the Re-Label Selected Items dialog box
   - Enter an alphanumeric prefix, the starting numeric number for the scheme to be added to the prefix and a numeric increment for the numeric sequence for Joints, Frames and Shells
   - Click on the Joint, Frame and/or Shell check boxes to further select which type of members within the total selection will be affected by the label change.
   - Select the first and second Re-Label order from the drop down boxes. See example below. The labels have been changed to add an alpha prefix and have been sorted first by X and then by Z.
4 Click OK.

EXAMPLE: 

VIEW MENU

Select 3-D Views

It is sometimes helpful to view the model in 3-D views. It is possible to chose a quick 3D view by clicking on the 3-d button in the Toolbar.

Setting 3-D View

1 On the View menu, click Set 3D View... This opens the 3D View dialog box.
2 Click the appropriate Fast View button.
3 Use the up/down scroll buttons or enter values in the text edit box for the Plan, Elevation and Aperture.
4 Click OK when finished.

Note: You can view the model orientation in the View window in the top left corner of the 3D View dialog box .

Tips: You can save time by saving 3D views by assigning them names and then recalling them later. Views may be saved through the View menu under Save Named Views.

Also See

» Select 2D Views

Select 2-D Views

It is sometimes easier to build a model while working in 2-D elevation or plan views.

Select an X-Y, X-Z or Y-Z Plane

1 On the View menu, click Set 2D View... This opens the 2D View dialog box
2 Select the X-Y, X-Z or Y-Z plane.
3 Fill in the edit box with the appropriate Z, Y or X coordinate or click on a relevant plane within the window showing an outline of the model.
4 Click OK.

Note: It is also possible to choose a plane by clicking on one of the 2D view buttons xy xz yz and then using the up or down arrow buttons to move to an appropriate plane.

You can use rubber band zoom to zoom into the window showing the outline of the model. Clicking once anywhere in the window will restore the original view

Tips: You can save time by saving 2D views by assigning them names and then recalling them later. Views may be saved through the View menu under Save Named Views.

Also See

Select 3D Views

Set Elements

Setting Elements allows you to selectively display various options associated with the elements. Use this method to selectively display various element types, associated element ID numbers and element section property types, labels and extrusions. You can also suppress element, element fills or shrink the elements about their corresponding centroids to enhance the clarity of the display.

1 On the View menu, click Set Elements or press the button on the toolbar. This will bring up the Set Elements dialog box.

2 In the Set Elements dialog box:

- Check the options to be activated using the check boxes. Check the Hide under each element's category if you don't want any of the elements and its options visible.
- Check Shrink Elements if you want the elements shrunk in size to improve viewing of the model or to check element connectivity. You can also do this from outside the Set Elements dialog box by pressing the button on the toolbar.
- Check Show Extrusions if you want to see the frame element section shape on the screen. This can help in seeing if the member is oriented properly.
- Check Fill Elements to see the elements shaded in as opposed to transparent.
- Check show edges to see the outline of the elements.
- Click the OK button to update the active screen.

Note: You Set Elements for individual windows not for the full model. So you can use this method to view different information in each window.

Set Limits

Use Set Limits menu when you do not want the whole structure to appear in the display. This option allows you to selectively display portions of the structure by defining limits on the X, Y and Z axes viewing range.

1 On the View menu, click Set Limits. This will bring up the Set Limits dialog box.

2 In the Set Limits dialog box.

Method 1

- Click on each of the XY, YZ and XZ planes.
- Using the mouse, draw a box around the area you want to view in the viewing screen that appears in the top left corner of the dialog box.
- Note that the limits of the box you define will be reflected in the Max and Min range for the corresponding Axis Limits at the bottom of the screen

Method 2

- Alternately directly enter in the Max and Min Axis Limits for the X, Y and Z axes.
- Click the Show All button to reset the full view for the corresponding axis.
- Checking the Ignore Limit Settings allows you to view the full structure without resetting the limits.
- Click OK.

Note: You can also use from the View menu the Show Selection Only option to view only the members that you have selected. Use the Show All option to go back viewing all the elements in the model.

DEFINE MENU

Define Materials

1 On the Define menu, click Materials. . This will display the Materials dialog box with default materials, CONC, OTHER and STEEL listed in the Materials list box.

What do you want to do?

Add a New Steel Material Type

Add a New Concrete Material Type

Add a New Other Material Type

Add a New Steel Material Type

To Add a New Steel Material:

1 In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.

2 In the Material Property Data box

- Choose Steel from the Design Type drop down list.
- Enter the Material name in the Material Name edit box or accept the default name.
- In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume, Modulus of Elasticity, Poisson's Ratio and Co-efficient of thermal expansion, if the default values are not acceptable.
- In the Design Property area, enter the value for Steel yield stress, fy or accept the default value.
- Click OK.

3 The new Material Name will be added to the Materials list box.

4 To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.

5 Click OK.

Add a New Concrete Material Type

To Add a New Concrete Material:

1 In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.

2 In the Material Property Data box

- Choose Concrete from the Design Type drop down list.

- Enter the Material name in the Material Name edit box or accept the default name.
- In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume, Modulus of Elasticity, Poisson's Ratio and Co-efficient of thermal expansion, if the default values are not acceptable.
- In the Design Property area, enter the value for Reinforcing yield stress (fy), Concrete strength (fc), Shear steel yield stress (fys) and Concrete shear strength (fcs) or accept the default values.
- Click OK.

3 The new Material Name will be added to the Materials list box

4 To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.

5 Click OK.

Add a New Other Material Type

To Add a New Other Material:

1 In the Materials dialog box, click on Add new Material button. This will display the Material Property Data box.

2 In the Material Property Data box

- Choose Other from the Design Type drop down list
- Enter the Material name in the Material Name edit box or accept the default name.
- In the Analysis Property Data area, enter new values for Mass per unit volume, Weight per unit volume, Modulus of Elasticity, Poisson's Ratio and Co-efficient of thermal expansion, if the default values are not acceptable.
- Click OK.

3 The new Material Name will be added to the Materials list box.

4 To Delete or Modify/Show Material, click on the material in the list box and then click on the Modify/Show Material or Delete Material button.

5 Click OK.

Define Section Properties

Section properties may be defined at any time before they are assigned to elements.

What do you want to do?

Import a Frame Section from the Section property file

Add a Frame Section by defining the dimensions and or properties manually

Add a Nonprismatic Frame Section

Add an Auto Select Frame Group

Add a Shell Section by defining the dimensions and/or properties

Add an NLLink Property

Define Frame Section Properties

Section properties may be defined at any time before they are assigned to elements. The define frame section dialog box is composed of a list of defined sections, a drop down list of sections that can be imported and a drop down list of sections that can be added by defining their dimensions There are also two buttons, one to Modify/Show Sections and one to Delete sections.

What do you want to do?

Import a Frame Section from the Section property file

»| Add a Frame Section by defining the dimensions and or properties manually

»| Add a Nonprismatic Frame Section

»| Add an Auto Select Frame Group

Import a Frame Section from the Section property file (e.g. Sections.pro)

1 On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
2 In the Frame Sections dialog box click on the Import drop down button and choose importing I/Wide Flange or Channel, Tee, Angle etc. This will display the Section Property File selection box.
3 Choose the file name from the Section Property File selection box.
4 Click Open. This will display a multiple selection list box.
5 Select one or more sections from the list box:
   - Clicking on a section name will select only that section.
   - Clicking on a section name and dragging the mouse to other sections will select a range of sections
   - Pressing the Ctrl key and clicking on different sections will select all of those sections.
6 Click OK. This will display a Section Property form that shows the physical shape and dimensions of the section.
7 The list of selected sections will appear in the Section Name drop down list box.
8 You can continue to load other sections by pressing the Import button which will display the multiple selection list box.
9 It is possible to look at the section properties by clicking on Section Properties.
10 You can modify the section properties by pressing the Modification Factors button. There you can change the default factor of 1 that the section properties are multiplied by.
11 Click OK to accept all the selected sections.
12 The new section names will be added to the Name list box.
13 To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.
14 Click OK.

Add a Frame Section by defining the dimensions and or properties manually

1 On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
2 In the Frame Sections dialog box click on the Add drop down button and choose Add I/Wide Flange or Channel, Tee, Angle etc. by double clicking on one of the section types. This will display a Section Property form that shows the physical shape and dimensions of the section.
3 Enter the section name or accept the default name.
4 Enter the physical dimensions of the section or accept the default values.
5 Choose the material type (e.g. Steel, Concrete or Other) from the Material drop down list box. For Concrete sections see below.
6 It is possible to look at the section properties by clicking on Section Properties.
7 You can modify the section properties by pressing the Modification Factors button. There you can change the default factor of 1 that the section properties are multiplied by.
8 Click OK to accept the selected section.
9 The new section name will be added to the Name list box.
10 To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button.
11 Click OK.

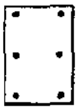
Concrete Section

For concrete sections you will need to also perform the following steps:

1 When you select a Concrete material type, an additional Reinforcement button will appear at the bottom of the form. Click the Reinforcement button.

- In the Reinforcement Data dialog box, select Element Class e.g. Column or Beam.

  For Columns
  - Select the Configuration of Reinforcement e.g. Rectangular or Circular.
  - Enter the Concrete Cover in the text edit box.
  - Enter the Number of Bars in the 3-Direction in the text edit box.
  - Enter the Number of Bars in the 2-Direction in the text edit box.



Example of Concrete Column
3 bars in the 3-Direction
2 bars in the 2-Direction

  - If Circular Reinforcement Configuration is selected then enter total number of bars.
  - Select and Enter the Area of 1 Bar or
  - Select Design Area of Steel for SAP2000 to automatically find the area of steel required during design mode.

  For Beams
  - Enter the Top and Bottom Concrete Cover in the text edit boxes.
  - If you want to specify top and bottom steel, enter reinforcement area for the section in the appropriate text edit box. Otherwise leave values of zero for SAP2000 to calculate the reinforcement automatically.
- Click OK to return to the Section Definition form.

Add a Nonprismatic Frame Section

1 On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.

2 In the Frame Sections dialog box click on the Add drop down button and choose Add Nonprismatic. This will display the Nonprismatic Section Definitions form.

Note: There must be at least two frame sections already defined for the Add Nonprismatic option to show in the add dropdown list box.

3 Enter a name for the Nonprismatic section if the default name is not acceptable.

4 Select from the drop down list box of already defined frame sections, the Start Section and End Section for the Nonprismatic member. These can both be the same section.

5 Enter the length of the section and Absolute Length Type or enter the Length ratio and Variable Length Type.

6 Select the EI variation in the 2-2 and 3-3 directions.

7 Press the Add button to add the section to the list.

8 Use the Insert, Modify and Delete buttons to edit the Nonprismatic section.

9 Press the OK button when finished.

Note: You can add more than one pair of frame sections to build the Nonprismatic section. By entering Absolute lengths (defined in the current units) you can define sections that are of fixed length. Then the sections defined as Variable Length Type will be extended between the fixed sections. The Variable Length value is the ratio of the length of the section relative to all the variable length sections defining the Nonprismatic member. E.g. if there are two Variable Length sections with lengths of 1 and 2, the section with length 1 will be 1/3 of the total variable length, and the section with a length of 2 will be 2/3 of the total variable length.

This method of defining Nonprismatic sections can be used to define sections that have top and bottom fixed length Nonprismatic sections with a prismatic section defined between them, like a bridge pier, where the prismatic part of the section can "telescope" to fit the space between the fixed ends.

Add an Auto Select Frame Group

Auto Select section is a list of frame sections that are used for doing steel optimization design. SAP2000 will use the average properties of all the sections in the group for the first analysis run. Then selecting Start Design/Check of Structure will design each frame member assigned to an Auto Select group with the lightest section in the Auto Select group that satisfies all the stress checks. Then selecting Update Analysis Sections will make the selected section from the Auto Select group the new analysis section.

1. On the Define menu, click Frame Sections... This will display the Frame Sections dialog box.
2. In the Frame Sections dialog box click on the Add drop down button and choose Add Auto Select. This will display the Auto Selection Sections form.
3. Enter a name for the Auto Select section if the default name is not acceptable.
4. Select and Add sections from the List of Sections to the Auto Selections List.
5. Click OK

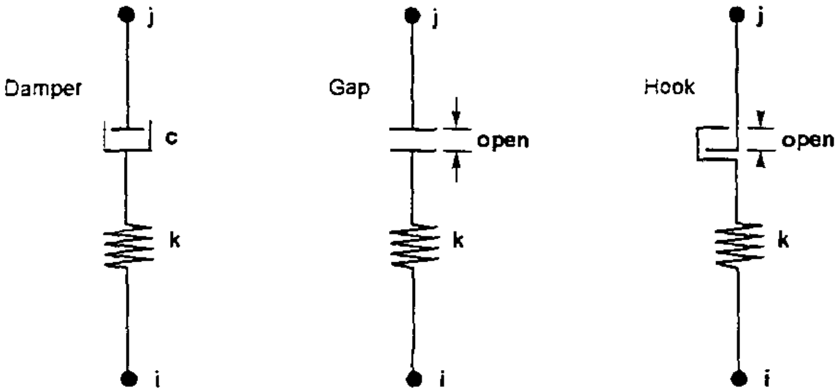
Add a Shell Section by defining the dimensions and / or properties

1. On the Define menu, click Shell Sections... This will display the Shell Sections dialog box.
2. In the Shell Sections dialog box click on the Add New Section button. This will display Shell Sections Data dialog box.
3. In the Shell Sections Data dialog box
   - Enter the Section Name or accept the default name.
   - Select the material type by clicking on the Material drop down box.
   - Enter the Membrane and Bending Thickness.
   - Choose the shell type to be Shell, Membrane or Plate.
   - Click OK.
4. The new section name will be added to the Shell Sections list box.
5. To Delete or Modify/Show Section, click on the Section name in the list box to select it and then click on the Modify/Show Section or Delete Section button
6. Click OK.

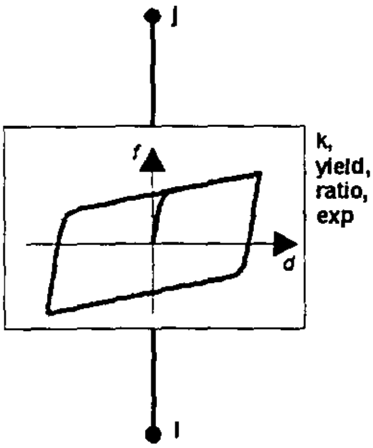
Add an NLLink Property

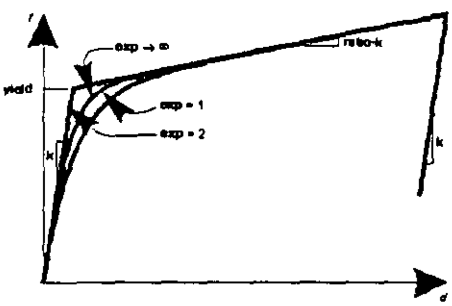
1. On the Define menu, click NLLink Properties... This will display the NLLink Properties dialog box.
2. In the NLLink Properties dialog box click on the Add New Property button. This will display NLLink Property Data dialog box.
3. In the NLLink Property Data dialog box
   - Enter the Property Name or accept the default name.
   - Select the Property type to be Damper , Gap , Hook , Plastic1 , Isolator1 or Isolator2 by clicking on the Type drop down box.
   - Enter the Mass, Weight and Rotational Inertias in the edit boxes.
   - Choose the appropriate directions (U1,U2,U3,R1,R2,R3) for this property by clicking the check boxes and also choose whether it is nonlinear or not.
   - Directional properties such as Stiffness, Damping, Yield Strength, Post Yield Stiffness Ratio and exponents etc. may be specified by clicking on the Modify/Show Properties button.
   - Click OK.
4. The new section name will be added to the NLLink Props list box.
5. To Delete or Modify/Show Property, click on the Property name in the list box to select it and then click on the Modify/Show Property or Delete Property button.
6. Click OK.

NL Damper, Gap and Hook Properties

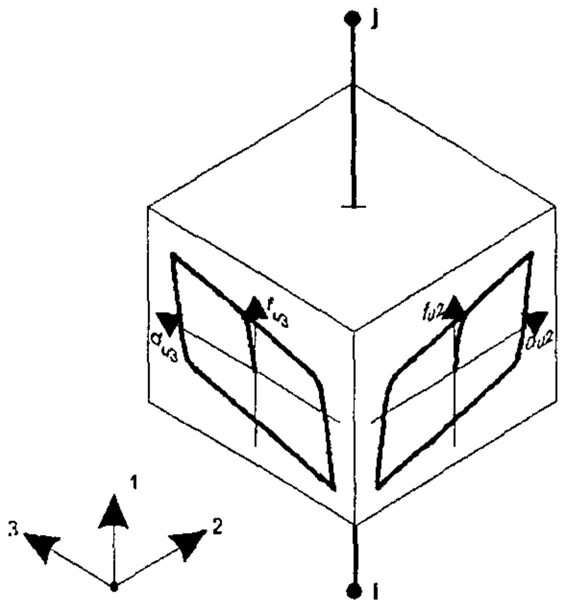


NL Plastic 1 Properties

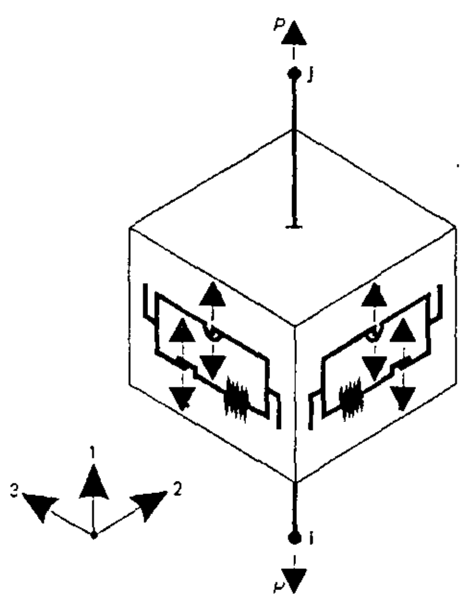




NL Isolator 1 Properties



NL Isolator 2 Properties



Define Group Names

Defining Groups is a powerful tool in SAP2000. It helps in selecting elements, displaying and printing results as well as helping with design.

1 On the Define menu, click Groups... This will display the Define Groups dialog box with default Group ALL listed in the Groups list box.
2 To add a new Group, type in the new name in the Groups edit box and then click the Add new Group Name button.

3 To Change or Delete a group name, click on the Group name in the list box to select it and then click on the Change Group Name or Delete Group Name button.

3 Click OK.

Also see:

» Assign Group Names

Define Static Load Cases

1 On the Define menu, click Static Load Cases... This will display the Static Load Case Names dialog box with a default Load Case, LOAD1 listed in the Loads list box

2 To add a new Load Case name

- Type in the new name in the Load edit box
- Choose the load type to be Dead, Live, Quake, Wind, Snow or Other from the Type drop down list
- Type in a self-weight multiplier in the edit box and
- Click Add new Load button.

3 To change a Load Case name click on the name in the list box and then type over the changes in the edit boxes or choose a different Load Type from the drop down list and click Change Load button.

4 To delete a Load Case name click on the name in the list box and click Delete Load button.

5 Click OK.

Tip: Load Case names defined here are used in Load Combinations. Load Case Types are used by SAP2000 to develop automatic load combinations for Design/Stress Check of the structure.

Bridge Analysis

The bridge analysis can be used to compute influence lines for traffic on bridge structures and to analyze these structures for the response due to vehicle live loads. The vehicle live loads can be combined with static and dynamic loads and envelopes of the response can be computed.

Displacements, spring forces and Frame-element internal forces can be determined due to the influence of Vehicle live loads. Other element types (Shell, Plane, Asolid, Solid and NLLink) may be used; they contribute to the stiffness of the structure, but they are not analyzed for the effects of Vehicle loads.

You may select Vehicle live loads from a set of standard highway and railway Vehicles, or you may create your own Vehicle live loads.

What do you want to do ?

» Define Lanes

» Assign FRAME Lanes

» Define Vehicles

» Define Vehicles Classes

» Define Bridge Response

» Define Moving Loads

» Define Joint Influence Lines

» Display Frame Influence Lines

Define Lanes

This information is needed to define the traffic Lanes that are required for bridge moving-load analysis. Frame elements are used to represent the traffic Lanes.

1 On the Define menu, click Moving Load Cases ... Lanes ... This will display the Define Bridge Lanes dialog

box.

2 Click Add new Lane button. This will display the Lane Data dialog box.

In the Lane Data dialog box

- Enter the Lane Name or accept the default name.

If you want to define the lanes explicitly:

- Enter the Label of the Frame representing the lane in the edit box
- Enter the distance of the Lane (in the current length units) from the Frame representing the Lane in the Eccentricity edit box.
- Press the Add button to add the new lane.
- Click OK when you are finished adding frame members to the lane.

If you want to define the lanes through the graphic interface by Assigning Lanes:

- Click OK to just add the lane name to the list of lanes.

HINT: You can use the Insert button to insert a new Lane above the selected Lane

3 The new Lane name will be added to the Lanes list box.

4 To Delete or Modify/Show a Lane, select it from the Lanes list box and then click on the Modify or Delete Lanes button.

5 Click OK.

Note: The frame members denoting the Lane should be nearly contiguous and progress in a consistent longitudinal direction In the same way that a real lean on a road would be defined.

Also see:

»| Assign FRAME Lanes

Define Vehicles

This information defines the Vehicle loads that are required for bridge moving-load analysis.

On the Define menu, click Moving Load Cases ... Vehicles ... This will display the Define Vehicles dialog box.

To Add A New Standard Vehicle

1 Click Add new Standard Vehicle from the drop down list box. This will display the Standard Vehicle Data dialog box.

In the Standard Vehicle Data dialog box

- Select the Vehicle type from the drop down list.
- Depending on the selected vehicle you may be required to enter a Scale Factor or Dynamic Allowance in the appropriate edit box.
- Click OK.

2 The new Vehicle name will be added to the Vehicles list box

3 To Delete or Modify/Show a Vehicle, select it from the Vehicles list box and then click on the Modify or Delete Lanes button.

4 Click OK.

To Add A New General Vehicle

1 Click Add General Vehicle from the drop down list box. This will display the General Vehicle Data dialog box.

In the General Vehicle Data dialog box

- Enter the Vehicle Name or accept the default name.
- Select what the Vehicle will be used for in the Usage area. More than one usage can be selected for a Vehicle.
- Enter uniform and first axle load values in the Leading and Trailing Loads area.
- Enter Floating Axial Load Values by selecting either a Single Valued Axle or a Double Valued Axle with separate values for Lane Moments.
- Enter Intermediate Loads.

- Enter Uniform load between axles in the edit box.
- Enter an Axle load in the edit box.
- Enter a Minimum and Maximum Distance between the current and previous axle.
- Press the Add button to add the axle information.

- Click OK.

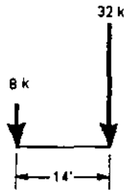
Note: A value of Zero for Max Distance indicates an infinite distance. Only one intermediate axle may have a value of Max Distance > Min Distance. The remaining intermediate axles must have a value of Max Distance = Min Distance.

2 The new Vehicle name will be added to the Vehicles list box.

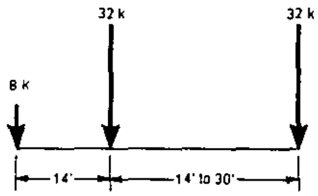
3 To Delete or Modify/Show a Vehicle, select it from the Vehicles list box and then click on the Modify or Delete Lanes button.

4 Click OK.

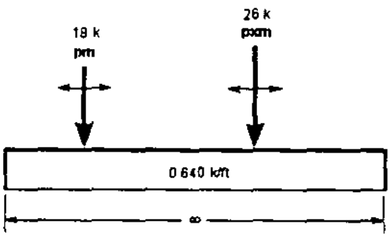
Standard Vehicles



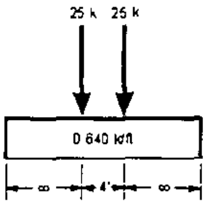
H20-44 Truck Load



HS20-44 Truck Load

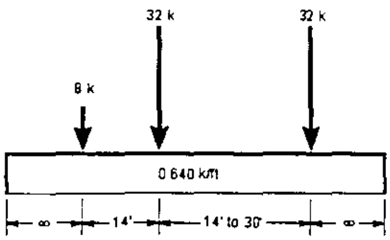


H20-44L and HS20-44L Lane Loads

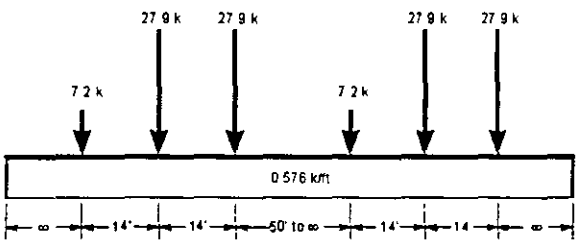


Note: All point loads will be increased by the dynamic load allowance, Im, expressed as a percentage

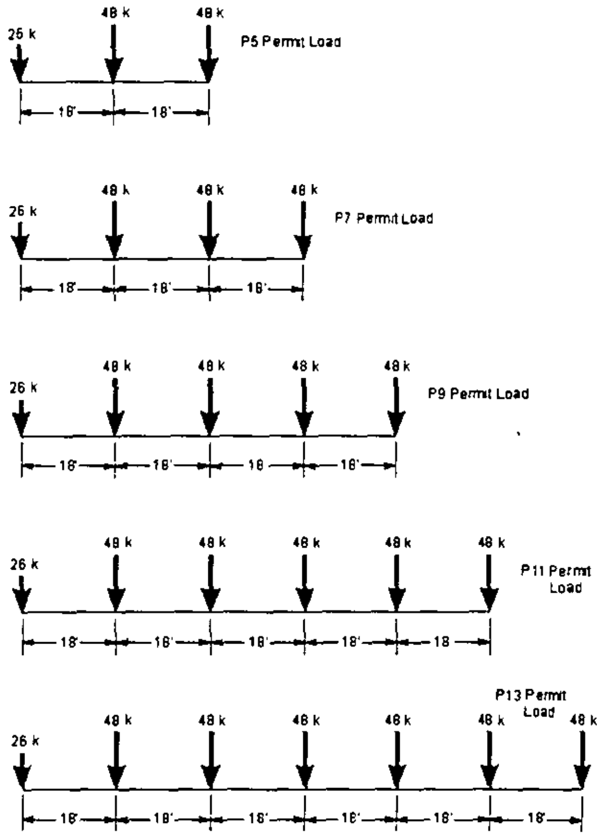
HL-93K Tandem and Lane Load

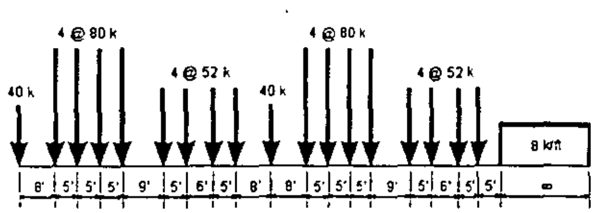


HL-93K Truck and Lane Load

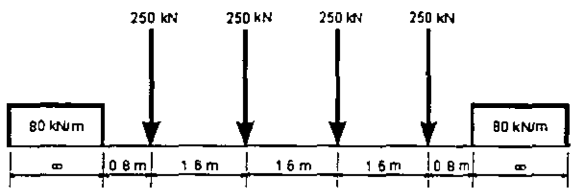


HL-93S Truck and Lane Load for Negative Moment and Reactions at Interior Piers

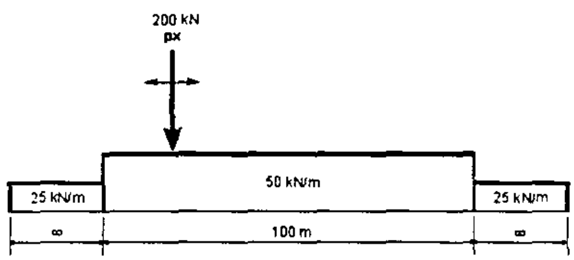

26 k
48 k
48 k
P5 Permit Load
18'
18'
26 k
48 k
48 k
48 k
P7 Permit Load
18'
18'
18'
26 k
48 k
48 k
48 k
48 k
P9 Permit Load
18'
18'
18
18'
26 k
48 k
48 k
48 k
48 k
48 k
P11 Permit Load
18'
18'
18'
18'
18
P13 Permit Load
26 k
48 k
48 k
48 k
48 k
48 k
48 k
18'
18'
18'
18'
18'
18'



Cooper E 80 Train Load

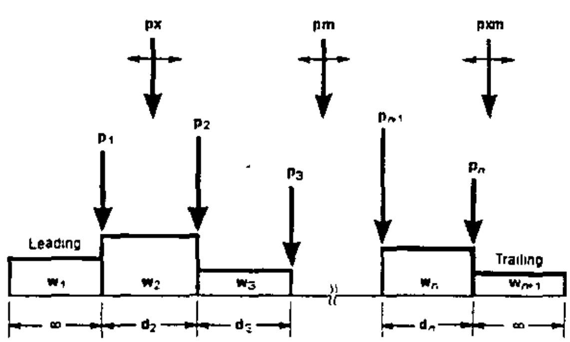


UIC80 Train Load



RL Train Load

General Vehicle



Notes

(1) All loads are point loads or uniform line loads acting on the Lane center line

(2) Any of the point loads or uniform line loads may be zero

(3) The number of axles, n, may be zero or more

(4) One of the inter-axle spacings, d_2 through d_n, may vary over a specified range

(5) The locations of loads **px**, **pm**, and **pxm** are arbitrary

Define Vehicle Classes

This information is needed to define the Vehicle Classes or groups that are used to perform the bridge moving-load analysis. The order in which the Vehicles are assigned to a Vehicle class are not important. The Lanes assigned to the Vehicle class will be checked for each Vehicle in that Vehicle Class.

1 On the Define menu, click Moving Load Cases ... Vehicle Classes ... This will display the Define Vehicle Classes dialog box

2 Click Add Class button. This will display the Vehicle Class Data dialog box.

In the Vehicle Class Data dialog box

- Enter the Vehicle Class Name or accept the default name.
- Select a Vehicle Name from the drop down list box.
- Enter a Scale Factor for the Vehicle in the edit box.
- Press the Add button to add the Vehicle.
- Click OK when you are finished adding Vehicles to the Vehicle Class

3 The new Vehicle Class name will be added to the Classes list box.

4 To Delete or Modify/Show a Vehicle Class, select it from the Classes list box and then click on the Modify or Delete Lanes button.

5 Click OK.

Define Bridge Response

This information is needed to allow you to selectively control what information is calculated for joints and frame elements in the computationally intensive moving-load analysis that is performed

1 On the Define menu, click Moving Load Cases ... Bridge Responses ... This will display the Bridge Response Requests dialog box.

2 Check each of the response categories for which you want analysis done

- From the Select Group drop down list choose the group for which you want the responses computed. The default is ALL.

3 Under the Method of Calculation select Exact or Refinement Level and enter a value in the text input box. The value entered can be any positive integer. This feature is provided to give the user a fast analysis option for preliminary analysis. The larger the integer the greater the level of refinement. The recommended Refinment Levels integers are 1 to 4.

4 Check Calculate Correspondence Values for Frames if you want to use the Max/Min Correspondence in your design of frame sections when using moving loads. This is a very time-intensive operation so you are advised to only use it when it is required.

5 Click OK.

Define Moving Loads

This information defines the Moving Load cases that determine the response to the Vehicles in the Vehicle Classes moving along the traffic Lanes.

1 On the Define menu, click Moving Load Cases ... Moving Load ... This will display the Define Moving Load Cases dialog box

2 Click Add Load button. This will display the Moving Load Cases Data dialog box.

In the Moving Load Cases Data dialog box.

- Enter the Moving Load Case Name or accept the default name.
- Enter a Scale Factor for each of the Number of Lanes combinations selected from the drop down list box. This number is limited by the number of lanes defined in the Lane Data dialog box.
- Press the Add Assign button to assign a Vehicle class to the Moving Load Case.
  - In the Moving Load Case Assignment Data dialog box .
  - Select the Vehicle Class.
  - Enter the Vehicle Class Load Scale Factor.
  - Enter the Minimum and Maximum Number of Loaded Lanes for the selected Vehicle Class.

HINT: A value of zero for Minimum and Maximum Number of Loaded Lanes means include all the lanes .

  - In the Assignment Lanes area Add the Lanes that the Vehicle Class can be assigned to.
  - Click OK.

- To Modify/Show a Vehicle Class – Lane Assignment, select it from the Assignment Number drop down list box and press the Modify/Show Assign button.

- Add as many Vehicle Class Lane Assignments as required.
- Click OK when finished.

3 The new Moving Load Case name will be added to the Moving Loads list box.

4 To Delete or Modify/Show a Moving Load, select it from the Moving Loads list box and then click on the Modify or Delete Load button.

5 Click OK.

Define Joint Patterns

1 On the Define menu, click Joint Patterns... This will display the Pattern Names dialog box with a default pattern, DEFAULT listed in the Patterns list box

2 To add a new pattern name type in the new name in the Patterns edit box and click Add new Pattern Name.

3 To change a pattern name click on the name in the list box and then type over the changes in the edit box and click Change Pattern Name.

4 To delete a pattern name click on the name in the list box and click Delete Pattern Name.

5 Click OK.

Tip: Pattern names defined here are used in Assign Joint Patterns .

Assign Dynamic Loads

Dynamic Loads (as ground motions) may be input as Response Spectra or Time Histories. Time Histories may also be applied as Joint Time Histories.

What do you want to do?

»| Define Time History Functions

»| Define Time History Cases

»| Define Response Spectrum Functions

»| Define Response Spectrum Cases

Define Time History Functions

1 On the Define menu, click Time History Functions... This will display the Time History Functions dialog box with a built-in RAMP function listed in the Functions list box.

To Define Function by specifying Time and Value:

- Click Add new Function button. This will display the Function Definition dialog box.
- Enter a pair of Time and Value (acceleration, displacement or velocity) in the Define Function edit box and then click Add. This will add the pair to the list box.
- Repeat to enter other pairs defining the function.
- Clicking on a value in the list box selects it and then it is possible to either modify or delete it.
- Click OK.

To Define Function from File:

- Click Add Function from File button. This will display the Time History Function Definition dialog box.
- Enter the function name in the Function Name edit box or accept the default name.
- Click on the Open File button to select a file.
- Choose the file name from the Pick Function Data File selection box.
- Click Open.
- Enter the number of points defined per line in the file.
- Click to check if the function is defined at equal intervals or as pairs of time and function values.
- If the function is defined at equal intervals then enter the time step in the edit box.

- Click OK

2 The new Function Name will be added to the Functions list box.
3 To Delete or Modify/Show Function, select the function name from the list box and then click on the Show or Delete Function button.
4 Click OK.

Define Time History Cases

1 On the Define menu, click Time History... This will display the Time History Cases dialog box.
2 Click Add new History button. This will display the Time History Case Data dialog box.
In the Time History Case Data dialog box
- Enter the Time History Case Name or accept the default name.
- In the Options area:
  - Choose Analysis Type to be Linear , Periodic or NonLinear from the drop down box.
  - Click Modify/Show button to specify Modal Damping.
  - Enter the Number of Output Time Steps in the edit box.
  - Enter the Output Time Step size in the edit box.
  - If desired, choose the name of the Time History from which this Time History should be started by clicking on the Start From drop down list.
- In the Load Assignments area:
  - Choose the Load Case from the Load drop down box (acc dir 1 , acc dir 2, acc dir 3 and LOAD1 are listed as default cases)
  - Choose the Function Name from the Function drop down box (RAMP function is listed as default function)
  - Enter the Scale Factor, Arrival Time and Angle of excitation direction in the edit boxes. The Angle is only available for base accelerations and is turned off for Load Cases.
  - Click Add. This will add a Load Assignment to the list box below.
  - To change or delete the definition of a Load Assignment, select the assignment from the list box and then click on the Modify or Delete button.
  - Repeat to add more load assignments to make up a Time History Case.
- Click OK.

3 The new Time History Case name will be added to the History Cases list box.
4 To Delete or Modify/Show History, select the history case from the list box and then click on the Modify or Delete History button.
5 Click OK.

Define Response Spectrum Functions

1 On the Define menu, click Response Spectrum Functions... This will display the Response Spectrum Functions dialog box with built in UNIT, UBC94S1 , UBC94S2 and UBC94S3 functions listed in the Functions list box.

To Define Function by specifying Time and Acceleration:

- Click Add new Function button..
- Enter the function name in the Function Name edit box or accept the default name.
- Enter a pair of Period and Acceleration values in the Define Function edit boxes and then click Add. This will add the pair to the list box.
- Repeat to enter other pairs defining the function.
- Clicking on a value in the list box selects it and then it is possible to either modify or delete it.
- Click OK.

To Define Function from File:

- Click Add Function from File button..
- Enter the function name in the Function Name edit box or accept the default name.

- Click on Open File button to select a file.
- Choose the file name from the Pick Function Data File selection box.
- Click Open.
- Enter the number of points defined per line in the file.
- Click to check if the function is defined at equal intervals or as pairs of Period and Acceleration values.
- If the function is defined at equal intervals then enter the Period step in the edit box.
- Click OK.

2 The new Function Name will be added to the Functions list box
3 To Delete or Modify/Show Function, select it from the Functions list box and then click on the Modify or Delete Function button.
4 Click OK.

Define Response Spectrum Cases

1 On the Define menu, click Response Spectrum Cases... This will display the Response Spectra dialog box
2 Click Add new Spectra button. This will display the Response Spectrum Data dialog box.
In the Response Spectrum Data dialog box

- Enter the Response Spectrum Name or accept the default name
- Enter the excitation angle in the edit box.
- Choose the Modal Combination Technique by clicking on either **CQC, SRSS, ABS or GMC.**
- Enter the Damping ratio in the edit box. This ratio is used for modal combination.
- If GMC is chosen then enter characteristic frequencies F1 and F2 as defined in ASCE 4 for GMC.
- Choose the Directional Combination Technique by clicking on either SRSS or ABS.
- In the Input Response Spectra area:
  - Select the Function Names for U1, U2 and U3 directions from the Function drop down boxes. Unit function is the default built in function
  - Enter the Scale Factor for each function in the Scale Factor edit box The default value is 1.0
- Click OK

3 The new Spectrum name will be added to the Spectra list box.
4 To Delete or Modify/Show Spectra, select it from the Spectra list box and then click on the Modify or Delete Spectra button.
5 Click OK.

Define Load Combinations

1 On the Define menu, click Load Combinations.. This will display the Load Combinations dialog box.
2 To add a new Load Combination, click Add new Combo button. This will display the Load Combination Data dialog box.
In the Load Combination Data dialog box

- Type in the combination name in the Load Combination Name edit box.
- Select the Load Combination Type: ADD, ENVE, ABS, SRSS
- Type in the title (description of load combo) in the Title edit box.
- Select an Analysis Case name from the Case Name drop down list box and type in the multiplier in the Scale Factor edit box and click ADD. This will add the Analysis Case and the multiplier to the list box.
- Similarly select other Analysis Cases and add them to the list box to complete the Load Combination definition.
- It is possible to modify or delete an Analysis Case from the Combination definition by selecting the Analysis Case from the list box and clicking the Modify or Delete button.
- Select whether the combo should be used as a part of the Steel or Concrete design loading combinations.
- Click OK This will add the Load Combination to the Combinations list box.

3 To include the default design load combinations for the code specified in Concrete and Steel Preferences, click on the Add Default Design Combo Button. The program will add Steel design combos if the Steel Design is selected in the Design menu and Concrete design combos if the Concrete Design is selected in the Design menu. The Type assignment for the Static Load Case is used to determine the appropriate load factors.
4 To modify a Load Combination, click on the name in the Combinations list box to select it and then click Modify/Show Combo button. This will display the Load Combination Data dialog box. Make the appropriate changes and then click OK.

5 To delete a Load Combination, click on the name in the Combinations list box to select it and then click Delete Combo button. This will delete the Load Combination.

6 Click OK.

Tip: Load Combinations defined here are used in Select Design Combos to define design loading combinations.

DRAW MENU

Draw Mode

Clicking on any of these buttons from the **Floating Toolbar** puts SAP2000 in the Draw mode.
The Draw mode allows for drawing of new members and editing one element or a joint at a time.
Draw Mode is also the default mode when a NEW model is started.
Select various Draw options by clicking as follows:

To Draw or Edit	Do this
Edit/Shape/Move Element	Click on and click on a joint or Element. Use the handles to move or stretch.
An extra or Special Joint	Click on and click on any point to add a Joint
A Frame Element (joint to joint)	Click on
Quick draw Frame Elements	Click on
A Shell Element (joint to joint)	Click on
Quick draw Shell Elements	Click on

Draw Members

There are two ways to draw Elements:

Quick Draw a single click on a grid segment (for FRAME) or an area bounded by four grids (for SHELL) will draw the element. The ESC key takes you out of the Draw Mode and puts you in the Select mode.

Joint to Joint Draw a sequential drawing of elements by clicking on previously defined joints or points in space. Double click on a joint or pressing ENTER terminates the sequential draw and the ESC key takes you out of the Draw Mode and puts you in the Select mode.

What do you want to do?

Draw a Quick FRAME Element

Draw a FRAME Element by clicking on 2 Joints

»] Draw a Quick SHELL Element

»] Draw a SHELL Element by clicking on 4 Joints

»] Draw an NLLINK Element

Reshape Element

Reshape Element option is used to edit element forms or shapes. With the help of this option it is possible to move FRAME or SHELL elements, stretch or shrink FRAME elements and reshape SHELL elements. To move or reshape an element do the following:

Add in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to reshape the member.
3. Click on [icon] to switch to Reshape Element mode.
4. Click once on a FRAME or a SHELL element to select it. This will display the member end handles
5. Grabbing one of the handles of the element and moving it will stretch, shrink, rotate or reshape the element.
6. Grabbing the element (any point away from the handles) and dropping it at another location will allow you to move the element.

Add in 3-D

In 3-D, the element editing works the same way as 2-D except that the handle may only be dropped at another pre-defined JOINT or Grid intersection.

Note: Only one Element at a time may be moved or reshaped through this option.

Add Special Joints

In building a model in SAP2000 it is not necessary to pre-define Joints. The Joints are automatically added to the ends or corners of the elements. Special Joints are those Joints which are added by the user. These Joints may be necessary to add in rare cases such as at one end of an NLLink Element - an end where there is no other element present and hence no automatically generated Joint. To add a Special Joint do the following:

Add in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to add a Joint.
3. Click on [icon] to switch to Add Special Joint mode.
4. Click on a Grid intersection or any other point in that plane to add a Joint.

Add in 3-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Click on [icon] to switch to Add Special Joint mode.
3. In the 3-D view, click on a Grid intersection to add a Joint.

Note: The Joints may only be added at grid intersections in 3-D view.
Right Click on a Joint will open Joint Information edit box. You may edit the Joint location in this edit box to locate the Joint precisely in the desired location.

Draw a Frame Element from Joint to Joint

Draw in 2-D

1 On the View menu, click Show Grids This puts a check mark next to it and works as a toggle to turn On or Off the Grids.
2 Select 2-D Plane in which you want to draw the member.
3 Click on to switch to drawing a FRAME element between 2 points.
4 Click on a Grid intersection, a previously defined Joint or any point in the plane. Click again on another point to add a single FRAME element. Every subsequent click will add another FRAME element unless a double click on the same joint is performed or the ENTER key is pressed. Pressing the ESC key will also terminate the sequential draw and take you out of the Draw mode.

Draw in 3-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2 Click on to switch to drawing a FRAME element between 2 points.
3 In the 3-D view click on a Grid intersection or a previously defined Joint (a small red ball appears to confirm that the joint has been selected).
4 Click on another Grid intersection or a Joint to add a FRAME element. Every subsequent click on a selected point will add another FRAME element unless a double click on the same joint is performed or the ENTER key is pressed. Pressing the ESC key will also terminate the sequential draw and take you out of the Draw mode.

Note: It is not possible to draw a FRAME element between any two arbitrary points in space in the 3-D view.

Draw a Quick Frame Element

Draw in 2-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.
2 Select 2-D Plane in which you want to draw the member.
3 Click on to choose the quick draw mode.
4 Click on a grid segment to draw a quick single FRAME element.
Clicking in a space bounded by 4 grid lines will add a Cross Brace.

Draw in 3-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.
2 In the 3-D view click on a grid segment. This will add a single element between two grid intersections.

Note: It is not possible to Quick Draw a Cross Brace in 3-D.

Draw a Shell Element between 4 Joints

Draw in 2-D

1 On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2 Select 2-D Plane in which you want to draw the member.
3 Click on to switch to drawing a SHELL element between 4 Joints.
4 Click on a Grid intersection, a previously defined Joint or any point in that plane.
5 Click again on 3 other points either clockwise or counterclockwise to add a SHELL element.

Note: If you want to add a triangular element the fourth point clicked should be the same as the first.

Draw in 3-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Click on to switch to drawing a SHELL element between 4 Joints.
3. In the 3-D view click on a Grid intersection or a previously defined Joint (a small red ball appears to confirm that the point has been selected).
4. Click on 3 other similar points either clockwise or counterclockwise to add a SHELL element .

Note: It is not possible to draw a SHELL element between any 4 arbitrary points in space in the 3-D view

Draw a Quick Shell Element

Draw in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. Select 2-D Plane in which you want to draw the member.
3. Click on to choose the quick draw mode.
4. Click in a grid space, bounded by 4 grid lines, to draw a quick single SHELL element.

Draw in 3-D

Note: It is not possible to draw a Quick SHELL Element in 3-D view.

Draw an NLLink Element

Draw in 2-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn On or Off the Grids.
2. Select 2-D Plane in which you want to draw the member.
3. On the Draw menu, click Draw NLLink Element. This puts you in the non-linear link drawing mode.
4. Click on a previously defined Joint in that plane. Click again on another Joint to add a single NLLink element. Click again on two Joints to add another NLLink element. Double clicking on the same Joint will add a zero length link element. ESC key will take you out of the NLLink drawing mode.

Draw in 3-D

1. On the View menu, click Show Grids. This puts a check mark next to it and works as a toggle to turn on or off the Grids.
2. On the Draw menu, click Draw NLLink Element. This puts you in the non-linear link drawing mode..
3. In the 3-D view, click on a previously defined Joint (a small red ball appears to confirm that the joint has been selected). Click again on another Joint to add a single NLLink element. Click again on two Joints to add another NLLink element. Double clicking on the same Joint will add a zero length link element. ESC key will take you out of the NLLink drawing mode.

Note: It is not possible to draw an NLLink element between any two arbitrary points in space. The link element may only be drawn between previously defined or generated Joints.

Edit Grids

After defining your coordinate system you can edit the location of the individual grids in the active coordinate system.

1. On the Draw menu, click Edit Grid or double click on a grid line in the graphics screen.

2 In the Modify Grids dialog box, Select Direction of the grids you want to edit.
3 In the location area:

To Move a Grid

- Select the grid location from the list.
- Edit the grid's location in the text edit box.
- Click the Move Grid Line button.

To Add a new Grid

- Enter the grid's location in the text edit box.
- Click the Add Grid Line button.

To Delete a Grid

- Select the grid location from the list.
- Click the Delete Grid Line button.

4 The Delete All button will delete all the grids in the coordinate system
5 Check the Lock Grid Lines if you want to lock the grid lines from being moved using the method discussed in the note below.
6 Check the Snap to Grid lines if you want the feature on
7 Check Glue Joints to Grid Lines if you want the joints to move with the grids. This is a very powerful way to quickly edit the structure without having to redefine joint locations.
8 Click OK when finished.

Note: From the display screen with the Reshape Element option set, you can add a new grid line by holding the CTRL-key down and clicking on an existing grid line and dragging away a duplicate.

Also See

»| Setup Coordinate System

Snap to Grid

1 On the Draw menu, click Snap to Grid. This option is available as an On and Off toggle.
2 If Snap to Grid is on, an element drawn or moved near a grid or a grid intersection snaps to it. It is a useful option for aligning objects with precision.
3 When all three snap options are on, Snap to Joints has highest priority, Snap to Grid has priority over Snap to Frame/Edge.

Tip: You can add a new grid by pressing the CTRL key and dragging an existing grid to a new location, or by using the Modify Grid command from the Draw menu. Double clicking on an existing grid opens the Modify Grid box where you may add or modify the location of a grid.

Snap to Joints

1 On the Draw menu, click Snap to Joints. This option is available as an On and Off toggle.
2 If Snap to Joints is on, the end of an Element being drawn snaps to the Joint when it comes close to it (within the default or user specified tolerance). It is very useful for drawing FRAME and SHELL elements when previously defined or generated Joints are available.
3 When all three snap options are on, Snap to Joints has highest priority, Snap to Grid has priority over Snap to Frame/Edge.

Tip: You can modify the default tolerances listed in Preferences within the Options pull down menu.

Snap to Frame/Edge

1 On the Draw menu, click Snap to Frame/Edge. This option is available as an On and Off toggle.
2 If Snap to Frame/Edge is on, an element drawn or moved near a Frame element or a Shell element Edge snaps to it. This option guarantees that the new generated joint will be exactly on the Frame element or the Edge of the Shell element.
3 When all three snap options are on, Snap to Joints has highest priority, , Snap to Grid has priority over Snap to Frame/Edge .

Initialize a New Labeling Scheme

The labels in SAP2000 are alphanumeric. By default the program automatically assigns a numeric numbering scheme to the Joints and Elements. However, it is possible assign an alphanumeric labeling scheme by giving an alpha prefix and a starting numeric sequence. All Joints and Elements, added after the scheme is initialized, will be affected by the scheme

To Initialize a New Labeling Scheme:

1 On the Draw menu, click New Labels... This will display the New Labels dialog box.
2 In the New Labels dialog box enter an alphanumeric prefix, the starting numeric number for the scheme to be added to the prefix and a numeric increment for the numeric sequence for Joints, Frames and Shells.
3 Click OK

Select Menu

Selection Procedures

Multiple selection of Objects (i.e. Elements or Joints) may be made in any session except while in the DRAW mode. In the DRAW mode only one Joint or Element may be selected for editing purposes Clicking on an object selects it. Clicking again on a selected object will deselect it.

You may Also

- Select Objects by drawing a window around them
- Select Objects by drawing a line intersecting them
- Select Objects lying in XY, XZ and YZ planes
- Select Objects by pre-assigned Group names
- Select Objects by Types of Frame Sections
- Select Objects by Types of Shell Sections
- Select Objects by NLLink Properties
- Select Objects by Types of Constraints
- Select Objects by Labels
- Select All Objects

Note: Deselect Options are the same as Select Options and are available through the Select menu. When Deselect options are activated they will deselect the previously selected objects.

Hints: brings back the previous selection.
clears the entire selection.

Select Objects by Window

1 Click [icon] from the floating toolbar or on the Select menu, click Select and then Pointer/Window from the sub-menu. This will switch you into the selection mode.
2 Select Objects by either clicking on them or drawing a window around them. Only those elements completely within the window are selected.

HINT: It is sometimes easier to select a single Joint by drawing a window around it

Select Objects by Intersecting Line

1 Click [icon] from the floating toolbar or on the Select menu, click Select and then Intersecting Line from the sub-menu. This will switch you into the selection mode.
2 Draw a line by clicking at one end and then holding down the left mouse button drag the pointer to the other end of the line. All Objects crossing this line will be selected.

HINT: It is sometimes easier to select a single or a few Elements by drawing a line across

Select Objects by 2D Planes

1 On the Select menu, click Select and then XY Plane or XZ Plane or YZ Plane from the sub-menu. This will switch you into the selection mode.
2 Pick any point on the plane desired and all Elements on that Plane will be selected

HINT: It is a useful selection for editing purposes e.g. adding or deleting a story may very easily be accomplished by this selection option.

Select Objects by Groups

1 On the Select menu, click Select and then Groups from the sub-menu. This will open the Select Groups list box. Click to select one or more of the previously defined groups.
2 Click OK. All Elements belonging to those groups will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output based upon group selections.

Select Objects by Frame Sections

1 On the Select menu, click Select and then Frame Sections... from the sub-menu. This will open the Select Sections list box. Click to select one or more of the previously defined sections.
2 Click OK. All Frame Elements belonging to the selected section types will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output Frame Elements based upon selected sections.

Select Objects by Shell Sections

1 On the Select menu, click Select and then Shell Sections... from the sub-menu. This will open the Select Sections list box. Click to select one or more of the previously defined sections.

2 Click OK. All Shell Elements belonging to the selected Sections will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output Shell Elements based upon selected sections.

Select Objects by NLLink Properties

1 On the Select menu, click Select and then NLLink Properties... from the sub-menu This will open the Select Properties list box. Click to select one or more of the previously defined nonlinear link properties.
2 Click OK. All nonlinear link Elements belonging to the selected properties will be selected.

HINT: It is a very useful selection for editing or viewing purposes You may display, delete, assign or output nonlinear link Elements based upon selected properties.

Select Objects by Constraints

1 On the Select menu, click Select and then Constraints... from the sub-menu. This will open the Select Constraints list box. Click to select one or more of the previously defined Constraints.
2 Click OK. All Joints belonging to the selected Constraints will be selected.

HINT: It is a very useful selection for editing or viewing purposes. You may display, delete, assign or output support Joints or Joints with other type of Constraints.

Select Objects by Labels

1 On the Select menu, click Select and then Labels from the sub-menu This will open the Select by Labels dialog box.
   - Select the Element Type from the drop down list box.
   - Enter Start and End labels.
   - Enter Increment between labels
   - Click OK to select the elements using the specified criteria.
2 The total number of Elements selected is displayed on the status bar.

HINT: It is a useful technique to find specific elements in a large model.

Select All Objects

1 Click from the floating toolbar or on the Select menu, click Select and then All from the sub-menu This will select all of the objects in the model.
2 The total number of all the Joints and Elements selected is displayed on the status bar.

HINT: It is a useful technique to find out the total number of Joints or Elements in the model.

ASSIGN MENU

Assign Options

Clicking on any of these buttons from the **Floating Toolbar** allows you to Assign properties, loads and restraints/constraints to your selection. Members or Joints have to be selected first in order to make an assignment.

Select various Assign options by clicking as follows:

To Assign	Do this
Supports or Restraints to Joints	Click on
Sections to Frame Elements	Click on
Thickness to Shell Elements	Click on
Loads on Joints	Click on
Span and Point Loads on Frame Elements	Click on
Uniform Loads on Shell Elements	Click on

Assign Section Properties

FRAME Elements

1. Select one or more Frame Elements to which you want to Assign the same Section properties.
2. On the Assign menu, click Frame and then Sections... in the sub-menu.
3. In the Frame Sections display box:
   - Select a previously defined Section
   - Click the OK button

Note: If you have not previously defined frame sections, you may define them by importing new sections from a Section property file or adding a user defined section from the "Frame Sections" display box.

SHELL Elements

1. Select one or more Shell Elements to which you want to Assign Section properties.
2. On the Assign menu, click Shell and then Sections... in the sub-menu.

3 In the Shell Sections display box

- Select a previously defined Section
- Click the OK button

Note: If you have not previously defined shell sections, you may add new sections by clicking on Add New Section

Assign Joint Constraints

Constraints are applied to selected joints.

1 Select the Joints to which you want to apply Constraints

2 On the Assign menu, click Joint and then Constraints... in the sub-menu This will display the Constraints dialog box.

3 In the Constraints dialog box:

- Either select default names for different types of Constraints such as Beam, Body, Diaphragm, Equal, Local, Plate, Rod or Weld or
- Add a new Constraint by clicking on the Add drop down list and selecting the appropriate type of constraint Type in the new name or accept the default name, choose the axis (X, Y or Z) or Auto to define the direction of the constraint and then click OK.

4 Click OK.

How do I ?

» Add some Joints to an existing Constraint

» Delete or Remove some Joints from an existing Constraint

Add Joints to an existing Constraint

1 Select the Joints you want to add to an existing Constraint

2 On the Assign menu, click Joint and then Constraints... in the sub-menu. This will display the Constraints dialog box.

3 In the Constraints dialog box:

- Choose the existing Constraint from the Constraints list box
- Click OK

4 The Joints will be added to the existing Constraint.

Delete or Remove Joints from an existing Constraint

1 Select the Joints you want to remove from an existing Constraint.

2 On the Assign menu, click Joint and then Constraints... in the sub-menu. This will display the Constraints dialog box.

3 In the Constraints dialog box:

- Choose the existing Constraint from the Constraints list box
- Click on Modify/Show Constraint
- Choose Remove Constraint
- Click OK

4 The Joints will be removed from the existing Constraint.

Generalized Constraints

Constraints are used to enforce certain types of rigid-body behavior, to connect together different parts of the model, and/or to impose certain types of symmetry conditions.

A constraint consists of a set of two of more **constrained joints**. The displacements of each pair of joints in the constraint are related by constraint equations. The types of behavior that can be enforced by constraints are, Rigid-body behavior, Equal-displacement behavior, Symmetry and anti-symmetry conditions.

Also See

- Rigid Body: fully rigid for all displacements
- Rigid Diaphragm: rigid for membrane behavior in a plane
- Rigid Plate: rigid for plate bending in a plane
- Rigid Rod: rigid for extension along an axis
- Rigid Beam: rigid for beam bending on an axis
- Equal-displacement behavior
- Partial Connection
- Local Constraint
- Symmetry About a Plane
- Anti-symmetry About a Plane
- Axisymmetry
- Cyclic symmetry
- Symmetry About a Point

Hint: The use of constraints reduces the number of equations in the system to be solved and will usually result in increased computational efficiency.

Body Constraint

A Body Constraint causes all of its constrained joints to move together as a three-dimensional rigid body. Effectively, all constrained joints are connected to each other by rigid links and cannot displace relative to each other.

This Constraint can be used to:

- Model rigid connections, such as where several beams and/or columns frame together
- Connect together different parts of the structural model that were defined using separate meshes
- Connect Frame elements that are acting as eccentric stiffeners to Shell elements

Joint Connectivity

Each Body Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space.

Local Coordinate System

Each Body Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. These correspond to the X, Y, and Z axes of the global coordinate system, respectively. The actual orientation of the local axes is not important since the constraint equations are independent of the coordinate system.

Diaphragm Constraint

A Diaphragm Constraint causes all of its constrained joints to move together as a planar diaphragm that is rigid against membrane deformation. Effectively, all constrained joints are connected to each other by links that are rigid in the plane, but do not affect out-of-plane (plate) deformation.

This Constraint can be used to:

- Model concrete floors (or concrete-filled decks) in building structures, which typically have very high in-plane stiffness
- Model diaphragms in bridge superstructures

The use of the Diaphragm Constraint for building structures eliminates the numerical-accuracy problems created when the large in-plane stiffness of a floor diaphragm is modeled with membrane elements. It is also very useful in the lateral (horizontal) dynamic analysis of buildings, as it results in a significant reduction in the size of the eigenvalue problem to be solved.

Joint Connectivity

Each Diaphragm Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space, but for best results all joints should lie in the plane of the constraint. Otherwise, in-plane moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Local Coordinate System

Each Diaphragm Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 3 is always normal to the plane of the constraint. The program arbitrarily chooses the orientation of axes 1 and 2 in the plane. The actual orientation of the planar axes is not important since only the normal direction affects the constraint equations

Plate Constraint

A Plate Constraint causes all of its constrained joints to move together as a flat plate that is rigid against bending deformation. Effectively, all constrained joints are connected to each other by links that are rigid for out-of-plane bending, but do not affect in-plane (membrane) deformation.

This Constraint can be used to:

- Connect structural-type elements (Frame and Shell) to solid-type elements (Plane and Solid); the rotation in the structural element can be converted to a pair of equal and opposite translations in the solid element by the Constraint
- Enforce the assumption that "plane sections remain plane" in detailed models of beam bending

Joint Connectivity

Each Plate Constraint connects a set of two or more joints together. The joints may have any arbitrary location in space. Unlike the Diaphragm Constraint, equilibrium is not affected by whether or not all joints lie in the plane of the Plate Constraint.

Local Coordinate System

Each Plate Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 3 is always normal to the plane of the constraint. The program arbitrarily chooses the orientation of axes 1 and 2 in the plane. The actual orientation of the planar axes is not important since only the normal direction affects the constraint equations.

Rod Constraint

A Rod Constraint causes all of its constrained joints to move together as a straight rod that is rigid against axial deformation. Effectively, all constrained joints maintain a fixed distance from each other in the direction parallel to the axis of the rod, but translations normal to the axis and all rotations are unaffected.

This Constraint can be used to:

- Prevent axial deformation in Frame elements
- Model rigid truss-like links

An example of the use of the Rod Constraint is in the analysis of the two-dimensional frame shown in Figure 30 (page 128) of the User's Manual. If the axial deformations in the beams are negligible, a single Rod Constraint could be defined containing the five joints. Instead of five equations, the program would use a single equation to define the X-displacement of the whole floor. However, it should be noted that this will result in the axial forces of the beams being output as zero, as the Constraint will cause the ends of the beams to translate together in the X-direction. Interpretations of such results associated with the use of

Constraints should be clearly understood.

Joint Connectivity

Each Rod Constraint connects a set of two or more joints together The joints may have any arbitrary location in space, but for best results all joints should lie on the axis of the constraint. Otherwise, transverse (bending) moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Local Coordinate System

Each Rod Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 1 is always the axis of the constraint. The program arbitrarily chooses the orientation of the transverse axes 2 and 3. The actual orientation of the transverse axes is not important since only the axial direction affects the constraint equations.

Beam Constraint

A Beam Constraint causes all of its constrained joints to move together as a straight beam that is rigid against bending deformation. Effectively, all constrained joints are connected to each other by links that are rigid for off-axis bending, but do not affect translation along or rotation about the axis.

This Constraint can be used to:

- Connect structural-type elements (Frame and Shell) to solid-type elements (Plane and Solid); the rotation in the structural element can be converted to a pair of equal and opposite translations in the solid element by tne Constraint
- Prevent bending deformation in Frame elements

Joint Connectivity

Each Beam Constraint connects a set of two or more joints together The joints may have any arbitrary location in space, but for best results all joints should lie on the axis of the constraint. Otherwise, torsional moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Local Coordinate System

Each Beam Constraint has its own local coordinate system, the axes of which are denoted 1, 2, and 3. Local axis 1 is always the axis of the constraint. The program arbitrarily chooses the orientation of the transverse axes 2 and 3. The actual orientation of the transverse axes is not important since only the axial direction affects the constraint equations.

Equal Constraint

An Equal Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the constraint local coordinate system. The other degrees of freedom are unaffected.

The Equal Constraint differs from the rigid-body types of Constraints in that there is *no coupling* between the rotations and the translations.

This Constraint can be used to:

- Model symmetry and anti-symmetry conditions with respect to a plane
- Partially connect together different parts of the structural model, such as at expansion joints and hinges

For fully connecting meshes, it is better to use the Body Constraint when the constrained joints are not in exactly the same location.

Joint Connectivity

Each Equal Constraint connects a set of two or more joints together. For a given Constraint, if *any* of the selected degrees of freedom are negative (i.e., opposite), only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space, but for best results all joints should share the same location in space if used for connecting meshes. Otherwise, moments may be generated that are restrained by the Constraint, which unrealistically stiffens the structure. If this happens, the constraint forces printed in the output file may not be in equilibrium.

Such restrained moments may also be generated when Equal Constraints are used for symmetry purposes. They are necessary to enforce the desired symmetry or anti-symmetry of the displacements when the applied loads are not correspondingly symmetric or anti-symmetric.

Local Coordinate System

Each Equal Constraint uses a fixed coordinate system, **csys**, that you specify. The default for **csys** is zero, indicating the global coordinate system. The axes of the fixed coordinate system are denoted X, Y, and Z.

Selected Degrees of Freedom

For each Equal Constraint you may specify a list, **cdofs**, of up to six degrees of freedom in coordinate system **csys** that are to be constrained. The degrees of freedom are indicated as UX, UY, UZ, RX, RY, and RZ. A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., -UX.

Partial Connection

When joints are being connected, all specified degrees of freedom are positive. For example, consider an idealized hinge connection of eight space-truss members. Only displacements are continuous across the hinge, not rotations. Each truss member is connected to a separate joint (node) at the connection. One Equal Constraint is defined for the eight constrained joints. The degrees of freedom that would be specified for this Constraint are: UX, UY, and UZ.

The eight joints should be coincident or the axes of the truss members should all intersect at the same point. Otherwise, moments may be generated that are unrealistically restrained by the Constraint.

Local Constraint

A Local Constraint causes all of its constrained joints to move together with the same (or opposite) displacements for each selected degree of freedom, taken in the separate joint local coordinate systems.

The other degrees of freedom are unaffected.

The Local Constraint differs from the rigid-body types of Constraints in that there is *no coupling* between the rotations and the translations. The Local Constraint is the same as the Equal Constraint if all constrained joints have the same local coordinate system.

This Constraint can be used to:

- Model symmetry conditions with respect to a line or a point
- Model displacements constrained by mechanisms

The behavior of this Constraint is dependent upon the choice of the local coordinate systems of the constrained joints.

Joint Connectivity

Each Local Constraint connects a set of two or more joints together. If *any* of the selected degrees of freedom for a given Constraint are negative (i.e., opposite) only two constrained joints are allowed for that Constraint. Otherwise any number of constrained joints are permitted.

The joints may have any arbitrary location in space. If the joints do not share the same location in space, moments may be generated that are restrained by the Constraint. If this happens, the constraint forces printed in the output file may not be in equilibrium These moments are necessary to enforce the desired symmetry of the displacements when the applied loads are not symmetric, or may represent the constraining action of a mechanism.

No Local Coordinate System

A Local Constraint does not have its own local coordinate system. The constraint equations are written in terms of constrained joint local coordinate systems, which may differ. The axes of these coordinate systems are denoted 1, 2, and 3.

Selected Degrees of Freedom

For each Local Constraint you may specify a list, **ldofs**, of up to six degrees of freedom in the joint local coordinate systems that are to be constrained The degrees of freedom are indicated as U1, U2, U3, R1, R2, and R3 A negative sign indicates a degree of freedom that is constrained to be opposite, e.g., -U1.

Constraint Equations

The constraint equations relate the displacements at any two constrained joints (subscripts I and j) in a Local Constraint. These equations are expressed in terms of the translations u1, u2, and u3 and the rotationsr1, r2 and r3, all taken in joint local coordinate systems The equations used depend upon the selected degrees of freedom and their signs. Some important cases are described next.

Symmetry About a Plane

For a structure that is symmetric about a plane, symmetric loading causes symmetric displacements as follows:

- Forces and displacements parallel to the plane of symmetry are equal
- Forces and displacements normal to the plane of symmetry are opposite
- Moments and rotations parallel to the plane of symmetry are opposite
- Moments and rotations normal to the plane of symmetry are equal

As an example, consider a structure that is symmetric with respect to a plane normal to the X axis and subjected to symmetric loading. A separate Equal Constraint must be defined for each pair of joints that is symmetrically located with respect to the plane. The degrees of freedom that would be specified for these Constraints are: -UX, UY, UZ, RX, RY, and -RZ.

Any joints on the plane of symmetry should not be constrained, but instead have their UX, RY, and RZ degrees of freedom restrained.

Anti-symmetry About a Plane

For a structure that is symmetric about a plane, anti-symmetric loading causes anti-symmetric displacements. All degrees of freedom that are equal when symmetric are opposite when anti-symmetric, and all degrees of freedom that are opposite when symmetric are equal when anti-symmetric. Thus the specification of the anti-symmetric degrees of freedom simply uses the opposite signs from the symmetric case.

Consider the example above of a structure that is symmetric with respect to a plane normal to the X axis, but now subjected to anti-symmetric loading. A separate Equal Constraint must be defined for each pair of joints that is symmetrically located with respect to the plane. The degrees of freedom that would be specified for these Constraints are: UX, -UY, -UZ, -RX, RY, and RZ. The signs of the constraint equations are corresponding changed from the symmetric case.

Axisymmetry

Axisymmetry is a type of symmetry about a line It is best described in terms of a cylindrical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be axisymmetric about a line if they do not vary with angular position around the line, i.e., they are independent of the angular coordinate CA.

To enforce axisymmetry using the Local Constraint:

- Model any cylindrical sector of the structure using any axisymmetric mesh of joints and elements
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +CR, +CA, and +CZ, respectively
- For each axisymmetric set of joints (i.e., having the same coordinates CR and CZ, but different CA), define a Local Constraint using all six degrees of freedom: U1, U2, U3, R1, R2, and R3
- Restrain joints that lie on the line of symmetry so that, at most, only axial translations (U3) and rotations (R3) are permitted

Cyclic symmetry

Cyclic symmetry is another type of symmetry about a line. It is best described in terms of a cylindrical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be cyclically symmetric about a line if they vary with angular position in a repeated (periodic) fashion.

To enforce cyclic symmetry using the Local Constraint.

- Model any number of adjacent, representative, cylindrical sectors of the structure; denote the size of a single sector by the angle q
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +CR, +CA, and +CZ, respectively
- For each cyclically symmetric set of joints (i.e., having the same coordinates CR and CZ, but with coordinate CA differing by multiples of q, define a Local Constraint using all six degrees of freedom: U1, U2, U3, R1, R2, and R3.
- Restrain joints that lie on the line of symmetry so that, at most, only axial translations (U3) and rotations (R3) are permitted

For example, suppose a structure is composed of six identical 600 sectors, identically loaded. If two adjacent sectors were modeled, each Local Constraint would apply to a set of two joints, except that three joints would be constrained on the symmetry planes at 00, 600, and 1200.

If a single sector is modeled, only joints on the symmetry planes need to be constrained.

Symmetry About a Point

Symmetry about a point is best described in terms of a spherical coordinate system having its Z axis on the line of symmetry. The structure, loading, and displacements are each said to be symmetric about a point if they do not vary with angular position about the point, i.e., they are independent of the angular coordinates SB and SA. Radial translation is the only displacement component that is permissible.

To enforce symmetry about a point using the Local Constraint:

- Model any spherical sector of the structure using any symmetric mesh of joints and elements
- Assign each joint a local coordinate system such that local axes 1, 2, and 3 correspond to the coordinate directions +SB, +SA, and +SR, respectively
- For each symmetric set of joints (i.e., having the same coordinate SR, but different coordinates SB and SA), define a Local Constraint using only degree of freedom U3
- For all joints, restrain the degrees of freedom U1, U2, R1, R2, and R3
- Fully restrain any joints that lie at the point of symmetry

It is also possible to define a case for symmetry about a point that is similar to cyclic symmetry around a line, e.g., where each octant of the structure is identical.

Assign Joint Springs

Springs are added to selected joints.

1 Select the Joints to which you want to apply Springs.

2 On the Assign menu, click Joint and then Springs... in the sub-menu. This will display the Joint Springs dialog box.
3 In the Joint Springs dialog box:
   - Type in the Spring Stiffness values in the local directions for the three Translations and the three Rotations.
   - In the Options area select the appropriate option - Add to existing springs, Replace existing springs or Delete existing springs.
   - If required specify the upper half of the Coupled 6X6 Spring matrix, accessed by clicking the Advanced button.
4 Click OK.

Assign Joint Masses

Masses are added to selected joints.

1 Select the Joints to which you want to apply Masses.
2 On the Assign menu, click Joint and then Masses... in the sub-menu This will display the Joint Masses dialog box.
3 In the Joint Masses dialog box:
   - Type in the Mass values in local Direction 1, Direction 2 and Direction 3.
   - Type in the Mass Moment of Inertia values in Rotation about 1, Rotation about 2 and Rotation about 3.
   - In the Options area select the appropriate option - Add to existing masses, Replace existing masses or Delete existing masses.
4 Click OK.

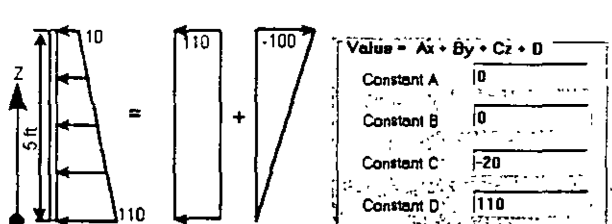
Assign Joint Restraints

Restraints are added to selected joints. 

1 Select the Joints to which you want to apply Restraints
2 On the Assign menu, click Joint and then Restraints... in the sub-menu. This will display the Joint Restraints dialog box.
3 In the Joint Restraint dialog box:
   - Select from the Restraints in Local Directions, the joint restraint(s) desired, or
   - Select from Fast Restraints, the icon that represents the desired restraint condition.
4 Click OK.

Assign Joint Patterns

1 Select the Joints to which you want to assign the pattern.
2 On the Assign menu, click Joint Patterns... This will display the Pattern Data dialog box.
3 Click on the Pattern Name drop down box and choose a previously defined pattern name by clicking on it.
4 Type in the values in the edit boxes for Constants A, B, C and D, relevant to defining the Value of **Ax+By+Cz+D** (x, y and z are joint coordinates). It is this Value that will be assigned to the Joints. It can define the variation of Temperature for Frame Elements and variation of Pressure for Shell Elements. These Values are then subsequently multiplied by the Temperature values assigned from the Frame Static Loads or Pressure values assigned from the Shell Static Loads. For instance if you want to assign a pattern which represents the variation of soil pressure on a wall



5 Options

- To add computed Values from Constants to existing Values from previously assigned patterns, click on Add to existing values under Options.
- To replace computed Values from Constants to existing Values from previously assigned patterns, click on Replace existing values under Options.
- To delete previously assigned patterns, click on Delete existing values under Options

6 Click to check Zero negative values if negative variation is not desired.

7 Click OK.

Also See:

»| Define Joint Patterns

Assign Local Axes

All Elements and Joints have default local axes. It is possible to change the definition of local axes by simply defining a rotation angle.

What do you want to do?

»| Assign/Change Local Axes for FRAME Elements

»| Assign/Change Local Axes for SHELL Elements

»| Assign/Change Local Axes for JOINTS

Assign Local Axes for Joints

1 Select one or more Joints to which you want to Assign the same Local Axes.

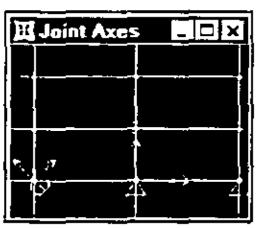
2 On the Assign menu, click Joint and then Local Axes... in the sub-menu.

3 In the Joint Local Axis display box:

- Type in a value for rotation angles about Z, Y' and X'' axes, in degrees (Y' and X'' instead of Y and X are used as axes names to emphasize that once an angle about Z is specified the local X and Y will change as a result)
- Check the Use default box if Global definition is to be used for the local axes.
- Click the OK button

Note: The local axes arrows are only shown when the local axes for the JOINT are different than the Global.

EXAMPLE:



Assign Local Axes for Frame Elements

1 Select one or more Frame Elements to which you want to Assign the same Local Axes.

2 On the Assign menu, click Frame and then Local Axes... in the sub-menu.

3 In the Frame Local Axis display box:

- Type in a value for Angle in Degrees. This is an angle by which local axis 2 of the Element will be rotated around local axis 1. Local axis 1 is along the length of the Element. By default local axis 2 is always in the 1-Z plane except if the Element is vertical and then it is parallel to the global X axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 1 is pointing towards you..

- Check the Reverse start and end connectivity box if you want to flip the I and J ends of the members i.e flip the local axis 1.
- Click the OK button

Note: By default global Z is taken as the 3D View Up Direction. You may change the UP direction from the Option pull down menu. If you change the UP direction, it will only affect the display of the model and does not affect the local axes definition in any way.

Assign Frame Releases

Releases are assigned to selected Frame Elements.

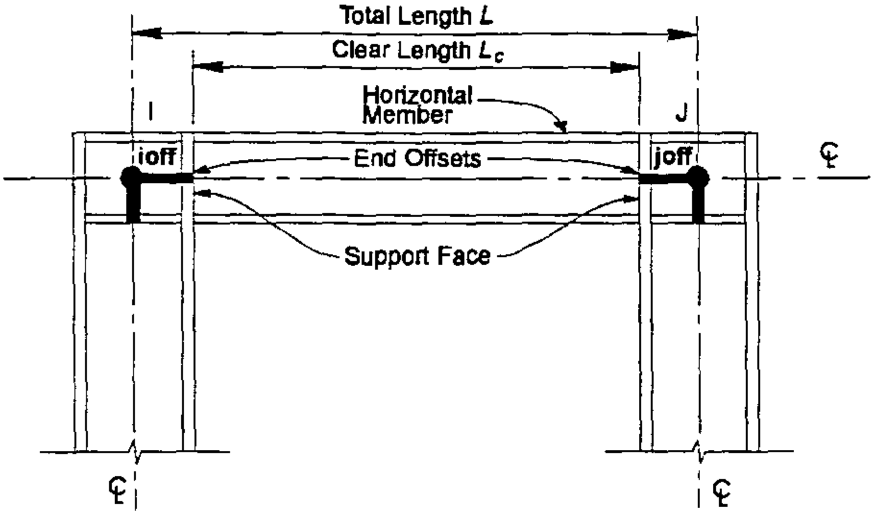
1. Select the FRAME Elements to which you want to apply the releases.
2. On the Assign menu, click Frame and then Releases... in the sub-menu. This will display the Frame Releases dialog box.
3. In the Frame Releases dialog box:
   - For each end of the FRAME Element check the type of release desired (Axial, Shear Force 2 (Major), Shear Force 3 (Minor), Torsion, Moment 22 (Minor) and Moment 33 (Major).
   - If no releases are desired then check the No Releases box.
4. Click OK.

Note: The releases specified always replace the existing releases.

Assign Frame End Offsets

Frame End Offsets are assigned to selected Frame Elements.

1. Select the FRAME Elements to which you want to apply the Frame End Offsets.
2. On the Assign menu, click Frame and then End Offsets ... in the sub-menu. This will display the Frame End Offsets dialog box.
3. In the Frame End Offsets dialog box.
   - If you want to use User Specified End Offsets then check Define Lengths and type in the values for the Offsets at End I and End J.
   - If you want the program to calculate the End Offsets from the connectivity of the model then check Update lengths from current connectivity. The program will automatically calculate the End Offsets from the Depth (Major) and Width (Minor) specified for the Frame Element properties.
   - Specify a Rigid Zone Factor in the edit box. This is a factor used to define the percentage of the Zone specified through End Offsets to be taken as fully rigid. 0 means no rigid zone and 1 means that the entire zone is taken as rigid.
4. Click OK.



Assign Frame Output Segments

Frame Output Segments are assigned to selected Frame Elements.

1. Select the FRAME Elements to which you want to assign the Frame Output Segments.
2. On the Assign menu, click Frame and then Output Segments... in the sub-menu. This will display the Frame Output Segments dialog box.
3. In the Frame Output Segments dialog box:
   - In the Number of Segments edit box specify the number of segments at which the output is desired..
4. Click OK.

Note: The output segments are specified on the Clear Length of the Elements.

Tip: It is advisable to specify a different number of output segments for Columns than for Beams (columns typically are designed using only the two end values of forces and moments whereas beams may be designed at the maximum values occurring somewhere in between the length because of the presence of point or distributed loads on the span of the beam).

Assign Frame Prestress

Assigning Frame Prestress loads is a way to model the prestress cable load on a frame member.

1. Select the FRAME Elements to which you want to assign a Prestress load.
2. On the Assign menu, click Frame and then Prestress... in the sub-menu. This will display the Frame Prestressing Patterns dialog box.
3. In the Frame Prestressing Patterns dialog box enter:
   - The Cable Tension in the text edit box
   - The Start, Middle and End cable locations
   - Select from Add, Replace or Delete.
4. Click OK.

Note: You can assign more than one Prestress load to an individual frame member. Once you have assigned prestress loads to elements you will need to assign the frame prestress loads to a static load case.

Also see:

» Assign Prestress to FRAME Static Load

Assign Frame Initial P-Delta Force

1. Select the FRAME Elements to which you want to assign a P-Delta load.
2. On the Assign menu, click Frame and then P-Delta .. in the sub-menu. This will display the Frame Initial P-Delta Forces dialog box
3. In the Frame Initial P-Delta Forces dialog box:
   - Select in which direction you want the load specified and enter the P-Delta axial force acting on the selected frame elements.
   - Select from Add, Replace or Delete.
4. Click OK.

Note: You can assign more than one P-Delta load to an individual frame member.

Assign Frame Lanes

1. Select the FRAME Elements you want assigned to an already defined lane.
2. On the Assign menu, click Frame and then Lane . . in the sub-menu. This will display the Assign Lane dialog box.
3. In the Assign Lane dialog box:
   - Select the lane you want the frames assigned to from the drop down list box.
   - Enter an eccentricity for the lane relative to the frame member in the text edit box. The units for the eccentricity are the same as the current length units.
   - Press the Modify/Show Lane button to edit the order of the frame elements and their eccentricities.
4. Click OK.

Note: A frame member can be a part of more then one lane

»| Define Lanes

Assign Prestress to Frame Static Load

Once the prestress has been assigned to the frame member the load from that prestress needs to be assigned to a Load Case for SAP2000 to be able to include it in the analysis.

1. Select the FRAME Elements for which you want the prestress load assigned to a defined Load Case
2. On the Assign menu, click Frame Static Loads and then Prestress ... in the sub-menu. This will display the Frame Prestress Loads dialog box.
   - Select the Load Case from the drop down list box.
   - Set a scale factor for the prestress load This value is independent of scale factor for Load Combinations .
   - Select Add, Replace, Delete existing load.
3. Click OK

Also see:

»| Assign FRAME Prestress

Assign Local Axes for Shell Elements

1. Select one or more Shell Elements to which you want to Assign the same Local Axes.

2 On the Assign menu, click Shell and then Local Axes... in the sub-menu
3 In the Shell Local Axis display box:

- Type in a value for Angle in Degrees. This is an angle by which local axis 2 of the Element will be rotated around local axis 3. Local axis 3 is normal to the surface of the SHELL Element. By default local axis 2 is always in the 3-Z plane (and lies in the plane of the SHELL) except if the Element is horizontal and then it is parallel to the global X axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 3 is pointing towards you..
- Check the Reverse direction of normal box if you want to flip the direction of local axis 3.
- Click the OK button

Note: The positive direction of the local axis 3 is determined by the definition of the SHELL connectivity. See Sign Convention for more info.

Assign Local Axis For NLLinks

1 Select one or more NLLink Elements to which you want to Assign the same Local Axes.
2 On the Assign menu, click NLLink and then Local Axes... in the sub-menu.
3 In the NLLink Local Axis display box:

- Type in a value for Angle in Degrees. This is an angle by which local axis 2 of the Element will be rotated around local axis 1. Local axis 1 is along the length of the Element. By default local axis 2 is always in the 1-Z plane except if the Element is vertical and then it is parallel to the global X axis. The definition of the local axes follow the right hand rule. The angle is measured anti-clockwise as positive if local axis 1 is pointing towards you..
- Check the Reverse start and end connectivity box if you want to flip the I and J ends of the members i.e flip the local axis 1.
- Click the OK button

Note: By default global Z is taken as the 3D View Up Direction. You may change the UP direction from the Option pull down menu. If you change the UP direction, it will only affect the display of the model and does not affect the local axes definition in any way.

Assign Static Loads

Loads are applied to selected JOINTS, FRAME elements or SHELL elements

What do you want to do?

- Assign Loads or Displacements to JOINTS
- Assign Gravity Loads to FRAMES
- Assign Point and Uniform Loads to FRAMES
- Assign Trapezoidal Loads to FRAMES
- Assign Temperature Loads to FRAMES
- Assign Gravity Loads to SHELLS
- Assign Uniform Loads to SHELLS
- Assign Pressure Loads to SHELLS
- Assign Temperature Loads to SHELLS

Assign Loads or Displacements to Joints

1 Select one or more JOINTS to assign loads to.
2 On the Assign menu, click Joint Static Loads and then Forces... or Displacements... in the sub-menu.
3 In the Joint Forces or Ground Displacement dialog box:

- Select the Load Case
- Provide Forces and Moments or Translations and Rotations.

- Select from Add, Replace, Delete

4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the Joints selected.

Note: Displacements may only be applied to previously RESTRAINED Joints.

Tip: Click to assign Joint forces.

Assign Gravity Loads to Frames

This loading method is a way to add the factored self weight of the members in as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the Static Load Cases .

1 Select one or more FRAMES to assign loads to.

2 On the Assign menu, click Frame Static Loads and then Gravity... in the sub-menu. This will display the Frame Gravity Loads dialog box.

3 In the Frame Gravity Loads dialog box:

- Select the Load Case
- Provide Gravity Multipliers
- Select from Add, Replace, Delete

4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Point and Uniform Loads to Frames

1 Select one or more FRAMES to assign loads to.

2 On the Assign menu, click Frame Static Loads and then Point and Uniform... in the sub-menu. This will display the Point and Uniform Span Loads dialog box.

3 In the Point and Uniform Span Loads dialog box:

- Select Load Case
- Choose Load Type and Direction
- Provide Point Loads and Distances
- Provide Uniform Load
- Select from Add, Replace, Delete

4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Tip: Click to assign Point and Uniform loads to FRAMES.

Assign Trapezoidal Loads to Frames

1 Select one or more FRAMES to assign loads to.

2 On the Assign menu, click Frame Static Loads and then Trapezoidal... in the sub-menu. This will display the Trapezoidal Span Loads dialog box.

3 In the Trapezoidal Span Loads dialog box:

- Select Load Case
- Choose Load Type and Direction
- Provide Loads and Distances
- Select from Add, Replace, Delete

4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Temperature Loads to Frames

1 Select one or more FRAMES to assign loads to.
2 On the Assign menu, click Frame Static Loads and then Temperature... in the sub-menu. This will display the Frame Temperature Loading dialog box.
3 In the Frame Temperature Loading dialog box:
   - Select Load Case
   - Select Type from Temperature, Temperature Gradient 2-2, Temperature Gradient 3-3
   - Select Temperature by Element or Joint Pattern
   - Select from Add, Replace, Delete
4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Gravity Loads to Shells

This loading method is a way to add the factored self weight of the members in as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the Static Load Cases

1 Select one or more SHELLS to assign loads to.
2 On the Assign menu, click Shell Static Loads and then Gravity... in the sub-menu. This will display the Shell Gravity Loads dialog box.
3 In the Shell Gravity Loads dialog box:
   - Select the Load Case
   - Provide Gravity Multipliers
   - Select from Add, Replace, Delete
4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Uniform Loads to Shells

1 Select one or more SHELLS to assign loads to.
2 On the Assign menu, click Shell Static Loads and then Uniform... in the sub-menu. This will display the Shell Uniform Loads dialog box.
3 In the Shell Uniform Loads dialog box:
   - Select Load Case
   - Provide Uniform Load value and Direction
   - Select from Add, Replace, Delete
4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Tip: Click  to assign Uniform loads to SHELLS.

Assign Pressure Loads to Shells

1 Select one or more SHELLS to assign loads to.
2 On the Assign menu, click Shell Static Loads and then Pressure... in the sub-menu. This will display the Shell Pressure Loads dialog box.
3 In the Shell Pressure Loads dialog box:
   - Select Load Case
   - Choose if pressure be applied by Element or by Joint Pattern
   - If by Element, then specify pressure value
   - If by Joint Pattern, then choose pattern and multiplier
   - Select from Add, Replace, Delete
4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Temperature Loads to Shells

1 Select one or more SHELLS to assign loads to
2 On the Assign menu, click Shell Static Loads and then Temperature... in the sub-menu. This will display the Shell Temperature Loading dialog box.
3 In the Shell Temperature Loading dialog box:
   - Select Load Case
   - Choose the type of Temperature Loading, i.e. Temperature or Gradient
   - Choose if temperature to be applied by Element or by Joint Pattern
   - If by Element, then specify temperature value
   - If by Joint Pattern, then choose pattern and multiplier
   - Select from Add, Replace, Delete
4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected

Assign Gravity Loads to NLLinks

This loading method is a way to add the factored self weight of the members in as a force in any of the global directions. It is recommended that the actual self weight of the structure be included in the definition of the Static Load Cases .

1 Select one or more NLLinks to assign loads to.
2 On the Assign menu, click NLLink Loads and then Gravity... in the sub-menu. This will display the NLLink Gravity Loads dialog box.
3 In the NLLink Gravity Loads dialog box:
   - Select the Load Case
   - Provide Gravity Multipliers
   - Select from Add, Replace, Delete
4 Click OK.

The screen will refresh with a graphical representation of the load assigned on the members selected.

Assign Group Names

Assigning Groups is a powerful tool in SAP2000. It helps in selecting elements, displaying and printing results as well as helping with design.

Group Name is assigned to selected Joints and Elements.

1 Select the Joints and Elements to which you want to assign a Group Name.
2 On the Assign menu, click Group Name... This will display the Assign Group dialog box.
3 In the Assign Group dialog box:
   - Click on the group name in the Groups list box. A default name ALL is always present in the list box.
   - It is possible to define a new group at this stage by typing in the new name in the Groups edit box and then clicking on the Add new Group Name button.
   - To Change or Delete a group name, click on the Group name in the list box to select it and then click on the Change Group Name or Delete Group Name button.
4 Click OK.

Tip: To add more Joints and/or Elements to an already assigned Group, first select by group name, select more Joints and/or Elements and then assign them to the Group. Group assignment always replaces the existing Elements in that group.

ANALYZE MENU

Analyzing a Model

1 Click on the Analyze menu and then Set Options... This will display the Analysis Options dialog box.

2 Click to check the appropriate degrees of freedom (U1, U2, U3, R1, R2, R3) to be available for either a 2-D or 3-D model in the Available DOF's area.

3 Alternatively, the available degrees of freedom may be automatically checked by clicking on the four Fast DOF's option i.e. Space Frame, Plane Frame, Plane Grid and Space Truss.

4 If a Dynamic Analysis is required, check the Dynamic Analysis box. Checking this box will display the Set Dynamic Parameters button. Clicking on this button will display the Dynamic Analysis Parameters dialog box.

In the Dynamic Analysis Parameters box:

- Type in the number of Modes
- Select the Type of Analysis, Eigenvectors or Ritz Vectors.
- If Eigenvector Analysis is selected, then Eigen Value parameters may be defined. The default values shown in the edit boxes are adequate for a majority of cases.
- Choosing Ritz Vector Analysis makes the Starting Ritz Vector dialog box available.
- Select Starting Ritz Load Vectors from the List of Loads. Use the ADD and REMOVE buttons to modify the list.
- Click OK

5 If P-Delta analysis is required, then check the Include P-Delta box. Checking this box will allow you to select the Set P-Delta Parameters button. Clicking on this button will display the P-Delta Parameters dialog box.

In the P-Delta Parameters dialog box:

- Enter maximum number of iterations or accept the default. A reasonable number is usually 5 or less.
- Enter displacement tolerance or accept the default. Failure to converge may result if too small a displacement is specified. The default tolerance is 0.001
- Enter force tolerance or accept the default.
- Define the load combination from existing Load Cases using appropriate scale factors. Use the Add, Modify and Delete buttons to set up this combination.
- Click OK.

6 If you want to have any analysis results saved to an output file, then check the Generate Output box . Checking this box will allow you to select the Select Output Options button. Click on this button to display the Select Output Results dialog box.

In the Select Output Results dialog box:

- Check the type of analysis results that you are interested in.
- Checking this box will allow you to select the Select/Show Loads button. You can press this button and select which load cases and load combinations to output information on for that category. You can not select the range of elements to include in the output.
- Click OK.

7 Specify the memory in Kilobytes (KB) The default memory size is 2000 kilobytes. For larger problems more memory may be necessary.

8 Clicking the OK button will save the analysis parameters and exit the form.

9 From the Analyze menu Click Run or Run minimized. Run minimized will perform the execution in the

background i.e will allow you to minimize SAP2000 while the analysis is being carried out.

Note: The program will seek the number of lowest frequency (longest period) Modes. Only the number of modes selected are available for any subsequent response-spectrum or time-history analysis processing. The number of modes is limited to the mass degrees of freedom.

Caution: P-Delta analysis in SAP2000 is an iterative analysis and the iterations do not include the change in geometry.

DISPLAY MENU

Display Options

Clicking on any of these buttons from the **Floating Toolbar** or some buttons from the Toolbar allows you to Display your selection with varied options.
Select various Display options by clicking as follows:

To Display	Do this
Undeformed geometry	Click on
Isometric view (toggle)	Click on
2-D View (XY, XZ or YZ plane)	Click on any one of the buttons to display an elevation view in that specific plane.
3-D View	Click on
Deformed Shape	Click on or Display menu and Show Deformed Shape
Element Stress Diagram	Click on or Display menu and Show Element Stress Diagram. Right click will show the details.
Joint, Frame or Shell Loads	Click on Display menu and Show Loads.
Patterns	Click on Display menu and Show Patterns
Tabular Input	Click on Display menu and Show Input Tables.
Mode Shapes	Click on or Display menu and Show Mode Shapes .
Tabular Output	Click on or Display menu and Show Output Tables .
Response Spectrum	Click on Display menu and Show Response Spectrum Curves .
Time History	Click on Display menu and Show Time History Traces .
Shrunken Elements (toggle)	Click on or click on and select shrink.
Selective Elements	Click on or View menu and Set Elements .

Graphical Output

What do you want to do?

- Display Undeformed Geometry
- Display Static Deformed Shape
- Display Mode Shape
- Display Member Force or Stress Diagram
- View Time History Results
- View Generated Response Spectra Curves
- Animate Deformed Shape
- View Real Time Animation for Time History Results
- Print Selected Graphical Output to a Printer or a File

Displaying Undeformed Geometry

On the Display menu, click Show Undeformed Shape, or click from the floating Toolbar.

Display Static Loads

To display graphically the loads on the structure:

1 On the Display menu, click Show Loads.., select the element type you are interested in. This will display the Show *member type* Loads dialog box.
2 In the dialog box:
    - Select the Load Case for the loads you want to see.
    - Select an item from the Load Type area.
    - If provided select to have the Load Values Shown.
3 Click OK to view the loads in the active window.

Display Joint Patterns

To display Joint Patterns :

1 On the Display menu, click Show Joint Patterns
2 In the Select Pattern dialog box, select the pattern you are interested in.
3 Click OK to view the load pattern in the active window.

Display Bridge Lanes

To graphically display lanes:

1 On the Display menu, click Show Lanes .
2 In the Show Lane dialog box,
    - Select the Lane you are interested in.
    - Check Show Eccentricity if you want to see the lane eccentricities.
3 Click OK to view the lane in the active window.

Also See:

Bridge Analysis

Display Input in Tabular Format

To view your input data as it appears in an S2K file or by printing Input Tables from the file menu:

1. On the Display menu, click Show Input Tables... Geometry Data or Loading Data.
2. In the Display Geometry/Loading Options dialog box
   - Select the Type of input information you are interested in.
3. Click OK to view the Input Table.
   - Under the File menu, in the form that appears, there are printing options
   - Close the form by clicking on the X in the top right hand corner of the form.

Display Static Deformed Shape

1. On the Display menu, click Show Deformed Shape..., or press  on the floating Toolbar. This will display the Deformed Shape dialog box.
2. In the Deformed Shape dialog box:
   - Select the Load Case or Combination Name from the Load drop down list.
   - Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the displacement vector components. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto option.
   - Checking the Wire Shadow box will also display the undeformed shape as a reference for comparison with the deformed shape.
   - Checking the Cubic Curve box will display the deformed elements with a cubic curve fit.
   - After selecting the options, click OK and the screen display will be updated.
3. Clicking the **Start Animation** button on the Status Line will animate the deformed shape of the model. Animation speed is controlled by + and - buttons next to the animation button

To view the displacement components for a single joint do the following:

- Right click once on the joint. The selected joint is highlighted and values are displayed in a floating window called the Joint Displacements window.
- Clicking on any other joint updates the display.
- Clicking anywhere else closes the floating window.

Caution: The deformed shape display for dynamic Load Cases is based on the absolute values of maximum displacements.

Note: Joint displacements can not be displayed during animation. To stop the animation, press the Stop Animation button.

Display Mode Shape

1. On the Display menu, click Show Mode Shape..., or click  on the floating Toolbar. This will display the Mode Shape dialog box.
2. In the Mode Shape dialog box:
   - Select the mode number by either typing it in the edit box or clicking on the spin button.
   - Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the vector components. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto option.
   - Checking the Wire Shadow box will also display the undeformed shape as a reference for comparison with the mode shape.
   - Checking the Cubic Curve box will display the deformed elements with cubic curve fit.
   - After selecting the options, click OK and the screen display will be updated.
3. Clicking the **Start Animation** button on the Status Line will animate the deformed shape of the model. Animation speed is controlled by a sliding button next to the animation button. Clicking on + and - buttons will change the display of the mode shape number by +1 or -1.

To view the normalized modal components for a single joint do the following:

- Right click once on the joint. The selected joint is highlighted and values are displayed in a floating window called the Joint Modal Components window.
- Clicking on any other joint updates the display.
- Clicking anywhere else closes the floating window.

Note: Modal components can not be displayed during animation. To stop the animation, press the Stop Animation button.

Animate Deformed Shape

1 Plot the desired deformed shape on the Screen.

2 Clicking the **Start Animation** button on the Status Line will animate the deformed shape of the model. Animation speed is controlled by a sliding button next to the animation button. Clicking on + and - buttons will change the Analysis Case for the display by +1 or -1.

Display Member Force or Stress Diagram

1 On the Display menu, click Show Element Force/Stresses, or click the **F** (Frame), **S** (Shell) or **J** (Joints) buttons on the Floating Toolbar. This will display the Member Force Diagram dialog box.

2 In the Member Force Diagram dialog box.

If **Frames** is selected

- Select the Load Case or Combination Name from the Load drop down list.
- Select the desired member force component, that is, Axial Force, Shear 2, Shear 3, Torsion, Moment 2-2, or Moment 3-3 .
- Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the Member Force diagrams. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto.
- Check on Fill Diagram to see the diagrams with colored fill.
- Check on Show Values on Diagram to print numerical values on the diagram.

If **Shells** is selected

- Select the Load Case or Combination Name from the Load drop down list
- Select output Type as Resultants or Stresses.
- Select the desired member force Component (If Resultants F11, F22, F12, FMAX, FMIN, FVM, M11, M22, M12, MMAX, MMIN, V13, V23, VMAX and if Stresses S11, S22, S12, SMAX, SMIN, SVM, S13, S23, SMAXV).
- Select the contour range.
- Select if the Stress should be averaged at joints.
- Select if the Stress contours to be displayed on deformed shape.

If **Joints** is selected

- Select the Load Case or Combination Name from the Load drop down list.
- Select either Reactions or Spring Forces

3 Click OK. The screen display will be updated with the options selected.

Hints: To display the details of a force diagram for a single member, right click on the member. This will open a floating window which displays the variation of the selected component over the length of the element If more than one element exists within the tolerance of the cursor, a list box of the elements is displayed. You can then select the desired element.

Move the pointer along the length of the member to view the Distance and Value at that point. Alternatively, click on the diagram in the floating window at the desired location. The magnitude of force component and associated location are displayed. The element ID number is displayed in this window.

Click anywhere in the main display window to close the Member Force Diagram window.

View Time History Results

Displaying of the time history results is a four-step process.

Step 1: Choose a Time History Case for which you want to see the traces.

Step 2: Select Joints or Elements for which you want to display Time History traces.

Step 3: Define Output Functions for Selected Joints or Elements and add to Display List.

Step 4: Plot Time History traces added to Display List.

What do you want to do?

» Display Input Time History Functions

» Display Joint Output Time History Trace

» Display Frame Element Force Output Time History Trace

» Display Shell Element Stress Output Time History Trace

» Display Structural Energy Time History Trace

» Display Base Function Time History Trace

Display Input Time History Functions

To display Input Time History Functions do the following:

1 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box.

2 To Add the Input Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Input Functions from the Add drop down list box.
   - In the Time History Input Function dialog box.
     - Select the Input Function from the Function Name drop down list box.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.

3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

» Plot the variations of Function against Time (F(t) vs. t)

» Change Line Type. Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

Display Joint Output Time History Trace

To display Joint Output Time History Trace by selecting the joint first, do the following:

1 Select the Joints for which you want to Display Time History traces.

2 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box with the Joint Labels listed as Functions.

3 To Modify an existing Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - In the Time History Function dialog box:
     - Click on a Function Name.
     - Click the Modify/Show TH Function button.
     - In the Time History Joint Function dialog box
       - Accept or modify the function name in the Function Name edit box.
       - Select the Vector Type (Displacement, Velocity, Acceleration, Absolute Acceleration, Reaction or Spring).

- Select the Vector Direction (X Translation, Y Translation, Z Translation, X Rotation, Y Rotation, Z Rotation).
- To recover the time history for a particular mode.
  - Select the Include One button.
  - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
- Click OK.

4 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box

To display Joint Output Time History Trace directly from the Time History Display Definition dialog box, do the following:

1 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box.

2 To Add a joint Function:

- Click the Define Functions button. This will display the Time History Function dialog box.
- Click on Add Joint Disps/Forces from the Add drop down list box.
- In the Time History Joint Function dialog box.
  - Enter the JointID.
  - A function name will automatically appear in the Function Name text box.
  - Select the Vector Type (Displacement, Velocity, Acceleration, Absolute Acceleration, Reaction or Spring).
  - Select the Vector Direction (X Translation, Y Translation, Z Translation, X Rotation, Y Rotation, Z Rotation).
  - To recover the time history for a particular mode:
    - Select the Include One button.
    - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
  - Click OK.
- The Function Name will be Added in to the Functions list box.

3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

Plot the variation of Function against Time (F(t) vs. t)

Plot the variation of Function against Function(F(t) vs. F(t))

Change Line Type, Line Color and Scale Factor for the Plots

Zoom in on a smaller area of the plot to see the details

Print Time History Plots or Tables

Display Frame Element Force Output Time History Trace

To display Frame Output Time History Trace by selecting the frame first, do the following:

1 Select the Frames for which you want to Display Time History traces.

2 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box with the Frame Labels listed as Functions.

3 To Modify an existing Function:

- Click the Define Functions button. This will display the Time History Function dialog box.
- In the Time History Function dialog box:
  - Click on a Function Name.
  - Click the Modify/Show TH Function button.
  - In the Time History Frame Function dialog box
    - Accept or modify the function name in the Function Name edit box.

- Select the Component (Axial Force, Shear 2, Shear 3, Torsion, Moment 2-2, Moment 3-3).
- Select the Location Station for the Frame Element.
- To recover the time history for a particular mode:
  - Select the Include One button.
  - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
- Click OK.

5 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box

To display Frame Output Time History Trace directly from the Time History Display Definition dialog box, do the following:

1 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box.
2 To Add a frame Function:
   - Click the Define Functions button This will display the Time History Function dialog box.
   - Click on Add Frame Element Forces from the Add drop down list box.
   - In the Time History Frame Function dialog box:
     - Enter the ElementID.
     - A function name will automatically appear in the Function Name text box.
     - Select the Component (Axial Force, Shear 2-2, Shear 3-3, Torsion, Moment 2-2, Moment 3-3).
     - Select the Location Station for the Frame Element.
     - To recover the time history for a particular mode:
       - Select the Include One button.
       - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
     - Click OK.
   - The Function Name will be Added in to the Functions list box
3 Click OK on the Time History Functions dialog box This will return focus back to the Time History Display Definition dialog box.

» Plot the variation of Function against Time (F(t) vs. t)

» Plot the variation of Function against Function(F(t) vs. F(t))

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

Display Shell Element Stress Output Time History Trace

To display Shell Output Time History Trace by selecting the shell first, do the following:

1 Select the Shells for which you want to Display Time History traces.
2 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box with the Shell Labels listed as Functions.
3 To Modify an existing Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - In the Time History Function dialog box:
     - Click on a Function Name.
     - Click the Modify/Show TH Function button.
     - In the Time History Shell Function dialog box
       - Accept or modify the function name in the Function Name edit box.
       - Select output Type as Resultants or Stresses (top or bottom).
       - Select the Component (If Resultants; F11, F22, F12,FMAX,FMAIN,FVM, M11, M22, M12,MMAX,MMIN, V13, V23,VMAX and if Stresses; S11, S22, S12, SMAX, SMIN, SVM, S13,

S23, SMAX)

- Select the Joint of the Shell at which the Resultants or Stresses are required from the Joint drop down list.
- To recover the time history for a particular mode:
  - Select the Include One button
  - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
- Click OK.

6 Click OK on the Time History Functions dialog box This will return focus back to the Time History Display Definition dialog box

To display Shell Output Time History Trace directly from the Time History Display Definition dialog box, do the following:

1 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box.

2 To Add a shell Function:

- Click the Define Functions button. This will display the Time History Function dialog box.
- Click on Add Shell Element Forces from the Add drop down list box.
- In the Time History Shell Function dialog box:
  - Enter the ElementID.
  - A function name will automatically appear in the Function Name text box.
  - Select output Type as Resultants or Stresses (top or bottom).
  - Select the Component (If Resultants; F11, F22, F12,FMAX,FMAIN,FVM, M11, M22, M12,MMAX,MMIN, V13, V23,VMAX and if Stresses; S11, S22, S12, SMAX, SMIN, SVM, S13, S23, SMAX).
  - Select the Joint of the Shell at which the Resultants or Stresses are required from the Joint drop down list.
  - To recover the time history for a particular mode:
    - Select the Include One button.
    - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
  - Click OK.
- The Function Name will be Added in to the Functions list box.

3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

» Plot the variation of Function against Time (F(t) vs. t)

» Plot the variation of Function against Function(F(t) vs. F(t))

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

Display Structural Energy Time History Trace

1 On the Display menu, click Show Time History Traces... This will display the Time History Display Definition dialog box.

2 To Add the Energy Functions:

- Click the Define Functions button. This will display the Time History Function dialog box.
- Click on Add Energy Functions from the Add drop down list box.
- In the Energy Functions dialog box:
  - Select the desired Energy Functions: Input Energy, Kinetic Energy, Potential Energy, MDamp Energy, NDamp Energy, NLLink , Energy Error .
  - Click OK.
- The Function Name will be Added in to the Functions list box.

3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

» Plot the variation of Function against Time (F(t) vs. t)

» Plot the variation of Function against Function(F(t) vs. F(t))

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

Display Base Function Time History Trace

1 On the Display menu, click Show Time History Traces.. This will display the Time History Display Definition dialog box.
2 To Add the Base Function:
   - Click the Define Functions button This will display the Time History Function dialog box.
   - Click on Add Base Functions from the Add drop down list box.
   - In the Base Functions dialog box:
     - Select the desired Base Functions: Base Shear X, Base Shear Y, Base Shear Z, Base Moment X, Base Moment Y, Base Moment Z.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box.

» Plot the variation of Function against Time (F(t) vs. t)

» Plot the variation of Function against Function(F(t) vs. F(t))

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

Display Group Summation Forces Time History

To display Group Summation Forces Time History Trace from the Time History Display Definition dialog box, do the following:

1 On the Display menu, click Show Time History Traces.. This will display the Time History Display Definition dialog box.
2 To Add a Group Summation Forces Function:
   - Click the Define Functions button. This will display the Time History Function dialog box.
   - Click on Add Group Summation Forces from the Add drop down list box.
   - In the Time History Group Function dialog box:
     - Select the Group name from the Group drop down list box.
     - A function name will automatically appear in the Function Name text box.
     - Select the Component Force or Moment in the three global directions.
     - To recover the time history for a particular mode:
       - Select the Include One button.
       - Enter the mode number directly, or, use the spin button up and down arrows to select from the list.
     - Click OK.
   - The Function Name will be Added in to the Functions list box.
3 Click OK on the Time History Functions dialog box. This will return focus back to the Time History Display Definition dialog box

» Plot the variation of Function against Time (F(t) vs. t)

» Plot the variation of Function against Function(F(t) vs F(t))

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

F(t) vs. t

To plot the variations of one function against time:

1 In the Time History Display Definition dialog box.

- Choose the F(t) vs. t Tab.
- From the drop down list box, select the Time History Case for which you want to display Input /Output Time History Traces.
- Click on the Function name to be plotted in the List of Functions box.
- Click on the Add button. The function name will be added to the Plot Functions list box.
- Repeat to include other function names (if any) for simultaneous display of several functions
- Use the Remove button to remove a function from the Plot Functions list box.
- Use the Show button to display the function's attributes (e.g. Vector Type, Direction etc.)
- Enter the time range in the From and To edit boxes. By default, the full time range is selected.
- Enter the axis labels in the Horizontal and Vertical edit boxes in the Axis Labels area.
- Removing the check mark from the Grid Overlay box will disable the generation of a grid through the major divisions of the axes.

3 Click on Display. This will open a floating window labeled Time History Traces in which functions are graphically displayed. A small Legend window displays the maximum and minimum numerical values for the functions and another small window just below it displays the coordinates of the pointer.

» Plot the variations of Function against Time (F(t) vs. F(t))

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

» Print Time History Plots or Tables

F(t) vs. F(t)

To plot the variations of one function against another function:

1 In the Time History Display Definition dialog box:

- Choose the F(t) vs. F(t) Tab
- From the drop down list box, select the Time History Case for which you want to display Input /Output Time History Traces
- Select the horizontal time function from the drop down list in the Choose Functions area.
- Select the vertical time function from the drop down list in the Choose Functions area.
- Enter the time range in the From and To boxes By default, the full time range is selected.
- Enter the axis labels in the Horizontal and Vertical edit boxes in the in the Axis Labels area.
- Removing the check mark from the Grid Overlay box will disable the generation of a grid through the major divisions of the axes.

4 Click on Display. This will open a floating window labeled Time History Traces in which functions are graphically displayed. A small Legend window displays the Function names for the Horizontal and Vertical axes together with their attributes and another small window just below it displays the coordinates of the pointer.

» Plot the variations of Function against Time (F(t) vs. t)

» Change Line Type, Line Color and Scale Factor for the Plots

» Zoom in on a smaller area of the plot to see the details

Scale Factor, Line Type and Color

To change the **Line Type**

- Click on the function name in the Plot Functions list box.
- Select the line type to be Solid, Dashed or Dotted from the Line Options area.

To change the **Line Color**

- Click the Color button in the Line Color area This will display the Color dialog box.
- Click on the desired color in the palette.
- Click OK.

To change the **Scale Factor**

The Scale Factors default to 1.

- Check the Horizontal/Vertical direction for which you want to change the Scale Factor.
  - Press the Define button that appears.
  - Enter in the Horizontal/Vertical Scale for the Maximum and Minimum Scale Factor in the appropriate edit box.
  - Click OK

Note: Maximum and Minimum Scale Factors of zero mean that the default scale factors are used.

Zoom into the Plot

To zoom on a specific part of the plot:

- Point to a corner of the region, hold down the mouse button and drag to define the rectangular zoom region, and release the mouse button.
- Clicking anywhere in the main display area brings back the full view.

Print Time History or Response Spectrum Plots or Tables

To print the **Plots** display do the following:

- From the Print menu, choose 'Print Graphics'.

To print the **Tables** (results in a numerically tabulated form, in an ASCII format,) do the following:

- From the Print menu, choose 'Print Tables'.

To save the **Tables** in a File, do the following:

- From the Print menu, choose 'Save Tables as File...'. This will display the Save File As dialog box
- In the Save File As dialog box:
  - Select the drive and the directory in which the file is to reside.
  - Enter a new filename or overwrite an existing file.
  - Click the OK button.

Clicking anywhere else closes the floating window.

View Generated Response Spectra Curves

1 Select the Joint for which you want to Display Response Spectrum Curve.

2 On the Display menu, click Show Response Spectrum Curves... This will display the Response Spectrum Generation dialog box. The dialog box has five Tabs labeled **Define**, **Axes**, **Options**, **Frequency** and

Damping. Clicking on any Tab highlights it.

- The Define Tab
- The Axes Tab
- The Options Tab
- The Frequency Tab
- The Damping Tab

After making the necessary changes in all the Tabs in the Response Spectrum Generation dialog box:

3. Click on Display to open a floating window labeled Response Spectrum Curves in which the response spectra are graphically displayed. A small Legend window displays the Joint number Direction, Scale Factor, Widening percentage and Damping Ratios color coded to correspond to the Spectra Curves Another small window just below it displays the coordinates of the pointer
4. Click on Done to save the changes made in all the Tabs and exit the Response Spectrum Generation dialog box.

- Zoom in on a smaller area of the plot to see the details
- Change Line Type, Line Color and Scale Factor for the Plots
- Print Response Spectrum Plots or Tables

Spectrum Generation Define Tab

In the **Define Tab**:

- From the drop down list select the Time History Case for which you want to generate a Response Spectrum.
- Click on a Joint in the **Choose a Joint** list and then click on the appropriate direction X, Y or Z from the **Vector Direction** option adjacent to it to define the direction for which you want to generate a spectrum. By default the X (global X-direction) is checked.

Spectrum Generation Axes Tab

In the **Axes Tab**:

- Select the **Abscissa** to show as either **Frequency** or **Period.**
- Select the **Ordinate** to be one of the five choices i.e. **SD** (Spectral Displacement), **SV** (Spectral Velocity), **PSV** (Pseudo Spectral Velocity), **SA** (Spectral Acceleration) and **PSA** (Pseudo Spectral Acceleration)

Spectrum Generation Options Tab

In the **Options Tab:**

- Set the **Abscissa** scale. By default **Log** is checked
- Changing the percentage of **Spectrum Widening** will change the peaks of response spectra curves. The default is set to zero.
- Set the **Ordinate** scale. By default **Arithmetic** is checked.
- Changing the ordinate scale factor will scale the spectral values The default is 1.0.
- Removing the check mark from the Grid Overlay box will disable the generation of a grid through the major divisions of the axes.

Spectrum Generation Frequency Tab

In the **Frequency Tab:**

- Removing the check mark from of the Default box will exclude the built-in set of frequencies in the generation of response spectrum.

- Removing the check mark from of the Structural box will disable merging of the structural frequencies into the set of frequencies used in the generation of response spectrum.
- To enter user-defined frequencies do the following:
  - Check the User box in the Include Frequencies area.
  - Enter the value in the User Frequencies edit box.
  - Click the **Add Value** button
  - Repeat to include all the frequencies.
  - Use the **Change Value** and **Delete Value** buttons to edit or delete the values included.

Spectrum Generation Damping Tab

In the **Damping Tab:**

- Use this option to add a new damping value or edit the default damping values.
- To Add a new damping value do the following:
  - Type in a new value in the edit box.
  - Click the **Add Value** button.
- To change one of the default damping values do the following:
  - Click on the damping value in the Damping values list box.
  - Change the value in the edit box.
  - Click the **Change Value** button.
- To remove one of the values from the set do the following:
  - Click on the damping value.
  - Click the **Delete Value** button.

Display Group Joint Force Sums

The option is an easy way to find the sum of forces and moments on a Group of joints.

1 On the Display menu, click Show Group Joint Force Sums. This will display the Select Groups dialog box.
2 Select the Group or Groups for which you want the sum.
3 Click OK to view in tabular form the sum of the shears and moments in the group.
4 You can print the table from the File menu.
5 When finished, close the table by clicking on the close button at the top right of the form.

Note: The group for the Group Sum must be selected carefully. The group must consist of a set of joints and the elements directly connected to only one side of them. For example if the base shear for a structure is required, the group should consist of the joints at the base of the structure and the frame/shell elements above them that are directly connected to them.

Display Joint Influence Lines

The influence lines can be displayed for any joint displacement, reaction or force component due to a unit load on a defined Bridge Lane in the structure.

1 On the Display menu, click Show Influence Lines ...Joints. This will display the Show Joints Influence Line dialog box.
2 Select the Lane for which you want to see influence lines.
3 Select the Joint for which the results are reported.
4 Select the Vector Type as Displacement, Spring Force or Reaction. There may be no influence lines for some of these Vector Types, depending on the structural configuration.
5 Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the diagrams. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto.
6 Pressing the Table button will show the influence line data points in tabular form. The table includes the Lane name, Frame name, Location relative to the starting point of the lane, Location relative to the i end of

the frame member, and the influence line value.

7 Click OK to view the Influence line or Cancel to close the dialog box without viewing the influence lines.

Display Frame Influence Lines

The influence lines can be displayed for any Frame moment, shear, torsion or axial load component due to a unit load on a defined Bridge Lane in the structure.

1 On the Display menu, click Show Influence Lines ...Frames. This will display the Show Frames Influence Line dialog box.
2 Select the Lane for which you want to see influence lines.
3 Select the Frame for which the results are reported
4 Select the Component as Moment, Shear, Torsion or Axial Load. There may be no influence lines for some of these Component, depending on the structural configuration.
5 Select the Station (Output Segment) for which you want the influence lines.
6 Select the Scaling method used. Selecting Auto will automatically set the scale factor. Selecting Scale Factor lets the user scale the diagrams. If the Auto was selected previously, then the scale factor text edit box will show the scale factor used by the Auto.
7 Pressing the Table button will show the influence line data points in tabular form. The table includes the Lane name, Frame name, Location relative to the starting point of the lane, Location relative to the i end of the frame member, and the influence line value.
8 Click OK to view the Influence line or Cancel to close the dialog box without viewing the influence lines.

Display Joint or Member Text Output on Screen

1 On the Display menu, click Set Output Table Mode..., or click on the Floating Toolbar. This will display the Select Output dialog box.
2 In the Select Output dialog box.
   - Select the Load Cases and/or the Load Combinations labels for which the output is to be displayed in a tabulated form Loads may be selected by clicking on the Load labels in the list box. For selecting a range of loads, click and drag, or for multiple loads hold down the Ctrl Key and click on different load labels.
   - Click OK to close the Select Output dialog box.
   - To display the member force, displacement or reaction output table, right click on the element or joint of interest. This will open a floating window which displays the output table. If more than one element or joint exists within the tolerance of the cursor, a list box of the elements and joints is displayed. You can then click on the desired element or joint.
   - Click anywhere in the main display window to close the output table window.

DESIGN MENU

Steel Design

On the Design menu, click Steel Design. This will switch the program into the Steel Design mode.

What do you want to do?

- Define Section Properties
- Setting Steel Design Parameters
- Group Elements for Design
- Select/Redefine Design Load Combinations
- Overwrite Element Design Properties
- Replace Auto with Optimal Sections
- Display Steel Stress Check/Design Ratios on Screen
- Display Steel Stress Check/Design Input on Screen
- Review Steel Stress Check/Design Detail
- Redesign a Steel Member
- Start Design/Check of Structure
- Update Analysis Sections
- Print Design Results to a Printer or a File
- Defining Elements as a single Member for bending

Concrete Design

On the Design menu, click Concrete Design. This will switch the program into the Concrete Design mode.

What do you want to do?

- Define Section Properties
- Setting Concrete Design Parameters
- Select/Redefine Design Load Combinations
- Overwrite Element Design Properties
- Display Concrete Design/Check Output on Screen
- Display Concrete Design/Check Input on Screen

» Review Concrete Design/Check Detail

» View Interaction Diagram for a Concrete Column

» Redesign a Concrete Member

» Start Design/Check of Structure

» Update Analysis Sections

» Print Design Results to a Printer or a File

» Defining Elements as a single Member for bending

Group Elements for Design

Grouping elements for design will design all the elements in the group to the same section or steel reinforcement area. Using the section or reinforcing area that will work for all of the members in the group.

NOTE: Grouping elements for design currently only works on steel members with Auto Select sections assigned to them

1 On the Design Menu, click Select Design Groups... This will display either the Steel Design Group Selection or Concrete Design Group Selection dialog box.

2 In the Steel or Concrete Design Group Selection box:

- Select previously defined Groups from the List of Groups
- Click on the Add button to add Groups to the Design Groups list
- Clicking on the Remove button will remove the selected Groups from the Design Groups list
- Click OK

Start Design/Check of Structure

1 Select the Frame Elements to be designed or checked

2 On the Design Menu, click Start Design/Check of Structure ..

3 This will immediately start the stress check or design of all the selected frame members in the model.

4 The ratio of actual to allowable stress/capacity or member design reinforcement, based upon the previously selected combos, will be displayed.

5 Subsequent member selections can be Designed/Checked and added to the display.

6 The Check/Design results display will only affect the selected elements and all other elements will show the previous values, if any.

Note: If no Elements are selected then the program will automatically select all Frame Elements and perform the Design or Check.

Selecting Design Load Combinations

1 On the Design Menu, click Select Design Combos... This will display the Design Load Combinations Selection dialog box.

2 In the Design Load Combinations Selection box:

- Select previously defined Combos from the List of Combos
- Click on Add button to add Combos to the Design Combos list
- Clicking on Remove button will remove the selected Combos from the Design Combos list
- Click OK

Note: The Design Load Combinations Selection menu lists a series of default load combinations labeled DCON# for Concrete design and DSTL# for Steel design. The number and type of default combinations depends upon the static Load Cases previously defined. Other user defined Load Combos that were indicated as design combos in the Load Combination Data dialog box will also be included in the Design Combos

If other load combinations have been defined and were not indicated as design combos in the Load Combination Data dialog box, they are shown under List of Combos. The Design Combos list can be modified by removing one or more default Combinations and/or by adding from the List of Combos.

Clicking on the Show button will present the Load Cases and Scale factors for the highlighted Combo.

Overwrite Element Design Data

1 Select the Elements for which you want to overwrite assignments.
2 On the Design Menu, click ReDefine Element Design Data. . This will display the Element Overwrite Assignment menu.
3 In the Element Overwrite Assignments dialog box·
   - Check the desired Element assignments to be overwritten by clicking on the relevant check boxes for K-factors, Unbraced Lengths, Cm, Cb, Live Load Reduction Factor or Magnification Factors
   - Provide the new value in the edit boxes
   - Change the Element Section by clicking on the Change button
   - If Concrete sections are selected, choose the Element Types to be Special, Intermediate or Ordinary.
   - If Steel sections are selected, click the Overwrite Allowable Stresses button to overwrite axial, bending and shear allowable stresses.
   - Click the OK button

Note: Entering a value of **Zero** sets the value back to the program default.

The Element Overwrite menu lists a series of assignments, which can be overwritten. The selection options are mutually exclusive and may be selected in any order.

Selecting and modifying the appropriate factor will overwrite the default or computed value.

The new value(s) are displayed in the active window.

The Effective Length Factors K33 (major) and K22 (minor) can be any positive reasonable number.

The Unbraced Length Ratios L33 (major) and L22 (minor) are to be input as a positive ratio of the full length, i.e. 0.33 0.75, etc.

The bending coefficient Cm can be any positive number less than or equal to 1.0.

The moment gradient coefficient Cb can be any positive number from 1.0 to 2.3.

The Live Load Reduction Factor is a positive number between 0 and 1.0 and represents a fraction of the full live load previously input.

The Non Sidesway and Sidesway Magnification Factors DB33, DB22, DS33, DS22 are numbers greater than or equal to 1.0 If P-delta is carried out DS33 and DS22 may be set to 1.0

Tip: Element Class specification (Column or Beam) for Concrete Sections is done at the Section definition level.

Defining Elements as a Single Member for Bending

It is often practical to divide structural members into multiple elements when developing your computer model. However for design it is important to know the actual member start and end point. This information is used for defining un braced length in the major and minor access bending, Cb and Cm values where applicable as well as the provisions relating to the distribution of moment and shear along a member for Special and Intermediate moment resisting frames.

SAP2000 provides various ways to enter this information for design. L22,L33 Cb and Cm can be defined directly for each element in the model using the Element Overwrite Assignments dialog box. This will however not supply the design module with the distribution of moment and shear along a member.

To Define a set of frame elements to be part of a single Frame Member for bending in the major and/or minor direction:

1 Select the frame elements that you want to assign as a single Member for bending in the major and/or minor direction.
2 On the Design Menu, click ReDefine Element Design Data... This will display the Element Overwrite Assignment menu.
3 In the Element Overwrite Assignments dialog box:

- Check Define Member for Major Bending to assign the selected frame elements to be part of the same frame member for major axis bending.
- Check Define Member for Minor Bending to assign the selected frame elements to be part of the same frame member for minor axis bending.
- Click the OK button.

Note: Values overwritten explicitly in the Element Overwrite Assignments dialog box will take precedence over the values calculated by the defined bending members. All elements selected as a part of the defined Bending Member must be connected. Only one Member can be defined at one time.

Replace Auto with Optimal Sections

This selection will replace the "AUTO" sections with the optimally section chosen from the auto select group. This is an irreversible action i.e. once the AUTO sections are replaced they will no longer be available for optimal design. You will have to re-assign them as AUTO if you want them included in Optimization.

1. Select the frame members that you want to replace the analysis sections for.
2. On the Design Menu, click Replace Auto w/Optimal Sections.
3. This will issue a warning message that Replacing Auto-Selected Sections will change the analysis model and cause analysis results to be deleted.
4. Click OK if acceptable otherwise click Cancel.

Note: This option should only be used after the final design cycle. Use Update Analysis Sections to update the sections chosen at the design/redesign level as it will update the sections properties for analysis without changing the sections from being "AUTO".

Display Design Input/Results on Screen

What do you want to do?

» Display Steel Stress Check/Design Input on Screen

» Display Steel Stress Check/Design Ratios on Screen

» Display Concrete Design/Check Input on Screen

» Display Concrete Design/Check Output on Screen

Display Steel Stress Check/Design Ratios on Screen

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose either P-M ratios or Shear Ratios to be displayed.
4. Click OK. This will refresh the window and display the Stress check Ratios.

Display Concrete Design/Check Output on Screen

1. On the Design Menu, click Display Design Info...
2. This will display the Display Design Results box.
3. In the Display Design Results box, click on Design Output, then from the drop down list box choose either Longitudinal Reinforcement or Shear Reinforcement to be displayed.
4. Click OK. This will refresh the window and display the area of Reinforcement.

Display Steel Stress Check/Design Input on Screen

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results, click on Design Input, then from the drop down list box you may choose Design Sections, K-factors, Effective Length Ratios, Cm Factors, Cb Factors, Live Load Reduction Factors, DS Factors, DB Factors, Design Type, Axial Allowables, Bending Allowables or Shear Allowables to be displayed.
4 Click OK This will refresh the window and display the input parameter selected.

Display Concrete Design/Check Input on Screen

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results, click on Design Input, then from the drop down list box you may choose Design Sections, K-factors, Effective Length Ratios, Cm Factors, Cb Factors, Live Load Reduction Factors, DS Factors, DB Factors or Design Type to be displayed.
4 Click OK This will refresh the window and display the input parameter selected.

Review Stress Check/Design Details

What do you want to do?

» Review Steel Stress Check/Design Detail

» Review Concrete Design/Check Detail

Review Steel Stress Check/Design Detail

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results box, click on Design Output, then from the drop down list box choose either P-M ratios or Shear Ratios to be displayed.
4 Click OK. This will refresh the window and display the Stress Check Ratios.
5 Right Click on a member for which you want see the Design Details. This will display the Steel Stress Check Information box with a breakdown of the Stress Ratio (Axial, Major and Minor Bending, and Major and Minor Shear).
6 In the Steel Stress Check Information box, click Details to see more details (Actual Stress, Allowable Stress, Cm, Cb, Fe' etc.) pertaining to the calculation of the Stress Ratio.

Review Concrete Design/Check Detail

1 On the Design Menu, click Display Design Info...
2 This will display the Display Design Results box.
3 In the Display Design Results box, click on Design Output, then from the drop down list box choose either Longitudinal Reinforcement or Shear Reinforcement to be displayed.
4 Click OK. This will refresh the window and display the area of Reinforcement.
5 Right Click on a member for which you want see the Design Details. This will display the Concrete Design

Information box with Flexural and Shear Reinforcement areas listed (Top Reinforcement, Bottom Reinforcement and Shear Reinforcement for Beams and Longitudinal and Major and Minor Shear Reinforcement for Columns).

6 In the Concrete Design Information box, click Details to see more details (Vn, Vc, Vs, Cm Major and Minor Magnification factors etc.) pertaining to the calculation of the Reinforcement for each of the load combinations.

View Interaction Diagram for a Concrete Column

1 On the Design Menu, click Display Design Info...

2 This will display the Display Design Results box.

3 In the Display Design Results box, click on Design Output, then from the drop down list box choose Longitudinal Reinforcement to be displayed.

4 Click OK. This will refresh the window and display the area of Longitudinal Reinforcement.

5 Right Click on a Column for which you want to see the Interaction Diagram. This will display the Concrete Design Information box.

6 In the Concrete Design Information box, click Interaction. This will display the interaction diagram. Click on pxy to see the 3-D interaction, xy to see the interaction between Mx and My, px to see the interaction between P and Mx and py to see the interaction between P and My.

7 Use the spin buttons to change the Plan and Elevation angles.

8 To zoom on a specific part of the plot:

- Point to a corner of the region, hold down the mouse button and drag to define the rectangular zoom region, and release the mouse button.
- Clicking anywhere in the main display area brings back the full view.

9 Use the Print menu to print Interaction Cures.

Redesign a Steel Member

1 On the Design Menu, click Display Design Info...

2 This will display the Display Design Results box.

3 In the Display Design Results box, click on Design Output, then from the drop down list box choose either P-M ratios or Shear Ratios to be displayed.

4 Click OK. This will refresh the window and display the Stress Check Ratios.

5 Right Click on a member for which you want see the Design Details. This will display the Steel Stress Check Information box with a breakdown of the Stress Ratio (Axial, Major and Minor Bending, and Major and Minor Shear).

6 In the Steel Stress Check Information box, click Redesign. This will display the Element Overwrite Assignments box.

7 Make appropriate changes to the assignments and click OK. This will bring you back to the Steel Stress Check Information box with the redesign carried out and the new results displayed in the design information box.

Redesign a Concrete Member

1 On the Design Menu, click Display Design Info...

2 This will display the Display Design Results box.

3 In the Display Design Results box, click on Design Output, then from the drop down list box choose either Longitudinal Reinforcement or Shear Reinforcement to be displayed.

4 Click OK. This will refresh the window and display the area of Reinforcement.

5 Right Click on a member for which you want see the Design Details. This will display the Concrete Design Information box with Flexural and Shear Reinforcement areas listed (Top Reinforcement, Bottom Reinforcement and Shear Reinforcement for Beams and Longitudinal and Major and Minor Shear Reinforcement for Columns).

6 In the Concrete Design Information box, click Redesign. This will display the Element Overwrite Assignments

box.

7 Make appropriate changes to the assignments and click OK. This will bring you back to the Concrete Design Information box with the redesign carried out and the new results displayed in the design information box.

Update Analysis Sections

SAP2000 allows you to change section properties while in the Design Mode (Steel or Concrete) and redesign the member. Redesign uses the analysis results from the original properties unless you update analysis section properties and re-run the analysis It is advisable to use the Update Analysis Section option after you have reviewed the entire design as the analysis results will no longer be available after the properties are updated.

1 Select the frame elements that you want to update.
2 On the Design Menu, click Update Analysis Sections...
3 This will issue a warning message that Updating Analysis Sections will change the analysis model and cause analysis results to be deleted.
4 Click OK if acceptable, otherwise click Cancel.

Reset Design Sections

SAP2000 allows you to change section properties while in the Design Mode (Steel or Concrete) and redesign the member. Some times you may find that you want to revert to having the analysis section used for design.

1 Select the frame elements that you want to update.
2 On the Design Menu, click Reset Design Sections...
3 This will reset the same sections used in the analysis for design.

OPTIONS MENU

Preferences

SAP2000 has some built-in Tolerances and default values for some parameters and Design Codes. It is possible to change most of those default values by editing them in the three Preference Tabs, namely Dimensions, Steel and Concrete Design Coeds and parameters.

1 On the Options menu, click Preferences... This will display the Preferences dialog box with three Tabs namely Dimensions, Steel and Concrete.
2 Click on the appropriate tab to modify the default options.
3 Click OK.

Also see:

»| Setting Steel Design Parameters

»| Setting Concrete Design Parameters

Setting Steel Design Parameters

1 Select Preferences from the Options menu.
2 Press on the Steel Tab to view the Steel Design Preferences .
6 From the Steel Design Code drop down list box select the code you want used in steel design.
7 If the Section Properties file is not the one you want used in the design then press the Choose File button to select another file.
8 Select the preferred design method for multi valued load cases if you have any in the design.
   - The envelope option will design for the maximum and minimum values in a member.
   - The Time Step option will check the section for each time step of a Time History analysis. This method is very time consuming and should be used mainly for the final design.
   - The Max/Min Correspondence will design for the maximum and minimum moment/ axial force and its corresponding axial force/moment.
9 Click on OK when finished.

Setting Concrete Design Parameters

1 Select Preferences from the Options menu.
2 Press on the Concrete Tab to view the Concrete Design Preferences .
3 From the Concrete Design Code drop down list box select the code you want used in concrete design.

4 Edit any Strength reduction factors that may not be satisfactory.
5 Edit the Interaction Diagram Parameters that are used to develop for the column design.
6 Select the preferred design method for multi valued load cases if you have any in the design.
   - The envelope option will design for the maximum and minimum values in a member.
   - The Time Step option will check the section for each time step of a Time History analysis This method is very time consuming and should be used mainly for the final design.
   - The Max/Min Correspondence will design for the maximum and minimum moment/ axial force and its corresponding axial force /moment.
7 Click on OK when finished

Setting Colors

In SAP2000 you have the ability to change the colors used by the program for both viewing in the graphic interface and for printing to a color or monochrome printer.

1 On the Options menu, click Colors... This will display the Assign Colors dialog box with a Display and Output tab.
   - Under the Display tab you can set colors for viewing the structural configuration including: Joints, Frames Shells, Grids and the Background.
   - Under the Output tab you can set the colors for the analysis and design results.
2 On either form you can choose to set the colors for the Screen and the Printer.
   - Select Screen or Printer.
   - Change the colors by pressing the appropriate button under the Display tab or by double clicking on the appropriate color under the Output tab.
   - Then select a new color and press the OK button.
3 If you want to set the colors back to the SAP2000 default values, press the Reset button.
4 Click OK.

Note: The Default printer colors are set for a monochrome printer. To print in color you will need to manually select the colors you want for the printer.

Editing Toolbars

Both the Toolbar and the Floating Toolbar can be edited to suit the way you work with SAP2000.

1 Double click on the Toolbar or Floating Toolbar away from a button, e.g. between two buttons or to the right of the last button
2 Add and Remove buttons as required.
3 Click on Move Up and Move Down to locate the buttons as desired.
4 The Reset button sets the Toolbar to its default settings.
5 Press the Close button when finished.

User Defined Coordinate System

The default coordinate system is the global coordinate system. All user defined coordinate systems are defined relative to this global coordinate system All coordinate systems follow the right-hand rule.

Also See:

»| Defining a new Coordinate System

FINAL NOTES

General Tips

1 You can view the time history displacements in real time video via the SAP2000 video creation option.
2 You can use the new nonlinear link element to model base isolators and dampers.
3 You can animate deformed shapes and mode shapes in 3-d perspective by a simple click of the 'Start Animation' button.
4 You can animate shell structures displaying stress contours with the corresponding deformed shapes in full in 3-d perspective
5 You can automatically break frame members into elements at their intersection points in 3-d space.
6 You can automatically generate a macro shell element by a single click on the working grid and then mesh it into finer elements with simple meshing parameters.
7 Large models can be generated automatically using the SAP2000 templates. The models can then be sculptured with on screen editing to satisfy specific situations.
8 In SAP2000 you only define elements All joints needed by the elements are automatically generated. No pre-definition of joints is needed.
9 You can display information about any joint or element by pointing to it and clicking the right mouse button"
10 The SAP2000 model generation features allow on screen linear and cylindrical replication and mirroring of selected portions of the model.
11 Joints can be glued to a grid line. Moving the grid line will modify the location of all joints on the grid line and stretch or shrink all elements connected to those joints.
12 You can double click a grid line to bring up an edit box that will allow you to accurately reposition the grid line.
13 You can unlock the gridlines and drag and drop them on the screen to new locations.
14 You can add a new grid line by holding the CTRL-KEY down and clicking on an existing grid line and dragging away a duplicate
15 Clicking on a view with the CTRL-KEY down will give you a list of the joints and elements within the click tolerance.
16 The SAP2000 nonlinear time history analysis is based upon a new special purpose algorithm called the Wilson-FNA method
17 You can produce scaled perspective extrusions of the structural sections by simply activating the SAP2000 'extrude' option.
18 You can copy a selected three dimensional region of your model to the clipboard and paste it back at any location in 3d space.
19 While displaying mode shapes or deformed shapes the vector being displayed can be instantaneously changed with the '+/-' buttons that will appear at the bottom of the screen.
20 You can switch units at any time while creating the model.
21 Back in 1970, with the first release, the slang name SAP was selected to remind users that this program, like all computer programs, lacks intelligence and that it is the responsibility of the engineer to use the program correctly.

22 'With good engineering judgment you can produce on the back of an envelope that which otherwise cannot be produced with a ton of computer output'. – Anonymous Circa 1974

Specific Tips

SAP2000 Versions and Limitations

SAP2000 may be purchased in the following four versions:

SAP2000 Standard

Capacity	1500 Nodes
Analysis	Static and Dynamic Response Spectrum Analysis
Elements	FRAME and SHELL Elements only
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

SAP2000 Plus

Capacity	No Practical Limit
Analysis	Static, Dynamic Response Spectrum, Time History and Bridge Analysis
Elements	FRAME, SHELL, PLANE, SOLID and ASOLID Elements
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

SAP2000 Nonlinear

Capacity	No Practical Limit
Analysis	Static, Dynamic Response Spectrum, Time History, Bridge and Dynamic Nonlinear Time History Analysis
Elements	FRAME, SHELL, PLANE, SOLID, ASOLID, and NLLINK (as external Damping , Base Isolators, Gap and Hook Elements)
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

SAP2000 Educational

Capacity	30 Nodes
Analysis	Static, Dynamic Response Spectrum, Time History, Bridge and Dynamic Nonlinear Time History Analysis
Elements	FRAME, SHELL, PLANE, SOLID, ASOLID, and NLLINK (as external Damping, Base Isolators, Gap and Hook Elements)
Design	Steel (AISC-ASD89, AISC-LRFD93, BS5950 90, CISC 95, EUROCODE 3-1992), Concrete (ACI 318-95, BS8110 89, CAN3-A23.2-M84, EUROCODE 2-1991)

Error Messages

Phone and Fax Support

Support is available, free of charge, for 90 days after purchase and for the term of the maintenance agreement. Please call CSI's offices to inquire about the maintenance agreement.

Standard phone and fax support is available in the United States, from CSI support engineers via a toll call between 8:30 A.M. and 5:00 P.M. Pacific time, Monday through Friday, excluding holidays.

If you are experiencing problems using the software:

- Consult the documentation and other printed information included with the software.
- Check the online Help.

If you cannot find a solution then:

- Contact CSI's Offices via **phone** at (510) 845-2177.
- Send a **fax** with questions and a sketch of the model to CSI at (510) 845-4096.
- You may also send the input file via e-mail to support@csiberkeley.com .

When you call, you should be at your computer and have the appropriate documentation (User's manual) at hand. Please have the following information:

- The version number of the program that you are using.
- The hardware configuration (computer make, operating system, hard disk size and RAM size).
- The exact wording of error messages that appeared on your screen.
- A description of what happened and what you were doing when the problem occurred.
- A description of how you tried to solve the problem.

When you send a Fax or e-mail, include the following information:

- Your name, Company name, and phone number.
- The version and version number of the program that you are using
- The hardware configuration (computer make, operating system, hard disk size and RAM size).
- The exact wording of any error messages that appeared on your screen.
- A full description of what you are having problem with.
- A full description of how you tried to solve the problem.
- An attachment including your model and any other pertinent data files (For e-mail only!)

Online Support

Online support is available by:

Sending an e-mail and your model file to support@csiberkeley.com

Visiting CSI's web site at http://www.csiberkeley.com (frequently asked questions)

When you send a Fax or e-mail, include the following information:

- Your name, Company name, and phone number.
- The version and version number of the program that you are using.
- The hardware configuration (computer make, operating system, hard disk size and RAM size).
- The exact wording of any error messages that appeared on your screen.
- A full description of what you are having problem with.
- A full description of how you tried to solve the problem.
- An attachment including your model and any other pertinent data files. (For e-mail only!)

Frequently Asked Questions

Q. ***Why do 2-D views show objects which are not on that plane?***

R The length tolerance, given as Auto-merge tolerance in the Dimension tab of Preferences in the Options pull down menu may be too large. The default value is 0.1 inches. The value of this tolerance determines the clipping thickness of the 2-D view plane and all joints within that clipping depth will be available on that plane.
Caution: This length tolerance is also used to merge together joints automatically generated.

Program Manuals

1 **SAP2000 Volume I**

This volume describes all of the theoretical concepts behind the modeling and analysis features offered by the SAP2000 structural analysis program. The focus of this manual is on the analysis portion of the program.

2 **SAP2000 Volume II**

This volume describes the contents of SAP2000 text input data file. Most users can skip this volume.

3 **SAP2000 Getting Started**

This manual is designed to help you become quickly productive with SAP2000. It gives an introduction to the basic concepts of the graphical user interface and includes a tutorial on using the program. It is strongly recommended that you read this manual and work the tutorial before attempting a real project with SAP2000.

4 **SAP2000 Verification Manual**

This manual contains example problems solved using SAP2000. The problems demonstrate many of the capabilities of SAP2000. For purposes of verification, key results from the examples are compared with theoretical or published results.

5 **SAP2000 Steel Design Manual / SAP2000 Concrete Design Manual**

This manual includes a steel/ concrete design tutorials and description of the implementation of the design coeds in SAP2000.

6 **SAP2000 Tutorial Manual**

This manual has various tutorial to help users make the most efficient use of SAP2000.

Primers

Newsletters

Text Books

1 **Three Dimensional Analysis of Structures**
With Emphasis on Earthquake Engineering
Edward L. Wilson

If you are interested in obtaining a copy of this book, contact CSI for cost and shipping.

References

AASHTO, 1992

Standard Specifications for Highways Bridges, 15th Edition, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

AASHTO, 1994

LRFD Bridge Design Specifications, Customary U.S. Units, 1st Edition, The American Association of State Highway and Transportation Officials, Inc., Washington, D.C.

ACI, 1995

Building Code Requirements for Structural Concrete (ACI 318-95) and Commentary (ACI 318R-95), American Concrete Institute, Farmington Hills, Mich.

AISC, 1994

Manual of Steel Construction, Load & Resistance Factor Design, 2nd Edition, American Institute of Steel Construction, Chicago, Ill.

K. J. Bathe, 1982

Finite Element Procedures in Engineering Analysis, Prentice-Hall, Englewood Cliffs, N.J.

K. J. Bathe and E. L. Wilson, 1976

Numerical Methods in Finite Element Analysis, Prentice-Hall, Englewood Cliffs, N.J.

K. J. Bathe, E. L. Wilson, and F. E. Peterson, 1974

SAP IV — A Structural Analysis Program for Static and Dynamic Response of Linear Systems, Report No. EERC 73-11, Earthquake Engineering Research Center, University of California, Berkeley.

J. L. Batoz and M. B. Tahar, 1982
"Evaluation of a New Quadrilateral Thin Plate Bending Element," *International Journal for Numerical Methods in Engineering*, Vol. 18, pp. 1655–1677.

Caltrans, 1995
Bridge Design Specifications Manual, as amended to December 31, 1995, State of California, Department of Transportation, Sacramento, Calif.

R. D. Cook, D. S. Malkus, and M. E. Plesha, 1989
Concepts and Applications of Finite Element Analysis, 3rd Edition, John Wiley & Sons, New York, N.Y.

R. D. Cook and W. C. Young, 1985
Advanced Mechanics of Materials, Macmillan, New York, N.Y.

A. K. Gupta, 1990
Response Spectrum Method in Seismic Analysis and Design of Structures, Blackwell Scientific Publications, Cambridge, Mass.

J. P. Hollings and E. L. Wilson, 1977
3–9 Node Isoparametric Planar or Axisymmetric Finite Element, Report No. UC SESM 78-3, Division of Structural Engineering and Structural Mechanics, University of California, Berkeley.

A. Ibrahimbegovic and E. L. Wilson, 1989
"Simple Numerical Algorithms for the Mode Superposition Analysis of Linear Structural Systems with Nonproportional Damping," *Computers and Structures*, Vol. 33, No. 2, pp. 523–531.

A. Ibrahimbegovic and E. L. Wilson, 1991
"A Unified Formulation for Triangular and Quadrilateral Flat Shell Finite Elements with Six Nodal Degrees of Freedom," *Communications in Applied Numerical Methods*, Vol. 7, pp. 1–9.

L. E. Malvern, 1969
Introduction to the Mechanics of a Continuous Medium, Prentice-Hall, Englewood Cliffs, N.J.

S. Nagarajaiah, A. M. Reinhorn, and M. C. Constantinou, 1991
3D-Basis: Nonlinear Dynamic Analysis of Three-Dimensional Base Isolated Structures: Part II, Technical Report NCEER-91-0005, National Center for Earthquake Engineering Research, State University of New York at Buffalo, Buffalo, N. Y.

Y. J. Park, Y. K. Wen, and A. H-S. Ang, 1986
"Random Vibration of Hysteretic Systems under Bi-Directional Ground Motions," *Earthquake Engineering and Structural Dynamics*, Vol. 14.

R. J. Roark and W. C. Young, 1975
Formulas for Stress and Strain 5th Edition, McGraw-Hill, New York, N.Y.

R. L. Taylor and J. C. Simo, 1985
"Bending and Membrane Elements for Analysis of Thick and Thin Shells," *Proceedings of the NUMEETA 1985 Conference*, Swansea, Wales.

K. Terzaghi and R. B. Peck, 1967
Soil Mechanics in Engineering Practice, 2nd Edition, John Wiley & Sons, New York, N.Y.

S. Timoshenko and S. Woinowsky-Krieger, 1959
Theory of Plates and Shells, 2nd Edition, McGraw-Hill, New York, N.Y

Y. K. Wen, 1976
"Method for Random Vibration of Hysteretic Systems," *Journal of the Engineering Mechanics Division*, ASCE, Vol. 102, No. EM2.

D. L. White and J. F. Hajjar, 1991

"Application of Second-Order Elastic Analysis in LRFD· Research to Practice," *Engineering Journal*, ACI, Vol. 28, No. 4, pp. 133–148.

E. L. Wilson, 1970
SAP — A General Structural Analysis Program, Report No. UC SESM 70-20, Structural Engineering Laboratory, University of California, Berkeley.

E. L. Wilson, 1972
SOLID SAP — A Static Analysis Program for Three Dimensional Solid Structures, Report No. UC SESM 71-19, Structural Engineering Laboratory, University of California, Berkeley.

E. L. Wilson, 1985
"A New Method of Dynamic Analysis for Linear and Non-Linear Systems," *Finite Elements in Analysis and Design*, Vol. 1, pp. 21–23.

E. L Wilson, 1993
"An Efficient Computational Method for the Base Isolation and Energy Dissipation Analysis of Structural Systems," ATC17-1, *Proceedings of the Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control*, Applied Technology Council, Redwood City, Calif.

E. L. Wilson and M. R. Button, 1982
"Three Dimensional Dynamic Analysis for Multicomponent Earthquake Spectra," *Earthquake Engineering and Structural Dynamics*, Vol. 10.

E. L. Wilson, A. Der Kiureghian, and E. P. Bayo, 1981
"A Replacement for the SRSS Method in Seismic Analysis," *Earthquake Engineering and Structural Dynamics*, Vol. 9.

E. L Wilson and I. J. Tetsuji, 1983
"An Eigensolution Strategy for Large Systems," *Computers and Structures*, Vol. 16.

E. L. Wilson, M. W. Yuan, and J. M. Dickens, 1982
"Dynamic Analysis by Direct Superposition of Ritz Vectors," *Earthquake Engineering and Structural Dynamics*, Vol. 10, pp. 813–823.

V. Zayas and S. Low, 1990
"A Simple Pendulum Technique for Achieving Seismic Isolation," *Earthquake Spectra*, Vol. 6, No. 2.

O. C. Zienkiewicz and R. L. Taylor, 1989
The Finite Element Method, 4th Edition, Vol. 1, McGraw-Hill, London.

O. C. Zienkiewicz and R. L. Taylor, 1991
The Finite Element Method, 4th Edition, Vol. 2, McGraw-Hill, London.

End User License Agreement for CSI Software

This is a legal agreement between you (either an individual or entity), the end user, (hereinafter referred to as "Licensee") and COMPUTERS AND STRUCTURES, INC. (hereinafter referred to as "CSI"). If Licensee does not agree to the terms of this Agreement, Licensee shall promptly return the unopened disk package and any accompanying items (including written materials and binders or other containers) to the place Licensee obtained them for a full refund. This package is to be opened only by the Licensee (or authorized representative thereof). By opening this sealed diskette package, the Licensee indicates acceptance of this Agreement.

GRANT OF LICENSE.

Dedicated Use. CSI grants to Licensee the right to use one (1) copy of the software program identified above (the "Software") on a single computer ("Dedicated Computer"). Licensee may transfer the Software to another single computer, provided Licensee does so no more often than once every ninety (90) days and no copies of the Software licensed herein are retained for use on any other computer.

Transferability. CSI grants to Licensee a nontransferable License to use this Software. Multiple licenses of the Software to same licensee are available at discounted rates. This License limits the use of this copy of the Software to projects of the Licensee. Application of the Software as a service to other engineering firms or individuals, or use of the Software by other engineering firms or individuals, is not allowed. Licensee shall not sell, rent, lease, transfer, network, publish, disclose, display or otherwise make available any portions of the Software or copies thereof to others.

For the purposes of this section, "use" means loading the Software into RAM, as well as installation on a hard disk or other storage device (other than a network server). Licensee may access the Software from a hard disk, over a network, or any other method Licensee chooses, so long as Licensee otherwise complies with this CSI License Agreement.

TERM OF LICENSE.

The term of this License shall begin from the date of this License and will last as long as the Licensee complies with the terms of this Agreement This License is subject to cancellation by CSI if Licensee fails to comply with the terms and conditions of this Agreement. Within five (5) days of such cancellation, Licensee shall return the Software to CSI.

COPYRIGHT.

The Software is owned by CSI and is protected by United States copyright laws and international treaty provisions. Therefore, Licensee must treat the Software like any other copyrighted material (e.g., a book or musical recording) except that Licensee may either make one (1) copy of the Software solely for backup or archival purposes, or transfer the Software to a single hard disk, provided Licensee keeps the original solely for backup or archival purposes. Licensee may not copy the written materials accompanying the Software.

OTHER RESTRICTIONS.

This CSI License Agreement is Licensee's proof of license to exercise the rights granted herein and must be retained by Licensee. Licensee agrees to secure and protect the Software and copies thereof in a manner consistent with the maintenance of the proprietary rights of CSI. Licensee agrees to take appropriate action, by instruction or agreement with its employees who are permitted access to the Software, to protect unauthorized proliferation of the Software. Licensee may not reverse engineer, decompile, or disassemble the Software. If the Software is an update, any transfer must include the update and all prior versions. If the Software package contains both 3.5" and 5.25" disks, then Licensee may use only the disks appropriate for Licensee's single designated computer or network server. Licensee may not use the other disks on another computer or computer network, or loan, rent, lease, or transfer them to another user except as part of a transfer or other use as expressly permitted by this CSI License Agreement.

PROGRAM UPGRADES.

From time to time, CSI may, at its discretion, offer program upgrades. If offered, such upgrades will be provided at no charge to the Licensee for a period of ninety (90) days from the date of this Agreement. Thereafter, upgrades will be provided under the terms defined by the CSI Software Maintenance Agreement.

SUPPORT.

User support will be available to one (1) contact individual designated by the Licensee, for a period of ninety (90) days from the date of this Agreement Thereafter, continued support will be available under the terms defined by the CSI Software Maintenance Agreement.

During the term of this License, CSI, at its discretion, will assist Licensee in resolving problems suspected of being caused by possible defects in the Software. However, if it is determined that the problem stems from a user data error, the Licensee agrees to reimburse CSI for the time spent in resolving the problem, at the prevailing professional rates of CSI.

LIMITED WARRANTY; NO OTHER WARRANTIES.

CSI warrants that the Software will perform substantially in accordance with the accompanying written materials for a period of ninety (90) days from the date of receipt. Any implied warranties on the Software are limited to ninety (90) days. Some states do not allow limitations on duration of an implied warranty, so the above limitation may not apply to Licensee.

CSI disclaims all other warranties, either express or implied, including but not limited to implied warranties of merchantability and fitness for a particular purpose, with respect to the Software, the accompanying written materials, and any accompanying hardware This limited warranty gives Licensee specific legal rights. Licensee may have others, which vary from state to state.

CUSTOMER REMEDIES.

CSI's entire liability and Licensee's exclusive remedy shall be, at CSI's option, either (a) return of the price paid or (b) repair or replacement of the Software that does not meet CSI's Limited Warranty and that is returned to CSI with a copy of Licensee's receipt. This Limited Warranty is void if failure of the Software has resulted from accident, abuse or misapplication. Any replacement Software will be warranted for the remainder of the original warranty period or thirty (30) days, whichever is longer. **These remedies are not available outside of the United States.**

NO LIABILITY FOR CONSEQUENTIAL DAMAGES.

In no event shall CSI or its suppliers be liable for any damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or other pecuniary loss) arising out of the use of or inability to use this CSI product, even if CSI has been advised of the possibility of such damages. Because some states do not allow the exclusion or limitation of liability for consequential damages, the above limitation may not apply to Licensee.

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT OF THE SOFTWARE, AND IT HAS BEEN THOROUGHLY TESTED AND USED HOWEVER, EXCEPT AS OTHERWISE SPECIFICALLY PROVIDED HEREIN, NO WARRANTY IS MADE ON ITS ACCURACY OR RELIABILITY. IT IS THE RESPONSIBILITY OF THE ENGINEER TO VERIFY THE RESULTS OBTAINED FROM THE SOFTWARE. IN THE EVENT THE SOFTWARE IS FOUND TO BE DEFECTIVE, CSI'S ONLY OBLIGATION IS TO REMEDY THE DEFECT CSI WILL IN NO EVENT HAVE OBLIGATIONS OR LIABILITIES FOR INCIDENTAL OR CONSEQUENTIAL DAMAGES ASSOCIATED WITH THE USE OF THE SOFTWARE NO OTHER WARRANTY IS MADE.

U.S. GOVERNMENT RESTRICTED RIGHTS

The Software and documentation are provided with RESTRICTED RIGHTS. Use, duplication, or disclosure by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs (c)(1) and (2) of the Commercial Computer Software--Restricted Rights at 48 CFR 52.227-19, as applicable.

GENERAL

This License shall be interpreted by the laws of the State of California In the event that any portion of this Agreement is invalidated by court or legislation, the remaining portions of the Agreement shall remain in binding effect. In the event of legal action brought by either party, the prevailing party shall be entitled to reimbursement of actual legal fees and related expenses. The Licensee agrees that this Agreement shall inure to the benefit of CSI, its administrators, successors, heirs and assignees. This Agreement may only be changed by mutual written consent.

Should Licensee have any questions concerning this Agreement, or if Licensee desires to contact CSI for any reason, please write: CSI, Consumer Sales and Service, 1995 University Avenue, Berkeley, California 94704.

Questions and Answers about the End User License Agreement

- Can I sell my end user license agreement for SAP2000 to someone else?
- Can I sell or give away old versions of SAP when I acquire an upgrade?
- Can I make a second copy for my personal use?

- Can I rent or lease the Program to someone else?
- When I upgrade from SAP90, do my license rights for SAP2000 change?
- In what ways can I use the software over a network?

Can I sell my end user license agreement for SAP2000 to someone else?

No. You may not sell, rent or lease SAP2000 on a temporary or permanent basis.

Can I sell or give away old versions of SAP when I acquire an upgrade?

No. The original SAP90 and the upgrade SAP2000 together are considered as a single software unit. You must retain the old SAP90 version as part of the license agreement. It is a good idea in any case to save the older versions in case you need to go back and run the old data files

Can I make a second copy for my personal use?

CSI grants to Licensee a **nontransferable** License to use this Software. Multiple licenses of the Software to same licensee are available at discounted rates. This License limits the use of this copy of the Software to projects of the Licensee Application of the Software as a service to other engineering firms or individuals, or use of the Software by other engineering firms or individuals, is not allowed.

Can I rent or lease the Program to someone else?

Licensee shall not sell, rent, lease, transfer, network, publish, disclose, display or otherwise make available any portions of the Software or copies thereof to others.

When I upgrade from SAP90, do my license rights for SAP2000 change?

Yes, the license agreement included with the SAP2000 upgrade sets forth the license rights for both the SAP90 and the SAP2000. With every new upgrade product, you will receive a new license agreement. Upgrade versions are treated as part of the whole product.

In what ways can I use the software over a network?

Network version of the software is available. Please contact CSI for pricing. For all other versions you must acquire a hardware protection device and dedicate one individual computer from which you use the software. For multiple computers you must purchase multiple licenses which are available at a discounted rate. A license to use a particular copy of the software may not be shared or used concurrently on different computers.

APPENDIX

Complete Quadratic Combination -- a modal combination technique which accounts for modal damping. Same as SRSS if damping is zero

Square Root of Sum of Squares -- a modal combination technique which does not account for modal damping.

General Modal Combination -- also known as Gupta Method.

Absolute -- arithmetic summation without signs

EIGENVECTORS

Eigenvalue analysis determines the undamped free-vibration mode shapes and frequencies of the system. These natural modes provide an excellent insight into the behavior of the structure.

RITZ

Ritz vectors take into account the spatial distribution of the dynamic loading.

Rigid-body behavior

Rigid-body behavior, in which the constrained joints translate and rotate together as if connected by rigid links. The types of rigid behavior that can be modeled are·

Equal-displacement behavior

Equal-displacement behavior, in which the translations and rotations are equal at the constrained joints.

Symmetry Conditions

For a structure that is symmetric about a plane, symmetric loading causes symmetric displacements as follows:

- Forces and displacements parallel to the plane of symmetry are equal

- Forces and displacements normal to the plane of symmetry are opposite
- Moments and rotations parallel to the plane of symmetry are opposite
- Moments and rotations normal to the plane of symmetry are equal

Anti-symmetry Conditions

For a structure that is symmetric about a plane, anti-symmetric loading causes anti-symmetric displacements. All degrees of freedom that are equal when symmetric are opposite when anti-symmetric, and all degrees of freedom that are opposite when symmetric are equal when anti-symmetric. Thus the specification of the anti-symmetric degrees of freedom simply uses the opposite signs from the symmetric case.

Von Mises Stress

The Von Mises Stress provides a measure of the shear, or distortional, stress in the material. This type of stress tends to cause yielding in metals. It is independent of the amount of hydrostatic stress
($\sigma 1= \sigma 2= \sigma 3$) action on the material.

The Von Mises Stress is identified in terms of the principal stresses as
$\sigma vm=\sqrt{1/2[(\sigma 1- \sigma 2)2+(\sigma 1- \sigma 3)2+(\sigma 2- \sigma 3)2]}$.

In a state of pure tension, say $\sigma 11=\sigma$ and all other stresses are zero, then $\sigma vm=\sigma$. In a state of pure shear, say $\sigma 12=\tau$ and all other stresses are zero, then $\sigma vm=\sqrt{3}\,\tau$.

For materials, initial yielding can be expected when $\sigma vm=\sigma y$, where σy is the tensile yield stress, or when $\sigma vm=\sqrt{3}\,\tau y$, where τy is the yield stress in shear. For other materials, particularly frictional materials such as soil and concrete, the Von Mises Stress may have no value in predicting yield or failure.

Analysis Cases

Analysis Cases include:

Static Load Cases
Response Spectrum Analysis Cases
Time History Analysis Cases
Moving Load Case
Modal case based on a specific vibration mode shape
Load Combos located above the present Load Combo in the Combinations list box

Load Combination Type

ADD = All Analysis case results are multiplied by their scale factor and added together. Combo type can be used for static loads.

ENVE = An Max/Min Envelope of the defined Analysis cases is evaluated for each frame output segment and element joints. The Analysis cases that give the Maximum and Minimum components are used for this combo. Therefore the Load Combo holds two values for each output segment and joint. Combo type can be used for moving loads and any Analysis case where the load producing the maximum or minimum force/stress is required.

ABS = The absolute of the individual Analysis case results are summed and positive and negative value are automatically produced for each output segment and joint. Combo type can be used for lateral loads.

SRSS = The Square Root Sum of the Squares calculation is done on the Analysis cases and positive and negative value are automatically produced for each output segment and joint. Combo type can be used for lateral loads.

Linear Time History Analysis

The Time History functions specified are induced on the structure and all elements are taken to be linear.

Periodic Time History Analysis

The Time History functions are assumed to be periodic and the steady state results are displayed taking all elements to be linear.

NonLinear Time History Analysis

The Time History Analysis is performed with nonlinear link elements being allowed to act nonlinearly.

Ground Acceleration

The specified Time History Function will be induced on the structure as a ground acceleration through all support joints.
acc dir 1 = Ground Acceleration in the global X direction.
acc dir 2 = Ground Acceleration in the global Y direction.
acc dir 3 = Ground Acceleration in the global Z direction.

UBC94 Response Spectrum

UBC94S1 Response Spectrum for Soil Type 1.
UBC94S2 Response Spectrum for Soil Type 2.
UBC94S3 Response Spectrum for Soil Type 3

(type popup definition text here)

(type popup definition text here)

Energy Functions

Input Energy	Work done on structure by force and acceleration functions. In an exact solution this would equal the sum of the Kinetic, Potential, Mdamp, Ndamp and NLLink Energies.
Kinetic Energy	A function of the sums of the masses and their corresponding velocity for the structure.
Potential Energy	A function of the sums of the elastic constants and their corresponding displacements for the structure. The sum includes NLLink members.
MDamp Energy	(Modal Damping Energy) Is the energy absorbed through modal damping.
NDamp Energy	(NLLink Damping Energy) Is the energy absorbed through NLLink damping.
NLLink Energy	(NLLink Hysteretic Energy) Is the energy absorption by Plastic1, Isolator1 and Isolator2 NLLink

elements. Damper, Gap and Hook NLLink elements are not included.

Error Energy — The is an estimate of the error in the analysis. It is the difference between the Input energy and the sum of the remaining energy components.

Base Functions

Base Shear: The total shear on all the supports in a given global direction.
Base Moment: The Overturning/Torsional moment on the supports of the structure in a given global direction.

DXF

DXF is a Computer Aided Drafting (CAD) file format used by many CAD programs

Joint Stress Averaging

None:

This option is used to display contours with no averaging. It is used to see if the model is meshed properly. By selecting this option you can see whether there are large stress variations between elements. This can be an indication that the model is not properly meshed and may need to be refined to properly capture the variation in stress.

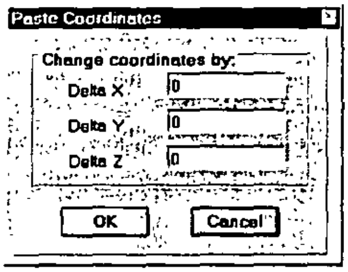
At All Joints:

Once you are confident that the model is acceptable, the stress Averaging will give better results of the stress in the elements by averaging the stress between elements and thus reducing the error due to the stress on each individual element.

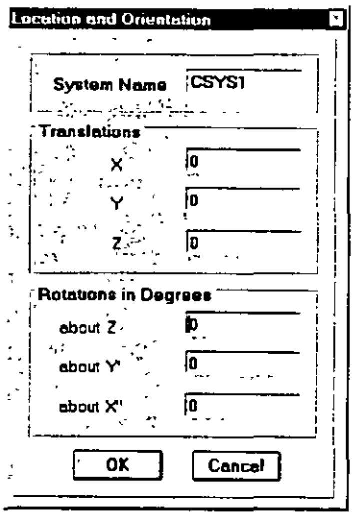
At Select Joints:

If you have a discontinuity in the model, like two planes meeting at an angle, then you will need to do the averaging on each plane independently by using this option. This will avoid the problem of averaging across the two planes which would give incorrect result. This is due to the stress along the two planes not being continues relative to the elements local axes.

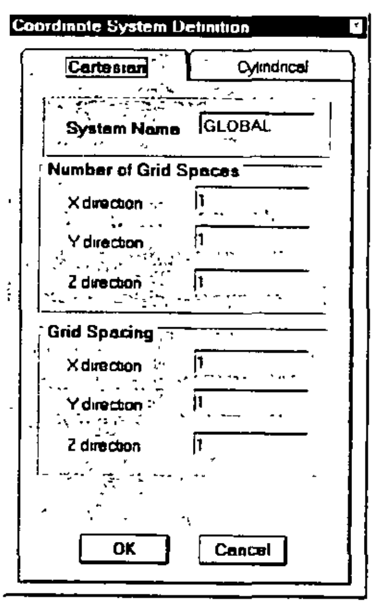
Paste Coordinates



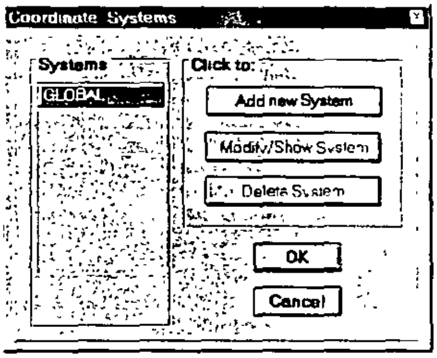
Insertion Point



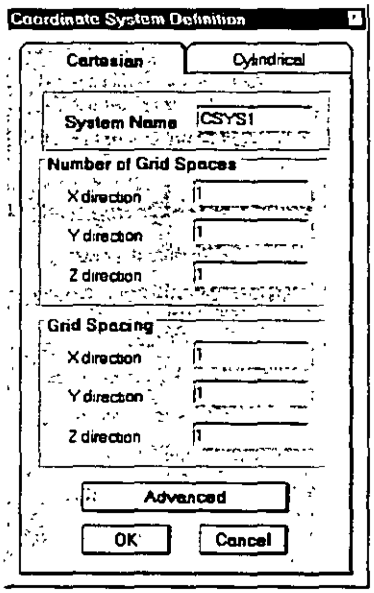
Coordinate System



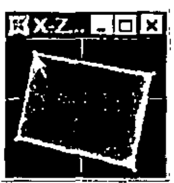
Set Coordinate System



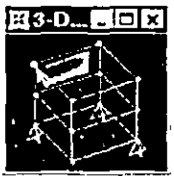
Coordinate System (Advanced)



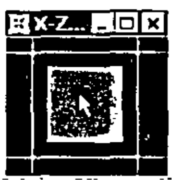
Shell Element (4 Joints)



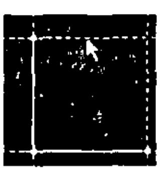
Shell Element (4 Joints 3-D)



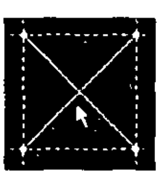
Shell Element (Quick)



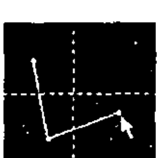
Frame Element (Quick)



Frame Element (Cross Brace)

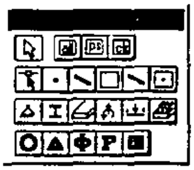


Frame Element (2 Joint)

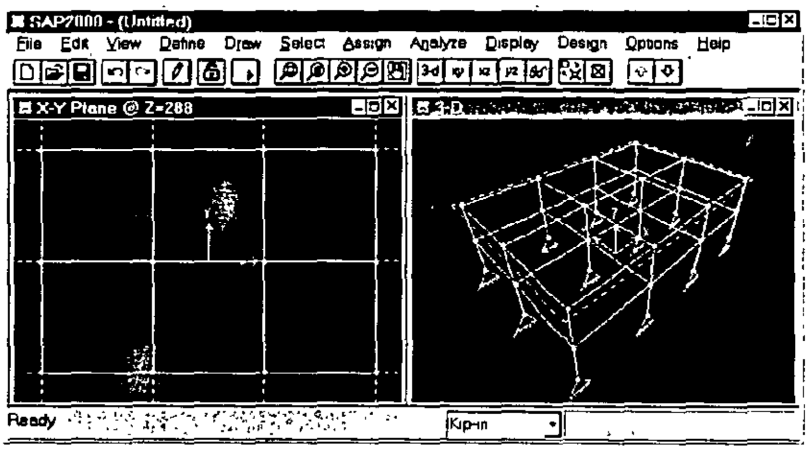


Frame Element (2 Joint 3-D)

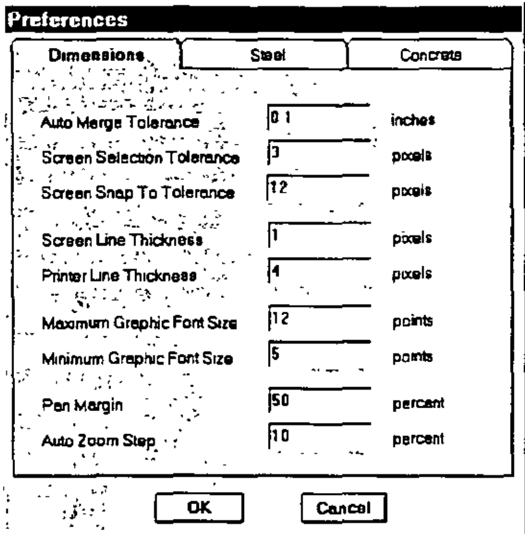
Floating Toolbar



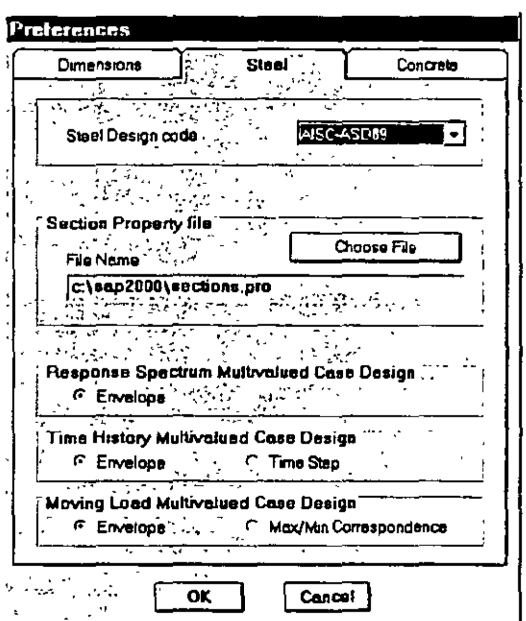
SAP2000 Screen



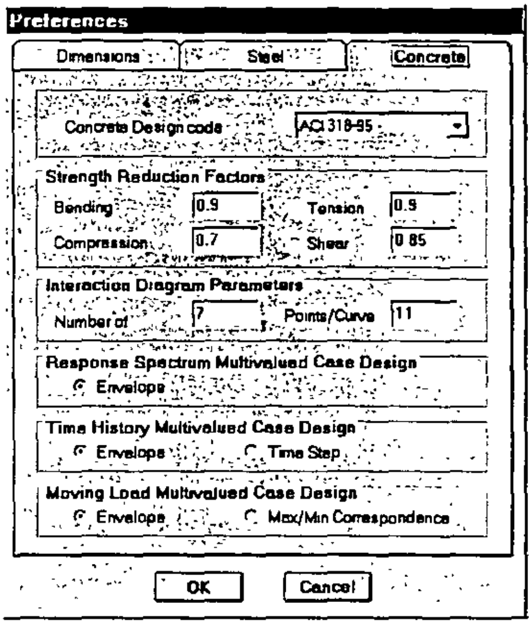
Dimension Tolerances



Steel Design Parameters



Concrete Design Parameters



INDEX

2

2 Modes of SAP2000 1

A

Add a Frame Section 25
Add a New Concrete Material Type 23
Add a New Other Material Type 24
Add a New Steel Material Type 23
Add a Nonprismatic Frame Section 26
Add a Shell Section 27
Add an Auto Select Frame Group 27
Add an NLLink Property 27
Add Joints to an existing Constraint 50
Add Special Joints 42
Analyzing a Model 67
Animate Deformed Shape 72
Animation 10, 72
Anti-symmetry About a Plane 55
Asolid 5
Asolid Element 5
Assign Dynamic Loads 37
Assign Frame End Offsets 59
Assign Frame Initial P-Delta Force 61
Assign Frame Lanes 61
Assign Frame Output Segments 60
Assign Frame Prestress 60
Assign Frame Releases 59
Assign Gravity Loads to Frames 63
Assign Gravity Loads to NLLinks 65
Assign Gravity Loads to Shells 64
Assign Group Names 65
Assign Joint Constraints 50
Assign Joint Masses 57
Assign Joint Patterns 57
Assign Joint Restraints 57
Assign Joint Springs 56
Assign Loads or Displacements to Joints 62
Assign Local Axes 58
Assign Local Axes for Frame Elements 58
Assign Local Axes for Joints 58
Assign Local Axes for Shell Elements 61
Assign Local Axis For NLLinks 62
Assign Options 49
Assign Point and Uniform Loads to Frames 63
Assign Pressure Loads to Shells 64
Assign Prestress to Frame Static Load 61
Assign Section Properties 49
Assign Static Loads 62

Assign Temperature Loads to Frames 64
Assign Temperature Loads to Shells 65
Assign Trapezoidal Loads to Frames 63
Assign Uniform Loads to Shells 64
Auto Select 27
Automatic Meshing 16
AVI 9
Axes 58, 59, 62
Axisymmetry 55, 56

B

Base Functions 77
Base Shear 77
Beam 25
Beam Constraint 53
Body Constraint 51
Break 16, 17
Bridge 30, 31, 32, 35, 36, 61, 70, 91
Bridge Analysis 30

C

Change Labels 18
Color 92
Column 26
Concrete 11, 23, 24, 25, 26, 83, 84, 85, 86, 87, 88, 89, 91
Concrete Design 83
Connect 18
Constraint 48, 50, 51, 52, 53, 54, 55, 56
Coordinate System 92
Copy 12, 13
Cut 12, 13
Cyclic symmetry 56

D

Define Bridge Response 36
Define Frame Section Properties 24
Define Group Names 29
Define Joint Patterns 37
Define Lanes 30
Define Load Combinations 39
Define Materials 23
Define Moving Loads 36
Define Response Spectrum Cases 39
Define Response Spectrum Functions 38
Define Section Properties 24
Define Static Load Cases 30
Define Time History Cases 38
Define Time History Functions 37
Define Vehicle Classes 36
Define Vehicles 31
Defining Elements as a Single Member for Bending 85
Deformed Shape 71, 72
Delete 13
Delete or Remove Joints from an existing Constraint 50
Deleting Members 13
Deselect 46
Design 11, 83, 84, 85, 86, 87, 88, 89, 91, 92
Designer's Convention 1
Diaphragm Constraint 51
Disconnect 17
Displacement 62
Display Base Function Time History Trace 77
Display Bridge Lanes 70
Display Concrete Design/Check Input on Screen 87
Display Concrete Design/Check Output on Screen 86
Display Design Input/Results on Screen 86
Display Frame Element Force Output Time History Trace 74
Display Frame Influence Lines 82

Display Group Joint Force Sums 81
Display Group Summation Forces Time History 77
Display Input in Tabular Format 71
Display Input Time History Functions 73
Display Joint Influence Lines 81
Display Joint or Member Text Output on Screen 82
Display Joint Output Time History Trace 73, 74
Display Joint Patterns 70
Display Member Force or Stress Diagram 72
Display Mode Shape 71
Display Options 69
Display Shell Element Stress Output Time History Trace 75
Display Static Deformed Shape 71
Display Static Loads 70
Display Steel Stress Check/Design Input on Screen 87
Display Steel Stress Check/Design Ratios on Screen 86
Display Structural Energy Time History Trace 76
Displaying Undeformed Geometry 70
Divide 16, 17
Divide or Break Frames 16
Draw a Frame Element from Joint to Joint 43
Draw a Quick Frame Element 43
Draw a Quick Shell Element 44
Draw a Shell Element between 4 Joints 43
Draw an NLLink Element 44
Draw Members 41
Draw Mode 41
Duplicates 18
DXF 9
Dynamic 67

E

Edit Grids 44
End Offset 59
End User License Agreement for CSI Software 98
Energy Functions 76
Equal Constraint 53
Export 8, 9
Export a SAP2000 text input file 8
Export an AVI file 9

F

F(t) vs F(t) 78
F(t) vs t 78
Force Diagram 72
frame 3, 16, 17, 24, 25, 26, 27, 42, 43, 47, 49, 58, 59, 60, 61, 63, 64, 69, 70, 71, 72, 74, 75, 82
Frame 46
Frame Element 3
Frequently Asked Questions 95

G

General Tips 93
General Vehicle 35
Generalized Constraints 50
Graphical Output 70
Gravity 63, 64, 65
Grid 2, 44, 45
Group 30, 47, 65, 77, 81, 84
Group Elements for Design 84

I

Import 8, 9
Import a DXF file 9
Import a Frame Section from the Section property file (e.g. Sections.pro) 25
Import a SAP2000 Job Run in DOS mode 9
Import a SAP2000 text input file 8
Import a SAP90 text input file 8

Import/Export Capabilities 8
Influence Lines 81, 82
Initialize a New Labeling Scheme 46
Input 71
Interaction Diagram 88
Intersecting Line 47

J

Join Frames 17
Joint 16, 37, 42, 45, 49, 50, 57, 58, 62, 63, 69, 70, 72, 73, 74, 81, 82
Joint and Element Labels 18

L

Label 18, 46, 48
Lane 31, 70
License 98, 99, 100, 101
Load 30, 37, 38, 39, 40, 49, 61, 62, 63, 64, 65, 69, 70, 71, 84, 85
Load Cases 30
Load Combinations 39, 40, 84
Local Axis 62
Local Constraint 54
Log 12

M

Mass 57
Material 23, 24
Merge Joints 16
Mesh 16, 17
Mesh Shells 17
Mirror 15
Mode Shape 71
Move 14
Moving Loads 37

N

Network 101
New Interface 1
NLLink 27, 44, 48, 62, 65
Nonlinear 5, 6
Nonlinear Analysis 5
Nonprismatic 26
Note 12
Noun-Verb 1

O

Online Support 95
Output 67
Output Segment 60
Overwrite Element Design Data 85

P

Partial Connection 54
Paste 12, 13
Patterns 70
P-Delta 61, 67, 68
Phone and Fax Support 94
Plate Constraint 52
Preferences 91
Pressure 64
prestress 60, 61
Print 10, 11, 12, 67, 79, 88, 92
Print Design Results to a Printer or a File 11
Print Input Tables to a Printer or a File 11
Print Output to a Printer or a File 11
Print Selected Graphical Output to a Printer or a File 10
Print Setup 10

Print Time History or Response Spectrum Plots or Tables 79
Program Manuals 95
Project Data 10

R

Radial 14, 15
Redesign a Concrete Member 88
Redesign a Steel Member 88
Redo 13
References 96
Re-Label Previously Assigned Labels 18
Release 59
Replace Auto with Optimal Sections 86
Replicate 14, 15
Replicate by Using the Mirroring Option 15
Replicate in a Linear Array 14
Replicate in a Radial Array 14
Replicate in a Radial Array by Shifting the Origin 15
Reshape Element 42
Response Spectrum 38, 39, 79, 80, 81, 91
Restraint 57
Review Concrete Design/Check Detail 87
Review Steel Stress Check/Design Detail 87
Review Stress Check/Design Details 87
Rod Constraint 52

S

SAP2000 8, 94, 95, 96, 98, 100, 101, 110
SAP2000 Modes 1
SAP2000 Screen 1, 110
SAP2000 Versions and Limitations 94
SAP90 8
Scale Factor
  Line Type and Color 79
Select 46
Select 2-D Views 21
Select 3-D Views 21
Select All Objects 48
Select Mode 2
Select Objects by 2D Planes 47
Select Objects by Constraints 48
Select Objects by Frame Sections 47
Select Objects by Groups 47
Select Objects by Intersecting Line 47
Select Objects by Lables 48
Select Objects by NLLink Properties 48
Select Objects by Shell Sections 47
Select Objects by Window 47
Selecting Design Load Combinations 84
Selection Procedures 46
Set Elements 22
Set Limits 22
Setting Colors 92
Setting Concrete Design Parameters 91
Setting Steel Design Parameters 91
Setup Coordinate System 2
Shell 17, 24, 27, 42, 43, 44, 47, 48, 49, 50, 61, 62, 64, 65, 69, 70, 72, 75, 76, 82
  Joint 71
Show Duplicates 18
Snap to Frame/Edge 46
Snap to Grid 45
Snap to Joints 45
Solid 5
Solid Element 5
Spectrum Generation Axes Tab 80
Spectrum Generation Damping Tab 81
Spectrum Generation Define Tab 80
Spectrum Generation Frequency Tab 80

Spectrum Generation Options Tab 80
Spring 57
Standard Vehicles 32
Start Design/Check of Structure 84
Static Load Cases 30
Steel 11, 23, 83, 85, 86, 87, 88, 91
Steel Design 83
Stress Diagram 72
Support 49, 94, 95
Symmetry About a Plane 55
Symmetry About a Point 56

T

Temperature 64, 65
Template 7
Text Books 96
Text Editor 12
Text Output 11
Time History 37, 38, 73, 74, 75, 76, 77, 78, 79, 91, 92

U

Undeformed Geometry 70
Undo 13
Undo and Redo Capabilities 13
Update Analysis Sections 89
User Comments and Session Log 12
User Defined Coordinate System 92

V

Vehicle 31, 32, 35, 36
View 10, 21, 22, 69, 70, 73, 79, 88
View Generated Response Spectra Curves 79
View Interaction Diagram for a Concrete Column 88
View Real Time Animation for Time History Results 10
View Time History Results 73

Z

Zoom 22, 79, 88
Zoom into the Plot 79

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

I
TEMA

SAP2000 INTEGRATED FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

VERIFICATION MANUAL

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 ● Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 ● Fax: 5510.0573

SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

VERIFICATION MANUAL

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

Version 6.1
Revised July 1997

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

tel: (510) 845-2177
fax: (510) 845-4096
e-mail: *info@csiberkeley.com*
web: *www.csiberkeley.com*

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table of Contents

	Introduction	1
Example 1	Two-Dimensional Frame — Static and Dynamic Loads	3
Example 2	Bathe and Wilson Frame — Eigenvalue Problem	13
Example 3	Three-dimensional Frame — Dynamic Loads	17
Example 4	ASME Frame — Eigenvalue Problem	23
Example 5	Three-dimensional Braced Frame — Dynamic Loads	29
Example 6	Beam — Steady-State Harmonic Loads	35
Example 7	Two-dimensional Truss — Static Loads	39
Example 8	Three-dimensional Building — Dynamic Loads	43
Example 9	Patch Tests — Prescribed Displacements	47
Example 10	Straight Beam — Static Loads	51
Example 11	Curved Beam — Static Loads	59
Example 12	Twisted Beam — Static Loads	63

Example 13	Beam On Elastic Foundation — Static Loads	67
Example 14	Rectangular Plate — Static Loads	71
Example 15	Cantilever Plate — Eigenvalue Problem	75
Example 16	Scordelis-Lo Roof — Static Loads	79
Example 17	Hemispherical Shell — Static Loads	83
Example 18	Portal with P-delta	87
Example 19	Pounding of Two Planar Frames — Nonlinear Time-History Analysis	91
Example 20	Friction-Pendulum Base-Isolated 3D Frame — Nonlinear Time-History Analysis	95

Introduction

This manual presents a set of example problems solved using the SAP2000 structural analysis program. These examples demonstrate many of the capabilities of the SAP2000 program. For purposes of verification, key results from these examples are compared with theoretical or published results from other computer programs, where such are available. The verification problems cover each type of element and include both static and dynamic examples.

For each example, this manual contains:

- A short description of the problem
- A list of significant SAP2000 options activated
- A description of the input data used to create the model
- A comparison of key results with theoretical results or results from other computer programs, if available

Some examples are solved using several different elements, mesh sizes and/or boundary conditions. Key results from these different solutions are presented for comparative purposes.

The data files and selected output files associated with the example problems are provided in subdirectory EXAMPLES of the SAP2000 directory. For each example, the following files may be provided:

- The SAP2000 model file, with extension .SDB. This file can be accessed in SAP2000 using the Open command under the File menu
- The input data text file, with extension .S2K. This is an alternative to the .SDB file and can be accessed in SAP2000 using the Import command under the File menu
- Binary results files, with various extensions. If these files are provided, analysis results can be viewed in SAP2000 without running the analysis

Some of the examples have more than one model file corresponding to the use of different element types or mesh sizes in the creation of the model. The filenames associated with a particular problem are identified (without extension) in each example.

Note that not all examples can be run with every version of the program. For examples, time-history analyses require the PLUS or Nonlinear version, and the nonlinear examples require the Nonlinear version.

Example 1

Two-Dimensional Frame — Static and Dynamic Loads

Description

This is a seven-story two-dimensional frame structure, subjected to static lateral and vertical loads and dynamic lateral loads due to earthquakes. The structure is analyzed once using earthquake loads specified as a response spectrum, and once using earthquake loads specified as a base acceleration time history. A solution to this problem using another computer program is documented in Reference [1]. The framing and the static loads are shown in Figure 1-1. The earthquake excitation is shown in Figure 1-8. It is the N-S component of the 1940 El Centro earthquake. The framing and the loads are all identical to the ones used in Reference [1].

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Diaphragm constraints
- Lateral joint loads
- Vertical span loads
- Response-spectrum analysis
- Time-history response to base excitation

Input Data

The computer model used is shown in Figures 1-1 and 1-2. Kip-inch units are used. Two different input files are used. The first file is to analyze the structure for static vertical and lateral loads and response spectrum dynamic loads. The input data file for this analysis is FRAME. The second file is to repeat the dynamic analysis but using base acceleration time-history as a loading. The input data file for this analysis is FRAMETH and the digitized base acceleration is given in file ELCENTRO. This is shown in Figure 1-8.

The file FRAME is described first. Vertical loads input as Load Case 1 are specified as span loading on beams. Static lateral loads input as Load Case 2 are specified as joint loads. The lateral (Y) displacements of the columns at each story level are constrained together using a separate Diaphragm Constraint for each floor. Also, masses are specified only in the lateral (Y) direction at each story level. These are common modeling techniques used to reduce the size of the equation system and are also utilized in the analysis reported in Reference [1]. The Diaphragm Constraints eliminate all axial deformations in the beam. This, and the absence of mass specification in the vertical direction reduces the dynamic problem to seven natural modes of vibration. All seven modes are included in the analysis.

It should also be noted that the AISC section properties in the database file SECTIONS.PRO are not used in this example and the required properties are explicitly entered. This is intentional as most of the sections shown are older sections not in the current AISC database.

The input file FRAMETH is identical to file FRAME for the structural model. However, no static or response spectrum loads are specified. Instead the base acceleration is specified in the Y direction. The acceleration data is discretized in unequal time steps. The output sampling time used is 0.02 seconds and the response is calculated for the first eight seconds. A damping value of 0.05 is used for all modes.

Comparison of Results

Reference [1] presents results only for the static lateral load analysis and the dynamic analysis. A comparison of key results for these analyses is presented in Figures 1-3, 1-4 and 1-5. The static results and the time periods are identical for the two programs. The comparison is excellent for the response spectrum results and good for the time-history results. Explicit time integration, not dependent on the size of the time steps, is used in SAP2000.

For the response-spectrum results, the program of Reference [1] uses the square root of the sum of the squares (SRSS) method of modal combination. SAP2000 allows both the SRSS method and the complete quadratic combination (CQC) method of modal combination. The CQC method is the default for SAP2000 and is generally recommended. Figure 1-5 presents two sets of results for SAP2000 response spectrum analysis: the default CQC results, and the SRSS results.

Plots of the deformed shape of the structure under the static lateral loads and of the sixth mode shape are shown in Figures 1-6 and 1-7. A plot of time versus displacement at the top of the structure is given in Figure 1-9.

References

1. *Static and Dynamic Analysis of Multistory Frame Structure Using DYNAMIC/EASE2*, Engineering Analysis Corporation and Computers/Structures International.

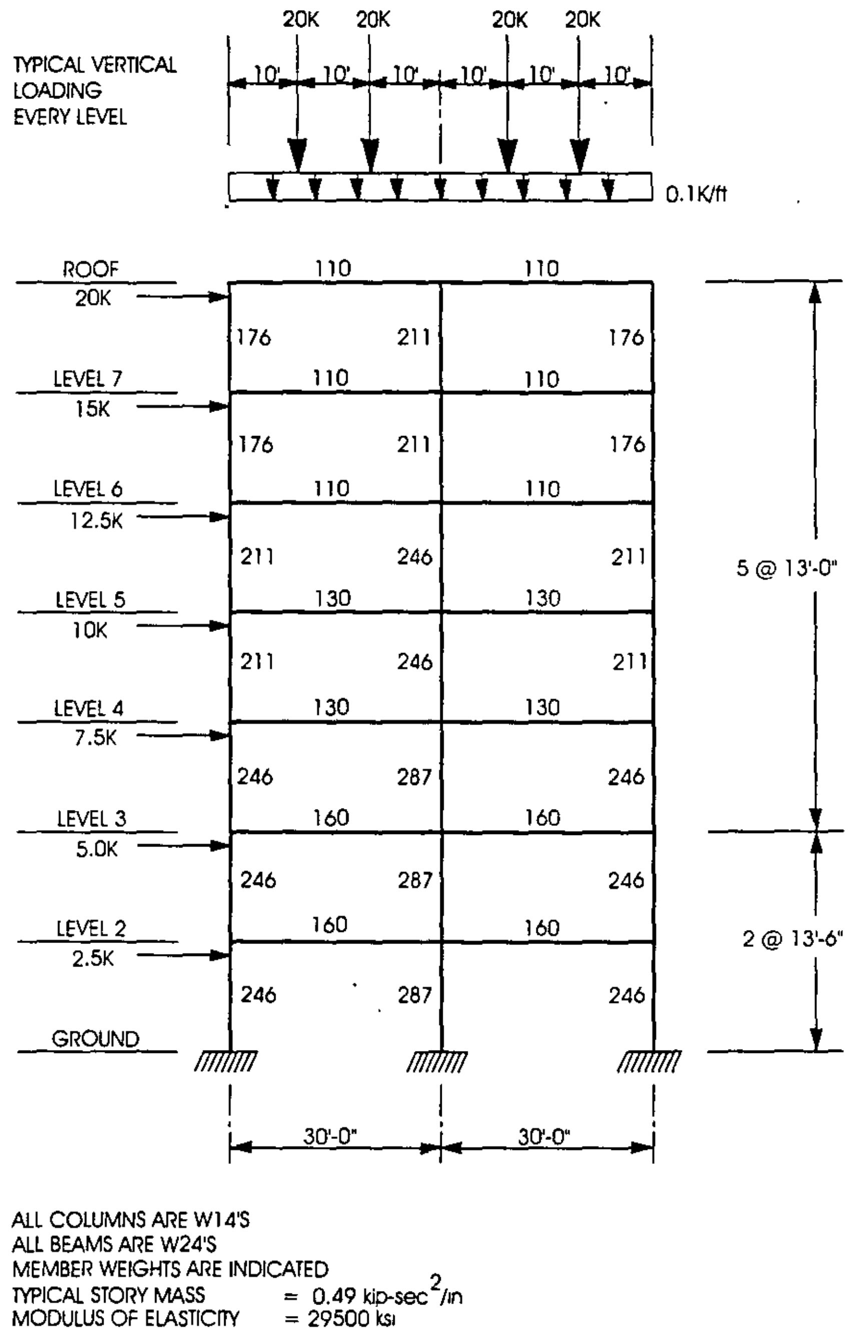


Figure 1-1

Two-dimensional Frame Example

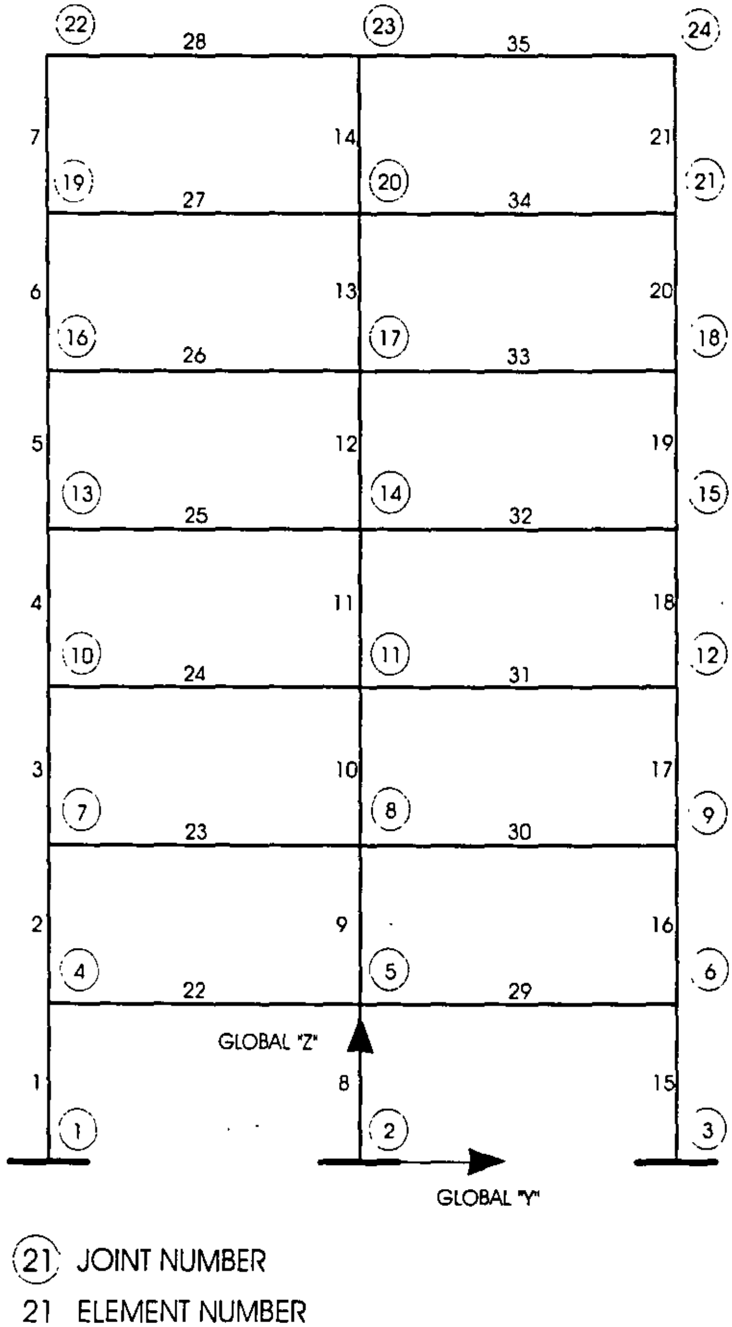


Figure 1-2
Two-dimensional Frame Example Model

Quantity	SAP2000	Reference [1]
Lateral Displacement at Node 22	1.450764	1.450764
Axial Force in Member 1	69.99	69.99
Moment in Member 1 at Node 1	2324.68	2324.68

Figure 1-3
Comparison of Results for Static Lateral Loads

Mode	SAP2000	Reference [1]
1	1.2732	1.2732
2	0.4313	0.4313
3	0.2420	0.2420
4	0.1602	0.1602
5	0.1190	0.1190
6	0.0951	0.0951
7	0.0795	0.0795

Figure 1-4
Comparison of Results for Periods of Vibration

Quantity	SAP2000			Reference [1]	
	Response Spectrum (CQC)	Response Spectrum (SRSS)	Time History	Response Spectrum (SRSS)	Time History
Lateral Displacement at Node 22	5.431	5.437	5.486	5.438	5.46
Axial Force in Member 1	261.5	261.7	263.0	261.8	258.0
Moment in Member 1 at Node 1	9916	9864	9104	9868	8740

Figure 1-5
Comparison of Results for Dynamic Analyses

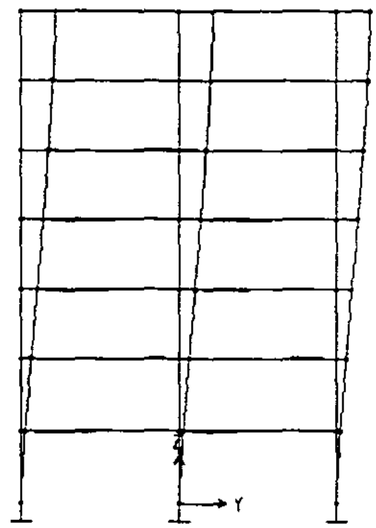


Figure 1-6
Deflection Due to Lateral Loads

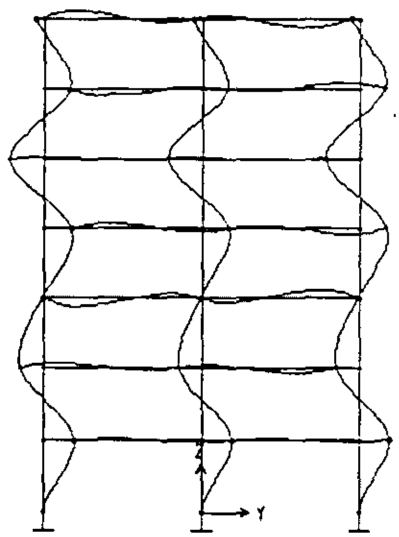


Figure 1-7
Mode Shape 6

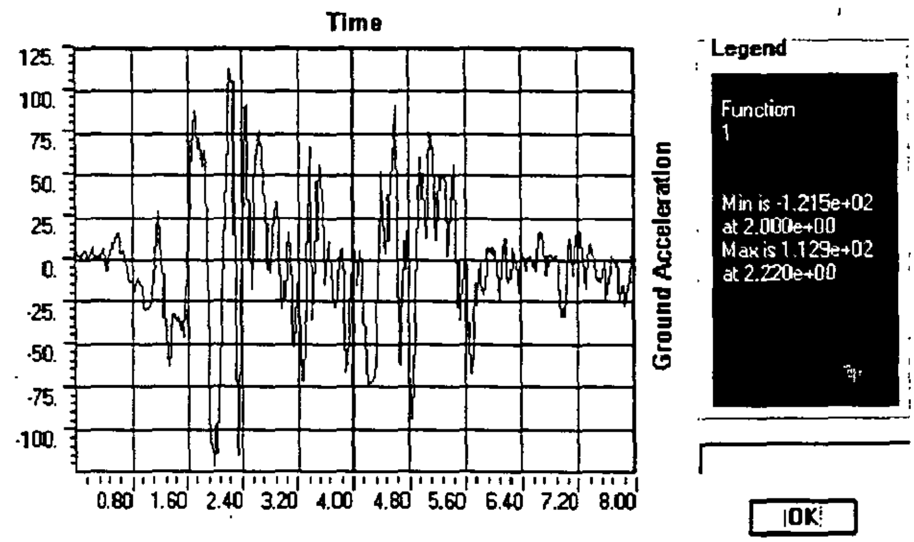


Figure 1-8
El Centro Ground Acceleration Input

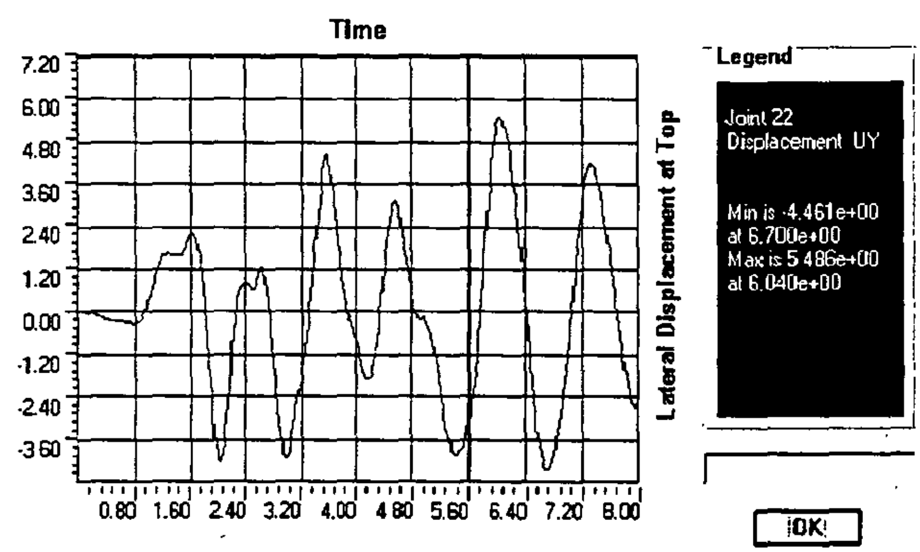


Figure 1-9
Lateral Displacement of Joint 22

Example 2

Bathe and Wilson Frame — Eigenvalue Problem

Description

This is a ten-bay, nine-story, two-dimensional frame structure solved in Reference [1]. The framing and the material and section properties are shown in Figure 2-1. The mass per unit length and other properties used are consistent with References [1] and [2], to which the results are compared. The first three eigenvalues are computed.

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Eigenvalue analysis

Input Data

The computer model used is shown in Figure 2-1. Kip-foot units are used. Mass per unit length of the members is specified. The program automatically computes the joint masses to be used in the eigenvalue analysis.

The input data file for this example is FRAMEBW.

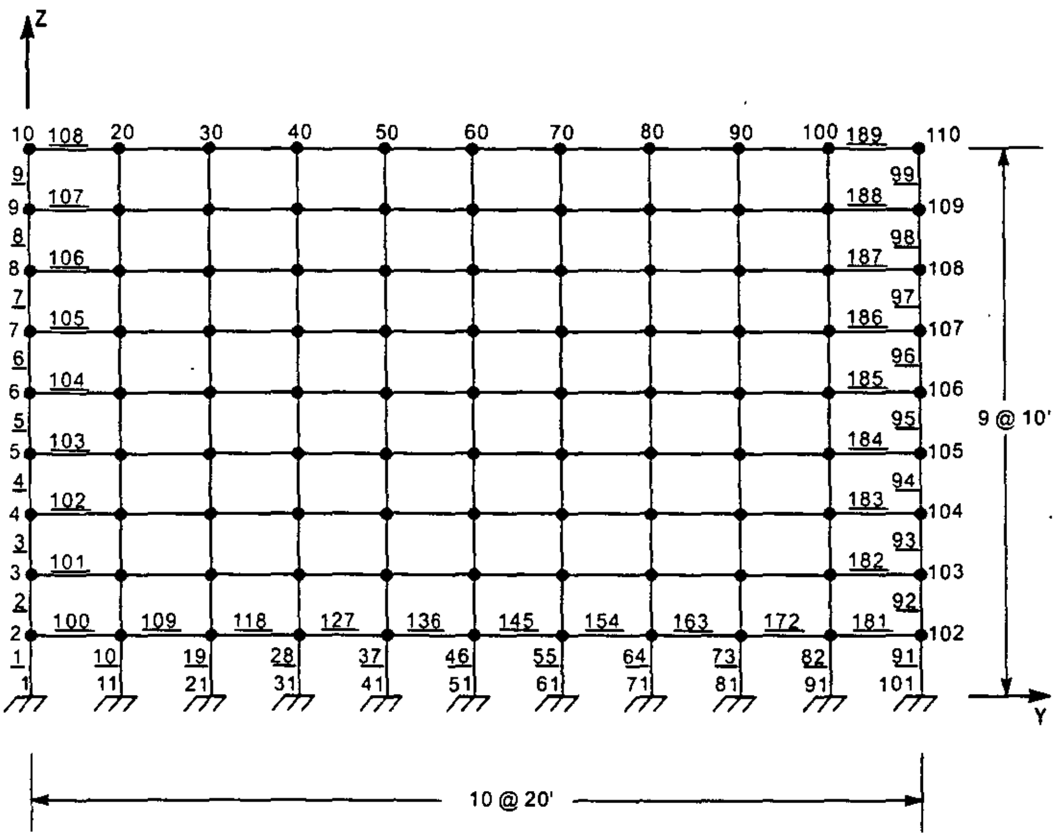
Comparison of Results

A comparison of the first three eigenvalues computed by SAP2000 with results from References [1] and [2] is presented in Figure 2-2. The comparison is excellent.

References

1. **Bathe, K. J. and Wilson, E. L.**
   Large Eigenvalue Problems in Dynamic Analysis, Journal of the Eng. Mech. Div., ASCE, Vol. 98, No. EM6, Proc. Paper 9433, Dec. 1972.

2. **Peterson, F. E.**
   EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual, Engineering Analysis Corporation, Berkeley, California, 1981.



4 JOINT NUMBER
4 MEMBER NUMBER

TYPICAL PROPERTIES

AREA	= 3 ft^2
MOMENT OF INERTIA	= 1 ft^4
MODULUS OF ELASTICITY	= 4.32×10^5 ksf
MASS PER UNIT LENGTH	= 3 kip-sec^2/ft/ft

Figure 2-1
Bathe and Wilson Frame Example

Mode	SAP2000	Reference [1]	Reference [2]
1	0.589541	0.589541	0.589541
2	5.52696	5.52695	5.52696

Figure 2-2
Comparison of Eigenvalues

Example 3

Three-dimensional Frame — Dynamic Loads

Description

This is a two-story, three-dimensional frame structure with rigid diaphragm floors. The problem is the same as the one solved in Reference [1]. The framing is shown in Figures 3-1 and 3-2. The structure is doubly symmetric in plan, except that the center of gravity at each story level is eccentric and is given by coordinates X = 38 feet and Y = 27 feet, represented in Figure 3-1 by joints 28 and 29.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Rigid diaphragm modeling
- Response spectrum analysis

Input Data

The computer model used is shown in Figures 3-1 and 3-2. Kip-foot units are used. An additional joint is added to each story at the center of gravity, and all story mass is given at these two joints.

Two rigid Diaphragm constraints are defined, one each for Stories 1 and 2. All joints on story 1 are constrained together, including the joint at the center of gravity. Similarly, all joints for story 2 are constrained together. For each story, the X and Y displacements and the Z rotations for all joints are dependent upon each other.

Masses at the centers of gravity are specified in the X and Y directions. No rotational mass inertia is used for consistency with Reference [1]. It should be noted that the problem has only four natural modes. All four modes are used in the analysis.

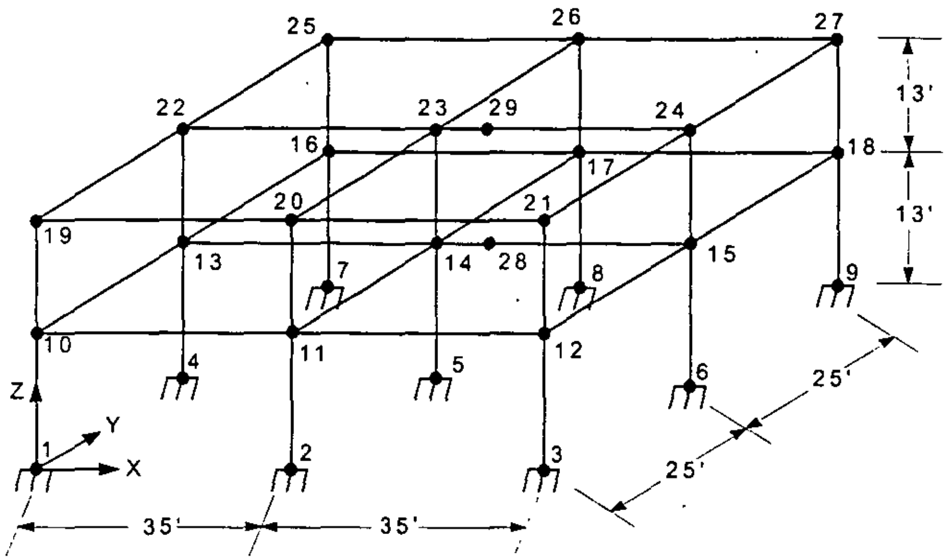
The input data file for this example is FRAME3D.

Comparison of Results

A comparison of the SAP2000 results with Reference [1] results for the four natural periods of vibration and the X-deflection at Joint 29 is presented in Figure 3-3. The comparison is excellent.

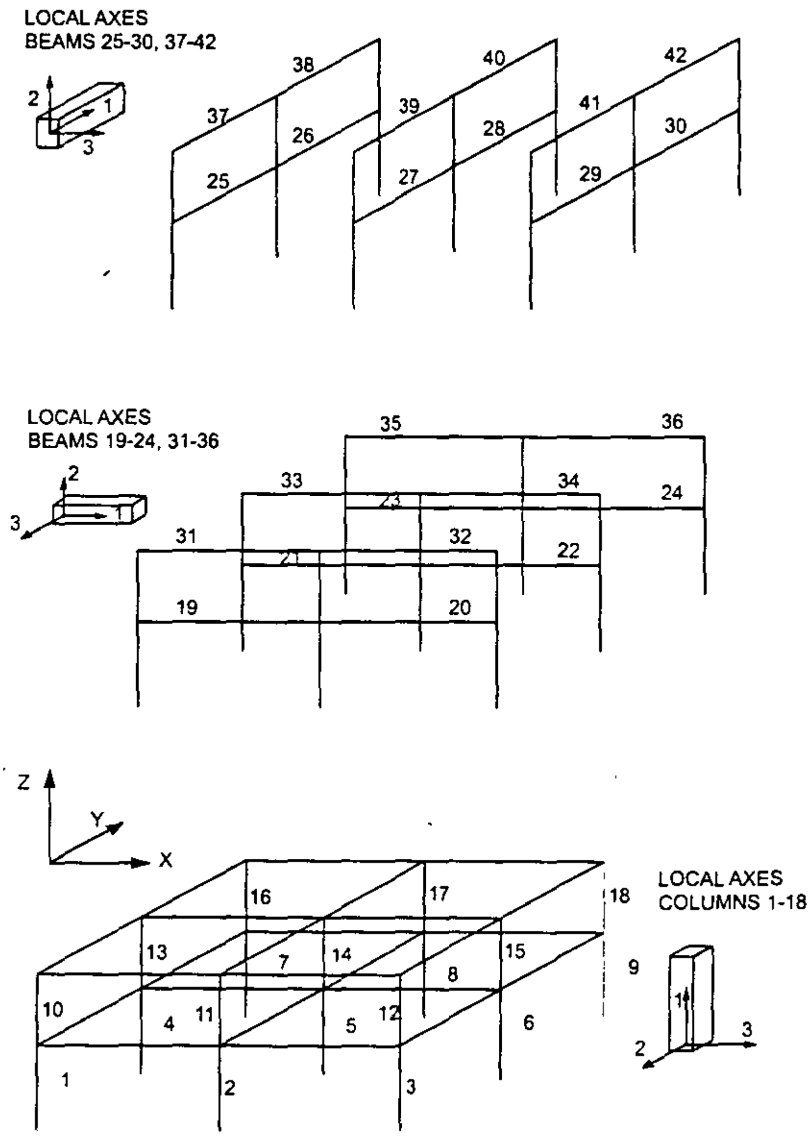
Reference

1. **Peterson, F. E.**
EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual, Engineering Analysis Corporation, Berkeley, California, 1981.



STORY 1 C.G. (NODE 28) AT (38,27,13)
STORY 2 C.G. (NODE 29) AT (38,27,26)
TYPICAL STORY MASS = 6.212 kip-sec²/ft

Figure 3-1
Three-dimensional Frame Example: Dimensions



TYPICAL PROPERTIES

	COLUMNS	BEAMS
AREA	= 4 ft^2	5 ft^2
MOMENT OF INERTIA, I_{22}	= 1.25 ft^4	1.67 ft^4
MOMENT OF INERTIA, I_{33}	= 1.25 ft^4	2.61 ft^4
MODULUS OF ELASTICITY	= 3.50×10^5 ksf	5.00×10^5 ksf

Figure 3-2
Three-dimensional Frame Example: Element Properties

Quantity	SAP2000	Reference [1]
Period, Mode 1	0.2271	0.2271
Period, Mode 2	0.2156	0.2156
Period, Mode3	0.0733	0.0733
Period, Mode 4	0.0720	0.0720
X Deflection, Joint 29	0.0201	0.0201

Figure 3-3
Comparison of Results

Example 4

ASME Frame — Eigenvalue Problem

Description

This is a single-story, single-bay in each direction, three-dimensional frame structure made of 2-inch steel pipe segments and 2.75-inch steel cubes as shown in Figure 4-1. The frame is the same as modeled in References [1] and [2] and is Problem 1 from the ASME 1972 Program Verification and Qualification Library (Reference [3]).

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Use of rigid end offsets on Frame elements
- Eigenvalue analysis
- Ritz vector analysis

Input Data

The computer model used is shown in Figure 4-2. The pipe segments are modeled using Frame elements. For consistency with Reference [1], masses are specified at the nodes instead of using the Frame member mass per unit length specification and additional nodal masses only for the solid cubes. Masses identical to those used in

Reference [1] are specified. Since masses at 14 nodes are specified in three directions, this problem has 3 x 14 = 42 dynamic degrees of freedom. The first 24 modes are calculated using both eigenvectors and Ritz vectors. The Ritz vectors are calculated using ground acceleration in the three global directions as the starting load vectors. The input data file for this example using Ritz vector analysis is FRAMASME. For eigenvector analysis, just change the type of modes requested.

Comparison of Output

A one-to-one comparison of SAP2000 results with References [1] and [2] results for this problem is not possible. This is because both References [1] and [2] use the Guyan reduction method to reduce the 42 dynamic degrees of freedom problem to 24. This introduces approximations into the solution. A comparison of the results is presented in Figure 4-3 for the first 12 natural frequencies. Two sets of results are presented for SAP2000: one using Ritz vector analysis, and one using eigenvector analysis.

The comparison between SAP2000 and References [1] and [2] is good considering the modeling differences between the different solutions. The SAP2000 eigenvector and Ritz-vector results are essentially the same for the first 11 modes, but begin to differ in the higher modes. In general, only the eigenvectors represent the natural modes of the structure. The Ritz vectors are a better basis for response-spectrum and time-history analyses, but may not have the same frequencies and mode shapes as the eigenvectors.

References

1. **Peterson, F. E.**
   EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual, Engineering Analysis Corporation, Berkeley, California, 1981.

2. **DeSalvo, G. J. and Swanson, J. A.**
   ANSYS, Engineering Analysis System, Examples Manual, Swanson Analysis Systems, Inc., Elizabeth, Pennsylvania, 1977.

3. *Program Verification and Qualification Library*, ASME Pressure Vessel and Piping Division, Committee on Computer Technology, 1972.

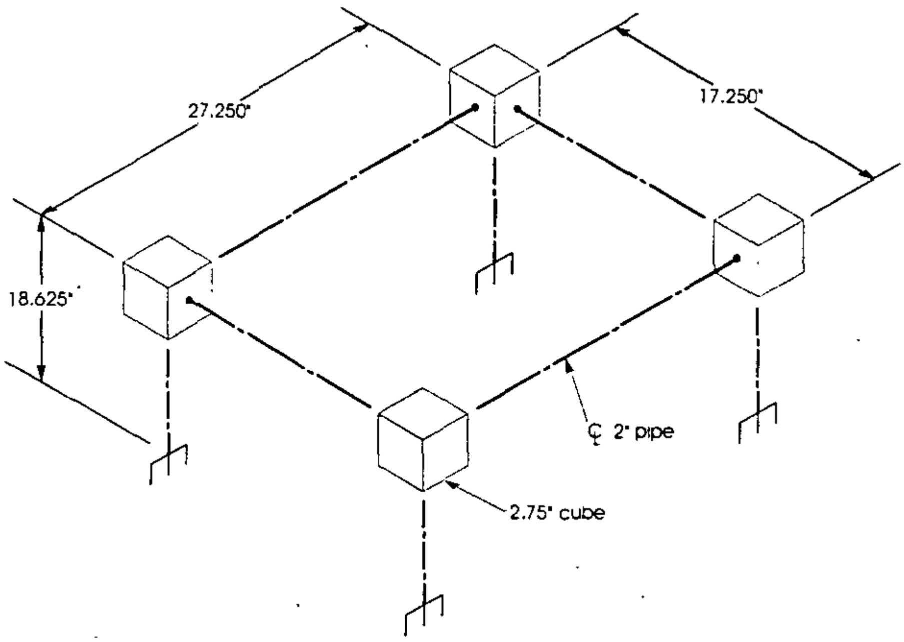
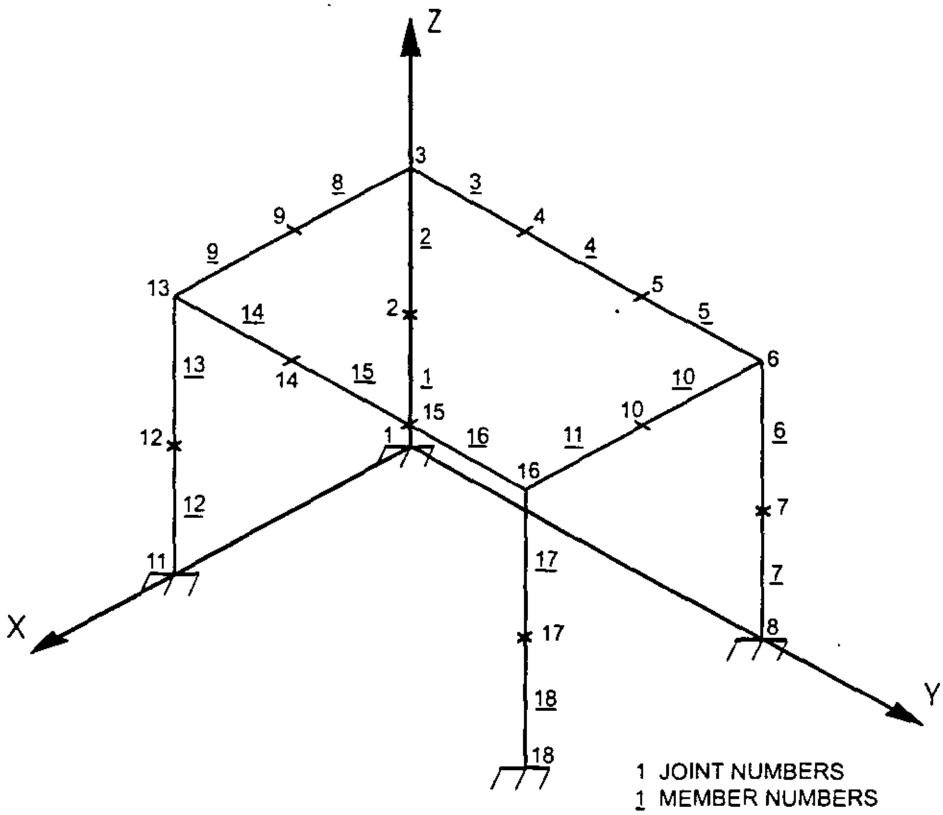


Figure 4-1
ASME Frame Example



TYPICAL PROPERTIES
2.375 in O.D. PIPE 0.154" THICK
MODULUS OF ELASTICITY = 27.9×10^6 psi
POISSON'S RATIO = 0.3
JOINT MASSES = 8.942×10^{-3} lb-sec^2/in
ADDITIONAL MASSES AT
JOINTS 3,6,13 and 16 = 16.439×10^{-3} lb-sec^2/in

Figure 4-2
ASME Frame Model

Mode	SAP2000 Ritz Analysis	SAP2000 Eigen Analysis	Reference [1] & Reference [2] *
1	114	114	112
2	119	119	116
3	141	141	138
4	222	222	218
5	399	399	404
6	422	422	423
7	450	450	452
8	550	550	554
9	774	774	736
10	800	799	762
11	909	909	853
12	955	946	894

* *Both results are based upon Guyan reduction from 42 to 24 dynamic degrees of freedom.*

Figure 4-3
Comparison of Modal Frequencies

Example 5

Three-dimensional Braced Frame — Dynamic Loads

Description

This is a three-story, L-shaped braced frame structure. The floors act as rigid diaphragms. The framing consists of four identical frames and the frame members carry only axial loads. The problem is identical to the one solved in Reference [1]. The framing is shown in Figure 5-1.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Axial-load-only frame members
- Rigid diaphragm modeling
- Response spectrum analysis

Input Data

The computer model used for Frame 1 is shown in Figure 5-2. Kip-inch units are used. The models for Frames 2, 3 and 4 are identical except that node numbers are incremented by 12 and member numbers by 21, for each successive frame. It must be noted that the common column between Frames 2 and 3 is modeled twice, once for each frame. This is done for consistency with the modeling in Reference [1].

Joints 49, 50 and 51 are specified at the center of gravity of stories 1, 2 and 3, respectively. X- and Y-direction masses and the mass moment of inertia about the Z-axis are specified at these joints. All joints on a given story are connected together using a Diaphragm constraint.

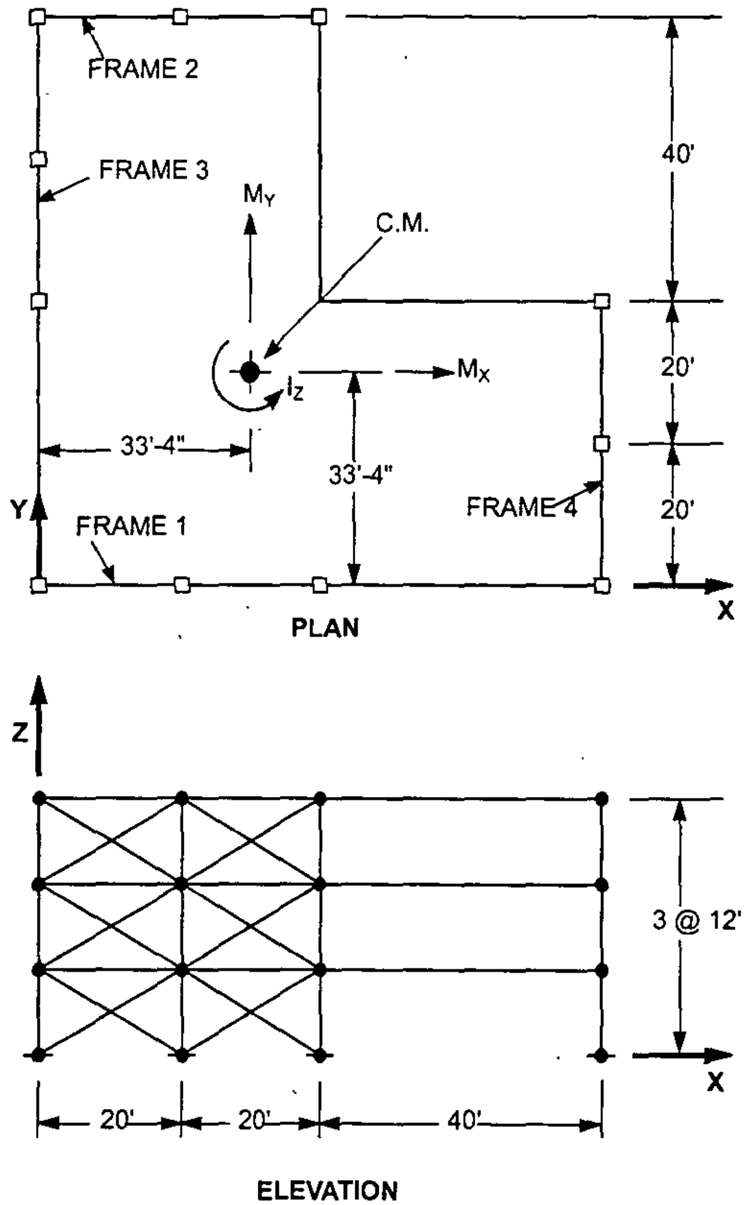
The modeling is identical to that used in Reference [1]. Also, only the first two modes are used in the response spectrum analysis. The input data file for this example is FRAMBRAC.

Comparison of Results

Key results from the SAP2000 analysis are compared with the Reference [1] solutions in Figure 5-3. The comparison is excellent.

Reference

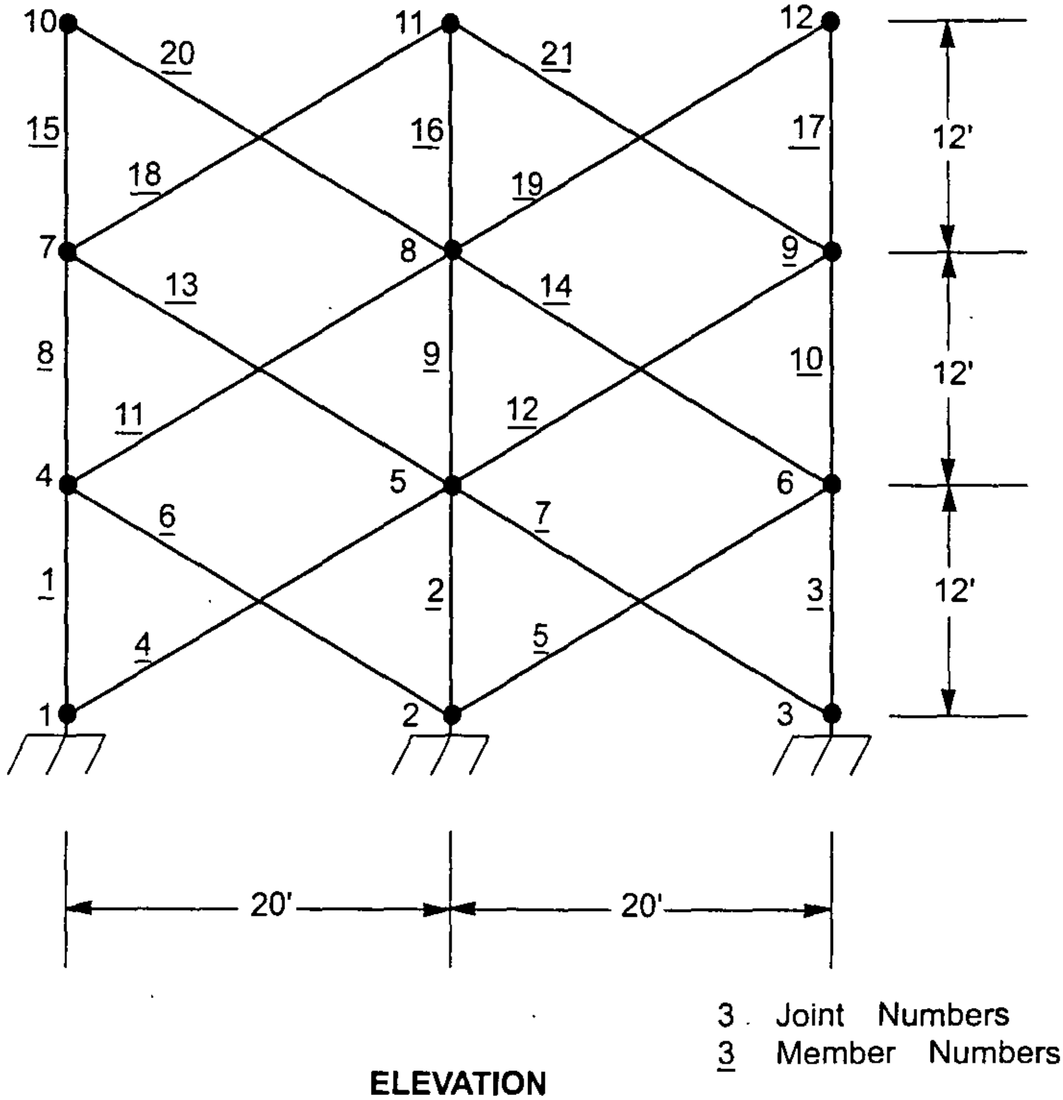
1. **Peterson, F. E.**
EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual, Engineering Analysis Corporation, Berkeley, California, 1981.



TYPICAL STORY MASSES, M_X and M_Y = 1.24224 kip-sec^2/in

TYPICAL STORY MASS MOMENT OF INERTIA, I_Z = 174.907X10^3 kip-sec^2-in

Figure 5-1
Three-dimensional Frame Example



TYPICAL PROPERTIES
AREA = 6 in^2
MODULUS OF ELASTICITY = 29500 ksi

Figure 5-2
Three-dimensional Braced Frame Example — Model of Frame 1

Quantity	SAP2000	Reference [1]
Period, Mode 1	0.326887	0.326887
Period, Mode 2	0.320640	0.320640
Axial Force, Member 1	279.47	279.48
Axial Force, Member 4	194.50	194.50
Axial Force, Member 5	120.52	120.52

Figure 5-3
Comparison of Results for Three-dimensional Braced Frame Example

Example 6

Beam — Steady-State Harmonic Loads

Description

This is a fixed-end beam in two dimensions subjected to a uniformly distributed load which varies harmonically with respect to time. The beam is shown in Figure 6-1. The problem is the same as given in Reference [1]. The beam is solved twice, once using the undamped, steady-state analysis option and once using the periodic loading, time-history analysis option.

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Steady-state analysis
- Time-history analysis for periodic loading

Input Data

The computer model used is shown in Figure 6-1. Pound-inch units are used.

For the steady-state option the frequency of the forcing function is provided in cycles per second. The input data file for this option is BEAM.

For the time-history option the time function (sine wave) portion of the loading is discretized at 37 points at equal intervals covering one complete cycle of loading. See Figure 6-2. The input data file for this example is BEAMTH.

Comparison of Results

Reference [1] computes the deflection at the center of the beam to be -0.0541 sin 300t. The SAP2000 result for the amplitude of this deflection for the steady-state option is -0.054535. It should be noted that Reference [1] uses only the first five mode shapes of the beam for its computation. The SAP2000 results for the time-history option are shown in Figure 6-3. The maximum amplitude being reported is -0.05440. The comparison of results for both methods of analysis in SAP2000 with the theoretical results is excellent.

Reference

1. **Paz, M.**
   Structural Dynamics, Theory and Computations, Van Nostrand Reinhold, 1985.

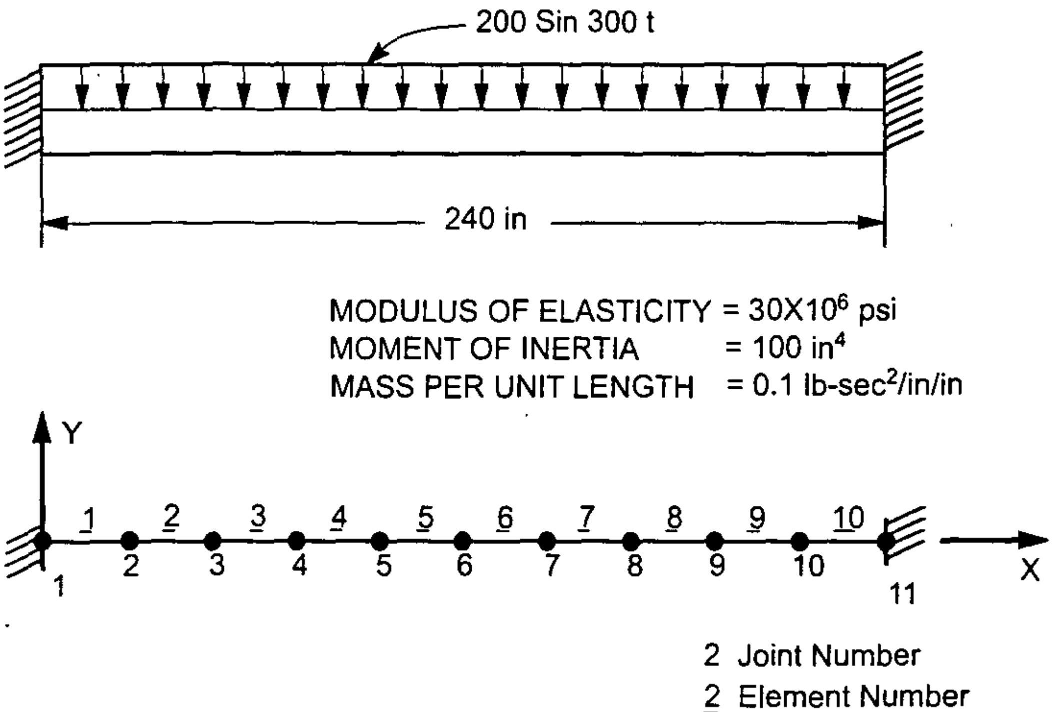


Figure 6-1
Beam Example

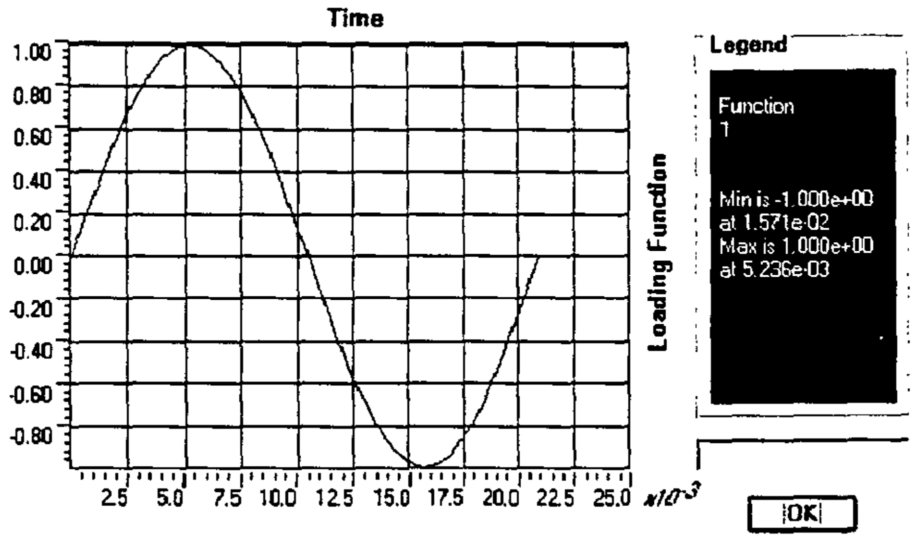


Figure 6-2
Time Variation of Loading

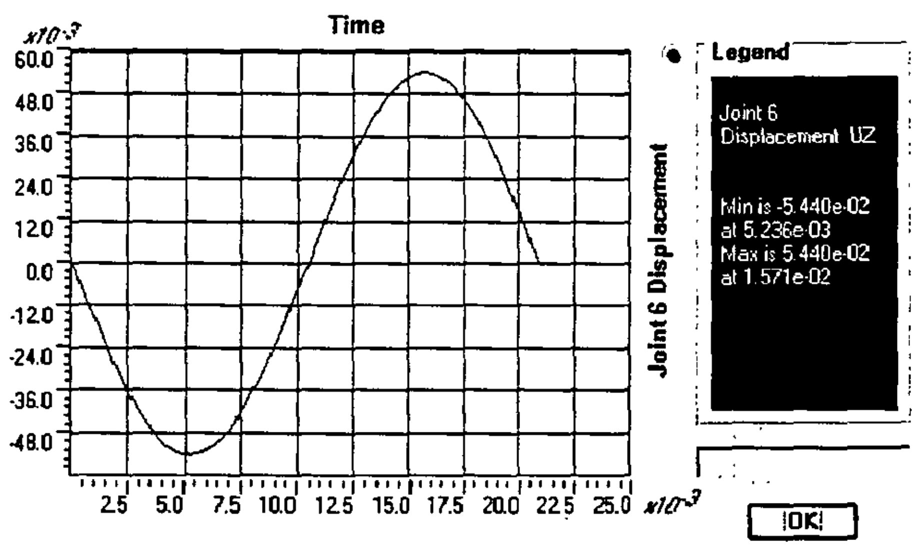


Figure 6-3
Center Span Displacement

Example 7

Two-dimensional Truss — Static Loads

Description

This is a two-dimensional truss structure. All members can carry only axial loads. The truss is shown in Figure 7-1.

Significant Options of SAP2000 Activated

- Two-dimensional truss analysis
- Vertical joint loads

Input Data

The computer model used is shown in Figure 7-1. Truss members are modeled using frame elements with zero moments of inertia. Load case 1 is live load, and Load case 2 is dead load. Load combination 1 is dead load plus live load.

The input data file for this example is TRUSS.

Comparison of Results

This example is included as a sample only. Other results for comparison are not available.

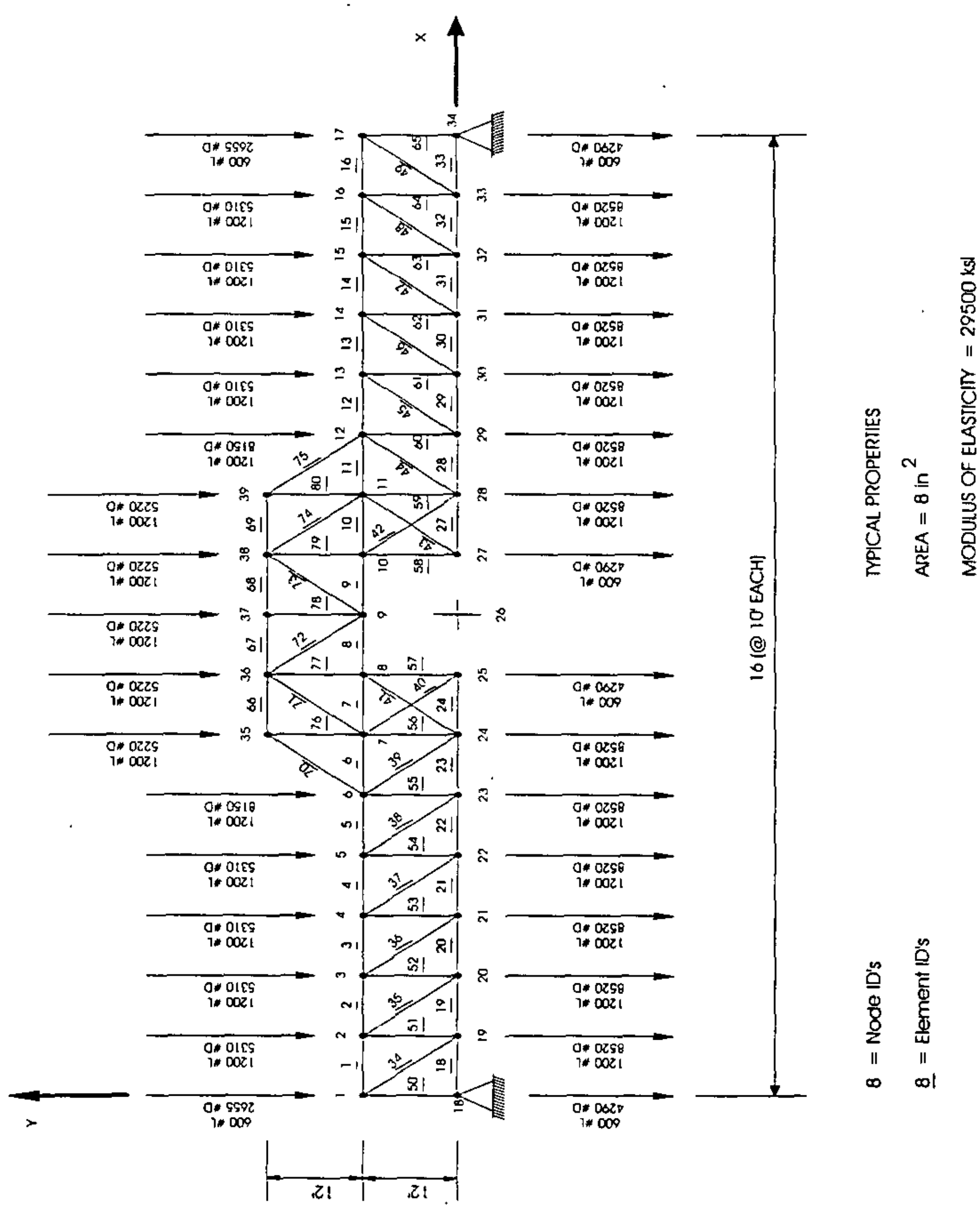


Figure 7-1
Two-dimensional Truss Example

Example 8

Three-dimensional Building — Dynamic Loads

Description

This is a two-story, three-dimensional framed building. The structure is shown in Figure 8-1. The floors act as rigid diaphragms. The building is unsymmetrical and is subjected to lateral dynamic loads along two horizontal axes at a 30° angle to the building axes.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Rigid diaphragm modeling
- Response-spectrum analysis in two directions

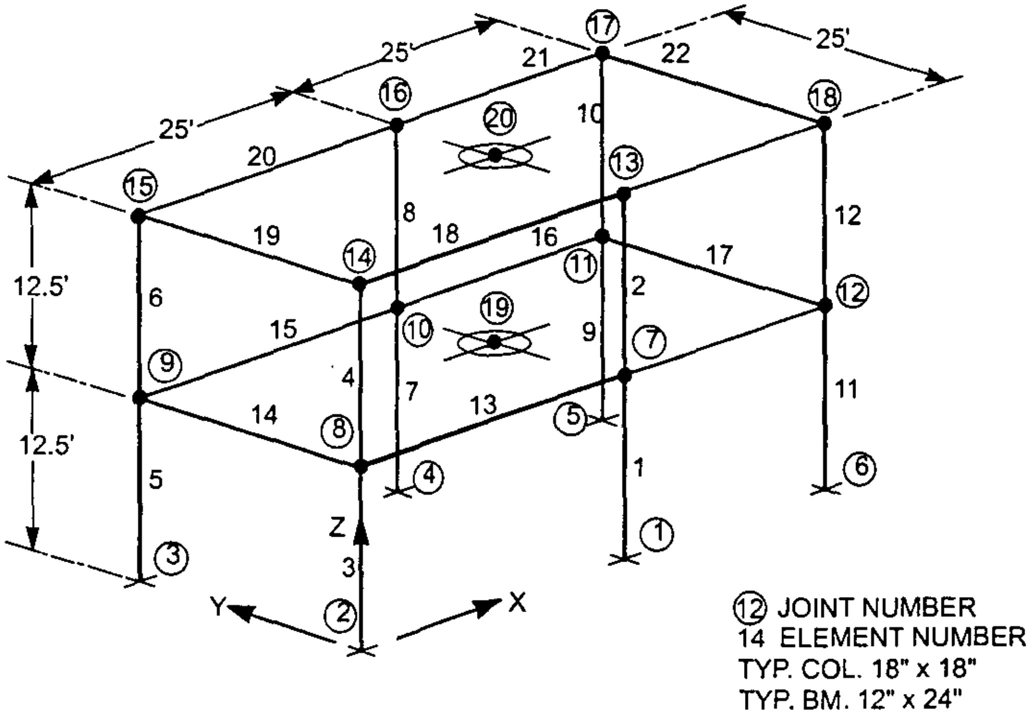
Input Data

The computer model used is shown in Figure 8-1. Kip-foot units are used. Joints 19 and 20 are specified at the center of gravity of Stories 1 and 2, respectively, and the X- and Y-direction masses and the mass moment of inertia about the Z axis is defined at these joints only. All six modes of the structure are used for the dynamic analysis. The response spectra are defined for two horizontal axes at a 30° angle to the building axes.

The input data file for this example is BUILDING.

Comparison of Results

This example is included as a sample only. Other results for comparison are not available.



MODULUS OF ELASTICITY = 432000 ksf

TYPICAL STORY MASSES, M_X and M_Y = 3.88 kip-sec²/ft

TYPICAL STORY MASS MOMENT OF INERTIA, I_Z = 1011 kip-sec²-ft

Figure 8-1
Three-dimensional Building Example

Example 9

Patch Tests — Prescribed Displacements

Description

This is a rectangular plate with irregularly-shaped elements and subjected to prescribed displacements at the edges. The plate is shown in Figure 9-1. The location of the inner nodes and the prescribed displacements are the same as suggested in Reference [1]. The problem is solved using both Plane Stress elements and Shell elements.

Significant Options of SAP2000 Activated

- Plane stress analysis using 4-node Plane elements
- Displacement specification
- Plate bending and membrane analysis using Shell elements

Input Data

The computer model used is shown in Figure 9-1. The prescribed displacements at the edges are calculated as:

$$u = 10^{-3}\left(x + \frac{y}{2}\right)$$

$$v = 10^{-3}\,(y + \frac{x}{2})$$

$$w = 10^{-3}\,\frac{(x^2 + xy + y^2)}{2}$$

$$\Theta_x = 10^{-3}\,(y + \frac{x}{2})$$

$$\Theta_y = 10^{-3}\,(-x - \frac{y}{2})$$

These represent constant stress fields.

For the Plane element solution, only the X- and Y-translations are unrestrained. All other degrees of freedom are restrained. For the Shell element solution, all six degrees of freedom are unrestrained because the membrane formulation of this shell element gives rotational stiffness components about a direction normal to the plane of the shell.

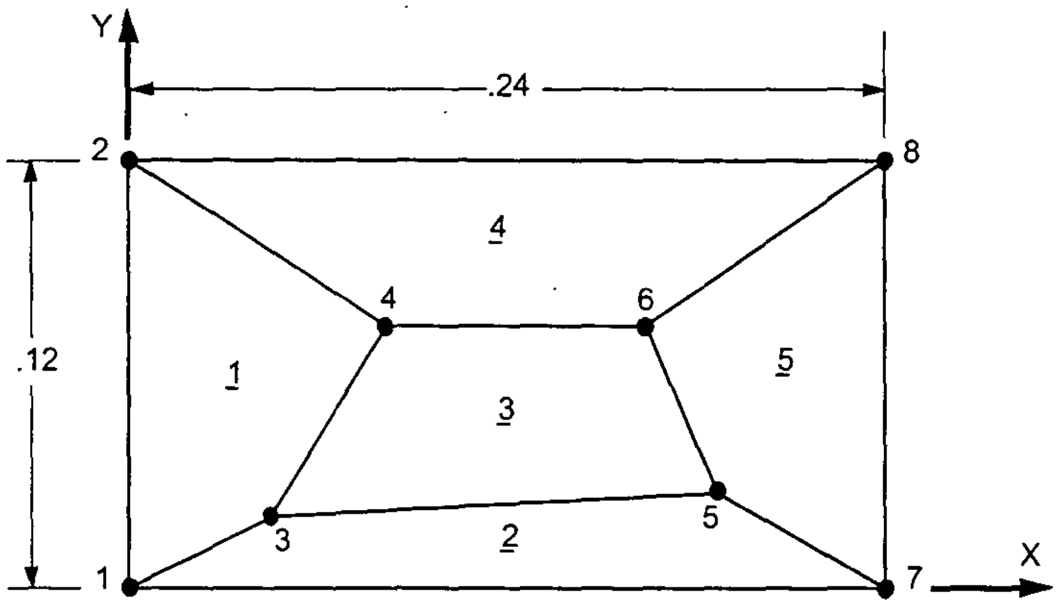
For this example, the input data file using the Plane element is PATCHPLN, and that using the Shell element is PATCHSHL.

Comparison of Results

The theoretical results for the problem are $S_{xx} = S_{yy} = 1333$ and $S_{xy} = 400$ for the membrane components; and $M_{xx} = M_{yy} = 1.111 \times 10^{-7}$ and $M_{xy} = 0.333 \times 10^{-7}$ for the plate bending components. These theoretical results are reproduced by SAP2000.

Reference

1. **MacNeal, R. H. and Harder, R. C.**
A *Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite* Elements in Analysis and Design 1 (1985), pp. - 20, North-Holland.



Location of inner nodes:

	x	y
3	.04	.02
4	.08	.08
5	.18	.03
6	.16	.08

3 Joint Number

3 Element Number

THICKNESS = 0.001

MODULUS OF ELASTICITY = $1.0X10^6$

POISSON'S RATIO = 0.25

Figure 9-1
Patch Test Example

Example 10

Straight Beam — Static Loads

Description

This is a straight cantilever beam of proportions shown in Figure 10-1. The beam has unit forces at the tip in the three orthogonal directions and a unit twist, each modeled as a separate Load case. The beam is modeled using different mesh geometries as suggested in Reference [1]. The problem is solved using Shell elements, nine-node Plane stress elements ,and Solid elements.

Significant Options of SAP2000 Activated

- Shell element analysis
- Plane element analysis with plane-stress option
- Solid element analysis with and without incompatible bending modes

Input Data

Several mesh geometries are used and key results for these are presented. The following data files are provided:

- STRBMSHL for the rectangular mesh of Shell elements.
- STRBMPLN for the rectangular mesh of Plane elements.

- STRBMSOL for the rectangular mesh of Solid elements with incompatible bending modes.

Figures 10-1 to 10-3 represent the models used for the Shell, Plane and Solid elements.

The unit forces at the tip are applied as 1/2 at each node for the Shell element; 1/6 at each corner node and 4/6 at the midside node for the Plane element; and 1/4 at each node for the Solid element. This represents a consistent set of forces for the elements. The unit twist for the Shell and Solid elements is applied as a couple.

For the model using Shell elements the nodes at the fixed end are restrained in the axial direction, in the out-of-plane direction, and for rotations that cause out of plane bending. The transverse, in-plane direction is restrained only at one node so as not to cause local Poisson's effect. The other node is, however, provided with the transverse reaction as a load.

For the model using Plane elements the nodes at the fixed end are restrained in the axial direction and one of them is also restrained in the transverse, in-plane direction for the same reason as for the Shell element. The other nodes are, however, provided with the transverse reactions as loads. All rotations and the out-of-plane direction for all nodes are also restrained.

For the model using Solid elements the nodes at the fixed end are restrained in the axial direction and one of them is also restrained in the two transverse directions for the same reason as for the Shell element. This, however, does not make the model stable, so an additional node is restrained in the vertical direction to prevent rotation about the axial direction. The other nodes are, however, provided with the transverse reactions as loads.

Comparison of Results

The displacements at the tip in the direction of the load is compared for each type of element and the different meshes with the theoretical results in Figures 10-4, 10-5, and 10-6.

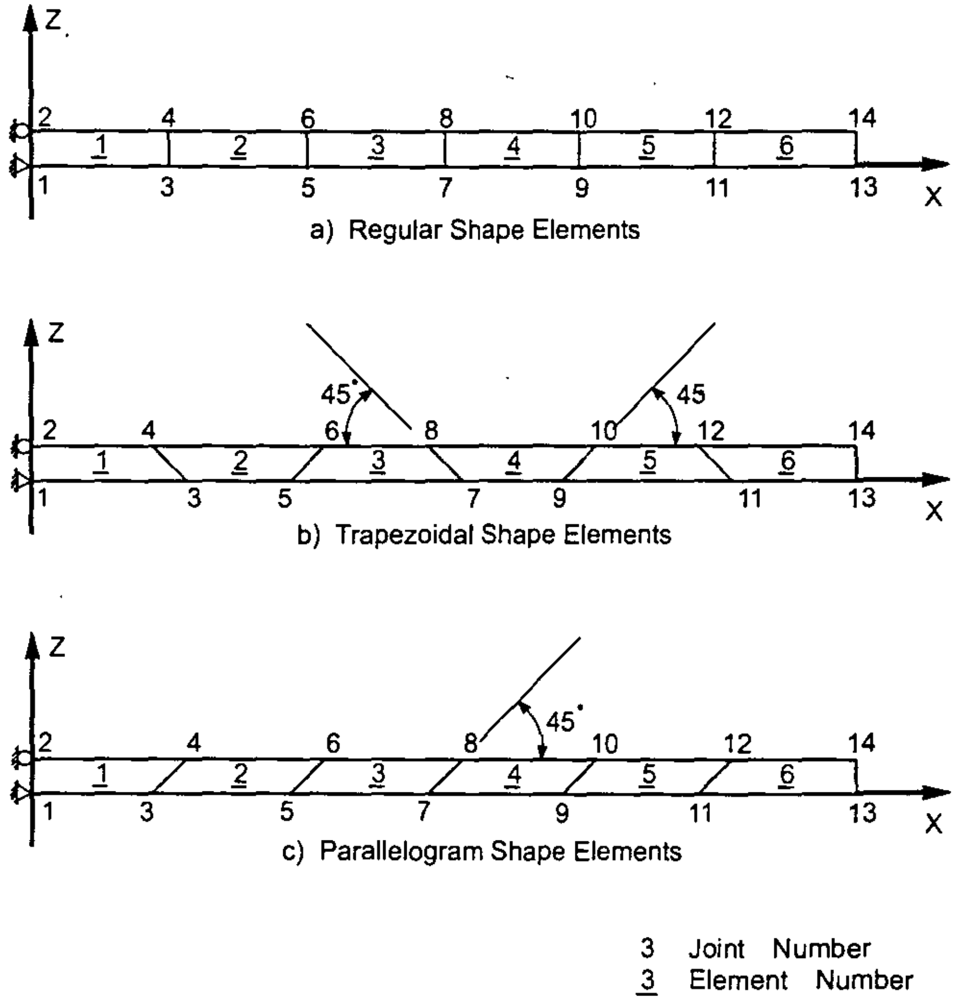
The results for the Shell element are good except for the in-plane shear results for the irregular meshes and the results for the unit twist. The twist results are too stiff. The element is too thick compared to the width for the thin plate twisting behavior the Shell element is capable of modeling. The irregular mesh behavior in in-plane shear can be improved by decreasing the skewness and aspect ratio of the elements and using more elements.

The results for the nine-node Plane element for all mesh geometries is excellent. The four-node Plane element is too stiff in bending and a finer mesh would be required to accurately capture the bending behavior.

The results for the Solid element with a rectangular mesh and using incompatible bending modes are good except for the case with the unit twist. To capture the twisting behavior accurately, more elements are needed across the beam section. The Solid element model without incompatible bending modes is too stiff in the bending mode. A finer mesh along the length of the beam would give better results.

Reference

1. **MacNeal, R. H. and Harder, R. C.**
   A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design 1 (1985), pp. 3-20, North-Holland.



LENGTH	= 6.0
WIDTH	= 0.2
THICKNESS	= 0.1
MODULUS OF ELASTICITY	= $1.0X10^7$
POISSON'S RATIO	= 0.30

LOADING: UNIT FORCES AT FREE END

Figure 10-1

Straight Beam Example - Shell Element Models

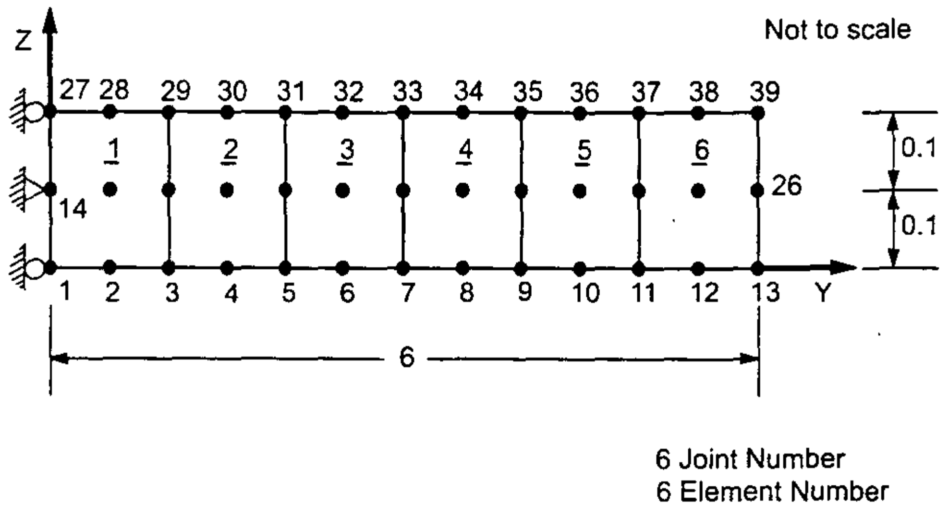


Figure 10-2
Straight Beam Example - Plane Stress Element Model

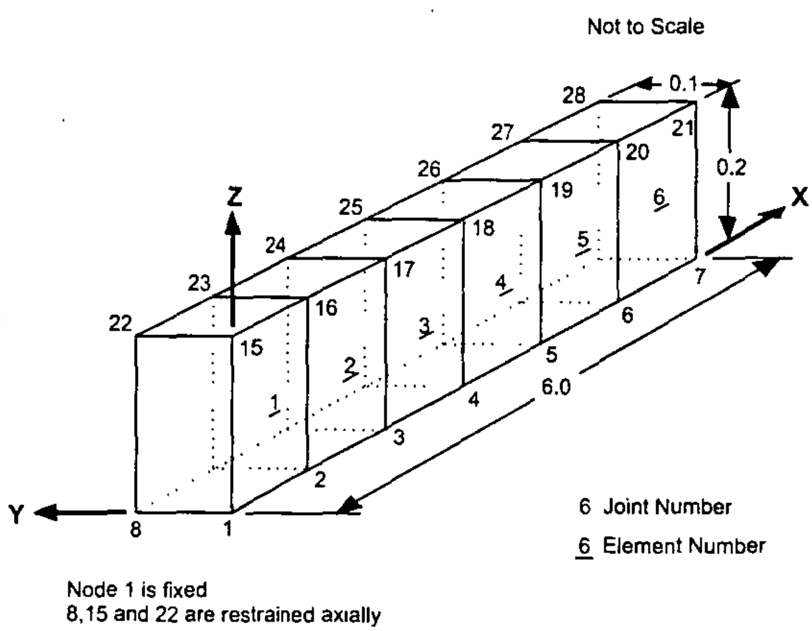


Figure 10-3
Straight Beam Example - Solid Element

Load Direction	SAP2000			Theoretical
	Rectangular Mesh	Trapezoidal Mesh	Parallelogram Mesh	
Extension	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$
Out-of-plane Shear	0.4263	0.4266	0.4266	0.4321
In-plane Shear	0.1072	0.0221	0.0790	0.1081
Twist	0.00233	0.00233	0.00233	0.00321

Figure 10-4
Comparison of Tip Displacements Using Shell Elements

Load Direction	SAP2000				Theoretical
	Rectangular Mesh 4-Node Element	Rectangular Mesh 9-Node Element	Trapezoidal Mesh 9-Node Element	Parallelo-gram Mesh 9-Node Element	
Extension	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$	$3.000x10^{-5}$	3.000x10-5
In-plane Shear	0.0101	0.1076	0.1063	0.1065	0.1081

Figure 10-5
Comparison of Tip Displacements Using Plane Stress Element

Load Direction	SAP2000		Theoretical
	Rectangular Mesh with Incompatible Bending Modes	Rectangular Mesh without Incompatible Bending Modes	
Extension	3.000×10^{-5}	3.000×10^{-5}	3.000×10^{-5}
Out-of-plane Shear	0.4283	0.0109	0.4321
In-plane Shear	0.1072	0.0101	0.1081
Twist	0.00286	0.00286	0.00321

Figure 10-6
Comparison of Tip Displacements Using Solid Elements

Example 11

Curved Beam — Static Loads

Description

This is a curved cantilever beam of proportions shown in Figure 11-1. The beam is loaded with unit shears at the free end. The problem is the same as suggested in Reference [1]. The problem is solved using Shell elements and Plane elements.

Significant Options of SAP2000 Activated

- Shell element analysis
- Plane element analysis using plane- stress option

Input Data

The input data file for the Shell element example is CRVBMSHL, and for the Plane element example is CRVBMPLN.

Comparison of Results

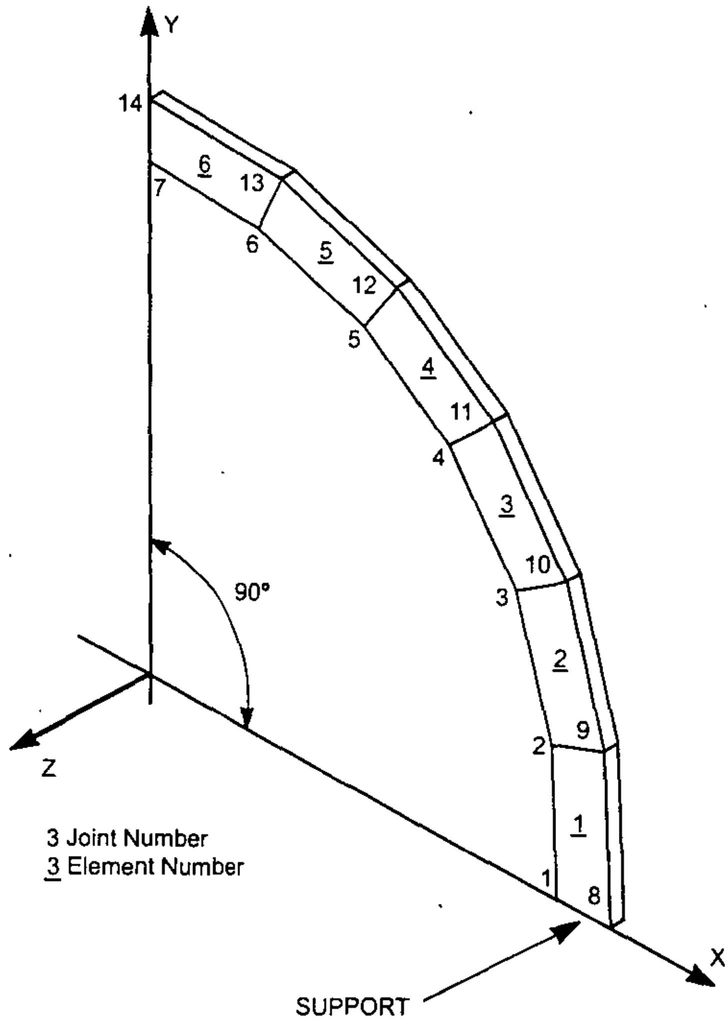
The displacement at the free end in the direction of the load is compared with theoretical results (Reference [1]) for both element types in Figure 11-2.

For the Plane element, more elements along the length of the beam would give better results. The aspect ratio of the element is quite large in this model.

For the Shell element, the in-plane (membrane) results are good: the out-of-plane (bending) results are not as good. This is because of the stiffer twisting behavior when the thickness is large compared with the width.

Reference

1. **MacNeal, R. H. and Harder, R. C.**
   A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design 1 (1985), pp. 3-20, North-Holland.



INNER RADIUS	= 4.12
OUTER RADIUS	= 4.32
ARC	= 90°
THICKNESS	= 0.1
MODULUS OF ELASTICITY	= $1.0X10^7$
POISSON'S RATIO	= 0.25

LOADING: UNIT FORCES AT FREE END

Figure 11-1
Curved Beam Example — Shell Element Model

Load Direction	SAP2000		Theoretical
	9-Node Plane Stress Element	Shell Element	
In-plane Shear	0.0775	0.0851	0.0873
Out-of-plane Shear		0.4518	0.5022

Figure 11-2
Comparison of Tip Displacements

Example 12

Twisted Beam — Static Loads

Description

This is a twisted cantilever beam of proportions shown in Figure 12-1. Unit loads are applied at the free end. The problem is the same as suggested in Reference [1]. The problem is solved using Shell elements.

Significant Options of SAP2000 Activated

- SHELL element analysis

Input Data

The computer model used is shown in Figure 12-1. The input data file for this example is TWSBMSHL.

Comparison of Results

The displacements at the tip in the direction of the loads are compared with theoretical results (Reference [1]) in Figure 12-2. The comparison is excellent.

Reference

1. **MacNeal, R. H. and Harder, R. C.**
A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design 1 (1985), pp. 3-20, North-Holland.

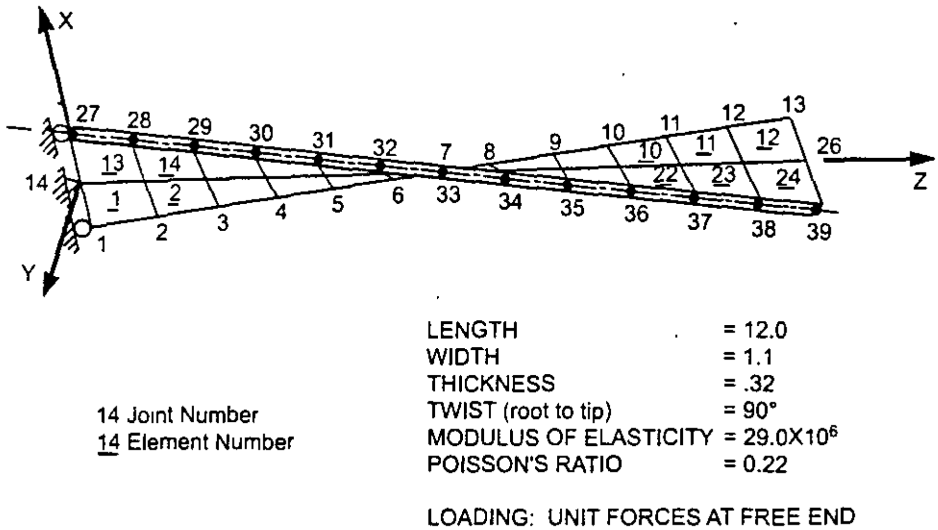


Figure 12-1
Twisted Beam Example

Load Direction	SAP2000 Shell Element 12x12 Mesh	Theoretical
In-plane Shear	0.005413	0.005424
Out-of-plane Shear	0.001770	0.001754

Figure 12-2
Comparison of Tip Deflections

Example 13

Beam On Elastic Foundation — Static Loads

Description

This is a simply supported beam on an elastic foundation. Half of the beam is modeled as shown in Figure 13-1. The geometry and the loads are the same as used in Reference [1].

Significant Options of SAP2000 Activated

- Plane element analysis using the plane stress option
- Spring supports representing elastic foundations

Input Data

The computer model used is shown in Figure 13-1. Pound-inch units are used. Nine-node Plane elements with the plane stress option are used. Half of the beam is modeled using 10 elements and symmetry is utilized to obtain the boundary conditions. Springs are used to model the elastic foundation.

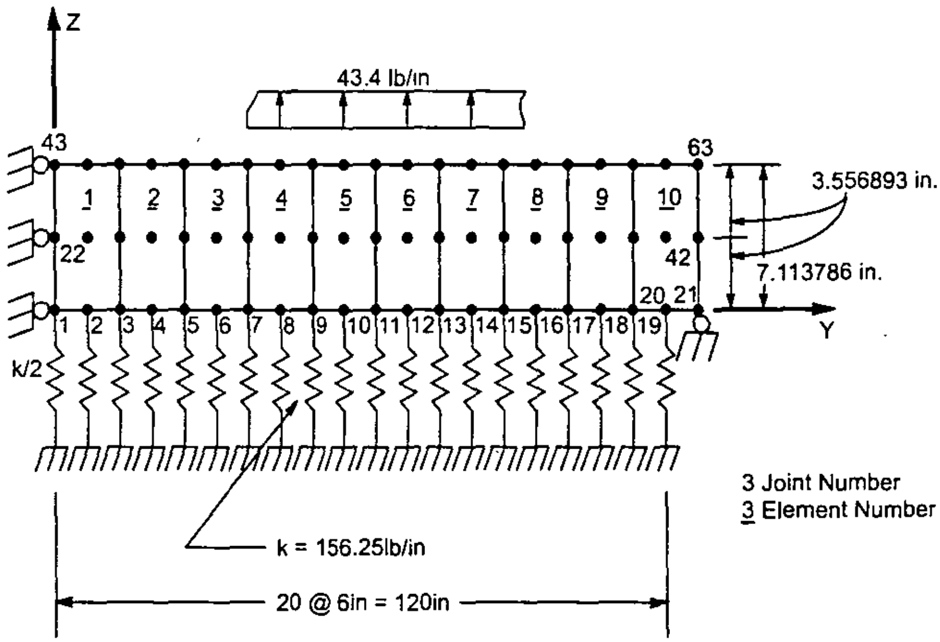
The input data file for this example is BEAMONFN.

Comparison of Results

The transverse displacements along the center of the beam and the maximum bending stress (using the average of the values at the top and bottom fiber) are compared with theoretical results (Reference [1]) in Figure 13-2. The comparison is good.

Reference

1. **Peterson, F. E.**
EASE2, Elastic Analysis for Structural Engineering, Example Problem Manual, Engineering Analysis Corporation, Berkeley, California, 1981.



THICKNESS = 1 in
MODULUS OF ELASTICITY = $30X10^6$ psi

Figure 13-1
Beam on Elastic Foundation Example

Axial Station	Transverse Displacements		Maximum Bending Stress	
	SAP2000	Theoretical	SAP2000	Theoretical
0	1.0458	1.0453		
6			18029	18052
12	1.0336	1.0331		
18			17751	17773
24	0.9973	0.9967		
30			17183	17206
36	0.9373	0.9367		
42			16304	16327
48	0.8546	0.8541		
54			15082	15106
60	0.7507	0.7502		
66			13476	13501
72	0.6275	0.6270		
78			11436	11462
84	0.4874	0.4870		
90			8902	8930
96	0.3335	0.3331		
102			5810	5839
108	0.1695	0.1693		
114			2089	2119
120	0.0002	0.0000		

Figure 13-2
Comparison of Results

Example 14

Rectangular Plate — Static Loads

Description

This is a rectangular plate as shown in Figure 14-1. This problem is solved using two different aspect ratios: one square plate of 2 x 2, and another rectangular plate of 2 x 10. Two Load cases are used: Load case 1 for a concentrated load at the center and Load case 2 for uniform load. Also, the problems are solved once with the edges clamped, and once with the edges simply supported. The problem is modeled using both Shell and Solid elements. The geometry, properties and loads used are those suggested in Reference [1].

Significant Options of SAP2000 Activated

- Plate bending analysis using Shell elements
- Plate bending analysis using Solid elements
- Static load analysis

Input Data

The computer model using Shell elements is shown in Figure 14-1. For both types of elements, a 6 x 6 mesh is used on a quarter of the plate, with symmetry conditions applied to represent the remainder of the plate. Incompatible bending modes are included for the Solid element solutions.

The input data file for the model using Shell elements with clamped edges is RCPLTSHL. The input data file for the model using Solid elements with simple supports is RCPLTSOL. Both are for a rectangular (2 x 10) plate.

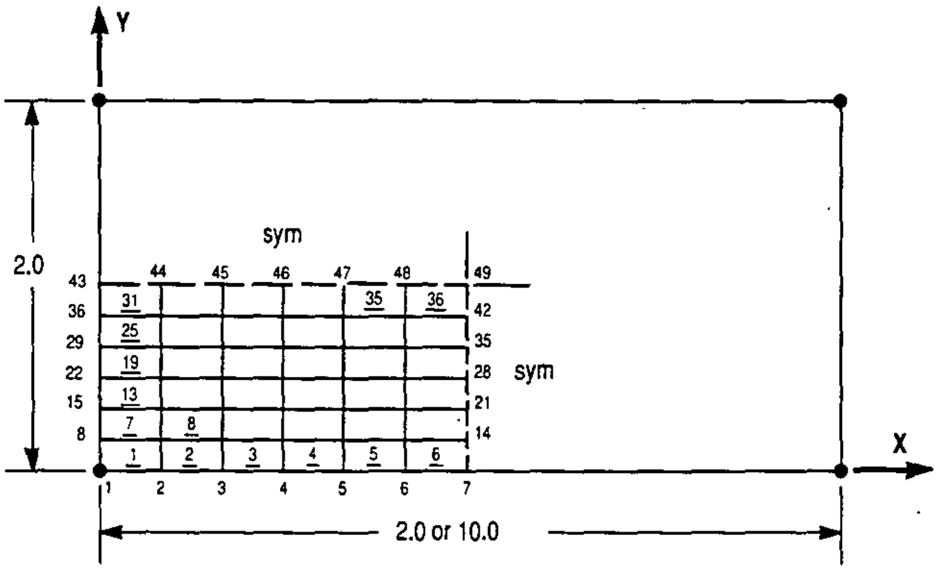
Two different thicknesses are used in models using Solid elements. One is 100 times thicker than the model using Shell elements, the other is 1000 times thicker. The loads for each model are inversely proportional to the cube of the thickness so that all displacements are of the same order of magnitude.

Comparison of Results

The central deflection results for the various boundary conditions, loading conditions, and element types are compared in Figure 14-2 with the theoretical results. The comparison is good. The thicker model using SOLID elements gives better results because of the improved aspect ratio.

Reference

1. **MacNeal, R. H. and Harder, R. C.**
A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design 1 (1985), pp. 3-20, North-Holland.



6 Joint Number
6 Element Number

THICKNESS (SHELL ELEMENT) = 0.0001
THICKNESS (SOLID ELEMENT) = 0.01 or 0.1
MODULUS OF ELASTICITY = 1.7472×10^7
POISSON'S RATIO = 0.3

LOADING: UNIFORM LOAD = 1.0×10^{-4}
CENTRAL LOAD = 4.0×10^{-4}

BOUNDARIES: SIMPLY SUPPORTED OR CLAMPED

Figure 14-1
Rectangular Plate Example

Loading	Boundary Condition	Aspect Ratio (b/a)	SAP2000			Theoretical
			0.0001 Thick Shell Element	0.01 Thick Solid Element	0.1 Thick Solid Element	
Uniform	Simply Supported	1.0	4.061	3.175	4.084	4.062
		5.0	12.92	9.37	12.91	12.97
	Clamped	1.0	1.29	0.83	1.30	1.26
		5.0	2.60	1.83	2.61	2.56
Concentrated	Simply Supported	1.0	11.77	8.76	11.89	11.60
		5.0	17.74	6.40	15.97	16.96
	Clamped	1.0	5.76	3.44	5.87	5.60
		5.0	7.80	1.78	6.39	7.23

Figure 14-2
Comparison of Deflection at Center

Example 15

Cantilever Plate — Eigenvalue Problem

Description

This is a square cantilever plate and the first five eigenvalues are computed. The plate is shown in Figure 15-1.

Significant Options of SAP2000 Activated

- Shell elements for plate bending analysis
- Eigenvalue analysis
- Large capacity analysis — over 4900 dynamic degrees of freedom

Input Data

Three different mesh sizes are used: a 10 x 10 mesh, a 19 x 19 mesh, and a 40 x 40 mesh. The computer model used for the 19 x 19 mesh is shown in Figure 15-1. Kip-inch units are used. Unit mass for the plate is specified to generate the mass matrix.

The input data file is PLATE for the 19 x 19 mesh and LARGEPLT for the 40 x 40 mesh.

Comparison of Output

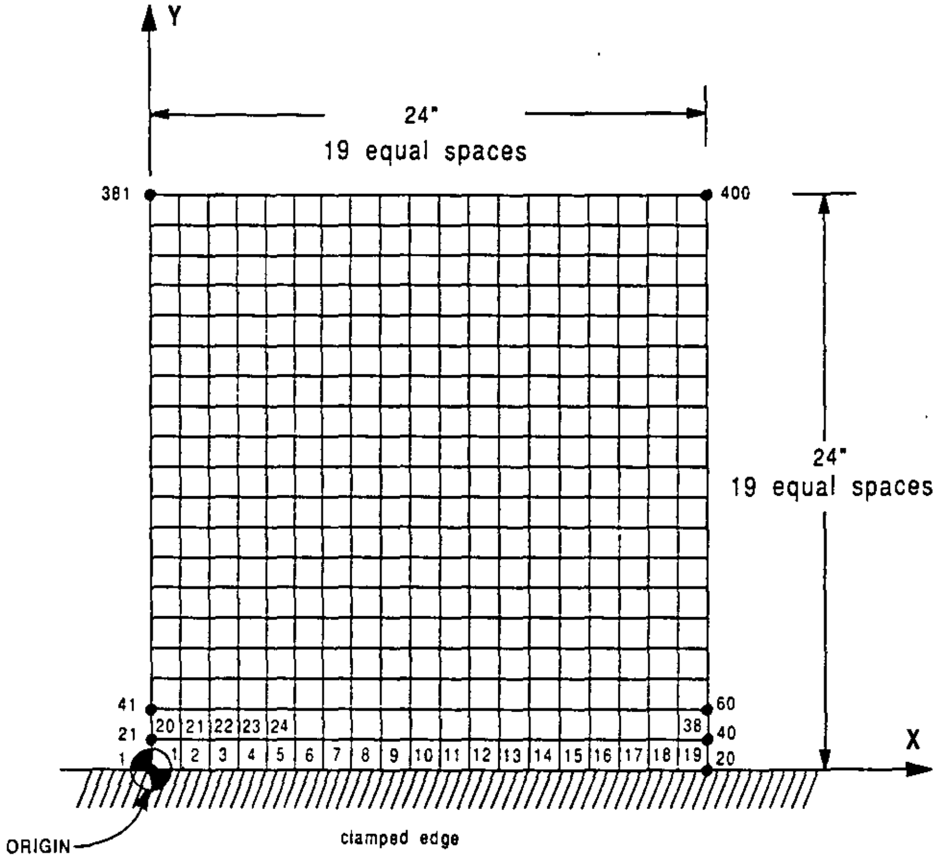
Reference [1] gives results for the first five natural frequencies of a square cantilever plate obtained by using the Ritz method with beam mode shapes. A comparison of the SAP2000 results with the Reference [1] results is given in Figure 15-2.

The comparisons of the periods of vibration for the first, third and fourth modes are excellent; however, the comparisons of the periods of vibration for the second and fifth modes are not as good because the results of Reference [1] involve approximating plate mode shapes with beam mode shapes.

The SAP2000 results using three different mesh sizes are very close.

Reference

1. **Harris, C. M. and Crede, C. E.**
   Shock and Vibration Handbook, McGraw-Hill, 1976.



PLAN

PLATE 24"x 24"x 1"
MODULUS OF ELASTICITY = 29500 ksi
MASS DENSITY = 0.49/1728/386.4 kip-sec^2/in/in^3

Figure 15-1
Cantilever Plate Example

Mode	SAP2000			Reference [1]*
	40 x 40 MESH	19 x 19 MESH	10 x 10 MESH	
1	0.01779	0.01781	0.01787	0.01790
2	0.00647	0.00648	0.00654	0.00732
3	0.00284	0.00285	0.00288	0.00292
4	0.00221	0.00223	0.00228	0.00228
5	0.00186	0.00187	0.00190	0.00201

* *Using Ritz's method with beam mode shapes*

Figure 15-2
Comparison of Natural Periods of Vibration

Example 16

Scordelis-Lo Roof — Static Loads

Description

This is a reinforced concrete single span cylindrical roof as shown in Figure 16-1. The roof is to be analyzed for gravity loads. The problem solved is the same as suggested in Reference [1].

Significant Options of SAP2000 Activated

- Three-dimensional shell analysis
- Gravity load analysis
- Large capacity analysis — 7991 static degrees of freedom

Input Data

Two different mesh sizes are used for this problem: one, a 6 x 6 mesh used on a quarter of the roof as shown in Figure 16-1; another, an 18 x 72 mesh used on half the roof. Pound-foot units are used. Symmetry boundary conditions is used to represent the omitted parts of the structure. Unit weight is specified to apply the uniformly distributed loading.

The input data file for the 6 x 6 mesh is ROOF. The input data file for the 18 x 72 mesh is LARGEROF.

Comparison of Results

The theoretical vertical deflection at the center of the free edge is 0.3086, and Reference [1] suggests a value of 0.3024 for comparison of finite element behavior. SAP2000 gives a value of 0.3068 for the 6 x 6 mesh and 0.3012 for the 18 x 72 mesh. The comparison is excellent.

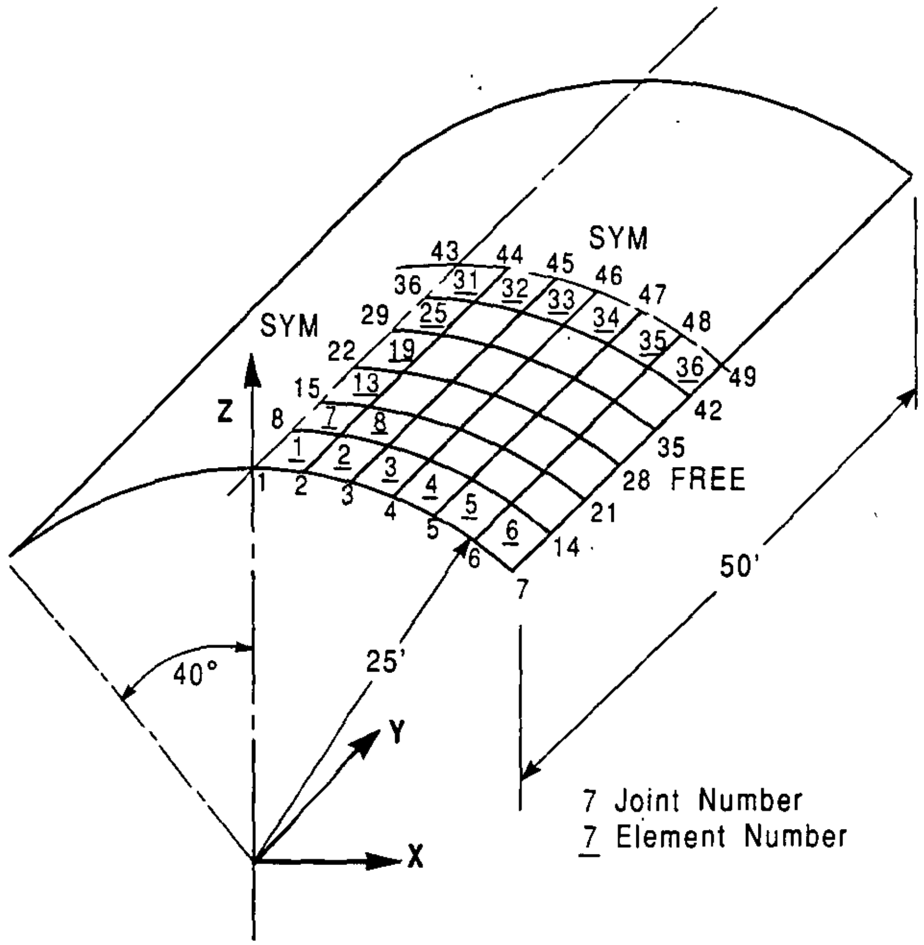
A comparison of SAP2000 displacement and bending moment results using the 6 x 6 mesh, and theoretical results provided in Reference [2] using the theory presented in Reference [3], is given in Figures 16-2 and 16-3. The theoretical results are measured from the figures given in Reference [2]. Nodal averages are used for the SAP2000 bending moments shown. The comparison is good.

References

1. **MacNeal, R. H. and Harder, R. L.**
   A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design 1 (1985), pp. 3-20, North-Holland.

2. **Zienkiewicz, O. C.**
   The Finite Element Method, McGraw-Hill, 1977.

3. **Scordelis, A. C. and Lo, K. S.**
   Computer Analysis of Cylindrical Shells, Journal of the American Concrete Institute, Vol. 61, May 1964.



THICKNESS = 3 in
MODULUS OF ELASTICITY = 4.32 $\times 10^8$ psf
POISSON'S RATIO = 0.0
GRAVITY LOAD = 90 psf
(UNIFORM ON SURFACE AREA)

BOUNDARY CONDITION: SIMPLY SUPPORTED ON CURVED EDGES

Figure 16-1
Scordelis-Lo Roof Example

Location	Axial Deformations at Support		Vertical Displacements at Central Section	
	SAP2000 6x6 Mesh	Theoretical	SAP2000 6x6 Mesh	Theoretical
0°	0.0000	0.0004	0.046	0.045
6.67°	0.0005	0.0009	0.031	0.027
13.33°	0.0018	0.0020	-0.013	-0.018
20.00°	0.0029	0.0030	-0.078	-0.082
26.67°	0.0024	0.0021	-0.155	-0.155
33.33°	-0.0017	-0.0016	-0.234	-0.241
40.00°	-0.0118	-0.0120	-0.307	-0.309

Location	Transverse Moments at Central Section		Twisting Moments at Support	
	SAP2000 6x6 Mesh	Theoretical	SAP2000 6x6 Mesh	Theoretical
0°	-2099	-2090	-91	0
6.67°	-1978	-2000	-359	-380
13.33°	-1614	-1620	-698	-670
20.00°	-1045	-1000	-982	-1000
26.67°	-408	-430	-1183	-1240
33.33°	37	100	-1248	-1290

Figure 16-2
Comparison of Results

Example 17

Hemispherical Shell — Static Loads

Description

This is a hemispherical shell loaded by point loads at the edge as shown in Figure 17-1. The problem solved is identical to the one suggested in Reference [1].

Significant Options of SAP2000 Activated

- Three-dimensional shell analysis

Input Data

The computer model used is shown in Figure 17-1. An 8 x 8 mesh of Shell elements is used on a quarter of the hemispherical shell. Symmetry is used to define the boundary conditions. A single restraint in the vertical direction is applied at the center of the free edge to provide stability.

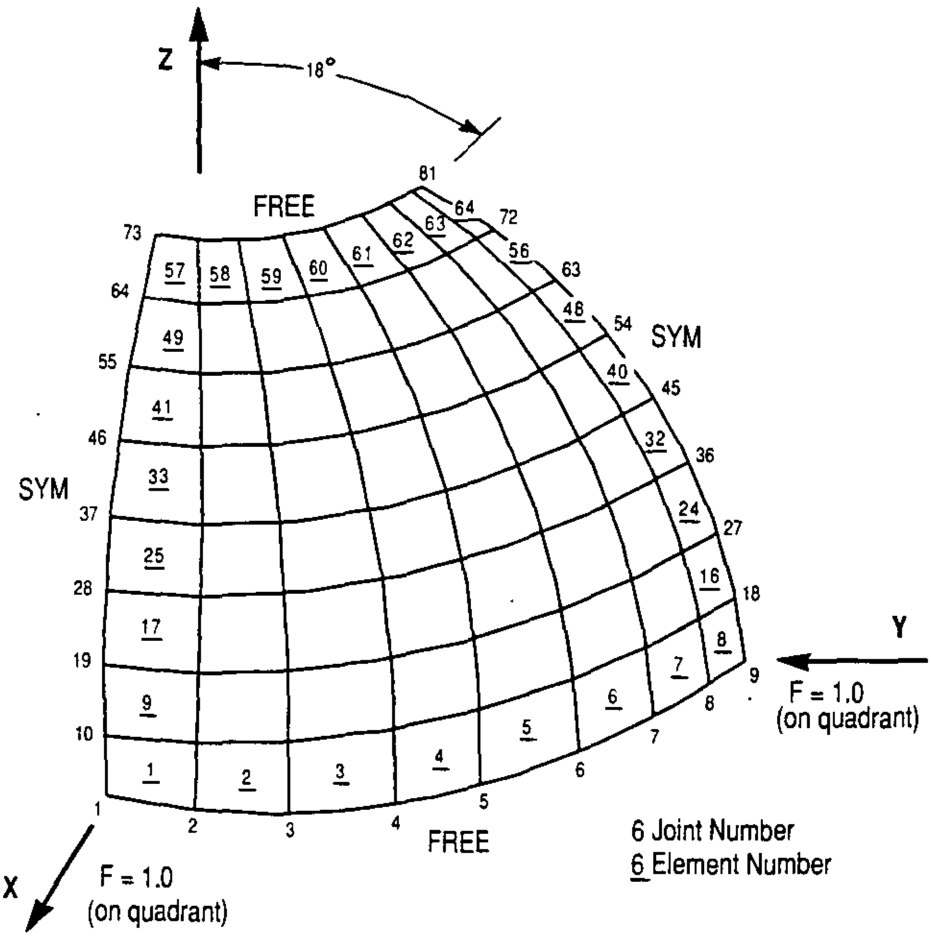
The input data file for this example is SHELL.

Comparison of Results

The theoretical lower bound for the displacement under the load in the direction of the load is 0.0924, where the hole at the center is not present. Reference [1] suggests a value of 0.094 for comparison of results. The SAP2000 solution gives a value of the displacement under the load of 0.0937. The comparison is excellent.

Reference

1. **MacNeal, R. H. and Harder, R. C.**
A Proposed Standard Set of Problems to Test Finite Element Accuracy, Finite Elements in Analysis and Design 1 (1985), pp. 3-20, North-Holland.



RADIUS = 10.0
THICKNESS = 0.4
MODULUS OF ELASTICITY = 6.825×10^7
POISSON'S RATIO = 0.3

LOADING: CONCENTRATED FORCES AS SHOWN

Figure 17-1
Hemispherical Shell Example

Example 18

Portal with P-delta

Description

These are two two-dimensional, one story, one bay, portal frames fixed at the base as shown in Figure 18-1. One frame carries a central load of 1000 pounds and the other carries an eccentric load of the same magnitude. The critical buckling load for the frame in the symmetrical, non-sway mode is 6082 pounds ($P_{cr} = \frac{2.55\pi^2 EI}{h^2}$) and in the asymmetrical, sway mode is 1784 pounds ($P_{cr} = \frac{.748\pi^2 EI}{h^2}$). The purpose of this example is to verify the adequacy of the P-Delta algorithm in SAP2000 for these type of problems. Theoretical results for these problems can be derived using the stability function approach. Tables for stability functions can be obtained from References [1].

Significant Options of SAP2000 Activated

- P-Delta analysis of frameworks

Input Data

The two frames are modeled together in SAP2000 using Frame elements. The loading is applied as span loads to the beams of the structures. Pound-inch units are used.

The theoretical results are computed assuming no axial deformation and no shear deformation in the members. To duplicate this behavior a large axial area is specified for the elements and the shear areas are defaulted as zero (program interprets this to mean that the shear deformation is to be ignored.) It is important to note that the axial deformations of the members were not forced to be zero by restraining or constraining the joints. Setting the axial deformation to be zero in this manner also sets the axial load in the member to be zero..This then completely eliminates the P-Delta effect from that member.

For the P-Delta analysis the default value of the relative displacement tolerance (.001) is used and the maximum number of iterations is set at 10 even though the solution converges at a much lower number of iterations. All loads are in a single Load case and this is used as the P-Delta load combination.

The input data for this example is PORTALPD.

Comparison of Results

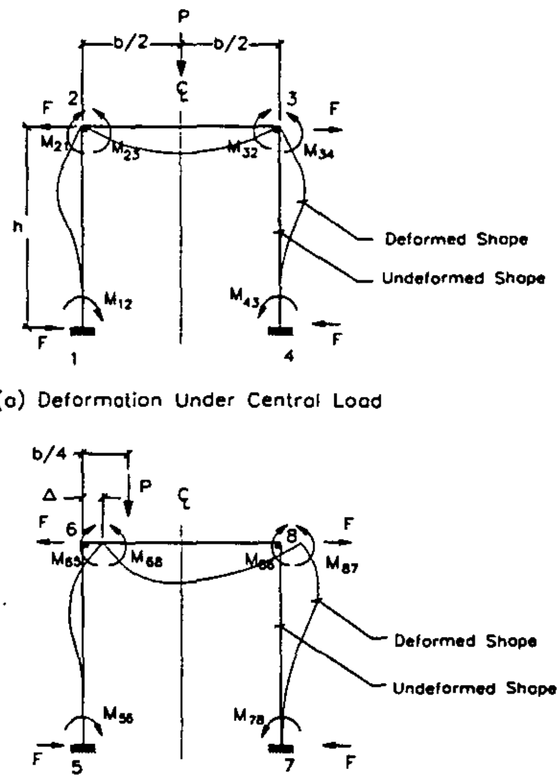
The lateral displacement, joint rotation and member end-moment results for the two load configurations are compared in Figure 18-2 with theoretical results and with results for a model in which the P-Delta effect was ignored. The agreement between the SAP2000 and theoretical results is excellent.

Theoretical values are obtained using the slope-deflection method of analysis by including the effect of axial loads in members. In this case the stiffness and carry-over factor of each member, instead of being constant, are functions of the axial load in the member. These functions are tabulated in Reference [1].

It is interesting to study the effect of P-Delta for these frames. In the symmetrical frame (central load), the decrease in the stiffness of the columns due to compressive axial loads causes a decrease in the beam end moments (top column moments) but increases the bottom column moments. For the asymmetrical frame (eccentric load), the same effect is seen in the column that carries the larger axial load. However, the column with the lesser axial load is now comparatively stiffer and attracts a larger moment at each end when P-Delta effects are considered.

Reference

1. **Livesley, R. K., and Chandler, D. B.**
Stability Functions for Structural Frameworks, Manchester University Press, UK, 1956.



(b) Deformation Under Eccentric Load

TYPICAL PROPERTIES :

SPAN, b	= 100 in
HEIGHT, h	= 100 in
AREA, A	= 1 in^2
MOMENT of INERTIA, I_{22}, I_{33}	= 1/12 in^4
MODULUS of ELASTICITY, E	= 29×10^6 psi

Figure 18-1
Portal Frame Example

QUANTITY	With P-Delta		Without P-Delta
	SAP2000	Theoretical	SAP2000 & Theoretical
Rotation Joint 2 (θ_2)	0.09178	0.09192	0.08620
Moment Joint 1 (M_{12})	4589.1	4606.6	4166.7
Moment Joint 2 (M_{21})	8260.4	8254.0	8333.3
Shear Force (F)	128.5	128.6	125.0

Frame Under Central Load

QUANTITY	With P-Delta		Without P-Delta
	SAP2000	Theoretical	SAP2000 & Theoretical
Lateral Displacement (Δ)	1.894	1.893	1.385
Rotation Joint 6 (θ_6)	0.1014	0.1013	0.0924
Rotation Joint 8 (θ_8)	0.0367	0.0367	0.0369
Moment Joint 5 (M_{56})	2550.9	2544.9	2455.4
Moment Joint 6 (M_{65})	6183.6	6088.6	6919.6
Moment Joint 7 (M_{78})	4503.5	4456.9	3794.6
Moment Joint 8 (M_{87})	6124.9	6153.0	5580.4
Shear Force (F)	101.6	101.4	93.75

Frame Under Eccentric Load

Figure 18-2
Comparison of Results for a Planar Rigid Portal Frame

Example 19

Pounding of Two Planar Frames — Nonlinear Time-History Analysis

Description

A two-bay; seven-story plane frame is linked to a one bay four story plane frame using Nllink Gap elements. The structure experiences pounding due to ground motion. El Centro 1940 (N-S) record is used in the nonlinear time history analysis.

The geometry of the structure is shown in Figure 19-1.

Significant Options of SAP2000 Activated

- Two-dimensional frame analysis
- Use of uniaxial Nllink gap elements
- Nonlinear time-history analysis

Input Data

A Diaphragm constraint is applied to each floor of the two frames. Kip-inch units are used.

The joints at column line 3 are connected to the corresponding joints at column line 4 by Nllink Gap elements. The local 1 axes of these elements are in the global X direction, and gap properties are specified for the local 1 direction. The opening displacement for the gaps is 0.25 inches, corresponding to the distance between the outer faces of the buildings.

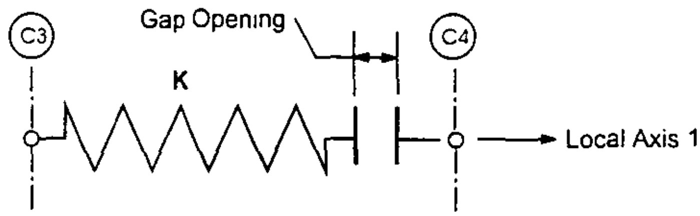
The nonlinear gap stiffness was set to 1000 k/in, an estimate of the stiffness of a tributary region around the point of contact. The linear effective stiffness, used for calculating the modes, was set to zero, since the gap elements are normally open. Care should be taken not to use values of nonlinear stiffness or linear effective stiffness that are too large relative to the stiffness of connecting elements. Overly large values of stiffness can cause numerical sensitivity, which can reduce the accuracy and the efficiency of the solution.

The modes were calculated using Ritz vectors, with the ground acceleration and the nonlinear deformation loads used as starting vectors. All eleven modes of the structure were requested.

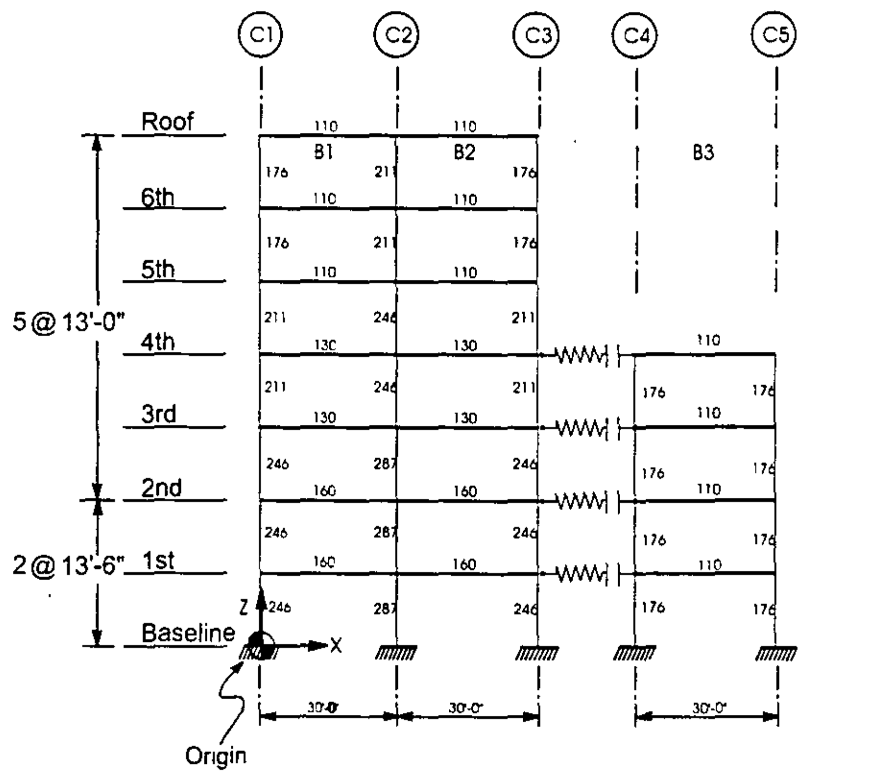
The input data for this example is POUND. The time history of the ground acceleration is given in file ELCN-THU

Comparison of Results

This example is included as a sample only. A typical results plot is shown in Figure 19-2. It shows the variations of displacement of Column lines 3 and 4 and the link force at Story level 4. It can be clearly seen that the link force is generated whenever the two column lines move in phase and their separation is less than the specified initial opening or if they move towards each other out of phase. For most part the pounding has the effect of keeping the buildings in phase. For display purposes the displacements are scaled up by a factor of 100.



Nllink Gap Element



- All columns are W14's
- All Beams are W24's

Elevation

Figure 19-11
Geometry of Two Pounding Frames

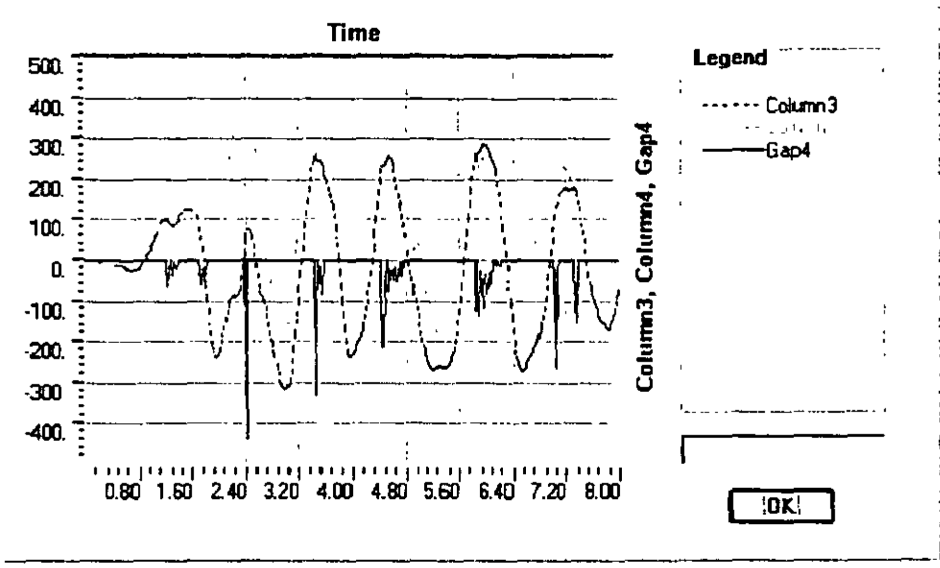


Figure 19-2
Displacements of Pounding Columns and Pounding Force at Story 4

Example 20

Friction-Pendulum Base-Isolated 3D Frame — Nonlinear Time-History Analysis

Description

This is a two-story, three-dimensional frame with base isolation using friction-pendulum base isolators. The structure is subjected to earthquake motion in two perpendicular directions using the Loma Prieta acceleration records.

The geometry of the structure is shown in Figure 20-1.

Significant Options of SAP2000 Activated

- Three-dimensional frame analysis
- Use of biaxial Nllink friction-pendulum elements
- Nonlinear quasistatic time-history analysis
- Nonlinear time-history analysis

Computer Model

The structure is modeled as a reinforced concrete frame with 9 column lines and 12 bays. Kip-inch units are used. The modulus of elasticity is taken to be 3000 ksi. The self-weight of the concrete is taken as 150 pcf.

The floor slab is taken to be 8" thick covering all the specified floor bays at the base and the 1st story levels. At the second story level the corner column as well as the two edge beams are eliminated together with the floor slab to render this particular level unsymmetric as depicted in Figure 20-1. A diaphragm constraint is applied to each story. The floor slab is modeled with Shell Membrane elements. These are used for the purpose of generating mass only; their stiffness is ignored due to the diaphragm Constraints.

Friction-pendulum type base isolators of the type described in Reference [1] are modeled using the SAP2000 Nllink Isolator2 elements.

The isolator properties are defined as follows:

Stiffness in direction 1 (vertical)	1E3
Stiffness in directions 2 and 3 (horizontal)	1E2
Coefficient of friction at fast speed	.04
Coefficient of friction at slow speed	.03
Parameter determining the variation of the coefficient of friction with velocity	20
Radius of contact surface in directions 2 and 3	60

A single Load case is defined that models the self-weight of the structure. This will be applied before the seismic analysis in order to generate the proper frictional response in the isolators. This self-weight must be applied as part of the time-history analysis, as described below.

Seventeen Ritz vectors are requested for the modal analysis, using the two horizontal ground accelerations, the gravity load, and all the nonlinear deformation loads as the starting load vectors. There are 27 nonlinear deformation loads, one for each of the three translational degrees of freedom for the nine isolators. However, only three of the 18 horizontal degrees of freedom are independent due to the diaphragm constraint. The program will automatically discard 15 dependent starting load vectors. Thus the total number of independent starting load vectors is 15. By listing the two ground accelerations as first, two Ritz vectors are generated for these two starting load vectors, and one for each of the remaining 13, for a total of 17 modes.

It is very important that the nonlinear deformation loads be used as starting load vectors in order to capture nonlinear behavior. In this case, the axial load in each isolator must be adequately modeled since the shear force depends strongly upon it.

Two time-history analyses are performed. The first history applies the self-weight quasistatically. This history consists of 10 time steps of one second, for a total time of 10 seconds. The load is increased linearly to full value for the first five seconds, and allowed to come to equilibrium for the remaining five seconds. A damping value of 99% is used for all modes to prevent vibration.

The second history starts from the first history, and applies the seismic acceleration. This history consists of 2000 time steps of 0.02 seconds, for a total time of 40 seconds. Zero damping is used for all modes since most energy dissipation is expected to be due to the friction properties of the isolators, with little damage occuring in the superstructure.

The input data file for this example is ISOLAT2 and the input time history files are LP-TH0 and LP-TH90.

Comparison of Results

This example is included as a sample only. No results are available for comparison. Typical plots are shown for the seismic time history. Figure 20-2 shows the time variation of input, kinetic, potential (strain), damping, and frictional energies; these energies do not start at zero since they continue from the self-weight time history. Figure 20-3 shows the time variation of the displacements of the second floor at Column line 1. Figure 20-4 shows the interaction diagram for the two shear forces in the isolator beneath Column line 1. Figure 20-5 shows the force-displacement plot for motion in the X direction of the isolator beneath Column line 1.

References

1. **Zayas, V. and Low, S.**
"A Simple Pendulum Technique for Achieving Seismic Isolation," *Earthquake Spectra*, Vol. 6, No. 2, Earthquake Engineering Research Institute, Oakland, California,1990.

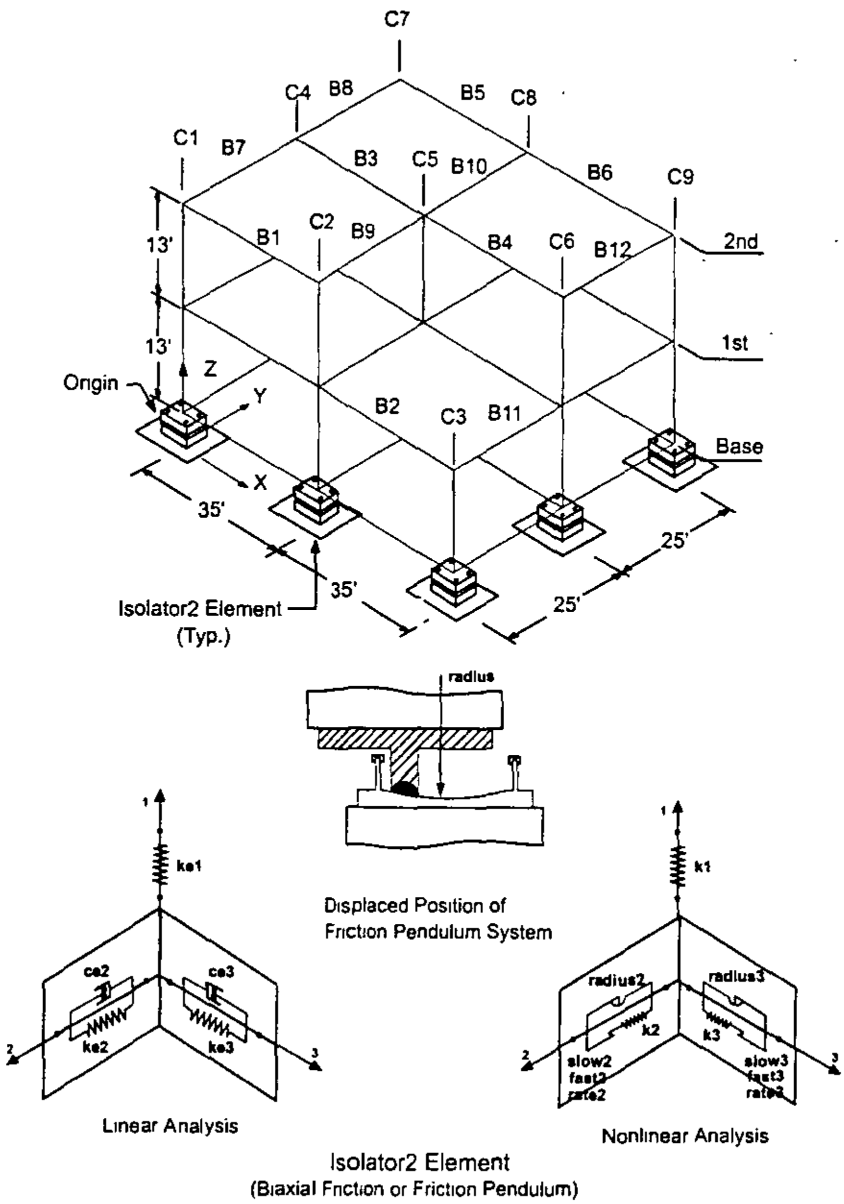


Figure 20-1
Model of Friction-Pendulum Base-Isolated 3-D Frame

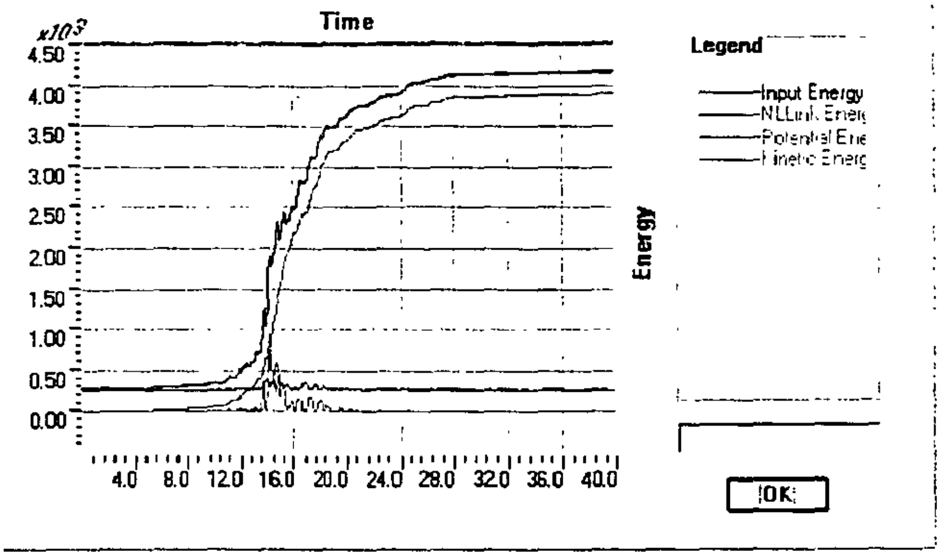


Figure 20-2
Variation of Energy during Seismic History

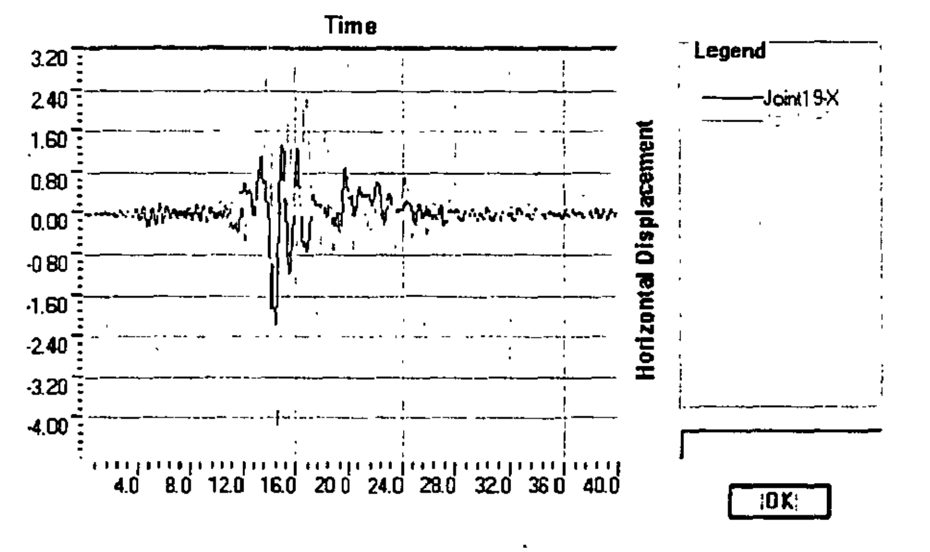


Figure 20-3
Horizontal Displacements at Top of Column 1 during Seismic History

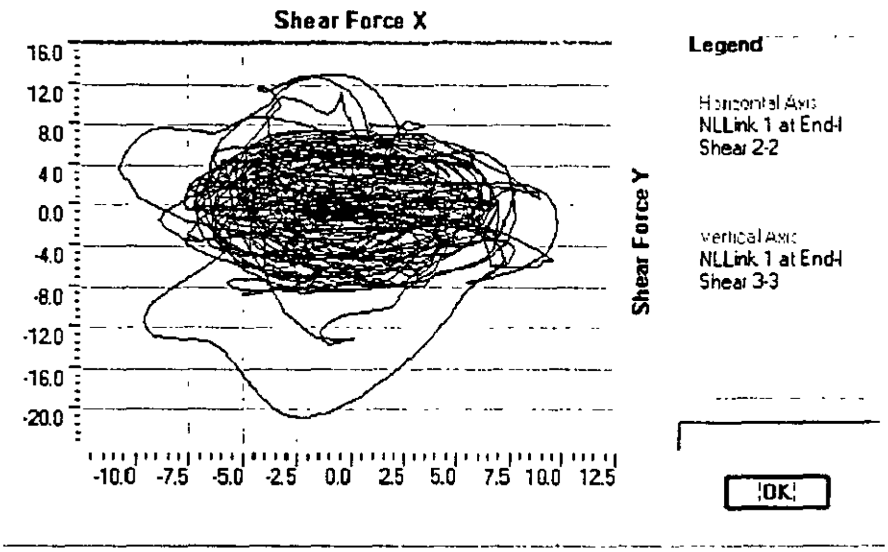


Figure 20-4
Shear Force Interaction in Isolator under Column 1 during Seismic History

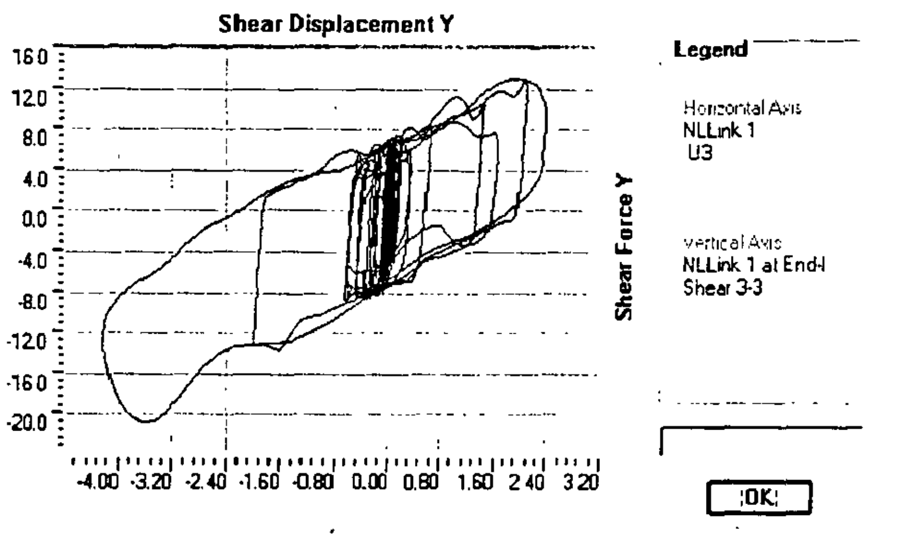


Figure 20-5
Shear Force vs. Deformation in Isolator under Column 1 during Seismic History

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 INTEGRATED FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

CONCRETE DESGIN MANUAL

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,

SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

CONCRETE DESIGN MANUAL

COMPUTERS & STRUCTURES INC

Computers and Structures, Inc.
Berkeley, California, USA

Version 6.1
October 1997

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Tel: (510) 845-2177
Fax: (510) 845-4096
E-mail: *info@csiberkeley.com*
Web: *www.csiberkeley.com*

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THIS PROGRAM IS A VERY PRACTICAL TOOL FOR THE DESIGN OF REINFORCED CONCRETE STRUCTURES. HOWEVER, THE USER MUST THOROUGHLY READ THE MANUAL AND CLEARLY RECOGNIZE THE ASPECTS OF REINFORCED CONCRETE DESIGN THAT THE PROGRAM ALGORITHMS DO NOT ADDRESS.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table of Contents

CHAPTER I **Introduction** 1

Overview 1
Organization 2
Recommended Reading 3

CHAPTER II **Quick Tutorial** 5

Overview 5
Description of the Model 6
Starting the Tutorial 8
Opening the Model Database File 9
Analyzing the Model 10
Selecting the Design Code 12
Starting Design 12
Changing Member Properties 16
Concluding Remarks 21

CHAPTER III **Design Algorithms** 23

Design Load Combinations 24
Design and Check Stations 25
Identifying Beams and Columns 25
Design of Beams 26
Design of Columns 26
P-Δ Effects 30
Element Unsupported Lengths 30
Special Considerations for Seismic Loads 32
Choice of Input Units 32

CHAPTER IV Design for ACI 318-95 33

Design Load Combinations 33
Strength Reduction Factors 36
Column Design 37
Generation of Biaxial Interaction Surfaces 37
Check Column Capacity 39
Determine Factored Moments and Forces 39
Determine Moment Magnification Factors 39
Determine Capacity Ratio 41
Design Column Shear Reinforcement 42
Determine Section Forces 43
Determine Concrete Shear Capacity 44
Determine Required Shear Reinforcement 45
Beam Design 46
Design Beam Flexural Reinforcement 46
Determine Factored Moments 47
Determine Required Flexural Reinforcement 47
Design Beam Shear Reinforcement 53
Determine Shear Force and Moment 55
Determine Concrete Shear Capacity 56
Determine Required Shear Reinforcement 56

CHAPTER V Design for CAN3-A23.3-M84 59

Design Load Combinations 62
Strength Reduction Factors 62
Column Design 63
Generation of Biaxial Interaction Surfaces 63
Check Column Capacity 65
Determine Factored Moments and Forces 65
Determine Moment Magnification Factors 65
Determine Capacity Ratio 67
Design Column Shear Reinforcement 68
Determine Section Forces 68
Determine Concrete Shear Capacity 70
Determine Required Shear Reinforcement 70
Beam Design 72
Design Beam Flexural Reinforcement 72
Determine Factored Moments 72
Determine Required Flexural Reinforcement 73
Design Beam Shear Reinforcement 79
Determine Shear Force and Moment 79
Determine Concrete Shear Capacity 82
Determine Required Shear Reinforcement 82

CHAPTER VI Design for BS 8110-85 83

Design Load Combinations 83
Design Strength 86
Column Design 87
Generation of Biaxial Interaction Surfaces 87
Check Column Capacity 88
Determine Factored Moments and Forces 89
Determine Additional Moments 89
Determine Capacity Ratio 91
Design Column Shear Reinforcement 92
Beam Design 94
Design Beam Flexural Reinforcement 94
Determine Factored Moments 94
Determine Required Flexural Reinforcement 95
Design Beam Shear Reinforcement 100

CHAPTER VII Design for Eurocode 2 103

Design Load Combinations 103
Design Strength 106
Column Design 107
Generation of Biaxial Interaction Surfaces 107
Check Column Capacity 109
Determine Factored Moments and Forces 109
Determine Code Total Moments 109
Determine Capacity Ratio 111
Design Column Shear Reinforcement 112
Beam Design 114
Design Beam Flexural Reinforcement 115
Determine Factored Moments 115
Determine Required Flexural Reinforcement 115
Design Beam Shear Reinforcement 121

CHAPTER VIII Design Output 123

Overview 123
Graphical Display of Design Output 124
Tabular Display of Design Output 125
Member Specific Information 127

References 131

Index 133

Chapter I

Introduction

Overview

SAP2000 features powerful and completely integrated modules for design of both steel and reinforced concrete structures (CSI 1997a, 1997b). The program provides the user with options to create, modify, analyze and design structural models, all from within the same user interface.

The program provides an interactive environment in which the user can study the stress conditions, make appropriate changes, such as member size revisions, and update the design without re-analyzing the structure. A single mouse click on an element brings up detailed design information. Members can be grouped together for design purposes. The output in both graphical and tabulated formats can be readily displayed and printed.

The program is structured to support a wide variety of design codes for the automated design and check of concrete frame members. The program currently supports the following design codes: U.S. (ACI 1995), Canadian (CSA 1984), British (BSI 1985), and European (CEN 1992).

The design is based upon a set of user-specified loading combinations. However, the program provides a set of default load combinations for each design code supported in SAP2000. If the default load combinations are acceptable, no definition of additional load combinations are required.

In the design of the columns, the program calculates the required longitudinal and shear reinforcement. However the user may specify the longitudinal steel, in which case a column capacity ratio is reported. The column capacity ratio gives an indication of the stress condition with respect to the capacity of the column.

Every beam member is designed for flexure and shear at a user defined number of stations along the beam span.

The presentation of the output is clear and concise. The information is in a form that allows the engineer to take appropriate remedial measures in the event of member overstress. Backup design information produced by the program is also provided for convenient verification of the results.

English as well as SI and MKS metric units can be used to define the model geometry and to specify design parameters.

Organization

This manual is organized in the following way:

Chapter II provides a quick tutorial aiming at giving the first time users hands-on experience. Several of the basic features of the SAP2000 concrete design modules are explored in this tutorial.

Chapter III outlines various aspects of the concrete design procedures of the SAP2000 program. This chapter describes the common terminology of concrete design as implemented in SAP2000.

Each of four subsequent chapters gives a detailed description of a specific code of practice as interpreted by and implemented in SAP2000. Each chapter describes the design loading combination, column and beam design procedures, and other special consideration required by the code.

Chapter IV gives a detailed description of the ACI code (ACI 1995) as implemented in SAP2000.

Chapter V gives a detailed description of the Canadian code (CSA 1984) as implemented in SAP2000.

Chapter VI gives a detailed description of the British code (BSI 1985) as implemented in SAP2000.

Chapter VII gives a detailed description of the Eurocode (CEN 1992) as implemented in SAP2000.

Chapter VIII outlines various aspects of the tabular and graphical output from SAP2000 related to concrete design.

Recommended Reading

It is recommended that first time users follow through the steps of the "Quick Tutorial" in Chapter II and read Chapter III "Design Algorithms" and one of four subsequent chapters corresponding to the code of interest to the user. Finally the user should read "Design Output" in Chapter VIII for understanding and interpreting SAP2000 output related to concrete design.

Chapter II

Quick Tutorial

Overview

Several of the basic features of the SAP2000 concrete design modules are explored in this tutorial. This introduction is aimed at giving the first time user hands-on experience for designing concrete frames with SAP2000. The program allows you to select from several U.S. and international codes to design and review concrete structures. A comprehensive on-line Help is included in the program for your quick reference. It is assumed that you have a working knowledge of concrete design procedures and are reasonably familiar with the current codes of practice and their underlying design concepts.

We will access the SAP2000 commands from both the Toolbar and from the menus. The Toolbar, however, provides quick access to most commonly used features available from the menus.

In the assignment sequence, there are two important points you must remember. First, you have to define an entity before you can assign an attribute to it, and second, you have to select member(s) before you can assign new attributes or modify old ones.

Description of the Model

The structure is a two-story, two-by-two bay office building located in Seismic Zone No. 4 (high seismic area). It is designed as a special moment resisting concrete frame using the ACI 318-95 code.

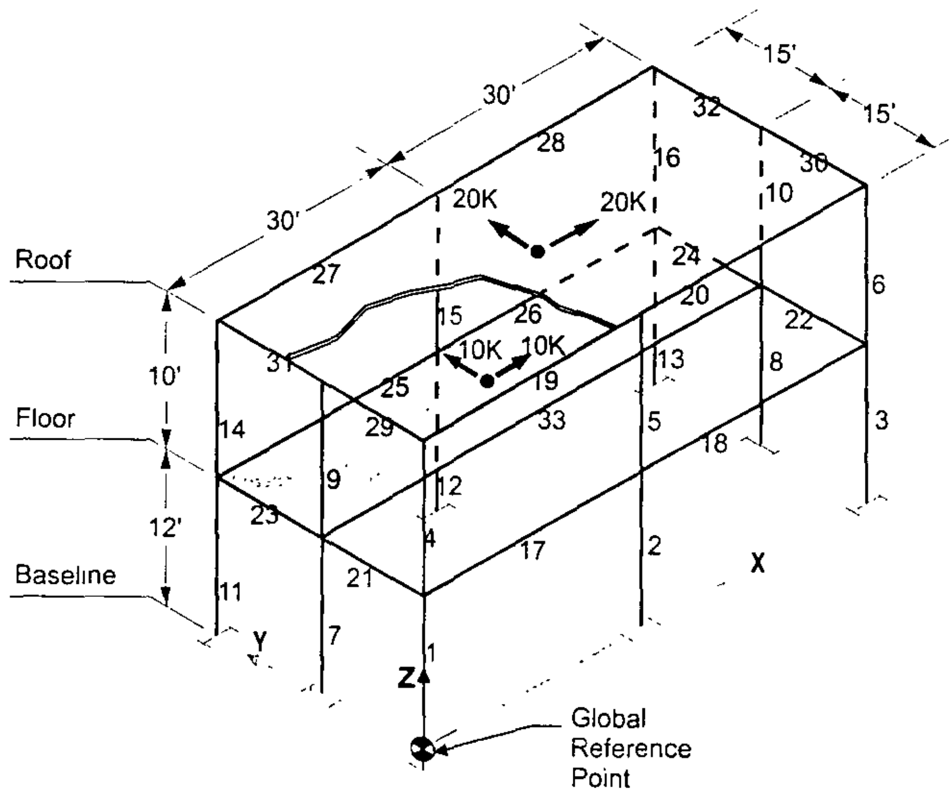


Figure II-1
Ductile Moment Resisting Concrete Frame (Tutorial Example)

Geometry

The two-story structure has a partial floor diaphragm and a full roof diaphragm. See Figure II-1. The story height of the top and bottom floor is taken as 10'0" and 12'0" respectively. The initial member sizes and reinforcement are given in Table II-1.

ID	Structural Component	Description
1	Typical columns at the top story	12" ×12", Rebar not specified, 2" cover to center of steel
2	Typical columns at the bottom story	18" ×18", Rebar not specified, 2" cover to center of steel
3	All other beams	12" ×24", Rebar not specified, 2" cover to center of steel
4	The longest span beam (Beam 33)	12" ×36", Rebar not specified, 2" cover to center of steel

Table II-1
Structural Property Data (Tutorial Example)

Material Properties

The properties of the materials used in the model are given in Table II-2. It is assumed that the materials used for the beams and columns are the same. However, the shear reinforcement is different from the longitudinal reinforcement.

Material Property	Magnitude
f_c	4.0 ksi
E_c	3600 ksi
f_y	60 ksi
f_{ys}	40 ksi

Table II-2
Material Specifications (Tutorial Example)

For analysis in SAP2000, the value of E_c is modified to account for cracking. A multiplier of 0.4 (as recommended in ACI 318-89) is used for columns assumed to have about 2% steel, and a multiplier of 0.5 (as recommended in ACI 318-89) is used for the beams. These multiplication factors are slightly different in ACI 318-95. See Section R10.11.1 of ACI 318-95.

Load Cases

Four load cases are considered in the analysis. The dead and live loads are defined as load cases DL and LL respectively. The lateral seismic loads, in turn, are designated as QX and QY respectively.

The dead and live loads are simplified as line loads on the beams. The equivalent static seismic forces are applied as lateral loads at the centroids of the diaphragms:

Load case 1 : DL — 1.0 kip/ft on all beams which are connected with the diaphragm along the X-direction (Self-weight included)

Load case 2 : LL — 0.5 kip/ft on all beams which are connected with the diaphragm along the X-direction

Load case 3 : QX — Static equivalent earthquake force in the X-direction

Load case 4 : QY — Static equivalent earthquake force in the Y-direction

Analysis

Two diaphragm constraints are applied for the two diaphragms at the two floors. These constraints prevent in-plane relative displacements of the nodes at each floor. The lateral earthquake loads are assumed to be applied at the centroid of the diaphragm. A P-Δ analysis is carried out with a load level of 0.75 (1.4 DL + 1.7 LL)/φ as recommended in the chapter "Design for ACI 318-95" on page 40, where φ is taken as 0.75.

Design

The design is performed in accordance with ACI 318-95. Kip-in units are used. The input database file for this model is **EXCONC.SDB**. This is supplied as part of the SAP2000 package.

Starting the Tutorial

A step-by-step procedure for the design of the model is outlined below. It is recommended that you actually perform these steps while reading this chapter. We assume that you have successfully started the program. You can do this by running SAP2000 from the Start Menu.

In this tutorial, whenever possible, we will use the Toolbars to access various options quickly. Most of the features available on the Toolbars can also be accessed from the Menu Bar. Use the on-line Help or refer to the "SAP2000 Getting Started" manual for a detailed description of the SAP2000 screen.

The input database file for the model (**EXCONC.SDB**) is in the EXAMPLES subdirectory under the main directory where the program has been installed. In this example, the analysis model is already created. This tutorial gives the highlights of the design phase. You are assumed to be familiar with creating and editing structural models using SAP2000

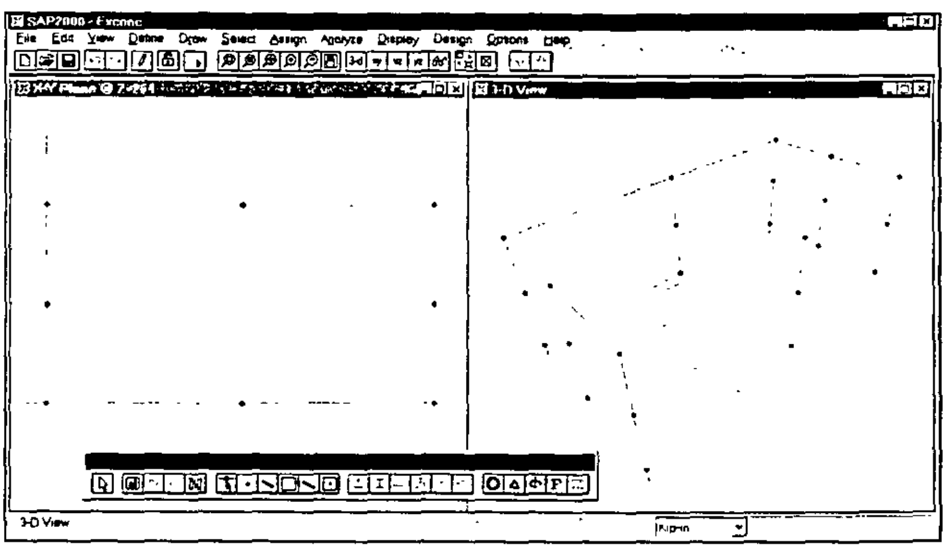
Opening the Model Database File

1. Click on the **Open** button from the **File** menu. This will display the **Open Model File** dialog box.

2. In this dialog box:

   - Select the **EXCONC.SDB** file.
   - Click on the **Open** button.

The screen will now show two vertically-tiled windows. The left window displays a plan view of the model at level + 264 in. Section labels are not displayed in this view. A three-dimensional view of the model is shown in the right window.



Note: when working with multiple windows, clicking anywhere in a particular window will activate that window.

Before we proceed further, we will make a copy of the data file by saving the model under a new name, say, TUTOR1.SDB. We will use the copy during the tutorial and leave the original file unaltered.

3. From the **File** menu, choose **Save As...**. This will display the **Save Model File As** dialog box.

4. In this dialog box:

   - Enter new filename, Tutor1.SDB.

     Note: Even if you do not type in the extension .SDB, the program automatically appends this extension to the filename.

   - Click on the **Save** button.

The new name is displayed in the Main Title Bar.

Analyzing the Model

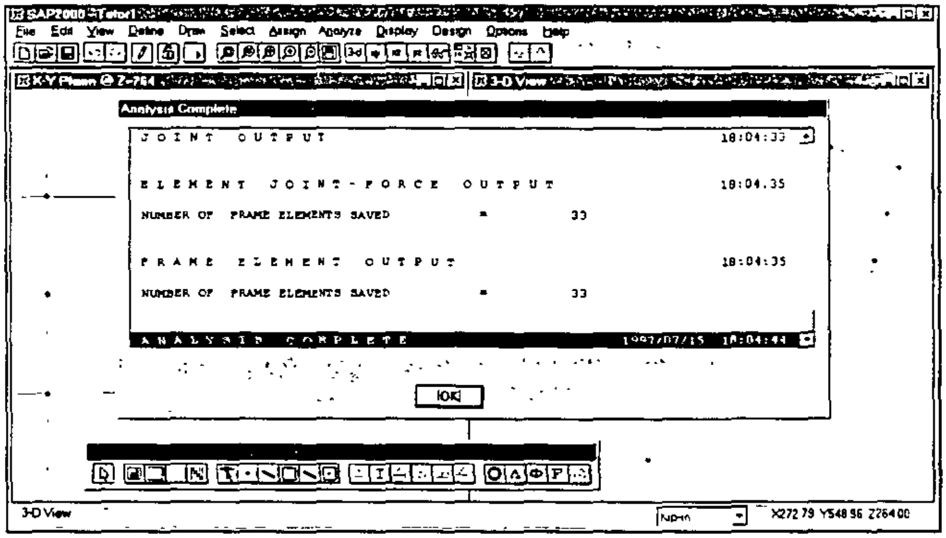
We will now analyze the model. Before analyzing the model we need to set the P-Δ force and other parameters for P-Δ analysis. To do this:

1. Select the **Set Options...** button in the **Analyze** menu. This will immediately bring up the **Analysis Options** dialog box. In this dialog box:

   - Check the **Include P-Delta** check box.
   - Click on the **Set P-Delta Parameters** button This will bring up the **P-Delta Parameters** dialog box. In this dialog box:
     - Set maximum iterations to 5.
     - Change the DL scale factor to 1.4 and click **Modify**.
     - Click on the **Load Case** drop down arrow.
     - Select LL.
     - Change the LL scale factor to 1.7 and click **Add**.
     - Click **OK** to close the **P-Delta Parameters** dialog box
   - Click **OK** to close **Analysis Options** dialog box.

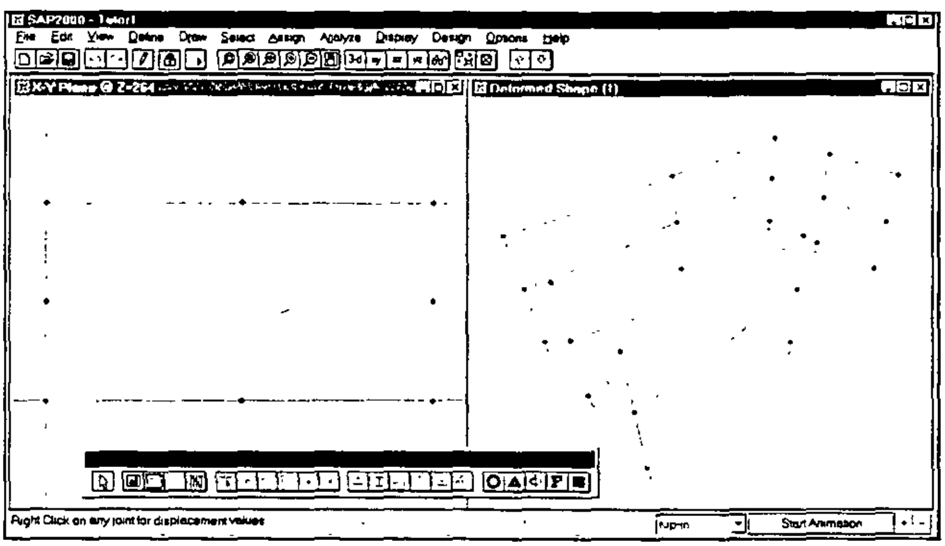
2. Click on the **Run Analysis** button on the main toolbar.

A top window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display the following:



3. Use the scroll bar on the top window to review the analysis messages and to check for any error or warning messages. In our case there should be none.

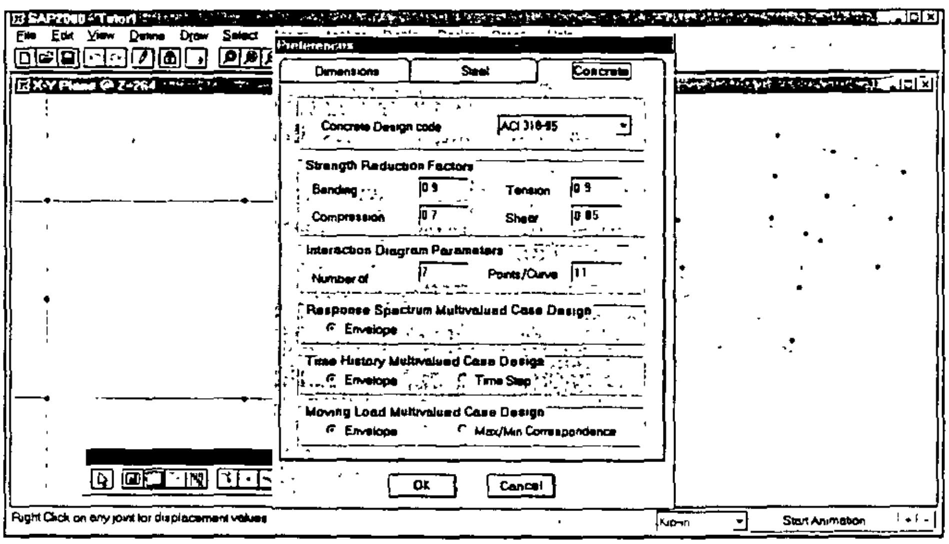
4. Click on the **OK** button in the top window to close it. This will display a deformed shape for the first load case (DL) in the active window (right window in this example) as follow:s



Selecting the Design Code

Selection of a design code is activated from **Preferences...** in the **Options** menu. The default design code is ACI 318-95 for reinforced concrete design. Since the default code is used in this tutorial, we can by-pass this step. To confirm, however, you can follow this:

1. Click on the **Preferences...** button from the **Options** menu. This will launch the **Options** dialog box.
2. Click on the **Concrete** tab.
3. You can see the currently selected concrete design code, strength reduction factors, interaction diagram parameters, and other parameters as shown in the next page. You do not need to change anything.
4. Click on the **Cancel** button to close the dialog box.

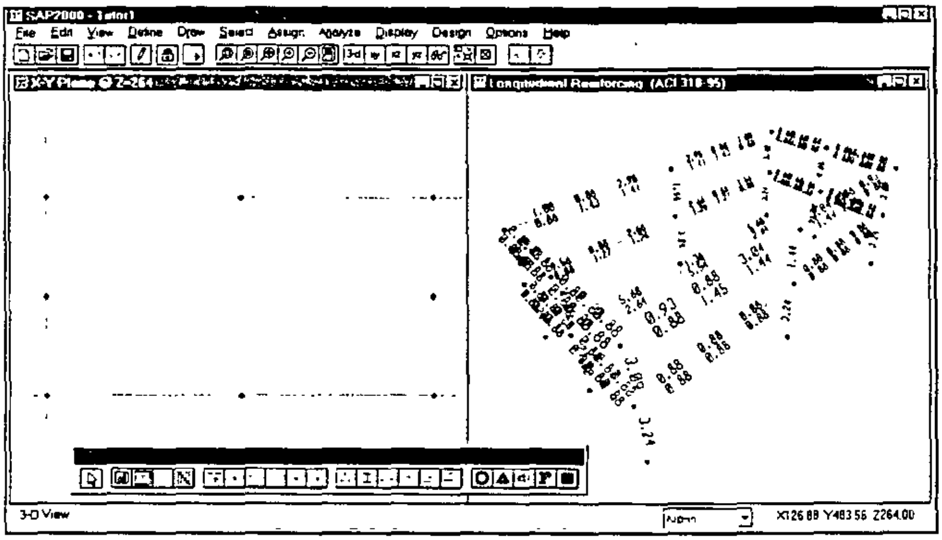


Starting Design

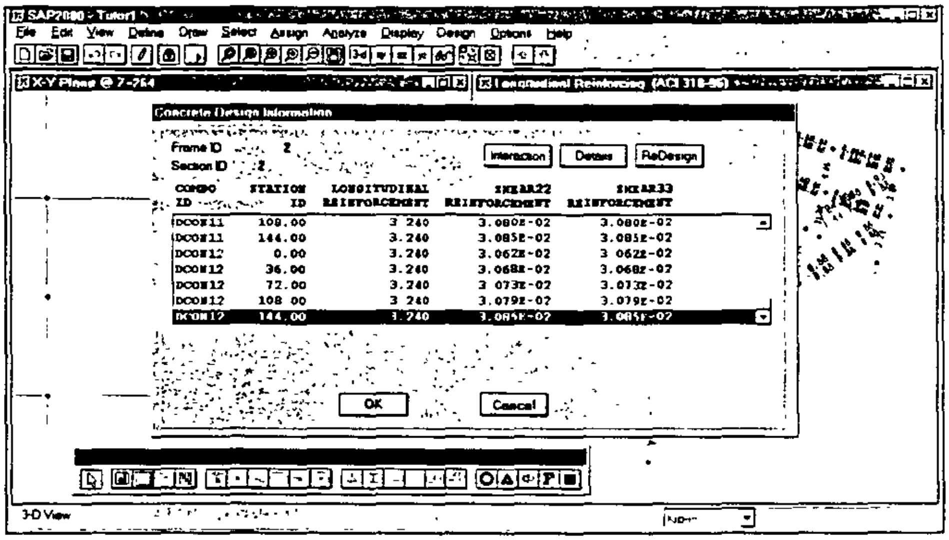
With the analysis phase and selection of the design code completed, we will now design the structure using the requirements of ACI 318-95.

1. From the **Design** menu, choose **Start Design/Check of Structure**. The program now designs each of the concrete frame members. (If we had selected some frame members, then only the selected frames would be designed). In a

few moments the longitudinal reinforcement requirements are displayed in the active window. For beams the compression and the tensile reinforcement are displayed separately. For columns the total overall reinforcement area is displayed. In the display, the reinforcement areas are reported for the governing design combination, by default.



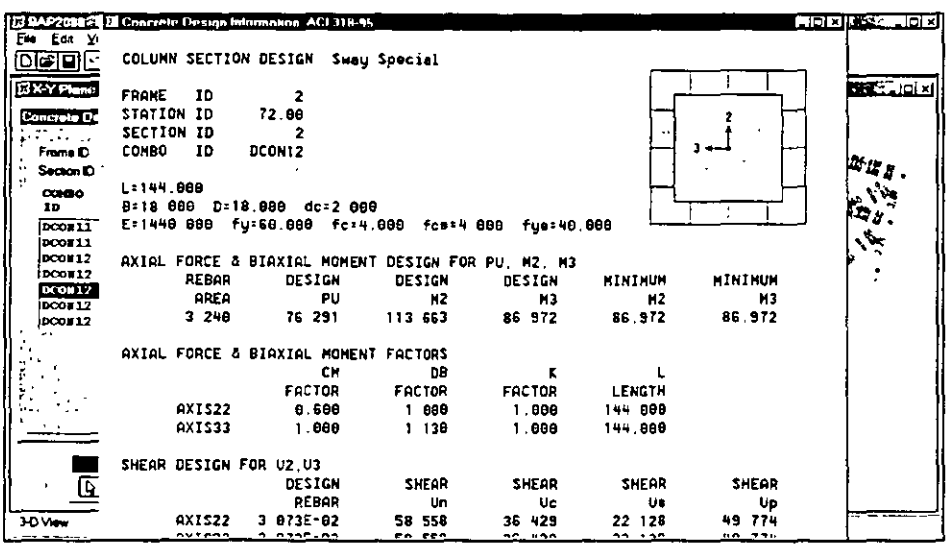
2. Right click on a column member, for example element 2 (see Figure II-1). This will open the **Concrete Design Information** dialog box showing longitudinal and shear reinforcement requirements at various stations along the element length for the various load combinations (see screen below). The dialog box also can show information regarding the **Details** of calculation for design, the element overwrite assignments for **ReDesign** for the selected member, and column **Interaction** properties. However, if the member is a beam, rather than a column, the **Interaction** properties are not relevant and are not available from the **Concrete Design Information** dialog box.



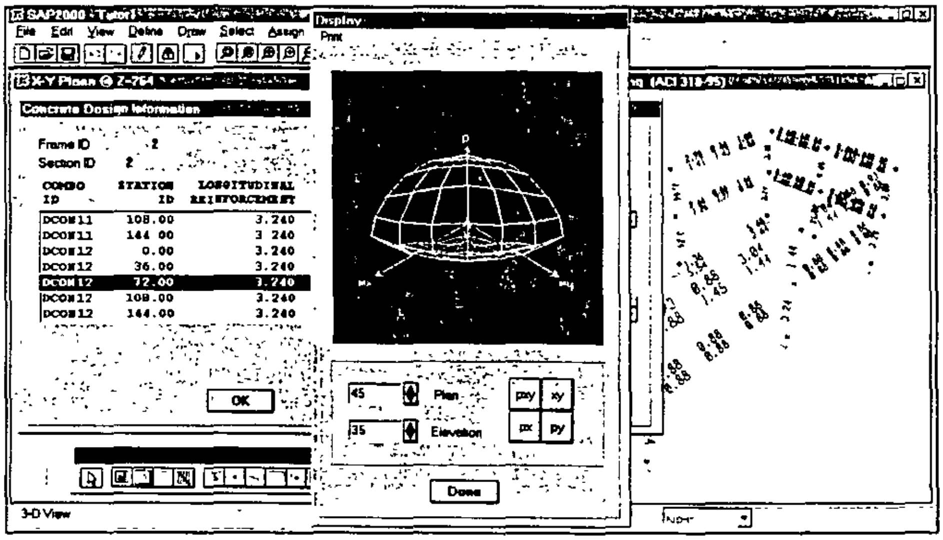
3. In this dialog box:

   - Select a design check station in the **Concrete Design Information** dialog box.

   - Click on the **Details** button. This will open the **Concrete Design Information ACI 318-95** screen showing the design parameters including the reinforcement areas and the factored member forces for the selected load combination at that particular station. See screen below.



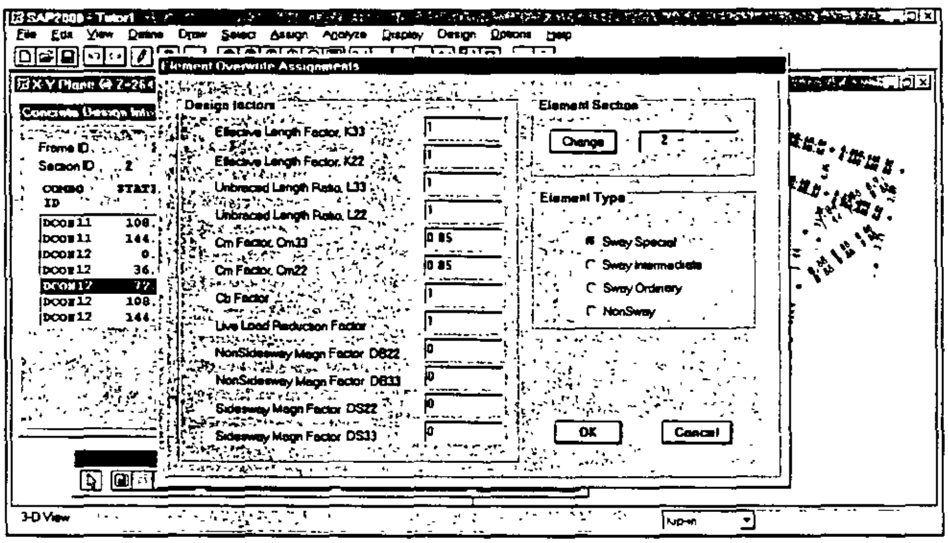
- Close the **Concrete Design Information ACI 318-95** window.
- Click on the **Interaction** button. This will open the dialog box showing the column interaction diagram and the current state of the design forces in the diagram for the selected load combination at that particular station. The interaction diagram can be rotated about any axis to view the diagram from different directions. See screen below.



- Click on the **Done** button to close the **Interaction** information dialog box.
- Click on the **ReDesign** button. This will open the **Element Overwrite Assignments** information dialog box showing the input design factors including the *K* factors. C_m factors, etc. These factors can be edited for redesigning. See screen on the next page. There is also an alternative way of editing the properties of a set of members which will be demonstrated in the next section "Changing Member Properties".
- Click on the **OK** button to close the **Element Overwrite Assignments** dialog box.
- Click on the **OK** button to close the **Concrete Design Information** dialog box.

Note: The number of stations (number of segments + 1) used in the design is set by the user through **Frame** *and* **Output Segment** *buttons from the* **Assign** *menu prior to the analysis phase The default number of segments is 4 for beams and 2 for columns.*

Till now we have analyzed and designed the concrete frame and reviewed some of the design information. SAP2000 allows you to interactively change the design code, member properties, remove or add new load combinations, etc. and re-run the analysis and design phases. As a demonstration in this tutorial, we will edit/change a member property for a set of frame members in the next section.



Changing Member Properties

With the analysis and preliminary design successfully completed, we will now modify the section properties of all the columns of the bottom story before performing re-analysis. Initially, in the analysis, the section type of each column member of the bottom story was taken to be 2. Referring to the screen that follows, we will change the section type of every column at the bottom story to be 1. Note that there are already four previously defined section types in the model which were named numerically as 1, 2, 3, and 4. In order to make these changes, we will change the view in the right window to make all the columns visible for selection. Notice that this window is currently showing the longitudinal reinforcing from the previous design.

1. Click on the **Show Undeformed Shape** button from the floating toolbar.

2. Click on the **2D View (xz)** button from the main toolbar for an elevation view.

3. Click on the **Perspective Toggle** button from the main toolbar. This will display a 3D view. All columns except the middle two will be visible. These two

columns will be overlapping each other. To look at them better, we need to rotate the model about a vertical axis.

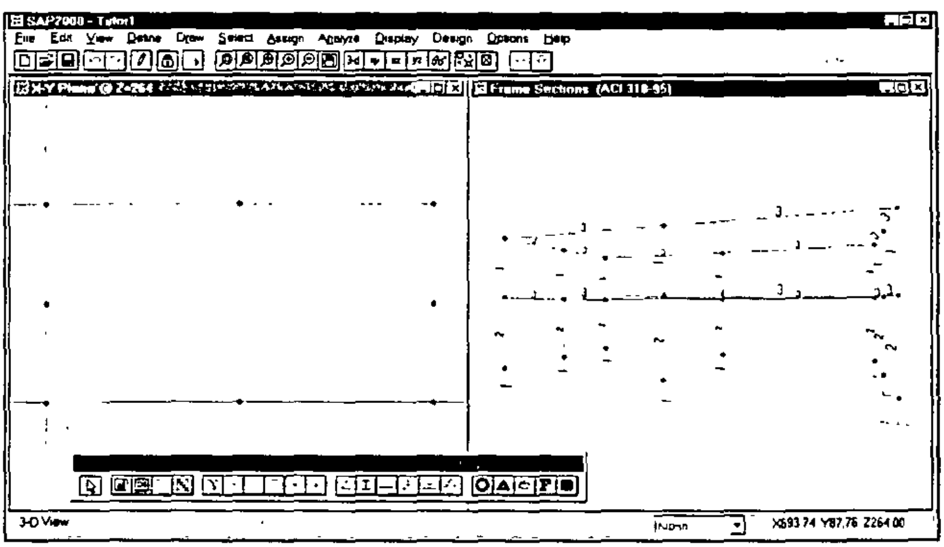
4. Click on the **Set 3D View ...** button on the **View** menu. Increase the **plan View Direction Angle** from 270 to 300 on the **Set 3D View** popup window and then click on the **OK** button.

Now, with all the columns visible, we can select and modify their design section information. Remember, SAP2000 maintains two sets of information for sections. One is for analysis and the other is for design. Changing section type here will affect the design section only. To update the analysis section, you need to explicitly request an update of the analysis information from the current design state using the menu item **Update Analysis Sections** in the **Design** menu.

5. To see the current setting of Design Sections do the following:

   - Click on the **Display Design Info ...** menu item from the **Design** menu. Select the **Design Input** option button.
   - Select **Design Sections** from the drop-down list.
   - Click **OK**.

This will display the design sections on the screen as shown below. Now we can select and modify the sections for the columns at the first story.

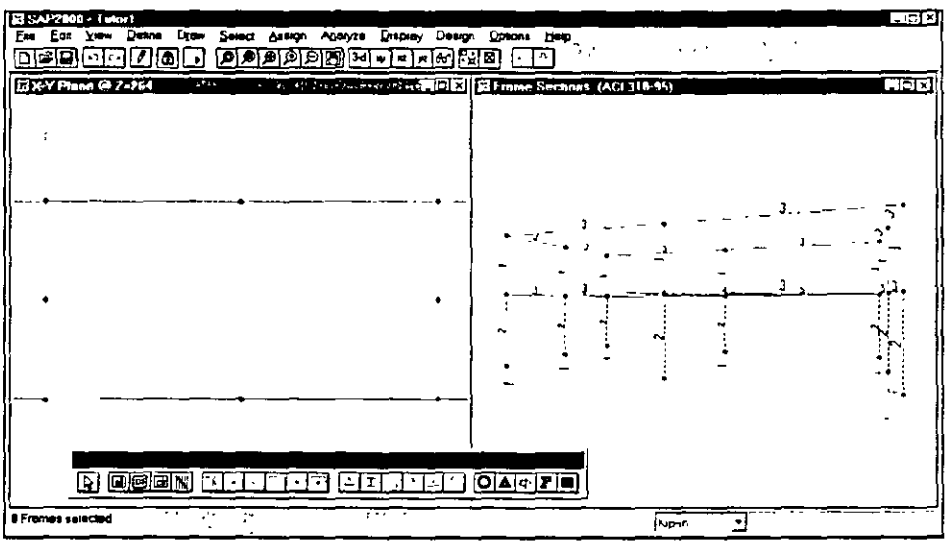


6. To select all the columns at the bottom story do the following:

   - Click the **Set Intersecting Line Select Mode** button on the Floating Toolbar.
   - Move the pointer to the left and middle of the leftmost corner column at the bottom story.
   - Click and hold down the left mouse button.
   - While holding, move the pointer horizontally to the right of the members intersecting all the columns at the bottom story. A "rubber-band line" will show the intersecting line.
   - Release the left mouse button to select all members that intersect the rubber-band line.

     Note: To select all the columns in the bottom story, we have to do this operation only once. Any member can also be selected just by clicking the member itself.

The selection of all the bottom story columns is now complete. The selected members appear as dashed lines.



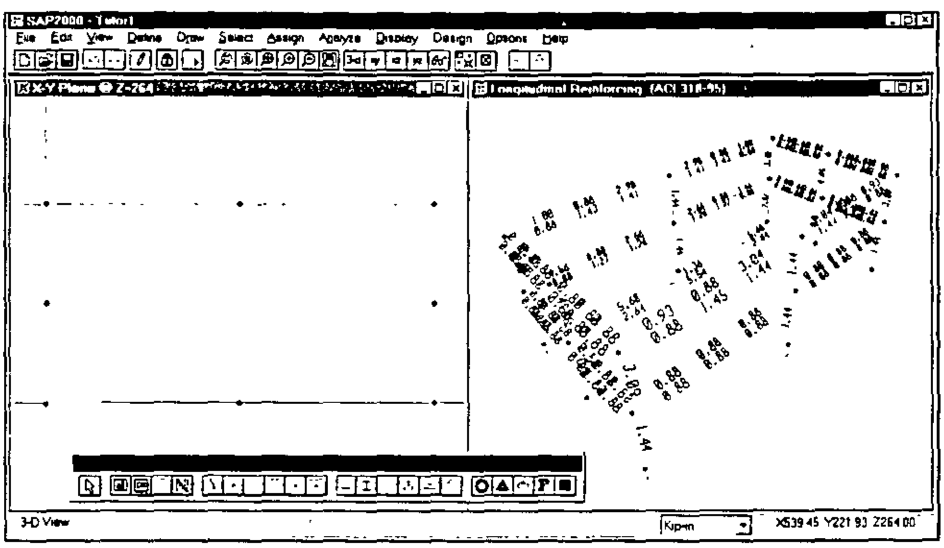
7. From the **Design** menu, choose **ReDefine Element Design Data....** This will display the **Element Overwrite Assignments** dialog box to edit the sections and the design factors. The design factors are code dependent. To change the sections from this dialog box:

- Click the **Change** button on the **Element Section** area. This will display **Select Sections** dialog box. In this dialog box:
  - Select 1 by clicking once.
  - Click on the **OK** button to accept the change.
- Click on the **OK** button on **Element Overwrite Assignment** dialog box.

This will recompute the longitudinal reinforcement based on the new section properties and the previous analysis results.

8. To see the recomputed longitudinal reinforcement, do the following:

   - Click on the **Display Design Info ...** menu item from the **Design** menu. Select the **Design Output** option button.
   - Select **Longitudinal Reinforcing** from the drop-down list.
   - Click **OK**. This will display the longitudinal reinforcement as recomputed based on the new section properties and the previous analysis results.
   - Click on the **3D View (3-d)** button from the main toolbar to display the results in an orientation used earlier.

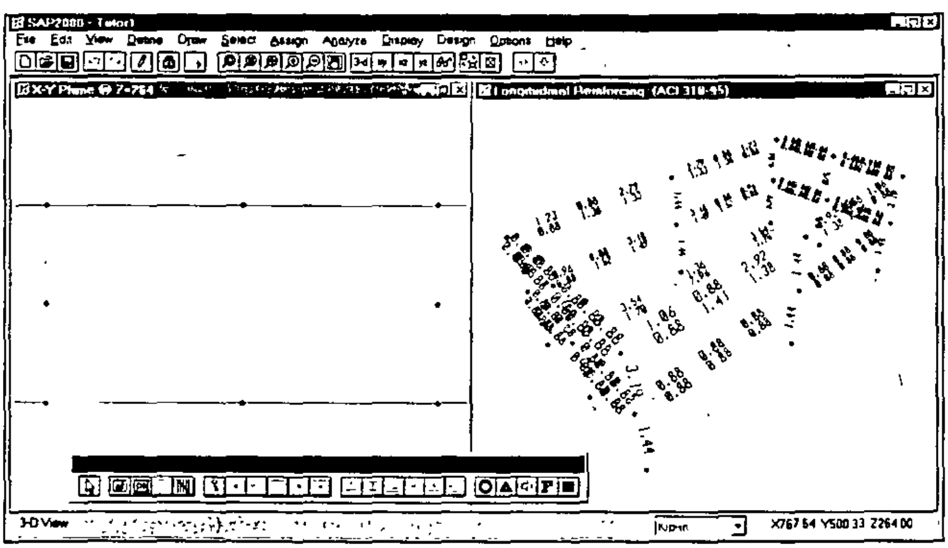


Notice that as a result of changing the section, the reinforcement areas in those particular columns are changed. To see the difference, compare this display with the one on page 13.

It is important to realize that changes made to member section properties in the design phase are **not** automatically reflected in the analysis results. These changes are only local to the post-processing phase unless a re-run of the analysis, with updated elements, is requested by the user. In other words, overwriting the section properties only affects the stress values and not the factored element forces obtained in the analysis preceding such changes. The redistribution of member forces due to change of stiffness (revision of section properties) is effected in a re-run of analysis.

- In the **Design** menu click on **Update Analysis Sections**. This will prompt a dialog box asking **"Updating Analysis Section will unlock model! OK to update?"**. Click **OK**.
- From the **Analyze** menu, choose **Run.** This will immediately start the analysis procedure. A top window is opened in which various phases of analysis are progressively displayed. The results will obviously differ from those produced in the initial analysis because of the change of section properties we made in the design stage. Click **OK** to close the top window.
- Click on the **Start Design/Check of Structures** from the **Design** menu. This will redesign the structure and display the new required longitudinal reinforcement.

You can see the difference after re-running the design based on the latest analysis results.



Concluding Remarks

We have come to the end of this tutorial on the SAP2000 concrete design options. The intent has been to highlight and demonstrate a few of the basic features in order to open up the path for you to explore and use the more advanced options. For more information on various topics consult the on-line **Help** provided with the program.

Chapter III

Design Algorithms

This chapter outlines various aspects of the concrete design and design-check procedures that are used by the SAP2000 program. The concrete design and check may be performed in SAP2000 according to one of the following design codes:

- The 1995 American Concrete Institute Building Code Requirements for Structural Concrete, **ACI 318-95** (ACI 1995).
- The 1984 Canadian Standards Association Design of Concrete Structures for Buildings, **CAN3-A23.3-M.84** (CSA 1984).
- The 1985 British Standards Institution Structural Use of Concrete, **BS 8110**, (BSI 1985).
- The 1992 European Committee for Standardization, Design of Concrete Structures, **EUROCODE 2** (CEN 1992).

Details of the algorithms associated with each of these codes as implemented in SAP2000 are described in the subsequent chapters. However, this chapter provides a background which is common to all the design codes.

In writing this manual it has been assumed that the user has an engineering background in the general area of structural reinforced concrete design and familiarity with at least one of the above mentioned design codes.

For referring to pertinent sections of the corresponding code, a unique prefix is assigned for each code. For example, all references to the ACI 318-95 code are preceded by the word "**ACI**". Similarly,

- References to the Canadian code (CSA 1984) carry the prefix of "**CAN**"
- References to the British code (BSI 1985) carry the prefix of "**BS**"
- References to the Eurocode (CEN 1992) carry the prefix of "**EC2**"

Design Load Combinations

The design load combinations are used for determining the various combinations of the load cases for which the structure needs to be designed/checked. The load combination factors to be used vary with the selected design code. The load combination factors are applied to the forces and moments obtained from the associated load cases and are then summed to obtain the factored design forces and moments for the load combination.

For multi-valued load combinations involving response spectrum, time history, moving loads and multi-valued combinations (of type enveloping, square-root of the sum of the squares or absolute) where any correspondence between interacting quantities is lost, the program automatically produces multiple sub combinations using maxima/minima permutations of interacting quantities. Separate combinations with negative factors for response spectrum cases are not required because the program automatically takes the minima to be the negative of the maxima for response spectrum cases and the above described permutations generate the required sub combinations.

When a design combination involves only a single multi-valued case of time history or moving load, further options are available. The program has an option to request that time history combinations produce sub combinations for each time step of the time history. Also an option is available to request that moving load combinations produce sub combinations using maxima and minima of each design quantity but with corresponding values of interacting quantities.

For normal loading conditions involving static dead load, live load, wind load, and earthquake load, and/or dynamic response spectrum earthquake load the program has built-in default loading combinations for each design code. These are based on the code recommendations and are documented for each code in the corresponding chapters.

For other loading conditions involving moving load, time history, pattern live loads, separate consideration of roof live load, snow load, etc., the user must define design loading combinations either in lieu of or in addition to the default design loading combinations.

The default load combinations assume all static load cases declared as dead load to be additive. Similarly, all cases declared as live load are assumed additive. However, each static load case declared as wind or earthquake, or response spectrum cases, is assumed to be non additive with each other and produces multiple lateral load combinations. Also wind and static earthquake cases produce separate loading combinations with the sense (positive or negative) reversed. If these conditions are not correct, the user must provide the appropriate design combinations.

The default load combinations are included in design if the user requests them to be included or if no other user defined combination is available for concrete design. If any default combination is included in design, then all default combinations will automatically be updated by the program any time the design code is changed or if static or response spectrum load cases are modified.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

The user is cautioned that if moving load or time history results are not requested to be recovered in the analysis for some or all the frame members, then the effects of these loads will be assumed to be zero in any combination that includes them.

Design and Check Stations

For each load combination, each element is designed or checked at a number of locations along the length of the element. The locations are based on equally spaced segments along the clear length of the element. The number of segments in an element is requested by the user before the analysis is made. The user can refine the design along the length of an element by requesting more segments.

Identifying Beams and Columns

Since SAP2000 is a general purpose analysis and design program, all beams and columns are represented as frame elements. But design of beams and columns requires separate treatment. Therefore, identifying each frame element as either a beam or a column is necessary. This identification for a concrete element is done by specifying the frame section assigned to the element to be of type beam or column.

Design of Beams

In the design of concrete beams, in general, SAP2000 calculates and reports the required areas of steel for flexure and shear based upon the beam moments, shears, load combination factors, and other criteria which are described in detail in the code specific chapters. The reinforcement requirements are calculated at a user-defined number of stations along the beam span.

All the beams are only designed for major direction flexure and shear. Effects due to any axial forces, minor direction bending, and torsion that may exist in the beams must be investigated independently by the user.

In designing the flexural reinforcement for the major moment at a particular section of a particular beam, the steps involve the determination of the maximum factored moments and the determination of the reinforcing steel. The beam section is designed for the maximum positive M_u^+ and maximum negative M_u^- factored moment envelopes obtained from all of the load combinations. Negative beam moments produce top steel. In such cases the beam is always designed as a rectangular section. Positive beam moments produce bottom steel. In such cases the beam may be designed as a rectangular- or a T-beam. For the design of flexural reinforcement, the beam is first designed as a singly reinforced beam. If the beam section is not adequate, then the required compression reinforcement is calculated.

In designing the shear reinforcement for a particular beam for a particular set of loading combinations at a particular station due to the beam major shear, the steps involve the determination of the factored shear force, the determination of the shear force that can be resisted by concrete, and the determination of the reinforcement steel required to carry the balance.

Special considerations for seismic design are incorporated in SAP2000 for ACI 318-95 and 1984 Canadian codes.

Design of Columns

In the design of the columns, the program calculates the required longitudinal steel, or if the longitudinal steel is specified, the column stress condition is reported in terms of a column capacity ratio, which is a factor that gives an indication of the stress condition of the column with respect to the capacity of the column. The design procedure for the reinforced concrete columns of the structure involves the following steps:

- Generate axial force-biaxial moment interaction surfaces for all of the different concrete section types of the model. A typical interaction surface is shown in Figure III-1.
- Check the capacity of each column for the factored axial force and bending moments obtained from each loading combination at each end of the column. This step is also used to calculate the required reinforcement (if none was specified) that will produce a capacity ratio of 1.0.
- Design the column shear reinforcement.

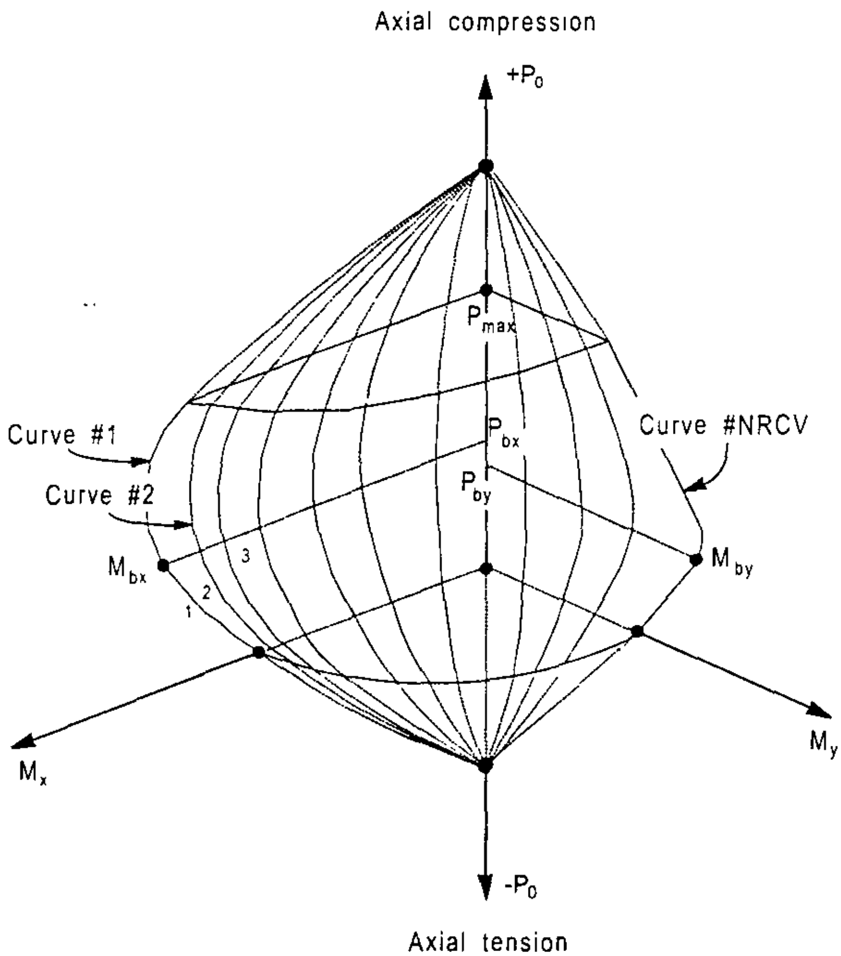


Figure III-1
A Typical Column Interaction Surface

The generation of the interaction surface is based on the assumed strain and stress distributions and some other simplifying assumptions. These stress and strain distributions and the assumptions vary from code to code. A typical assumed strain distribution is described in Figure III-2.

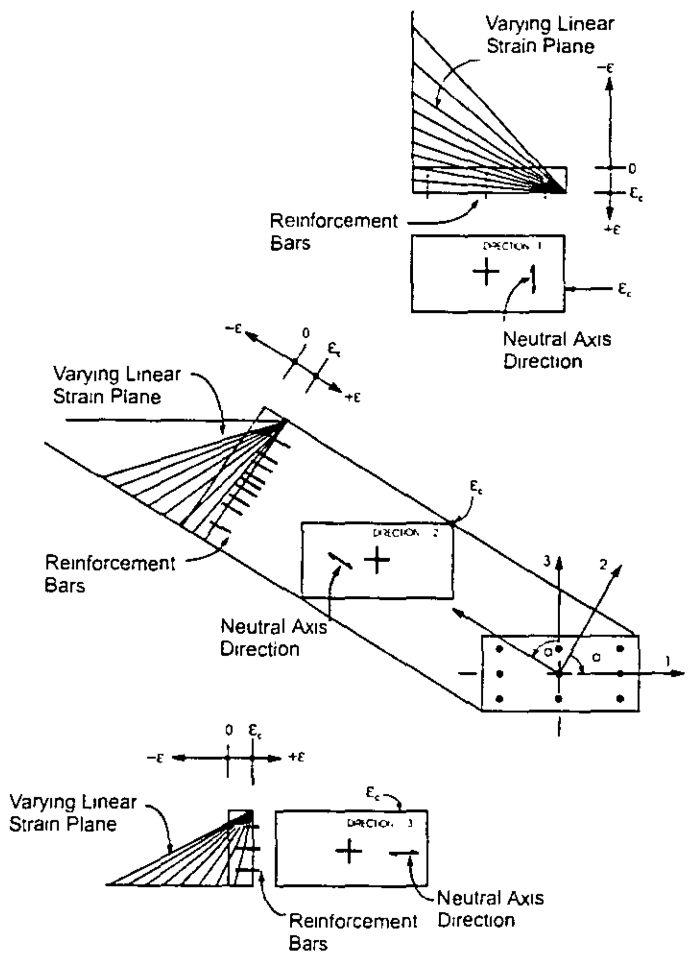


Figure III-2
Idealized Strain Distribution for Generation of Interaction Surfaces

Here maximum compression strain is limited to ε_c. For most of the design codes, this assumed distribution remains valid. However, the value of ε_c varies from code to code. For example, $\varepsilon_c = 0.003$ for ACI and Canadian codes, and $\varepsilon_c = 0.0035$ for

British and European codes. The details of the generation of interaction surfaces differ from code to code. These are described in the chapters specific to the code.

The capacity check is based on whether the design load points lie inside the interaction volume in a force space, as shown in Figure III-3. If the point lies inside the volume, the column capacity is adequate, and vice versa.

The shear reinforcement design procedure for columns is very similar to that for beams, except that the effect of the axial force on the concrete shear capacity needs to be considered.

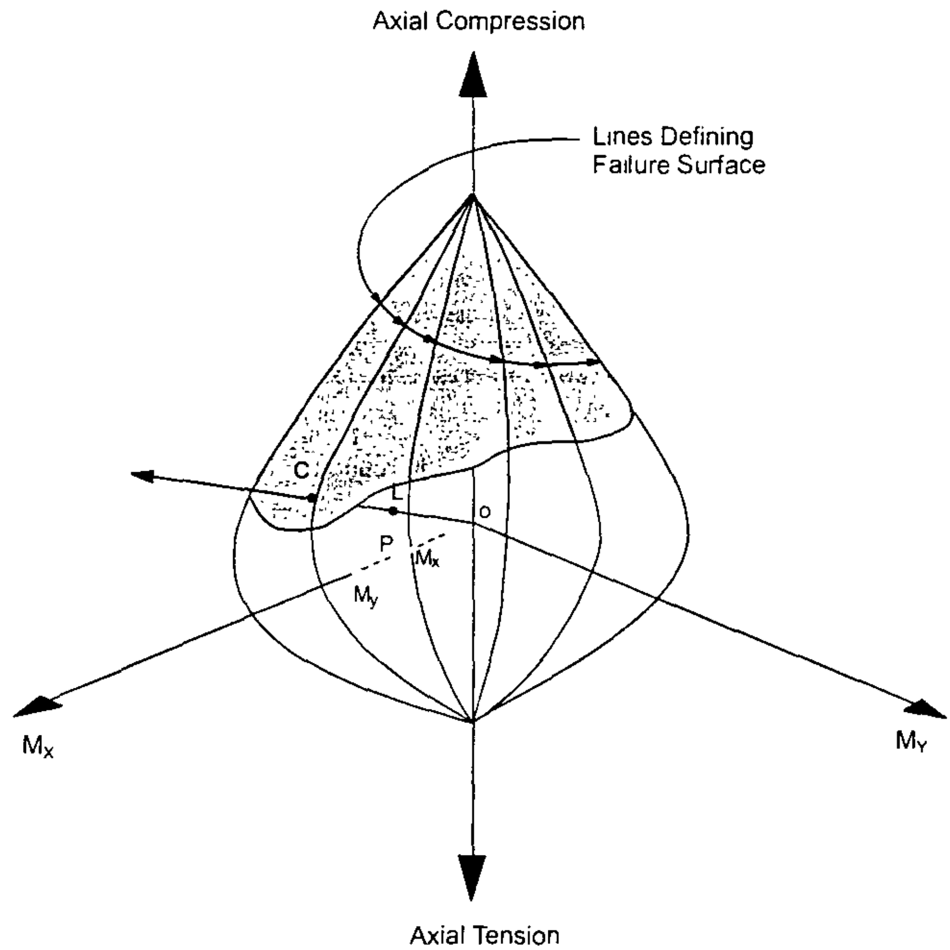


Figure III-3
Geometric Representation of Column Capacity Ratio

P-Δ Effects

The SAP2000 design algorithms require that the analysis results include the P-Δ effects. The P-Δ effects are considered differently for "braced" or "nonsway" and "unbraced" or "sway" columns or frames. For the braced columns, the effect of P-Δ is limited to "individual member stability". For unbraced frames, "lateral drift effects" should be considered in addition to individual member stability effect.

For the individual member stability effects, the moments are magnified with moment magnification factors as in the ACI and Canadian codes or with additional moments as in the British and Eurocode codes.

For lateral drift effects of unbraced or sway frames, SAP2000 assumes that the amplification is already included in the results because P-Δ effects are considered. The moments and forces obtained from P-Δ analysis are further amplified for individual column stability effect if required by the governing code as in the ACI and Canadian codes.

The users of SAP2000 should be aware that the default analysis option in SAP2000 is turned OFF for P-Δ effect. The user can turn the P-Δ analysis ON and set the maximum number of iterations for the analysis. The default number of iteration for P-Δ analysis is 1. For further reference, the user is referred to "SAP2000 Reference Manual" (CSI 1997b).

The user is also cautioned that SAP2000 currently considers P-Δ effects due to axial loads in frame members only. Forces in other types of elements do not contribute to this effect. If significant forces are present in other type of elements, for example, huge axial loads in shear walls modeled as shell elements, then the additional forces computed for P-Δ will not be accurate.

Element Unsupported Lengths

To account for column slenderness effects the column unsupported lengths are required. The two unsupported lengths are l_{33} and l_{22}. These are the lengths between support points of the element in the corresponding directions. The length l_{33} corresponds to instability about the 3-3 axis (major axis), and l_{22} corresponds to instability about the 2-2 axis (minor axis).

Normally, the unsupported element length is equal to the length of the element, i.e., the distance between END-I and END-J of the element. See Figure III-4. The program, however, allows users to assign several elements to be treated as a single member for design. This can be done differently for major and minor bending.

Therefore, extraneous joints as shown in Figure III-5 that affect the unsupported length of an element are automatically taken into consideration.

In determining the values for l_{22} and l_{33} of the elements, the program recognizes various aspects of the structure that have an effect on these lengths, such as member connectivity, diaphragm constraints and support points. The program automatically locates the element support points and evaluates the corresponding unsupported element length.

Therefore, the unsupported length of a column may actually be evaluated as being greater than the corresponding element length. If a beam frames into only one direction of the column, the beam is assumed to give lateral support only in that direction.

The user has options to specify the unsupported lengths of the elements on an element-by-element basis.

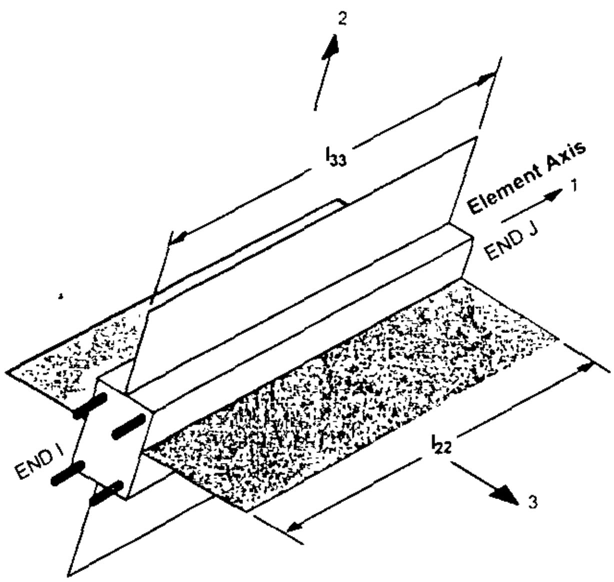


Figure III-4
Axes of Bending and Unsupported Length

Special Considerations for Seismic Loads

The ACI code (ACI 1995) imposes a special ductility requirement for frames in seismic regions by specifying frames either as Ordinary, Intermediate, or Special moment resisting frames. The Special moment resisting frame can provide the required ductility and energy dissipation in the nonlinear range of cyclic deformation. The Canadian code (CSA 1984) also requires that the concrete frame must be designed as either an Ordinary, Nominal, or Ductile moment resisting frame.

Unlike the ACI (ACI 1995) and the Canadian code (CSA 1984), the current implementation of the British code (BSI 1985) and the Eurocode (CEN 1992) in SAP2000 does not account for any special requirements for seismic design.

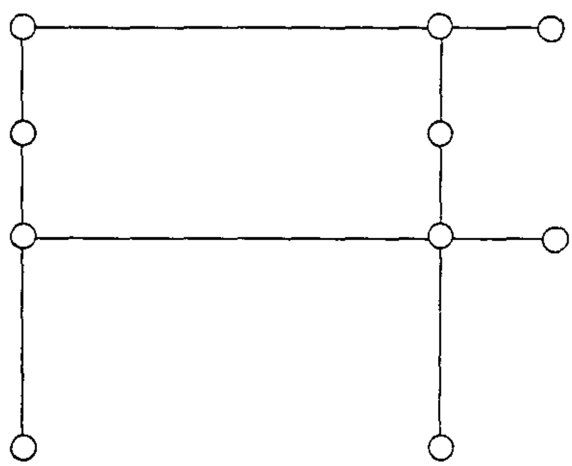


Figure III-5
Unsupported Lengths and Interior Nodes

Choice of Input Units

English as well as SI and MKS metric units can be used for input. But the codes are based on a specific system of units. All equations and descriptions presented in the subsequent chapters correspond to that specific system of units unless otherwise noted. For example, the ACI code is published in inch-pound-second units. By default, all equations and descriptions presented in the chapter "Design for ACI 318-95" correspond to inch-pound-second units. However, any system of units can be used to define and design the structure in SAP2000.

Chapter IV

Design for ACI 318-95

This chapter describes in detail the various aspects of the concrete design procedure that is used by SAP2000 when the user selects the ACI 318-95 Design Code (ACI 1995). Various notations used in this chapter are listed in Table IV-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

SAP2000 provides options to design or check Ordinary, Intermediate (moderate seismic risk areas), and Special (high seismic risk areas) moment resisting frames as required for seismic design provisions. The details of the design criteria used for the different framing systems are described in the following sections.

English as well as SI and MKS metric units can be used for input. But the code is based on Inch-Pound-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Inch-Pound-Second** units unless otherwise noted.

Design Load Combinations

The design load combinations are the various combinations of the prescribed load cases for which the structure needs to be checked. For the ACI 318-95 code, if a

A_{cv}	Area of concrete used to determine shear stress, sq-in
A_g	Gross area of concrete, sq-in
A_s	Area of tension reinforcement, sq-in
A_s'	Area of compression reinforcement, sq-in
$A_{s(required)}$	Area of steel required for tension reinforcement, sq-in
A_{st}	Total area of column longitudinal reinforcement, sq-in
A_v	Area of shear reinforcement, sq-in
a	Depth of compression block, in
a_b	Depth of compression block at balanced condition, in
b	Width of member, in
b_f	Effective width of flange (T-Beam section), in
b_w	Width of web (T-Beam section), in
C_m	Coefficient, dependent upon column curvature, used to calculate moment magnification factor
c	Depth to neutral axis, in
c_b	Depth to neutral axis at balanced conditions, in
d	Distance from compression face to tension reinforcement, in
d'	Concrete cover to center of reinforcing, in
d_s	Thickness of slab (T-Beam section), in
E_c	Modulus of elasticity of concrete, psi
E_s	Modulus of elasticity of reinforcement, assumed as 29,000,000 psi
f_c'	Specified compressive strength of concrete, psi
f_y	Specified yield strength of flexural reinforcement, psi
f_{ys}	Specified yield strength of shear reinforcement, psi
h	Dimension of column, in
I_g	Moment of inertia of gross concrete section about centroidal axis, neglecting reinforcement, in^4
I_{se}	Moment of inertia of reinforcement about centroidal axis of member cross section, in^4
k	Effective length factor
L	Clear unsupported length, in

Table IV-1

List of Symbols Used in the ACI code

M_1	Smaller factored end moment in a column, lb-in
M_2	Larger factored end moment in a column, lb-in
M_c	Factored moment to be used in design, lb-in
M_{ns}	Nonsway component of factored end moment, lb-in
M_s	Sway component of factored end moment, lb-in
M_u	Factored moment at section, lb-in
M_{ux}	Factored moment at section about X-axis, lb-in
M_{uy}	Factored moment at section about Y-axis, lb-in
P_b	Axial load capacity at balanced strain conditions, lb
P_c	Critical buckling strength of column, lb
P_{max}	Maximum axial load strength allowed, lb
P_0	Axial load capacity at zero eccentricity, lb
P_u	Factored axial load at section, lb
r	Radius of gyration of column section, in
V_c	Shear resisted by concrete, lb
V_E	Shear force caused by earthquake loads, lb
V_{D+L}	Shear force from span loading, lb
V_u	Factored shear force at a section, lb
V_p	Shear force computed from probable moment capacity, lb
α	Reinforcing steel overstrength factor
β_1	Factor for obtaining depth of compression block in concrete
β_d	Absolute value of ratio of maximum factored axial dead load to maximum factored axial total load
δ_s	Moment magnification factor for sway moments
δ_{ns}	Moment magnification factor for nonsway moments
ε_c	Strain in concrete
ε_s	Strain in reinforcing steel
φ	Strength reduction factor

Table IV-1

List of Symbols Used in the ACI code (continued) structure is subjected to dead load (DL) and live load (LL) only, the stress check may need only one load combination, namely 1.4 DL + 1.7 LL (ACI 9.2.1). However, in addition to the dead and live loads, if the structure is subjected to wind (WL) and earthquake (EL) loads, and considering that wind and earthquake forces are reversible, then the following load combinations have to be considered (ACI 9.2).

$$1.4\ DL$$
$$1.4\ DL + 1.7\ LL \qquad \text{(ACI 9.2.1)}$$

$$0.9\ DL \pm 1.3\ WL$$
$$0.75\ (1.4\ DL + 1.7\ LL \pm 1.7\ WL) \qquad \text{(ACI 9.2.2)}$$

$$0.9\ DL \pm 1.3 * 1.1\ EL$$
$$0.75\ (1.4\ DL + 1.7\ LL \pm 1.7 * 1.1\ EL) \qquad \text{(ACI 9.2.3)}$$

These are also the default design load combinations in SAP2000 whenever the ACI 318-95 code is used.

Live load reduction factors can be applied to the member forces of the live load condition on an element-by-element basis to reduce the contribution of the live load to the factored loading.

Strength Reduction Factors

The strength reduction factors, φ, are applied on the nominal strength to obtain the design strength provided by a member. The φ factors for flexure, axial force, shear, and torsion are as follows:

φ = 0.90 for flexure, (ACI 9.3.2.1)

φ = 0.90 for axial tension, (ACI 9.3.2.2)

φ = 0.90 for axial tension and flexure, (ACI 9.3.2.2)

φ = 0.75 for axial compression, and axial compression and flexure (spirally reinforced column), (ACI 9.3.2.2)

φ = 0.70 for axial compression, and axial compression and flexure (tied column), and (ACI 9.3.2.2)

φ = 0.85 for shear and torsion. (ACI 9.3.2.3)

Column Design

The user may define the geometry of the reinforcing bar configuration of each concrete column section. If the area of reinforcing is provided by the user, the program checks the column capacity. However, if the area of reinforcing is not provided by the user, the program calculates the amount of reinforcing required for the column. The design procedure for the reinforced concrete columns of the structure involves the following steps:

- Generate axial force-biaxial moment interaction surfaces for all of the different concrete section types of the model. A typical biaxial interaction surface is shown in Figure III-1. When the steel is undefined, the program generates the interaction surfaces for the range of allowable reinforcement — 1 to 8 percent for Ordinary and Intermediate moment resisting frames (ACI 10.9.1) and 1 to 6 percent for Special moment resisting frames (ACI 21.4.3.1).
- Calculate the capacity ratio or the required reinforcing area for the factored axial force and biaxial (or uniaxial) bending moments obtained from each loading combination at each station of the column. The target capacity ratio is taken as one when calculating the required reinforcing area.
- Design the column shear reinforcement.

The following three subsections describe in detail the algorithms associated with the above-mentioned steps.

Generation of Biaxial Interaction Surfaces

The column capacity interaction volume is numerically described by a series of discrete points that are generated on the three-dimensional interaction failure surface. In addition to axial compression and biaxial bending, the formulation allows for axial tension and biaxial bending considerations. A typical interaction diagram is shown in Figure III-1.

The coordinates of these points are determined by rotating a plane of linear strain in three dimensions on the section of the column. See Figure III-2. The linear strain diagram limits the maximum concrete strain, ε_c, at the extremity of the section to 0.003 (ACI 10.2.3).

The formulation is based consistently upon the general principles of ultimate strength design (ACI 10.3), and allows for any doubly symmetric rectangular, square, or circular column section.

The stress in the steel is given by the product of the steel strain and the steel modulus of elasticity, $\varepsilon_s E_s$, and is limited to the yield stress of the steel, f_y (ACI 10.2.4). The area associated with each reinforcing bar is assumed to be placed at the actual location of the center of the bar and the algorithm does not assume any further simplifications in the manner in which the area of steel is distributed over the cross section of the column, such as an equivalent steel tube or cylinder. See Figure IV-1.

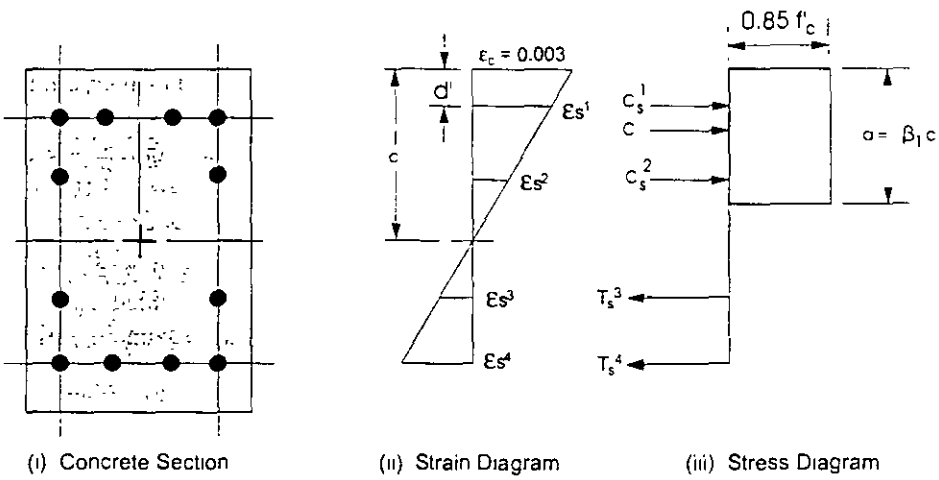


Figure IV-1
Idealization of Stress and Strain Distribution in a Column Section

The concrete compression stress block is assumed to be rectangular, with a stress value of $0.85 f'_c$ (ACI 10.2.7.1). See Figure IV-1. The interaction algorithm provides correction to account for the concrete area that is displaced by the reinforcement in the compression zone.

The effects of the strength reduction factor, φ, are included in the generation of the interaction surfaces. The maximum compressive axial load is limited to P_{max}, where

$P_{max} = 0.85\ \varphi[0.85 f'_c (A_g - A_{st}) + f_y A_{st}]$ spiral column, (ACI 10.3.5.1)

$P_{max} = 0.80\ \varphi\ [0.85 f'_c (A_g - A_{st}) + f_y A_{st}]$ tied column, (ACI 10.3.5.2)

$\varphi = 0.70$ for tied columns, and

$\varphi = 0.75$ for spirally reinforced columns.

The value of φ used in the interaction diagram varies from φ_{min} to 0.9 based on the axial load. For low values of axial load, φ is increased linearly from φ_{min} to 0.9 as the axial load decreases from the smaller of $0.1 f'_c A_g$ or P_b to zero, where P_b is the axial force at the balanced condition. In cases involving axial tension, φ is always 0.9 (ACI 9.3.2.2).

Check Column Capacity

The column capacity is checked for each loading combination at each check station of each column. In checking a particular column for a particular loading combination at a particular station, the following steps are involved:

- Determine the factored moments and forces from the analysis load cases and the specified load combination factors to give P_u, M_{ux}, and M_{uy}.
- Determine the moment magnification factors for the column moments.
- Apply the moment magnification factors to the factored moments. Determine whether the point, defined by the resulting axial load and biaxial moment set, lies within the interaction volume.

The factored moments and corresponding magnification factors depend on the identification of the individual column as either "sway" or "non-sway".

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Determine Factored Moments and Forces

The factored loads for a particular load combination are obtained by applying the corresponding load factors to all the load cases, giving P_u, M_{ux}, and M_{uy}. The factored moments are further increased for non-sway columns, if required, to obtain minimum eccentricities of $(0.6 + 0.03h)$ inches, where h is the dimension of the column in the corresponding direction (ACI 10.12.3.2).

Determine Moment Magnification Factors

The moment magnification factors are calculated separately for sway (overall stability effect), δ_s and for non-sway (individual column stability effect), δ_{ns}. Also the moment magnification factors in the major and minor directions are in general different.

The program assumes that a P-Δ analysis has been performed in SAP2000 and, therefore, moment magnification factors for moments causing sidesway are taken as unity (ACI 10.10.2). For the P-Δ analysis the load should correspond to a load combination of 0.75 (1.4 dead load + 1.7 live load) / φ, where φ is the understrength factor for stability which is taken as 0.75 (ACI 10.12.3). See also White and Hajjar (1991).

The moment obtained from analysis is separated into two components: the sway (M_s) and the non-sway (M_{ns}) components. The non-sway components which are identified as "*ns*" subscripts are predominantly caused by gravity load. The sway components are identified by the "*s*" subscripts. The sway moments are predominantly caused by lateral loads, and are related to the cause of side sway.

For individual columns or column-members in a floor, the magnified moments about two axes at any station of a column can be obtained as

$$M = M_{ns} + \delta_s M_s . \qquad \text{(ACI 10.13.3)}$$

The factor δ_s is the moment magnification factor for moments causing side sway. The moment magnification factors for sway moments, δ_s, is taken as 1 because the component moments M_s and M_{ns} are obtained from a "second order elastic (P-Δ) analysis" (ACI R10.13).

The computed moments are further amplified for individual column stability effect (ACI 10.13.5) by the nonsway moment magnification factor, δ_{ns}, as follows:

$$M_c = \delta_{ns} M_2 \text{, where} \qquad \text{(ACI 10.12.3)}$$

M_c is the factored moment to be used in design, and

M_2 is the larger factored and amplified end moment.

The non-sway moment magnification factor, δ_{ns}, associated with the major or minor direction of the column is given by (ACI 10.12.3)

$$\delta_{ns} = \frac{C_m}{1 - \dfrac{P_u}{0.75\, P_c}} \geq 1.0, \quad \text{where}$$

$$P_c = \frac{\pi^2 EI}{(kL)^2},$$

k is conservatively taken as 1, however SAP2000 allows the user to override this value, and

EI is associated with a particular column direction given by the larger of:

$$EI = \frac{0.2E_c\, I_g + E_s I_{se}}{1 + \beta_d},$$

$$EI = \frac{0.4\, E_c\, I_g}{1 + \beta_d},$$

$$\beta_d = \frac{\text{maximum factored axial dead load}}{\text{maximum factored axial total load}}, \text{ and}$$

$$C_m = 0.6 + 0.4\frac{M_a}{M_b} \geq 0.4. \qquad \text{(ACI 10.12.3.1)}$$

M_a and M_b are the moments at the ends of the column, and M_b is numerically larger than M_a. M_a/M_b is positive for single curvature bending and negative for double curvature bending. The above expression of C_m is valid if there is no transverse load applied between the supports. If transverse load is present on the span, or the length is overwritten, or for any other case, $C_m = 1$. C_m can be overwritten by the user on an element by element basis.

The magnification factor, δ_{ns}, must be a positive number and greater than one. Therefore P_u must be less than $0.75P_c$. If P_u is found to be greater than or equal to $0.75P_c$, a failure condition is declared.

The above calculations use the unsupported length of the column. The two unsupported lengths are l_{22} and l_{33} corresponding to instability in the minor and major directions of the element, respectively. See Figure III-4. These are the lengths between the support points of the element in the corresponding directions.

If the program assumptions are not satisfactory for a particular member, the user can explicitly specify values of δ_s and δ_{ns}.

Determine Capacity Ratio

As a measure of the stress condition of the column, a capacity ratio is calculated. The capacity ratio is basically a factor that gives an indication of the stress condition of the column with respect to the capacity of the column.

Before entering the interaction diagram to check the column capacity, the moment magnification factors are applied to the factored loads to obtain P_u, M_{ux}, and M_{uy}. The point (P_u, M_{ux}, M_{uy}) is then placed in the interaction space shown as point L in Figure III-3. If the point lies within the interaction volume, the column capacity is adequate; however, if the point lies outside the interaction volume, the column is overstressed.

This capacity ratio is achieved by plotting the point L and determining the location of point C. The point C is defined as the point where the line OL (if extended outwards) will intersect the failure surface. This point is determined by three-dimensional linear interpolation between the points that define the failure surface. See Figure III-3. The capacity ratio, CR, is given by the ratio $\frac{OL}{OC}$.

- If OL = OC (or CR=1) the point lies on the interaction surface and the column is stressed to capacity.
- If OL < OC (or CR<1) the point lies within the interaction volume and the column capacity is adequate.
- If OL > OC (or CR>1) the point lies outside the interaction volume and the column is overstressed.

The maximum of all the values of CR calculated from each load combination is reported for each check station of the column along with the controlling P_u, M_{ux}, and M_{uy} set and associated load combination number.

If the reinforcing area is not defined, SAP2000 computes the reinforcement that will give an interaction ratio of unity.

Design Column Shear Reinforcement

The shear reinforcement is designed for each loading combination in the major and minor directions of the column. In designing the shear reinforcing for a particular column for a particular loading combination due to shear forces in a particular direction, the following steps are involved:

- Determine the factored forces acting on the section, P_u and V_u. Note that P_u is needed for the calculation of V_c.
- Determine the shear force, V_c, that can be resisted by concrete alone.
- Calculate the reinforcement steel required to carry the balance.

For special moment resisting frames (ductile frames), the shear design of the columns is based upon the probable moment capacities of the members. Effects of the axial forces on the column moment capacities are included in the formulation.

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Determine Section Forces

- In the design of the column shear reinforcement of an **Ordinary moment resisting concrete frame**, the forces for a particular load combination, namely, the column axial force, P_u, and the column shear force, V_u, in a particular direction are obtained by factoring the SAP2000 analysis load cases with the corresponding load combination factors.
- In the shear design of **Special moment resisting frames** (seismic design) the following are checked in addition to the requirement for the ordinary moment resisting frames. In the design of Special moment resisting concrete frames, the design shear force in a column, V_u, in a particular direction is also calculated from the probable moment capacities of the column associated with the factored axial force acting on the column.

  For each load combination, the factored axial load, P_u, is calculated. Then, the positive and negative moment capacities, M_u^+ and M_u^-, of the column in a particular direction under the influence of the axial force P_u is calculated using the uniaxial interaction diagram in the corresponding direction. The design shear force, V_u, is then given by (ACI 21.4.5.1)

  $$V_u = V_p + V_{D+L} \qquad \text{(ACI 21.4.5.1)}$$

  where, V_p is the shear force obtained by applying the calculated probable ultimate moment capacities at the two ends of the column acting in two opposite directions. Therefore, V_p is the maximum of V_{P_1} and V_{P_2}, where

  $$V_{P_1} = \frac{M_I^- + M_J^+}{L}, \text{ and}$$

  $$V_{P_2} = \frac{M_I^+ + M_J^-}{L}, \text{ where}$$

  M_I^+, M_I^- = Positive and negative moment capacities at end I of the column using a steel yield stress value of αf_y and no φ factors,

  M_J^+, M_J^- = Positive and negative moment capacities at end J of the column using a steel yield stress value of αf_y and no φ factors, and

  L = Clear span of column.

For Special moment resisting frames α is taken as 1.25 (ACI R21.3.4.1). V_{D+L} is the contribution of shear force from the in-span distribution of gravity loads. For most of the columns, it is zero.

- For **Intermediate moment resisting frames**, the design shear force is taken to be the minimum of that based on the nominal moment capacity and factored shear force The procedure for calculating nominal moment capacity is the same as that for computing the probable moment capacity for special moment resisting frames, except that α is taken equal to 1 rather than 1.25. The factored shear forces are based on the specified load factors except the earthquake load factors are doubled (ACI 21.8.3).

Determine Concrete Shear Capacity

Given the design force set P_u and V_u, the shear force carried by the concrete, V_c, is calculated as follows:

- If the column is subjected to axial compression, i.e. P_u is positive,

$$V_c = 2\sqrt{f'_c}\left(1 + \frac{P_u}{2000A_g}\right)A_{cv}, \qquad \text{(ACI 11.3.1.2)}$$

where,

$$\sqrt{f'_c} \leq 100 \text{ psi, and} \qquad \text{(ACI 11.1.2)}$$

$$V_c \leq 3.5\sqrt{f'_c}\sqrt{\left(1 + \frac{P_u}{500A_g}\right)}A_{cv}. \qquad \text{(ACI 11.3.2.2)}$$

The term $\frac{P_u}{A_g}$ must have psi units. A_{cv} is the effective shear area which is shown shaded in Figure IV-2.

- If the column is subjected to axial tension, P_u is negative,

$$V_c = 2\sqrt{f'_c}\left(1 + \frac{P_u}{500A_g}\right)A_{cv} \geq 0 \qquad \text{(ACI 11.3.2.3)}$$

- For **Special moment resisting concrete frame** design, V_c is set to zero if the factored axial compressive force, P_u, including the earthquake effect is small ($P_u < f'_c A_g/20$) and if the shear force contribution from earthquake, V_E, is more than half of the total factored maximum shear force over the length of the member V_u ($V_E \geq 0.5 V_u$) (ACI 21.4.5.2).

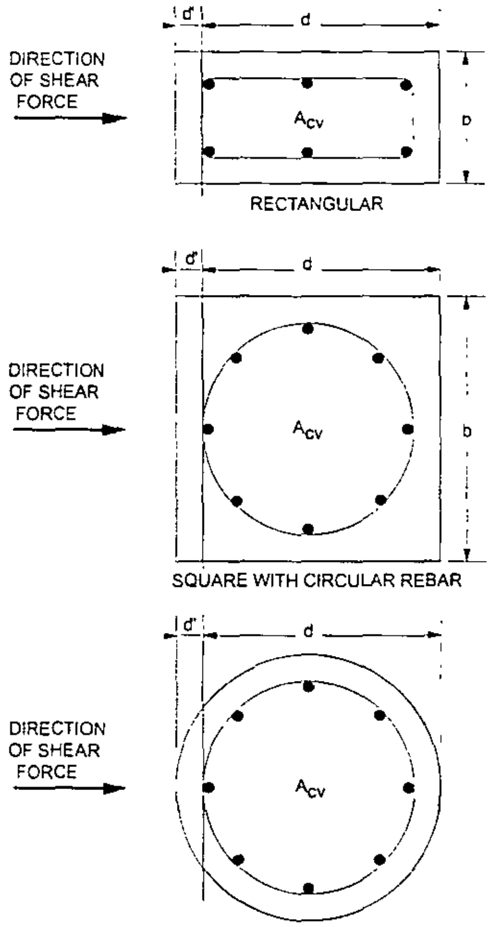


Figure IV-2
Shear Stress Area, A_{cv}

Determine Required Shear Reinforcement

Given V_u and V_c, the required shear reinforcement in the form of stirrups or tie within a spacing, s, is given by

$$A_v = \frac{(V_u/\varphi - V_c)\,s}{f_{ys}\,d}, \qquad \text{(ACI 11.5.6.2)}$$

$$(V_u/\varphi - V_c) \leq 8\sqrt{f'_c}A_{cv} \qquad \text{(ACI 11.5.6.8)}$$

Otherwise redimensioning of the concrete section is required. Here φ, the strength reduction factor, is 0.85 (ACI 9.3.2.3). The maximum of all the calculated A_v values obtained from each load combination are reported for the major and minor directions of the column along with the controlling shear force and associated load combination label.

The column shear reinforcement requirements reported by the program are based purely upon shear strength consideration. Any minimum stirrup requirements to satisfy spacing considerations or transverse reinforcement volumetric considerations must be investigated independently of the program by the user.

Beam Design

In the design of concrete beams, SAP2000 calculates and reports the required areas of steel for flexure and shear based upon the beam moments, shears, load combination factors, and other criteria described below. The reinforcement requirements are calculated at a user defined number of check/design stations along the beam span.

All the beams are only designed for major direction flexure and shear. Effects due to any axial forces, minor direction bending, and torsion that may exist in the beams must be investigated independently by the user.

The beam design procedure involves the following steps:

- Design beam flexural reinforcement
- Design beam shear reinforcement

Design Beam Flexural Reinforcement

The beam top and bottom flexural steel is designed at check/design stations along the beam span. In designing the flexural reinforcement for the major moment for a particular beam for a particular section, the following steps are involved:

- Determine the maximum factored moments
- Determine the reinforcing steel

Determine Factored Moments

In the design of flexural reinforcement of Special, Intermediate, or Ordinary moment resisting concrete frame beams, the factored moments for each load combination at a particular beam section are obtained by factoring the corresponding moments for different load cases with the corresponding load factors.

The beam section is then designed for the maximum positive M_u^+ and maximum negative M_u^- factored moments obtained from all of the load combinations.

Negative beam moments produce top steel. In such cases the beam is always designed as a rectangular section. Positive beam moments produce bottom steel. In such cases the beam may be designed as a Rectangular- or a T-beam.

Determine Required Flexural Reinforcement

In the flexural reinforcement design process, the program calculates both the tension and compression reinforcement. Compression reinforcement is added when the applied design moment exceeds the maximum moment capacity of a singly reinforced section. The user has the option of avoiding the compression reinforcement by increasing the effective depth, the width, or the grade of concrete.

The design procedure is based on the simplified rectangular stress block as shown in Figure IV-3 (ACI 10.2). Furthermore it is assumed that the compression carried by concrete is less than 0.75 times that which can be carried at the balanced condition (ACI 10.3.3). When the applied moment exceeds the moment capacity at this designed balanced condition, the area of compression reinforcement is calculated on the assumption that the additional moment will be carried by compression and additional tension reinforcement.

The design procedure used by SAP2000, for both rectangular and flanged sections (L- and T-beams) is summarized below. It is assumed that the design ultimate axial force does not exceed $0.1f'_cA_g$ (ACI 10.3.3), hence all the beams are designed for major direction flexure and shear only.

Design for Rectangular Beam

In designing for a factored negative or positive moment, M_u, (i.e. designing top or bottom steel) the depth of the compression block is given by a (see Figure IV-3), where,

$$a = d - \sqrt{d^2 - \frac{2\,|M_u|}{0.85 f'_c\, \varphi\, b}},$$

where, the value of φ is 0.90 (ACI 9.3.2.1) in the above and the following equations. Also β_1 and c_b are calculated as follows:

$$\beta_1 = 0.85 - 0.05\left(\frac{f'_c - 4000}{1000}\right), \qquad 0.65 \le \beta_1 \le 0.85, \qquad \text{(ACI 10.2.7.3)}$$

$$c_b = \frac{87000}{87000 + f_y}\, d.$$

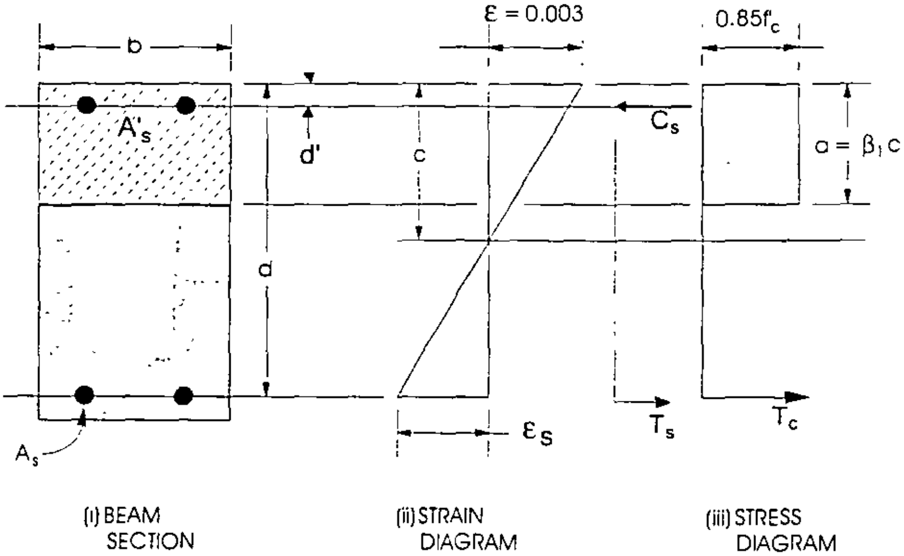


Figure IV-3
Design of Rectangular Beam Section

- If $a \le 0.75\beta_1 c_b$, the area of tensile steel reinforcement is then given by

$$A_s = \frac{M_u}{\varphi f_y\left(d - \frac{a}{2}\right)}.$$

  This steel is to be placed at the bottom if M_u is positive, or at the top if M_u is negative.

- If $a > 0.75\beta_1 c_b$, compression reinforcement is required (ACI 10.3.3) and is calculated as follows:

- The depth of compression block is given by

$$a_b = 0.75\beta_1 c_b \,. \qquad \text{(ACI 10.3.3)}$$

- The compressive force developed in concrete alone is given by

$$C = 0.85 f'_c b a_b \,, \text{ and} \qquad \text{(ACI 10.2.7.1)}$$

the moment resisted by concrete compression and tensile steel is

$$M_{uc} = C\left(d - \frac{a_b}{2}\right)\varphi \,.$$

- Therefore the moment resisted by compression steel and tensile steel is

$$M_{us} = M_u - M_{uc} \,.$$

- So the required compression steel is given by

$$A'_s = \frac{M_{us}}{f'_s(d - d')\varphi} \,, \text{ where}$$

$$f'_s = 0.003\, E_s \left[\frac{c - d'}{c}\right]. \qquad \text{(ACI 10.2.4)}$$

- The required tensile steel for balancing the compression in concrete is

$$A_{s1} = \frac{M_{uc}}{f_y(d - \frac{a_b}{2})\varphi} \,, \text{ and}$$

the tensile steel for balancing the compression in steel is given by

$$A_{s2} = \frac{M_{us}}{f_y(d - d')\varphi} \,.$$

- Therefore, the total tensile reinforcement, $A_s = A_{s1} + A_{s2}$, and total compression reinforcement is A'_s. A_s is to be placed at bottom and A'_s is to be placed at top if M_u is positive, and vice versa if M_u is negative.

Design for T-Beam

In designing for a factored negative moment, M_u, (i.e. designing top steel), the calculation of the steel area is exactly the same as above, i.e., no T-Beam data is to be used. See Figure IV-4. If $M_u > 0$, the depth of the compression block is given by

$$a = d - \sqrt{d^2 - \frac{2\,M_u}{0.85 f'_c\,\varphi\,b_f}}\,.$$

The depth of compression block under design balanced condition is given by

$$a_b = 0.75\beta_1 c_b\,.$$

- If $a \le d_s$, the subsequent calculations for A_s are exactly the same as previously defined for the rectangular section design. However, in this case the width of the compression flange is taken as the width of the beam for analysis. Whether compression reinforcement is required depends on whether $a > a_b$.
- If $a > d_s$, calculation for A_s is done in two parts. The first part is for balancing the compressive force from the flange, C_f, and the second part is for balancing the compressive force from the web, C_w, as shown in Figure IV-4. C_f is given by

$$C_f = 0.85 f'_c (b_f - b_w) d_s\,.$$

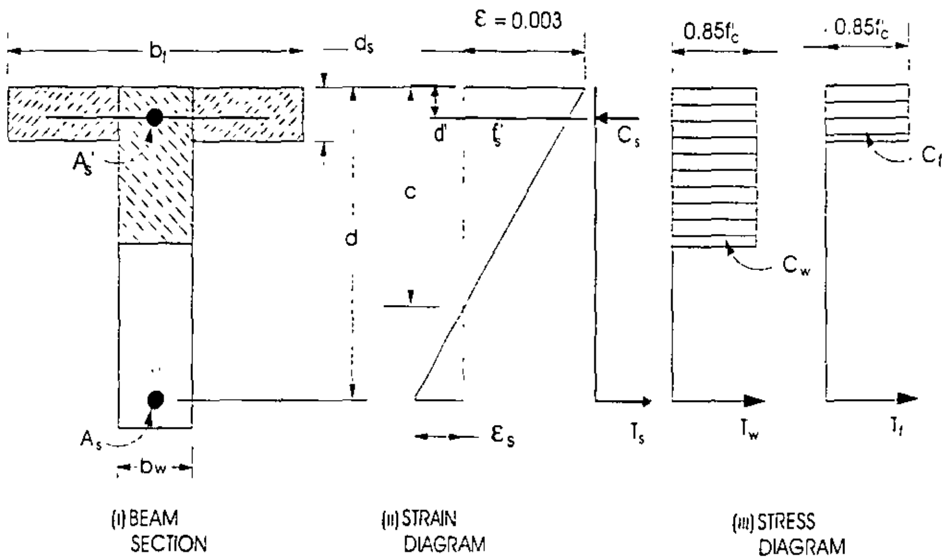


Figure IV-4
Design of a T-Beam Section

Therefore, $A_{s1} = \dfrac{C_f}{f_y}$ and the portion of M_u that is resisted by the flange is given by

$$M_{uf} = C_f\left(d - \frac{d_s}{2}\right)\varphi .$$

Again, the value for φ is 0.90. Therefore, the balance of the moment, M_u to be carried by the web is given by

$$M_{uw} = M_u - M_{uf} .$$

The web is a rectangular section of dimensions b_w and d, for which the design depth of the compression block is recalculated as

$$a_1 = d - \sqrt{d^2 - \frac{2M_{uw}}{0.85 f'_c\, \varphi\, b_w}} .$$

- If $a_1 \le a_b$, the area of tensile steel reinforcement is then given by

  $$A_{s2} = \frac{M_{uw}}{\varphi f_y\left(d - \frac{a_1}{2}\right)} \text{, and}$$

  $$A_s = A_{s1} + A_{s2} .$$

  This steel is to be placed at the bottom of the T-beam.

- If $a_1 > a_b$, compression reinforcement is required (ACI 10.3.3) and is calculated as follows:
  - The compressive force in web concrete alone is given by

    $$C = 0.85 f'_c b a_b . \qquad \text{(ACI 10.2.7.1)}$$

  - Therefore the moment resisted by concrete web and tensile steel is

    $$M_{uc} = C\left(d - \frac{a_b}{2}\right)\varphi \text{ , and}$$

    the moment resisted by compression steel and tensile steel is

    $$M_{us} = M_{uw} - M_{uc} .$$

  - Therefore, the compression steel is computed as

$$A_s' = \frac{M_{us}}{f_s'(d - d')\varphi}, \text{ where}$$

$$f_s' = 0.003\, E_s \left[\frac{c - d'}{c}\right]. \qquad \text{(ACI 10.2.4)}$$

– The tensile steel for balancing compression in web concrete is

$$A_{s2} = \frac{M_{uc}}{f_y(d - \frac{a_2}{2})\varphi}, \text{ and}$$

the tensile steel for balancing compression in steel is

$$A_{s3} = \frac{M_{us}}{f_y(d - d')\varphi}.$$

– The total tensile reinforcement, $A_s = A_{s1} + A_{s2} + A_{s3}$, and total compression reinforcement is A_s'. A_s is to be placed at bottom and A_s' is to be placed at top.

Minimum Tensile Reinforcement

The minimum flexural tensile steel provided in a rectangular section in an Ordinary moment resisting frame is given by the minimum of the two following limits:

$$A_s \geq \max\left\{\frac{3\sqrt{f_c'}}{f_y} b_w d \text{ and } \frac{200}{f_y} b_w d\right\} \text{ or} \qquad \text{(ACI 10.5.1)}$$

$$A_s \geq \frac{4}{3} A_{s(required)} \qquad \text{(ACI 10.5.3)}$$

Special Consideration for Seismic Design

For Special moment resisting concrete frames (seismic design), the beam design satisfies the following additional conditions (see also Table IV-2 for comprehensive listing) :

- The minimum longitudinal reinforcement shall be provided at both at the top and bottom. Any of the top and bottom reinforcement shall not be less than $A_{s(min)}$ (ACI 21.3.2.1).

$$A_{s(min)} \geq \max\left\{\frac{3\sqrt{f'_c}}{f_y} b_w d \quad \text{and} \quad \frac{200}{f_y} b_w d\right\} \text{ or} \qquad \text{(ACI 10.5.1)}$$

$$A_{s(min)} \geq \frac{4}{3} A_{s(required)}. \qquad \text{(ACI 10.5.3)}$$

- The beam flexural steel is limited to a maximum given by

$$A_s \geq 0.025\, b_w d. \qquad \text{(ACI 21.3.2.1)}$$

- At any end (support) of the beam, the beam positive moment capacity (i.e. associated with the bottom steel) would not be less than 1/2 of the beam negative moment capacity (i.e. associated with the top steel) at that end (ACI 21.3.2.2).
- Neither the negative moment capacity nor the positive moment capacity at any of the sections within the beam would be less than 1/4 of the maximum of positive or negative moment capacities of any of the beam end (support) stations (ACI 21.3.2.2).

For Intermediate moment resisting concrete frames (seismic design), the beam design would satisfy the following conditions:

- At any support of the beam, the beam positive moment capacity would not be less than 1/3 of the beam negative moment capacity at that end (ACI 21.8.4.1).
- Neither the negative moment capacity nor the positive moment capacity at any of the sections within the beam would be less than 1/5 of the maximum of positive or negative moment capacities of any of the beam end (support) stations (ACI 21.8.4.1.).

Design Beam Shear Reinforcement

The shear reinforcement is designed for each load combination at a user defined number of stations along the beam span. In designing the shear reinforcement for a particular beam for a particular loading combination at a particular station due to the beam major shear, the following steps are involved:

- Determine the factored shear force, V_u.
- Determine the shear force, V_c, that can be resisted by the concrete.
- Determine the reinforcement steel required to carry the balance.

Type of Check/ Design	Ordinary Moment Resisting Frames (non-Seismic)	Intermediate Moment Resisting Frames (Seismic)	Special Moment Resisting Frames (Seismic)
Column Check (interaction)	NLD† Combinations	NLD† Combinations	NLD† Combinations
Column Design (Interaction)	NLD† Combinations $1\% < \rho < 8\%$	NLD† Combinations $1\% < \rho < 8\%$	NLD† Combinations $\alpha = 1.0$ $1\% < \rho < 6\%$
Column Shears	NLD† Combinations	Modified NLD† Combinations (earthquake loads doubled) Column Capacity $\varphi = 1.0$ and $\alpha = 1.0$	NLD† Combinations and Column shear capacity $\varphi = 1.0$ and $\alpha = 1.25$
Beam Design Flexure	NLD† Combinations	NLD† Combinations	NLD† Combinations $\rho \leq 0.025$ $\rho \geq \frac{3\sqrt{f'_c}}{f_y}, \rho \geq \frac{200}{f_y}$
Beam Min. Moment Override Check	No Requirement	$M^+_{uEND} \geq \frac{1}{3} M^-_{uEND}$ $M^+_{uSPAN} \geq \frac{1}{5}\max\{M^+_u, M^-_u\}_{END}$ $M^-_{uSPAN} \geq \frac{1}{5}\max\{M^+_u, M^-_u\}_{END}$	$M^+_{uEND} \geq \frac{1}{2} M^-_{uEND}$ $M^+_{uSPAN} \geq \frac{1}{4}\max\{M^+_u, M^-_u\}_{END}$ $M^-_{uSPAN} \geq \frac{1}{4}\max\{M^+_u, M^-_u\}_{END}$
Beam Design Shear	NLD† Combinations	Modified NLD† Combinations (earthquake loads doubled) Beam Capacity Shear (V_p) with $\alpha = 1.0$ and $\varphi = 1.0$ plus V_{D+L}	NLD† Combinations Beam Capacity Shear (V_p) with $\alpha = 1.25$ and $\varphi = 1.0$ plus V_{D+L} $V_c = 0$

† Number of specified loading

Table IV-2
Design Criteria Table

For Special and Intermediate moment resisting frames (ductile frames), the shear design of the beams is based upon the probable and nominal moment capacities of the members, respectively.

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Determine Shear Force and Moment

- In the design of the beam shear reinforcement of an **Ordinary moment resisting concrete frame**, the shear forces and moments for a particular load combination at a particular beam section are obtained by factoring the associated shear forces and moments with the corresponding load combination factors.
- In the design of **Special moment resisting concrete frames** (seismic design), however, the shear force, V_u, is calculated from the probable moment capacities of each end of the beam and the gravity shear forces. The procedure for calculating the design shear force in a beam from probable moment capacity is the same as that described for a column in section "Column Design" on page 43. See also Table IV-2 for details.

  The design shear force V_u is then given by (ACI 21.4.5.1)

  $$V_u = V_p + V_{D+L} \qquad \text{(ACI 21.4.5.1)}$$

  where, V_p is the shear force obtained by applying the calculated probable ultimate moment capacities at the two ends of the beams acting in two opposite directions. Therefore, V_p is the maximum of V_{P_1} and V_{P_2}, where

  $$V_{P_1} = \frac{M_I^- + M_J^+}{L}, \text{ and}$$

  $$V_{P_2} = \frac{M_I^+ + M_J^-}{L}, \text{ where}$$

  M_I^- = Moment capacity at end I, with top steel in tension, using a steel yield stress value of αf_y and no φ factors,

  M_J^+ = Moment capacity at end J, with bottom steel in tension, using a steel yield stress value of αf_y and no φ factors,

M_I^+ = Moment capacity at end I, with bottom steel in tension, using a steel yield stress value of αf_y and no φ factors,

M_J^- = Moment capacity at end J, with top steel in tension, using a steel yield stress value of αf_y and no φ factors, and

L = Clear span of beam.

For Special moment resisting frames α is taken as 1.25 (ACI R21.3.4.1). V_{D+L} is the contribution of shear force from the load in-span distribution of gravity loads.

- For **Intermediate moment resisting frames**, the design shear force in beams is taken to be the maximum of that based on the nominal moment capacity and factored shear force. The procedure for calculating nominal moment capacity is the same as that for computing the probable moment capacity for Special moment resisting frames, except that α is taken equal to 1 rather than 1.25. The factored shear forces are based on the specified load factors except the earthquake load factors are doubled (ACI 21.8.3). The computation of the design shear force in a beam of an **Intermediate moment resisting frame**, is also the same as that for columns, which is described earlier on page 44. See also Table IV-2 for details.

Determine Concrete Shear Capacity

The allowable concrete shear capacity is given by

$$V_c = 2\sqrt{f'_c}\, b_w d . \qquad \text{(ACI 11.3.1.1)}$$

For Special moment resisting frame concrete design, V_c is set to zero if both the factored axial compressive force including the earthquake effect P_u is less than $f'_c A_g/20$ and the shear force contribution from earthquake V_E is more than half of the total maximum shear force over the length of the member V_u (i.e. $V_E \geq 0.5 V_u$) (ACI 21.3.4.2).

Determine Required Shear Reinforcement

Given V_u and V_c, the required shear reinforcement in area/unit length is calculated as

$$A_v = \frac{(V_u/\varphi - V_c)\ s}{f_{ys}\ d}, \qquad \text{(ACI 11.5.6.2)}$$

The shear force resisted by steel is limited by

$$(V_u/\varphi - V_c) \leq 8\sqrt{f'_c}\,bd\ , \qquad \text{(ACI 11.5.6.8)}$$

where, φ, the strength reduction factor, is 0.85 (ACI 9.3.2.3). The maximum of all the calculated A_v values, obtained from each load combination, is reported along with the controlling shear force and associated load combination number.

The beam shear reinforcement requirements displayed by the program are based purely upon shear strength considerations. Any minimum stirrup requirements to satisfy spacing and volumetric considerations must be investigated independently of the program by the user.

Chapter V

Design for CAN3-A23.3-M84

This chapter describes in detail the various aspects of the concrete design procedure that is used by SAP2000 when the user selects the Canadian code, **CAN3-A23.3-M84** (CSA 1984) Various notations used in this chapter are listed in Table V-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

SAP2000 provides options to design or check Ordinary, Nominal (moderate seismic risk areas), and Ductile (high seismic risk areas) moment resisting frames as required for seismic design The details of the design criteria used for the different framing systems are described in the following sections.

English as well as SI and MKS metric units can be used for input. But the code is based on Newton-Millimeter-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Newton-Millimeter-Second** units unless otherwise noted.

A_{cv}	Area of concrete used to determine shear stress, sq-mm
A_g	Gross area of concrete, sq-mm
A_s	Area of tension reinforcement, sq-mm
A_s'	Area of compression reinforcement, sq-mm
$A_{s(required)}$	Area of steel required for tension reinforcement, sq-mm
A_{st}	Total area of column longitudinal reinforcement, sq-mm
A_v	Area of shear reinforcement, sq-mm
a	Depth of compression block, mm
a_b	Depth of compression block at balanced condition, mm
b	Width of member, mm
b_f	Effective width of flange (T-Beam section), mm
b_w	Width of web (T-Beam section), mm
C_m	Coefficient, dependent upon column curvature, used to calculate moment magnification factor
c	Depth to neutral axis, mm
c_b	Depth to neutral axis at balanced conditions, mm
d	Distance from compression face to tension reinforcement, mm
d'	Concrete cover to center of reinforcing, mm
d_s	Thickness of slab (T-Beam section), mm
E_c	Modulus of elasticity of concrete, MPa
E_s	Modulus of elasticity of reinforcement, assumed as 200,000 MPa
f_c'	Specified compressive strength of concrete, MPa
f_y	Specified yield strength of flexural reinforcement, MPa
f_{ys}	Specified yield strength of shear reinforcement, MPa
h	Dimension of column, mm
I_g	Moment of inertia of gross concrete section about centroidal axis, neglecting reinforcement, mm^4
I_{se}	Moment of inertia of reinforcement about centroidal axis of member cross section, mm^4
k	Effective length factor
L	Clear unsupported length, mm

Table V-1

List of Symbols Used in the Canadian code

M_1	Smaller factored end moment in a column, N-mm
M_2	Larger factored end moment in a column, N-mm
M_c	Factored moment to be used in design, N-mm
M_{ns}	Nonsway component of factored end moment, N-mm
M_s	Sway component of factored end moment, N-mm
M_u	Factored moment at section, N-mm
M_{ux}	Factored moment at section about X-axis, N-mm
M_{uy}	Factored moment at section about Y-axis, N-mm
P_b	Axial load capacity at balanced strain conditions, N
P_c	Critical buckling strength of column, N
P_{max}	Maximum axial load strength allowed, N
P_0	Axial load capacity at zero eccentricity, N
P_u	Factored axial load at section, N
V_c	Shear resisted by concrete, N
V_{D+L}	Shear force from span loading, N
V_p	Shear force computed from probable moment capacity, N
V_u	Factored shear force at a section, N
V_s	Shear force at a section resisted by steel, N
α	Reinforcing steel overstrength factor
β_1	Factor for obtaining depth of compression block in concrete
β_d	Absolute value of the ratio of the maximum factored axial dead load moment to the maximum factored total load moment
δ_b	Moment magnification factor for nonsway moments
δ_s	Moment magnification factor for sway moments
ε_c	Strain in concrete
ε_s	Strain in reinforcing steel
φ_c	Strength reduction factor for concrete
φ_s	Strength reduction factor for steel
φ_m	Strength reduction factor for member
λ	Shear strength factor

Table V-1

List of Symbols Used in the Canadian code (continued)

Design Load Combinations

The design load combinations are the various combinations of the prescribed load cases for which the structure needs to be checked. For this code, if a structure is subjected to dead load (DL), live load (LL), wind (WL), and earthquake (EL) loads, and considering that wind and earthquake forces are reversible, then the following load combinations may have to be considered for design of concrete frames (CAN 9.2):

1.25 DL
1.25 DL + 1.50 LL (CAN 9.2)

1.25 DL ± 1.50 WL
0.85 DL ± 1.50 WL
1.25 DL + 0.7 (1.50 LL ± 1.50 WL) (CAN 9.2)

1.25 DL ± 1.50 EL
0.85 DL ± 1.50 EL
1.25 DL + 0.7 (1.50 LL ± 1.50 EL) (CAN 9.2)

These are also the default design load combinations in SAP2000 whenever the Canadian Code is used.

In generating the above default loading combinations, the importance factor is taken as 1. The user should use other appropriate loading combinations if roof live load is separately treated, other types of loads are present, or pattern live loads are to be considered.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

Strength Reduction Factors

The strength reduction factor, φ, is material dependent and is defined as

φ_c = 0.60 for concrete and (CAN 9.3.2)

φ_s = 0.85 for steel. (CAN 9.3.3)

In some special cases, a member resistance factor, φ_m, is used as an additional reduction factor in addition to φ_c and φ_s (CAN 9.3.1).

Column Design

The user may define the geometry of the reinforcing bar configuration of each concrete column section. If the area of reinforcing is provided by the user, the program checks the column capacity. However, if the area of reinforcing is not provided by the user, the program calculates the amount of reinforcing required for the column. The design procedure for the reinforced concrete columns of the structure involves the following steps:

- Generate axial force-biaxial moment interaction surfaces for all of the different concrete section types of the model. A typical biaxial interaction surface is shown in Figure III-1. When the steel is undefined, the program generates the interaction surfaces for the range of allowable reinforcement — 1 to 8 percent for ordinary moment resisting frames (CAN 10.9.1 and CAN 10.9.2) and 1 to 6 percent for ductile moment resisting frames (CAN 21.4.3.1).
- Calculate the capacity ratio or the required reinforcing area for the factored axial force and biaxial (or uniaxial) bending moments obtained from each loading combination at each station of the column. The target capacity ratio is taken as one when calculating the required reinforcing area.
- Design the column shear reinforcement.

The following three subsections describe in detail the algorithms associated with the above-mentioned steps.

Generation of Biaxial Interaction Surfaces

The column capacity interaction volume is numerically described by a series of discrete points that are generated on the three-dimensional interaction failure surface. In addition to axial compression and biaxial bending, the formulation allows for axial tension and biaxial bending considerations. A typical interaction diagram is shown in Figure III-1.

The coordinates of these points are determined by rotating a plane of linear strain in three dimensions on the section of the column. See Figure III-2. The linear strain diagram limits the maximum concrete strain, ε_c, at the extremity of the section, to 0.003 (CAN 10.2.3).

The formulation is based consistently upon the general principles of ultimate strength design (CAN 10.3), and allows for any doubly symmetric rectangular, square, or circular column section.

The stress in the steel is given by the product of the steel strain and the steel modulus of elasticity, $\varepsilon_s E_s$, and is limited to the yield stress of the steel, f_y (CAN 10.2.4). The area associated with each reinforcing bar is assumed to be placed at the actual location of the center of the bar and the algorithm does not assume any further simplifications in the manner in which the area of steel is distributed over the cross section of the column (such as an equivalent steel tube or cylinder). See Figure V-1.

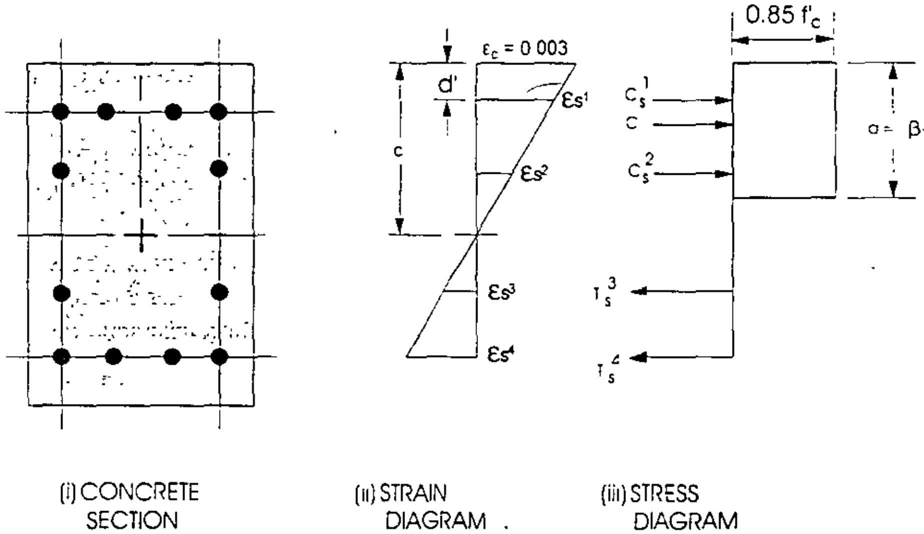


Figure V-1

Idealization of Stress and Strain Distribution in a Column Section

The concrete compression stress block is assumed to be rectangular, with a stress value of $0.85 f'_c$ (CAN 10.2.7). The interaction algorithm provides correction to account for the concrete area that is displaced by the reinforcement in the compression zone.

The effects of the strength reduction factor, φ, are included in the generation of the interaction surfaces. The maximum compressive axial load is limited to P_{max}, where the maximum factored axial load resistance is given by

$$P_{max} = 0.80[0.85\varphi_c f'_c (A_g - A_{st}) + \varphi_s f_y A_{st}] \text{(tied column), (CAN 10.3.5.3)}$$

$$P_{max} = 0.85\,[0.85\varphi_c f'_c (A_g - A_{st}) + \varphi_s f_y A_{st}] \text{(spiral column). (CAN 10.3.5.2)}$$

Check Column Capacity

The column capacity is checked for each loading combination at each check stations of each column. In checking a particular column for a particular loading combination at a particular location, the following steps are involved:

- Determine the factored moments and forces from the analysis load cases and the specified load combination factors to give P_u, M_{ux}, and M_{uy}.
- Determine the moment magnification factors for the column moments.
- Apply the moment magnification factors to the factored loads obtained in the first step. Determine whether the point, defined by the resulting axial load and biaxial moment set, lies within the interaction volume.

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Determine Factored Moments and Forces

The factored loads for a particular load combination are obtained by applying the corresponding load factors to all the load conditions, giving P_u, M_{ux}, and M_{uy}. The factored moments are further increased, if required, to obtain minimum eccentricities of $(15 + 0.03h)$ mm, where h is the dimension of the column in the corresponding direction (CAN 10.11.6.4). The computed moments are further amplified by using "Moment Magnification Factors" to allow for "Lateral Drift Effect" and "Member Stability Effect".

Determine Moment Magnification Factors

The moment magnification factors are applied in two different stages. First the moments are separated into their "sway" and "non-sway" components. The non-sway components are amplified for lateral drift effect. Although this amplification may be avoided for "braced" frames according to the code, SAP2000 treats all frames uniformly to amplify non-sway components of moments. These amplified moments are further amplified for individual member stability effect.

Lateral Drift Effect

For all frames, the moment magnification factor for lateral drift effect is applied only to the "sway" moment in SAP2000.

$$M = M_{ns} + \delta_s M_s \qquad \text{(ACI 10.11.5.2)}$$

The moment magnification factors in the major and minor directions can be different. The moment magnification factors for moments causing sidesway, δ_{sx} and δ_{sy}, can be taken as 1.0 if a P-Δ analysis is carried out (CAN 10.11.7). **The program assumes that the SAP2000 analysis models P-Δ effects, therefore, δ_{sx} and δ_{sy} are taken as 1.0.**

It is suggested that the P-Δ analysis be done at the factored load level (White and Hajjar 1991). The necessary factors for a P-Δ analysis for the CAN3-A23.2-M84 code should be (1.25 DL + 0.7*1.50 LL)/φ_m, where φ_m is the strength reduction factor for stability and is equal to 0.65.

The user is reminded of the special analysis requirements, especially those related to the value of EI used in analysis (CAN 10.11.2.2). If the program assumptions are not satisfactory for a particular member, the user can explicitly specify values of δ_{sx} and δ_{sy}.

Member Stability Effects

All compression members are designed using the factored axial load, P_u, from the analysis and a magnified factored moment, M_c. The magnified moment is computed as,

$$M_c = \delta_b M_2 \, , \qquad \text{(CAN 10.11.6)}$$

where M_2 is the column maximum end moment obtained from elastic analysis after considering minimum eccentricity and lateral drift effect, and M_c is the maximum moment occurring either at the end or at an interior point within the span of the column. The moment magnification factor, δ_b, for moments not causing sidesway associated with the major or minor direction of the column is given by

$$\delta_b = \frac{C_m}{1 - \dfrac{P_u}{\varphi_m P_c}} \geq 1.0 \text{, where} \qquad \text{(CAN 10.11.6.1)}$$

$$\varphi_m = 0.65 \, ,$$

$$P_c = \frac{\pi^2 EI}{(kL)^2} \, ,$$

k is conservatively taken as 1, however the user can override the value.

EI is associated with a particular column direction given by the larger of:

$$EI = \frac{0.2\, E_c\, I_g + E_s\, I_{sc}}{1 + \beta_d} \quad \text{or} \quad EI = 0.25\, E_c\, I_g \text{, and} \qquad \text{(CAN 10.11.6.2)}$$

$$\beta_d = \frac{\text{Maximum factored dead load moment}}{\text{Maximum factored total load moment}}, \qquad \text{(CAN 10.0)}$$

$$C_m = 0.6 + 0.4 \frac{M_a}{M_b} \geq 0.4, \qquad \text{(CAN 10.11.6.3)}$$

M_a and M_b are the moments at the ends of the column, and M_b is numerically larger than M_a. M_a / M_b is positive for single curvature bending and negative for double curvature bending. The above expression of C_m is valid if there is no transverse load applied between the supports. If transverse load is present on the span, or the length is overwritten, or for any other case, $C_m = 1$. C_m can be overwritten by the user on an element by element basis.

The magnification factor, δ_b, must be a positive number and greater than one. Therefore P_u must be less than $\varphi_m P_c$. If P_u is found to be greater than or equal to $\varphi_m P_c$, a failure condition is declared.

The above calculations use the unsupported length of the column. The two unsupported lengths are l_{22} and l_{33} corresponding to instability in the minor and major directions of the element, respectively. See Figure III-4. These are the lengths between the support points of the element in the corresponding directions.

Determine Capacity Ratio

As a measure of the stress condition of the column, a capacity ratio is calculated. The capacity ratio is basically a factor that gives an indication of the stress condition of the column with respect to the capacity of the column.

Before entering the interaction diagram to check the column capacity, the moment magnification factors are applied to the factored loads to obtain P_u, M_{ux}, and M_{uy}. The point (P_u, M_{ux}, M_{uy}) is then placed in the interaction space shown as point L in Figure III-3. If the point lies within the interaction volume, the column capacity is adequate; however, if the point lies outside the interaction volume, the column is overstressed.

This capacity ratio is achieved by plotting the point L and determining the location of point C. The point C is defined as the point where the line OL (if extended outwards) will intersect the failure surface. This point is determined by three-dimensional linear interpolation between the points that define the failure surface. See Figure III-3. The capacity ratio, CR, is given by the ratio $\frac{OL}{OC}$.

- If OL = OC (or CR=1) the point lies on the interaction surface and the column is stressed to capacity.
- If OL < OC (or CR<1) the point lies within the interaction volume and the column capacity is adequate.
- If OL > OC (or CR>1) the point lies outside the interaction volume and the column is overstressed.

The maximum of all the values of CR calculated from each load combination is reported for each check station of the column along with the controlling P_u, M_{ux}, and M_{uy} set and associated load combination number.

If the reinforcing area is not defined, SAP2000 computes the reinforcement that will give an interaction ratio of unity.

Design Column Shear Reinforcement

The shear reinforcement is designed for each loading combination in the major and minor directions of the column. In designing the shear reinforcing for a particular column for a particular loading combination due to shear forces in a particular direction, the following steps are involved:

- Determine the factored forces acting on the section, P_u and V_u. Note that P_u is needed for the calculation of V_c.
- Determine the shear force, V_c, that can be resisted by concrete alone.
- Calculate the reinforcement steel required to carry the balance.

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Determine Section Forces

- In the design of the column shear reinforcement of an **Ordinary moment resisting concrete frame**, the forces for a particular load combination, namely, the column axial force, P_u, and the column shear force, V_u, in a particular direction are obtained by factoring the SAP2000 analysis load cases with the corresponding load combination factors.

- In the shear design of **Ductile moment resisting frames** (seismic design) the following are checked in addition to the requirement for the ordinary moment resisting frames. In the design of Ductile moment resisting concrete frames, the design shear force, V_u, in a particular direction is also calculated from the probable moment capacities of the column associated with the factored axial force acting on the column.

  For each load combination, the factored axial load, P_u, is calculated. Then, the positive and negative moment capacities, M_u^+ and M_u^-, of the column in a particular direction under the influence of the axial force P_u is calculated using the uniaxial interaction diagram in the corresponding direction. The design shear force, V_u, is then given by

  $$V_p + V_{D+L} \qquad \text{(CAN 21.7.2.1)}$$

  where, V_p is the shear force obtained by applying the calculated probable ultimate moment capacities at the two ends of the column acting in two opposite directions. Therefore, V_p is the maximum of V_{P_1} and V_{P_2}, where

  $$V_{P_1} = \frac{M_I^- + M_J^+}{L}, \text{ and}$$

  $$V_{P_2} = \frac{M_I^+ + M_J^-}{L}, \text{ where}$$

  M_I^+, M_I^- = Positive and negative moment capacities at end I of the column using a steel yield stress value of αf_y and no φ factors,

  M_J^+, M_J^- = Positive and negative moment capacities at end J of the column using a steel yield stress value of αf_y and no φ factors, and

  L = Clear span of column.

  For Ductile moment resisting frames α is taken as 1.25 (CAN 21.1). V_{D+L} is the contribution of shear force from the in-span distribution of gravity loads. For most of the columns, it is zero.

- For **Nominal moment resisting frames** (seismic), the design shear force is taken to be the maximum of that based on the nominal moment capacity and factored shear force. The procedure for calculating nominal moment capacity is the same as that for computing the probable moment capacity for Ductile moment resisting frames, except that α is taken equal to 1 rather than 1.25 (CAN 21.1). The factored shear forces are based on the specified load factors.

Determine Concrete Shear Capacity

Given the design force set P_u and V_u, the shear force carried by the concrete, V_c, is calculated as follows:

- If the column is subjected to flexure and shear only, i.e. P_u is zero,

  $$V_c = 0.2\,\varphi_c\,\lambda\,\sqrt{f'_c}\;b_w\,d \qquad \text{(CAN 11.3.4.1)}$$

  where λ is taken as 1 for normal weight concrete For other types of sections $b_w\,d$ is replaced by A_{cv}, the effective shear area which is shown in Figure V-2

- If the column is subjected to axial tension, i.e. P_u is negative,

  $$V_c = 0.2\varphi_c\lambda\sqrt{f'_c}\left(1 + \frac{P_u}{0.6\,\varphi_c\,\lambda\sqrt{f'_c}\,A_g}\right)b_w\,d\,. \qquad \text{(CAN 11.3.4.2)}$$

- If the column is subjected to axial compression, i.e. P_u is positive,

  $$V_c = 0.2\,\varphi_c\,\lambda\sqrt{f'_c}\left(1 + 3\frac{P_u}{A_g\,f'_c}\right)b_w\,d\,. \qquad \text{(CAN 11.3.4.3)}$$

- For Ductile moment resisting frames, the shear strength at any beam or column section is taken as zero if axial force is tensile or compression is very small. This is given as

  $$V_c = 0 \;\text{ if }\; P_u \leq 0.10 f'_c\,A_g\,. \qquad \text{(CAN 21.7.3.1 and CAN 21.3.1)}$$

Determine Required Shear Reinforcement

Given V_u and V_c, the required shear reinforcement for a spacing s is given by

$$A_v = \frac{(V_u - V_c)s}{\varphi_s f_{ys}\,d}\,. \qquad \text{(CAN 11.3.6.1)}$$

The shear resistance due to the reinforcement, V_s, is limited by

$$V_s \leq 0.8\varphi_c\lambda\sqrt{f'_c}\;b_w\,d\,. \qquad \text{(CAN 11.3.6.6)}$$

The maximum of all the calculated A_v values obtained from each load combination are reported for the major and minor directions of the column along with the controlling shear force and associated load combination label.

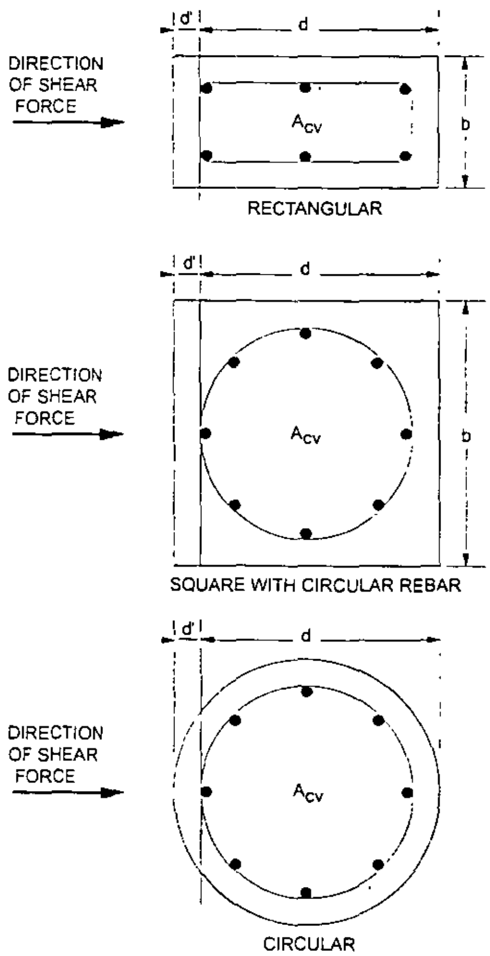


Figure V-2
Shear Stress Area, A_{cv}

The column shear reinforcement requirements reported by the program are based purely upon shear strength consideration. Any minimum stirrup requirements to satisfy spacing and/or volumetric requirements must be investigated independently of the program by the user.

Beam Design

In the design of concrete beams, SAP2000 calculates and reports the required areas of steel for flexure and shear based upon the beam moments, and shears, load combination factors and other criteria described below. The reinforcement requirements are calculated at a user defined number of check stations along the beam span.

All the beams are only designed for major direction flexure and shear. Effects due to any axial forces, minor direction bending, and torsion that may exist in the beams must be investigated independently by the user.

The beam design procedure involves the following steps:

- Design beam flexural reinforcement
- Design beam shear reinforcement

Design Beam Flexural Reinforcement

The beam top and bottom flexural steel is designed at a user defined number of design stations along the beam span. In designing the flexural reinforcement for the major moment of a particular beam for a particular section, the following steps are involved:

- Determine the maximum factored moments
- Determine the reinforcing steel

Determine Factored Moments

In the design of flexural reinforcement of ductile, nominal, or ordinary moment resisting concrete frame beams, the factored moments for each load combination at a particular beam station are obtained by factoring the corresponding moments for different load cases with the corresponding load factors.

The beam section is then designed for the maximum positive M_u^+ and maximum negative M_u^- factored moments obtained from all of the load combinations

Negative beam moments produce top steel. In such cases the beam is always designed as a rectangular section. Positive beam moments produce bottom steel. In such cases the beam may be designed as a Rectangular- or T-beam.

Determine Required Flexural Reinforcement

In the flexural reinforcement design process, the program calculates both the tension and compression reinforcement. Compression reinforcement is added when the applied design moment exceeds the maximum moment capacity of a singly reinforced section. The user has the option of avoiding the compression reinforcement by increasing the effective depth, the width, or the grade of concrete.

The design procedure is based on the simplified rectangular stress block as shown in Figure V-3 (CAN 10.2). Furthermore it is assumed that the compression carried by concrete is less than that which can be carried at the balanced condition (CAN 10.3.2). When the applied moment exceeds the moment capacity at the balanced condition, the area of compression reinforcement is calculated on the assumption that the additional moment will be carried by compression and additional tension reinforcement.

The design procedure used by SAP2000, for both rectangular and flanged sections (L- and T-beams) is summarized below. It is assumed that the design ultimate axial force does not exceed $0.15 f'_c A_g$ (CAN 10.3.3), hence all the beams are designed for major direction flexure and shear only.

Design for Flexure of a Rectangular Beam

In designing for a factored negative or positive moment, M_u, (i.e. designing top or bottom steel) the depth of the compression block is given by a, where,

$$a = d - \sqrt{d^2 - \frac{2\left|M_u\right|}{0.85 f'_c \varphi_c b}},$$

where the value of φ_c is 0.60 (CAN 9.3.2) in the above and following equations. See Figure V-3. Also β_1 and c_b are calculated as follows:

$$\beta_1 = 0.85 - \frac{0.08(f'_c - 30)}{10}, \quad 0.65 \le \beta_1 \le 0.85, \text{ and} \qquad \text{(CAN 10.2.7)}$$

$$c_b = \frac{600}{600 + f_y} d. \qquad \text{(CAN 10.3.3)}$$

- If $a \le \beta_1 c_b$, the area of tensile steel reinforcement is then given by

$$A_s = \frac{M_u}{\varphi_s f_y \left(d - \frac{a}{2}\right)}$$

This steel is to be placed at the bottom if M_u is positive, or at the top if M_u is negative.

- If $a > \beta_1 c_b$, compression reinforcement is required (CAN 10.3.2) and is calculated as follows:

  - The balanced depth of compression block is given by

    $a_b = \beta_1 c_b$. (CAN 10.3.2)

  - The compressive force developed in concrete alone is given by

    $C = 0.85 f'_c b a_b$, and (CAN 10.2.7)

    the moment resisted by concrete and bottom steel is

    $$M_{uc} = C\left(d - \frac{a_b}{2}\right)\varphi_c \, .$$

  - The moment resisted by compression steel and tensile steel is

    $M_{us} = M_u - M_{uc}$.

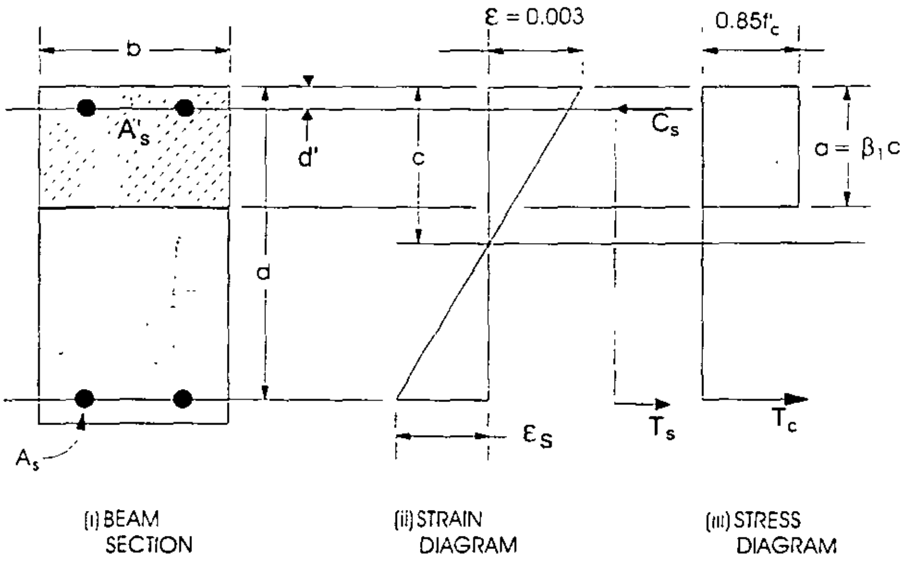


Figure V-3
Design of a Rectangular Beam Section

- So the required compression steel is given by

$$A_s' = \frac{M_{us}}{f_s'(d - d')\varphi_s}, \text{ where}$$

$$f_s' = 0.003\, E_s \left[\frac{c - d'}{c}\right]. \qquad \text{(ACI 10.2.4)}$$

- The required tensile steel for the balancing the compression in concrete is

$$A_{s1} = \frac{M_{uc}}{f_y(d - \frac{a_b}{2})\varphi_s}, \text{ and}$$

the tensile steel for balancing the compression in steel is

$$A_{s2} = \frac{M_{us}}{f_y(d - d')\varphi_s}.$$

- Therefore, the total tensile reinforcement, $A_s = A_{s1} + A_{s2}$, and total compression reinforcement is A_s'. A_s is to be placed at bottom and A_s' is to be placed at top if M_u is positive, and vice versa.

Design for Flexure of a T-Beam

In designing for a factored negative moment, M_u, (i.e. designing top steel), the calculation of the steel area is exactly the same as above, i.e., no T-Beam data is to be used. If $M_u > 0$, the depth of the compression block is given by (see Figure V-4).

$$a = d - \sqrt{d^2 - \frac{2\, M_u}{0.85 f_c' \varphi_c\, b_f}}.$$

The depth of compression block under balanced condition is given by

$$a_b = \beta_1 c_b\,.$$

- If $a \leq d_s$, the subsequent calculations for A_s are exactly the same as previously done for the rectangular section design. However, in this case the width of the compression flange is taken as the width of the beam for analysis. Whether compression reinforcement is required depends on whether $a > a_b$.
- If $a > d_s$, calculation for A_s is done in two parts. The first part is for balancing the compressive force from the flange, C_f, and the second part is for balancing the compressive force from the web, C_w. As shown in Figure V-4,

$$C_f = 0.85 f'_c (b_f - b_w) d_s .$$

Therefore, $A_{s1} = \dfrac{C_f \varphi_c}{f_y \varphi_s}$ and the portion of M_u that is resisted by the flange is given by

$$M_{uf} = C_f \left(d - \frac{d_s}{2} \right) \varphi_c .$$

Therefore, the balance of the moment, M_u to be carried by the web is given by

$$M_{uw} = M_u - M_{uf} .$$

The web is a rectangular section of dimensions b_w and d, for which the depth of the compression block is recalculated as

$$a_1 = d - \sqrt{d^2 - \frac{2 M_{uw}}{0.85 f'_c \varphi_c b_w}} .$$

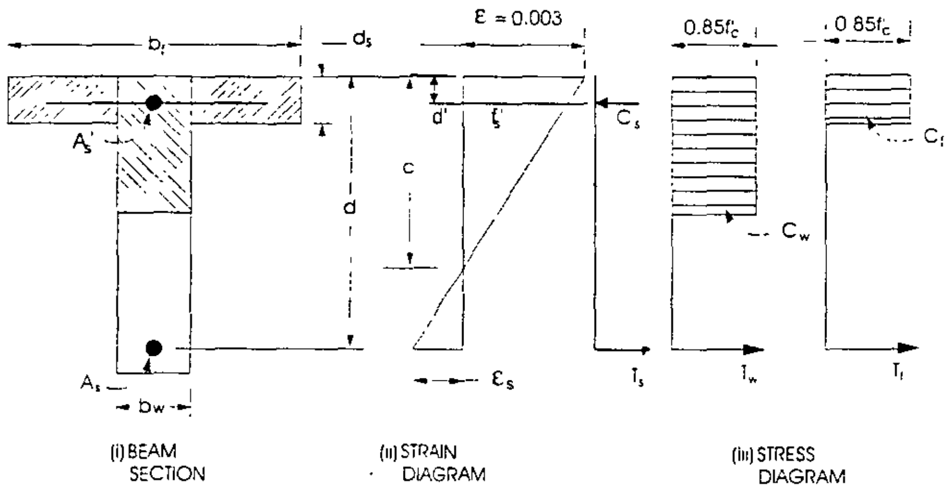


Figure V-4
Design of a T-Beam Section

- If $a_1 \le a_b$, the area of tensile steel reinforcement is then given by

  $$A_{s2} = \frac{M_{um}}{\varphi_s f_y \left(d - \frac{a_1}{2} \right)}, \text{ and}$$

  $$A_s = A_{s1} + A_{s2}.$$

  This steel is to be placed at the bottom of the T-beam.

- If $a_1 > a_b$, compression reinforcement is required (CAN 10.3.2) and is calculated as follows:

  – The compressive force in concrete web alone is given by

  $$C = 0.85 f'_c b a_b, \text{ and} \qquad \text{(CAN 10.2.7)}$$

  the moment resisted by concrete web and tensile steel is

  $$M_{uc} = C\left(d - \frac{a_b}{2} \right)\varphi_c.$$

  – The moment resisted by compression steel and tensile steel is

  $$M_{us} = M_{um} - M_{uc}.$$

  – Therefore, the compression steel is computed as

  $$A'_s = \frac{M_{us}}{f'_s (d - d')\varphi_s}, \text{ where}$$

  $$f'_s = 0.003\, E_s \left[\frac{c - d'}{c} \right]. \qquad \text{(CAN 10.2.4)}$$

  – The tensile steel for balancing compression in web concrete is

  $$A_{s2} = \frac{M_{uc}}{f_y (d - \frac{a_b}{2})\varphi_s}, \text{ and}$$

  the tensile steel for balancing compression in steel is

  $$A_{s3} = \frac{M_{us}}{f_y (d - d')\varphi_s}.$$

– Total tensile reinforcement, $A_s = A_{s1} + A_{s2} + A_{s3}$, and total compression reinforcement is A_s'. A_s is to be placed at bottom and A_s' is to be placed at top.

Minimum Tensile Reinforcement

The minimum flexural tensile steel provided in a rectangular section in an ordinary moment resisting frame is given by the minimum of the two limits

$$A_s \geq \frac{1.4}{f_y} b_w d, \text{ or} \qquad \text{(CAN 10.5.1)}$$

$$A_s \geq \frac{4}{3} A_{s(required)}. \qquad \text{(CAN 10.5.2)}$$

Special Consideration for Seismic Design

For Ductile moment resisting concrete frames (seismic design), the beam design should satisfy the following additional conditions (see also Table V-2 for comprehensive listing):

- The minimum longitudinal reinforcement shall be provided at both at the top and bottom. Any of the top and bottom reinforcement shall not be less than $A_{s(min)}$.

$$A_{s(min)} \geq \frac{1.4}{f_y} b_w d \qquad \text{(CAN 21.3.2.1)}$$

- The beam flexural steel is limited to a maximum given by

$$A_s \leq 0.025\, b_w d. \qquad \text{(CAN 21.3.2.1)}$$

- At any end (support) of the beam, the beam positive moment capacity (i.e. associated with the bottom steel) would not be less than 1/2 of the beam negative moment capacity (i.e. associated with the top steel) at that end (CAN 21.3.2.2).
- Neither the negative moment capacity nor the positive moment capacity at any of the sections within the beam would be less than 1/4 of the maximum of positive or negative moment capacities of any of the beam end (support) stations (CAN 21.3.2.2).

For Nominal moment resisting concrete frames (seismic design), the beam design would satisfy the following conditions:

- At any support of the beam, the beam positive moment capacity would not be less than 1/3 of the beam negative moment capacity at that end (CAN 21.9.2.1).
- Neither the negative moment capacity nor the positive moment capacity at any of the sections within the beam would be less than 1/5 of the maximum of positive or negative moment capacities of any of the beam end (support) stations (CAN 21.9.2.1).

Design Beam Shear Reinforcement

The shear reinforcement is designed for each load combination at user defined number of stations along the beam span. In designing the shear reinforcement for a particular beam for a particular loading combination at a particular station due to the beam major shear, the following steps are involved:

- Determine the factored shear force, V_u.
- Determine the shear force, V_c, that can be resisted by the concrete.
- Determine the reinforcement steel required to carry the balance.

For Ductile and Intermediate moment resisting frames, the shear design of the beams is based on the probable and nominal moment capacities of the members, respectively.

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Determine Shear Force and Moment

- In the design of the beam shear reinforcement of an **Ordinary moment resisting concrete frame**, the shear forces and moments for a particular load combination at a particular beam section are obtained by factoring the associated shear forces and moments with the corresponding load combination factors.
- In the design of **Ductile moment resisting concrete frames** (seismic design), however, the shear force, V_u, is calculated from the probable moment capacities of each end of the beam, and the gravity shear forces. The procedure for calculating the design shear force in a beam from probable moment capacity is the same as that described for a column in section "Column Design" on page 69. See also Table V-2 for more details.

  The design shear force V_u is then given by

  $$V_u = V_p + V_{D-L} \qquad \text{(CAN 21.7.2.1)}$$

where, V_p is the shear force obtained by applying the calculated probable ultimate moment capacities at the two ends of the beams acting in two opposite directions. Therefore, V_p is the maximum of V_{P_1} and V_{P_2}, where

$$V_{P_1} = \frac{M_I^- + M_J^+}{L}, \text{ and}$$

$$V_{P_2} = \frac{M_I^+ + M_J^-}{L}, \text{ where}$$

M_I^- = Moment capacity at end I, with top steel in tension, using a steel yield stress value of αf_y and no φ factors,

M_J^+ = Moment capacity at end J, with bottom steel in tension, using a steel yield stress value of αf_y and no φ factors,

M_I^+ = Moment capacity at end I, with bottom steel in tension, using a steel yield stress value of αf_y and no φ factors,

M_J^- = Moment capacity at end J, with top steel in tension, using a steel yield stress value of αf_y and no φ factors, and

L = Clear span of beam.

The overstrength factor α is always taken as 1.25 for Ductile moment resisting frames (CAN 21.1). V_{D+L} is the contribution of shear force from the load in-span distribution of gravity loads.

- In the design of **Nominal moment resisting frames** (seismic), the design shear force in beams is taken to be the maximum of that based on the nominal moment capacity and factored shear force. The procedure for calculating nominal moment capacity is the same as that for computing the probable moment capacity for Ductile moment resisting frames, except that α is taken equal to 1 rather than 1.25 (CAN 21.1). The factored shear forces are based on the specified load factors. The computation of the design shear force in a beam of an **Intermediate moment resisting frame**, is also the same as that for columns, which is described earlier in section "Column Design" on page 70. The value of V_{D+L} is based on the span load. See also Table V-2 for details.

Type of Check/ Design	Ordinary Moment Resisting Frames (non-Seismic)	Nominal Moment Resisting Frames (Seismic)	Ductile Moment Resisting Frames (Seismic)
Column Check (interaction)	NLD† Combinations	NLD† Combinations	NLD† Combinations
Column Design (Interaction)	NLD† Combinations $1\% < \rho < 8\%$	NLD† Combinations $1\% < \rho < 8\%$	NLD† Combinations $\alpha = 1.0$ $1\% < \rho < 6\%$
Column Shears	NLD† Combinations	Modified NLD† Combinations Column Capacity Shear (V_p) $\varphi = 1.0$ and $\alpha = 1.0$	NLD† Combinations and Column Capacity Shear (V_p) $\varphi = 1.0$ and $\alpha = 1.25$
Beam Design Flexure	NLD† Combinations	NLD† Combinations	NLD† Combinations $\rho \le 0.025$ $\rho \ge \frac{1.4}{f_y}$
Beam Min. Moment Override Check	No Requirement	$M^{+}_{uEND} \ge \frac{1}{3} M^{-}_{uEND}$ $M^{+}_{uSPAN} \ge \frac{1}{5}\max\{M^{+}_{u}, M^{-}_{u}\}_{END}$ $M^{-}_{uSPAN} \ge \frac{1}{5}\max\{M^{+}_{u}, M^{-}_{u}\}_{END}$	$M^{+}_{uEND} \ge \frac{1}{2} M^{-}_{uEND}$ $M^{+}_{uSPAN} \ge \frac{1}{4}\max\{M^{+}_{u}, M^{-}_{u}\}_{END}$ $M^{-}_{uSPAN} \ge \frac{1}{4}\max\{M^{+}_{u}, M^{-}_{u}\}_{END}$
Beam Design Shear	NLD† Combinations	NLD† Combinations Beam Capacity Shear (V_p) with $\alpha = 1.0$ and $\varphi = 1.0$ plus V_{D+L}	NLD† Combinations Beam Capacity Shear (V_p) with $\alpha = 1.25$ and $\varphi = 1.0$ plus V_{D+L} $V_c = 0$

† Number of specified loading

Table V-2

Comparison of Ordinary, Ductile, and Nominal Moment Resisting Frame Design

Determine Concrete Shear Capacity

The allowable shear capacity for ordinary and nominal moment resisting frames is given by

$$V_c = 0.2\varphi_c \lambda \sqrt{f'_c}\, b_w d \qquad \text{(CAN 11.3.4.1)}$$

V_c is taken as zero for ductile moment resisting beams (CAN 21.7.3.1).

Determine Required Shear Reinforcement

Given V_u and V_c, the required shear reinforcement in area/unit length is calculated as

$$A_v = \frac{(V_u - V_c)s}{\varphi_s f_{ys}\, d}. \qquad \text{(CAN 11.3.6.1)}$$

The shear force to be resisted by steel, V_s, is limited by

$$V_s < 0.8\, \varphi_c\, \lambda \sqrt{f'_c}\, b_w d\ . \qquad \text{(CAN 11.3.6 6)}$$

The beam shear reinforcement requirements displayed by the program are based purely upon shear strength considerations. Any minimum stirrup requirements to satisfy spacing and volumetric requirements must be investigated independently of the program by the user.

Chapter VI

Design for BS 8110-85

This chapter describes in detail the various aspects of the concrete design procedure that is used by SAP2000 when the user selects the British limit state design code **BS 8110** (BSI 1985) Various notations used in this chapter are listed in Table VI-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

English as well as SI and MKS metric units can be used for input. But the code is based on Newton-Millimeter-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Newton-Millimeter-Second** units unless otherwise noted.

Design Load Combinations

The design loading combinations define the various factored combinations of the load cases for which the structure is to be checked. The design loading combinations are obtained by multiplying the characteristic loads by appropriate partial factors of safety, γ_f (BS 2.4.1.3). If a structure is subjected to dead load (DL) and live load (LL) only, the design will need only one loading combination, namely 1.4 DL + 1.6 LL. However, in addition to the dead load and live load, if the structure is subjected to wind (WL) and/or earthquake (EL) loads, and considering that those loads

A_{cv}	Area of section for shear resistance, mm^2
A_s	Area of tension reinforcement, mm^2
A_s'	Area of compression reinforcement, mm^2
A_{sc}	Total area of column longitudinal reinforcement, mm^2
A_{sv}	Total cross-sectional area of links at the neutral axis, mm^2
a	Depth of compression block, mm
b	Width or effective width of the section in the compression zone, mm
b'	Shorter section dimension, mm
	Shorter effective depth of biaxially bent column, mm
b_f	Width or effective width of flange, mm
b_w	Average web width of a flanged beam, mm
C	Compression force, N
d	Effective depth of tension reinforcement, mm
d'	Depth to center of compression reinforcement, mm
E_c	Modulus of elasticity of concrete, MPa
E_s	Modulus of elasticity of reinforcement, assumed as 200,000 MPa
e_{min}	Minimum or nominal eccentricity, mm
f_{cu}	Characteristic cube strength at 28 days, MPa
f_s'	Compressive stress in a beam compression steel, MPa
f_y	Characteristic strength of reinforcement, MPa
f_{yv}	Characteristic strength of link reinforcement, MPa (< 460 MPa)
h	Overall depth of a section in the plane of bending, mm
h_f	Flange thickness, mm
K'	Maximum $\dfrac{M_u}{bd^2 f_{cu}}$ for a singly reinforced concrete section taken as 0.156
	by assuming that moment redistribution is limited to 10%
k_1	Shear strength enhancement factor
k_2	Concrete shear strength factor, $[f_{cu}/25]^{1/3}$
l_e	Effective height of a column, mm
l_0	Clear height between end restraints, mm

Table VI-1

List of Symbols Used in the BS code

M	Design moment at a section, MPa
M_1, M_2	Smaller and larger end moments in a slender column, N-mm
M_{add}	Maximum additional moment column, N-mm
M_i	Initial moment at the point of maximum additional moment, N-mm
M_x, M_y	Applied moments about the major and minor axes of a column, N-mm
N	Ultimate axial load, N
s_v	Spacing of links, mm
T	Tension force, N
V	Shear force at ultimate design load, N
v	Shear stress, MPa
v_c	Design ultimate shear stress resistance of a concrete beam, MPa
v_c'	Design concrete shear stress corrected for axial forces, MPa
v_x, v_y	Design ultimate shear stress of a concrete section, MPa
x	Neutral axis depth, mm
x_{bal}	Depth of neutral axis in a balanced section, mm
z	Lever arm, mm
β	Effective length factor
β_b	Moment redistribution factor in a member
γ_f	Partial safety factor for load
γ_m	Partial safety factor for material strength
ε_c	Concrete strain
ε_s	Strain in tension steel
ε'_s	Strain in compression steel

Table VI-1

List of Symbols Used in the BS code (continued) are subject to reversals, the following load combinations for ultimate limit state might have to be considered (BS 2.4.3):

1.4 DL
1.4 DL + 1.6 LL (BS 2.4.3)

1.0 DL ± 1.4 WL
1.4 DL ± 1.4 WL
1.2 DL + 1.2 LL ± 1.2 WL (BS 2.4.3)

1.0 DL ± 1.4 EL
1.4 DL ± 1.4 EL
1.2 DL + 1.2 LL ± 1.2 EL

These are the default load combinations. In addition to the above load combinations, the code requires that all buildings should be capable of resisting a notional design ultimate horizontal load applied at each floor or roof level. The notional load should be equal to 0.015 times the dead load (BS 3.1.4.2). It is recommended that the user define additional load cases for considering the notional load in SAP2000.

Live load reduction factors, as allowed by some design codes, can be applied to the member forces of the live load case on a member-by-member basis to reduce the contribution of the live load to the factored loading.

Design Strength

The design strength for concrete and steel are obtained by dividing the characteristic strength of the material by a partial factor of safety, γ_m. The values of γ_m used in the program are listed below (BS 2.4.4.1).

Values of γ_m for the ultimate limit state	
Reinforcement	1.15
Concrete in flexure and axial load	1.50
Shear strength without shear reinforcement	1.25

Column Design

The user may define the geometry of the reinforcing bar configuration of each concrete column section. If the area of reinforcing is provided by the user, the program checks the column capacity. However, if the area of reinforcing is not provided by the user, the program calculates the amount of reinforcing required for the column. The design procedure for the reinforced concrete columns of the structure involves the following steps:

- Generate axial force-biaxial moment interaction surfaces for all of the different concrete section types of the model. A typical biaxial interaction surface is shown in Figure III-1. When the steel is undefined, the program generates the interaction surfaces for the range of allowable reinforcement from 1 to 8 percent (BS 3.12.6.2).
- Calculate the capacity ratio or the required reinforcing area for the factored axial force and biaxial (or uniaxial) bending moments obtained from each loading combination at each station of the column. The target capacity ratio is taken as one when calculating the required reinforcing area.
- Design the column shear reinforcement.

The following three subsections describe in detail the algorithms associated with the above-mentioned steps.

Generation of Biaxial Interaction Surfaces

The column capacity interaction volume is numerically described by a series of discrete points that are generated on the three-dimensional interaction failure surface. In addition to axial compression and biaxial bending, the formulation allows for axial tension and biaxial bending considerations (BS 3.8.4.1). A typical interaction diagram is shown in Figure III-1.

The coordinates of these points are determined by rotating a plane of linear strain in three dimensions on the section of the column (BS 3.4.4.1). See Figure III-2. The linear strain diagram limits the maximum concrete strain, ε_c, at the extremity of the section, to 0.0035 (BS 3.4.4.1).

The formulation is based consistently upon the basic principles of ultimate strength design and allows for any doubly symmetric rectangular, square, or circular column section (BS 3.8.4).

The stress in the steel is given by the product of the steel strain and the steel modulus of elasticity, $\varepsilon_s E_s$ and is limited to the design strength the steel, $f_y/1.15$ ($0.87 f_y$). The area associated with each reinforcing bar is placed at the actual location of the center of the bar and the algorithm does not assume any simplifications in the manner in which the area of steel is distributed over the cross section of the column (such as an equivalent steel tube or cylinder). See Figure VI-1.

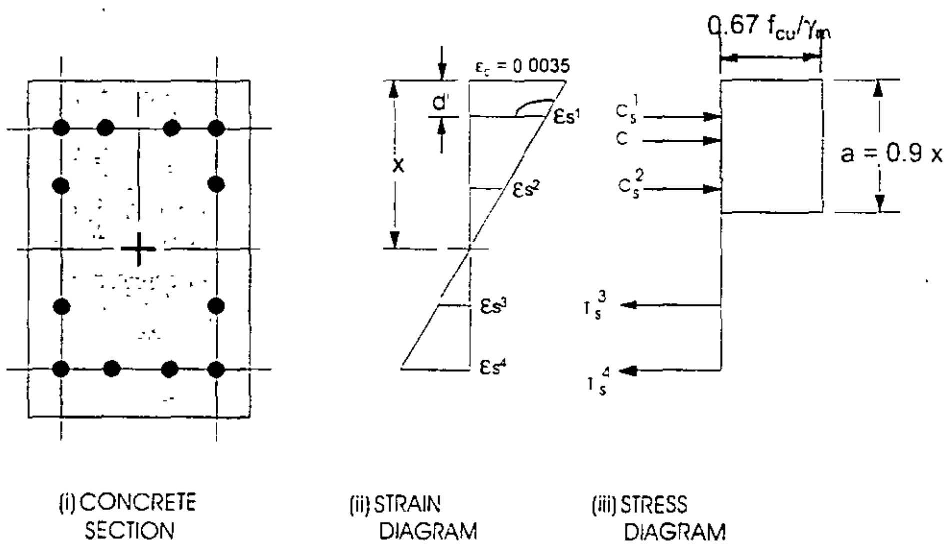


Figure VI-1
Idealized Stress and Strain Distribution in a Column Section

The concrete compression stress block is assumed to be rectangular, with a stress value of $0.67 f_{cu}/\gamma_m = 0.45 f_{cu}$ (BS 3.4.4.1). See Figure VI-1. The interaction algorithm provides corrections to account for the concrete area that is displaced by the reinforcement in the compression zone.

Check Column Capacity

The column capacity is checked for each loading combination at each output station of each column. In checking a particular column for a particular loading combination at a particular location, the following steps are involved:

- Determine the factored moments and forces from the analysis load cases and the specified load combination factors to give N, V_x, V_y, M_x, and M_y.
- Determine the additional moments due to slender column effect. Compute moments due to minimum eccentricity.
- Determine total design moments by adding the corresponding additional moments to the factored moments obtained from the analysis. Determine whether the point, defined by the resulting axial load and biaxial moment set, lies within the interaction volume.

The following three subsections describe in detail the algorithms associated with the above-mentioned steps.

Determine Factored Moments and Forces

Each load combination is defined with a set of load factors corresponding to the load cases. The factored loads for a particular load combination are obtained by applying the corresponding load factors to the load cases, giving N, V_x, V_y, M_x and M_y.

Determine Additional Moments

The determination of additional moments depends on whether the frame is "braced" or "unbraced" against side-sway (BS 3.8.1.5). For "unbraced" columns additional moment is automatically considered in the P-Δ analysis. But for "braced" columns, further calculation is required for stability of individual column members.

Braced Column

The additional moment in a braced column in a particular plane is the product of the axial load and the lateral deflection of the column in that plane (BS 3.8.3),

$$M_{add} = N\,a_u\,, \qquad \text{(BS 3.8.3.1)}$$

where, a_u is the deflection at the ultimate limit state which is obtained from

$$a_u = \beta_c K h \quad \text{and} \qquad \text{(BS 3.8.3.1)}$$

$$\beta_c = \frac{1}{2000}\left(\frac{l_e}{b'}\right)^2. \qquad \text{(BS 3.8.3.1)}$$

In the above equations,

- l_e is the effective length in the plane under consideration. It is obtained from

  $$l_e = \beta \, l_0 \,, \qquad \text{(BS 3.8.1.6.1)}$$

  where β is the effective length factor, and l_0 the unsupported length corresponding to instability in the major or minor direction of the element, l_x or l_y in Figure III-4. In calculating the value of the effective length, the β factor is conservatively taken as 1. However, SAP2000 allows the user to override this default value.
- b' is the dimension of the column in the plane of bending considered,
- h is also the dimension of the column in the plane of bending considered, and
- K is the correction factor to the deflection to take care of the influence of the axial force and K is conservatively taken as 1.

SAP2000 then calculates the total design moments by combining the factored moments obtained from analysis and the additional moments. If M_1 and M_2 ($M_2 > M_1$) are the initial end moments in a column member in a particular plane, then the maximum design moment for the column is taken as the greatest of the following:

$$M_2 \qquad \text{(BS 3.8.3.2)}$$

$$M_1 + M_{add} \qquad \text{(BS 3.8.3.2)}$$

$$M_i + M_{add}/2 \qquad \text{(BS 3.8.3.2)}$$

$$N \, e_{min} \qquad \text{(BS 3.8.3.2)}$$

where,

M_i is the initial moment in a column due to design ultimate loads at the point of maximum additional moment and is given by

$$M_i = 0.4 \, M_1 + 0.6 \, M_2 \quad \geq 0.4 \, M_2 \, . \qquad \text{(BS 3.8.3.2)}$$

M_1 and M_2 are the smaller and the larger end moments respectively. Both moments are assumed to be positive if the column is in single curvature. If the column is in double curvature, M_1 is assumed to be negative.

e_{min} is the minimum eccentricity which is taken as 0.05 times the overall dimension of the column in the plane of bending considered but not more than 20 mm (BS 3.8.2.4).

$$e_{min} = \frac{h}{20} \leq 20 \text{ mm} \qquad \text{(BS 3.8.2.4)}$$

Unbraced Column

In the case of the unbraced column, it is assumed that the SAP2000 analysis includes P-Δ effects so that the analysis results include the effects of the additional moments. Therefore, no additional computation is required. That means, moment magnification factors for moments causing sidesway are taken as unity. However, it is recommended that for P-Δ analysis a factor be used to obtain a P equivalent to 1.2 DL + 1.2 LL (White and Hajjar 1991).

Also, the minimum eccentricity requirement are satisfied so the design moment should at least be

$$M_u \geq N e_{min}, \qquad \text{(BS 3.8.3.2)}$$

where, e_{min} is the minimum eccentricity which is described in the previous section. In biaxial bending the algorithm ensures that the eccentricity exceeds the minimum about both the axes simultaneously.

Determine Capacity Ratio

As a measure of the stress condition of the column, a capacity ratio is calculated. The capacity ratio is basically a factor that gives an indication of the stress condition of the column with respect to the capacity of the column.

Before entering the interaction diagram to check the column capacity, the design forces N, M_x and M_y are obtained according to the previous subsections. The point (N, M_x, M_y) is then placed in the interaction space shown as point L in Figure III-3. If the point lies within the interaction volume, the column capacity is adequate; however, if the point lies outside the interaction volume, the column is overstressed.

This capacity ratio is achieved by plotting the point L and determining the location of point C. The point C is defined as the point where the line OL (if extended outwards) will intersect the failure surface. This point is determined by three-dimensional linear interpolation between the points that define the failure surface. See Figure III-3. The capacity ratio, CR, is given by the ratio $\frac{\text{OL}}{\text{OC}}$.

- If OL = OC (or CR=1) the point lies on the interaction surface and the column is stressed to capacity.
- If OL < OC (or CR<1) the point lies within the interaction volume and the column capacity is adequate.

- If OL > OC (or CR>1) the point lies outside the interaction volume and the column is overstressed.

The maximum of all the values of CR calculated from each load combination is reported for each check station of the column along with the controlling N, M_x, and M_y set and associated load combination number.

If the reinforcing area is not defined, SAP2000 computes the reinforcement that will give an interaction ratio of unity.

Design Column Shear Reinforcement

The shear reinforcement is designed for each loading combination in the major and minor directions of the column. In designing the shear reinforcement for a particular column for a particular loading combination due to shear forces in a particular direction, the following steps are involved (BS 3.8.4.6):

- Calculate the design shear stress from

$$v = \frac{V}{A_{cv}}, \; A_{cv} = bd \text{, where} \qquad \text{(BS 3.4.5.2)}$$

$$v \leq 0.8\sqrt{f_{cu}} \text{, and} \qquad \text{(BS 3.4.5.12)}$$

$$v \leq 5 \text{ N/mm}^2. \qquad \text{(BS 3.4.5.12)}$$

If v exceeds either $0.8\sqrt{f_{cu}}$ or 5 N/mm^2, the section area should be increased.

- If axial load is tensile then the shear resistance of concrete is totally neglected, i.e., the design shear stress is assumed to be resisted by the links whose area per unit length is given by

$$\frac{A_{sv}}{s_v} \geq \frac{v\,b}{0.87 f_{yv}}.$$

- If axial load is compressive and $M/N < 0.75h$, where h is the dimension of the column in the direction of shear, provide minimum links given by (BS 3.8.4.6)

$$\frac{A_{sv}}{s_v} \geq \frac{0.4\,b}{0.87 f_{yv}}, \qquad \text{(BS 3.4.5.3)}$$

where f_{yv} can not be greater than 460 MPa (BS 3.4.5.1).

- If axial load is compressive and $M/N \geq 0.75h$, calculate the design concrete shear stress from (BS 3.8.4.6)

$$v'_c = v_c + 0.75 \frac{N}{A_c} \frac{Vd}{M}, \text{ with} \qquad \text{(BS 3.4.5.12)}$$

$$v_c = \frac{0.79\,k_1 k_2}{\gamma_m} \left(\frac{100\,A_s}{bd} \right)^{1/3} \left(\frac{400}{d} \right)^{1/4}, \qquad \text{(BS 3.4.5.4)}$$

where,

k_1 is the enhancement factor for support compression and taken conservatively as 1, (BS 3.4.5.8)

$$k_2 = \left(\frac{f_{cu}}{25} \right)^{1/3}, \qquad \text{(BS 3.4.5.4)}$$

$$\gamma_m = 1.25,$$

$$0.15 \leq \frac{100\,A_s}{bd} \leq 3, \qquad \text{(BS 3.4.5.4)}$$

$$\frac{400}{d} \geq 1, \qquad \text{(BS 3.4.5.4)}$$

$$\frac{Vd}{M} \leq 1, \qquad \text{(BS 3.4.5.12)}$$

$f_{cu} \leq 40$ N/mm^2, and (BS 3.4.5.4)

A_s is the area of tensile steel layer.

If $v \leq v'_c + 0.4$, provide minimum links defined by

$$\frac{A_{sv}}{s_v} \geq \frac{0.4\,b}{0.87 f_{yv}}, \qquad \text{(BS 3.4.5.3)}$$

else if $v > v'_c + 0.4$, provide links given by

$$\frac{A_{sv}}{s_v} \geq \frac{(v - v'_c)\,b}{0.87 f_{yv}}, \qquad \text{(BS 3.4.5.3)}$$

where f_{yv} can not be greater than 460 MPa (BS 3.4.5.1).

Beam Design

In the design of concrete beams, SAP2000 calculates and reports the required areas of steel for flexure and shear based upon the beam moments, and shears, load combination factors, and other criteria described below. The reinforcement requirements are calculated at a user defined number of check stations along the beam span.

All the beams are only designed for major direction flexure and shear. Effects due to any axial forces, minor direction bending, and torsion that may exist in the beams must be investigated independently by the user.

The beam design procedure involves the following steps:

- Design beam flexural reinforcement
- Design beam shear reinforcement

Design Beam Flexural Reinforcement

The beam top and bottom flexural steel is designed at a user defined number of check stations along the beam span. In designing the flexural reinforcement for the major moment for a particular beam at a particular section, the following steps are involved:

- Determine the maximum factored moments
- Determine the reinforcing steel

Determine Factored Moments

In the design of flexural reinforcement of concrete frame beams, the factored moments for each load combination at a particular beam station are obtained by factoring the corresponding moments for different load cases with the corresponding load factors.

The beam section is then designed for the maximum positive and maximum negative factored moments obtained from all of the load combinations at that section.

Negative beam moments produce top steel. In such cases the beam is always designed as a rectangular section. Positive beam moments produce bottom steel. In such cases, the beam may be designed as a rectangular section, or T-Beam effects may be included.

Determine Required Flexural Reinforcement

In the flexural reinforcement design process, the program calculates both the tension and compression reinforcement. Compression reinforcement is added when the applied design moment exceeds the maximum moment capacity of a singly reinforced section. The user has the option of avoiding the compression reinforcement by increasing the effective depth, the width, or the grade of concrete.

The design procedure is based on the simplified rectangular stress block as shown in Figure VI-2 (BS 3.4.4.1). Furthermore it is assumed that moment redistribution in the member does not exceed 10% (i.e. $\beta_b \geq 0.9$) (BS 3.4.4.4). The code also places a limitation on the neutral axis depth, $x/d \leq 0.5$, to safeguard against non-ductile failures (BS 3.4.4.4). In addition, the area of compression reinforcement is calculated on the assumption that the neutral axis depth remains at the maximum permitted value.

The design procedure used by SAP2000, for both rectangular and flanged sections (L- and T-beams) is summarized below. It is assumed that the design ultimate axial force does not exceed $0.1 f_{cu} A_g$ (BS 3.4..4.1), hence all the beams are designed for major direction flexure and shear only.

Design of a Rectangular beam

For rectangular beams, the moment capacity as a singly reinforced beam, M_{single}, is obtained first for a section. The reinforcing steel area is determined based on whether M is greater than, less than, or equal to M_{single}. See Figure VI-2.

- Calculate the ultimate moment of resistance of the section as singly reinforced.

$$M_{single} = K' f_{cu} bd^2, \text{ where} \qquad \text{(BS 3.4.4.4)}$$

$$K' = 0.156.$$

- If $M \leq M_{single}$ the area of tension reinforcement, A_s, is obtained from

$$A_s = \frac{M}{(0.87 f_y) z}, \text{ where} \qquad \text{(BS 3.4.4.4)}$$

$$z = d\left\{0.5 + \sqrt{0.25 - \frac{K}{0.9}}\right\} \leq 0.95d, \text{ and}$$

$$K = \frac{M}{f_{cu} bd^2}.$$

This is the top steel if the section is under negative moment and the bottom steel if the section is under positive moment.

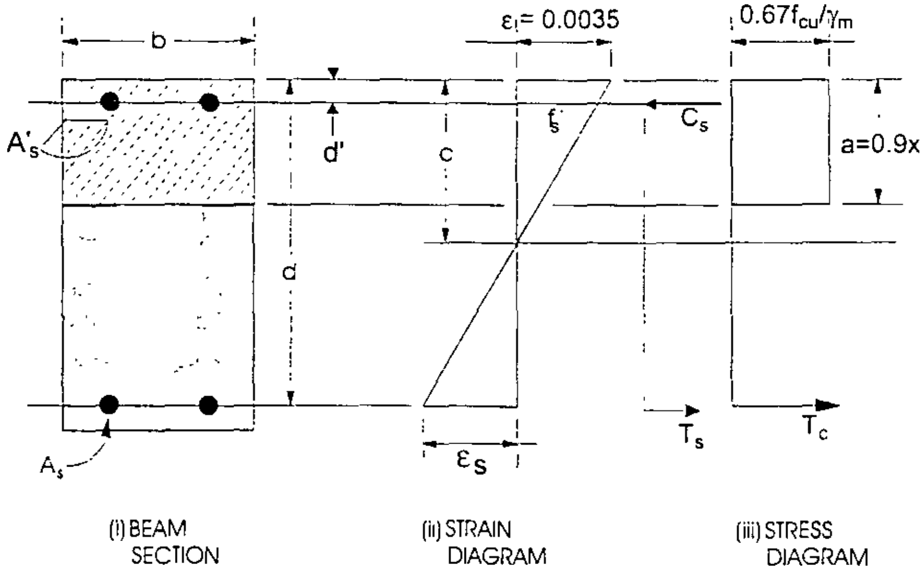


Figure VI-2
Design of Rectangular Beam Section

- If $M > M_{single}$, the area of compression reinforcement, A'_s, is given by

$$A'_s = \frac{M - M_{single}}{f'_s(d - d')},$$

where d' is the depth of the compression steel from the concrete compression face, and

$$f'_s = 0.87 f_y \qquad if \quad \frac{d'}{d} \le \frac{1}{2}\left[1 - \frac{f_y}{800}\right],$$

$$f'_s = 700\left[1 - \frac{2d'}{d}\right] \quad if \quad \frac{d'}{d} > \frac{1}{2}\left[1 - \frac{f_y}{800}\right].$$

This is the bottom steel if the section is under negative moment. From equilibrium, the area of tension reinforcement is calculated as

$$A_s = \frac{M_{single}}{(0.87 f_y)\, z} + A'_s \text{ , where} \qquad \text{(BS 3.4.4.4)}$$

$$z = 0.775d .$$

Design as a T-Beam

(i) *Flanged beam under negative moment*

The contribution of the flange to the strength of the beam is ignored. The design procedure is therefore identical to the one used for rectangular beams except that in the corresponding equations b is replaced by b_w. See Figure VI-3.

(ii) *Flanged beam under positive moment*

With the flange in compression, the program analyzes the section by considering alternative locations of the neutral axis. Initially the neutral axis is assumed to be located in the flange. Based on this assumption, the program calculates the exact depth of the neutral axis. If the stress block does not extend beyond the flange thickness the section is designed as a rectangular beam of width b_f. If the stress block extends beyond the flange width, then the contribution of the web to the flexural strength of the beam is taken into account. See Figure VI-3.

Assuming the neutral axis to lie in the flange, the normalized moment is computed as

$$K = \frac{M}{f_{cu} b_f\, d^2} .$$

Then the moment arm is computed as

$$z = d\left\{0.5 + \sqrt{0.25 - \frac{K}{0.9}}\right\} \leq 0.95d ,$$

the depth of neutral axis is computed as

$$x = \frac{1}{0.45}(d - z) \text{ , and}$$

the depth of compression block is given by

$a = 0.9x$.

- If $a \le h_f$, the subsequent calculations for A_s are exactly the same as previously defined for the rectangular section design. However, in this case the width of the compression flange, b_f, is taken as the width of the beam, b, for analysis. Whether compression reinforcement is required depends on whether $K > K'$.
- If $a > h_f$, calculation for A_s is done in two parts. The first part is for balancing the compressive force from the flange, C_f, and the second part is for balancing the compressive force from the web, C_w, as shown in Figure VI-3.

  In this case, the ultimate resistance moment of the flange is given by

  $$M_f = 0.45 f_{cu} (b_f - b_w) h_f \, (d - 0.5\, h_f),$$

  the balance of moment taken by the web is computed as

  $$M_w = M - M_f \text{, and}$$

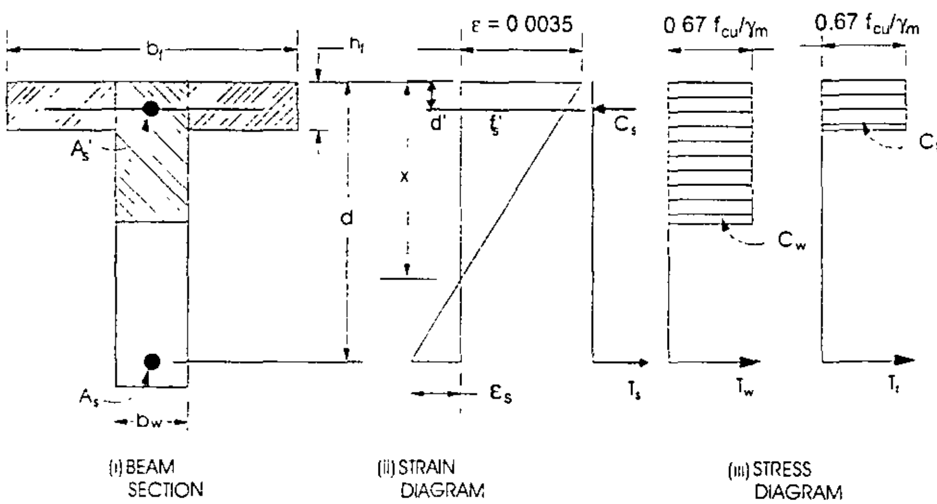


Figure VI-3
Design of a T-Beam Section

the normalized moment resisted by the web is given by

$$K_w = \frac{M_w}{f_{cu} b_w d^2}.$$

- If $K_w \le 0.156$, the beam is designed as a singly reinforced concrete beam. The area of steel is calculated as the sum of two parts, one to balance compression in the flange and one to balance compression in the web.

$$A_s = \frac{M_f}{0.87 f_y (d - 0.5 h_f)} + \frac{M_w}{0.87 f_y z}, \text{ where}$$

$$z = d\left\{0.5 + \sqrt{0.25 - \frac{K_w}{0.9}}\right\} \le 0.95d.$$

- If $K_w > 0.156$, compression reinforcement is required and is calculated as follows:

  The ultimate moment of resistance of the web only is given by

  $$M_{uw} = 0.156 f_{cu} b_w d^2.$$

  The compression reinforcement is required to resist a moment of magnitude $M_w - M_{uw}$. The compression reinforcement is computed as

  $$A_s' = \frac{M_w - M_{uw}}{f_s'(d - d')},$$

  where, d' is the depth of the compression steel from the concrete compression face, and

  $$f_s' = 0.87 f_y, \quad \text{if } \frac{d'}{d} \le \frac{1}{2}\left[1 - \frac{f_y}{800}\right], \text{ and}$$

  $$f_s' = 700\left[1 - \frac{2d'}{d}\right], \quad \text{if } \frac{d'}{d} > \frac{1}{2}\left[1 - \frac{f_y}{800}\right].$$

  The area of tension reinforcement is obtained from equilibrium

  $$A_s = \frac{1}{0.87 f_y}\left[0.45 f_{cu}(b - b_w) h_f + \left(0.156 f_{cu} b_w d^2\right)/\left(0.777d\right)\right] + A_s'$$

Design Beam Shear Reinforcement

The shear reinforcement is designed for each loading combination in the major and minor directions of the column. In designing the shear reinforcement for a particular beam for a particular loading combination due to shear forces in a particular direction, the following steps are involved (BS 3.4.5):

- Calculate the design shear stress as

$$v = \frac{V}{A_{cv}},\ A_{cv} = bd\text{, where} \qquad \text{(BS 3.4.5.2)}$$

$$v \leq 0.8\sqrt{f_{cu}}\text{, and} \qquad \text{(BS 3.4.5.2)}$$

$$v \leq 5\ \text{N/mm}^2. \qquad \text{(BS 3.4.5.2)}$$

- Calculate the design concrete shear stress from

$$v_c = \frac{0.79\, k_1 k_2}{\gamma_m}\left(\frac{100\, A_s}{bd}\right)^{1/3}\left(\frac{400}{d}\right)^{1/4}, \qquad \text{(BS 3.4.5.4)}$$

where,

k_1 is the enhancement factor for support compression, and is conservatively taken as 1, (BS 3.4.5.8)

$$k_2 = \left(\frac{f_{cu}}{25}\right)^{1/3} \geq 1\text{, and} \qquad \text{(BS 3.4.5.4)}$$

$$\gamma_m = 1.25.$$

However, the following limitations also apply:

$$0.15 \leq \frac{100\, A_s}{bd} \leq 3, \qquad \text{(BS 3.4.5.4)}$$

$$\frac{400}{d} \geq 1\text{, and} \qquad \text{(BS 3.4.5.4)}$$

$$f_{cu} \leq 40\ \text{N/mm}^2 \text{ (for calculation purpose only).} \qquad \text{(BS 3.4.5.4)}$$

A_s is the area of tensile steel.

- If $v \leq v_c + 0.4$, provide minimum links defined by

$$\frac{A_{sv}}{s_v} \geq \frac{0.4\, b}{0.87\, f_{yv}}, \qquad \text{(BS 3.4.5.3)}$$

else if $v > v_c + 0.4$, provide links given by

$$\frac{A_{sv}}{s_v} \geq \frac{(v - v_c)\, b}{0.87\, f_{yv}}, \qquad \text{(BS 3.4.5.3)}$$

where f_{yv} can not be greater than 460 MPa (BS 3.4.5.1)

Chapter VII

Design for Eurocode 2

This chapter describes in detail the various aspects of the concrete design procedure that is used by SAP2000 when the user selects the **1992 Eurocode 2** (CEN 1992). Various notations used in this chapter are listed in Table VII-1.

The design is based on user-specified loading combinations. However, the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

English as well as SI and MKS metric units can be used for input. But the code is based on Newton-Millimeter-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Newton-Millimeter-Second** units unless otherwise noted.

Design Load Combinations

The design loading combinations define the various factored combinations of the load cases for which the structure is to be checked. The design loading combinations are obtained by multiplying the characteristic loads by appropriate partial factors of safety. If a structure is subjected to dead load (DL) and live load (LL) only, the design will need only one loading combination, namely 1.35 DL + 1.5 LL.

A_c	Area of concrete, mm^2
A_{cv}	Area of section for shear resistance, mm^2
A_g	Gross cross-sectional area of a frame member, mm^2
A_s	Area of tension reinforcement, mm^2
A'_s	Area of compression reinforcement, mm^2
A_{sw}	Total cross-sectional area of links at the neutral axis, mm^2
a	Depth of compression block, mm
b	Width or effective width of the section in the compression zone, mm
b_f	Width or effective width of flange, mm
b_w	Average web width of a flanged beam, mm
d	Effective depth of tension reinforcement, mm
d'	Effective depth of compression reinforcement, mm
E_c	Modulus of elasticity of concrete, MPa
E_s	Modulus of elasticity of reinforcement, assumed as 200,000 MPa
e	Eccentricity of axial load in a column, mm
e_{min}	Minimum or nominal eccentricity, mm
e_{tot}	Total eccentricity for a braced column, mm
f_{cd}	Design concrete strength $= \frac{f_{ck}}{\gamma_c}$, MPa
f_{ck}	Characteristic compressive cylinder strength of concrete at 28 days, MPa
f_{yd}	Design yield strength of reinforcing steel $= \frac{f_{yk}}{\gamma_s}$, MPa
f_{yk}	Characteristic yield strength of reinforcement, MPa
f'_s	Compressive stress in a beam compression steel, MPa
f_y	Characteristic strength of reinforcement, MPa
f_{ywd}	Design strength of shear reinforcement, MPa
f_{ywk}	Characteristic strength of shear reinforcement, MPa
h	Overall depth of a section in the plane of bending, mm
h_f	Flange thickness, mm
l_0	Effective height of a column, mm

Table VII-1

List of Symbols Used in the Eurocode 2

l_{col}	Clear height between end restraints, mm
M	Design moment at a section, N-mm
M_1, M_2	Smaller and larger end moments in a slender column, N-mm
M_x, M_y	Applied moments about the major and minor axes of a column, N-mm
M_{Rd}	Design moment of resistance of a section N-mm
M_{Sd}	Moment at a section obtained from analysis, N-mm
m	Normalized design moment, $\dfrac{M}{bd^2 \alpha f_{cd}}$
N	Ultimate axial load, N
s_v	Spacing of links, mm
V_{Rd1}	Design shear resistance from concrete alone, N
V_{Rd2}	Design limiting shear resistane of a cross-section, N
V_{Sd}	Shear force at ultimate design load, N
V_x, V_y	Shear force at ultimate design load in two directions, N
V_{wd}	Shear force from reinforcement, N
α	Concrete strength reduction factor for sustained loading
β	Effective length factor,
	Enhancement factor of shear resistance for concentrated load
γ_f	Partial safety factor for load
γ_c	Partial safety factor for concrete strength
γ_m	Partial safety factor for material strength
γ_s	Partial safety factor for steel strength
δ	Redistribution factor
ε_c	Concrete strain
ε_s	Strain in tension steel
ε'_s	Strain in compression steel
ν	Effectiveness factor for shear resistance without concrete crushing,
	Out of plumbness factor
ρ	Tension reinforcement ratio, A_s/bd
σ_{cp}	Effective average compressive stress in concrete column, MPa
ω	Normalized tensile steel ratio, $A_s f_{yd}/\alpha f_{cd} bd$
ω'	Normalized compression steel ratio, $A_s' f_{yd}/\alpha f_{cd} bd$
ω_{lim}	Normalized limiting tensile steel ratio

Table VII-1
List of Symbols Used in the Eurocode 2 (continued)

However, in addition to the dead load and live load, if the structure is subjected to wind (WL) and earthquake (EL) forces, and considering that wind and earthquake forces are subject to reversals, the following load combinations might have to be considered (EC2 2.3.3):

1.35 DL
1.35 DL + 1.50 LL (EC2 2.3.3)

1.35 DL ± 1.50 WL
1.00 DL ± 1.50 WL
1.35 DL + 1.35 LL ± 1.35 WL (EC2 2.3.3)

1.00 DL ± 1.00 EL
1.00 DL + 1.5*0.3 LL ± 1.0 EL (EC2 2.3.3)

These are the default load combinations. These default loading combinations are produced for persistent and transient design situations (EC2 2.2.1.2) by combining load due to dead, live, wind, and earthquake loads according to the simplified formula (EC2 2.3.3.1) for ultimate limit states.

In addition to the above load combinations, the code requires that all buildings should be capable of resisting a notional design ultimate horizontal load applied at each floor or roof level (EC2 2.5.1.3). It is recommended that the user define additional load cases for considering the notional load in SAP2000.

Live load reduction factors, as allowed by some design codes, can be applied to the member forces of the live load condition on a member-by-member basis to reduce the contribution of the live load to the factored loading.

Design Strength

The design strength for concrete and steel are obtained by dividing the characteristic strength of the material by a partial factor of safety, γ_m. The values of γ_m used in the program are listed below. These values are recommended by the code to give an acceptable level of safety for normal structures under regular design situations (EC2 2.3.3.2). For accidental and earthquake situations, the recommended values are less than the tabulated value. The user should consider those separately.

Considering the partial safety factors for the materials, the design strengths of concrete and steel are given below:

f_{yd} = Design yield strength of reinforcing steel = $\frac{f_{yk}}{\gamma_s}$, and

f_{yk} = Characteristic yield strength of reinforcement,

f_{cd} = Design cylinder concrete strength = $\frac{f_{ck}}{\gamma_c}$, where,

f_{ck} = Characteristic compressive cylinder strength of concrete at 28 days,

γ_s = Partial safety factor for steel = 1.15, and

γ_c = Partial safety factor for concrete = 1.5.

Column Design

The user may define the geometry of the reinforcing bar configuration of each concrete column section. If the area of reinforcing is provided by the user, the program checks the column capacity. However, if the area of reinforcing is not provided by the user, the program calculates the amount of reinforcing required for the column. The design procedure for the reinforced concrete columns of the structure involves the following steps:

- Generate axial force-biaxial moment interaction surfaces for all of the different concrete sections types of the model (EC2 4.3.1.2). A typical biaxial interaction surface is shown in Figure III-1. When the steel is undefined, the program generates the interaction surfaces for the range of allowable reinforcement from 1 to 8 percent (EC2 5.4.1.2.1).
- Calculate the capacity ratio or the required reinforcing area for the factored axial force and biaxial (or uniaxial) bending moments obtained from each loading combination at each station of the column. The target capacity ratio is taken as one when calculating the required reinforcing area.
- Design the column shear reinforcing.

The following three sections describe in detail the algorithms associated with the above-mentioned steps.

Generation of Biaxial Interaction Surfaces

The column capacity interaction volume is numerically described by a series of discrete points that are generated on the three-dimensional interaction failure surface. In addition to axial compression and biaxial bending, the formulation allows for axial tension and biaxial bending considerations as shown in Figure III-1. The coordinates of these points are determined by rotating a plane of linear strain in three dimensions on the section of the column. See Figure III-2.

The formulation is based consistently upon the basic principles of ultimate strength design and allows for any doubly symmetric rectangular, square, or circular column section. The linear strain diagram limits the maximum concrete strain, ε_c, at the extremity of the section to 0.0035 and at a depth of $\frac{3}{7}d$ from the most compressed face to 0.0020 (EC2 4.3.1.2). See Figure VII-1.

The stress in the steel is given by the product of the steel strain and the steel modulus of elasticity, $\varepsilon_s E_s$, and is limited to the design yield strength the steel, f_{yd} (EC2 4.2.3.3.3). The area associated with each reinforcing bar is placed at the actual location of the center of the bar and the algorithm does not assume any simplifications in the manner in which the area of steel is distributed over the cross section of the column (such as an equivalent steel tube or cylinder).

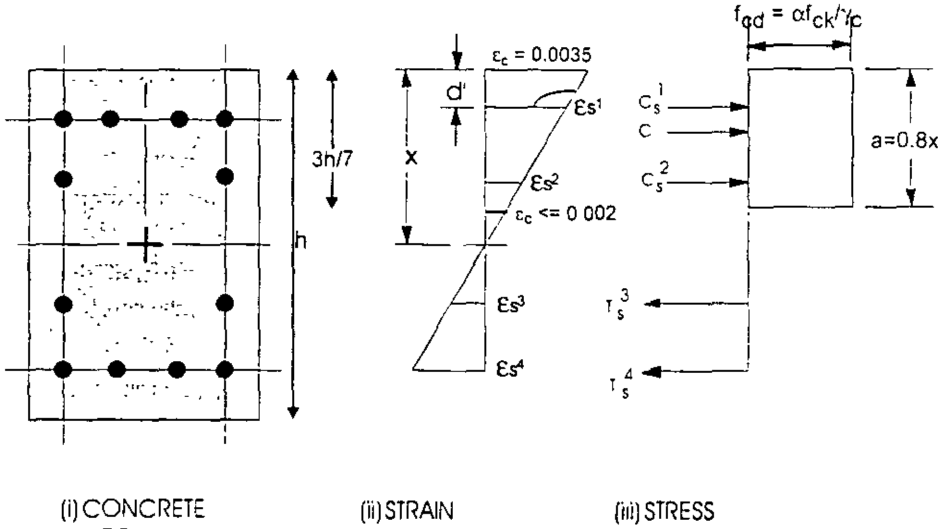


Figure VII-1
Idealized Stress and Strain Distribution in a Column Section

The concrete compression stress block is assumed to be rectangular, with a stress value of αf_{cd}, where f_{cd} is the design value of concrete cylinder compressive strength and α is the reduction factor to account for sustained compression. α is generally assumed to be 0.80 (EC2 4.2.1.3). See Figure VII-1. The interaction algorithm provides corrections to account for the concrete area that is displaced by the reinforcement in the compression zone.

Check Column Capacity

The column capacity is checked for each loading combination of each column. In checking a particular column for a particular loading combination at a particular location, the following steps are involved:

- Determine the factored moments and forces from the analysis load cases and the specified load combination factors to give N, V_x, V_y, M_x, and M_y.
- Determine the code total moments due to slender column effect. Compute moments due to minimum eccentricity.
- Check the column capacity ratio or compute the reinforcement for the column for resisting the factored moments, the code total moments, and the moments from minimum eccentricity.

The following three subsections describe in detail the algorithms associated with the above-mentioned steps.

Determine Factored Moments and Forces

Each load combination is defined with a set of load factors corresponding to the load cases. The factored loads for a particular load combination are obtained by applying the corresponding load factors to the load cases, giving N, V_x, V_y, M_x, and M_y.

Determine Code Total Moments

The determination of code total moments depends on whether the frame is "braced" or "unbraced" against side-sway.

Braced Column

Eurocode specifies that for braced columns the total moment should be computed from a set of eccentricities, such that

$$e_{tot} = e_0 + e_a + e_2 \text{, where} \qquad \text{(EC2 4.3.5.6.2)}$$

$$e_0 = 0.4\frac{M_1}{N} + 0.6\frac{M_2}{N} \geq 0.4\ \frac{M_2}{N}\text{, where } |M_1| \leq |M_2|, \qquad \text{(EC2 4.3.5.6.2)}$$

$$e_a = \frac{\nu l_0}{2}, \qquad \text{(EC2 4.3.5.4)}$$

ν is taken as 1/100, however the user can override this value (EC2 2.5.1.3),

l_0 is the effective length of a column in a given plane and is obtained from

$$l_0 = \beta\, l_{col} , \qquad \text{(EC2 4.3.5.3.5)}$$

where β is the effective length factor depending on the end conditions and resistance against side-sway, β is conservatively taken as 1 for braced frames, and l_{col} is the unsupported length corresponding to instability in the major or minor direction of the element, l_x and l_y in Figure III-4.

$$e_2 = \frac{k_1 k_2 l_0^2 f_{yd}}{45 E_s d} , \qquad \text{(EC2 4.3.5.6.3)}$$

$$k_1 = \begin{cases} 0 & \text{for } \lambda \le 15, \\ \dfrac{\lambda}{20} - 0.75, & \text{for } 15 < \lambda \le 35, \\ 1 & \text{for } \lambda > 35, \end{cases}$$

$$\lambda = \beta l_0 / r_G ,$$

r_G = the radius of gyration about the axis of bending, and

$$k_2 = \frac{N_{ud} - N_{sd}}{N_{ud} - N_{bal}} \le 1, \quad k_2 \text{ is taken as } 1.0.$$

However, the minimum eccentricity requirement is satisfied such that

$$M_{Rd} > N_{Sd}\, e_{min} , \text{ where} \qquad \text{(EC2 4.3.5.5.3)}$$

M_{Rd} = Design moment resistance of the section,

N_{Sd} = The axial force obtained from analysis, and

e_{min} is the minimum eccentricity which is taken as 0.05 times the overall dimension of the column in the plane of bending and is given by

$$e_{min} = h/20 . \qquad \text{(EC2 4.3.5.5.3)}$$

Finally the design moments are computed from the maximum of the three,

$$M_{Rd} = \max(N_{Rd} e_{tot},\quad N_{Rd} e_{min},\quad M_{factored}) . \qquad \text{(EC2 4.3.5.6)}$$

In biaxial bending, the program calculates the design moments at any station about two axes.

Unbraced Column

In the case of the unbraced column, it is assumed that the SAP2000 analysis includes P-Δ effects so that the analysis results include the effects of the additional moments. Therefore, any additional computation is not required. That means, the moment magnification factors for moments causing sidesway are taken as unity. However, it is recommended that a factor be used to obtain a P equivalent to 1.35 DL + 1.35 LL for P-Δ analysis (White and Hajjar 1991).

In addition, the minimum eccentricity requirement needs to be satisfied so that the design moment should at least be

$$M_{Rd} \geq N e_{min} , \qquad \text{(EC2 4.3.5.5.3)}$$

where, e_{min} is the minimum eccentricity which is described in the previous section. In biaxial bending the algorithm ensures that the eccentricity exceeds the minimum about both the axes simultaneously.

Determine Capacity Ratio

As a measure of the stress condition of the column, a capacity ratio is calculated. The capacity ratio is basically a factor that gives an indication of the stress condition of the column with respect to the capacity of the column.

Before entering the interaction diagram to check the column capacity, the design forces N, M_x, and M_y are obtained according to the previous subsections. The point (N, M_x, M_y) is then placed in the interaction space shown as point L in Figure III-3. If the point lies within the interaction volume, the column capacity is adequate; however, if the point lies outside the interaction volume, the column is overstressed.

This capacity ratio is achieved by plotting the point L and determining the location of point C. The point C is defined as the point where the line OL (if extended outwards) will intersect the failure surface. This point is determined by three-dimensional linear interpolation between the points that define the failure surface. See Figure III-3. The capacity ratio, CR, is given by the ratio $\frac{\text{OL}}{\text{OC}}$.

- If OL = OC (or CR=1) the point lies on the interaction surface and the column is stressed to capacity.
- If OL < OC (or CR<1) the point lies within the interaction volume and the column capacity is adequate.

- If OL > OC (or CR>1) the point lies outside the interaction volume and the column is overstressed.

The maximum of all the values of CR calculated from each load combination is reported for each check station of the column along with the controlling N, M_x, and M_y set and associated load combination number.

If the reinforcing area is not defined. SAP2000 computes the reinforcement that will give an interaction ratio of unity.

Design Column Shear Reinforcement

The shear reinforcement is designed for each loading combination in the major and minor directions of the column. The assumptions in designing the shear reinforcement are as follows:

- The column sections are assumed to be prismatic. The effect of any variation of width in the column section on the concrete shear capacity is neglected.
- The effect on the concrete shear capacity of any concentrated or distributed load in the span of the column between two beams is ignored. Also, the effect of the direct support on the columns provided by the beams is ignored.
- All shear reinforcement is provided through shear reinforcement which are perpendicular to the longitudinal reinforcement.
- The effect of any torsion is neglected for the design of shear reinforcement:

In designing the shear reinforcement for a particular column for a particular loading combination due to shear forces in a particular direction, the following steps of the standard method are involved (EC2 4.3.2.1):

- Obtain the design value of the applied shear force V_{Sd} from the SAP2000 analysis results.

  $$V_{Sd} = V_2 \quad \text{or} \quad V_3$$

- Calculate the design shear resistance of the member without shear reinforcement.

  $$V_{Rd1} = \beta\left[\tau_{Rd} k\,(1.2 + 40\rho_1) + 0.15\sigma_{cp}\right]\left(b_w d\right), \text{ where} \qquad \text{(EC2 4.3.2.3)}$$

  β = enhancement factor for shear resistance for members with concentrated loads located near the face of the support. β is taken as 1.

τ_{Rd} = basic design shear strength of concrete $= \frac{0.25 f_{ctk\,0.05}}{\gamma_c}$,

$f_{ctk\,0.05} = 0.7 f_{ctm}$,

$f_{ctm} = 0.3 f_{ck}^{2/3}$.

k = strength magnification factor for curtailment of longitudinal reinforcement and is considered to be 1,

ρ_1 = tension reinforcement ratio $= \frac{A_{s1}}{b_w d} \leq 0.02$,

A_{s1} = area of tension reinforcement,

σ_{cp} = average stress in concrete due to axial force $= \frac{N_{Sd}}{A_c}$,

N_{Sd} is the design value of the applied axial force in section, and

A_c is the total area of concrete cross-section.

- Calculate the maximum design shear force that can be carried without crushing of the notional concrete compressive struts, V_{Rd2}.

$$V_{Rd2} = \frac{1}{2} \nu \left[\frac{f_{ck}}{\gamma_c}\right]\left(0.9 b_w d\right), \text{ where} \qquad \text{(EC2 4.3.2.3)}$$

ν is the effectiveness factor $= 0.7 - \frac{f_{ck}}{200} \geq 0.5$. (EC2 4.3.2.3)

If the effective average stress in concrete $\sigma_{cp,eff}$ is more than $0.4 \frac{f_{ck}}{\gamma_c}$, V_{Rd2} should be reduced as follows:

$$V_{Rd2\,red} = 1.67 V_{Rd2}\left(1 - \frac{\sigma_{cp,eff}}{f_{cd}}\right) \leq V_{Rd2}, \text{ where} \qquad \text{(EC2 4.3.2.2)}$$

$$\sigma_{cp,eff} = \frac{N_{Sd} - f_{yd} A_{s2}}{A_c},$$

N_{Sd} is the design axial force,

f_{yd} is the design yield stress of compression steel, $f_{yd} \leq 400$ MPa,

A_{s2} is the area of reinforcement in the compression zone, A_{s2} is conservatively taken as zero for a conservative design, and

A_c is the total area of concrete cross-section.

- If $V_{Sd} \geq V_{Rd2,red}$, the notional concrete-struts will be crushed. The section is not adequate to carry the shear force.
- If $V_{Sd} < V_{Rd2,red}$, either the minimum shear reinforcement or the design shear reinforcement is required.
  - If $V_{Sd} \leq V_{Rd1}$, the concrete shear capacity is adequate to carry the shear force. Only minimum shear reinforcement is required.
  - If $V_{Sd} > V_{Rd1}$, the concrete alone is not enough to carry the shear force. Designed shear reinforcement is required. The required shear resistance from reinforcement, V_{wd} is given by

$$V_{wd} = V_{Sd} - V_{Rd1} . \qquad \text{(EC2 4.3.2.4.3)}$$

The required shear reinforcement per unit length of the column is given by

$$\frac{A_{sw}}{s_v} = \frac{V_{wd}\, \gamma_s}{0.9 d\, f_{ywd}}, \qquad \text{(EC2 4.3.2.4.3)}$$

where, f_{ywd} is the design yield strength of the shear reinforcement.

Beam Design

In the design of concrete beams, SAP2000 calculates and reports the required areas of steel for flexure and shear based upon the beam moments, shears, load combination factors, and other criteria described below. The reinforcement requirements are calculated at a user defined number of check stations along the beam span.

All the beams are only designed for major direction flexure and shear. Effects due to any axial forces, minor direction bending, and torsion that may exist in the beams must be investigated independently by the user.

The beam design procedure involves the following steps:

- Design beam flexural reinforcement
- Design beam shear reinforcement

Design Beam Flexural Reinforcement

The beam top and bottom flexural steel is designed at the design stations along the beam span. In designing the flexural reinforcement for a particular beam for a particular section, for the beam major moment, the following steps are involved:

- Determine the maximum factored moments
- Determine the reinforcing steel

Determine Factored Moments

In the design of flexural reinforcement, the factored moments for each load combination at a particular beam station are obtained by factoring the corresponding moments for different load cases with the corresponding load factors. The beam section is then designed for the maximum positive M_u^+ and maximum negative M_u^- factored moments obtained from all of the load combinations

Negative beam moments produce top steel. In such cases the beam is always designed as a rectangular section. Positive beam moments produce bottom steel. In such cases, the beam may be designed as a rectangular section, or T-Beam effects may be included.

Determine Required Flexural Reinforcement

In the flexural reinforcement design process, the program calculates both the tension and compression reinforcement. Compression reinforcement is added when the applied design moment exceeds the maximum moment capacity of a singly reinforced section. The user has the option of avoiding the compression reinforcement by increasing the effective depth, the width, or the grade of concrete.

The design procedure is based on the simplified rectangular stress block as shown in Figure VII-2 (EC2 4.3.1.2). Furthermore, it is assumed that moment redistribution in the member does not exceed the code specified limiting value. The code also places a limitation on the neutral axis depth, to safeguard against non-ductile failures (EC2 2.5.3.4.2). When the applied moment exceeds M_u, the area of compression reinforcement is calculated on the assumption that the neutral axis depth remains at the maximum permitted value.

The design procedure used by SAP2000, for both rectangular and flanged sections (L- and T-beams) is summarized below. It is assumed that the design ultimate axial force does not exceed $0.08 f_{ck} A_g$ (EC2 4.3.1.2), hence all the beams are designed for major direction flexure and shear only.

Design as a Rectangular Beam

For rectangular beams, the normalized moment, m, and the normalized section capacity as a singly reinforce beam, m_{lim}, are obtained first. The reinforcing steel area is determined based on whether m is greater than, less than, or equal to m_{lim}.

- Calculate the normalized design moment, m.

$$m = \frac{M}{bd^2 \alpha f_{cd}}, \text{ where}$$

α is the reduction factor to account for sustained compression. α is generally assumed to be 0.80 for assumed rectangular stress block, (EC2 4.2.1.3). See also page 108 for α. The concrete compression stress block is assumed to be rectangular, with a stress value of αf_{cd}, where f_{cd} is the design concrete strength and is equal to $\frac{f_{ck}}{\gamma_c}$. See Figure VII-2.

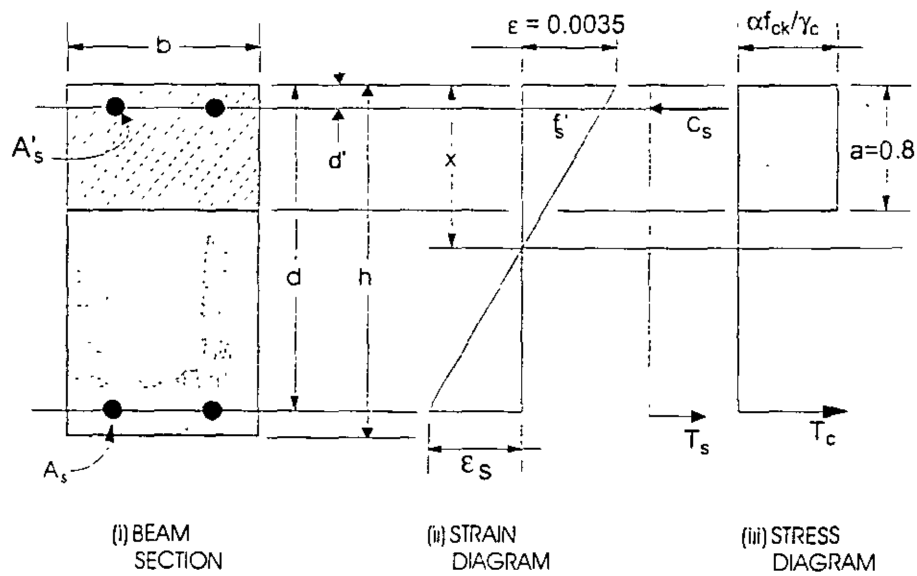


Figure VII-2
Design of a Rectangular Beam

- Calculate the normalized concrete moment capacity as a singly reinforced beam, m_{lim}.

$$m_{lim} = \left(\frac{x}{d}\right)_{lim}\left[1 - 0.4\left(\frac{x}{d}\right)_{lim}\right],$$

where the limiting value of the ratio, $\frac{x}{d}$, of the neutral axis depth at the ultimate limit state after redistribution to the effective depth, is expressed as a function of the ratio of the redistributed moment to the moment before redistribution, δ, as follows:

$$\left(\frac{x}{d}\right)_{lim} = \frac{\delta - 0.44}{1.25}, \quad \text{if} \quad f_{ck} \leq 35, \qquad \text{(EC2 2.5.3.4.1)}$$

$$\left(\frac{x}{d}\right)_{lim} = \frac{\delta - 0.56}{1.25}, \quad \text{if} \quad f_{ck} > 35, \qquad \text{(EC2 2.5.3.4.1)}$$

δ is assumed to be 1.

- If $m \leq m_{lim}$, a singly reinforced beam will suffice. Calculate the normalized steel ratio,

$$\omega = 1 - \sqrt{1 - 2m}.$$

Calculate the area of tension reinforcement, A_s, from

$$A_s = \omega\left[\frac{\alpha f_{cd} bd}{f_{yd}}\right].$$

This is the top steel if the section is under negative moment and the bottom steel if the section is under positive moment.

- If $m > m_{lim}$, the beam will not suffice as a singly reinforced beam. Both top and bottom steel are required.
  - Calculate the normalized steel ratios ω', ω_{lim}, and ω.

$$\omega_{lim} = 0.807\left(\frac{x}{d}\right)_{lim},$$

$$\omega' = \frac{m - m_{lim}}{1 - d'/d}, \text{ and}$$

$$\omega = \omega_{lim} + \omega' .$$

- Calculate the area of compression and tension reinforcement, A'_s and A_s, as follows:

$$A'_s = \omega' \left[\frac{\alpha f'_{cd} bd}{f_{yd}} \right], \text{ and}$$

$$A_s = \omega \left[\frac{\alpha f_{cd} bd}{f_{yd}} \right].$$

Design as a T-Beam

(i) Flanged beam under negative moment

The contribution of the flange to the strength of the beam is ignored if the flange is in the tension side. See Figure VII-3. The design procedure is therefore identical to the one used for rectangular beams. However, the width of the web, , b_w is taken as the width of the beam.

(ii) Flanged beam under positive moment

With the flange in compression, the program analyzes the section by considering alternative locations of the neutral axis. Initially the neutral axis is assumed to be located within the flange. Based on this assumption, the program calculates the depth of the neutral axis. If the stress block does not extend beyond the flange thickness the section is designed as a rectangular beam of width b_f. If the stress block extends beyond the flange, additional calculation is required. See Figure VII-3.

- Calculate the normalized design moment, m.

$$m = \frac{M}{b_f d^2 \alpha f_{cd}}, \text{ where}$$

 α is the reduction factor to account for sustained compression. α is generally assumed to be 0.80 for assumed rectangular stress block, (EC2 4.2.1.3). See also page 108 for α. The concrete compression stress block is assumed to be rectangular, with a stress value of αf_{cd}.

- Calculate the limiting value of the ratio, $\left(\frac{x}{d} \right)_{lim}$, of the neutral axis depth at the ultimate limit state after redistribution to the effective depth, which is expressed as a function of the ratio of the redistributed moment to the moment before redistribution, δ, as follows:

$$\left(\frac{x}{d}\right)_{lim} = \frac{\delta - 0.44}{1.25}, \text{ if } f_{ck} \leq 35, \qquad \text{(EC2 2.5.3.4.1)}$$

$$\left(\frac{x}{d}\right)_{lim} = \frac{\delta - 0.56}{1.25}, \text{ if } f_{ck} > 35, \qquad \text{(EC2 2.5.3.4.1)}$$

δ is assumed to be 1.

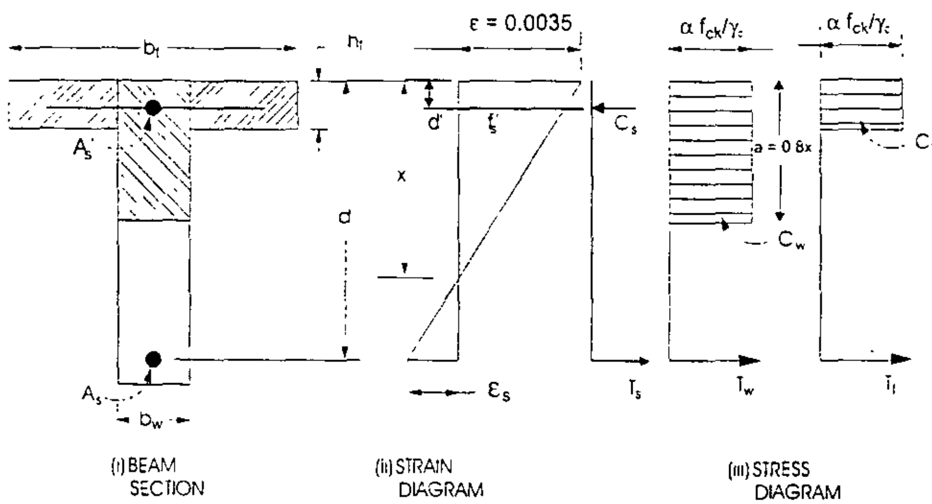


Figure VII-3
Design of a T-Beam Section

- Calculate the normalized steel ratio,

  $$\omega = 1 - \sqrt{1 - 2m}.$$

- Calculate the ratio, $\frac{x}{d}$, as follows:

  $$\frac{x}{d} = \frac{\omega}{0.807}.$$

- If $\left(\frac{x}{d}\right) \leq \left(\frac{h_f}{d}\right)$, the neutral axis lies within the flange. Calculate the area of tension reinforcement, A_s, as follows:

$$A_s = \omega\left[\frac{\alpha f_{cd} b_f d}{f_{yd}}\right].$$

- If $\left(\frac{x}{d}\right) > \left(\frac{h_f}{d}\right)$, the neutral axis lies below the flange.

  Calculate steel area required for equilibrating the flange compression, A_{s2}.

$$A_{s2} = \frac{(b_f - b_w)h_f \alpha f_{cd}}{f_{yd}},$$

  and the corresponding resistive moment is given by

$$M_2 = A_{s2} f_{yd}\left(d - \frac{h_f}{2}\right).$$

  Calculate steel area required for rectangular section of width b_w to resist moment, $M_1 = M - M_2$, as follows:

$$m_1 = \frac{M_1}{b_w d^2 \alpha f_{cd}}, \text{ and}$$

$$m_{lim} = \left(\frac{x}{d}\right)_{lim}\left[1 - 0.4\left(\frac{x}{d}\right)_{lim}\right],$$

  - If $m_1 \leq m_{lim}$,

$$\omega_1 = 1 - \sqrt{1 - 2m_1}, \text{ and}$$

$$A_{s1} = \omega_1\left[\frac{\alpha f_{cd} b_w d}{f_{yd}}\right].$$

  - If $m_1 > m_{lim}$,

$$\omega' = \frac{m_1 - m_{lim}}{1 - d'/d},$$

$$\omega_{lim} = 0.807\left(\frac{x}{d}\right)_{lim},$$

$$\omega_1 = \omega_{lim} + \omega',$$

$$A'_s = \omega'\left[\frac{\alpha f_{cd} b_w d}{f_{yd}}\right], \text{ and}$$

$$A_{s1} = \omega_1\left[\frac{\alpha f_{cd} b_w d}{f_{yd}}\right].$$

– Calculate total steel area required for the tension side.

$$A_s = A_{s1} + A_{s2}$$

Design Beam Shear Reinforcement

The shear reinforcement is designed for each loading combination at various check stations along the beam span. The assumptions in designing the shear reinforcements are as follows:

- The beam sections are assumed to be prismatic. The effect of any variation of width in the beam section on the concrete shear capacity is neglected.
- The effect on the concrete shear capacity of any concentrated or distributed load in the span of the beam between two columns is ignored. Also, the effect of the direct support on the beams provided by the columns is ignored.
- All shear reinforcements are assumed to be perpendicular to the longitudinal reinforcement.
- The effect of any torsion is neglected for the design of shear reinforcement.

In designing the shear reinforcement for a particular column for a particular loading combination due to shear forces in a particular direction, the following steps of the standard method are involved.

- Obtain the design value of the applied shear force V_{Sd} from the SAP2000 analysis results.

$$V_{Sd} = V_2$$

- Calculate the design shear resistance of the member without shear reinforcement.

$$V_{Rd1} = \beta\left[\tau_{Rd} k\,(1.2+40\rho_1)+0.15\sigma_{cp}\right]\left(b_w d\right) \qquad \text{(EC2 4.3.2.3)}$$

- Calculate the maximum design shear force that can be carried without crushing of the notional concrete compressive struts, V_{Rd2}.

$$V_{Rd2} = \frac{1}{2}\nu\left[\frac{f_{ck}}{\gamma_c}\right]\left(0.9 b_w d\right) \qquad \text{(EC2 4.3.2.4.3)}$$

- If the effective average stress in concrete $\sigma_{cp,eff}$, as defined earlier on page 113, is more than $0.4\frac{f_{ck}}{\gamma_c}$, V_{Rd2} is reduced as follows:

$$V_{Rd2,red} = 1.67\,V_{Rd2}\left(1-\frac{\sigma_{cp,eff}}{f_{cd}}\right) \le V_{Rd2}. \qquad \text{(EC2 4.3.2.2)}$$

- If $V_{Sd} \ge V_{Rd2,red}$, the notional concrete-struts will be crushed. The section is not adequate to carry the shear force.
- If $V_{Sd} < V_{Rd2,red}$, either the minimum shear reinforcement or the design shear reinforcement is required.
  - If $V_{Sd} \le V_{Rd1}$, the concrete shear capacity is enough to carry the shear force. Only minimum shear reinforcement is required.
  - If $V_{Sd} > V_{Rd1}$, the concrete alone is not enough to carry the shear force De-signed shear reinforcement is required. The required shear resistance from reinforcement, V_{wd}, is given by

$$V_{wd} = V_{Sd} - V_{Rd1}. \qquad \text{(EC2 4.3.2.4.3)}$$

The required shear reinforcement per unit length of the beam is given by

$$\frac{A_{sw}}{s_v} = \frac{V_{wd}\,\gamma_s}{0.9 d f_{ywd}}. \qquad \text{(EC2 4.3.2 4.3)}$$

Chapter VIII

Design Output

Overview

SAP2000 creates design output in three major different formats — graphical display, tabular output, and member specific detailed design information.

The graphical display of design output includes input and output design information. Input design information includes design section label, K-factors, live load reduction factor, and other design parameters. The output design information includes longitudinal reinforcing, shear reinforcing, and column capacity ratio. All graphical output can be printed.

The tabular output can be saved in a file or printed. The tabular output includes most of the information which can be displayed. This is generated for added convenience to the designer.

The member specific detailed design information shows the details of the calculation from the designer's point of view. It shows the design forces, design section dimensions, reinforcement, and some intermediate results for all the load combinations at all the design sections of a specific frame member. For a column member, it can also show the position of the current state of design forces on the column interaction diagram.

In the following sections, some of the typical graphical display, tabular output, and member specific detailed design information are described. Some of the design information is specific to the chosen concrete design codes which are available in the program and are only described where required. The ACI 318-95 design code is described in the later part of this chapter. For all other codes, the design outputs are similar.

Graphical Display of Design Output

The graphical output can be produced either as color screen display or in gray-scaled printed form. Moreover, the active screen display can be sent directly to the printer. The graphical display of design output includes input and output design information.

Input design information, for the ACI 318-95 code, includes

- Design section labels,
- K-factors for major and minor direction of buckling,
- Unbraced Length Ratios,
- C_m-factors,
- Live Load Reduction Factors,
- δ_s-factors, and
- δ_b-factors.

The output design information which can be displayed is

- Longitudinal Reinforcing,
- Shear Reinforcing, and
- Column Capacity Ratios.

The graphical displays can be accessed from the **Design** menu. For example, the longitudinal reinforcement can be displayed by selecting **Display Design Info...** from the **Design** menu. This will pop up a dialog box called **Display Design Results**. Then the user should switch ON the **Design Output** option button (default) and select **Longitudinal Reinforcing** in the drop-down box. Then clicking the **OK** button will show the longitudinal reinforcing in the active window.

The graphics can be displayed in either 3D or 2D mode. The SAP2000 standard view transformations are available for all concrete design output displays. For switching between 3D or 2D view of graphical displays, there are several buttons on the main toolbar. Alternatively, the view can be set by choosing **Set 3D View...** from the **View** menu

The graphical display in an active window can be printed in gray scaled black and white from the SAP2000 program. To send the graphical output directly to the printer, click on the **Print Graphics** button in the **File** menu. A screen capture of the active window can also be made by following the standard procedure provided by the Windows operating system.

Tabular Display of Design Output

The tabular design output can be sent directly either to a printer or to a file. The printed form of tabular output is the same as that produced for the file output with the exception that for the printed output font size is adjusted.

The tabular design output includes input and output design information which depends on the design code of choice. For the ACI 318-95 code, the tabular output includes the following. All tables have formal headings and are self-explanatory, so further description of these tables is not given.

Input design information includes the following:

- Concrete Column Property Data
  - Material label,
  - Column dimensions,
  - Reinforcement pattern,
  - Concrete cover, and
  - Bar area.
- Concrete Beam Property Data
  - Material label,
  - Beam dimensions,
  - Top and bottom concrete cover, and
  - Top and bottom reinforcement areas.
- Load Combination Multipliers
  - Combination name,
  - Load types, and
  - Load factors.

- Concrete Design Element Information (code dependent)
  - Design Section ID,
  - K-factors for major and minor direction of buckling,
  - Unbraced Length Ratios,
  - C_m-factors,
  - Live Load Reduction Factors.
- Concrete Moment Magnification Factors (code dependent)
  - Section ID,
  - Element Type,
  - Framing Type,
  - δ_s-factors, and
  - δ_b-factors.

The output design information includes the following:

- Column Design Information
  - Section ID,
  - Station location,
  - Total longitudinal reinforcement and the governing load combination,
  - Major shear reinforcement and the governing load combination, and
  - Minor shear reinforcement and the governing load combination.
- Beam Design Information
  - Section ID,
  - Station location,
  - Top longitudinal reinforcement and the governing load combination,
  - Bottom reinforcement and the governing load combination, and
  - Major shear reinforcement and the governing load combination.

The tabular output can be accessed by selecting **Print Design Tables...** from the **File** menu. This will pop up a dialog box. Then the user can specify the design quantities for which the results are to be tabulated. By default, the output will be sent to the printer. If the user wants the output stream to be redirected to a file, he/she can check the **Print to File** box. This will provide a default filename. The default filename can be edited. Alternatively, a file list can be obtained by clicking the **File Name** button to chose a file from. Then clicking the **OK** button will direct the tabular output to the requested stream — the file or the printer.

Member Specific Information

The member specific design information shows the details of the calculation from the designer's point of view. It provides an access to the geometry and material data, other input data, design forces, design section dimensions, reinforcement details, and some of the intermediate results for a member. The design detail information can be displayed for a specific load combination and for a specific station of a frame member. For a column member, it can also show the position of the current state of design forces on the column interaction diagram.

The detailed design information can be accessed by **right clicking** on the desired frame member. This will pop up a dialog box called **Concrete Design Information** which includes the following tabulated information for the specific member. If the selected member is a column, the dialog box includes

- Load combination ID,
- Station location,
- Longitudinal reinforcement area,
- Major shear reinforcement area, and
- Minor shear reinforcement area.

If the selected member is a beam, the dialog box includes

- Load combination ID,
- Station location,
- Top reinforcement area,
- Bottom reinforcement area, and
- Shear reinforcement area.

Additional information can be accessed for column members by clicking on the **Re-Design**, **Details**, and **Interaction** buttons in the dialog box. For beams additional information can be accessed by clicking on the **ReDesign** and **Details** buttons in the dialog box.

Additional information that is available by clicking on the **ReDesign** button is as follows:

- Design Factors (code dependent)
  - Effective length factors, K, for major and minor direction of buckling.
  - Unbraced Length Ratios,
  - C_m-factors,
  - Live Load Reduction Factors,
  - δ_s-factors, and
  - δ_b-factors.
- Element Section ID
- Element Framing Type

Additional information that is available by clicking on the **Details** button is given below. The details of this information depends on whether the selected member is a beam or a column. If the member is a column, the information includes:

- Frame, Section, Station, and Load Combination IDs,
- Section geometric information and graphical representation,
- Material properties of steel and concrete,
- Design axial force and biaxial moments,
- Minimum design moments,
- Moment factors,
- Design shear forces, and
- Shear capacities of concrete and steel.

If the member is a beam, the information includes:

- Frame, Section, Station, and Load Combination IDs,
- Section geometric information and graphical representation,
- Material properties of steel and concrete,
- Design moments and shear forces,
- Minimum design moments,
- Top and bottom reinforcing areas,
- Shear capacities of concrete and steel, and
- Shear reinforcing area.

Clicking on the **Interaction** button displays the interaction diagram in a three dimensional space for the column section. The design axial force and the biaxial moments are plotted on the interaction diagram to show the state of stress in the column. The interaction diagram can be viewed in any orientation and the view can be manipulated from the interaction dialog box. The interaction diagram can be printed for hard-copy output.

References

ACI, 1995

Building Code Requirements for Reinforced Concrete (ACI 318-95) and Commentary (ACI 318R-95), American Concrete Institute, Detroit, Michigan, 1995.

BSI, 1985

BS 8110 . Part 1, Structural Use of Concrete. Part 1, Code of Practice for Design and Construction, British Standards Institution, London, UK, 1985.

CEN, 1992

ENV 1992-1-1, Eurocode 2: Design of Concrete Structures, Part 1, General Rules and Rules for Buildings, European Committee for Standardization, Brussels, Belgium, 1992.

CEN, 1994

ENV 1991-1, Eurocode 1: Basis of Design and Action on Structures — Part 1, Basis of Design, European Committee for Standardization, Brussels, Belgium, 1994.

CSA, 1984

CAN3-A23.3-M84, Design of Concrete Structures for Buildings, A National Standard of Canada, Canadian Standards Association, Rexdale, Ontario, Canada, 1984.

CSI, 1997a

SAP2000 Getting Started, Computers and Structures, Inc., Berkeley, California, 1997.

CSI, 1997b

SAP2000 Analysis Reference. Vols. I and II, Computers and Structures, Inc., Berkeley, California, 1997.

ICBO, 1997

Uniform Building Code, International Conference of Building Officials, Whittier, California, 1997.

PCA, 1996

Notes on ACI 318-95, Building Code Requirements for Reinforced Concrete, with Design Applications, Portland Cement Association, Skokie, Illinois, 1996.

D. W. White and J. F. Hajjar, 1991

"Application of Second-Order Elastic Analysis in LRFD: Research to Practice," *Engineering Journal, American Institute of Steel Construction, Inc.*, Vol. 28, No. 4, 1991.

Index

Additional moment
British, 89

Balanced condition
ACI, 48, 51
Canadian, 73, 75

Beam flexural design, 2, 26
ACI, 46
British, 94
Canadian, 72
Eurocode, 114

Beam shear design, 26
ACI, 53
British, 100
Canadian, 79
Eurocode, 121

Braced frames
ACI, 40
British, 89
Canadian, 65
Eurocode, 109

Check station, 25

Code total moment
Eurocode, 109

Column capacity ratio, 29
ACI, 39, 41
British, 88, 91
Canadian, 65, 67
Eurocode, 109, 111

Column flexural design, 26
ACI, 37
British, 87
Canadian, 63
Eurocode, 107

Column shear design, 29
ACI, 42
British, 92
Canadian, 68
Eurocode, 112

Compression reinforcement
ACI, 48, 51
British, 96, 99
Canadian, 74 - 75
Eurocode, 117, 120

Concrete shear capacity
ACI, 44, 56
British, 92, 100
Canadian, 70, 82
Eurocode, 112, 122

Demonstration
accessing detailed information, 14

column interaction, 15
design, 12
editing member properties, 16
redesign, 15

Design load combinations, 1, 24
ACI, 33
British, 83
Canadian, 62
Eurocode, 103

Design of T-beams
ACI, 49
British, 97
Canadian, 75
Eurocode, 118

Detailed output, 128
demonstration, 14

Ductile detailing
ACI, 53
Canadian, 78

Earthquake resisting frames
ductile, 42, 69, 78 - 79
intermediate, 33, 53, 56
nominal, 59, 78, 80
ordinary, 33
shear force in special frames, 43
shear in intermediate frames, 44, 56
special, 33, 52, 55

Element unsupported length, 30

Factored moments and forces
ACI, 39, 47
British, 89, 94
Canadian, 65, 72
Eurocode, 109, 115

Flexural reinforcement
ACI, 46 - 47
British, 94 - 95
Canadian, 72 - 73
Eurocode, 115

Generation of biaxial interaction surfaces, 28
ACI, 37
British, 87
Canadian, 63
Eurocode, 107

Graphical output, 124

Identification of beams, 25

Identification of columns, 25

Interaction diagram, 27
ACI, 37
British, 87
Canadian, 63
demonstration, 15
Eurocode, 107

Interactive environment, 1

Lateral drift effect, 30, 65
See also P-Delta analysis

Live load reduction factor, 25, 36, 62, 86, 106

Maximum column reinforcement
ACI, 37
British, 87
Canadian, 63
Eurocode, 107

Member specific output, 127

Minimum column reinforcement
ACI, 37
British, 87
Canadian, 63
Eurocode, 107

Minimum eccentricity
ACI, 39
British, 90
Canadian, 65
Eurocode, 110 - 111

Minimum tensile reinforcement
ACI, 52
Canadian, 78

Moment magnification
ACI, 39
British (additional moment), 89
Canadian, 65
Eurocode (total moment), 109

Nominal moment capacity, 69

Nonsway frames
ACI, 40
British, 89
Canadian, 65
Eurocode, 109

Output, 1
details, 128
graphical, 123 - 124
interaction diagram, 129
member specific, 123, 127
tabular, 123, 125

Overstrength factor, 80

P-Delta analysis, 30
ACI, 40
British, 91
Canadian, 66
Eurocode, 111

Probable moment capacity, 42, 69

Rectangular beam design
ACI, 47
British, 95
Canadian, 73
Eurocode, 116

Redesign, 127
demonstration, 15

Shear reinforcement
ACI, 45, 56
British, 92, 101
Canadian, 70, 82
Eurocode, 114, 122

Special considerations for seismic loads, 26, 32
ACI, 33, 43, 52, 55
Canadian, 59, 69, 79

Strength reduction factors
ACI, 36
British, 86
Canadian, 62
Eurocode, 106

Supported design codes, 1
ACI, 23, 33
British, 23, 83
Canadian, 23, 59
Eurocode, 23, 103

Sway frames
ACI, 40
British, 89
Canadian, 65
Eurocode, 109

Tabular output, 125

T-Beam design
ACI, 49
British, 97
Canadian, 75
Eurocode, 118

Unbraced frames, 89
ACI, 40
British, 91
Canadian, 65
Eurocode, 109, 111

Units, 2, 32
ACI, 33
British, 83
Canadian, 59
Eurocode, 103

Unsupported length, 41

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA
"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

SAP2000 INTEGRATED FINITE ELEMENT ANALYSIS AND DESIGN OF STRUCTURES

STEEL DESGIN MANUAL

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DE 2001

Palacio de Minería, Calle de Tacuba No. 5, Primer piso, Delegación Cuauhtémoc, CP 06000, Col. Centro, México D.F.,
APDO Postal M-2285 • Tels: 5521.4021 al 24, 5623.2910 y 5623.2971 • Fax: 5510.0573

SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

STEEL DESIGN MANUAL

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

Version 6.1
October 1997

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Tel: (510) 845-2177
Fax: (510) 845-4096
E-mail: *info@csiberkeley.com*
Web: *www.csiberkeley.com*

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THIS PROGRAM IS A VERY PRACTICAL TOOL FOR THE DESIGN/ CHECK OF STEEL STRUCTURES. HOWEVER, THE USER MUST THOROUGHLY READ THE MANUAL AND CLEARLY RECOGNIZE THE ASPECTS OF STEEL DESIGN THAT THE PROGRAM ALGORITHMS DO NOT ADDRESS.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table of Contents

CHAPTER I **Introduction** 1
Overview 1
Organization 2
Recommended Reading 3

CHAPTER II **Quick Tutorial** 5
Overview 5
Description of the Model 6
Starting the Tutorial 8
Opening the Model Database File 8
Analyzing the Model 10
Selecting the Design Code 11
Starting Design and Stress Check 12
Modifying Member Properties 16
Selecting Sections Automatically 19
Re-analyzing with Updated Elements 22
Concluding Remarks 23

CHAPTER III **Design Algorithms** 25
Design Load Combinations 26
Design and Check Stations 27
Element Stress Ratios 27
P-Δ Effects 28
Element Unsupported Lengths 28
Effective Length Factor (K) 30
Choice of Input Units 32

CHAPTER IV **Check/Design for AISC-ASD89** 33

Design Loading Combinations 36
Classification of Sections 36
Calculation of Actual Stresses 39
Calculation of Allowable Stresses 39
Allowable Stress in Compression. 39
Allowable Stress in Tension 40
Allowable Stress in Bending 40
I-sections, C-sections, T-sections, Angles and Double angles. 40
Box Sections and Rectangular Tubes 42
Pipe Sections 43
Rectangular and Round Bars 43
General sections 43
Allowable Stress in Shear 43
Calculation of Stress Ratios 44
Axial and Bending Stresses. 44
Shear Stresses. 45

CHAPTER V **Check/Design for AISC-LRFD93** 47

Design Loading Combinations 50
Classification of Sections 50
Calculation of Factored Forces and Moments. 54
Calculation of Nominal Strengths 55
Compression Capacity 55
Tension Capacity 56
Nominal Strength in Bending. 57
Yielding 57
Lateral-Torsional Buckling 57
Local Buckling. 59
Shear Capacities 60
Calculation of Capacity Ratios 61
Axial and Bending Stresses. 61
Shear Stresses. 62

CHAPTER VI **Check/Design for CISC94** 63

Design Loading Combinations 66
Classification of Sections 67
Calculation of Factored Forces and Moments 67
Calculation of Factored Strengths 67
Compression Strength 70
Tension Strength 70
Bending Strengths 71
I-shapes and Boxes. 71

Rectangular Bar 72
Pipes and Circular Rods 73
Channel Sections 73
T-shapes and double angles 74
Single Angle and General Sections 74
Shear Strengths 74
Calculation of Capacity Ratios 76
Axial and Bending Stresses 76
Shear Stresses 78

CHAPTER VII Check/Design for BS 5950 **79**

Design Loading Combinations 82
Classification of Sections 83
Calculation of Factored Forces and Moments 83
Calculation of Section Capacities 85
Compression Resistance 87
Tension Capacity 88
Moment Capacity 89
Plastic and Compact Sections 89
Semi-compact Sections 90
Lateral-Torsional Buckling Moment Capacity 90
Shear Capacities 92
Calculation of Capacity Ratios 92
Local Capacity Check 92
Under Axial Tension 93
Under Axial Compression 93
Overall Buckling Check 93
Shear Capacity Check 93

CHAPTER VIII Check/Design for EUROCODE 3 **95**

Design Loading Combinations 98
Classification of Sections 99
Calculation of Factored Forces 99
Calculation of Section Resistances 104
Tension Capacity 104
Compression Resistance 104
Shear Capacity 107
Moment Resistance 108
Lateral-torsional Buckling 108
Calculation of Capacity Ratios 110
Bending, Axial Compression, and Low Shear 110
Bending, Axial Compression, and High Shear 111
Bending, Compression, and Flexural Buckling 111
Bending, Compression, and Lateral-Torsional Buckling 112

Bending, Axial Tension, and Low Shear 113
Bending, Axial Tension, and High Shear 113
Bending, Axial Tension, and Lateral-Torsional Buckling 114
Shear 114

CHAPTER IX Design Output **115**

Overview 115
Graphical Display of Design Output 116
Tabular Display of Design Output 117
Member Specific Information 118

References **121**

Index **123**

Chapter I

Introduction

Overview

SAP2000 features powerful and completely integrated modules for design of both steel and reinforced concrete structures. The program provides the user with options to create, modify, analyze and design structural models, all from within the same user interface. The program is capable of performing initial member sizing and optimization from within the same interface.

The program provides an interactive environment in which the user can study the stress conditions, make appropriate changes, such as revising member properties, and re-examine the results without the need to re-run the analysis. A single mouse click on an element brings up detailed design information. Members can be grouped together for design purposes. The output in both graphical and tabulated formats can be readily printed.

The program is structured to support a wide variety of design codes for the automated design and check of steel frame members. The program currently supports the following steel design codes: U.S. ASD and LRFD (AISC 1989, AISC 1994), Canadian (CISC 1995), British (BSI 1990), and European (CEN 1992).

The design is based upon a set of user-specified loading combinations. However, the program provides a set of default load combinations for each design code supported in SAP2000. If the default load combinations are acceptable, no definition of additional load combination is required.

In the design process the program picks the least weight section required for strength for each element to be designed, from a set of user specified sections. Different sets of available sections can be specified for different groups of elements. Also several elements can be grouped to be designed to have the same section.

In the check process the program produces demand/capacity ratios for axial load and biaxial moment interactions and shear. The demand/capacity ratios are based on element stress and allowable stress for allowable stress design, and on factored loads (actions) and factored capacities (resistances) for limit state design.

The checks are made for each user specified (or program defaulted) load combination and at several user controlled stations along the length of the element. Maximum demand/capacity ratios are then reported and/or used for design optimization.

All allowable stress values or design capacity values for axial, bending and shear actions are calculated by the program. Tedious calculations associated with evaluating effective length factors for columns in moment frame type structures are automated in the algorithms.

The presentation of the output is clear and concise. The information is in a form that allows the designer to take appropriate remedial measures if there is member overstress. Backup design information produced by the program is also provided for convenient verification of the results.

Special requirements for seismic design are not implemented in the current version of SAP2000.

English as well as SI and MKS metric units can be used to define the model geometry and to specify design parameters.

Organization

This manual is organized in the following way:

Chapter II provides a quick tutorial aimed at giving the first time users hands-on experience. Several of the basic features of the SAP2000 steel design modules are explored in this tutorial.

Chapter III outlines various aspects of the steel design procedures of the SAP2000 program. This chapter describes the common terminology of steel design as implemented in SAP2000.

Each of the subsequent chapters gives a detailed description of a specific code of practice as interpreted by and implemented in SAP2000. Each chapter describes the design loading combinations to be considered; allowable stress or capacity calculations for tension, compression, bending, and shear; calculations of demand/capacity ratios; and other special considerations required by the code.

Chapter IV gives a detailed description of the AISC ASD code (AISC 1989) as implemented in SAP2000.

Chapter V gives a detailed description of the AISC LRFD code (AISC 1994) as implemented in SAP2000.

Chapter VI gives a detailed description of the Canadian code (CISC 1994) as implemented in SAP2000.

Chapter VII gives a detailed description of the British code BS 5950 (BSI 1990) as implemented in SAP2000.

Chapter VIII gives a detailed description of the Eurocode 3 (CEN 1992) as implemented in SAP2000.

Chapter IX outlines various aspects of the tabular and graphical output from SAP2000 related to steel design.

Recommended Reading

It is recommended that first time users follow the steps of the "Quick Tutorial" in Chapter II and read Chapter III "Design Algorithms" and the Chapter corresponding to the code of interest to the user. Finally the user should read "Design Output" in Chapter IX for understanding and interpreting SAP2000 output related to steel design.

Chapter II

Quick Tutorial

Overview

Several of the basic features of the SAP2000 steel design module are explored in this tutorial. This introduction is aimed at giving the first time users hands-on experience. The program allows you to select from several U.S. and international codes to stress check and design a steel structure. It is assumed that you have a working knowledge of steel design procedures and are reasonably familiar with the current codes of practice and their underlying design concepts. A comprehensive on-line Help is included in the program for your quick reference.

We will access the SAP2000 commands from both the Toolbar and from the menus. The Toolbars, however, provides quick access to most commonly used features available from the menus.

In the assignment sequence, there are two important points you must remember. First, you have to define an entity before you can assign an attribute to it, and second, you have to select member(s) before you can assign new attributes or modify old ones.

Description of the Model

The structure is a two-story, two-by-three bay office building. The frame will be designed in accordance with the AISC-LRFD93 code. Earthquake induced force is considered in the analysis and design of this frame. However, special requirements for the design of moment resisting ductile steel frames are not currently implemented in SAP2000.

Geometry

The moment frame is of structural steel as shown in Figure II-1. The second floor has metal deck and lightweight concrete fill, whereas the roof has metal deck only. Typical story height is taken as 13'0". The initial member sizes are given in Figure II-1.

Material Properties

Material specifications are:

Beams and girders:	ASTM A36	(F_y = 36 ksi)
Columns:	ASTM A572	(F_y = 50 ksi)

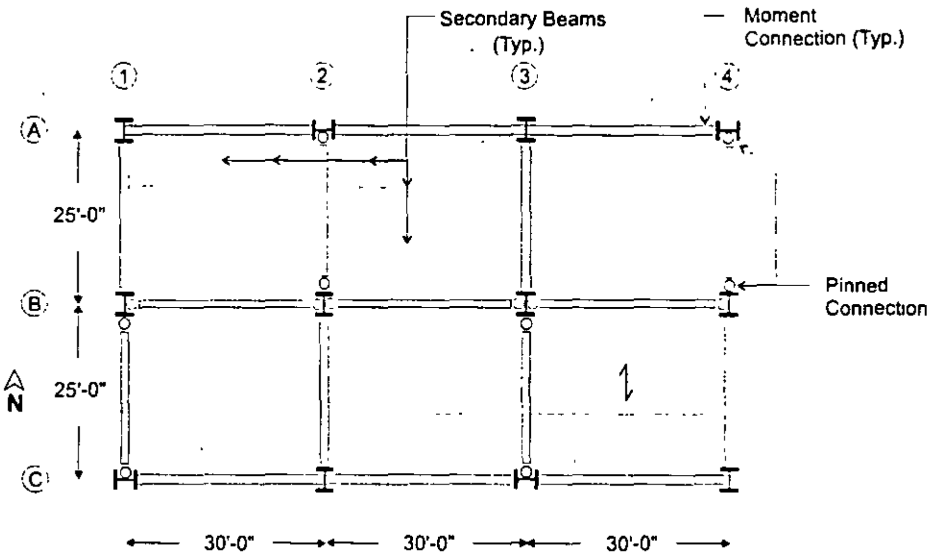
Load Cases

Four load cases are considered in the analysis — DL, LL, QX, and QY. The dead and live loads are defined as the load cases DL and LL respectively. The lateral seismic loads, in turn, are designated as the load cases QX and QY.

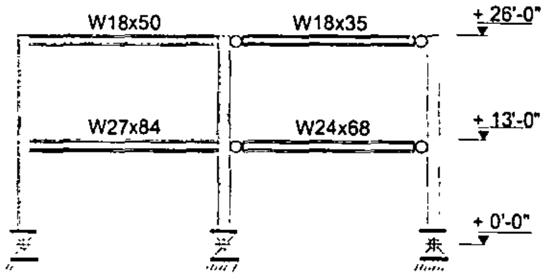
The dead and live loads are applied as beam span loads based on the loading intensities given below:

Roof	Dead load	= 30 psf
	Live load	= 20 psf
Floor	Dead load	= 75 psf
	Live load	= 80 psf
Wall	Dead load	= 15 psf

Equivalent static seismic forces are applied as lateral joint loads in the global X and global Y directions, separately. The total base shear in each direction is computed as 51 kips.



Framing Plan



All columns are W14x132

Transverse Elevation

Loading:

Roof	Dead load	= 30 psf
	Live load	= 20 psf
Floor	Dead load	= 75 psf
	Live load	= 80 psf
Extenor	Wall	= 15 psf

Steel Grades:

For beams F_y = 36 ksi
For columns F_y = 50 ksi

Figure II-1
Description of the Model (Tutorial Example)

Analysis

Two diaphragm constraints are applied for the two diaphragms at the two floor levels. These constraints prevent in-plane relative displacements of the nodes at each floor. The lateral earthquake loads are applied at nodal points on the floor levels. A P-Δ analysis is carried out with a load combination of 1.2 DL + 0.5 LL as described in Chapter "Check/Design for AISC-LRFD93". The initial unbraced length is taken as the full member length.

Design

The stress check and design are performed in accordance with AISC-LRFD93. Kip-in units are used. The input data file for this model is **EXSTL.SDB**. This is supplied as part of the SAP2000 package.

Starting the Tutorial

A step-by-step procedure for the stress check and design of the model is outlined below. It is recommended that you actually perform these steps while reading this chapter. We assume that you have successfully started the program. You can do this by running SAP2000 from the Start Menu of the Windows operating system.

In this tutorial we will typically use the Toolbars to access various options quickly. Most of the features available on the Toolbars can also be accessed from the Menu Bar. Use the on-line Help or refer to the "SAP2000 Getting Started" (CSI 1997a) manual for a detailed description of the SAP2000 screen.

The input data file for the model (**EXSTL.SDB**) is in the EXAMPLES subdirectory under the directory where the program has been installed. A section property file of name **SECTIONS.PRO** is also required for the tutorial. The SECTIONS.PRO file is also available in the directory where the program has been installed.

Opening the Model Database File

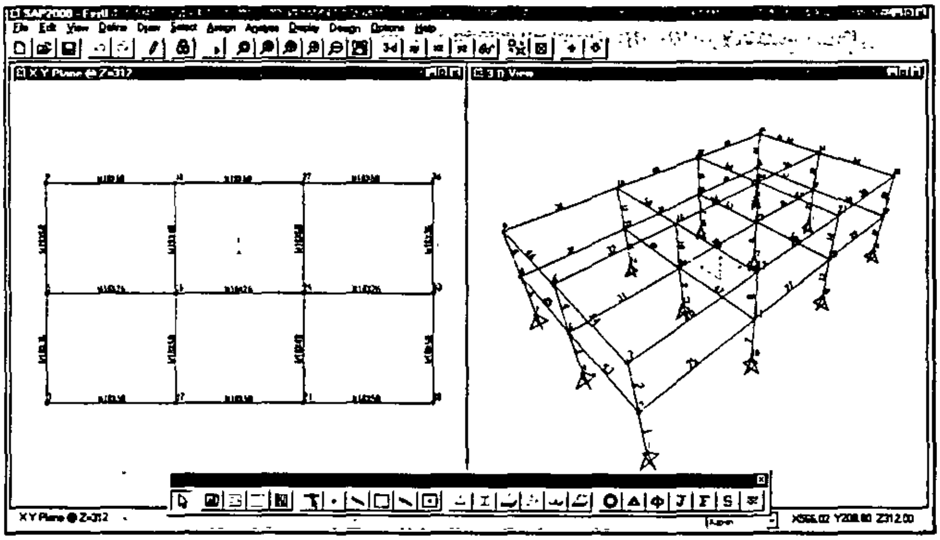
1. Click on the **Open** button in the **File** menu. This will display the **Open Model File** dialog box.

2. In this dialog box:

   - Select the **EXSTL.SDB** file.

   - Click on the **Open** button.

The screen will now show two vertically-tiled windows. The left window displays a plan view of the model at level + 312 in. Section labels are displayed in this view. A three-dimensional view of the model is shown in the right window. Element and node IDs are shown in this view. In addition, a floating toolbar is displayed at the bottom of the main window along with the main toolbar at the top of the window.



Note: When working with multiple windows, clicking anywhere in a particular window will activate that window.

Before we proceed further, we will make a copy of the data file by saving the model under a new name, say, **TUTOR2.SDB**. We will use this copy during the tutorial and leave the original file unaltered.

3. From the **File** menu, choose **Save As....** This will display the **Save Model File As** dialog box.

4. In this dialog box:

   - Enter new filename, Tutor2.SDB.

     Note: Even if you do not type in the extension .SDB, the program automatically appends this extension to the filename.

   - Click on the **Save** button.

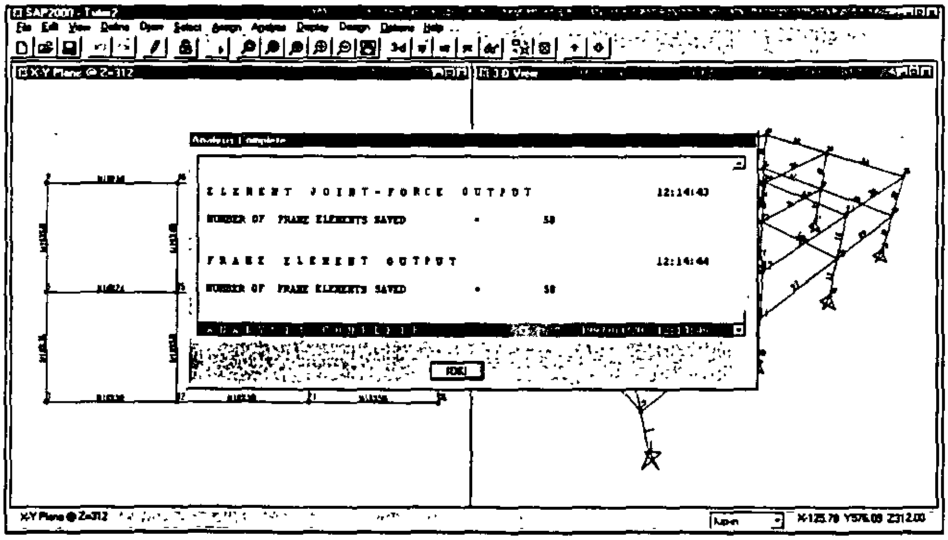
The new name is displayed in the Main Title Bar.

Analyzing the Model

We will now analyze the model. Before analyzing the model we need to set the P-Δ force (1.2 DL + 0.5 LL) and other parameters for P-Δ analysis. To do this:

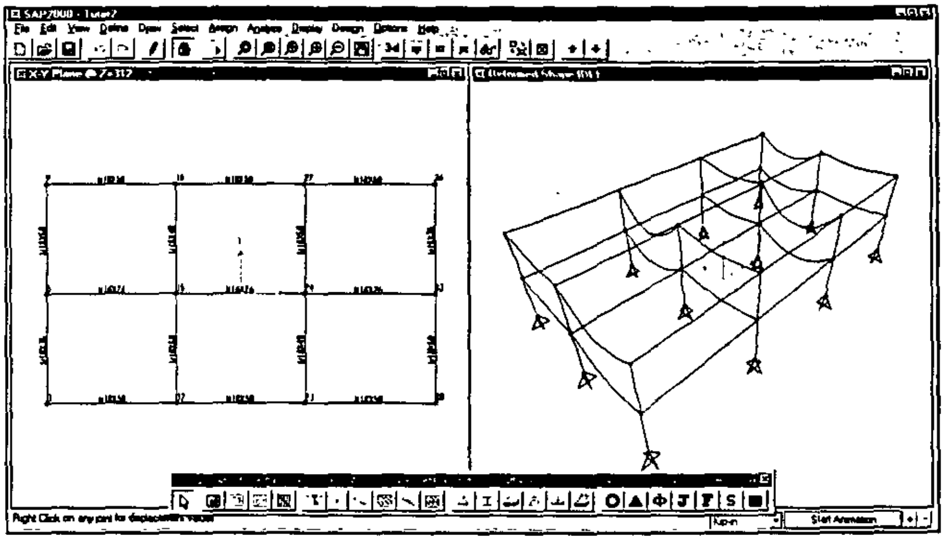
1. Select the **Set Options...** menu item in the **Analyze** menu. This will immediately bring up the **Analysis Options** dialog box. In this dialog box:
    - Click the **Include P-Delta** check box.
    - Click on the **Set P-Delta Parameters** button. This will bring up the **P-Delta Parameters** dialog box. In this dialog box:
        - Set maximum iteration to 5.
        - Change the DL scale factor to 1.2 and click **Modify**.
        - Click on the **Load Case** drop down arrow.
        - Select LL.
        - Change the LL scale factor to 0.5 and click **Add**.
        - Click **OK** to close the **P-Delta Parameters** dialog box
    - Click **OK** to close the **Analysis Options** dialog box.
2. Click on the **Run Analysis** button from the main toolbar.

A top window is opened in which various phases of analysis are progressively reported. When the analysis is complete the screen will display the following:



3. Use the scroll bar in the top window to review the analysis messages and to check for any error or warning messages. In our case there should be none.

4. Click on the **OK** button in the top window to close it. This will respond by displaying the deformed shape in the right window for the first load case (DL).



The floating toolbar occupies some of the display area. To get it back, move the floating toolbar below the SAP2000 main window.

Selecting the Design Code

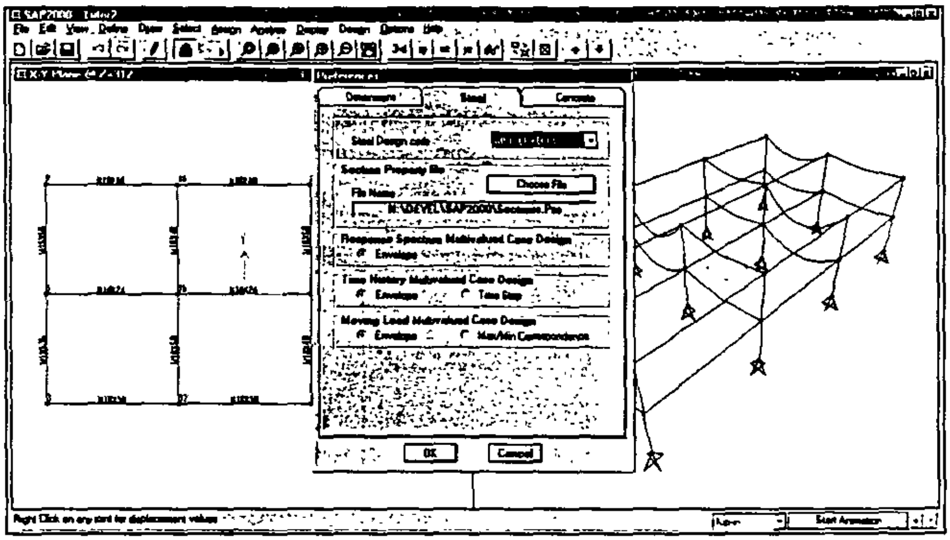
The default design code is AISC-ASD89 for design of steel structures. To choose the AISC-LRFD93 design code, do the following:

1. Click on **Preferences...** from the **Options** menu. This will launch the **Preferences** dialog box.

2. In the dialog box click on the **Steel** tab. You can see the currently selected steel design code, default section property file, and some other options as shown below.

3. You need to change the default design code (AISC-ASD89) to AISC-LRFD93. to do this:

   - Click on the drop down arrow in the design code box.
   - Select the **AISC-LRFD93** design code.



4. Click on the **OK** button to close the dialog box.

5. To make sure that SAP2000 will design the steel members, select **Steel Design** menu item from the **Design** menu.

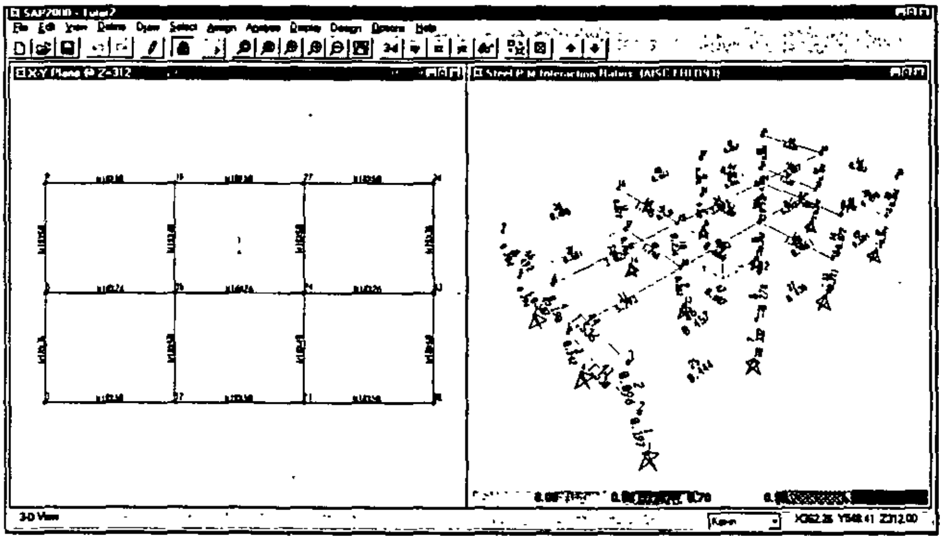
Starting Design and Stress Check

With the analysis phase and selection of design code completed, we will now examine member stress ratios using the requirements of AISC-LRFD93.

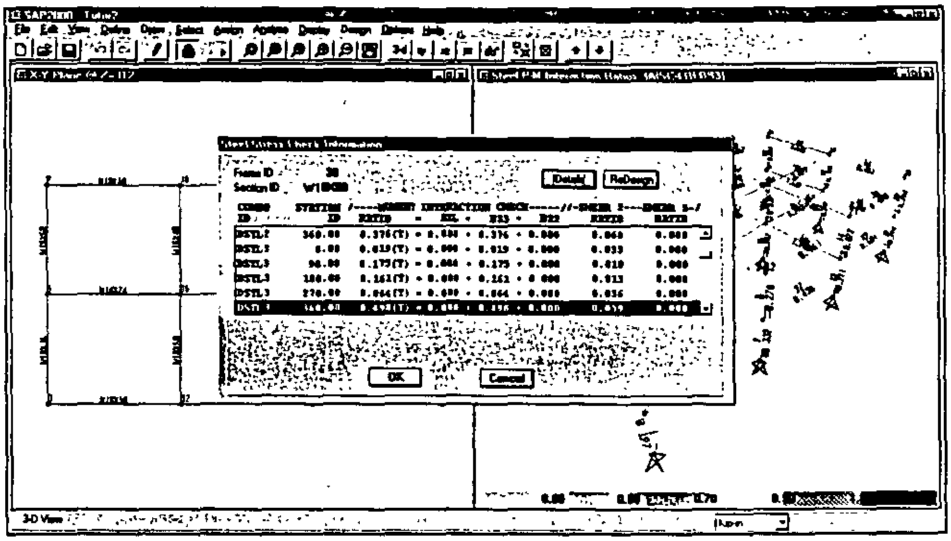
1. From the **Design** menu, choose **Start Design/Check of Structure**. The program now calculates the interaction ratios for each of the steel frame members. (If we had selected some of the frame members, then stress ratios would have been calculated for only the selected frame members). In a few moments the color-coded stress ratios are displayed for each member. By default, these are the axial force-moment interaction ratios which, according to the code, should not exceed 1.0. In the display, the stress ratios are reported for the governing design load combination.

Note: Since no load combinations were defined in the model, SAP2000 design automatically provided a set of design load combinations for the selected design code (AISC-LRFD93 for Steel). The default load combinations can be added by any of the three following ways.

- *Clicking* **Add Default Load Combinations** *in the* **Load Combinations** *dialog box launched from the* **Define** *menu by clicking* **Define Load Combinations**.
- *Clicking* **Select Design Combos...** *from the* **Design** *menu when there are no design load combinations defined in the model.*
- *Clicking* **Start Design/Check of Structure** *from the* **Design** *menu when there are no design load combinations defined in the model.*

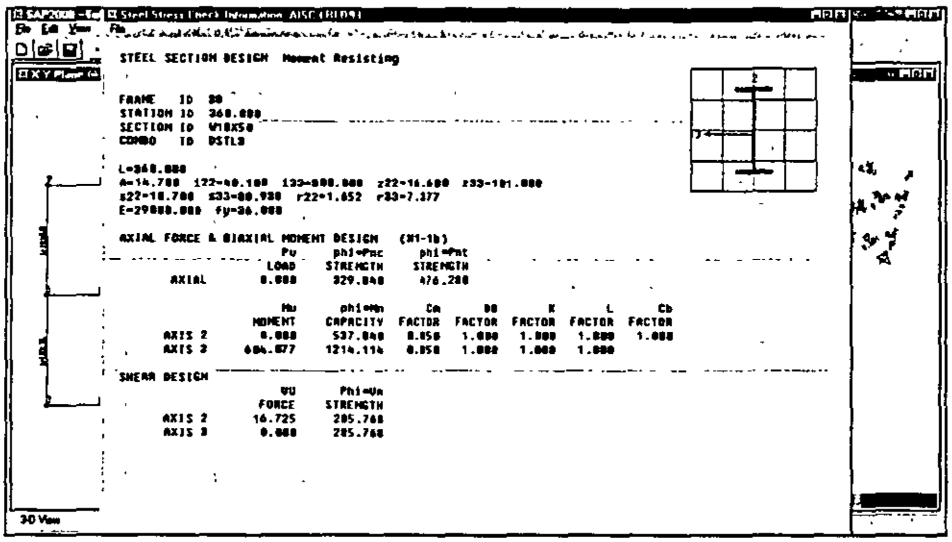


2. **Right click** on a member, say the beam on gridline A, lines 1-2 (element ID 38). See Figure II-1 on page 7 and the screen above. This will open the **Steel Stress Check Information** dialog box showing stress ratios at various stations along the element length for different load combinations. The dialog box also shows information regarding the structural section (W18x50) assigned to the member. The largest stress ratio is highlighted in this box. The dialog box also can show information regarding the **Details** of calculation for design and the element overwrite assignments for **ReDesign** for the selected member.



3.. In this dialog box:

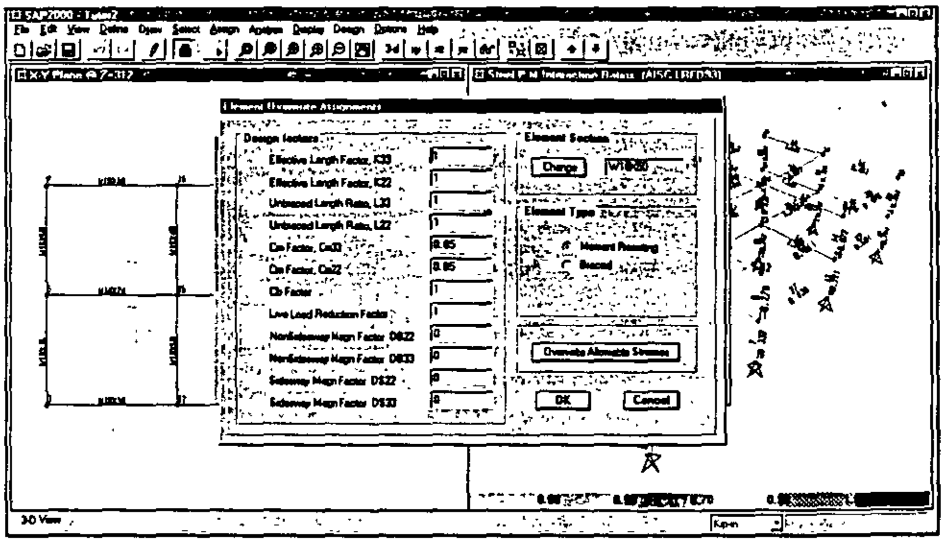
- Select a design check station by clicking once (select the default highlighted one by doing nothing).
- Click on the **Details** button. This will open the **Steel Section Design** detailed information window showing the steel section design parameters including the member identification, geometric parameters, material properties, nominal strength values, the factored member forces for the selected load combination, and other design parameters at that particular station.



- Close the **Steel Section Design** information window.

4. Click on the **ReDesign** button on the **Steel Stress Check Information** dialog box. This will open the **Element Overwrite Assignments** information dialog box showing the input design factors including the K factors, C_m factors, etc. These factors can be edited for redesigning. See screen below. There is also an alternate way of editing the properties of a set of members which will be demonstrated in the next section "Modifying Member Properties of a Group".

    - Click on the **Cancel** button to close the **Element Overwrite Assignments** information dialog box.
    - Click on the **Cancel** button to close the **Steel Stress Check Information** dialog box.



Note: The number of stations (number of segments + 1) used in the design and stress check is set by the user through **Frame** *and* **Output Segment** *menu items from the* **Assign** *menu prior to the analysis phase. The default number of segments is 4 for beams and 2 for columns.*

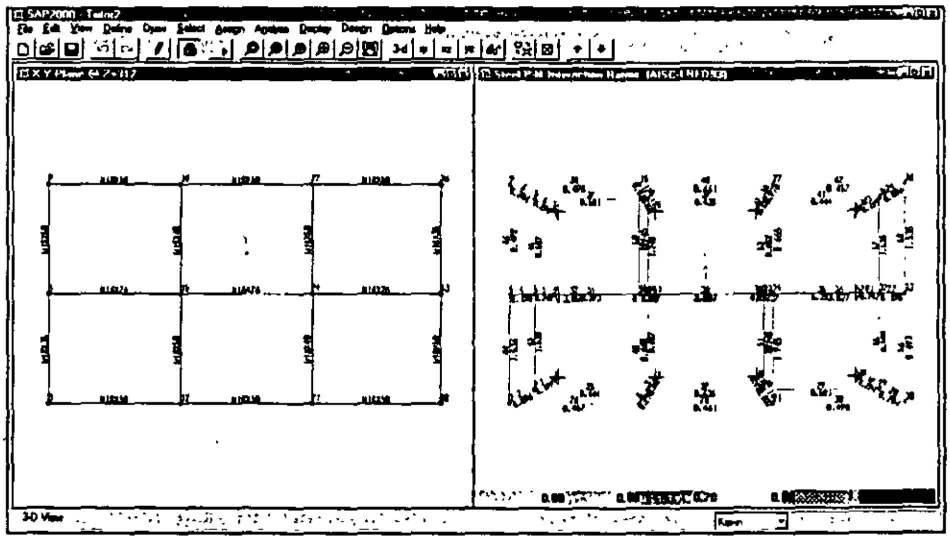
At this point we have analyzed and designed the steel frame and reviewed some of the design information. SAP2000 allows you to interactively change the design code, member properties, remove or add new load combinations, etc. and re-run the analysis and design phases. As a demonstration in the tutorial, we will edit/change a member property for a set of frame members.

Modifying Member Properties

With the analysis and stress check successfully completed, we will now modify the lateral unsupported length of all the beams and girders because the stress ratios are greater than 1 for some of the beams and girders and $l/r > 200$ for many of the beams. Initially, in the design check, the unbraced length of each member was taken to be the member full length in both the major and minor directions, i.e. $l_{22} = l_{33} = L$. However, the secondary beams of the floor structure provide restraints against lateral displacement of the compression flange. Referring to Figure II-1 (page 7) we will take the minor direction unbraced length, l_{22}, of every beam spanning N-S (Y-Y) as 33% of the actual length. Likewise, for beams spanning E-W (X-X), l_{22} is assumed to be 25% of the actual length. In order to make these modifications we will change the view in the right window to make all the beams visible for selection. Notice that this window is currently showing the stress ratios from the previous check in a 3D view.

1. Click on the **2D View (xy)** button on the main toolbar.

2. Click on the **Perspective Toggle** button on the main toolbar.



Now, with all the beams visible, we can select beams and modify their lateral unbraced lengths.

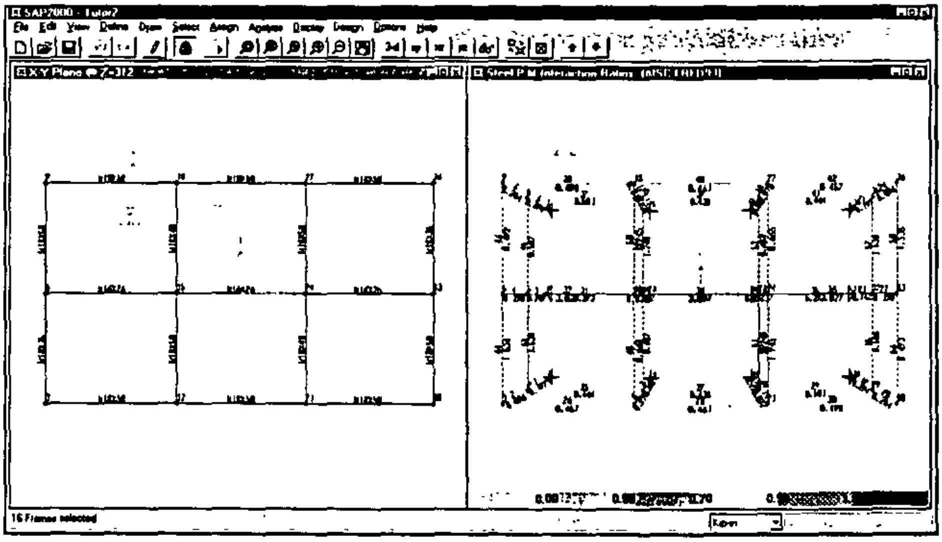
3. To select the N-S (Y-Y) beams do the following:

   - Click the **Set Intersecting Line Select Mode** button on the Floating Toolbar.

- Move the pointer to the left of the beam on line 1, bay A-B.
- Click and hold down the left mouse button.
- While holding, move the pointer to the right intersecting all the beams in the bay between lines A and B. A "rubber-band line" in the Y-Y direction will show the intersecting line.
- Release the left mouse button to select all members that intersect this line. The message area at the bottom-left corner of SAP2000 responds by showing that **8 Frames Selected.**

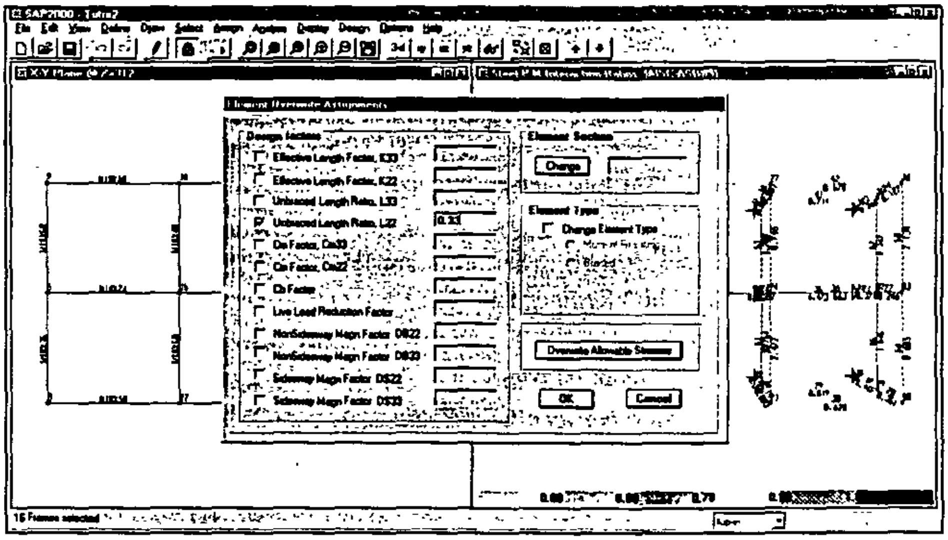
  Note: To select all the N-S beams we have to do this operation (Step 3) twice; once, for all the beams between lines A and B, and once for beams spanning between lines B and C.

The selection of all N-S beams is now complete. The selected members appear as dashed lines.



4. From the **Design** menu, choose the menu item **ReDefine Element Design Data....** This will display the **Element Overwrite Assignments** dialog box. In this dialog box:

   - Check the **Unbraced Length Ratio, L22** box in the **Design factors** area.
   - Replace 1 by 0.33.

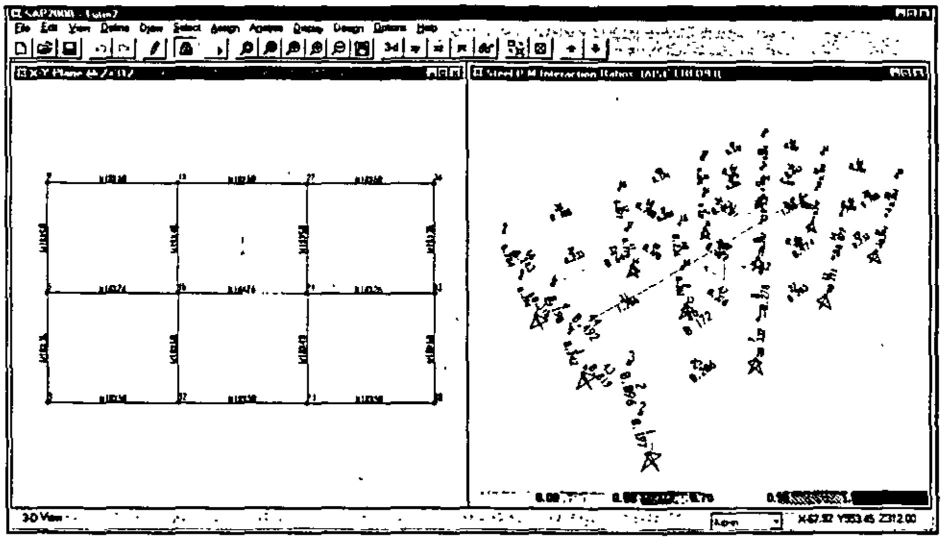


*Note: The **Design Factors** shown in this dialog box are code-dependent.*

- Click on the **OK** button to accept the unbraced length ratio.
- Click on the **Refresh Window** button from the main toolbar. Notice that as a result of using shorter unsupported lengths, the stress ratios in these particular members have decreased significantly.

5. Repeat Steps 3 and 4 to modify all the E-W (X-X) beams, except enter the unbraced length ratio as 0.25.

6. The stress ratios after redefining the unbraced member length can be made more visible in the 3D display. Click **3D View** button on the main toolbar. Compare the following display with the one on page 13.



Selecting Sections Automatically

After changing the unbraced length, most of the beam members are found to be understressed except only 3 members which are overstressed. These three overstressed members are color coded with red. As an exercise, we will select sections for some of these overstressed members automatically.

These three overstressed beams are identified by element numbers 31, 33, and 35. Currently all of these members have a W18x35 section. If we gradually replace these sections with larger ones, the stress ratio can be made close to 1 but less than 1. W18x50 section will satisfy the requirements. Instead we will do an exercise showing automatic selection of members.

In the right side window, where the steel stress ratios are displayed in a 3D view:

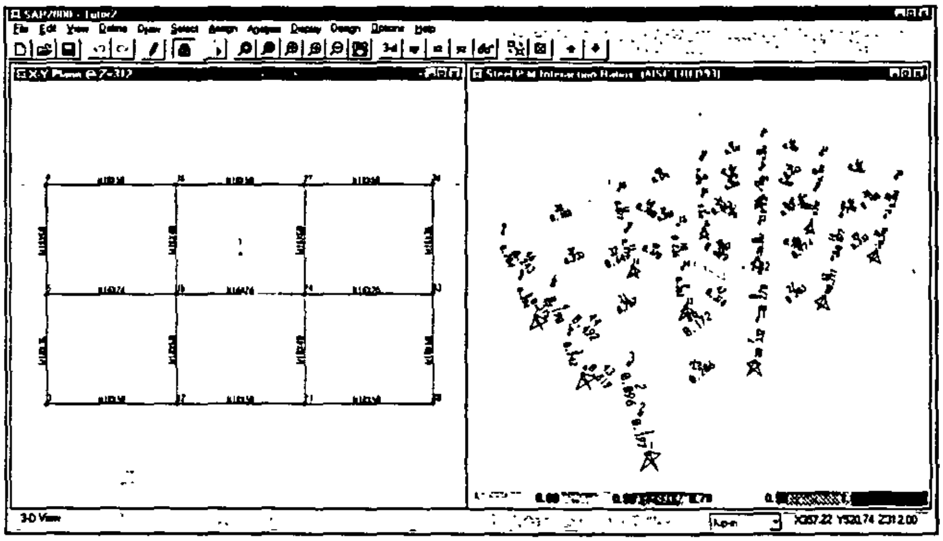
- Select these three overstressed members, which are color coded with red, by clicking on them one by one.
- Unlock the model by clicking on the **Lock/Unlock Model** from the mail toolbar.
  - This will prompt a dialog box asking **"Unlocking model will delete analysis results! OK to delete?"**. Click **OK**.
- Click on the **Frame Sections...** in the **Define** menu. This will bring up a **Define Frame Sections** dialog box. This dialog box shows the available sections in the model. We will make some more sections available for the model so that the

program can choose the automatically selected section from a wider group. Since the W18x35 is too big for some members, we will add some smaller sections, especially wide flange sections with 12 inch depth, for the domain of AUTO section. To achieve this, in this dialog box:

- Click on the **Import** pull down arrow.
- Scroll through the sections and choose **Import I/Wide Flange**. This will bring a **Section Property File** dialog box. In this dialog box, the default section property files are listed.
- Select the default section property file **Section.pro**.
- Click **Open** to open the file Sections.pro. This will show a list of available Wide Flange sections.
- Scroll down through the sections and select **W12x96** by clicking. Scroll down through the sections again and select **W12x14** by holding down the Shift Key and clicking. This will select all the sections ranging from W12x96 to W12x14.
- Click **OK** to close the Sections.pro file. The response will be a display of information in the **I/Wide Flange Sections** dialog box about the last selected section, i.e., **W12x14**.
- Click **OK** in the **I/Wide Flange Sections** dialog box. This will complete importing the newly selected sections into the model from the database. The imported sections are added to the **Frame Sections** list in the **Define Frame Sections** dialog box.
- Click on the **Add** pull down arrow in the **Define Frame Sections** dialog box.
- Scroll through the sections and choose **Add Auto Select**. This will bring an **Auto Selection Sections** dialog box. In this dialog box, the default domain of the Auto Sections is listed. You can edit the list by adding and deleting new sections. Scroll through the sections down to W14x132 and select it by clicking. The Remove button is highlighted. Click on the **Remove** button to remove this specific section from the domain of the auto section because the W14x132 is specifically used in this model for columns. The default name of the auto section is given as AUTO1. Click **OK** to accept the default name and the list of sections.
- Click the **OK** button to close the **Define Frame Sections** dialog box.

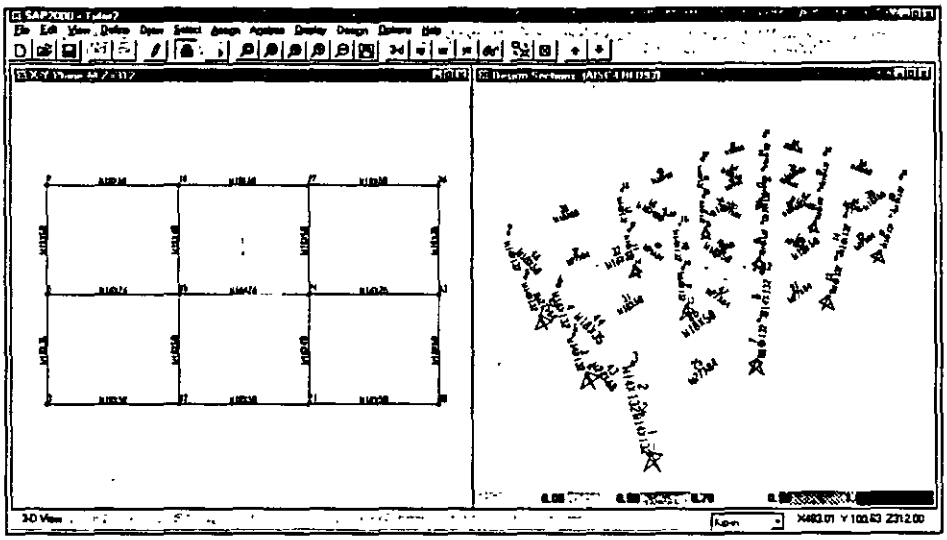
• Click on the **Assign** menu, select the **Frame** menu item, and then select **Sections...**. This will open the **Define Frame Sections** dialog box. In this dialog box you select **AUTO1** and then click **OK**. This will change the display recognizing that the selected members have **AUTO1** section.

- Reanalyze the model by clicking on the **Run Analysis** button on the main toolbar. Click **OK** to close the top window.
- Recalculate the stress ratio by clicking on the **Start Design/Check of Structure** menu item in the **Design** menu. This will respond by displaying new stress ratios on a 3D View in the right window.



Notice that as a result of changing the section, the stress ratios for those particular beams are changed. To see the difference, compare this display with the one on page 19.

- To see the newly selected sections, click on the **Display Design Info...** menu item in the **Design** menu. This will bring up the **Display Design Results** dialog box. In this dialog box, select the **Design Input** option button, accept the **Design Sections** from the **Design Input** drop down list item, and click **OK**. This will show the newly selected sections on the 3D view at the right window.



Note: The displayed sections are the design sections. Analysis sections can also be displayed by clicking on the **Set Elements** *button on the main toolbar and clicking on the* **Sections** *check box for* **Frames** *on the* **Set Elements** *dialog box.*

Re-analyzing with Updated Elements

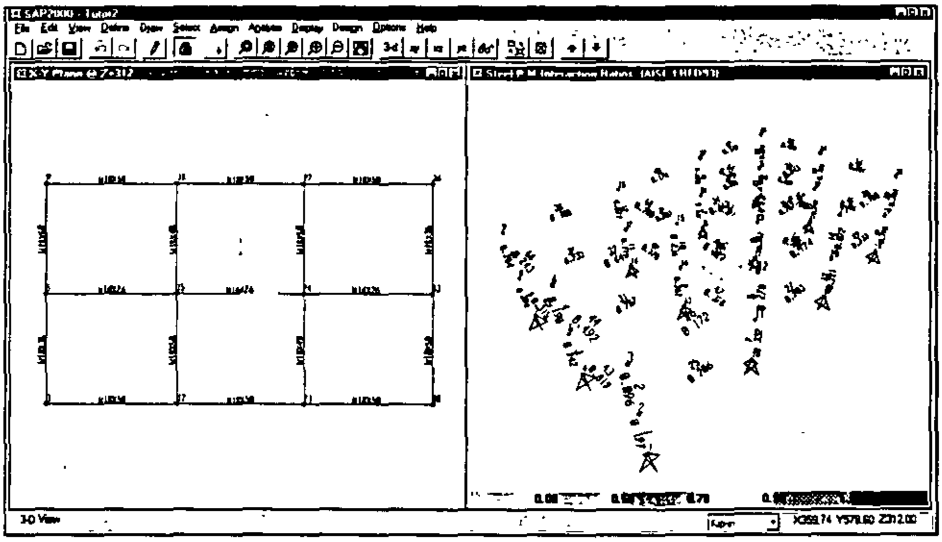
It is important to realize that changes made to member section properties in the stress/design phase are **not** automatically reflected in the analysis results. These changes are only local to the post-processing phase unless a re-run of the analysis, with updated elements, is requested by the user. In other words, overwriting the section properties only affects the stress values and not the factored element forces obtained in the analysis preceding such changes. The redistribution of member forces due to change of stiffness (revision of section properties) is effected in a re-run of analysis. We need to refresh the analysis model, reanalyze the model, and re-design the structure. To do this:

- In the **Design** menu click on the menu item **Update Analysis Sections.** This will prompt a dialog box asking **"Updating Analysis Section will unlock model! OK to update?"**. Click **OK**.
- From the **Analyze** menu, choose **Run.** This will immediately start the analysis procedure. A top window is opened in which various phases of analysis are progressively displayed. The results will obviously differ from those produced in the initial analysis because of the change of section properties we made in the design stage. Click **OK** to close the top window.

- Click on the **Start Design/Check of Structures** from the **Design** menu. This will redesign the structure and display the new stress ratios.
- Click on the **Replace Auto w/Optimal Sections** menu item from the **Design** menu.
  - This will ask "**Replacing Analysis Sections with Auto-Selected Sections will unlock model! OK to replace?**". Click **OK** to replace.

Note: Once you are satisfied with the selected sections, selecting the **Replace Auto w/Optimal Sections** *menu item will permanantly replace the auto sections with the current design sections. This effectively replaces the analysis sections with the optimal design sections and removes the auto tag. So selecting this menu item is one of the last things you should do.*

You can see the difference after re-running the design based on the latest analysis results.



Concluding Remarks

We have come to the end of this tutorial on the SAP2000 steel design options. The intent has been to highlight and demonstrate a few of the basic features in order to open up the path for you to explore and use the more advanced options. For more information on various topics consult the on-line Help provided.

Chapter III

Design Algorithms

This chapter outlines various aspects of the steel check and design procedures that are used by the SAP2000 program. The steel design and check may be performed according to one of the following codes of practice.

- American Institute of Steel Construction's "Allowable Stress Design and Plastic Design Specification for Structural Steel Buildings", **AISC-ASD** (AISC 1989).
- American Institute of Steel Construction's "Load and Resistance Factor Design Specification for Structural Steel Buildings", **AISC-LRFD** (AISC 1994).
- Canadian Institute of Steel Construction's "Limit States Design of Steel Structures", **CAN/CSA-S16.1-94** (CISC 1995).
- British Standards Institution's "Structural Use of Steelwork in Building", **BS 5950** (BSI 1990).
- European Committee for Standardization's "Eurocode 3: Design of Steel Structures — Part 1.1: General Rules and Rules for Buildings", **ENV 1993-1-1** (CEN 1992).

Details of the algorithms associated with each of these codes as implemented in SAP2000 are described in subsequent chapters. However, this chapter provides a background which is common to all the design codes.

It is assumed that the user has an engineering background in the general area of structural steel design and familiarity with at least one of the above mentioned design codes.

For referring to pertinent sections of the corresponding code, a unique prefix is assigned for each code. For example, all references to the AISC-ASD89 code carry the prefix of "**ASD**". Similarly,

- References to the AISC-LRFD93 code carry the prefix of "**LRFD**"
- References to the Canadian code carry the prefix of "**CISC**"
- References to the British code carry the prefix of "**BS**"
- References to the Eurocode carry the prefix of "**EC3**"

Design Load Combinations

The design load combinations are the various combinations of the load cases for which the structure needs to be designed/checked. The load combination factors to be used vary with the selected design code. The load combination factors are applied to the forces and moments obtained from the associated load cases and are then summed to obtain the factored design forces and moments for the load combination.

For multi-valued load combinations involving response spectrum, time history, moving loads and multi-valued combinations (of type enveloping, square-root of the sum of the squares or absolute) where any correspondence between interacting quantities is lost, the program automatically produces multiple sub combinations using maxima/minima permutations of interacting quantities. Separate combinations with negative factors for response spectrum cases are not required because the program automatically takes the minima to be the negative of the maxima for response spectrum cases and the above described permutations generate the required sub combinations.

When a design combination involves only a single multi-valued case of time history or moving load, further options are available. The program has an option to request that time history combinations produce sub combinations for each time step of the time history. Also an option is available to request that moving load combinations produce sub combinations using maxima and minima of each design quantity but with corresponding values of interacting quantities.

For normal loading conditions involving static dead load, live load, wind load, and earthquake load, and/or dynamic response spectrum earthquake load, the program has built-in default loading combinations for each design code. These are based on the code recommendations and are documented for each code in the corresponding chapters. For other loading conditions involving moving load, time history, pattern live loads, separate consideration of roof live load, snow load, etc., the user must define design loading combinations either in lieu of or in addition to the default design loading combinations.

The default load combinations assume all static load cases declared as dead load to be additive. Similarly, all cases declared as live load are assumed additive. However, each static load case declared as wind or earthquake, or response spectrum cases, are assumed to be non additive with each other and produce multiple lateral load combinations. Also wind and static earthquake cases produce separate loading combinations with the sense (positive or negative) reversed. If these conditions are not correct, the user must provide the appropriate design combinations.

The default load combinations are included in design if the user requests them to be included or if no other user defined combination is available for steel design. If any default combination is included in the design, then all default combinations will automatically be updated by the program any time the design code is changed or if static or response spectrum load cases are modified.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

The user is cautioned that if moving load or time history results are not requested to be recovered in the analysis for some or all frame members then the effects of these loads will be assumed to be zero in any combination that includes them.

Design and Check Stations

For each load combination, each element is designed or checked at a number of locations along the length of the element. The locations are based on equally spaced segments along the clear length of the element. The number of segments in an element is requested by the user before the analysis is made. The user can refine the design along the length of an element by requesting more segments.

Element Stress Ratios

For each element, the axial force/biaxial moment stress ratios as well as shear stress ratios are calculated for each station along the length of the member for each load combination. The actual member stress components and corresponding allowable stresses are calculated. Then, the stress ratios are evaluated according to the code. The controlling compression and/or tension stress ratio is then obtained, along with the corresponding identification of the station, load combination, and code-equation. A stress ratio greater than 1.0 indicates an overstress or exceeding a limit state.

P-Δ Effects

The SAP2000 design algorithms require that the analysis results include the P-Δ effects. The P-Δ effects are considered differently for "braced" or "nonsway" and "unbraced" or "sway" columns or frames. For the braced columns, the effect of P-Δ is limited to "individual member stability". For unbraced frames, "lateral drift effect" should be considered in addition to individual member stability effect.

For the individual member stability effects, the moments are magnified with moment magnification factors as in the AISC-LRFD code or are considered directly in the design equations as in the Canadian, British, and European codes. .

For lateral drift effects of unbraced or sway frames, SAP2000 assumes that the amplification is already included in the results because P-Δ effects are considered.

Users should be aware that the default analysis option in SAP2000 is OFF for P-Δ effect. The default number of iterations for P-Δ analysis is 1. **The user should turn the P-Δ analysis ON and set the maximum number of iterations for the analysis.** For further reference, the user is referred to "SAP2000 Reference Manual" (CSI 1997b).

The user is also cautioned that SAP2000 currently considers P-Δ effects due to axial loads in frame members only. Forces in other types of elements do not contribute to this effect. If significant forces are present in other types of elements, for example, large axial loads in shear walls modeled as shell elements, then the additional forces computed for P-Δ will be inaccurate.

Element Unsupported Lengths

To account for column slenderness effects the column unsupported lengths are required. The two unsupported lengths are l_{33} and l_{22}. See Figure III-1. These are the lengths between support points of the element in the corresponding directions. The length l_{33} corresponds to instability about the 3-3 axis, and l_{22} corresponds to instability about the 2-2 axis. The length l_{22} is also used for lateral-torsional buckling caused by major direction bending (i.e., about the 3-3 axis). See Figure III-2 for correspondence between the SAP2000 axes and the axes in the design codes.

Normally, the unsupported element length is equal to the length of the element, i.e., the distance between END-I and END-J of the element. See Figure III-1. The program, however, allows users to assign several elements to be treated as a single member for design. This can be done differently for major and minor bending. Therefore, extraneous joints as shown in Figure III-3 that affect the unsupported length of an element are automatically taken into consideration.

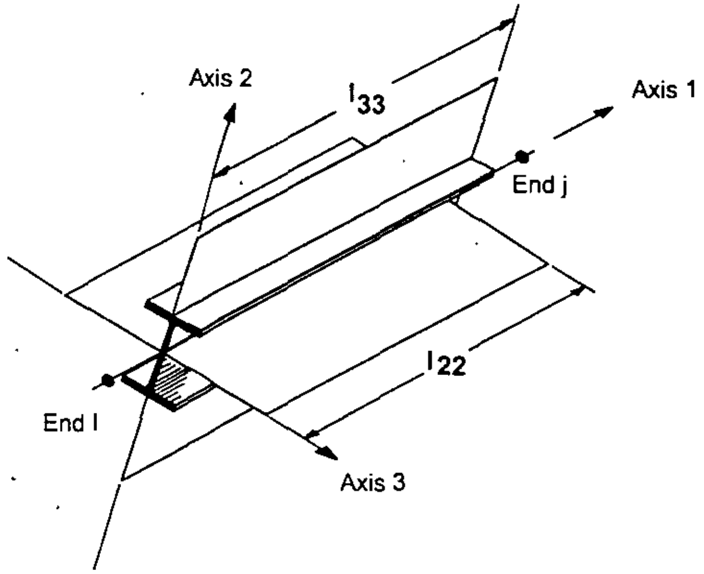


Figure III-1
Major and Minor Axes of Bending

In determining the values for l_{22} and l_{33} of the elements, the program recognizes various aspects of the structure that have an effect on these lengths, such as member connectivity, diaphragm constraints and support points. The program automatically locates the element support points and evaluates the corresponding unsupported element length.

Therefore, the unsupported length of a column may actually be evaluated as being greater than the corresponding element length. If a beam frames into only one direction of the column, the beam is assumed to give lateral support only in that direction.

The user has options to specify the unsupported lengths of the elements on an element-by-element basis.

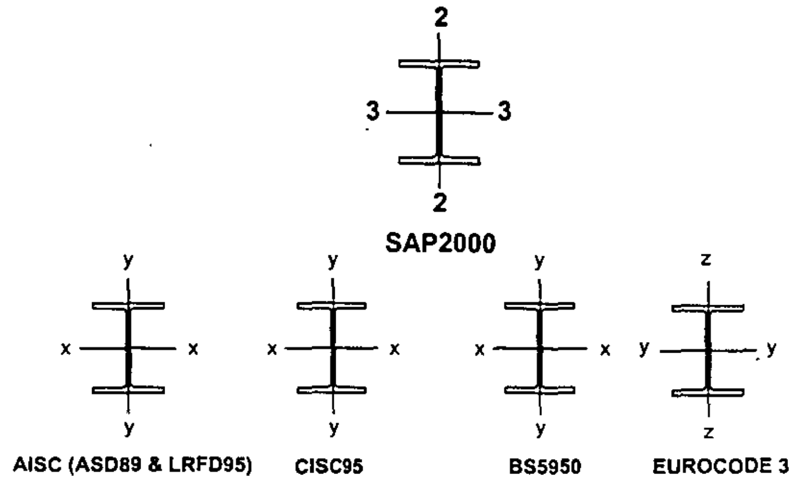


Figure III-2
Correspondence between SAP2000 Axes and Code Axes

Effective Length Factor (*K*)

The column K-factor algorithm has been developed for building-type structures, where the columns are vertical and the beams are horizontal, and the behavior is basically that of a moment-resisting nature for which the K-factor calculation is relatively complex. For the purpose of calculating K-factors, the elements are identified as columns, beams and braces. All elements parallel to the Z-axis are classified as columns. All elements parallel to the X-Y plane are classified as beams. The rest are braces.

The beams and braces are assigned K-factors of unity. In the calculation of the K-factors for a column element, the program first makes the following four stiffness summations for each joint in the structural model:

$$S_{cx} = \sum \left(\frac{E_c I_c}{L_c} \right)_x \qquad S_{bx} = \sum \left(\frac{E_b I_b}{L_b} \right)_x$$

$$S_{cy} = \sum \left(\frac{E_c I_c}{L_c} \right)_y \qquad S_{by} = \sum \left(\frac{E_b I_b}{L_b} \right)_y$$

where the x and y subscripts correspond to the global X and Y directions and the c and b subscripts refer to column and beam. The local 2-2 and 3-3 terms EI_{22}/l_{22} and EI_{33}/l_{33} are rotated to give components along the global X and Y directions to form the $(EI/l)_x$ and $(EI/l)_y$ values. Then for each column, the joint summations at END-I and the END-J of the member are transformed back to the column local 1-2-3 coordinate system and the G-values for END-I and the END-J of the member are calculated about the 2-2 and 3-3 directions as follows:

$$G^I{}_{22} = \frac{S^I{}_{c22}}{S^I{}_{b22}} \qquad G^J{}_{22} = \frac{S^J{}_{c22}}{S^J{}_{b22}}$$

$$G^I{}_{33} = \frac{S^I{}_{c33}}{S^I{}_{b33}} \qquad G^J{}_{33} = \frac{S^J{}_{c33}}{S^J{}_{b33}}$$

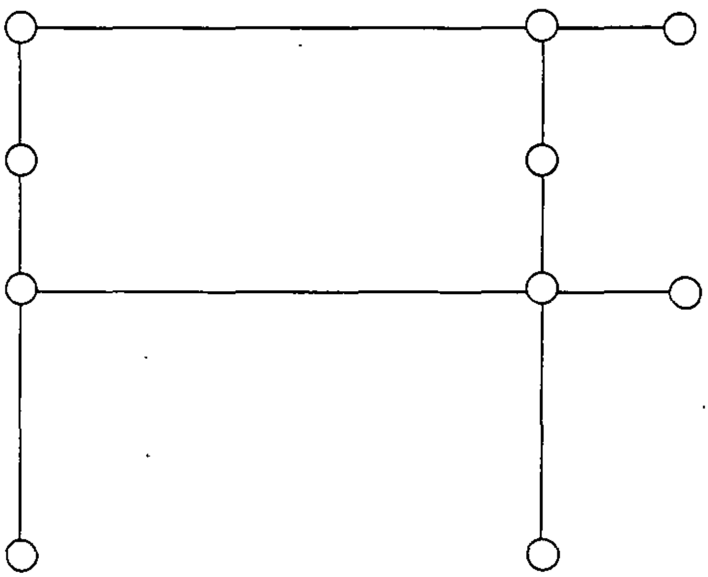


Figure III-3
Unsupported Lengths are not Affected by Intermediate Nodal Points

If a rotational release exists at a particular end (and direction) of an element, the corresponding value is set to 10.0. If all degrees of freedom for a particular joint are deleted, the G-values for all members connecting to that joint will be set to 1.0 for the end of the member connecting to that joint. Finally, if GI and GJ are known for a particular direction, the column K-factor for the corresponding direction is calculated by solving the following relationship for α:

$$\frac{\alpha^2 G^I G^J - 36}{6(G^I + G^J)} = \frac{\alpha}{\tan\alpha}$$

from which $K = \pi / \alpha$. This relationship is the mathematical formulation for the evaluation of K factors for moment-resisting frames assuming sidesway to be uninhibited. For other structures, such as braced frame structures, trusses, space frames, transmission towers, etc., the K-factors for all members are usually unity and should be set so by the user. The following are some important aspects associated with the column K-factor algorithm:

- An element that has a pin at the joint under consideration will not enter the stiffness summations calculated above. An element that has a pin at the far end from the joint under consideration will contribute only 50% of the calculated EI value. Also, beam elements that have no column member at the far end from the joint under consideration, such as cantilevers, will not enter the stiffness summation.
- If there are no beams framing into a particular direction of a column element, the associated G-value will be infinity. If the G-value at any one end of a column for a particular direction is infinity, the K-factor corresponding to that direction is set equal to unity.
- If rotational releases exist at both ends of an element for a particular direction, the corresponding K-factor is set to unity.
- The automated K-factor calculation procedure can occasionally generate artificially high K-factors, specifically under circumstances involving skewed beams, fixed support conditions, and under other conditions where the program may have difficulty recognizing that the members are laterally supported and K-factors of unity are to be used.
- All K-factors produced by the program can be overwritten by the user. These values should be reviewed and any unacceptable values should be replaced.

Choice of Input Units

English as well as SI and MKS metric units can be used for input. But the codes are based on a specific system of units. All equations and descriptions presented in the subsequent chapters correspond to that specific system of units unless otherwise noted. For example, AISC-ASD code is published in kip-inch-second units. By default, all equations and descriptions presented in the chapter "Check/Design for AISC ASD89" correspond to kip-inch-second units. However, any system of units can be used to define and design the structure in SAP2000.

Chapter IV

Check/Design for AISC-ASD89

This chapter describes the details of the structural steel design and stress check algorithms that are used by SAP2000 when the user selects the AISC-ASD89 design code (AISC 1989). Various notations used in this chapter are described in Table IV-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

In the evaluation of the axial force/biaxial moment capacity ratios at a station along the length of the member, first the actual member force/moment components and the corresponding capacities are calculated for each load combination. Then the capacity ratios are evaluated at each station under the influence of all load combinations using the corresponding equations that are defined in this section. The controlling capacity ratio is then obtained. A capacity ratio greater than 1.0 indicates overstress. Similarly, a shear capacity ratio is also calculated separately.

English as well as SI and MKS metric units can be used for input. But the code is based on Kip-Inch-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Kip-Inch-Second** units unless otherwise noted.

A	=	Cross-sectional area, in^2
A_f	=	Area of flange , in^2
A_g	=	Gross cross-sectional area, in^2
A_{v2}, A_{v3}	=	Major and minor shear areas, in^2
A_w	=	Web shear area, dt_w, in^2
C_b	=	Bending Coefficient
C_m	=	Moment Coefficient
C_w	=	Warping constant, in^6
D	=	Outside diameter of pipes, in
E	=	Modulus of elasticity, ksi
F_a	=	Allowable axial stress, ksi
F_b	=	Allowable bending stress, ksi
F_{b33}, F_{b22}	=	Allowable major and minor bending stresses, ksi
F_{cr}	=	Critical compressive stress, ksi
F'_{e33}	=	$\dfrac{12\pi^2 E}{23\left(K_{33} l_{33} / r_{33}\right)^2}$
F'_{e22}	=	$\dfrac{12\pi^2 E}{23\left(K_{22} l_{22} / r_{22}\right)^2}$
F_v	=	Allowable shear stress, ksi
F_y	=	Yield stress of material, ksi
K	=	Effective length factor
K_{33}, K_{22}	=	Effective length K-factors in the major and minor directions
M_{33}, M_{22}	=	Major and minor bending moment moments in member, kip-in
P	=	Axial force in member, kips
P_e	=	Euler buckling load, kips
S	=	Section modulus, in^3
S_{33}, S_{22}	=	Major and minor section moduli, in^3
V_2, V_3	=	Shear forces in major and minor directions, kips
b	=	Nominal dimension of longer leg of angles, in $(b_f - 2t_w)$ for welded and $(b_f - 3t_f)$ for rolled BOX (TS) sections, in
b_f	=	Flange width, in
d	=	Overall depth of member, in

Table IV-1
AISC-ASD Notations

f_a	=	Axial stress either in compression or in tension, ksi
f_b	=	Normal stress in bending, ksi
f_{b33}, f_{b22}	=	Normal stress in major and minor direction bending, ksi
f_v	=	Shear stress, ksi
f_{v2}, f_{v3}	=	Shear stress in major and minor direction bending, ksi
h_c	=	Clear distance between flanges less fillets, in assumed $(d-2k)$ for rolled sections and $(d-2t_f)$ for welded sections
k	=	Distance from outer face of flange to web toe of fillet, in
k_c	=	Parameter used for classification of sections, $\dfrac{4}{[h/t_w]^{0.46}}$ if $h/t_w > 70$, 1 if $h/t_w \leq 70$.
l_{33}, l_{22}	=	Major and minor direction unbraced member lengths, in
l_c	=	Critical length, in
r	=	Radius of gyration, in
r_{33}, r_{22}	=	Radii of gyration in the major and minor directions, in
r_z	=	Minimum Radius of gyration for angles, in
t	=	Thickness, in
t_f	=	Flange thickness, in
t_w	=	Web thickness, in
λ	=	Slenderness parameter

Table IV-1

AISC-ASD Notations (cont.)

Design Loading Combinations

The design load combinations are the various combinations of the prescribed load cases for which the structure needs to be checked. For the AISC-ASD89 code, if a structure is subjected to dead load (DL) and live load (LL) only, the stress check may need only one load combination, namely DL + LL (AISC A.4). However, if in addition to the dead and live loads, the structure is subjected to wind (WL) and earthquake (EL) induced loads, and considering that wind and earthquake forces are reversible, then the following load combinations may have to be considered for the design of steel frames (ASD A.4):

DL
DL + LL (ASD A4.1)

DL ± WL
DL + LL ± WL (ASD A4.1)

DL ± EL
DL + LL ± EL (ASD A4.1)

These are also the default design load combinations in SAP2000 whenever the AISC-ASD89 code is used.

When designing for combinations involving earthquake and wind loads, allowable stresses is increased by 33% of the allowable value (ASD A5.2).

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

Classification of Sections

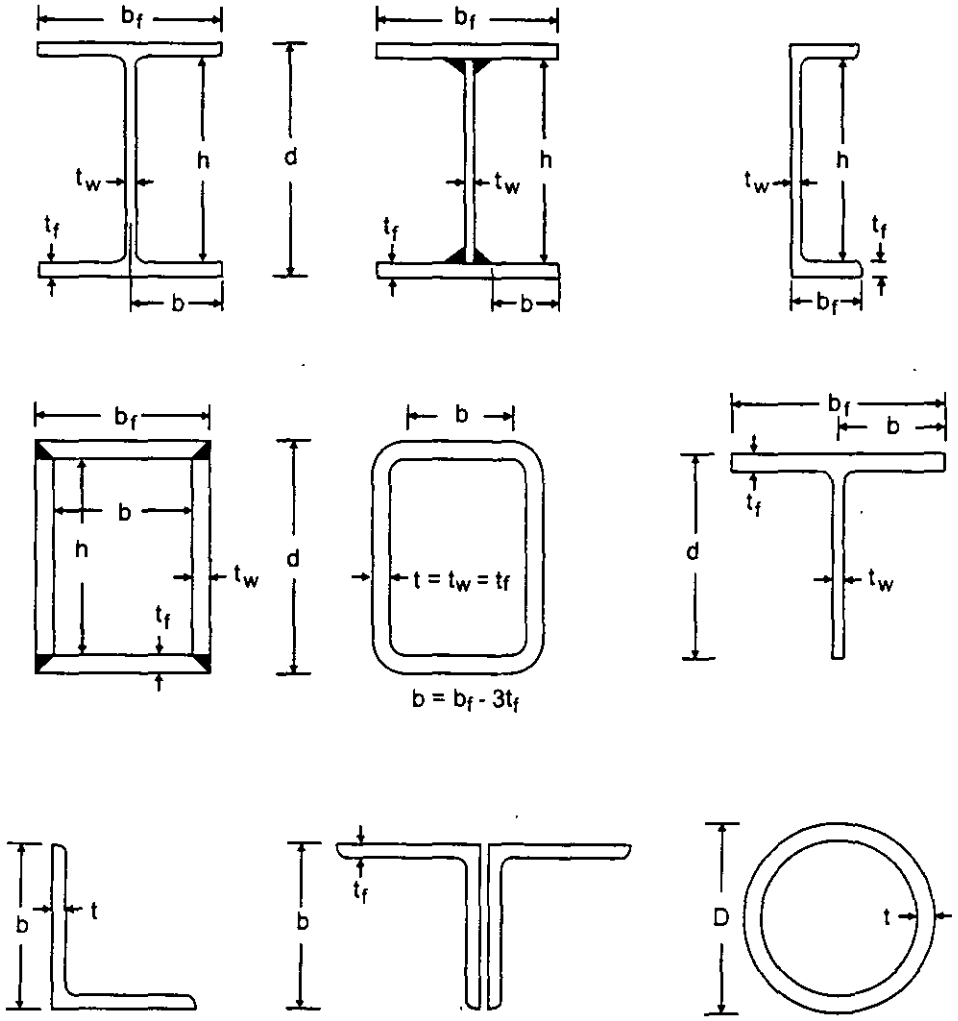
The allowable stresses for axial compression and flexure are dependent upon the classification of sections as either Compact, Noncompact, or Slender. SAP2000 classifies the individual members according to Table IV-2 (ASD B5.1). The variables related to the section dimensions are shown in Figure IV-1 and the nomenclature of the other variables is shown in Table IV-1.

If the section dimensions satisfy the limits shown in the table, the section is classified as either Compact or Noncompact. **If the limits for Noncompact sections are not met, the section is classified as Slender. Currently SAP2000 does not check stresses for Slender sections.**

Section Description	Ratio Checked	Compact Section	Noncompact Section	Noncompact (Compression)
GENERAL	—	Assumed Noncompact		
RECTANGLE	—	Assumed Compact		
I-SHAPE	$b_f/2t_f$ (rolled)	$\leq 65/\sqrt{F_y}$	$\leq 95/\sqrt{F_y}$	$\leq 95/\sqrt{F_y}$
	$b_f/2t_f$ (welded)	$\leq 65/\sqrt{F_y}$	$\leq 95/\sqrt{F_y/k_c}$	$\leq 95/\sqrt{F_y}$
	d/t_w	$\leq \frac{640}{\sqrt{F_y}}(1-3.74\frac{f_a}{F_y})$ for $f_a/F_y \leq 0.16$; $\leq 257/\sqrt{F_y}$ for $f_a/F_y > 0.16$	—	$\leq 253/\sqrt{F_y}$
	h/t_w	—	$\leq 760/\sqrt{F_b}$	—
BOX	b/t_f	$\leq 190/\sqrt{F_y}$ (rolled)	$\leq 238/\sqrt{F_y}$	$\leq 253/\sqrt{F_y}$
	d/t_w	As for I-shapes	—	—
	h/t_w	—	As for I-shapes	$\leq 253/\sqrt{F_y}$
CHANNEL	b_f/t_f b_f/t_f h/t_w	As for I-section, for $P \leq 0$ Not applicable, for $P = 0$ Not applicable	$\leq 95/\sqrt{F_y}$ $\leq 95/\sqrt{F_y}$ As for I-shapes	$\leq 95/\sqrt{F_y}$
T-SHAPE	$b_f/2t_f$ d/t_w	Not applicable Not applicable	$\leq 95/\sqrt{F_y}$ $\leq 127/\sqrt{F_y}$	$\leq 95/\sqrt{F_y}$ $\leq 127/\sqrt{F_y}$
ANGLE	b/t	Not applicable	$\leq 76/\sqrt{F_y}$	$\leq 76/\sqrt{F_y}$
ROUND BAR	—	Assumed Compact		
PIPE	D/t	$\leq 3300/F_y$	—	$\leq 3300/F_y$
DOUBLE ANGLES	b/t	Not applicable	$\leq 95/\sqrt{F_y}$ (sep.) $\leq 127/\sqrt{F_y}$ (con.)	$\leq 95/\sqrt{F_y}$ (sep.) $\leq 127/\sqrt{F_y}$ (con.)

Table IV-2

Classification of Sections According to AISC-ASD
Limiting Width-Thickness Ratios for Compression Elements



AISC-ASD89 : Axes Conventions

3-3 is the cross-section axis parallel to the flanges or the smaller leg in angle sections.

2-2 is the cross-section axis perpendicular to the flanges or the smaller leg in angle sections.

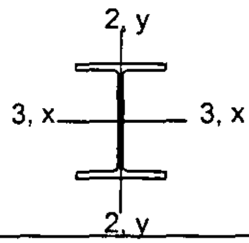


Figure IV-1
AISC-ASD Definition of Geometric Properties

Calculation of Actual Stresses

The stresses are calculated at each of the previously defined stations. The member stresses that are calculated for each load combination are:

$$f_a = P/A$$
$$f_{b33} = M_{33}/S_{33}$$
$$f_{b22} = M_{22}/S_{22}$$
$$f_{v2} = V_2/A_{v2}$$
$$f_{v3} = V_3/A_{v3}$$

Calculation of Allowable Stresses

The allowable stresses in compression, tension, bending, and shear are computed for Compact and Noncompact sections according to the following subsections.

For Slender sections and any singly symmetric and unsymmetric sections requiring consideration of flexural-torsional and torsional buckling, a reduction factor in the allowable stress may be applicable. The user must separately investigate this reduction if such elements are used.

*If the user specifies nonzero allowable stresses for one or more elements in the SAP2000 "Redefine Element Design Data", **those values will override all the allowable stresses calculated by the program** as defined in the following subsections.*

Allowable Stress in Compression

The allowable axial compressive stress value, F_a, for compact or non-compact sections, depends on the slenderness ratio Kl/r and a corresponding critical value, C_c. $\frac{Kl}{r}$ is the larger of $\frac{K_{33}\, l_{33}}{r_{33}}$ and $\frac{K_{22}\, l_{22}}{r_{22}}$ and $C_c = \sqrt{(2\pi^2 E) / F_y}$. F_a is evaluated as follows:

- If $\frac{Kl}{r} \le C_c$,

$$F_a = \frac{\left\{1.0 - \frac{(Kl/r)^2}{2C_c^2}\right\} F_y}{\frac{5}{3} + \frac{3(Kl/r)}{8\,C_c} - \frac{(Kl/r)^3}{8C_c^3}} . \qquad \text{(ASD E2-1)}$$

- If $\frac{Kl}{r} > C_c$,

$$F_a = \frac{12\pi^2 E}{23(Kl/r)^2}. \qquad \text{(ASD E2-2)}$$

For single angles r_z is used in place of r_{22} and r_{33}. For members in compression, if $\frac{Kl}{r}$ is greater than 200, a message is printed.

Allowable Stress in Tension

The allowable axial tensile stress value F_a is assumed to be $0.60\,F_y$. **It should be noted that net section checks are not made.** For members in tension, if l/r is greater than 300, a message to that effect is printed.

$$F_a = 0.6F_y \qquad \text{(ASD D1)}$$

Allowable Stress in Bending

The allowable bending stress depends on the following criteria: the geometric shape of the cross-section, the axis of bending, the compactness of the section, and a slenderness parameter.

I-sections, C-sections, T-sections, Angles and Double angles

For all I-sections, C-sections, T-sections, Angles, and Double angles, the slenderness parameter is taken as the laterally unbraced length, l, measured compared to a critical length, l_c. The critical length is defined as

$$l_c = \min\left\{\frac{76b_f}{\sqrt{F_y}}, \frac{20000A_f}{dF_y}\right\} \qquad \text{(ASD F1-2)}$$

Major Axis of Bending

If l_{33} is less than l_c, the major allowable bending stress is taken as

$$F_{b33} = 0.66\,F_y \text{ (For Compact section),} \qquad \text{(ASD F1-1)}$$

$$F_{b33} = 0.60\,F_y \text{ (For Noncompact section),} \qquad \text{(ASD F1-5)}$$

and if the unbraced length l_{33} is greater than l_c, then for both Compact and Noncompact I-sections the allowable bending stress depends on the $\frac{l_{33}}{r_T}$ ratio.

For $\sqrt{\frac{102 \times 10^3 C_b}{F_y}} \leq \frac{l_{33}}{r_T} \leq \sqrt{\frac{510 \times 10^3 C_b}{F_y}}$,

$$F_{b33} = \left[\frac{2}{3} - \frac{F_y (l_{33} / r_T)^2}{1530 \times 10^3 C_b} \right] F_y \leq 0.60 F_y , \qquad \text{(ASD F1-6)}$$

and for $\frac{l_{33}}{r_T} > \sqrt{\frac{510 \times 10^3 C_b}{F_y}}$,

$$F_{b33} = \left[\frac{170 \times 10^3 C_b}{(l_{33} / r_T)^2} \right] \leq 0.60 F_y , \qquad \text{(ASD F1-7)}$$

and F_{b33} is not less than that given by the following formula:

$$F_{b33} = \frac{12 \times 10^3 C_b}{l_{33} \left(d / A_f \right)} \leq 0.6 F_y \qquad \text{(ASD F1-8)}$$

For C-sections, only formula ASD F1-8 is used.

In all the above cases,

r_T is the radius of gyration of a section comprising the compression flange and 1/3 the compression web taken about an axis in the plane of the web,

$$C_b = 1.75 + 1.05 \left(\frac{M_a}{M_b} \right) + 0.3 \left(\frac{M_a}{M_b} \right)^2 \leq 2.3, \qquad \text{(ASD F1.3)}$$

M_a and M_b are the end moments of any unbraced segment of the member and M_a is numerically less than M_b; M_a / M_b being positive for double curvature bending and negative for single curvature bending. Also, if any moment within the segment is greater than M_b, C_b is taken as 1.0. Also, C_b is taken as 1.0 for cantilevers and frames braced against joint translation (ASD F1). SAP2000 defaults C_b to 1.0 if the unbraced length, l, of the member is redefined by the user (i.e. it is not equal to the length of the member). The user can overwrite the value of C_b for any member by specifying it.

Minor Axis of Bending

The minor direction allowable bending stress F_{b22} is taken as

$$F_{b22} = 0.60\, F_y\,, \qquad \text{(ASD F2-2)}$$

except in the case of Compact I-sections, where it is taken as

$$F_{b22} = 0.75\, F_y\,. \qquad \text{(ASD F2-1)}$$

Box Sections and Rectangular Tubes

For all Box sections and Rectangular tubes, the slenderness parameter is taken as the laterally unbraced length, l, measured compared to a critical length, l_c. The critical length is defined as

$$l_c = \max\left\{(1950 + 1200\, M_a/M_b)\frac{b}{F_y}, \frac{1200\, b}{F_y}\right\} \qquad \text{(ASD F3-2)}$$

where M_a and M_b have the same definition as noted earlier in the formula for C_b. If l is specified by the user, l_c is taken as $\frac{1200\, b}{F_y}$ in SAP2000.

Major Axis of Bending

If l_{33} is less than l_c, the allowable bending stress in the major direction of bending is taken as:

$$F_{b33} = 0.66\, F_y \quad \text{(for Compact section)} \qquad \text{(ASD F3-1)}$$

$$F_{b33} = 0.60\, F_y \quad \text{(for Noncompact section)} \qquad \text{(ASD F3-3)}$$

If l_{33} exceeds l_c, the allowable bending stress in the major direction of bending is taken, irrespective of compactness, as:

$$F_{b33} = 0.60\, F_y \qquad \text{(ASD F3-3)}$$

Minor Axis of Bending

For box sections and rectangular tubes, the allowable bending stress about the minor axis of bending is the same as that about the major axis of bending. The allowable stresses are given by ASD equations F3-1 and F3-3 which are described earlier.

Pipe Sections

For Pipe sections, the allowable bending stress for both major and minor axes of bending is taken as

$F_b = 0.66\,F_y$ (for compact section), and (ASD F3-1)

$F_b = 0.60\,F_y$ (for noncompact section). (ASD F3-3)

Rectangular and Round Bars

For solid square and round bars, the allowable bending stresses are the same in both major and minor directions of bending. The allowable stress for both the major and minor axis of bending of square and round bars and minor axis of bending of rectangular bars is taken as,

$F_b = 0.75\,F_y$. (ASD F2-1)

For solid rectangular bars bent about their major axes, the allowable stress is given by

$F_b = 0.66\,F_y$. (ASD F1-1)

General sections

For General sections the allowable bending stress in both major and minor directions is taken as,

$F_b = 0.60\,F_y$.

Allowable Stress in Shear

For webs with $\frac{h}{t_w} \leq \frac{380}{\sqrt{F_y}}$, the allowable shear stress F_v is assumed to be

$F_v = 0.40\,F_y$. (ASD F4-1).

For webs with $\frac{h}{t_w} > \frac{380}{\sqrt{F_y}}$, a reduction in the allowable shear stress applies and must be separately investigated by the user.

Calculation of Stress Ratios

In the calculation of the axial and bending stress capacity ratios, first, for each station along the length of the member, the actual stresses are calculated for each load combination. Then the corresponding allowable stresses are calculated. Then, the capacity ratios are calculated at each station for each member under the influence of each of the design load combinations. The controlling capacity ratio is then obtained, along with the associated station and load combination. A capacity ratio greater than 1.0 indicates an overstress.

During the design, the effect of the presence of bolts or welds is not considered. Also, the joints are not designed.

Axial and Bending Stresses

With the computed allowable axial and bending stress values and the factored axial and bending member stresses at each station, an interaction stress ratio is produced for each of the load combinations as follows:

- If f_a is compressive and $f_a/F_a > 0.15$, the combined stress ratio is given by the larger of

$$\frac{f_a}{F_a} + \frac{C_{m33} f_{b33}}{\left(1 - \frac{f_a}{F'_{e33}}\right) F_{b33}} + \frac{C_{m22} f_{b22}}{\left(1 - \frac{f_a}{F'_{e22}}\right) F_{b22}}, \text{ and} \qquad \text{(ASD H1-1)}$$

$$\frac{f_a}{0.60 F_y} + \frac{f_{b33}}{F_{b33}} + \frac{f_{b22}}{F_{b22}}, \text{ where} \qquad \text{(ASD H1-2)}$$

f_a, f_{b33}, f_{b22}, F_a, F_{b33}, and F_{b22} are defined earlier in this chapter,

C_{m33} and C_{m22} are coefficients representing distribution of moment along member length and are assumed to be 1.0 for all cases except for columns in unbraced frames when they are taken as 0.85. However, users can specify overriding values and

$$F'_e = \frac{12\pi^2 E}{23(Kl/r)^2}.$$

- If f_a is compressive and $f_a/F_a \leq 0.15$, a relatively simplified formula is used for the combined stress ratio.

$$\frac{f_a}{F_a} + \frac{f_{b33}}{F_{b33}} + \frac{f_{b22}}{F_{b22}} \qquad \text{(ASD H1-3)}$$

- If f_a is tensile or zero, the combined stress ratio is given by

$$\frac{f_a}{F_a} + \frac{f_{b33}}{F_{b33}} + \frac{f_{b22}}{F_{b22}}, \text{ where} \qquad \text{(ASD H2-1)}$$

f_a, f_{b33}, f_{b22}, F_a, F_{b33}, and F_{b22} are defined earlier in this chapter. However, either F_{b33} or F_{b22} need not be less than $0.6F_y$.

For circular sections, an SRSS combination is first made of the two bending components before adding the axial load component, instead of the simple addition implied by the above formulae.

Shear Stresses

From the allowable shear stress values and the factored shear stress values at each station, shear stress ratios for major and minor directions are produced for each of the load combinations as follows:

$$\frac{f_{v2}}{F_v}, \quad \text{and}$$

$$\frac{f_{v3}}{F_v}.$$

Chapter V

Check/Design for AISC-LRFD93

This chapter describes the details of the structural steel design and stress check algorithms that are used by SAP2000 when the user selects the AISC-LRFD93 design code (AISC 1994). Various notations used in this chapter are described in Table V-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

In the evaluation of the axial force/biaxial moment capacity ratios at a station along the length of the member, first the actual member force/moment components and the corresponding capacities are calculated for each load combination. Then the capacity ratios are evaluated at each station under the influence of all load combinations using the corresponding equations that are defined in this section. The controlling capacity ratio is then obtained. A capacity ratio greater than 1.0 indicates exceeding a limit state. Similarly, a shear capacity ratio is also calculated separately.

English as well as SI and MKS metric units can be used for input. But the code is based on Kip-Inch-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Kip-Inch-Second** units unless otherwise noted.

A	=	Cross-sectional area, in^2
A_g	=	Gross cross-sectional area, in^2
A_{v2}, A_{v3}	=	Major and minor shear areas, in^2
A_w	=	Shear area, equal dt_w per web, in^2
B_1	=	Moment magnification factor for moments not causing sidesway
B_2	=	Moment magnification factor for moments causing sidesway
C_b	=	Bending coefficient
C_m	=	Moment coefficient
C_w	=	Warping constant, in^6
D	=	Outside diameter of pipes, in
E	=	Modulus of elasticity, ksi
F_{cr}	=	Critical compressive stress, ksi
F_r	=	Compressive residual stress in flange assumed 10.0 for rolled sections and 16.5 for welded sections, ksi
F_y	=	Yield stress of material, ksi
G	=	Shear modulus, ksi
I_{22}	=	Minor moment of inertia, in^4
J	=	Torsional constant for the section, in^4
K	=	Effective length factor
K_{33}, K_{22}	=	Effective length K-factors in the major and minor directions
L_b	=	Laterally unbraced length of member, in
L_p	=	Limiting laterally unbraced length for full plastic capacity, in
L_r	=	Limiting laterally unbraced length for inelastic lateral-torsional buckling, in
M_{cr}	=	Elastic buckling moment, kip-in
M_{lt}	=	Factored moments causing sidesway, kip-in
M_{nt}	=	Factored moments not causing sidesway, kip-in
M_{n33}, M_{n22}	=	Nominal bending strength in major and minor directions, kip-in
M_{p33}, M_{p22}	=	Major and minor plastic moments, kip-in
M_{r33}, M_{r22}	=	Major and minor limiting buckling moments, kip-in
M_u	=	Factored moment in member, kip-in
M_{u33}, M_{u22}	=	Factored major and minor moments in member, kip-in
P_e	=	Euler buckling load, kips
P_n	=	Nominal axial load strength, kip
P_u	=	Factored axial force in member, kips
P_y	=	$A_g F_y$, kips

Table V-1
AISC-LRFD Notations

S	=	Section modulus, in^3
S_{33}, S_{22}	=	Major and minor section moduli, in^3
V_{n2}, V_{n3}	=	Nominal major and minor shear strengths, kips
V_{u2}, V_{u3}	=	Factored major and minor shear loads, kips
Z	=	Plastic modulus, in^3
Z_{33}, Z_{22}	=	Major and minor plastic moduli, in^3
b	=	Nominal dimension of longer leg of angles, in $b_f - 2t_w$ for welded and $b_f - 3t_w$ for rolled BOX (TS) sections
b_f	=	Flange width, in
d	=	Overall depth of member, in
h_c	=	Clear distance between flanges less fillets, in assumed $d - 2k$ for rolled sections and $d - 2t_f$ for welded sections
k	=	Distance from outer face of flange to web toe of fillet, in
k_c	=	Parameter used for section classification, $\frac{4}{\sqrt{h / t_w}}$, $0.35 \le k_c \le 0.763$
l_{33}, l_{22}	=	Major and minor direction unbraced member lengths, in
r	=	Radius of gyration, in
r_{33}, r_{22}	=	Radii of gyration in the major and minor directions, in
r_z	=	Minimum Radius of gyration for angles, in
t	=	Thickness, in
t_f	=	Flange thickness, in
t_w	=	Thickness of web, in
λ	=	Slenderness parameter
λ_c	=	Column slenderness parameter
λ_p	=	Limiting slenderness parameter for compact element
λ_r	=	Limiting slenderness parameter for non-compact element
λ_s	=	Limiting slenderness parameter for seismic element
φ	=	Resistance factor
φ_b	=	Resistance factor for bending, 0.9
φ_c	=	Resistance factor for compression, 0.85
φ_t	=	Resistance factor for tension, 0.9
φ_v	=	Resistance factor for shear, 0.9

Table V-1
AISC-LRFD Notations (cont.)

Design Loading Combinations

The design load combinations are the various combinations of the load cases for which the structure needs to be checked. For the AISC-LRFD93 code, if a structure is subjected to dead load (DL), live load (LL), wind load (WL), and earthquake induced load (EL), and considering that wind and earthquake forces are reversible, then the following load combinations may have to be defined (LRFD A4.1):

1.4 DL	(LRFD A4-1)
1.2 DL + 1.6 LL	(LRFD A4-2)
0.9 DL ± 1.3 WL	(LRFD A4-6)
1.2 DL ± 1.3 WL	(LRFD A4-4)
1.2 DL + 0.5 LL ± 1.3 WL	(LRFD A4-4)
0.9 DL ± 1.0 EL	(LRFD A4-6)
1.2 DL ± 1.0 EL	(LRFD A4-4)
1.2 DL + 0.5 LL ± 1.0 EL	(LRFD A4-4)

These are also the default design load combinations whenever the AISC-LRFD93 code is used. The user should use other appropriate loading combinations if roof live load is separately treated, if other types of loads are present, or if pattern live loads are to be considered.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

When using the AISC-LRFD93 code, SAP2000 design assumes that a P-Δ analysis has been performed so that moment magnification factors for moments causing sidesway can be taken as unity. It is recommended that the P-Δ analysis be done at the factored load level of 1.2 DL plus 0.5 LL (White and Hajjar 1991).

Classification of Sections

The nominal strengths for axial compression and flexure are dependent on the classification of the section as Compact, Noncompact, or Slender. SAP2000 classifies individual members according to the width/thickness ratio quantities given in Tables V-2 and V-3 (LRFD B.5.1). The definition of the section properties required in these tables are given in Figure V-1 and Table V-1. In SAP2000, a section is classified as Compact or Noncompact. Moreover, special considerations are required regarding the limits of width-thickness ratios for Compact sections in Seismic zones and Noncompact sections with compressive force as given in Table V-3. **If**

the limits for non-compact are not met, the section is classified as Slender. Currently SAP2000 does not check stresses for Slender sections.

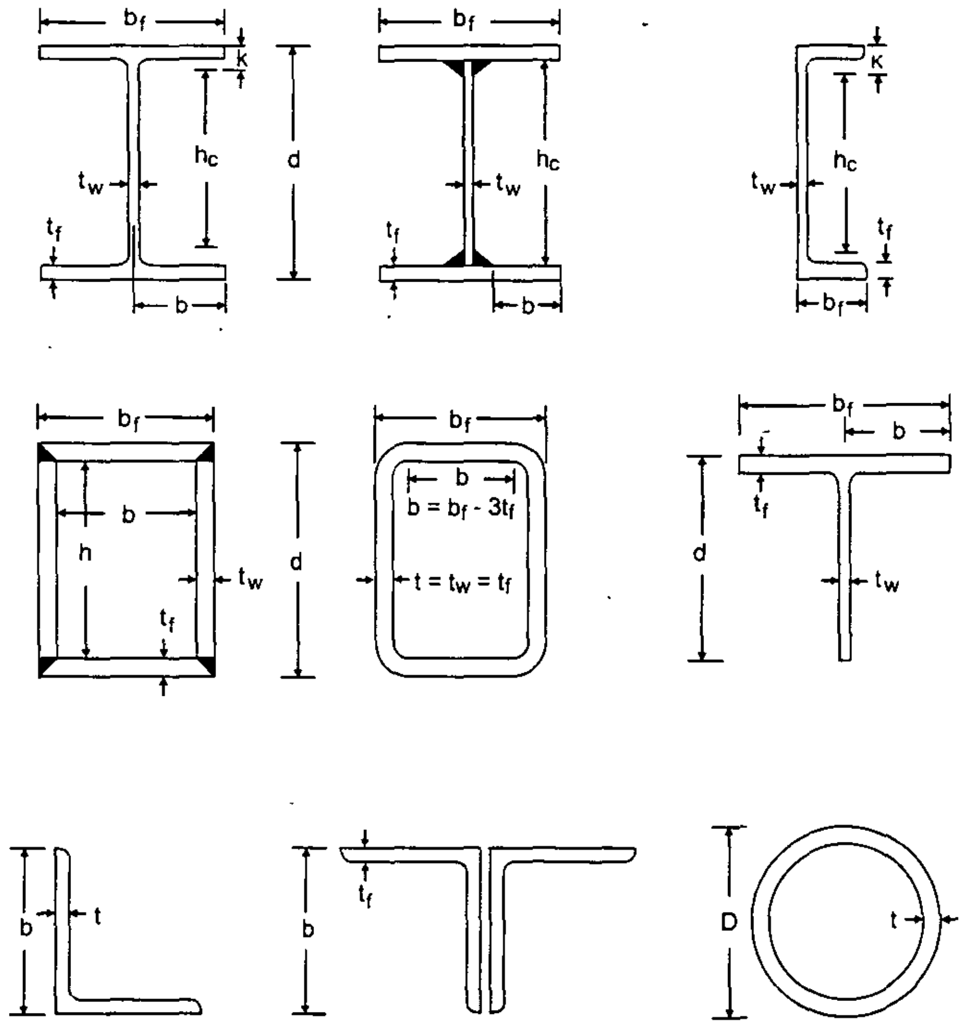
Description of Section	Check (λ)	COMPACT (λ_p)	NONCOMPACT (λ_r)
GENERAL	—	Assumed Noncompact	
RECTANGULAR	—	Assumed Compact	
I-SHAPE	$b_f/2t_f$ (rolled)	$\leq 65/\sqrt{F_y}$	$\leq 141/\sqrt{F_y - 10.0}$
	$b_f/2t_f$ (welded)	$\leq 65/\sqrt{F_y}$	$\leq 162/\sqrt{(F_y - 16.5)/k_c}$
	h_c/t_w	For $P_u/\varphi_b P_y \leq 0.125$, $\leq \frac{640}{\sqrt{F_y}}\left(1 - \frac{2.75P_u}{\varphi_b P_y}\right)$ For $P_u/\varphi_b P_y > 0.125$ $\leq \left\{\frac{191}{\sqrt{F_y}}\left(2.33 - \frac{P_u}{\varphi_b P_y}\right) \geq \frac{253}{\sqrt{F_y}}\right\}$	$\leq \frac{970}{\sqrt{F_y}}\left[1 - 0.74\frac{P_u}{\varphi_b P_y}\right]$
BOX	b/t_f b/t_f h_c/t_w	$\leq 190/\sqrt{F_y}$ (rolled) N/A for welded section As for I-shapes	$\leq 238/\sqrt{F_y}$ (rolled) $\leq 253/\sqrt{F_y}$ (welded) As for I-shapes
CHANNEL	b_f/t_f h_c/t_w	As for I-shapes As for I-shapes	As for I-shapes As for I-shapes
T-SHAPE	$b_f/2t_f$ d/t_w	As for I-Shapes Not applicable	As for I-Shapes $\leq 127/\sqrt{F_y}$
ANGLE	b/t	Not applicable	$\leq 76/\sqrt{F_y}$
ROUND BAR	—	Assumed compact	
PIPE	D/t	$\leq 2070/F_y$	$\leq 8970/F_y$
DOUBLE-ANGLE (Separated)	b/t	Not applicable	$\leq 76/\sqrt{F_y}$

Table V-2
Limiting Width-Thickness Ratio for Flexure Classification of Sections According to AISC-LRFD

Description of Section	Width-Thickness Ratio (λ)	COMPACT (SEISMIC ZONE) (λ_s)	NONCOMPACT (Uniform Compression) ($M_{22} = M_{33} = 0$) (λ_r)
GENERAL	—	Assumed Noncompact	
RECTANGULAR	—	Assumed Compact	
I-SHAPE	$b_f / 2t_f$ (rolled)	$\leq 52/\sqrt{F_y}$	$\leq 95/\sqrt{F_y}$
	$b_f / 2t_f$ (welded)	$\leq 52/\sqrt{F_y}$	$\leq 95/\sqrt{F_y}$
	h_c / t_w	For $P_u / \varphi_b P_y \leq 0.125$, $\leq \frac{520}{\sqrt{F_y}}\left(1 - 1.54\frac{P_u}{\varphi_b P_y}\right)$ For $P_u / \varphi_b P_y > 0.125$ $\leq \left\{\frac{191}{\sqrt{F_y}}\left(2.33 - \frac{P_u}{\varphi_b P_y}\right) \geq \frac{253}{\sqrt{F_y}}\right\}$	$\leq 253/\sqrt{F_y}$
BOX	b/t_f h_c/t_w	Not applicable Not applicable	$\leq 238/\sqrt{F_y}$ $\leq 253/\sqrt{F_y}$
CHANNEL	b_f/t_f h_c/t_w	Not applicable Not applicable	As for I-shapes As for I-shapes
T-SHAPE	$b_f/2t_f$ d/t_w	Not applicable Not applicable	$\leq 95/\sqrt{F_y}$ $\leq 127/\sqrt{F_y}$
ANGLE	b/t	Not applicable	$\leq 76/\sqrt{F_y}$
ROUND BAR	—	Not applicable	—
PIPE	D/t	Not applicable	$\leq 3300/F_y$
DOUBLE-ANGLE (Separated)	b/t	Not applicable	$\leq 76/\sqrt{F_y}$

Table V-3
Limiting Width-Thickness Ratios
for Classification of Sections According to AISC-LRFD

꜀├°ídг┴©1°½╚┴©2°»¿àπù



AISC-LRFD93 : Axes Conventions

3-3 is the cross-section axis parallel to the flanges or the smaller leg in angle sections.

2-2 is the cross-section axis perpendicular to the flanges or the smaller leg in angle sections.

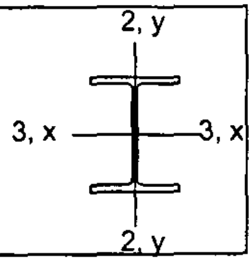


Figure V-1

AISC-LRFD Definition of Geometric Properties

Calculation of Factored Forces and Moments

The factored member loads that are calculated for each load combination are P_u, M_{u33}, M_{u22}, V_{u2} and V_{u3} corresponding to factored values of the axial load, the major moment, the minor moment, the major direction shear force and the minor direction shear force, respectively. These factored loads are calculated at each of the previously defined stations.

For loading combinations that cause compression in the member, the factored moment M_u (M_{u33} and M_{u22} in the corresponding directions) is magnified to consider second order effects. The magnified moment in a particular direction is given by:

$$M_u = B_1 M_{nt} + B_2 M_{lt} \text{, where} \qquad \text{(LRFD C1-1)}$$

B_1 = Moment magnification factor for non-sidesway moments,
B_2 = Moment magnification factor for sidesway moments,
M_{nt} = Factored moments not causing sidesway, and
M_{lt} = Factored moments causing sidesway.

The moment magnification factors are associated with corresponding directions. The moment magnification factor B_1 for moments not causing sidesway is given by

$$B_1 = \frac{C_m}{\left(1 - P_u / P_e\right)} \geq 1.0 \text{, where} \qquad \text{(LRFD C1-2)}$$

P_e is the Euler buckling load ($P_e = \dfrac{A_g F_y}{\lambda^2}$, with $\lambda = \dfrac{Kl}{r\pi}\sqrt{\dfrac{F_y}{E}}$), and

$$C_m = 0.6 - 0.4 \frac{M_a}{M_b} \text{, where} \qquad \text{(LRFD C1-3)}$$

M_a / M_b is the ratio of the smaller to the larger moment at the ends of the member, M_a / M_b being positive for double curvature bending and negative for single curvature bending. For compression members with transverse load on the member, C_m is assumed as 1.0. When M_b is zero, C_m is taken as 1.0. The program defaults C_m to 1.0 if the unbraced length, l, of the member is redefined by the user (i.e. it is not equal to the length of the member). The user can overwrite the value of C_m for any member.

The magnification factor B_1, must be a positive number. Therefore P_u must be less than P_e. If P_u is found to be greater than or equal to P_e, a failure condition is declared.

SAP2000 design assumes the analysis includes P-Δ effects, therefore B_2 is taken as unity for bending in both directions. It is suggested that the P-Δ analysis be done at the factored load level of 1.2 DL plus 0.5 LL. See also White and Hajjar (1991). If the program assumptions are not satisfactory for a particular structural model or member, the user has a choice of explicitly specifying the values of B_1 and B_2 for any member.

Calculation of Nominal Strengths

The nominal strengths in compression, tension, bending, and shear are computed for Compact and Non-compact sections according to the following subsections. The strength reduction factor, φ, is taken as follows (LRFD A5.3):

φ_t	=	Resistance factor for tension, 0.9	(LRFD D3)
φ_c	=	Resistance factor for compression, 0.85	(LRFD E2)
φ_b	=	Resistance factor for bending, 0.9	(LRFD F1)
φ_v	=	Resistance factor for shear, 0.9	(LRFD F2)

For Slender sections and any singly symmetric and unsymmetric sections requiring consideration of local buckling (LRFD Appendix B5), flexural-torsional and torsional buckling (LRFD Appendix E3), or web buckling (LRFD Appendix G), reduced nominal strengths may be applicable. The user must separately investigate this reduction if such elements are used.

If the user specifies nominal strengths for one or more elements in the "Redefine Element Design Data", these values ***will override all above the mentioned calculated values for those elements*** *as defined in the following subsections.*

Compression Capacity

The nominal axial compressive strength, P_n, depends on the slenderness ratio, $\frac{Kl}{r}$, and its critical value, λ_c. $\frac{Kl}{r}$ is the larger of $\frac{K_{33}\, l_{33}}{r_{33}}$ and $\frac{K_{22}\, l_{22}}{r_{22}}$, and

$$\lambda_c = \frac{Kl}{r\pi}\sqrt{\frac{F_y}{E}}. \qquad \text{(LRFD E2-4)}$$

P_n for compact or non-compact sections is evaluated for flexural buckling as follows:

$$P_n = A_g F_{cr}, \text{ where} \qquad \text{(LRFD E2-1)}$$

$$F_{cr} = \left(0.658^{\lambda_c^2}\right) F_y, \quad \text{for} \quad \lambda_c \leq 1.5, \text{ and} \qquad \text{(LRFD E2-2)}$$

$$F_{cr} = \left[\frac{0.877}{\lambda_c^2}\right] F_y, \quad \text{for} \quad \lambda_c > 1.5. \qquad \text{(LRFD E2-3)}$$

For single angles r_z is used instead of r_{33} and r_{22}. For members in compression, if $\frac{Kl}{r}$ is greater than 200, a message to that effect is printed (LRFD B7).

P_n for flexural-torsional buckling of double-angle and T-shaped compression members whose elements have width-thickness ratios less than λ_r is given by

$$P_n = A_g F_{crft}, \text{ where} \qquad \text{(LRFD E3-1)}$$

$$F_{crft} = \left(\frac{F_{cr2} + F_{crz}}{2H}\right)\left[1 - \sqrt{1 - \frac{4 F_{cr2} F_{crz} H}{(F_{cr2} + F_{crz})^2}}\right], \text{ where} \qquad \text{(LRFD E3-1)}$$

$$F_{crz} = \frac{GJ}{A \bar{r}_0^2},$$

$\bar{r}_0$ = Polar radius of gyration about the shear center,

$$H = 1 - \left(\frac{x_0^2 + y_0^2}{\bar{r}_0^2}\right),$$

x_0, y_0 are the coordinates of shear center with respect to the centroid, $x_0 = 0$ for double-angle and T-shaped members (y-axis of symmetry),

F_{cr2} is determined according to the equation LRFD E2-1 for flexural buckling about minor axis of symmetry for $\lambda_c = \frac{Kl}{r_{22}\pi}\sqrt{\frac{F_y}{E}}$.

For double-angles and T-shaped members with width-thickness ratios greater than λ_r or other symmetric and unsymmetric columns, the user is advised to separately perform a complete check for lateral-torsional buckling.

Tension Capacity

The nominal axial tensile strength value P_n is based on the gross cross-sectional area and the yield stress.

$$P_n = A_g F_y \qquad \text{(LRFD D1-1)}$$

It should be noted that no net section checks are made. For members in tension, if l/r is greater than 300, a message to that effect is printed (LRFD B7).

Nominal Strength in Bending

The nominal bending strength depends on the following criteria: the geometric shape of the cross-section, the axis of bending, the compactness of the section, and a slenderness parameter for lateral-torsional buckling. The nominal bending strength is the minimum value obtained from yielding, lateral-torsional buckling, flange local buckling, and web local buckling, as follows:

Yielding

For laterally braced compact (and seismic) members with $L_b \leq L_p$,

$$M_p = Z F_y \leq 1.5 S F_y \text{, where} \qquad \text{(LRFD F1-1)}$$

L_b = Laterally unbraced length, l_{22} and

L_p = Limiting laterally unbraced length for full plastic capacity,

$$\frac{300\, r_{22}}{\sqrt{F_y}} \quad \text{for I-shapes and channels, and} \qquad \text{(LRFD F1-4)}$$

$$\frac{3750\, r_{22}}{M_{p33}} \sqrt{JA} \quad \text{for boxes and rectangular bars.} \qquad \text{(LRFD F1-5)}$$

Lateral-Torsional Buckling

Doubly Symmetric Shapes and Channels

For I-shapes, channels, boxes and rectangular bars bent about the major axis, if $L_b \leq L_r$

$$M_{n33} = C_b \left[M_{p33} - (M_{p33} - M_{r33}) \left(\frac{L_b - L_p}{L_r - L_p} \right) \right] \leq M_{p33}, \qquad \text{(LRFD F1-2)}$$

and if $L_b > L_r$,

$$M_{n33} = M_{cr33} \leq M_{p33} \text{, where} \qquad \text{(LRFD F1-12)}$$

M_{n33} = Nominal major bending strength,

M_{p33} = Major plastic moment, $Z_{33}F_y \leq 1.5 S_{33}F_y$,

M_{r33} = Major limiting buckling moment,

$(F_y - F_r)S_{33}$ for I-shapes and channels, (LRFD F1-7)

and $F_y S_{33}$ for rectangular bars and boxes, (LRFD F1-11)

M_{cr33} = Critical elastic moment,

$$\frac{C_b \pi}{L_b}\sqrt{EI_{22}GJ + \left(\frac{\pi E}{L_b}\right)^2 I_{22}C_w}$$

for I-shapes and channels, and (LRFD F1-13)

$$\frac{57000\, C_b \sqrt{JA}}{L_b / r_{22}}$$ for boxes and rectangular bars, (LRFD F1-14)

L_b = Laterally unbraced length, l_{22} ,

L_p = Limiting laterally unbraced length for full plastic capacity,

$$\frac{300\, r_{22}}{\sqrt{F_y}}$$ for I-shapes and channels, and (LRFD F1-4)

$$\frac{3750\, r_{22}}{M_{p33}}\sqrt{JA}$$ for boxes and rectangular bars, (LRFD F1-5)

L_r = Limiting laterally unbraced length for inelastic lateral-torsional buckling,

$$\frac{r_{22}X_1}{F_y - F_r}\left\{1 + \left[1 + X_2(F_y - F_r)^2\right]^{1/2}\right\}^{1/2}$$

for I-shapes and channels, and (LRFD F1-6)

$$\frac{57000\, r_{22}\sqrt{JA}}{M_{r33}}$$ for boxes and rectangular bars, (LRFD F1-10)

$$X_1 = \frac{\pi}{S_{33}}\sqrt{\frac{EGJA}{2}},$$ (LRFD F1-8)

$$X_2 = 4\frac{C_w}{I_{22}}\left(\frac{S_{33}}{GJ}\right)^2,$$ (LRFD F1-9)

$$C_b = \frac{12.5\, M_{max}}{2.5\, M_{max} + 3\, M_A + 4\, M_B + 3\, M_C},$$ and (LRFD F1-3)

M_{max}, M_A, M_B, and M_C are absolute values of maximum moment, 1/4 point, center of span and 3/4 point major moments respectively, in the member. C_b should be taken as 1.0 for cantilevers. However, the program is unable to detect whether the member is a cantilever. The user should overwrite C_b for cantilevers. The program also defaults C_b to 1.0 if the unbraced length, l, of the member is redefined by the user (i.e. it is not equal to the length of the member). The user can overwrite the value of C_b for any member.

For pipes and circular bars bent about any axis,

$$M_n = M_p = ZF_y \leq 1.5SF_y \ .$$

For all sections bent about their minor axis;

$$M_{n22} = M_{p22} = Z_{22}F_y \leq 1.5S_{22}F_y \ .$$

T-sections and Double Angles

For T-shapes and double angles the nominal major bending strength is given as,

$$M_{n33} = \frac{\pi\sqrt{EI_{22}GJ}}{L_b}\left[B + \sqrt{1 + B^2}\right] \leq F_y S_{33} \text{, where} \qquad \text{(LRFD F1-15)}$$

$$B = \pm 2.3\frac{d}{L_b}\sqrt{\frac{I_{22}}{J}} \ . \qquad \text{(LRFD F1-16)}$$

The positive sign for B applies for tension in stem of T or outstanding legs of double angles (positive moments) and the negative sign applies for compression in stem (negative moments).

For T-shapes and double angles the nominal minor bending strength is assumed as,

$$M_{n22} = F_y S_{22} \ .$$

For single angles and for GENERAL sections the nominal major and minor direction bending strengths are assumed as,

$$M_n = SF_y \ .$$

Local Buckling

For non-compact I-shapes, channels and boxes, the nominal bending strengths are given by the lowest value calculated in the formulas below for the various local buckling modes possible for these sections. The nominal flexural strength M_n for the limit state of flange and web local buckling is:

$$M_{n33} = M_{p33} - (M_{p33} - M_{r33})\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right), \qquad \text{(LRFD A-F1-3)}$$

for major direction bending, and

$$M_{n22} = M_{p22} - (M_{p22} - M_{r22})\left(\frac{\lambda - \lambda_p}{\lambda_r - \lambda_p}\right), \qquad \text{(LRFD A-F1-3)}$$

for minor direction bending, where

M_{n33} = Nominal major bending strength,
M_{n22} = Nominal minor bending strength,
M_{p33} = Major plastic moment, $Z_{33}F_y \leq 1.5\, S_{33}F_y$,
M_{p22} = Minor plastic moment, $Z_{22}F_y \leq 1.5\, S_{22}F_y$,
M_{r33} = Major limiting buckling moment, (LRFD Table A-F1.1)
$(F_y - F_r)S_{33}$ for flange buckling of I-shapes and channels,
$F_y S_{33}$ for web buckling of I-shapes and channels,
and $F_y S_{33}$ for flange and web buckling of boxes,
M_{r22} = Minor limiting buckling moment, (LRFD Table A-F1.1)
$F_y S_{22}$ or flange buckling of I-shapes, channels and boxes,
λ = Controlling slenderness parameter,
λ_p = Largest value of λ for which $M_n = M_p$, and
λ_r = Largest value of λ for which buckling is inelastic.

For Noncompact pipe sections the nominal major and minor direction bending strength is

$$M_{n33} = M_{n22} = \left(\frac{600}{D/t} + F_y\right)S. \qquad \text{(LRFD Table A-F1.1)}$$

Shear Capacities

Major Axis of Bending

The nominal shear strength, V_{n2}, for major direction shears in I-shapes, boxes and channels is evaluated as follows:

For $\dfrac{h}{t_w} \leq \dfrac{418}{\sqrt{F_y}}$,

$$V_{n2} = 0.6F_y A_w, \qquad \text{(LRFD F2-1)}$$

for $\dfrac{418}{\sqrt{F_y}} < \dfrac{h}{t_w} \leq \dfrac{523}{\sqrt{F_y}}$,

$$V_{n2} = 0.6 F_y A_w \frac{418}{\sqrt{F_y}} / \frac{h}{t_w}, \text{ and} \qquad \text{(LRFD F2-2)}$$

for $\frac{523}{\sqrt{F_y}} < \frac{h}{t_w} \leq 260$,

$$V_{n2} = 132000 \frac{A_w}{[h/t_w]^2}. \qquad \text{(LRFD F2-3 and A-F2-3)}$$

The nominal shear strength for all other sections is taken as:

$$V_{n2} = 0.6\, F_y A_{v2}.$$

Minor Axis of Bending

The nominal shear strength for minor direction shears is assumed as:

$$V_{n3} = 0.6 F_y A_{v3}$$

Calculation of Capacity Ratios

In the calculation of the axial force/biaxial moment capacity ratios, first, for each station along the length of the member, the actual member force/moment components are calculated for each load combination. Then the corresponding capacities are calculated. Then, the capacity ratios are calculated at each station for each member under the influence of each of the design load combinations. The controlling compression and/or tension capacity ratio is then obtained, along with the associated station and load combination. A capacity ratio greater than 1.0 indicates exceeding a limit state.

During the design, the effect of the presence of bolts or welds is not considered. Also, the joints are not designed.

Axial and Bending Stresses

The interaction ratio is determined based on the ratio $\frac{P_u}{\varphi P_n}$. If P_u is tensile, P_n is the nominal axial tensile strength and $\varphi = \varphi_t = 0.9$; and if P_u is compressive, P_n is the nominal axial compressive strength and $\varphi = \varphi_c = 0.85$. In addition, the resistance factor for bending, $\varphi_b = 0.9$.

For $\frac{P_u}{\varphi P_n} \geq 0.2$, the capacity ratio is given as

$$\frac{P_u}{\varphi P_n} + \frac{8}{9}\left(\frac{M_{u33}}{\varphi_b M_{n33}} + \frac{M_{u22}}{\varphi_b M_{n22}}\right). \qquad \text{(LRFD H1-1a)}$$

For $\frac{P_u}{\phi P_n} < 0.2$, the capacity ratio is given as

$$\frac{P_u}{2\varphi P_n} + \left(\frac{M_{u33}}{\varphi_b M_{n33}} + \frac{M_{u22}}{\varphi_b M_{n22}}\right). \qquad \text{(LRFD H1-1b)}$$

For circular sections an SRSS (Square Root of Sum of Squares) combination is first made of the two bending components before adding the axial load component instead of the simple algebraic addition implied by the above formulas.

Shear Stresses

Similarly to the normal stresses, from the factored shear force values and the nominal shear strength values at each station for each of the load combinations, shear capacity ratios for major and minor directions are produced as follows:

$$\frac{V_{u2}}{\varphi_v V_{n2}}, \text{ and}$$

$$\frac{V_{u3}}{\varphi_v V_{n3}},$$

where $\varphi_v = 0.9$.

Chapter VI

Check/Design for CISC94

This chapter describes the details of the structural steel design and stress check algorithms that are used by SAP2000 when the user selects the CAN/CSA-S16.1-94 design code (CISC 1995). Various notations used in this chapter are described in Table VI-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

In the evaluation of the axial force/biaxial moment capacity ratios at a station along the length of the member, first the actual member force/moment components and the corresponding capacities are calculated for each load combination. Then the capacity ratios are evaluated at each station under the influence of all load combinations using the corresponding equations that are defined in this section. The controlling capacity ratio is then obtained. A capacity ratio greater than 1.0 indicates exceeding a limit state. Similarly, a shear capacity ratio is also calculated separately.

English as well as SI and MKS metric units can be used for input. But the code is based on Newton-Millimeter-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Newton-Millimeter-Second** units unless otherwise noted.

A	=	Cross-sectional area, mm^2
A_g	=	Gross cross-sectional area, mm^2
A_{v2}, A_{v3}	=	Major and minor shear areas, mm^2
A_w	=	Shear area, mm^2
C_e	=	Euler buckling strength, N
C_f	=	Factored compressive axial load, N
C_r	=	Factored compressive axial strength, N
C_w	=	Warping constant, mm^6
C_y	=	Compressive axial load at yield stress, $A_g F_y$, N
D	=	Outside diameter of pipes, mm
E	=	Modulus of elasticity, MPa
F_y	=	Specified minimum yield stress, MPa
G	=	Shear modulus, MPa
I_{22}	=	Minor moment of inertia, mm^4
J	=	Torsional constant for the section, mm^4
K	=	Effective length factor
K_{33}, K_{22}	=	Effective length K-factors in the major and minor direcions (assumed as 1.0 unless overwritten by user)
L	=	Laterally unbraced length of member, mm
M_{f33}, M_{f22}	=	Factored major and minor bending loads, N-mm
M_{p33}, M_{p22}	=	Major and minor plastic moments, N-mm
M_{r33}, M_{r22}	=	Factored major and minor bending strengths, N-mm
M_u	=	Critical elastic moment, N-mm
M_{y33}, M_{y22}	=	Major and minor yield moments, N-mm
S_{33}, S_{22}	=	Major and minor section moduli, mm^3
T_f	=	Factored tensile axial load, N
T_r	=	Factored tensile axial strength, N
U_1	=	Moment magnification factor to account for deformation of member between ends
U_2	=	Moment magnification factor (on sidesway moments) to account for P-Δ
V_{f2}, V_{f3}	=	Factored major and minor shear loads, N
V_{r2}, V_{r3}	=	Factored major and minor shear strengths, N
Z_{33}, Z_{22}	=	Major and minor plastic moduli, mm^3

Table VI-1

CISC 94 Notations

b	=	Nominal dimension of longer leg of angles $(b_f - 2t_w)$ for welded $(b_f - 3t_f)$ for rolled box sections, mm
b_f	=	Flange width, mm
d	=	Overall depth of member, mm
h	=	Clear distance between flanges, taken as $(d-2t_f)$, mm
k	=	Web plate buckling coefficient, assumed as 5.34 (no stiffeners)
k	=	Distance from outer face of flange to web toe of fillet, mm
l	=	Unbraced length of member, mm
l_{33}, l_{22}	=	Major and minor direction unbraced member lengths, mm
r	=	Radius of gyration, mm
r_{33}, r_{22}	=	Radii of gyration in the major and minor directions, mm
r_z	=	Minimum Radius of gyration for angles, mm
t	=	Thickness, mm
t_f	=	Flange thickness, mm
t_w	=	Web thickness, mm
λ	=	Slenderness parameter
φ	=	Resistance factor, taken as 0.9
ω_1	=	Moment Coefficient
ω_{13}, ω_{12}	=	Major and minor direction moment coefficients
ω_2	=	Bending coefficient

Table VI-1
CISC 94 Notations (cont.)

Design Loading Combinations

The design load combinations are the various combinations of the load cases for which the structure needs to be checked. For the CAN/CSA-S16.1-94 code, if a structure is subjected to dead load (DL), live load (LL), wind load (WL), and earthquake induced load (EL), and considering that wind and earthquake forces are reversible, then the following load combinations may have to be defined (CISC 7.2):

1.25 DL 1.25 DL + 1.50 LL	(CISC 7.2.2)
1.25 DL ± 1.50 WL 0.85 DL ± 1.50 WL 1.25 DL + 0.7 (1.50 LL ± 1.50 WL)	(CISC 7.2.2)
1.00 DL ± 1.00 EL 1.00 DL + 0.50 LL ± 1.00 EL	(CISC 7.2.6)

These are also the default design load combinations whenever CISC Code is used. In generating the above default loading combinations, importance factor is taken as 1.

The user should use other appropriate loading combinations if roof live load is separately treated, other types of loads are present, or if pattern live loads are to be considered.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

When using the CISC code, SAP2000 design assumes that a P-Δ analysis has been performed so that moment magnification factors for moments causing sidesway can be taken as unity. It is suggested that the P-Δ analysis be done at the factored load level of 1.25 DL plus 1.05 LL. See also White and Hajjar (1991).

For the gravity load case only, the code (CISC 8.6.2) requires that notional lateral loads be applied at each story, equal to 0.005 times the factored gravity loads acting at each story. If extra load cases are used for such analysis, they should be included in the loading combinations with due consideration to the fact that the notional lateral forces can be positive or negative.

Classification of Sections

For the determination of the nominal strengths for axial compression and flexure, the sections are classified as either Class 1 (Plastic), Class 2 (Compact), Class 3 (Noncompact), or Class 4 (Slender). The program classifies the individual sections according to Table VI-2 (CISC 11.2). According to this table, a section is classified as either Class 1, Class 2, or Class 3 as applicable.

If a section fails to satisfy the limits for Class 3 sections, the section is classified as Class 4 Currently SAP2000 does not check stresses for Class 4 sections.

Calculation of Factored Forces and Moments

The factored member forces for each load combination are calculated at each of the previously defined stations. These member forces are T_f or C_f, M_{f33}, M_{f22}, V_{f2} and V_{f3} corresponding to factored values of the tensile or compressive axial load, the major moment, the minor moment, the major direction shear, and the minor direction shear, respectively.

Because SAP2000 design assumes that the analysis includes P-Δ effects, any magnification of sidesway moments due to the second order effects are already included, therefore U_2 for both directions of bending is taken as unity. It is suggested that the P-Δ analysis be done at the factored load level of 1.25 DL plus 1.05 LL. See also White and Hajjar (1991).

However, the user can overwrite the values of U_2 for both major and minor direction bending. In this case M_f in a particular direction is taken as:

$$M_f = M_{fg} + U_2 M_{ft}, \text{ where} \qquad \text{(CISC 8.6.1)}$$

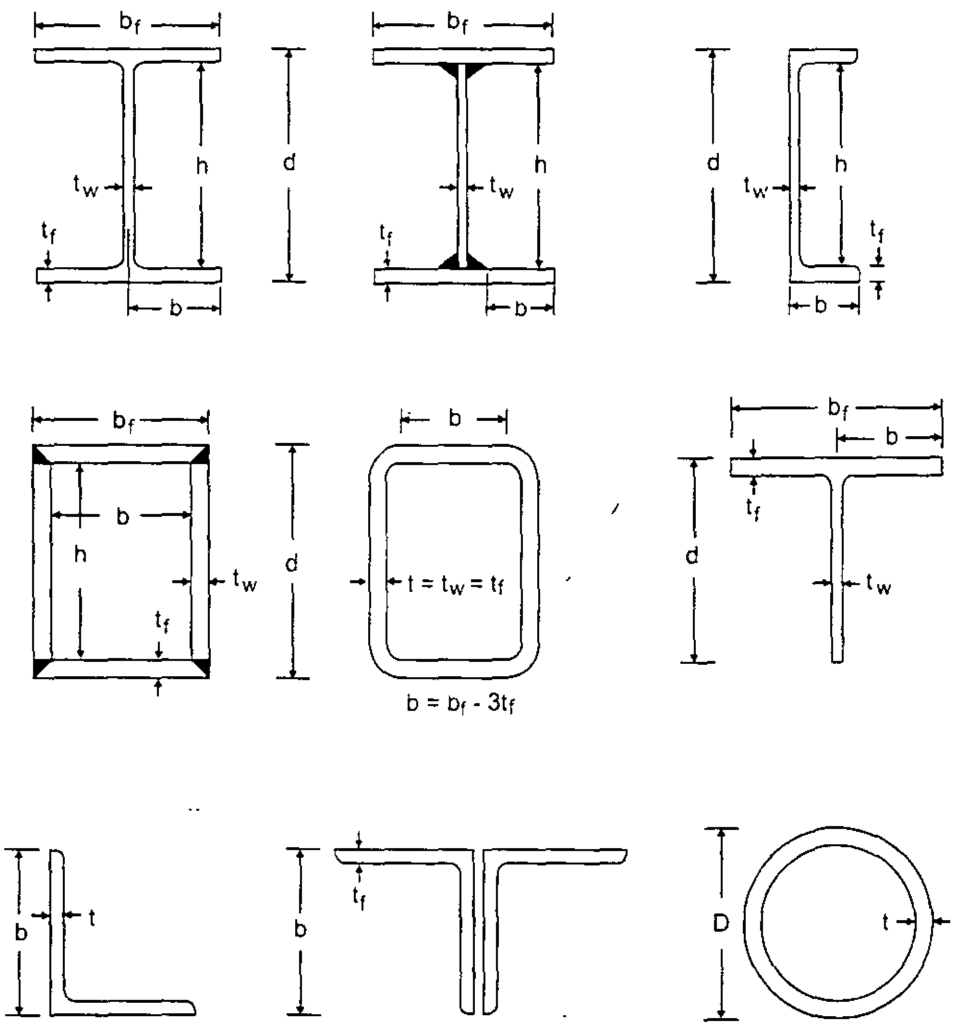
U_2 = Moment magnification factor for sidesway moments,
M_{fg} = Factored moments not causing translation, and
M_{ft} = Factored moments causing sidesway.

Calculation of Factored Strengths

The factored strengths in compression, tension, bending, and shear are computed for Class 1, 2, and 3 sections in SAP2000. The strength reduction factor, φ, is taken as 0.9 (CISC 13.1).

Description of Section	Ratio Checked	Class 1 (Plastic)	Class 2 (Compact)	Class 3 (Noncompact)
GENERAL	—	Assumed Class 3		
RECTAN-GULAR	—	Assumed Class 2		
I-SHAPE	$b_f/2t_f$	$\leq 145/\sqrt{F_y}$	$\leq 170/\sqrt{F_y}$	$\leq 200/\sqrt{F_y}$
	h/t_w	$\leq \frac{1100}{\sqrt{F_y}}\left(1 - 0.39\frac{C_f}{C_y}\right)$	$\leq \frac{1700}{\sqrt{F_y}}\left(1 - 0.61\frac{C_f}{C_y}\right)$	$\leq \frac{1900}{\sqrt{F_y}}\left(1 - 0.65\frac{C_f}{C_y}\right)$
BOX	b/t_f	$\leq 420/\sqrt{F_y}$ (rolled) $\leq 525/\sqrt{F_y}$ (welded)	$\leq 525/\sqrt{F_y}$	$\leq 670/\sqrt{F_y}$
	h/t_w	As for I-shapes	As for I-shapes	As for I-shapes
CHANNEL	b_f/t_f h/t_w	Not applicable Not applicable	Not applicable Not applicable	$\leq 200/\sqrt{F_y}$ As for I-shapes
T-SHAPE	$b_f/2t_f$ d/t_w	Not applicable Not applicable	Not applicable Not applicable	$\leq 200/\sqrt{F_y}$ $\leq 340/\sqrt{F_y}$
ANGLE	b/t	Not applicable	Not applicable	$\leq 200/\sqrt{F_y}$
ROUND BAR	—	Assumed Class 2		
PIPE (Flexure)	D/t	$\leq 13000/F_y$	$\leq 18000/F_y$	$\leq 66000/F_y$
PIPE (Axial)	D/t	—	—	$\leq 23000/F_y$
DOUBLE ANGLE	b/t	Not applicable	Not applicable	$\leq 200/\sqrt{F_y}$

Table VI-2
Limiting Width-Thickness Ratio
Classification of Sections in CISC 94



CISC95 : Axes Conventions

3-3 is the cross-section axis parallel to the flanges or the smaller leg in angle sections. This is the same as the x-x axis.

2-2 is the cross-section axis perpendicular to the flanges or the smaller leg in angle sections. This is the same as the y-y axis.

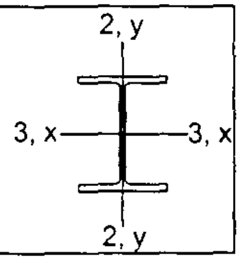


Figure VI-1
CISC 94 Definition of Geometric Properties

For Class 4 (Slender) sections and any singly symmetric and unsymmetric sections requiring consideration of local buckling, flexural-torsional and torsional buckling, or web buckling, reduced nominal strengths may be applicable. The user must separately investigate this reduction if such elements are used.

If the user specifies nominal strengths for one or more elements in the "Redefine Element Design Data", these values ***will override all the above mentioned calculated values for those elements*** *as defined in the following subsections.*

Compression Strength

The factored axial compressive strength value, C_r, for Class 1, 2, or 3 sections depends on a factor, λ, which eventually depends on the slenderness ratio, $\frac{Kl}{r}$, which is the larger of $\frac{K_{33}\, l_{33}}{r_{33}}$ and $\frac{K_{22}\, l_{22}}{r_{22}}$, and is defined as

$$\lambda = \frac{Kl}{r\pi}\sqrt{\frac{F_y}{E}}.$$

Then the factored axial strength is evaluated as follows (CISC 13.3.1):

$$C_r = \varphi A F_y \left(1+\lambda^{2n}\right)^{-\frac{1}{n}}, \text{ where} \qquad \text{(CISC 13.3.1)}$$

$n = 1.34$, for all types of sections except wide-flange sections with thickness less than or equal to 25.4 mm, and (CISC Table 1-2 and 6-2)

$n = 0.98$, for all wide-flange sections with thickness greater than or equal to 25.4 mm. (CISC Table 1-2 and 6-2)

For single angles r_z is used in place of r_{33} and r_{22}. The above calculated value of C_r is conservative for hot-formed or stress-relieved hollow sections where the code allows a higher value (CISC 13.3.1). For members in compression, if $\frac{Kl}{r}$ is greater than 200, a message is printed (CISC 10.2.1).

Tension Strength

The factored axial tensile strength value, T_r, is taken as $\varphi A_g F_y$ (CISC 13.2.(a).(i)). For members in tension, if l/r is greater than 300, a message is printed accordingly (CISC 10.2.2).

$$T_r = \varphi A_g F_y \qquad \text{(CISC 13.2)}$$

Bending Strengths

The factored bending strength in the major and minor directions is based on the geometric shape of the section, the section classification for compactness, and the unbraced length of the member. The bending strengths are evaluated according to CISC as follows (CISC 13.5 and 13.6):

For laterally supported members, the moment capacities are considered to be as follows:

For Class 1 and 2, $M_r = \varphi ZF_y$, and (CISC 13.5)

For Class 3, $M_r = \varphi SF_y$. (CISC 13.5)

Special considerations are required for laterally unsupported members. The procedure for the determination of moment capacities for laterally unsupported members (CISC 13.6) is described in the following subsections.

I-shapes and Boxes

Major Axis of Bending

For Class 1 and 2 sections of I-shapes and boxes bent about the major axis,

when $M_u > 0.67\, M_{p33}$,

$$M_{r3} = 1.15\varphi\, M_{p33}\left(1\text{-}0.28\,\frac{M_{p33}}{M_u}\right) \leq \varphi\, M_{p33}\text{, and} \qquad \text{(CISC 13.6)}$$

when $M_u \leq 0.67\, M_{p33}$,

$$M_{r33} = \varphi\, M_u\text{, where} \qquad \text{(CISC 13.6)}$$

M_{r33} = Factored major bending strength,
M_{p33} = Major plastic moment, $Z_{33}F_y$,
M_u = Critical elastic moment,

$$\frac{\omega_2 \pi}{L}\sqrt{EI_{22}GJ + \left(\frac{\pi E}{L}\right)^2 I_{22}C_w}\,, \qquad \text{(CISC 13.6)}$$

L = Laterally unbraced length, l_{22},
C_w = Warping constant assumed as 0.0,

for boxes, pipes, rectangular and circular bars, and

$$\omega_2 \quad = \quad 1.75 + 1.05\left(\frac{M_a}{M_b}\right) + 0.30\left(\frac{M_a}{M_b}\right)^2 \leq 2.5. \qquad \text{(CISC 13.6)}$$

M_a and M_b are end moments of the unbraced segment and M_a is less than M_b, $\left(\frac{M_a}{M_b}\right)$ being positive for double curvature bending and negative for single curvature bending. If any moment within the segment is greater than M_b, ω_2 is taken as 1.0. The program defaults ω_2 to 1.0 if the unbraced length, l of the member is overwritten by the user (i.e. it is not equal to the length of the member). ω_2 should be taken as 1.0 for cantilevers. However, the program is unable to detect whether the member is a cantilever. The user can overwrite the value of ω_2 for any member by specifying it.

For Class 3 sections of I-shapes, channels, boxes bent about the major axis,

when $M_u > 0.67 M_{y33}$,

$$M_{r33} = 1.15\varphi\, M_{y33}\left(1 - 0.28\frac{M_{y33}}{M_u}\right) \leq \varphi\, M_{y33}, \text{ and} \qquad \text{(CISC 13.6)}$$

when $M_u \leq 0.67 M_{y33}$

$$M_{r33} = \varphi M_u, \text{ where} \qquad \text{(CISC 13.6)}$$

M_{r33} and M_u are as defined earlier for Class 1 and 2 sections and M_{y33} is the major yield moment, $S_{33}F_y$.

Minor Axis of Bending

For Class 1 and 2 sections of I-shapes and boxes bent about their minor axis,

$$M_{r22} = \varphi\, M_{p22} = \varphi\, Z_{22}F_y\ .$$

For Class 3 sections of I-shapes and boxes bent about their minor axis,

$$M_{r22} = M_{y22} = S_{22}F_y.$$

Rectangular Bar

Major Axis of Bending

For Class 2 rectangular bars bent about the major axis, when $M_u > 0.67\, M_{p33}$,

$$M_{r33} = 1.15\varphi\, M_{p33}\left(1\text{-}0.28\frac{M_{p33}}{M_u}\right) \leq \phi\, M_{p33}\text{, and} \qquad \text{(CISC 13.6)}$$

when $M_u \leq 0.67\, M_{p33}$,

$$M_{r33} = \varphi\, M_u\,. \qquad \text{(CISC 13.6)}$$

Minor Axis of Bending

For Class 2 sections of rectangular bars bent about their minor axis.

$$M_{r22} = \varphi\, M_{p22} = \varphi\, Z_{22} F_y\,.$$

Pipes and Circular Rods

For pipes and circular rods bent about any axis

When $M_u > 0.67\, M_{p33}$,

$$M_{r33} = 1.15\,\varphi\, M_{p33}\left(1\text{-}0.28\frac{M_{p33}}{M_u}\right) \leq \varphi\, M_{p33}\text{, and} \qquad \text{(CISC 13.6)}$$

when $M_u \leq 0.67\, M_{p33}$,

$$M_{r33} = \varphi\, M_u\,. \qquad \text{(CISC 13.6)}$$

Channel Sections

Major Axis of Bending

For Class 3 channel sections bent about the major axis,

when $M_u > 0.67\, M_{y33}$,

$$M_{r33} = 1.15\varphi\, M_{y33}\left(1 - 0.28\frac{M_{y33}}{M_u}\right) \leq \varphi\, M_{y33}\text{, and} \qquad \text{(CISC 13.6)}$$

when $M_u \leq 0.67\, M_{y33}$,

$$M_{r33} = \varphi\, M_u\,.$$

Minor Axis of Bending

For Class 3 channel sections bent about their minor axis,

$$M_{r22} = M_{y22} = S_{22}F_y \, .$$

T-shapes and double angles

Major Axis of Bending

For Class 3 sections of T-shapes and double angles the factored major bending strength is assumed to be (CISC 13.6d),

$$M_{r33} = \varphi \frac{\omega_2 \pi \sqrt{EI_{22}GJ}}{L}\left[B + \sqrt{1 + B^2}\right] \leq \phi \, F_y S_{33} \, , \text{ where}$$

$$B = \pm \; 2.3(d/L)\sqrt{I_{22}/J} \, .$$

The positive sign for B applies for tension in stem of T-sections or outstanding legs of double angles (positive moments) and the negative sign applies for compression in stem (negative moments).

Minor Axis of Bending

For Class 3 sections of T-shapes and double angles the factored minor bending strength is assumed as,

$$M_{r22} = \varphi \, F_y S_{22} \, .$$

Single Angle and General Sections

For Class 3 single angles and for General sections, the factored major and minor direction bending strengths are assumed as,

$$M_{r33} = \varphi \, F_y S_{33} \, , \text{ and}$$

$$M_{r22} = \varphi \, F_y S_{22} \, .$$

Shear Strengths

The factored shear strength, V_{r2}, for major direction shears in I-shapes, boxes and channels is evaluated as follows (CISC 13.4.1.1):

- For $\frac{h}{t_w} \leq 439\sqrt{\frac{k_v}{F_y}}$,

$$V_{r2} = \varphi A_w \{0.66 F_y\}. \qquad \text{(CISC 13.4.1.1)}$$

- For $439\sqrt{\frac{k_v}{F_y}} < \frac{h}{t_w} \leq 502\sqrt{\frac{k_v}{F_y}}$,

$$V_{r2} = \varphi A_w \left\{290\frac{\sqrt{k_v F_y}}{h/t_w}\right\}. \qquad \text{(CISC 13.4.1.1)}$$

- For $502\sqrt{\frac{k_v}{F_y}} < \frac{h}{t_w} \leq 621\sqrt{\frac{k_v}{F_y}}$,

$$V_{r2} = \varphi A_w \{F_{cri} + F_t\}, \text{ where} \qquad \text{(CISC 13.4.1.1)}$$

$$F_{cri} = 290\frac{\sqrt{k_v F_y}}{h/t_w}, \text{ and}$$

$$F_t = \left(0.5F_y - 0.866F_{cri}\right)\left\{\frac{1}{\sqrt{1+(a/h)^2}}\right\}.$$

Assuming no stiffener is used, the value of F_t is taken as zero.

- For $\frac{h}{t_w} > 621\sqrt{\frac{k_v}{F_y}}$,

$$V_{r2} = \varphi A_w \{F_{cre} + F_t\}, \text{ where} \qquad \text{(CISC 13.4.1.1)}$$

$$F_{cre} = \frac{180000\,k_v}{(h/t_w)^2}.$$

In the above equations, k_v is the shear buckling coefficient, and it is defined as:

$$k_v = 4 + \frac{5.34}{(a/h)^2}, \qquad a/h < 1$$

$$k_v = 5.34 + \frac{4}{(a/h)^2}, \qquad a/h \geq 1$$

and the aspect ratio a/h is the ratio of the distance between the stiffeners to web depth. Assuming no stiffener is used, the value of k_v is taken as 5.34.

The factored shear strength for minor direction shears in I-shapes, boxes and channels is assumed as

$$V_{r2} = 0.66\ \varphi\ F_y A_{v3}\ . \qquad \text{(CISC 13.4.2)}$$

The factored shear strength for major and minor direction shears for all other sections is assumed as (CISC 13.4.2):

$$V_{r2} = 0.66\ \varphi\ F_y A_{v2}\ , \text{ and} \qquad \text{(CISC 13.4.2)}$$

$$V_{r3} = 0.66\ \varphi\ F_y A_{v3}\ . \qquad \text{(CISC 13.4.2)}$$

Calculation of Capacity Ratios

In the calculation of the axial force/biaxial moment capacity ratios, first, for each station along the length of the member, for each load combination, the actual member force/moment components are calculated. Then the corresponding capacities are calculated. Then, the capacity ratios are calculated at each station for each member under the influence of each of the design load combinations. The controlling compression and/or tension capacity ratio is then obtained, along with the associated station and load combination. A capacity ratio greater than 1.0 indicates exceeding a limit state.

During the design, the effect of the presence of bolts or welds is not considered. Also, the joints are not designed.

Axial and Bending Stresses

From the factored axial loads and bending moments at each station and the factored strengths for axial tension and compression and major and minor bending, an interaction capacity ratio is produced for each of the load combinations as follows:

Compressive Axial Load

If the axial load is compressive, the capacity ratio for all sections except Class 1 I-shaped sections is given by:

$$\frac{C_f}{C_r} + \frac{U_{13}\ M_{f33}}{M_{r33}} + \frac{U_{12}\ M_{f22}}{M_{r22}} \qquad \text{(CISC 13.8.1)}$$

For Class 1 I-shaped sections the capacity is given by:

$$\frac{C_f}{C_r} + 0.85\frac{U_{13}\,M_{f33}}{M_{r33}} + 0.6\frac{U_{12}\,M_{f22}}{M_{r22}} \qquad \text{(CISC 13.8.2)}$$

The above ratios are calculated for each of the following conditions and the largest ratio is reported:

- Cross-sectional Strength:
  - M_{r33} and M_{r22} are calculated assuming laterally braced members,
  - $C_r = \varphi A F_y$, and
  - $U_{13} = U_{12} = 1.0$.
- Overall Member Strength:
  - M_{r33} and M_{r22} are calculated assuming laterally braced members, and
  - C_r is evaluated with $K = 1.0$ in the direction of bending.
- Lateral-Torsional Buckling Strength:
  - C_r is calculated based on $\frac{K_{22}\,l_{22}}{r_{22}}$ only, and
  - $U_{13} \geq 1$.

For Class 1 I-shapes, the following ratio is also checked:

$$\frac{M_{f33}}{M_{r33}} + \frac{M_{f22}}{M_{r22}}. \qquad \text{(CISC 13.8.2)}$$

In the above expressions,

$$U_1 = \frac{\omega_1}{1 - C_f/C_e}, \qquad \text{(CISC 13.8.3)}$$

$$C_e = \frac{\pi^2 EI}{L^2},$$

$$\omega_1 = 0.6 - 0.4\, M_a / M_b \geq 0.4, \text{ and}$$

M_a / M_b is the ratio of the smaller to the larger moment at the ends of the member, M_a / M_b being positive for double curvature bending and negative for single curvature bending. ω_1 is assumed as 1.0 for beams with transverse load and when M_b is zero.

The program defaults ω_1 to 1.0 if the unbraced length, l, of the member is redefined by the user (i.e. it is not equal to the length of the member). The user can overwrite the value of ω_1 for any member by specifying it. The factor U_1 must be a positive number. Therefore C_f must be less than C_e. If this is not true, a failure condition is declared.

Tensile Axial Load

If the axial load is tensile the capacity ratio is given by the larger of:

$$\left[\frac{T_f}{T_r}\right] + \left[\frac{M_{f33}}{M_{r33}} + \frac{M_{f22}}{M_{r22}}\right] \qquad \text{(CISC 13.9)}$$

assuming M_{r33} and M_{r22} are calculated based on fully supported ($l = 0$), and

$$\left[\frac{M_{f33}}{M_{r33}} + \frac{M_{f22}}{M_{r22}}\right] - \left[\frac{T_f\, Z_{33}}{M_{r33}\, A}\right] \quad \text{(for Class 1 and 2) or} \qquad \text{(CISC 13.9)}$$

$$\left[\frac{M_{f33}}{M_{r33}} + \frac{M_{f22}}{M_{r22}}\right] - \left[\frac{T_f\, S_{33}}{M_{r33}\, A}\right] \quad \text{(for Class 3).} \qquad \text{(CISC 13.9)}$$

For circular sections an SRSS combination is first made of the two bending components before adding the axial load component instead of the simple algebraic addition implied by the above interaction formulas.

Shear Stresses

From the factored shear force values and the factored shear strength values at each station, for each of the load combinations, shear capacity ratios for major and minor directions are produced as follows:

$$\frac{V_{f2}}{V_{r2}} \quad \text{and}$$

$$\frac{V_{f3}}{V_{r3}}.$$

Chapter **VII**

Check/Design for BS 5950

This chapter describes the details of the structural-steel design and stress check algorithms that are used by SAP2000 when the user selects the BS 5950 design code (BSI 1990). Various notations used in this chapter are described in Table VII-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

In the evaluation of the axial force/biaxial moment capacity ratios at a station along the length of the member, first the actual member force/moment components and the corresponding capacities are calculated for each load combination. Then the capacity ratios are evaluated at each station under the influence of all load combinations using the corresponding equations that are defined in this section. The controlling capacity ratio is then obtained. A capacity ratio greater than 1.0 indicates exceeding a limit state. Similarly, a shear capacity ratio is also calculated separately.

English as well as SI and MKS metric units can be used for input. But the code is based on Newton-Millimeter-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Newton-Millimeter-Second** units unless otherwise noted.

A	=	Cross-sectional area, mm^2
A_g	=	Gross cross-sectional area, mm^2
A_{v2}, A_{v3}	=	Major and minor shear areas, mm^2
B	=	Breadth, mm
D	=	Depth of section, mm or outside diameter of pipes, mm
E	=	Modulus of elasticity, MPa
F_c	=	Axial compression, N
F_t	=	Axial tension, N
F_{v2}, F_{v3}	=	Major and minor shear loads, N
G	=	Shear modulus, MPa
H	=	Warping constant, mm^6
I_{33}	=	Major moment of inertia, mm^4
I_{22}	=	Minor moment of inertia, mm^4
J	=	Torsional constant for the section, mm^4
K	=	Effective length factor
K_{33}, K_{22}	=	Major and minor effective length factors
M	=	Applied moment, N-mm
M_{33}	=	Applied moment about major axis, N-mm
M_{22}	=	Applied moment about minor axis, N-mm
M_{a33}	=	Major maximum bending moment, N-mm
M_{a22}	=	Minor maximum bending moment, N-mm
M_b	=	Buckling resistance moment, N-mm
M_c	=	Moment capacity, N-mm
M_{c33}	=	Major moment capacity, N-mm
M_{c22}	=	Minor moment capacity, N-mm
M_E	=	Elastic critical moment, N-mm
P_c	=	Compression resistance, N
P_{c33}, P_{c22}	=	Major and minor compression resistance, N
P_t	=	Tension capacity, N
P_{v2}, P_{v3}	=	Major and minor shear capacities, N
S_{33}, S_{22}	=	Major and minor plastic section moduli, mm^3
T	=	Thickness of flange or leg, mm
Y_s	=	Specified yield strength, MPa
Z_{33}, Z_{22}	=	Major and minor elastic section moduli, mm^3

Table VII-1
BS 5950 Notations

a	=	Robertson constant
b	=	Outstand width, mm
d	=	Depth of web, mm
h	=	Story height, mm
k	=	Distance from outer face of flange to web toe of fillet , mm
l	=	Unbraced length of member, mm
l_{33}, l_{22}	=	Major and minor direction unbraced member lengths, mm
l_{e33}, l_{e22}	=	Major and minor effective lengths, mm
m	=	Equivalent uniform moment factor
q_e	=	Elastic critical shear strength of web panel, MPa
q_{cr}	=	Critical shear strength of web panel, MPa
r_{33}, r_{22}	=	Major and minor radii of gyration, mm
r_z	=	Minimum radius of gyration for angles, mm
t	=	Thickness, mm
t_f	=	Flange thickness, mm
t_w	=	Thickness of web, mm
u	=	Buckling parameter
v	=	Slenderness factor
β	=	Ratio of smaller to larger end moments
ε	=	Constant $\left(\frac{275}{p_y}\right)^{1/2}$
λ	=	Slenderness parameter
λ_o	=	Limiting slenderness
λ_{LT}	=	Equivalent slenderness
λ_{Lo}	=	Limiting equivalent slenderness
η	=	Perry factor
η_{LT}	=	Perry coefficient
p_c	=	Compressive strength, MPa
p_E	=	Euler strength, MPa
p_y	=	Yield strength, MPa
ψ	=	Monosymmetry index

Table VII-1

BS 5950 Notations (cont.)

Design Loading Combinations

The design load combinations are the various combinations of the load cases for which the structure needs to be checked. According to the BS 5950 code, if a structure is subjected to dead load (DL), live load (LL), wind load (WL), and earthquake load (EL); and considering that wind and earthquake forces are reversible, then the following load combinations may have to be considered (BS 2.4):

1.4 DL
1.4 DL + 1.6 LL (BS 2.4.1.1)

1.0 DL ± 1.4 WL
1.4 DL ± 1.4 WL
1.2 DL + 1.2 LL ± 1.2 WL (BS 2.4.1.1)

1.0 DL ± 1.4 EL
1.4 DL ± 1.4 EL
1.2 DL + 1.2 LL ± 1.2 EL

These are also the default design load combinations whenever BS 5950 Code is used. The user should use other appropriate loading combinations if roof live load is separately treated, other types of loads are present, or if pattern live loads are to be considered.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

In addition to the above load combinations, the code requires that all buildings should be capable of resisting a notional design horizontal load applied at each floor or roof level. The notional load should be equal to the maximum of 0.01 times the factored dead load and 0.005 times the factored dead plus live loads (BS 2.4.2.3). The notional forces should be assumed to act in any one direction at a time and should be taken as acting simultaneously with the factored dead plus vertical imposed live loads. They should not be combined with any other horizontal load cases (BS 5.1.2.3). It is recommended that the user should define additional load cases for considering the notional load in SAP2000 and define the appropriate design combinations.

When using the BS 5950 code, SAP2000 design assumes that a P-Δ analysis has already been performed, so that moment magnification factors for the moments causing side-sway can be taken as unity. It is suggested that the P-Δ analysis be done at the factored load level corresponding to 1.2 dead load plus 1.2 live load. See also White and Hajjar (1991).

Classification of Sections

The nominal strengths for axial compression and flexure are dependent on the classification of the section as Plastic, Compact, Semi-compact, or Slender. SAP2000 checks the sections according to Table VII-2 (BS 3.5.2). In this Table:

- R is the ratio of mean longitudinal stress in the web to ρ_y in a semi-compact section. This implies that for a semi-compact section in pure bending R is zero. In calculating R, compression is taken as positive and tension is taken as negative.
- α is given as $2\gamma_c/d$, where γ_c is the distance from the plastic neutral axis to the edge of the web connected to the compression flange. For $\alpha > 2$, the section is treated as having compression throughout.
- ε is defined as $(275/\rho_y)^{1/2}$.

The section is classified as either Class 1 (Plastic), Class 2 (Compact), or Class 3 (Semi-compact) as applicable. **If a section fails to satisfy the limits for Class 3 (Semi-compact) sections, the section is classified as Class 4 (Slender). Currently SAP2000 does not check stresses for Slender sections.**

Calculation of Factored Forces and Moments

The factored member loads that are calculated for each load combination are F_t or F_c, M_{33}, M_{22}, F_{v2}, and F_{v3} corresponding to factored values of the tensile or compressive axial load, the major moment, the minor moment, the major direction shear load, and the minor direction shear load, respectively. These factored loads are calculated at each of the previously defined stations.

The moment magnification for non-sidesway moments is included in the overall buckling interaction equations.

$$M = M_g + \left\{ \frac{1}{1 - 200\,\varphi_{s\,max}} \right\} M_s, \text{ where} \qquad \text{(BS 5.6.3)}$$

$\varphi_{s\,max}$ = Maximum story-drift divided by the story-height,
M_g = Factored moments not causing translation, and
M_s = Factored moments causing sidesway.

Description of Section	Ratio Checked	Class 1 (Plastic)	Class 2 (Compact)	Class 3 (Semi-compact)
GENERAL	—	Assumed Semi-compact		
SOLID RECTANGLE	—	Assumed Compact		
I-SHAPE	b/T (Rolled)	$\leq 8.5\,\varepsilon$	$\leq 9.5\,\varepsilon$	$\leq 15\,\varepsilon$
	b/T (welded)	$\leq 7.5\,\varepsilon$	$\leq 8.5\,\varepsilon$	$\leq 13\,\varepsilon$
	d/t webs ($\alpha < 2$)	$\leq \dfrac{79\,\varepsilon}{0.4 + 0.6\,\alpha}$	$\leq \dfrac{98\,\varepsilon}{\alpha}$	For $R > 0$: $\leq \dfrac{120\,\varepsilon}{1 + 1.5R}$ and $\leq \left(\dfrac{41}{R} - 13\right)\varepsilon$ (welded) $\leq \dfrac{120\,\varepsilon}{1 + 1.5R}$ and $\leq \left(\dfrac{41}{R} - 2\right)\varepsilon$ (rolled) For $R = 0$: $\leq 120\,\varepsilon$ For $R < 0$: $\leq \dfrac{120\,\varepsilon}{(1 - R)^2}$ and $\leq 250\,\varepsilon$
	d/t webs ($\alpha \geq 2$) (rolled)	$\leq 39\,\varepsilon$	$\leq 39\,\varepsilon$	$\leq 39\,\varepsilon$
	d/t webs ($\alpha \geq 2$) (welded)	$\leq 28\,\varepsilon$	$\leq 28\,\varepsilon$	$\leq 28\,\varepsilon$
BOX	b/T (Rolled)	$\leq 26\,\varepsilon$	$\leq 32\,\varepsilon$	$\leq 39\,\varepsilon$
	b/T (welded)	$\leq 23\,\varepsilon$	$\leq 25\,\varepsilon$	$\leq 28\,\varepsilon$
	d/t	As for I-shapes	As for I-shapes	As for I-shapes
CHANNEL	b/T d/t	As for I-shapes	As for I-shapes	As for I-shapes

Table VII-2
Classification of Sections According to BS 5950

Description of Section	Ratio Checked	Class 1 (Plastic)	Class 2 (Compact)	Class 3 (Semi-compact)
T-SHAPE	b/T d/t	$\leq 8.5\,\varepsilon$ $\leq 8.5\,\varepsilon$	$\leq 9.5\,\varepsilon$ $\leq 9.5\,\varepsilon$	$\leq 19\,\varepsilon$ $\leq 19\,\varepsilon$
ANGLE	b/T $\frac{b+d}{T}$	$\leq 8.5\,\varepsilon$ —	$\leq 9.5\,\varepsilon$ —	$\leq 15\,\varepsilon$ $\leq 23\,\varepsilon$
SOLID CIRCLE	—	Assumed Compact		
PIPE	D/t	$\leq 40\,\varepsilon^2$	$\leq 57\,\varepsilon^2$	$\leq 80\,\varepsilon^2$
DOUBLE ANGLE (separated)	d/T $\frac{b+d}{T}$	$\leq 8.5\,\varepsilon$ —	$\leq 9.5\,\varepsilon$ —	$\leq 15\,\varepsilon$ $\leq 23\,\varepsilon$

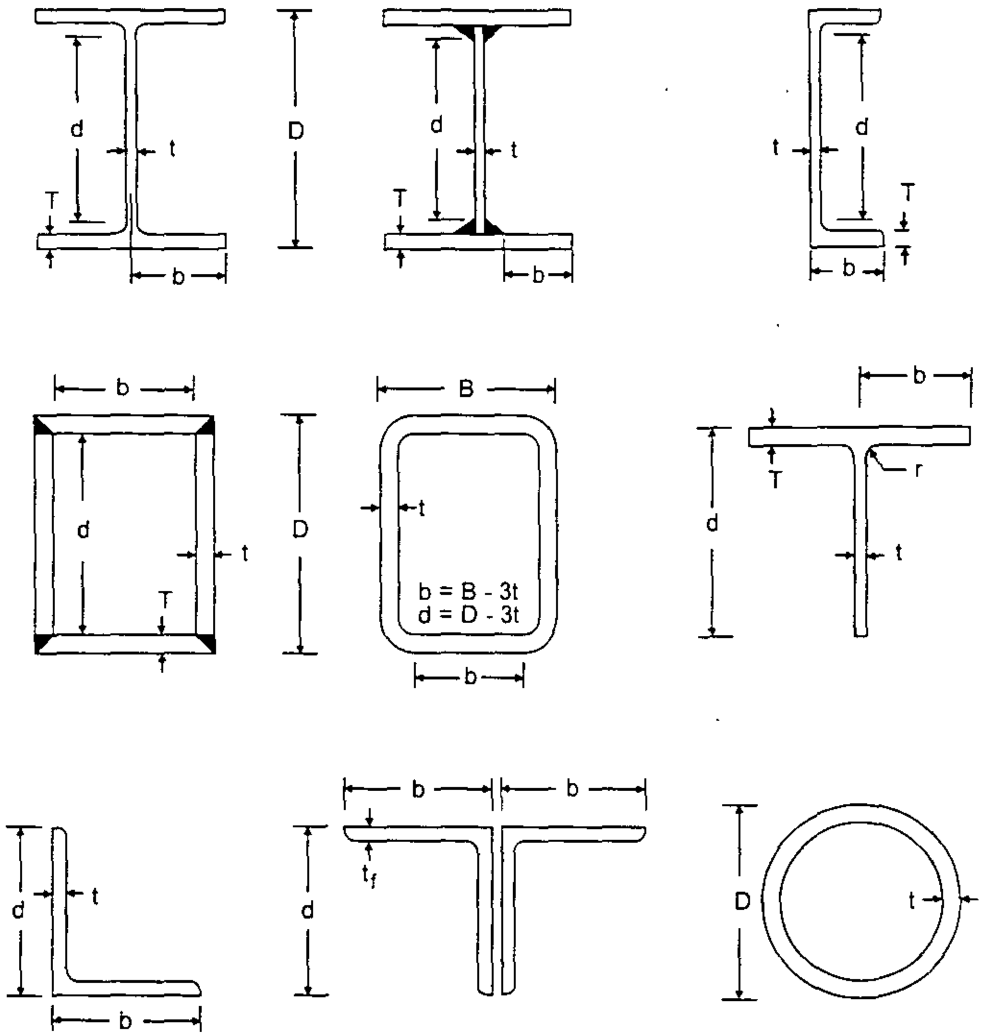
Table VII-2 (cont.)
Classification of Sections According to BS 5950

The moment magnification factor for moments causing sidesway can be taken as unity if a P-Δ analysis is carried out. SAP2000 design assumes a P-Δ analysis has been done and, therefore, $\varphi_{s,max}$ for both major and minor direction bending is taken as 0 in the following equation. It is suggested that the P-Δ analysis be done at the factored load level of 1.2 DL plus 1.2 LL. See also White and Hajjar (1991).

Calculation of Section Capacities

The nominal strengths in compression, tension, bending, and shear are computed for Class 1, 2, and 3 sections according to the following subsections. By default, SAP2000 takes the design strength, p_y, to be 1.0 times the minimum yield strength of steel, Y_s, as specified by the user. In inputting values of the yield strength, the user should ensure that the thickness and the ultimate strength limitations given in the code are satisfied (BS 3.1.1).

$$p_y = 1.0 Y_s \qquad \text{(BS 3.1.1)}$$



BS 5950 : Axes Conventions

3-3 is the cross-section axis parallel to the flanges or the smaller leg in angle sections. This is the same as the x-x axis.

2-2 is the cross-section axis perpendicular to the flanges or the smaller leg in angle sections. This is the same as the y-y axis.

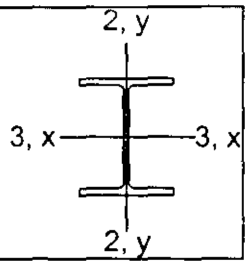


Figure VII-1

BS 5950 Definition of GeometricProperties

For Class 4 (Slender) sections and any singly symmetric and unsymmetric sections requiring special treatment, such as the consideration of local buckling, flexural-torsional and torsional buckling, or web buckling, reduced section capacities may be applicable. The user must separately investigate this reduction if such elements are used.

If the user specifies nominal stresses for one or more elements in the "Redefine Element Design Data", these values ***will override all above the mentioned calculated values for those elements*** *as defined in the following subsections.*

Compression Resistance

The compression resistance for plastic, compact, or semi-compact sections is evaluated as follows:

$$P_c = A_g \, p_c \, , \qquad \text{(BS 4.7.4)}$$

where p_c is the compressive strength given by

$$p_c = \frac{p_E \, p_y}{\varphi + (\varphi^2 - p_E \, p_y)^{1/2}} \, , \text{ where} \qquad \text{(BS C.1)}$$

$$\varphi = \frac{p_y + (\eta + 1) p_E}{2} \, ,$$

p_E = Euler strength, $\pi^2 E / \lambda^2$,

η = Perry factor, $0.001\, a(\lambda - \lambda_0) \geq 0$, (BS C.2)

a = Robertson constant from Table VII-3,

λ_0 = Limiting slenderness, $0.2\left(\dfrac{\pi^2 E}{p_y}\right)^{1/2}$, and (BS C.2)

λ is the slenderness ratio in either the major, $\lambda_{33} = l_{e33} / r_{33}$, or in the minor, $\lambda_{22} = l_{e22} / r_{22}$ direction. The larger of the two values is used in the above equations to calculate P_c.

For members in compression, if λ is greater than 180, a message to that effect is printed (BS 4.7.3.2). For single angles r_z is used instead of r_{33} and r_{22}.

Description of Section	Thickness (mm)	Axis of Bending	
		Major	Minor
I-SHAPE (rolled)	any	2.0	3.5
H-SHAPE (rolled)	≤40 >40	3.5 5 5	5.5 8.0
I-SHAPE (welded)	≤40 >40	3.5 3.5	5.5 8.0
BOX (welded)	≤40 >40	3.5 5.5	3.5 5.5
RECTANGULAR or CIRCLE	≤40 >40	3 5 5.5	3.5 5.5
ANGLE, CHANNEL, T-SHAPE	any	5.5	5.5
GENERAL	any	5 5	5 5

Table VII-3
Robertson Constant in BS 5950

Tension Capacity

The tension capacity of a member is given by

$$P_t = A_g \, p_y \, . \qquad \text{(BS 4.6.1)}$$

It should be noted that no net section checks are made. For main members in tension, the slenderness, λ, should not be greater than 250 (BS 4.7.3.2). If λ is greater than 250, a message is displayed accordingly. For brace members, a similar message is displayed when slenderness exceeds 350 (BS 4.7.3.2).

The user may have to separately investigate the members which are connected eccentrically to the axis of the member, for example angle sections.

Moment Capacity

The moment capacities in the major and minor directions, M_{c33} and M_{c22} are based on the design strength and the section modulus, the co-existent shear and the possibility of local buckling of the cross-section. Local buckling is avoided by applying a limitation to the width/thickness ratios of elements of the cross-section. The moment capacities are calculated as follows:

Plastic and Compact Sections

For plastic and compact sections, the moment capacities about the major and the minor axes of bending depend on the shear force, F_v, and the shear capacity, P_v.

For **I-, Box-, and Channel-sections** bending about the 3-3 axis the moment capacities considering the effects of shear force are computed as

$$M_c = \rho_y S \leq 1.2\, \rho_y Z, \qquad \text{for} \quad F_v \leq 0.6 P_v, \qquad \text{(BS 4.2.5)}$$

$$M_c = \rho_y (S - S_v \rho_1) \leq 1.2\, \rho_y Z, \qquad \text{for} \quad F_v > 0.6 P_v, \qquad \text{(BS 4.2.6)}$$

where

S = Plastic modulus of the gross section about the relevant axis,

Z = Elastic modulus of the gross section about the relevant axis,

S_v = Plastic modulus of the gross section about the relevant axis less the the plastic modulus of that part of the section remaining after deduction of shear area i.e. plastic modulus of shear area. For example, for rolled I-shapes S_{v2} is taken to be $tD^2/4$ and for welded I-shapes it is taken as $td^2/4$,

P_v = The shear capacity described later in this chapter,

ρ_1 = $\dfrac{2.5 F_v}{P_v} - 1.5$.

The combined effect of shear and axial forces is not being considered because practical situations do not warrant this. In rare cases, however, the user may have to investigate this independently, and if necessary, overwrite values of the section moduli.

For **all other cases**, the reduction of moment capacities for the presence of shear force is not considered. The user should investigate the reduced moment capacity separately. The moment capacity for these cases is computed in SAP2000 as

$$M_c = \rho_y S \leq 1.2\, \rho_y Z. \qquad \text{(BS 4.2.5)}$$

Semi-compact Sections

Reduction of moment capacity due to coexistent shear does not apply for semi-compact sections.

$$M_c = \rho_y Z \qquad \text{(BS 4.2.5)}$$

Lateral-Torsional Buckling Moment Capacity

The lateral torsional buckling resistance moment, M_b, of a member is calculated from the following equations. The program assumes the members to be uniform (of constant properties) throughout their lengths. Furthermore members are assumed to be symmetrical about at least one axis.

For **I-, Box-, and Channel-sections** M_b is obtained from

$$M_b = \frac{\rho_y S_{33} M_E}{\varphi_B + (\varphi_B^2 - \rho_y S_{33} M_E)^{1/2}}, \text{ where} \qquad \text{(BS B.2.1)}$$

$$\varphi_B = \frac{\rho_y S_{33} + (\eta_{LT} + 1) M_E}{2},$$

$M_E =$ The elastic critical moment, $\dfrac{S_{33} \pi^2 E}{\lambda_{LT}^2}$, and (BS B2.2)

$\eta_{LT} =$ The Perry coefficient.

The Perry coefficient, η_{LT}, for rolled and welded sections is taken as follows:

For rolled sections

$$\eta_{LT} = \alpha_b \{\lambda_{LT} - \lambda_{L0}\} \geq 0, \text{ and} \qquad \text{(BS B2.2)}$$

for welded sections

$$\eta_{LT} = 2\alpha_b \lambda_{L0} \geq 0, \text{ with } \alpha_b(\lambda_{LT} - \lambda_{L0}) \leq \eta_{LT} \leq 2\alpha_b(\lambda_{LT} - \lambda_{L0}). \text{(BS B2.2)}$$

In the above definition of η_{LT}, λ_{L0} and λ_{LT} are the limiting equivalent slenderness and the equivalent slenderness, respectively, and α_b is a constant. α_b is taken as 0.007. For flanged members symmetrical about at least one axis and uniform throughout their length, λ_{L0} and λ_{LT} are defined as follows:

$$\lambda_{L0} = 0.4\sqrt{\frac{\pi^2 E}{p_y}}, \qquad \text{(BS B2.4)}$$

$$\lambda_{LT} = n\,u\,v\,\lambda, \text{ where} \qquad \text{(BS B2.5)}$$

- λ is the slenderness and is equivalent to l_{22}/r_{22}.
- n is the slenderness correction factor. For flanged members in general, not loaded between adjacent lateral restraints, and for cantilevers without intermediate lateral restraints, n is taken as 1.0. For members with equal flanges loaded between adjacent lateral restraints, the value of n is conservatively taken as given by the following formula. This, however, can be overwritten by the user for any member by specifying it (BS Table 13).

  $$n = \frac{1}{\sqrt{C_b}}, \text{ where}$$

  $$C_b = \frac{12.5\, M_{max}}{2.5\, M_{max} + 3\, M_A + 4\, M_B + 3\, M_C}, \text{ and}$$

  M_{max}, M_A, M_B, and M_C are absolute values of maximum moment, 1/4 point, center of span and 3/4 point major moments respectively, in the member. The program also defaults C_b to 1.0 if the unbraced length, l, of the member is redefined by the user (i.e. it is not equal to the length of the member). C_b should be taken as 1.0 for cantilevers. However, the program is unable to detect whether the member is a cantilever. The user can overwrite the value of C_b for any member.
- u is the buckling parameter. It is conservatively taken as 0.9 for rolled I-shapes and channels. For any other section, u is taken as 1.0 (BS 4.3.7.5).
- v is the slenderness factor. It is given by the following formula for symmetric sections. This equation is a simplified version of equation (BS B.2.5). The torsional index used in the calculation of the slenderness factor is simplified to D/T (BS 4.3.7.5).

$$v = \frac{1}{\left\{1 + \frac{1}{20}\left[\frac{\lambda T}{D}\right]^2\right\}^{\frac{1}{4}}} . \qquad \text{(BS B.2.5)}$$

For **all other sections**, lateral torsional buckling is not considered. The user should investigate moment capacity considering lateral-torsional buckling separately.

Shear Capacities

The shear capacities for both the major and minor direction shears in I-shapes, boxes or channels are evaluated as follows:

$$P_{v2} = 0.6\,\rho_y\,A_{v2}\text{, and} \qquad \text{(BS 4.2.3)}$$

$$P_{v3} = 0.6\,\rho_y\,A_{v3}\,. \qquad \text{(BS 4.2.3)}$$

Moreover, the shear capacity computed above is valid only if $d/t \leq 63\,\varepsilon$, strictly speaking. For $d/t > 63\,\varepsilon$, the shear buckling of the thin members should be checked independently by the user in accordance with the code (BS 4.4.5).

Calculation of Capacity Ratios

In the calculation of the axial force/biaxial moment capacity ratios, first, for each station along the length of the member, for each load combination, the actual member force/moment components are calculated. Then the corresponding capacities are calculated. Then, the capacity ratios are calculated at each station for each member under the influence of each of the design load combinations. The controlling compression and/or tension capacity ratio is then obtained, along with the associated station and load combination. A capacity ratio greater than 1.0 indicates exceeding a limit state.

During the design, the effect of the presence of bolts or welds is not considered. Also, the joints are not designed.

Local Capacity Check

For members under axial load and moments, local capacity ratios are calculated as follows:

Under Axial Tension

A simplified approach allowed by the code is used to check the local capacity for plastic and compact sections.

$$\frac{F_t}{A_g \rho_y} + \frac{M_{33}}{M_{c33}} + \frac{M_{22}}{M_{c22}} \qquad \text{(BS 4.8.2)}$$

Under Axial Compression

Similarly, the same simplified approach is used for axial compression.

$$\frac{F_c}{A_g \rho_y} + \frac{M_{33}}{M_{c33}} + \frac{M_{22}}{M_{c22}} \qquad \text{(BS 4.8.3.2)}$$

Overall Buckling Check

In addition to local capacity checks, which are carried out at section level, a compression member with bending moments is also checked for overall buckling in accordance with the following interaction ratio:

$$\frac{F_c}{A_g \rho_c} + \frac{m_{33} M_{33}}{M_{b33}} + \frac{m_{22} M_{22}}{\rho_y Z_{22}} \qquad \text{(BS 4.8.3.3.1)}$$

The equivalent uniform moment factor, m, for members of uniform section and with flanges, not loaded between adjacent lateral restraints, is defined as

$$m = 0.57 + 0.33\beta + 0.10\beta^2 \, . \qquad \text{(BS Table 18)}$$

For other members, the value of m is taken as 1.0. The program defaults m to 1.0 if the unbraced length, l, of the member is overwritten by the user (i.e. if it is not equal to the length of the member). The user can overwrite the value of m for any member by specifying it. β is the ratio of the smaller end moment to the larger end moment on a span equal to the unrestrained length, being positive for single curvature bending and negative for double curvature bending.

Shear Capacity Check

From the factored shear force values and the shear capacity values at each station, shear capacity ratios for major and minor directions are produced for each of the load combinations as follows:

$\frac{F_{v2}}{P_{v2}}$, and

$\frac{F_{v3}}{P_{v3}}$.

Chapter VIII

Check/Design for EUROCODE 3

This chapter describes the details of the structural steel design and stress check algorithms that are used by SAP2000 when the user selects the Eurocode 3 design code (CEN 1992). The program investigates the limiting states of strength and stability but does not address the serviceability limit states. Various notations used in this chapter are described in Table VIII-1.

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

In the evaluation of the axial force/biaxial moment capacity ratios at a station along the length of the member, first the actual member force/moment components and the corresponding capacities are calculated for each load combination. Then the capacity ratios are evaluated at each station under the influence of all load combinations using the corresponding equations that are defined in this section. The controlling capacity ratio is then obtained. A capacity ratio greater than 1.0 indicates exceeding a limit state. Similarly, a shear capacity ratio is calculated separately.

English as well as SI and MKS metric units can be used for input. But the code is based on Newton-Millimeter-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Newton-Millimeter-Second** units unless otherwise noted.

A	=	Gross cross-sectional area, mm^2
A_{v2}, A_{v3}	=	Areas for shear in the 2- and 3-directions, mm^2
E	=	Modulus of elasticity, Mpa
G	=	Shear modulus, Mpa
I_t	=	Torsion constant, mm^4
I_w	=	Warping constant, mm^6
I_{33}	=	Major moment of inertia, mm^4
I_{22}	=	Minor moment of inertia, mm^4
K	=	Effective length factor
L	=	Length, span, mm
K_{33}, K_{22}	=	Major and minor effective length factors
$M_{b\,Rd}$	=	Design buckling resistance moment, N-mm
M_{cr}	=	Elastic critical moment for lateral-torsional buckling, N-mm
$M_{g\,Sd}$	=	Design moments not causing sidesway, N-mm
$M_{s\,Sd}$	=	Design moments causing sidesway, N-mm
$M_{33\,Sd}$	=	Design value of moment about the major axis, N-mm
$M_{22\,Sd}$	=	Design value of moment about the minor axis, N-mm
$M_{33\,Rd}$	=	Design moment resistance about the major axis, N-mm
$M_{22\,Rd}$	=	Design moment resistance about the minor axis, N-mm
$N_{b\,Rd}$	=	Design buckling resistance of a compression member, N
$N_{b33\,Rd}$	=	Design buckling resistance of a compression member about the major axis, N
$N_{b22\,Rd}$	=	Design buckling resistance of a compression member about the minor axis, N
$N_{c\,Sd}$	=	Design value of compressive force, N
$N_{c\,Rd}$	=	Design compression resistance, N
$N_{t\,Sd}$	=	Design value of tensile force, N
$N_{t\,Rd}$	=	Design tension resistance, N
$N_{pl\,Rd}$	=	Design plastic shear resistance, N
$V_{2\,Sd}$	=	Design value of shear force in the major direction, N
$V_{3\,Sd}$	=	Design value of shear force in the minor direction, N
$V_{2\,Rd}$	=	Design shear resistance in the major direction, N
$V_{3\,Rd}$	=	Design shear resistance in the minor direction, N
$W_{el\,33}, W_{el\,22}$	=	Major and minor elastic section moduli, mm^3
$W_{pl\,33}, W_{pl\,22}$	=	Major and minor plastic section moduli, mm^3

Table VIII-1
Eurocode 3 Notations

b	=	Width, mm
c	=	Distance, mm
d	=	Depth of web, mm
f_y	=	Nominal yield strength of steel, MPa
h	=	Overall depth, mm
l_{33}, l_{22}	=	Major and minor direction unbraced member lengths, mm
i_{33}, i_{22}	=	Major and minor radii of gyration, mm
i_z	=	Minimum radius of gyration for angles, mm
k_{33}, k_{22}	=	Factors applied to the major and minor design moments in the interaction equations
k_{LT}	=	Factor applied to the major design moments in the interaction equation checking for failure due to lateral-torsional buckling
t	=	Thickness, mm
t_f	=	Flange thickness, mm
t_w	=	Web thickness, mm
α	=	Ratio used in classification of sections
γ_{M0}, γ_{M1}	=	Material partial safety factors
ε	=	$\left[\dfrac{235}{f_y}\right]^{1/2}$ (f_y in MPa)
ρ	=	Reduction factor
τ_{ba}	=	Post-critical shear strength, MPa
χ_{33}, χ_{22}	=	Reduction factors for buckling about the 3-3 and 2-2 axes
χ_{LT}	=	Reduction factor for lateral-torsional buckling
ψ	=	Ratio of smaller to larger end moment of unbraced segment
ψ_s	=	Amplification factor for sway moments

Table VIII-1

Eurocode 3 Notations (cont.)

Design Loading Combinations

The design loading combinations define the various factored combinations of the load cases for which the structure is to be checked. The design loading combinations are obtained by multiplying the characteristic loads with appropriate partial factors of safety. If a structure is subjected to dead load (DL) and live load (LL) only, the design will need only one loading combination, namely 1.35 DL + 1.5 LL.

However, in addition to the dead load and live load, if the structure is subjected to wind (WL) or earthquake induced forces (EL), and considering that wind and earthquake forces are subject to reversals, the following load combinations may have to be considered (EC3 2.3.3):

1.35 DL
1.35 DL + 1.50 LL (EC3 2.3.3)

1.35 DL ± 1.50 WL
1.00 DL ± 1.50 WL
1.35 DL + 1.35 LL ± 1.35 WL (EC3 2.3.3)

1.00 DL ± 1.00 EL
1.00 DL + 1.5*0.3 LL ± 1.0 EL (EC3 2.3.3)

In fact, these are the default load combinations which can be used or overwritten by the user to produce other critical design conditions. These default loading combinations are produced for persistent and transient design situations (EC3 2.3.2.2) by combining forces due to dead, live, wind, and earthquake loads for ultimate limit states. See also section 9.4 of Eurocode 1 (CEN 1994) and Table 1.3, and 4 and section 4 of United Kingdom National Application Document (NAD).

The default load combinations will usually suffice for most building design. The user should use other appropriate loading combinations if roof live load is separately treated, other types of loads are present, or if pattern live loads are to be considered.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

In addition to the loads described earlier, equivalent lateral load cases for geometric imperfection should be considered by the user. This equivalent load is similar to the notional load of the British code, and depends on the number of stories and number of columns in any floor (EC3 5.2.4.3). Additional load combinations are also needed for these load cases.

When using Eurocode 3, SAP2000 design assumes that a P-Δ analysis has been performed so that moment magnification factors for moments causing sidesway can be taken as unity. It is suggested that the P-Δ analysis should be done at the factored load level corresponding to 1.35 dead load plus 1.35 live load. See also White and Hajjar (1991).

Classification of Sections

The design strength of a cross-section subject to compression due to moment or axial load depends on its classification as Class 1 (Plastic), Class 2 (Compact), Class 3 (Semi-compact), or Class 4 (Slender).

According to Eurocode 3, the classification of sections depends on the classification of flange and web elements. The classification also depends on whether the compression elements are in pure compression, pure bending, or under the influence of combined axial force and bending (EC3 5.3.2). SAP2000 conservatively classifies the compression elements for only pure compression according to Table VIII-2 and for pure bending according to Table VIII-3, depending on whether the member is a column or a beam. The section dimensions used in the tables are given in Figure VIII-1. If the section dimensions satisfy the limits shown in the tables, the section is classified as Class 1, Class 2, or Class 3 as applicable. A cross-section is classified by reporting the highest (least favorable) class of its compression elements.

If a section fails to satisfy the limits for Class 3 sections, the section is classified as Class 4. Currently SAP2000 does not check stresses for Class 4 sections.

One of the major factors in determining the limiting width-thickness ratio is ε. This parameter is used to reflect the influence of yield stress on the section classification.

$$\varepsilon = \sqrt{\frac{235}{f_y}} \qquad \text{(EC3 5.3.2)}$$

Calculation of Factored Forces

The internal design loads which are calculated for each load combination are $N_{t\,Sd}$ or $N_{c\,Sd}$, $M_{33\,Sd}$, $M_{22\,Sd}$, $V_{2\,Sd}$ and $V_{3\,Sd}$ corresponding to design values of the tensile or compressive axial load, the major moment, the minor moment, the major direction shear and the minor direction shear respectively. These design loads are calculated at each of the previously defined stations of each frame element.

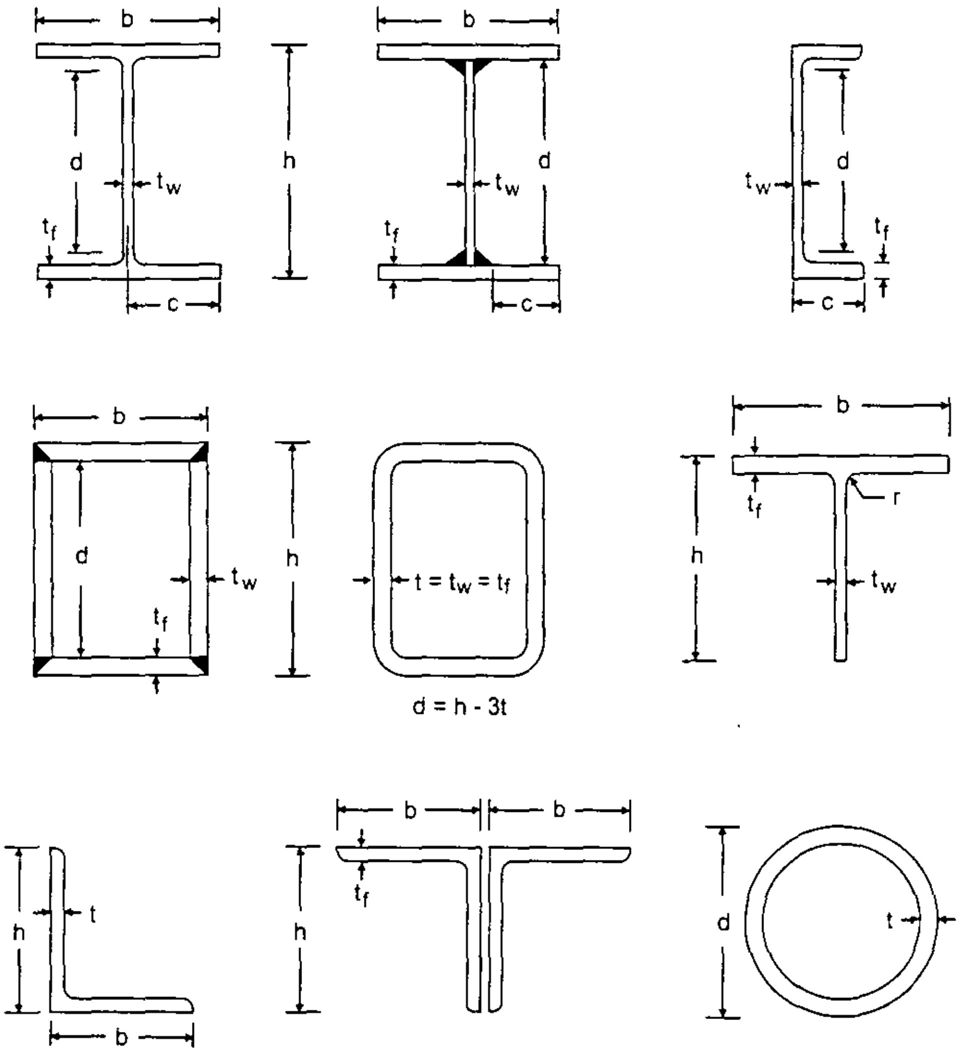
Section	Element	Ratio Checked	Class 1	Class 2	Class 3
GENERAL	—	None	Assumed Class 3		
RECTANGLE	—	None	Assumed Class 2		
I-SHAPE	web	d/t_w	33 ε	38 ε	42 ε
	flange	c/t_f (rolled)	10 ε	11 ε	15 ε
		c/t_f (welded)	9 ε	10 ε	14 ε
BOX	web	d/t_w	33 ε	38 ε	42 ε
	flange	$(b-3t_f)/t_f$ (rolled)	42 ε	42 ε	42 ε
		b/t_f (welded)	42 ε	42 ε	42 ε
CHANNEL	web	d/t_w	33 ε	38 ε	42 ε
	flange	b/t_f	10 ε	11 ε	15 ε
T-SHAPE	web	d/t_w	33 ε	38 ε	42 ε
	flange	$b/2t_f$ (rolled)	10 ε	11 ε	15 ε
		$b/2t_f$ (welded)	9 ε	10 ε	14 ε
ANGLE	—	h/t $(b+h)/(2t)$	Not applicable	Not applicable	15ε 11.5ε
ROUND BAR	—	None	Assumed Class 1		
PIPE	—	d/t	$50\varepsilon^2$	$70\varepsilon^2$	$90\varepsilon^2$
DOUBLE ANGLES	—	h/t $(b+h)/(2t)$	Not applicable	Not applicable	15ε 11.5ε

Table VIII-2
Classification of Sections According to Eurocode 3
Limiting Width-Thickness Ratios for Compression Elements

Section	Element	Ratio Checked	Class 1	Class 2	Class 3
GENERAL	—	None	Assumed Class 3		
RECTANGLE	—	None	Assumed Class 2		
I-SHAPE	web	d/t_w	72 ε	83 ε	124 ε
	flange	c/t_f (rolled)	10 ε	11 ε	15 ε
		c/t_f (welded)	9 ε	10 ε	14 ε
BOX	web	d/t_w	72 ε	83 ε	124 ε
	flange	$(b-3t_f)/t_f$ (rolled)	33 ε	38 ε	42 ε
		b/t_f (welded)	33 ε	38 ε	42 ε
CHANNEL	web	d/t_w (Major axis)	72 ε	83 ε	124 ε
		d/t_w (Minor axis)	33 ε	38 ε	42 ε
	flange	b/t_f	10 ε	11 ε	15 ε
T-SHAPE	web	d/t_w	33 ε	38 ε	42 ε
	flange	$b/2t_f$ (rolled)	10 ε	11 ε	15 ε
		$b/2t_f$ (welded)	9 ε	10 ε	14 ε
ANGLE	—	h/t $(b+h)/(2t)$	Not applicable	Not applicable	15.0ε 11.5ε
ROUND BAR	—	None	Assumed Class 1		
PIPE	—	d/t	$50\varepsilon^2$	$70\varepsilon^2$	$90\varepsilon^2$
DOUBLE ANGLES	—	h/t $(b+h)/(2t)$	Not applicable	Not applicable	15.0 ε 11.5 ε

Table VIII-3

Classification of Sections According to Eurocode 3
Limiting Width-Thickness Ratios for Flexure Elements



EUROCODE 3 : Axes Conventions

3-3 is the cross-section axis parallel to the flanges or the smaller leg in angle sections. This is the same as the y-y axis.

2-2 is the cross-section axis perpendicular to the flanges or the smaller leg in angle sections. This is the same as the z-z axis.

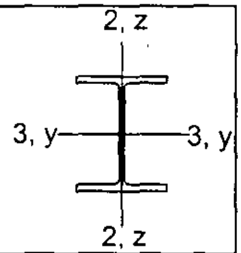


Figure VIII-1
Eurocode 3 Definition of Geometric Properties

The design moments and forces need to be corrected for second order effects. This correction are different for the so called "sway" and "nonsway" components of the moments. The code requires that the additional sway moments introduced by the horizontal deflection of the top of a story relative to the bottom must be taken into account in the elastic analysis of the frame in one of the following ways (EC3 5.2.6.2):

- Directly — by carrying out the global frame analysis using P-Δ analysis. Member design can be carried out using inplane buckling lengths for nonsway mode.
- Indirectly — by modifying the results of a linear elastic analysis using an approximate method which makes allowance for the second order effects. There are two alternative ways to do this — "amplified sway moment method" or "sway mode inplane buckling method".

The advantage of the direct second order elastic analysis is that this method avoids uncertainty in approximating the buckling length and also avoids splitting up moments into their "sway" and "nonsway" components.

SAP2000 design assumes that P-Δ effects are included in the analysis. Therefore any magnification of sidesway moments due to second order effects is already accounted for, i. e. ψ_s in the following equation is taken as 1.0. It is suggested that the P-Δ analysis be done at the factored load level of 1.35 DL plus 1.35 LL. See also White and Hajjar (1991). However, the user can overwrite the values of ψ_s for both major and minor direction bending in which case M_{Sd} in a particular direction is taken as:

$$M_{Sd} = M_{g\,Sd} + \psi_s \, M_{s\,Sd}, \text{ where} \qquad \text{(EC3 5.2.6.2)}$$

$M_{g,Sd}$ = Design moments not causing translation, and
$M_{s\,Sd}$ = Design moments causing sidesway.

Moment magnification for non-sidesway moments is included in the overall buckling interaction equations.

Sway moments are produced in a frame by the action of any load which results in sway displacements. The horizontal loads can be expected always to produce sway moments. However, they are also produced by vertical loads if either the load or the frame are unsymmetrical. In the case of a symmetrical frame with symmetrical vertical loads, the sway moments are simply the internal moments in the frames due to the horizontal loads (EC3 5.2.6.2).

Calculation of Section Resistances

The nominal strengths in compression, tension, bending, and shear are computed for Class 1, 2, and 3 sections according to the following subsections. The material partial safety factors used by the program are:

$\gamma_{M0} = 1.1$, and (EC3 5.1.1)
$\gamma_{M1} = 1.1$. (EC3 5.1.1)

For Class 4 (Slender) sections and any singly symmetric and unsymmetric sections requiring special treatment, such as the consideration of local buckling, flexural-torsional and torsional buckling, or web buckling, reduced section capacities may be applicable. The user must separately investigate this reduction if such elements are used.

If the user specifies nominal stresses for one or more elements in the "Redefine Element Design Data", these values are ***will override all the above mentioned calculated values for those elements*** *as defined in the following subsections.*

Tension Capacity

The design tension resistance for all classes of sections is evaluated in SAP2000 as follows:

$$N_{t\,Rd} = A f_y / \gamma_{M0} \qquad \text{(EC3 5.4.3)}$$

It should be noted that the design ultimate resistance of the net cross-section at the holes for fasteners is not computed and checked. The user is expected to investigate this independently.

Compression Resistance

The design compressive resistance of the cross-section is taken as the smaller of the design plastic resistance of the gross cross-section ($N_{pl\,Rd}$) and the design local buckling resistance of the gross cross-section ($N_{b\,Rd}$).

$$N_{c\,Rd} = \min(N_{pl\,Rd}, N_{b\,Rd}) \qquad \text{(EC3 5.4.4)}$$

The plastic resistance of Class 1, Class 2, and Class 3 sections is given by

$$N_{pl\,Rd} = A f_y / \gamma_{M0}\,. \qquad \text{(EC3 5.4.4)}$$

The design buckling resistance of a compression member is taken as

$$N_{b,Rd} = \chi_{min}\, \beta_A\, A f_y / \gamma_{M1}\,, \quad \text{where} \qquad \text{(EC3 5.5.1)}$$

$\beta_A = 1$, for Class 1, 2 or 3 cross-sections.

χ is the reduction factor for the relevant buckling mode. This factor is calculated below based on the assumption that all members are of uniform cross-section.

$$\chi = \frac{1}{\varphi + [\varphi^2 - \bar{\lambda}^2]^{1/2}} \leq 1, \quad \text{in which} \qquad \text{(EC3 5.5.1.2)}$$

$$\varphi = 0.5[1 + \alpha(\bar{\lambda} - 0.2) + \bar{\lambda}^2],$$

$$\bar{\lambda} = \left\{\frac{\lambda}{\lambda_1}\right\}[\beta_A]^{0.5},$$

$\lambda = \dfrac{K_{33}\, l_{33}}{i_{33}}$ or $\dfrac{K_{22}\, l_{22}}{i_{22}}$. The two values of λ give χ_3 and χ_2. χ_{min} is the lesser of the two.

$K = \dfrac{l}{L} \leq 1$, K is conservatively taken as 1 in SAP2000 design (EC3 5.5.1.5). The user can however override this default option if it is deemed necessary. An accurate estimate of K can be obtained from the Annex E of the code.

l is the buckling length,

L is the length of the column,

i is the radius of gyration about the neutral axis, and is determined using the properties of the gross cross-section,

$$\lambda_1 = \pi\left[\frac{E}{f_y}\right]^{1/2}, \text{ and}$$

α is an imperfection factor and is obtained from Table VIII-4. Values of this factor for different types of sections, axes of buckling, and thickness of materials are obtained from Tables 5.5.1 and 5.5.3 of the code.

Section	Limits	α (major axis)	α (minor axis)
GENERAL	any	0.49	0.49
RECTANGLE	any	0.49	0.49
I-SHAPE (rolled) h/b > 1.2	$t_f \leq 40$ mm	0.21	0.34
	$t_f > 40$ mm	0.34	0.49
I-SHAPE (rolled) h/b ≤ 1.2	$t_f \leq 100$ mm	0.34	0.49
	$t_f > 100$ mm	0.76	0.76
I-SHAPE (welded)	$t_f \leq 40$ mm	0.34	0.49
	$t_f > 40$ mm	0.49	0.76
BOX	Rolled welded	0.21 0.34	0.21 0.34
CHANNEL	any	0.49	0.49
T-SHAPE	any	0.49	0.49
ANGLE	any	0.49	0.49
ROUND BAR	any	0.49	0.49
PIPE	any	0.21	0.21
DOUBLE ANGLES	any	0.49	0.49

Table VIII-4

The α factor for different sections and different axes of buckling

Angle, Channel, and T-sections in compression are subjected to an additional moment due to the shift of the centroidal axis of the effective cross-section (EC3 5.4.4). SAP2000 does not currently considers this eccentricity. The user is expected to investigate this issue separately.

Shear Capacity

The design shear resistance of a section is the minimum of the plastic shear capacity and the buckling shear capacity. For all types of sections, the plastic shear resistance is computed as

$$V_{Rd} = V_{pl\,Rd} = \frac{A_v f_y}{\sqrt{3}} / \gamma_{M0}, \qquad \text{(EC3 5.4.6)}$$

where A_v is the effective shear area for the section and the appropriate axis of bending.

The buckling shear capacities are only computed for the I-, Box-, and Channel-sections if the width-thickness ratio is large ($d/t_w > 69\varepsilon$). The capacities are computed as

$$V_{Rd} = V_{ba\,Rd} = d\,t_w\,\tau_{ba} / \gamma_{M1}, \qquad \left(\text{for } \frac{d}{t_w} > 69\varepsilon\right) \qquad \text{(EC3 5.6.3)}$$

where, τ_{ba} is the simple post-critical shear strength which is determined as follows:

$$\tau_{ba} = \frac{f_{yw}}{\sqrt{3}}, \qquad \text{for } \bar{\lambda}_w \leq 0.8, \qquad \text{(EC3 5.6.3)}$$

$$\tau_{ba} = \left[1 - 0.625(\bar{\lambda}_w - 0.8)\right]\frac{f_{yw}}{\sqrt{3}}, \qquad \text{for } 0.8 < \bar{\lambda}_w < 1.2, \text{ and} \qquad \text{(EC3 5.6.3)}$$

$$\tau_{ba} = \left[0.9 / \bar{\lambda}_w\right]\frac{f_{yw}}{\sqrt{3}}, \qquad \text{for } \bar{\lambda}_w \geq 1.2. \qquad \text{(EC3 5.6.3)}$$

in which $\bar{\lambda}_w$ is the web slenderness ratio,

$$\bar{\lambda}_w = \frac{d / t_w}{37.4\,\varepsilon\sqrt{k_\tau}}, \text{ and} \qquad \text{(EC3 5.6.3)}$$

and k_τ is the buckling factor for shear. For webs with transverse stiffeners at the supports but no intermediate transverse stiffeners,

$$k_\tau = 5.34. \qquad \text{(EC3 5.6.3)}$$

Moment Resistance

The moment resistance in the major and minor directions is based on the section classification. Moment capacity is also influenced by the presence of shear force and axial force at the cross section. If the shear force is less than half of the shear capacity, the moment capacity is almost unaffected by the presence of shear force. If the shear force is greater than half of the shear capacity, additional factors need to be considered.

If $V_{Sd} \leq 0.5 V_{pl\,Rd}$

- For Class 1 and Class 2 Sections

$$M_{c\,Rd} = M_{pl\,Rd} = W_{pl} f_y / \gamma_{M0}\,. \qquad \text{(EC3 5.4.5.2)}$$

- For Class 3 Sections

$$M_{c\,Rd} = M_{el\,Rd} = W_{el} f_y / \gamma_{M0}\,. \qquad \text{(EC3 5.4.5.2)}$$

If $V_{Sd} > 0.5 V_{pl\,Rd}$

- For I-, Box-, and Channel-sections bending about the 3-3 axis the moment capacities considering the effects of shear force are computed as

$$M_{v\,Rd} = \left[W_{pl} - \frac{\rho A_v^2}{4 t_w} \right] \frac{f_y}{\gamma_{M0}} \leq M_{c\,Rd}\,, \text{ where} \qquad \text{(EC3 5.4.7)}$$

$$\rho = \left[\frac{2\, V_{Sd}}{V_{pl.Rd}} - 1 \right]^2 .$$

- For all other cases, the reduction of moment capacities for the presence of shear force is not considered. The user should investigate the reduced moment capacity separately.

Lateral-torsional Buckling

For the determination of lateral-torsional buckling resistance, it is assumed that the section is uniform, doubly symmetric, under standard conditions of restraint at each end, and loaded through its shear center. The lateral-torsional buckling resistance of a beam is evaluated as,

$$M_{b\,Rd} = \chi_{LT}\, \beta_w\, W_{pl33} f_y / \gamma_{M1}\,, \text{ where} \qquad \text{(EC3 5.5.2)}$$

$\beta_w = 1$, for Class 1 and Class 2 sections,

$\beta_w = \dfrac{W_{el33}}{W_{pl33}}$, for Class 3 sections,

$$\chi_{LT} = \frac{1}{\varphi_{LT} + [\varphi_{LT}^2 - \bar{\lambda}_{LT}^2]^{1/2}} \le 1, \text{ in which}$$

$$\varphi_{LT} = 0.5\left[1 + \alpha_{LT}(\bar{\lambda}_{LT} - 0.2) + \bar{\lambda}_{LT}^2\right], \text{ where}$$

$\alpha_{LT} = 0.21$, for rolled sections,

$\alpha_{LT} = 0.49$, for welded sections, and

$$\bar{\lambda}_{LT} = \left[\frac{\beta_w W_{pl33} f_y}{M_{cr}}\right]^{0.5}, \text{ where}$$

$$M_{cr} = C_1 \frac{\pi^2 E I_{22}}{L^2}\left[\frac{I_w}{I_{22}} + \frac{L^2 G I_t}{\pi^2 E I_{22}}\right]^{0.5}, \qquad \text{(EC3 F1.1)}$$

I_t = The torsion constant,

I_w = The warping constant,

L = Laterally unbraced length for buckling about the minor axis,

$C_1 = 1.88 - 1.40\psi + 0.52\psi^2 \le 2.7$, and

ψ = The ratio of smaller to larger end moment of unbraced segment, $\dfrac{M_a}{M_b}$. ψ varies between -1 and 1 ($-1 \le \psi \le 1$). A negative value implies double curvature. M_a and M_b are end moments of the unbraced segment and M_a is less than M_b, $\left(\dfrac{M_a}{M_b}\right)$ being negative for double curvature bending and positive for single curvature bending. If any moment within the segment is greater than M_b, C_1 is taken as 1.0. The program defaults C_1 to 1.0 if the unbraced length, l of the member is overwritten by the user (i.e. it is not equal to the length of the member). C_1 should be taken as 1.0 for cantilevers. However, the program is unable to detect whether the member is a cantilever. The user can overwrite the value of C_1 for any member by specifying it.

If $\overline{\lambda}_{LT} \leq 0.4$, no allowance for lateral torsional buckling is required by code and is not made in the design.

Calculation of Capacity Ratios

In the calculation of the axial force/biaxial moment capacity ratios, first, for each station along the length of the member, for each load combination, the actual member force/moment components are calculated. Then the corresponding capacities are calculated. Then, the capacity ratios are calculated at each station for each member under the influence of each of the design load combinations. The controlling compression and/or tension capacity ratio is then obtained, along with the associated station and load combination. A capacity ratio greater than 1.0 indicates exceeding a limit state.

During the design, the effect of the presence of bolts or welds is not considered. Also, the joints are not designed.

Bending, Axial Compression, and Low Shear

When the design value of the coexisting shear, V_{Sd}, is less than half of the corresponding capacities for plastic resistance, $V_{pl,Rd}$ and buckling resistance, $V_{ba,Rd}$, i.e.

$$V_{Sd} \leq 0.5 V_{pl,Rd}, \text{ and} \qquad \text{(EC3 5.4.9)}$$

$$V_{Sd} \leq 0.5 V_{ba,Rd}, \qquad \text{(EC3 5.4.9)}$$

the capacity ratios are computed for different types of sections as follows:

For Class 1 and Class 2 sections, the capacity ratio is conservatively taken as

$$\frac{N_{c,Sd}}{N_{pl,Rd}} + \frac{M_{33,Sd}}{M_{pl,33,Rd}} + \frac{M_{22,Sd}}{M_{pl,22,Rd}}. \qquad \text{(EC3 5.4.8.1)}$$

For Class 3 sections, the capacity ratio is conservatively taken as

$$\frac{N_{c,Sd}}{A f_{yd}} + \frac{M_{33,Sd}}{W_{el,33} f_{yd}} + \frac{M_{22,Sd}}{W_{el,22} f_{yd}}, \text{ where} \qquad \text{(EC3 5.4.8.1)}$$

$$f_{yd} = \frac{f_y}{\gamma_{M0}}.$$

Bending, Axial Compression, and High Shear

When the design value of the coexisting shear, V_{Sd}, is more than half the corresponding capacities for plastic resistance, $V_{pl\,Rd}$ or buckling resistance, $V_{ba\,Rd}$, the shear is considered to be high, i.e. the shear is high if

$$V_{Sd} > 0.5\, V_{pl\,Rd}\,, \text{ or} \qquad \text{(EC3 5.4.9)}$$

$$V_{Sd} > 0.5\, V_{ba\,Rd}\,. \qquad \text{(EC3 5.4.9)}$$

Under these conditions, the capacity ratios are computed for different types of sections as follows (EC3 5.4.9):

For Class 1, 2, and 3 sections, the capacity ratio is conservatively taken as

$$\frac{N_{c\,Sd}}{N_{pl\,Rd}} + \frac{M_{33\,Sd}}{M_{V33\,Rd}} + \frac{M_{22\,Sd}}{M_{V22\,Rd}}\text{, where} \qquad \text{(EC3 5.4.8.1)}$$

$M_{V33\,Rd}$ and $M_{V22\,Rd}$ are the design moment resistances about the major and and the minor axes, respectively, considering the effect of high shear (see page 108).

Bending, Compression, and Flexural Buckling

For all members of Class 1, 2, and 3 sections subject to axial compression, N_{Sd}, major axis bending, $M_{33\,Sd}$, and minor axis bending, $M_{22\,Sd}$, the capacity ratio is given by

$$\frac{N_{c\,Sd}}{N_{b\,min\,Rd}} + \frac{k_{33}\,M_{33\,Sd}}{\eta M_{c\,33\,Rd}} + \frac{k_{22}\,M_{22\,Sd}}{\eta M_{c\,22\,Rd}}\text{, where} \qquad \text{(EC3 5.5.4)}$$

$$N_{b\,min\,Rd} = \min\left\{N_{b\,33\,Rd},\ N_{b\,22\,Rd}\right\},$$

$$\eta = \frac{\gamma_{M0}}{\gamma_{M1}},$$

$$k_{33} = 1 - \frac{\mu_{33}\,N_{c\,Sd}}{\chi_{33}\,A f_y} \leq 1.5,$$

$$k_{22} = 1 - \frac{\mu_{22}\,N_{c\,Sd}}{\chi_{22}\,A f_y} \leq 1.5,$$

$$\mu_{33} = \bar{\lambda}_{33}\,(2\beta_{M33} - 4) + \left[\frac{W_{pl33} - W_{el33}}{W_{el33}}\right] \leq \; 0.9, \text{ (Class 1 and Class 2)},$$

$$\mu_{22} = \bar{\lambda}_{22}\,(2\beta_{M22} - 4) + \left[\frac{W_{pl22} - W_{el22}}{W_{el22}}\right] \leq \; 0.9, \text{ (Class 1 and Class 2)},$$

$$\mu_{33} = \bar{\lambda}_{33}\,(2\beta_{M33} - 4) \leq \; 0.9, \qquad \text{(for Class 3 sections)},$$

$$\mu_{22} = \bar{\lambda}_{22}\,(2\beta_{M22} - 4) \leq \; 0.9, \qquad \text{(for Class 3 sections)},$$

β_{M33} = Equivalent uniform moment factor for flexural buckling about the 3-3 (major) axis between points braced in 2-2 direction, and

β_{M22} = Equivalent uniform moment factor for flexural buckling about the 2-2 (minor) axis between points braced in 3-3 direction.

The equivalent uniform moment factors, β_{M33} and β_{M22}, are determined from

$$\beta_M = (1.8 - 0.7\,\psi) + \frac{M_Q}{\Delta M}\,(0.7\,\psi - 0.5), \text{ and}$$

M_Q = Absolute maximum moment due to lateral load only assuming simple support at the ends,

ψ = Absolute value of the ratio of smaller to larger end moment. ψ varies between -1 and 1 ($-1 \leq \psi \leq 1$). A negative value implies double curvature.

ΔM = Absolute maximum value of moment for moment diagram without change of sign, and

ΔM = Sum of absolute maximum and absolute minimum value of moments for moment diagram with change of sign.

Bending, Compression, and Lateral-Torsional Buckling

For all members of Class 1, 2, and 3 sections subject to axial compression, N_{Sd}, major axis bending, $M_{33\,Sd}$, and minor axis bending, $M_{22\,Sd}$, the capacity ratio is given by

$$\frac{N_{c\,Sd}}{N_{b\,22\,Rd}} + \frac{k_{LT}\,M_{33\,Sd}}{M_{b\,Rd}} + \frac{k_{22}\,M_{22\,Sd}}{\eta\,M_{c\,22\,Rd}}, \text{ where} \qquad \text{(EC3 5.5.4)}$$

k_{22} and η are as defined in the previous subsection "Bending, Compression, and Flexural Buckling",

$$k_{LT} = 1 - \frac{\mu_{LT} N_{c\,Sd}}{\chi_{22} A f_y} \leq 1, \text{ where}$$

$$\mu_{LT} = 0.15\,\bar{\lambda}_{22}\,\beta_{M\,LT} - 0.15 \leq 0.9, \text{ and}$$

$\beta_{M\,LT}$ = Equivalent uniform moment factor for lateral-torsional buckling. It is determined for bending about the y-y axis and between two points braced in the y-y direction.

Bending, Axial Tension, and Low Shear

When the design value of the coexisting shear, V_{Sd}, is less than half of the corresponding capacities for plastic resistance, $V_{pl\,Rd}$ and buckling resistance, $V_{ba\,Rd}$, i.e.

$$V_{Sd} \leq 0.5 V_{pl\,Rd}, \text{ and} \qquad \text{(EC3 5.4.9)}$$

$$V_{Sd} \leq 0.5 V_{ba\,Rd}, \qquad \text{(EC3 5.4.9)}$$

the capacity ratios are computed for different types of sections as follows:

For Class 1 and Class 2 sections, the capacity ratio is conservatively taken as

$$\frac{N_{t\,Sd}}{N_{t\,Rd}} + \frac{M_{33\,Sd}}{M_{pl\,33\,Rd}} + \frac{M_{22\,Sd}}{M_{pl\,22\,Rd}}. \qquad \text{(EC3 5.4.8.1)}$$

For Class 3 sections, the capacity ratio is conservatively taken as

$$\frac{N_{t\,Sd}}{A f_{yd}} + \frac{M_{33\,Sd}}{W_{el\,33} f_{yd}} + \frac{M_{22\,Sd}}{W_{el\,22} f_{yd}}. \qquad \text{(EC3 5.4.8.1)}$$

Bending, Axial Tension, and High Shear

When the design values of the coexisting shear, V_{Sd}, is more than half the corresponding capacities for plastic resistance, $V_{pl\,Rd}$ or buckling resistance, $V_{ba\,Rd}$, the shear is considered to be high, i.e. the shear is high if

$$V_{Sd} > 0.5\,V_{pl\,Rd}, \text{ or} \qquad \text{(EC3 5.4.9)}$$

$$V_{Sd} > 0.5\,V_{ba\,Rd}. \qquad \text{(EC3 5.4.9)}$$

Under these conditions, the capacity ratios are computed for different types of sections as follows (EC3 5.4.9):

For Class 1, 2, and 3 sections, the capacity ratio is conservatively taken as

$$\frac{N_{t\,Sd}}{N_{t\,Rd}} + \frac{M_{33\,Sd}}{M_{V33\,Rd}} + \frac{M_{22\,Sd}}{M_{V22\,Rd}}. \qquad \text{(EC3 5.4.8.1)}$$

Bending, Axial Tension, and Lateral-Torsional Buckling

The axial tensile force has a beneficial effect for lateral-torsional buckling. In order to check whether the member fails under lateral-torsional buckling, the effective internal moment about the 3-3 axis is calculated as follows:

$$M_{eff.33\,Sd} = M_{33\,Sd} - \psi_{vec}\frac{N_{t\,Sd}W_{com\,33}}{A}, \text{ where} \qquad \text{(EC3 5.5.3)}$$

ψ_{vec} = 0.8 (according to the EC3 box value), and

$W_{com\,33}$ is the elastic section modulus for the extreme compression fiber.

For all members of Class 1, 2, and 3 sections subject to axial tension, $N_{t\,Sd}$, major axis bending, $M_{33\,Sd}$, and minor axis bending, $M_{22\,Sd}$, the capacity ratio is taken as

$$\frac{N_{t\,Sd}}{N_{t\,Rd}} + \frac{k_{LT}\,M_{33\,Sd}}{M_{b\,Rd}} + \frac{k_{22}\,M_{22\,Sd}}{\eta\,M_{c\,22\,Rd}} - \psi_{vec}\,k_{LT}\,\frac{N_{t,Sd}W_{com\,33}}{A\,M_{b\,Rd}}, \qquad \text{(EC3 5.5.4)}$$

where k_{LT}, k_{22} and η are as defined in the previous subsections.

Shear

From the design values of shear force at each station, for each of the load combinations and the shear resistance values, shear capacity ratios for major and minor directions are produced as follows:

$$\frac{V_{2\,Sd}}{V_{2\,Rd}} \text{ and } \frac{V_{3\,Sd}}{V_{3\,Rd}}.$$

Chapter IX

Design Output

Overview

SAP2000 creates design output in three different major formats — graphical display, tabular output, and member specific detailed design information.

The graphical display of steel design output includes input and output design information. Input design information includes design section labels, K-factors, live load reduction factors, and other design parameters. The output design information includes axial force-bending moment interaction ratios and shear stress ratios. All graphical output can be printed.

The tabular output can be saved in a file or printed. The tabular output includes most of the information which can be displayed. This is generated for added convenience to the designer.

The member specific detailed design information shows details of the calculation from the designer's point of view. It shows the design section dimensions, material properties, design and allowable stresses or factored and nominal strengths, and some intermediate results for all the load combinations at all the design sections of a specific frame member.

In the following sections, some of the typical graphical display, tabular output, and member specific detailed design information are described. Some of the design information is specific to the chosen steel design codes which are available in the program and is only described where required. The AISC-ASD89 design code is described in the latter part of this chapter. For all other codes, the design outputs are similar.

Graphical Display of Design Output

The graphical output can be produced either as color screen display or in gray-scaled printed form. Moreover, the active screen display can be sent directly to the printer. The graphical display of design output includes input and output design information.

Input design information, for the AISC-ASD89 code, includes

- Design section labels,
- K-factors for major and minor direction of buckling,
- Unbraced Length Ratios,
- C_m-factors,
- C_b-factors,
- Live Load Reduction Factors,
- δ_s-factors,
- δ_b-factors,
- design type,
- allowable stresses in axial, bending, and shear.

The output design information which can be displayed is

- Color coded P-M interaction ratios with or without values, and
- Color coded shear stress ratios.

The graphical displays can be accessed from the **Design** menu. For example, the color coded P-M interaction ratios with values can be displayed by selecting the **Display Design Info...** from the **Design** menu. This will pop up a dialog box called **Display Design Results**. Then the user should switch on the **Design Output** option button (default) and select **P-M Ratios Colors & Values** in the drop-down box. Then clicking the **OK** button will show the interaction ratios in the active window.

The graphics can be displayed in either 3D or 2D mode. The SAP2000 standard view transformations are available for all steel design input and output displays. For switching between 3D or 2D view of graphical displays, there are several buttons on the main toolbar. Alternatively, the view can be set by choosing **Set 3D View...** from the **View** menu.

The graphical display in an active window can be printed in gray scaled black and white from the SAP2000 program. To send the graphical output directly to the printer, click on the **Print Graphics** button in the **File** menu. A screen capture of the active window can also be made by following the standard procedure provided by the Windows operating system.

Tabular Display of Design Output

The tabular design output can be sent directly either to a printer or to a file. The printed form of tabular output is the same as that produced for the file output with the exception that for the printed output font size is adjusted.

The tabular design output includes input and output design information which depends on the design code of choice. For the AISC-ASD89 code, the tabular output includes the following. All tables have formal headings and are self-explanatory, so further description of these tables is not given.

Input design information includes the following:

- Load Combination Multipliers
  - Combination name,
  - Load types, and
  - Load factors.
- Steel Stress Check Element Information (code dependent)
  - Frame ID,
  - Design Section ID,
  - K-factors for major and minor direction of buckling,
  - Unbraced Length Ratios,
  - C_m-factors,
  - C_b-factors, and
  - Live Load Reduction Factors.
- Steel Moment Magnification Factors (code dependent)

- Frame ID,
- Section ID,
- Framing Type,
- δ_b-factors, and
- δ_s-factors.

The output design information includes the following:

- Steel Stress Check Output (code dependent)
  - Frame ID,
  - Section location,
  - Controlling load combination ID for P-M interaction,
  - Tension or compression indication,
  - Axial force-bending moment interaction ratio,
  - Controlling load combination ID for major and minor shear forces, and
  - Shear stress ratios.

The tabular output can be accessed by selecting **Print Design Tables...** from the **File** menu. This will pop up a dialog box. Then the user can specify the design quantities for which the results are to be tabulated. By default, the output will be sent to the printer. If the user wants the output stream to be redirected to a file, he/she can check the **Print to File** box. This will provide a default filename. The default filename can be edited. Alternatively, a file list can be obtained by clicking the **File Name** button to chose a file from. Then clicking the **OK** button will direct the tabular output to the requested stream — the file or the printer.

Member Specific Information

The member specific design information shows the details of the calculation from the designer's point of view. It provides an access to the geometry and material data, other input data, design section dimensions, design and allowable stresses, reinforcement details, and some of the intermediate results for a member. The design detail information can be displayed for a specific load combination and for a specific station of a frame member.

The detailed design information can be accessed by **right clicking** on the desired frame member. This will pop up a dialog box called **Steel Stress Check Information** which includes the following tabulated information for the specific member.

- Frame ID,
- Section ID,
- Load combination ID,
- Station location,
- Axial force-bending moment interaction ratio, and
- Shear stress ratio along two axes.

Additional information can be accessed by clicking on the **ReDesign** and **Details** buttons in the dialog box. Additional information that is available by clicking on the **ReDesign** button is as follows:

- Design Factors (code dependent)
  - Effective length factors, K, for major and minor direction of buckling,
  - Unbraced Length Ratios,
  - C_m-factors,
  - C_b-factors,
  - Live Load Reduction Factors,
  - δ_s-factors, and
  - δ_b-factors.
- Element Section ID
- Element Framing Type
- Overwriting allowable stresses

Additional information that is available by clicking on the **Details** button is given below.

- Frame, Section, Station, and Load Combination IDs,
- Section geometric information and graphical representation,
- Material properties of steel,
- Moment factors,
- Design and allowable stresses for axial force and biaxial moments, and
- Design and allowable stresses for shear.

References

AISC, 1989

Manual of Steel Construction. Allowable Stress Design, 9th Edition, American Institute of Steel Construction, Chicago, Ill, 1989.

AISC, 1994

Manual of Steel Construction, Load & Resistance Factor Design, 2nd Edition, American Institute of Steel Construction, Chicago, Ill, 1994.

BSI, 1990

Structural Use of Steelwork in Building, Part 1, Code of Practice for Design in Simple and Continuous Construction: Hot Rolled Sections, BS 5950 : Part 1 : 1990, British Standards Institution, London, UK, 1990.

CEN, 1992

Design of Steel Structures, Part 1.1 : General Rules and Rules for Buildings, ENV 1993-1-1 : 1992, European Committee for Standardization, Brussels, Belgium, 1992.

CISC, 1995

Handbook of Steel Construction, CAN/CSA-S16.1-94, 6th Edition, Canadian Institute of Steel Construction, Willodale, Ontario, Canada, 1995.

CSI, 1997a

SAP2000 Getting Started, Computers and Structures, Inc., Berkeley, California, 1997.

CSI, 1997b

SAP2000 Analysis Reference, Vols. I and II, Computers and Structures, Inc., Berkeley, California, 1997.

ICBO, 1997

Uniform Building Code, 1997, International Conference of Building Officials, Whittier, California, 1997.

D. W. White and J. F. Hajjar, 1991

"Application of Second-Order Elastic Analysis in LRFD: Research to Practice," *Engineering Journal, American Institute of Steel Construction, Inc.*, Vol. 28, No. 4, 1991.

Index

Automatic member selection, 19

Bending strength
- ASD (allowable), 40
- BS, 89
- CISC, 71
- Eurocode, 108
- LRFD, 57

Braced frames, 28
- BS, 85
- CISC, 67
- Eurocode, 103
- LRFD, 54

Capacity ratio, 2, 27
- ASD, 33, 44
- BS, 79
- CISC, 63, 76
- Eurocode, 44, 61, 76, 92, 95, 110
- LRFD, 47

Check stations, 27

Classification of sections
- ASD, 36
- BS, 83
- CISC, 67
- Eurocode, 99
- LRFD, 50

Compact section
- See Classification of sections

Compressive strength
- ASD (allowable), 39
- BS, 87
- CISC, 70
- Eurocode, 104
- LRFD, 55

Demonstration
- accessing detailed information, 14
- analysis, 10
- automatic member selection, 19
- editing member properties, 16, 19
- reanalysis, 21
- redesign, 15
- stress check, 12

Design output, 115
- graphical, 116
- member specific, 118
- tabular, 117

Effective length factor, 30

Euler buckling load
- ASD, 40
- BS, 87
- LRFD, 54

Factored forces and moments
BS, 83
CISC, 67
Eurocode, 99
LRFD, 54

Flexural buckling
ASD, 39
CISC, 70, 87
Eurocode, 104

Graphical output, 116

Interaction equations
See Capacity ratio

Interactive environment, 1

Lateral drift effect, 28
See Also P-Delta analysis

Lateral-torsional buckling
ASD, 40
BS, 90
CISC, 71
Eurocode, 108
LRFD, 57

Live load reduction factor, 27, 36, 50, 66, 82, 98

Loading combinations, 1, 26
ASD, 36
BS, 82
CISC, 66
Eurocode, 98
LRFD, 50

Member specific output, 118

Member stability effect, 28
See Also P-Delta analysis

Moment magnification
CISC, 67
Eurocode, 103
LRFD, 54

Noncompact section
See Classification of sections

Nonsway, 28
BS, 85
CISC, 67
Eurocode, 103
LRFD, 54

Notional load
BS, 82
CISC, 66
Eurocode, 98

Output, 2
details, 14, 119
graphical, 115
member specific, 13
tabular, 115

P-Delta analysis, 28
BS, 82, 85
CISC, 66 - 67
Eurocode, 99, 103
LRFD, 50, 55

P-Delta effects, 28

Perry factor, 87

Plastic section
See Classification of sections

Redesign, 119

Robertson constant, 87

Second order effects
See P-Delta effects

Shear strength
ASD (allowable), 43
BS, 92
CISC, 74
Eurocode, 107
LRFD, 60

Slender section
See Classification of sections

Strength reduction factors
- BS, 85

Supported design codes, 1
- ASD, 25, 33
- BS, 25, 79
- CISC, 25, 63
- Eurocode, 25, 95
- LRFD, 25, 47

Sway, 28
- BS, 85
- CISC, 67
- Eurocode, 103
- LRFD, 54

Tabular output, 117

Tensile strength
- ASD (allowable), 40
- BS, 88
- CISC, 70
- Eurocode, 104
- LRFD, 56

Unbraced frames, 28
- BS, 85
- CISC, 67
- Eurocode, 103
- LRFD, 54

Units, 2, 32
- ASD, 33
- BS, 79
- CISC, 63
- Eurocode, 95
- LRFD, 47

Unsupported length, 28

FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

SAP 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

NOTAS DEL CURSO

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
NOVIEMBRE DEL 2001

UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO
FACULTAD DE INGENIERÍA

INSTRUCTIVO PARA LA UTILIZACION DEL PROGRAMA DE COMPUTADORA SAP 2000

Análisis y Diseño Integrado de Estructuras por el Método de Elementos Finitos

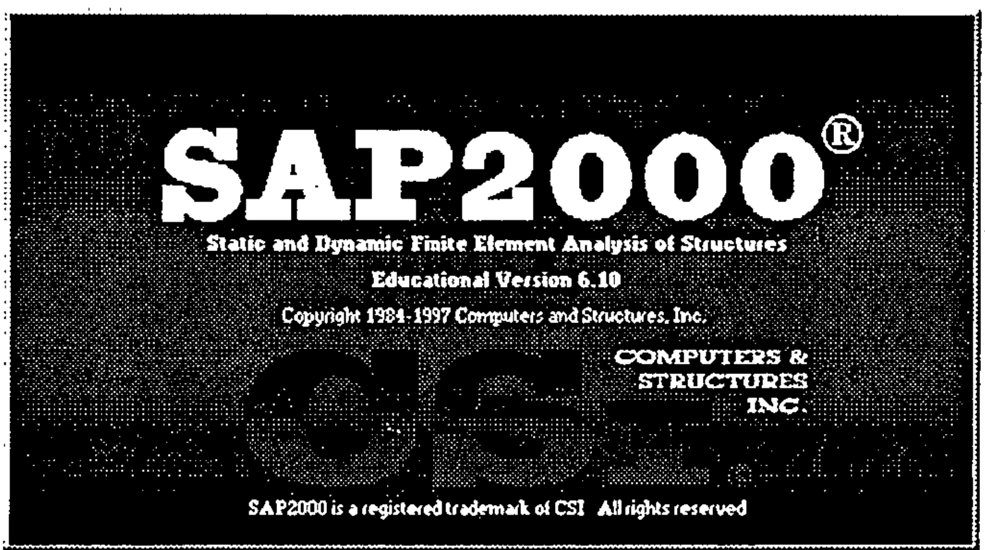
FERNANDO MONROY MIRANDA

DIVISIÓN DE INGENIERÍA CIVIL, TOPOGRÁFICA Y GEODÉSICA
DEPARTAMENTO DE ESTRUCTURAS
LABORATORIO DE CÓMPUTO ESTRUCTURAL.

UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO
FACULTAD DE INGENIERÍA

SAP-2000

(STRUCTURAL ANALYSIS PROGRAM
Análisis y Diseño Integrado de Estructuras por el Método de Elementos Finitos)



INSTRUCTIVO PARA LA UTILIZACION DEL PROGRAMA DE COMPUTADORA SAP 2000

FERNANDO MONROY MIRANDA
DIVISIÓN DE INGENIERÍA CIVIL, TOPOGRÁFICA Y GEODÉSICA
DEPARTAMENTO DE ESTRUCTURAS

PRÓLOGO

La serie de programas SAP son quizá los programas más conocidos, probados y utilizados en el campo de la Ingeniería Estructural, particularmente en el Análisis Estructural, desde las primeras versiones SOLIDSAP, SAP 3, SAP IV, etc., hasta la más reciente **SAP 2000**, han sido utilizadas por un gran número de ingenieros en nuestro país y en muchas partes del mundo, cuenta con respaldo y soporte técnico al que tiene derecho el usuario autorizado así como a los manuales respectivos.

Por lo anterior, desde hace algunos años el Departamento de Estructuras de la División de Ingeniería Civil Topográfica y Geodesia del la Facultad de Ingeniería de la UNAM consideró conveniente impartir una serie de cursos para enseñar a manejar el programa, para ello el contar con un instructivo que permita introducir al usuario de una manera fácil al programa facilitará el objetivo anterior, por lo que se sugiere que el lector asista a los cursos que organiza del Departamento de Estructuras o la División de Educación Continua de la FI de la UNAM.

En este instructivo se describen algunos de los principales elementos que intervienen en el uso del programa de computadora para Análisis y Diseño Estructural **SAP-2000**, cuya principal utilización será para los alumnos de la materia "Diseño Estructural" de la carrera de Ingeniero Civil que se imparte en la Facultad de Ingeniería de la UNAM.

Se ha procurado realizar este instructivo de una manera sencilla y resumida para que el usuario no emplee demasiado tiempo en leerlo y pueda resolver su problema en lo que respecta al Análisis y Diseño de Estructuras utilizando el programa **SAP-2000**.

Se recomienda que si algunos de los elementos no son descritos ampliamente se consulten los manuales respectivos o la ayuda en línea incluida en el programa y se observen los ejemplos que se desarrollan al final del instructivo. Se supone que el usuario esta familiarizado con la nomenclatura y terminología utilizada en el Análisis y Diseño Estructural y que cuenta con conocimientos básicos de computación en lo que respecta a manejo de información (archivos) y ejecución de programas en ambiente Windows 95, 98.

El autor agradece al Ing. Miguel Ángel Rodríguez Vega, Jefe del Departamento de Estructuras el apoyo para el desarrollo de este tipo de actividades, por las facilidades otorgadas para la realización de este trabajo así como la revisión del presente instructivo.

FERNANDO MONROY MIRANDA

Cd. Universitaria, Marzo del 2000

CONTENIDO

PRÓLOGO

CAPÍTULO 1 EL PROGRAMA SAP 2000

1.1 Introducción al programa SAP 2000

CAPÍTULO 2 RECOMENDACIONES PARA EL USO DEL PROGRAMA

2.1 Paso 1. Tipo de estructura
2.2 Paso 2. Definición de la geometría
2.3 Paso 3. Definición de las propiedades elásticas de los materiales
2.4 Paso 4. Definición de las propiedades geométricas de los elementos
2.5 Paso 5. Definición las características de las fuerzas y de las combinaciones
2.6 Paso 6. Elección del tipo de análisis y resultados
2.7 Paso 7. Diseño de elementos

CAPÍTULO 3 MÓDULOS DEL PROGRAMA DESCRIPCION GENERAL

3.1 Ejecución del programa, módulos que lo componen
3.2 Descripción general

CAPÍTULO 4 OPCIONES PARA LA GENERACIÓN DE LA ESTRUCTURA

4.1 Introducción
4.2 Descripción General
4.3 Generación de la Geometría
4.4 Definición y asignación de propiedades geométricas
4.5 Definición y Asignación materiales
4.6 Condiciones de Frontera, tipos de apoyo

4.7 Asignación de Fuerzas y combinaciones

4.8 Opciones de Análisis y Diseño, selección de resultados

CAPÍTULO 5 ANÁLISIS DE LA ESTRUCTURA

5.1 Verificando algunos elementos del proceso de análisis

CAPÍTULO 6 SELECCIÓN E INTERPRETACION DE RESULTADOS

6.1 Introducción

6.2 Ver la estructura deformada

6.3 Ver los diagramas de elementos mecánicos

6.4 Ver los resultados de diseño

6.5 Otras características

CAPÍTULO 7 OPCIONES ADICIONALES

7.1 Introducción

7.2 Ver el archivo de entrada

7.3 Ver el archivo de salida

7.4 Relación con AUTO CAD

CAPÍTULO 8 EJEMPLOS E INTERPRETACIÓN DE RESULTADOS

Ejemplo No. 1

Ejemplo No. 2

Ejemplo No. 3

Ejemplo No. 4

Ejemplo No. 5

CAPÍTULO 9 COMENTARIOS FINALES

EL PROGRAMA SAP 2000

CAPÍTULO I

1.1 INTRODUCCIÓN

En los últimos años, el desarrollo de los equipos y sistemas de computo ha permitido una comunicación mucho más rápida, directa y sencilla entre el usuario y la computadora logrando la posibilidad de desarrollar programas que, utilizando las características de las computadoras de hoy en día, nos permitan usarlas mas eficientemente y entre otras cosas facilitándonos la posibilidad de explorar varias alternativas de solución de problemas estructurales o bien considerar más variables en el comportamiento de las estructuras con el objeto de lograr un mejor modelo de la estructura.

Tomando en cuenta lo anterior, **SAP 2000** es el resultado de un trabajo desarrollado en los Estados Unidos de Norteamérica cuyo principal objetivo fue desarrollar un programa para Análisis y Diseño de Estructuras en donde el usuario tenga gran versatilidad en el manejo del mismo a través de una interacción directa en la mayor parte de la ejecución de los módulos que componen el programa y junto con la sencillez y facilidad de uso son algunas de sus principales características

El Sistema **SAP 2000** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis y Diseño de Estructuras bajo uno o varios sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura.

SAP 2000 fue desarrollado bajo la hipótesis de que la estructura está formada por barras prismáticas (aunque también maneja cierto tipo de barras de sección variable) de eje recto, considerando también la posibilidad de modelar elementos placa y sólido (Elementos finitos)

Consta básicamente de una serie de menús (Véase Figura 1) que se despliegan en la pantalla al inicio del programa y por lo general después de terminada la ejecución de cada una de las opciones, con ellas, el usuario puede introducir y/o modificar datos, o bien almacenarlos para su procesamiento posterior, analizar la estructura, ver resultados en la pantalla o imprimirlos, ver resultados de diseño, etc.

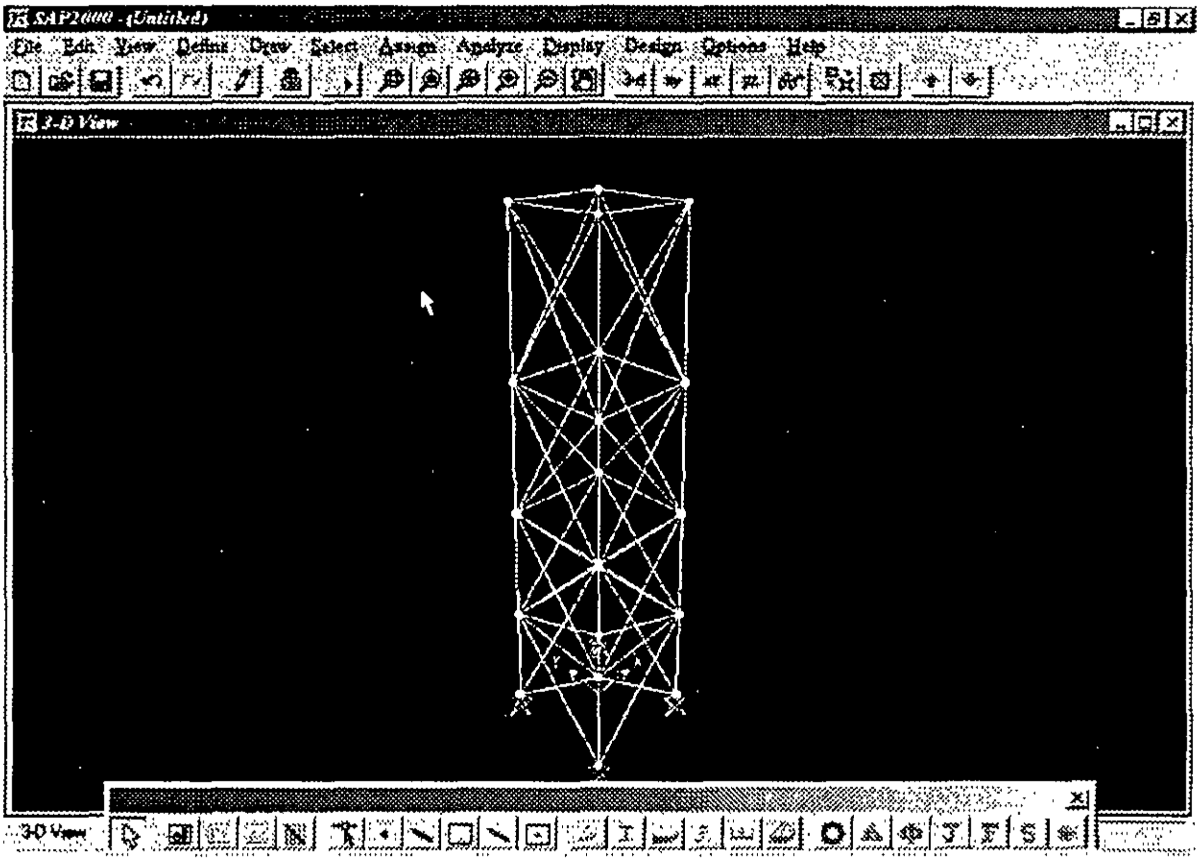


Figura 1.1 SAP 2000, menú principal.

Una de las principales características del programa es la interacción que se puede establecer entre éste y el usuario, y debido al número de opciones que el usuario puede activar, se requiere aprender su lenguaje específico para poder utilizarlo, ya que, el usuario puede seleccionar varias opciones y la ejecución de cada una de ellas genera otras más, **SAP 2000** es un programa orientado a eventos (seleccionar un elemento con el ratón, elegir una opción, activar/desactivar sucesos, etc.) y no siempre solicita textualmente los elementos (datos) que se vayan requiriendo para la ejecución completa de ese módulo, por otro lado además es necesario saber las convenciones de signos empleadas, los sistemas de referencia utilizados así como algunas recomendaciones para su uso, éstas y algunas características más son descritas en los capítulos posteriores.

En el capítulo 2 se dan las recomendaciones necesarias para facilitar la preparación e introducción de datos, en el capítulo 3 se comentan los módulos que componen el programa, el capítulo 4 describe el módulo para crear la estructura, en el capítulo 5 se presenta el módulo de análisis, en el capítulo 6 se presentan las opciones para ver resultados del Análisis y Diseño, en el capítulo 7 se describen algunas opciones adicionales o complementarias, el capítulo 8 contiene algunos ejemplos con la correspondiente interpretación de los resultados obtenidos por el programa **SAP 2000**, por último en el capítulo 9 se incluyen algunos comentarios y sugerencias finales.

RECOMENDACIONES PARA EL USO DEL PROGRAMA

CAPÍTULO 2

2.1 INTRODUCCION

El programa **SAP 2000** posee una interfase gráfica como una opción que le permite al usuario modelar, analizar, diseñar y desplegar tanto datos como resultados de una estructura, una vez que se cuenta con los datos de geometría, propiedades de los materiales de los cuales están hechos los elementos estructurales así como las cargas y desde luego un completo y correcto entendimiento del problema, se esta en condiciones de utilizar el programa, para ello habrá necesidad de modelar a los elementos anteriores, una vez definido el modelo que se utilizará para esos elementos se introducirá el modelo completo utilizando por ejemplo la interfase gráfica.

La estructura idealizada estará formada por:

- Elementos barra (FRAME) usados para representar a las vigas, columnas, diagonales, etc.
- Elementos placa (SHELL) usados para representar muros, losas, rampas, etc.
- Elementos sólidos (SOLID) usados para modelar estructuras continuas tridimensionales.
- Nudos (JOINTS) que representan la conexión entre los elementos barra, placa y sólido.
- Propiedades físicas y elásticas de los materiales
- Apoyos y resortes que representan las restricciones de desplazamiento del nudo.
- Cargas (concentradas, uniformes, etc.) que representan a las acciones (peso propio, viento, sismo, ocupación, etc.).

2.1 PASO 1. TIPO DE ESTRUCTURA

SAP 2000 permite manejar a la estructura en un sistema coordenado tridimensional, sin embargo, antes de realizar el análisis se pueden seleccionar determinados grados de libertad (ver figura 2.1) y así aunque la estructura este referida a un sistema tridimensional se pueden analizar:

Marcos y vigas en un plano vertical
Retículas (en un plano horizontal)

Desde luego se permite modelar y analizar Armaduras y marcos tridimensionales.

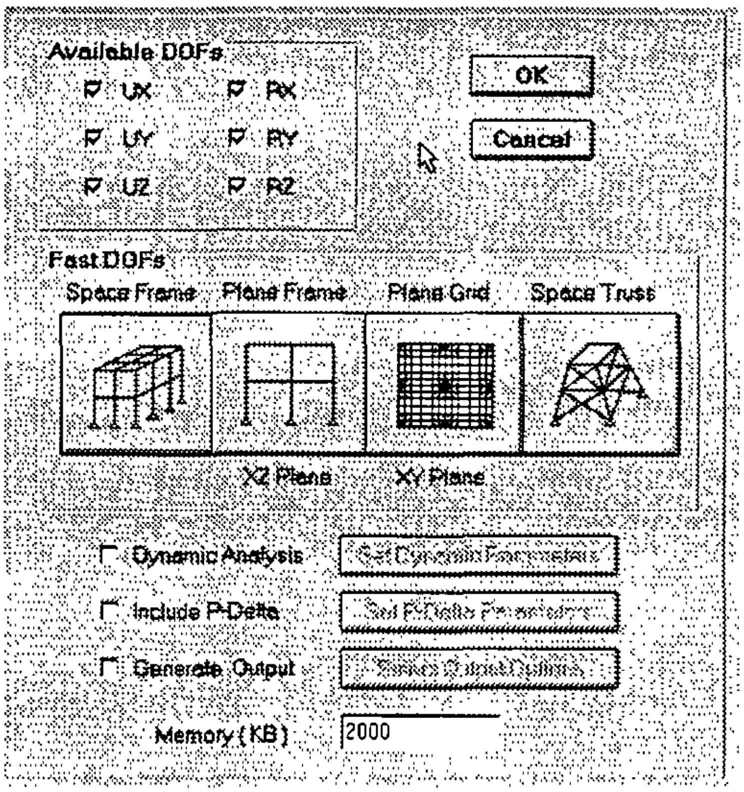


Figura 2.1. Selección de grados de libertad de acuerdo al tipo de estructura

Para el caso de las estructuras tipo armadura sólo se considerará el efecto axial en el análisis.

En las estructuras planas se consideran cortante y axial en el plano de la estructura y flexión perpendicular a ese plano.

El tipo retícula permite analizar estructuras con acciones perpendiculares a su plano considerando flexión en el plano, torsión y cortante.

El caso general lo constituye el tipo marco tridimensional en donde se consideran flexión y cortante en dos direcciones, torsión y axial con seis grados de libertad por nudo, desde luego que se pueden liberar extremos de las barras a algún elemento mecánico y suprimir o ligar grados de libertad (diafragma rígido por ejemplo).

2.2 PASO 2. DEFINICIÓN DE LA GEOMETRÍA

Antes de iniciar la ejecución del programa SAP 2000 es conveniente como segundo paso definir completamente la geometría del modelo. La estructura real se idealizará mediante una serie de elementos estructurales conectados entre sí, los cuales, de acuerdo a sus caracteristicas se podrán modelar como elementos barra (trabes, columnas, diagonales), elementos placa (losas, muros) o elementos sólidos tridimensionales (elementos continuos), estos elementos estarán unidos en puntos comunes (nudos), algunos nudos estarán completamente o parcialmente restringidos (apoyos), en uno o varios grados de libertad.

La definición de los elementos (barra, placa, sólido, etc.) se logra localizando sus nudos extremos (incidencias) en un sistema coordenado cartesiano proporcionando las coordenadas de esos nudos.

No es necesario numerar en ningún orden a los nudos que forman parte de la estructura ya que el programa los numera. Es conveniente localizar nudos en donde se tenga cambio de propiedades geométricas o elásticas, recordando que el elemento barra requiere de dos nudos para localizarlo, el elemento placa 3 ó 4 y el sólido comunmente 8 nudos.

Como se verá posteriormente el editor gráfico permite introducir la geometría de la estructura de una manera bastante sencilla y directa, ya que con la ayuda del "ratón" (dispositivo tipo puntero o *mouse*) simple y sencillamente por ejemplo haciendo clic en las coordenadas de los puntos extremos de la barra automáticamente se definen sus incidencias así como las coordenadas de esos nudos.

2.3 PASO 3. DEFINICIÓN DE LAS PROPIEDADES GEOMÉTRICAS DE LOS ELEMENTOS

SAP 2000 permite manejar una gran variedad de formas predefinidas para la sección transversal de las barras que componen la estructura (ver figura 2.2), como por ejemplo:

- Secciones I, canal, T, ángulos, ángulos dobles, cajón, tubos, etc.
- Secciones rectangulares, circulares.
- Secciones cualquiera (proporcionando sus propiedades)
- Sección no prismáticas (propiedades variables).

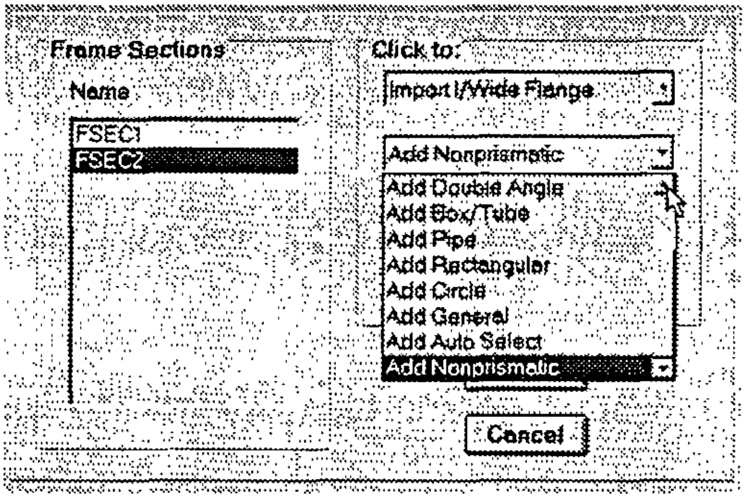


Figura 2.2. Algunas formas para la sección transversal de las barras

Una vez elegida la forma de la sección transversal será necesario introducir los datos relativos a las dimensiones (tamaño) de la forma seleccionada (ver figura 2.3).

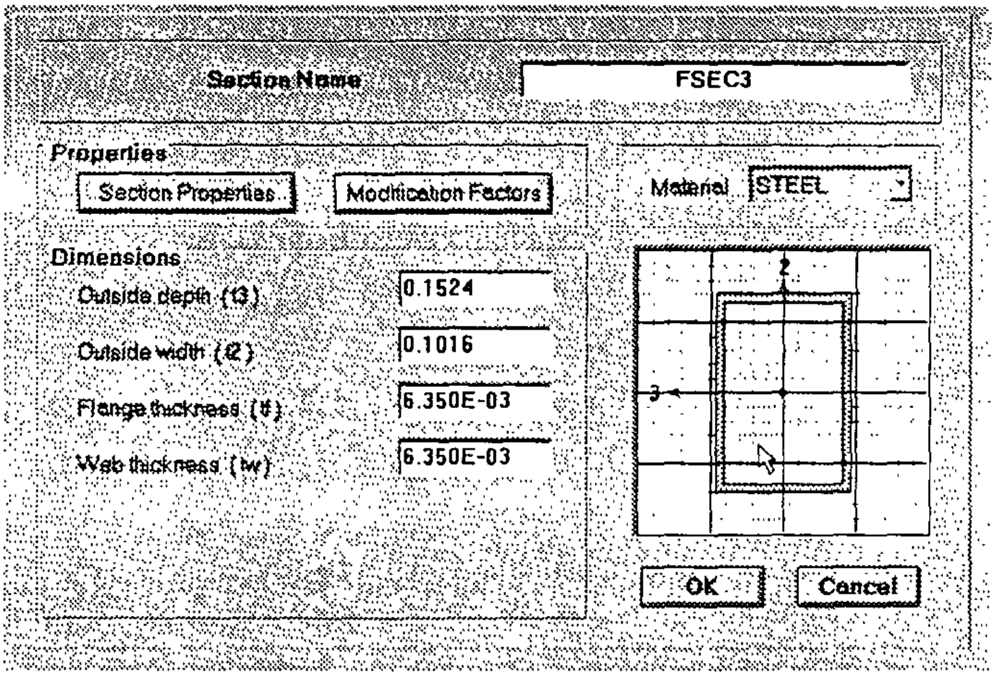


Figura 2.3. Dimensiones de una forma de sección transversal específica.

Para los elementos barra prismáticos (general) de una estructura tridimensional se requiere proporcionar las siguientes propiedades referidas a ejes locales, centroidales y principales de la barra (ver figura 2.4).

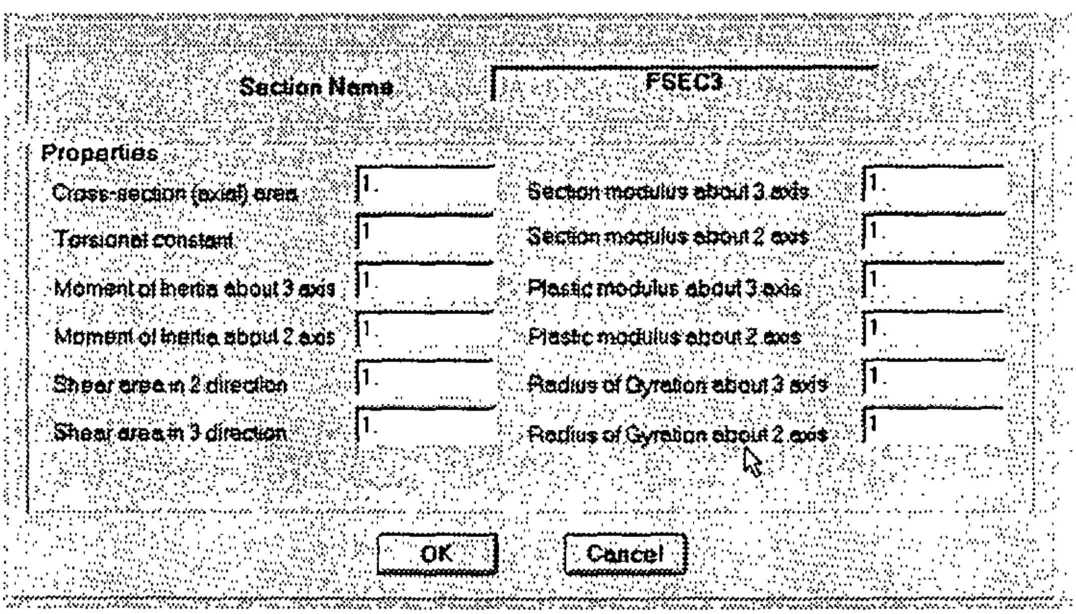


Figura 2.4. Características del tipo de sección transversal "general".

Dependiendo del tipo de estructura, en la tabla 2.1 se muestran las propiedades geométricas mínimas que es necesario proporcionar para que el análisis se pueda realizar.

Tipo de estructura	Propiedad requerida
TRUSS	AX
PLANE	AX, IZ ó IY
FLOOR	IX, IZ ó IY
SPACE	AX, IX, IY, IZ

Tabla 2.1 Propiedades geométricas mínimas requeridas.

El programa **SAP 2000** permite asignar las propiedades de los elementos barra de acuerdo a una tabla de perfiles de acero estándar (P. ej. tabla AISC, ver figura 2.5) o tomarlas de una tabla definida por el usuario.

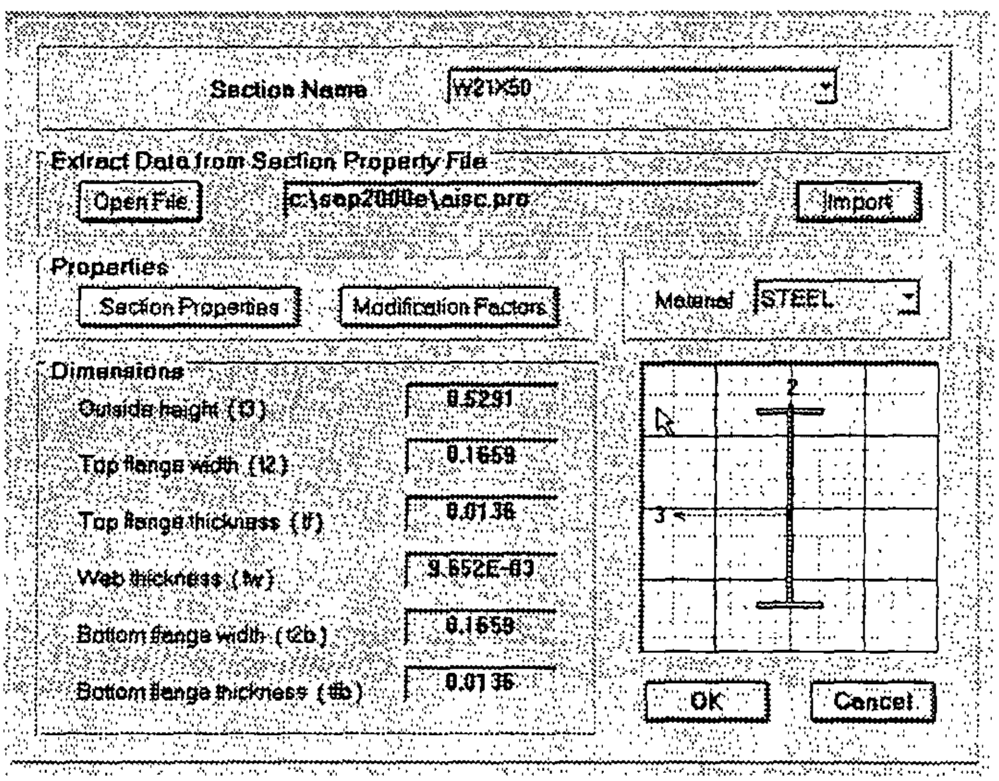


Figura 2.5. forma y propiedades geométricas tomadas de una tabla de perfiles.

Para el caso de los elementos placa será necesario proporcionar el espesor de la placa y seleccionar el tipo de trabajo de esta ("Shell", "Membrane" o "Plate", ver figura 2.6), para el sólido no es necesario proporcionar propiedades geométricas, sólo constantes elásticas

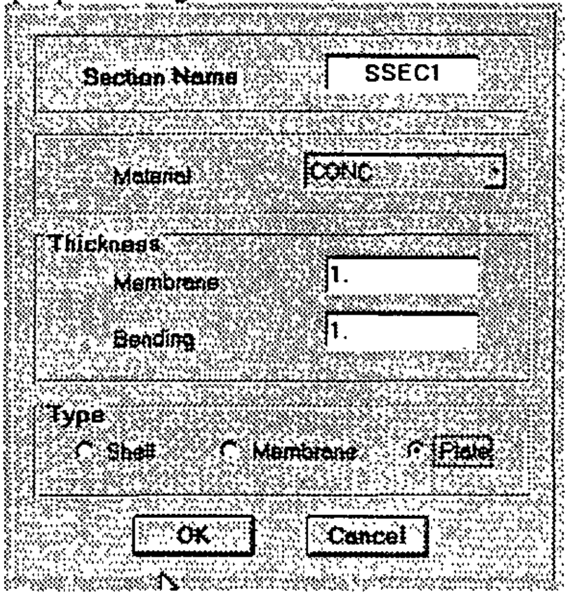


Figura 2.6. Datos para los elementos placa.

2.4 PASO 4. DEFINICIÓN DE LAS PROPIEDADES ELÁSTICAS DE LOS MATERIALES

Para realizar el análisis se requiere tener definidas las constantes del material del cual están o estarán hechos los elementos (barra, placa, sólido) como son E (Módulo elástico), y ν (relación de Poisson). Para incluir el peso propio es necesario proporcionar el peso volumétrico, si se desea emplear alguna opción de análisis dinámico entonces es necesario proporcionar la masa por unidad de longitud (en un modelo de masas distribuidas), masas en los nudos (modelo de masas concentradas), si quiere que se considere efectos de temperatura será necesario especificar el coeficiente lineal de dilatación térmica (ver figura 2.7).

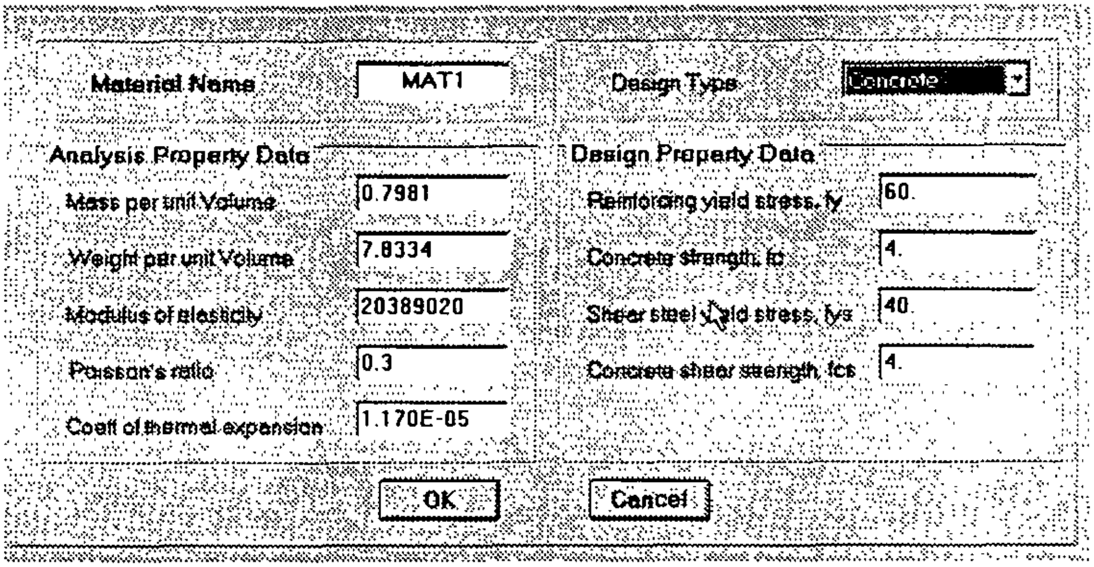


Figura 2.7. Datos para las propiedades de un material.

2.5 PASO 5. TIPOS DE FUERZAS Y COMBINACIONES

Es necesario tener completamente identificados comunmente los sistemas o conjuntos d· fuerzas (condiciones de carga) bajo los que se realizará el análisis (P. ej. peso propio, carga viva, sismo, viento, etc.) y para cada condición de carga las características de las fuerzas (tipo, magnitud, dirección, etc) que forman parte de ese sistema de fuerzas.

Por ejemplo una condición de carga puede ser la carga muerta que puede estar formada por ejemplo por: fuerzas uniformes en algunas barras simulando el peso de los muros divisorios, fuerzas concentradas simulando el peso de tanques, etc.

Otra condición de carga puede ser el sismo, que por ejemplo pudiera ser representado por una serie de fuerzas estáticas (sismo estático) aplicadas en determinados nudos.

Una condición más puede ser la carga viva, idealizada como una fuerza por unidad de área actuando en una determinada zona de la estructura (P. ej. azotea, entrepiso, escaleras, etc.).

Los sistemas de carga independientes pueden ser utilizados para formar sistemas de carga dependientes es decir combinaciones, si lo anterior se desea, es necesario saber de antemano el número de combinaciones a incluir en el análisis y para cada combinación las condiciones de carga que se incluirán así como su participación respectiva (factor de carga), por ejemplo teniendo como marco al Reglamento de Construcciones para el D.F. pensando en una estructura del grupo A, localizada en el D. F. una combinación es 1.5 de la carga muerta + 1.5 de la carga viva máxima, por lo que el factor de carga (o de participación) de las condiciones anteriores (1 y 2) es 1.5, siendo 1 y 2 las condiciones de carga respectivas.

2.6 PASO 6. ELECCIÓN DEL TIPO DE ANÁLISIS Y RESULTADOS

SAP 2000 permite realizar un análisis elástico lineal de 1er. orden, también se pueden incluir efectos P-Δ o bien un análisis dinámico, por lo anterior habrá que decidir sobre el tipo de análisis a efectuar por el programa.

En cuanto a los resultados que el programa puede proporcionar, será necesario saber cuales se requerirán, por ejemplo: desplazamientos, elementos mecánicos, gráficas y diseño, y de que elementos se requieren; por ejemplo: de algunos o de todos los nudos, de algunos o de todas las barras, gráficas de la deformada, de algún marco o de toda la estructura, etc., lo anterior se tendrá que definir para una, algunas o todas las condiciones de carga y/o combinaciones. Si el usuario no selecciona o define los elementos (nudos, barras, etc.), condiciones y/o combinaciones la impresión la realiza para todos los elementos y todos los sistemas de fuerzas existentes.

2.7 PASO 7. DISEÑO DE ELEMENTOS

SAP 2000 permite diseñar elementos de acero y concreto por lo que será necesario definir un código o especificaciones a utilizar (ACI, AISC, LRFD, ASSTHO, etc.) y proporcionar los valores de los parámetros a utilizar (f'c, fy, etc.), así como especificar los elementos que se diseñarán y el criterio a seguir para su diseño (viga, columna, etc.).

DESCRIPCION GENERAL

CAPÍTULO 3

3.1 INTRODUCCION

Una vez que se ha modelado la estructura (previo al uso del programa), es decir, seleccionada la forma de la sección transversal de las barras, definidas las características físicas y mecánicas de los materiales estructurales, especificados los sistemas de fuerzas (definidas cada una de las fuerzas que componen a cada sistema o condición de carga y combinaciones) bajo las cuales se analizará el modelo estructural, seleccionado el tipo de análisis así como el tipo de resultados, entonces se esta en condiciones de introducir los datos antes mencionados utilizando la interface gráfica que ofrece el programa con la cual es posible:

Manejar (Definir, mover, copiar, borrar) elementos estructurales (barra, placa, etc.).

Definir Tipos de apoyo (fijo o con grados de libertad, resortes).

Definir y asignar propiedades geométricas a los elementos barra de acuerdo a una tabla de perfiles estándar (AISC por ejemplo) o usar secciones prismáticas (circular, rectangular, Te, etc.), también es posible la utilización de secciones no prismáticas o de sección variable.

Definir el espesor de los elementos placa.

Definir y asignar propiedades a uno o varios elementos o grupo de elementos (barra, placas), las propiedades pueden ser densidad, módulo elástico, relación de Poisson, coeficiente de dilatación térmica, etc. Así como definir la posición de la sección dentro de la estructura (posición de ejes locales con respecto a los globales). Algunas de las propiedades se tienen predefinidas para ciertos materiales (acero y concreto) o se pueden introducir valores particulares.

Es posible seleccionar barras para liberarlos de algunos elementos mecánicos en sus extremos, también se pueden definir diafragmas rígidos.

Desde luego se permite introducir fuerzas estáticas aplicadas a los nudos, desplazamientos prescritos en ellos, en el caso de barras se puede incluir el peso propio, fuerzas uniformes, concentradas, con variación lineal, de presfuerzo y debidas a incrementos de temperatura, a ajustes en la longitud inicial de los elementos y algunas otras.

Además de las fuerzas de tipo estático, se puede incluir cargas variables (móviles), de acuerdo a AASHTO (HS2O, HS15, H2O, H15, etc.), o al UBC, o bien especificadas por el usuario. Una buena variedad de fuerzas dinámicas (fuerza-tiempo o aceleración-tiempo) pueden incluirse como sistemas de fuerzas, especificadas de acuerdo a sus características dinámicas (amplitud y frecuencia), definiendo el lapso de tiempo de actuación de la fuerza.

Una vez introducida la geometria, propiedades y fuerzas que actúan sobre la estructura, **SAP 2000** permite la realización del Análisis operando sobre el contenido del archivo que se ha seleccionado o definido previamente el cual desde luego debe contener los datos de la estructura en estudio, el módulo de análisis interpreta cada una de las ordenes o definiciones indicadas en el archivo de datos en el orden en que se encuentran, el contenido del archivo de datos e instrucciones puede introducirse manualmente vía algún editor previo a la ejecución de **SAP 2000** o bien mediante la instrucción **Save** al estar creando la estructura a través del editor gráfico caracteristico del programa, ambas opciones se describirán posteriormente.

Después de ejecutada la opción de análisis, **SAP 2000** genera archivos conteniendo los resultados de la fase de análisis, si este concluye satisfactoriamente se desplegará la configuración deformada de la estructura. Enseguida se podrán seleccionar opciones y elementos para que de ellos se muestren en el monitor los resultados numéricos y gráficos obtenidos por el programa como resultado del análisis.

3.2 EJECUCIÓN DEL PROGRAMA, MENU DE OPCIONES

Para iniciar el programa se puede hacer doble clic en el icono del programa o bien desde el menú de inicio hacer clic en la carpeta programas **SAP 2000 educacional** (versión educativa) o **SAP 2000 NonLinear** (versión profesional), enseguida se ejecuta el programa presentándose la imagen mostrada en la figura 3.1, una vez haciendo clic en la caja OK de la ventana en la parte central ("*Tip of the day*") desaparece esta dejando lugar a la ventana principal del programa **SAP 2000**.

En el "renglón" superior de esta ventana se encuentra en su extremo izquierdo el nombre del programa (**SAP2000**) seguido del nombre de archivo en donde se almacenarán los datos o de donde han sido tomados, en el extremo derecho se encuentran los iconos de minimizar, restaurar la ventana y cerrarla (una forma de finalizar la ejecución del programa es haciendo clic en este icono), debajo de lo anterior se localiza la barra de menú conteniendo las opciones que el programa tiene disponibles (File, Edit, View, etc.) las cuales se describirán posteriormente, debajo de esas opciones se encuentran una serie de iconos que realizan acciones de uso frecuente (seleccionar elementos, cambiar alguna opción de presentación, elegir algún tipo de resultado, etc.), se recomiendo al lector consultar las tablas que se presentan al final de este trabajo en donde se describe cada uno de esos iconos (incluyendo los de la barra flotante que también forma parte de la ventana de **SAP 2000**).

Debajo de los iconos está el área de presentación (con fondo negro) en la que se muestra gráficamente el modelo de la estructura por analizar así como diversa información en forma de ventanas que serán desplegadas por el programa después de que el usuario seleccione alguna de las opciones disponibles de **SAP 2000**.

Por último, en la parte inferior debajo de la barra flotante de iconos se muestra información acerca de las características del área de dibujo (vista o plano de presentación, coordenadas de algún nudo, etc.) y un poco a la derecha esta el cuadro de selección de unidades en las que se introducirá la información, antes de este cuadro se muestra información acerca del estado que guarda alguna instrucción o del programa.

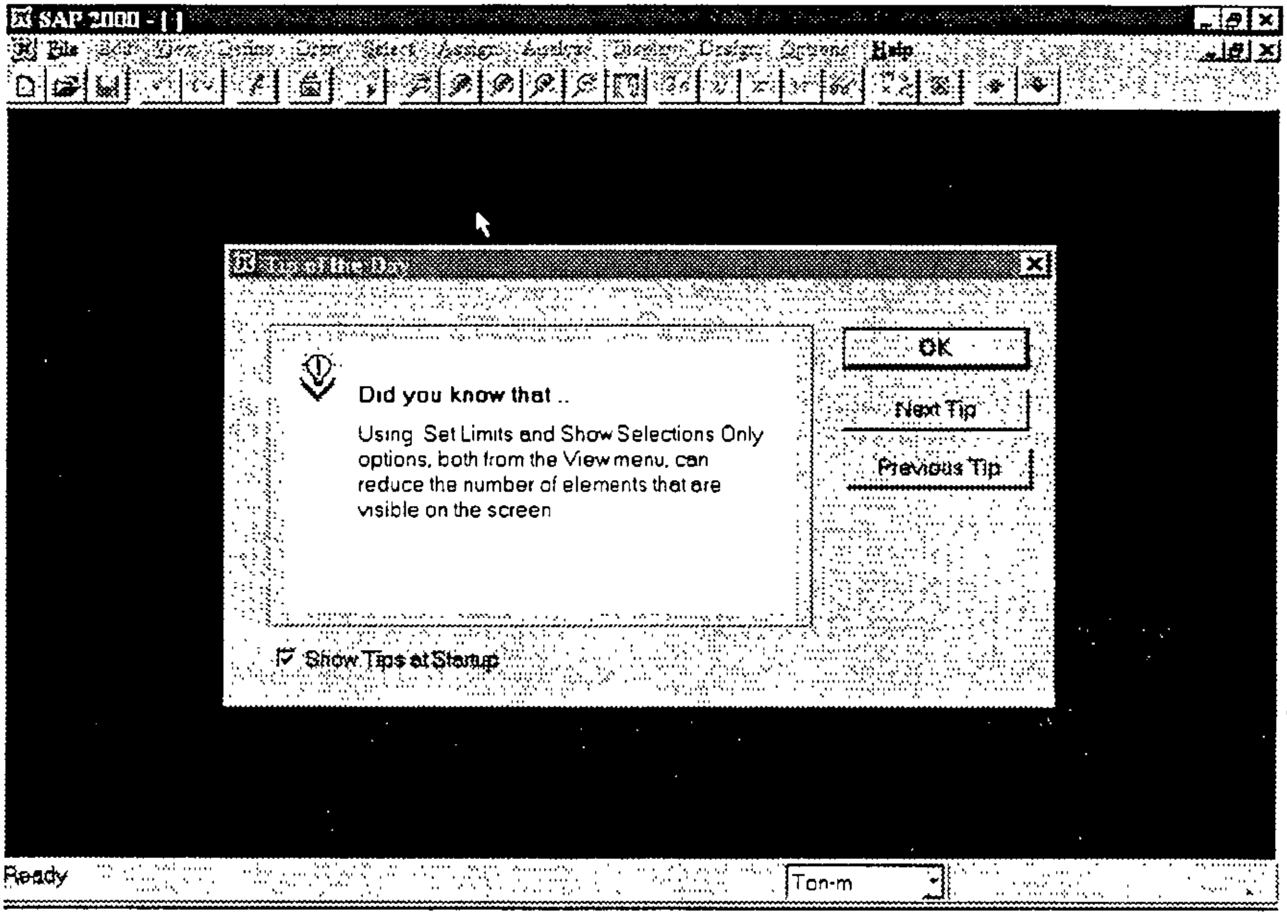


Figura 3.1 Iniciando el programa **SAP 2000**

En la versión 6.1 del programa SAP 2000 se pueden seleccionar varias opciones, las que se describen a continuación pueden ser las de uso más frecuente.

3.3 El menú File

EL menú File (ver figura 3.2) permite entre otras opciones manejar la información de alguna estructura contenida en un archivo, esa información pudo haberse generado previamente a la ejecución del programa o durante su uso, las opciones de este menú permiten:

New Model Iniciar un problema nuevo.

New Model from template Iniciar un problema nuevo, seleccionando una geometría típica de algunas formas estructurales como las mostradas en la figura 3.3.

Open... Abrir un archivo existente con datos de alguna estructura.

Save	Guarda los datos de la estructura.
Save As	Guarda los datos de la estructura en otro archivo.
Import	Permite ingresar los datos de un archivo generado con AutoCad, o bien para SAP90.
Export	Proporciona la flexibilidad de poder enviar los datos de la estructura existente a una archivo para **SAP2000** con extensión .S2K el cual puede ser modificado por ciertos procesadores de texto (p.ej. WordPad) y poder ser utilizado nuevamente por **SAP2000,** o bien enviarlos a un archivo .DXF y poder ser interpretado por AutoCad por ejemplo.
Print...	Nos permite configurar características de impresión, imprimir el contenido del área de dibujo así como una lista de datos y resultados.
Exit	Cerrar el programa y regresar a Windows.

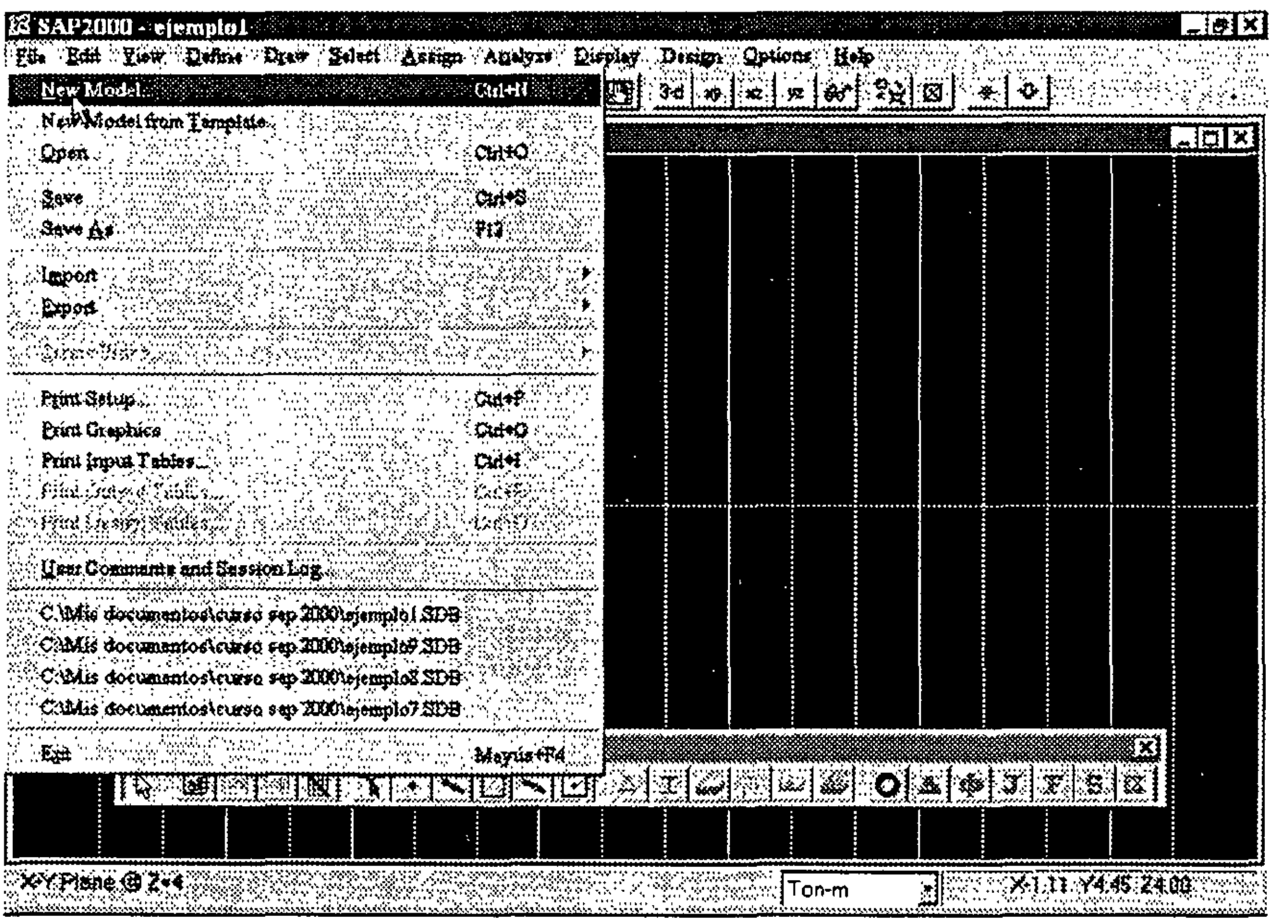


Figura 3.2 Módulos principales del menú File.

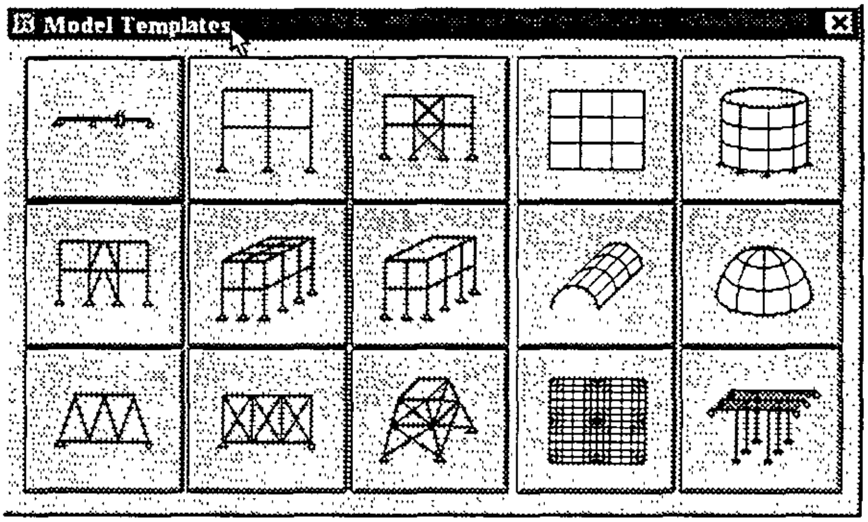


Figura 3.3. Geometrías predefinidas en la opción **New Model from Template**

Existen dentro de este menú otras opciones de uso no muy frecuente. Al iniciar **SAP 2000** s‹ recomienda seleccionar las unidades en las que se van a introducir los datos de la estructura a analizar, por ejemplo si estas fueron ton-m (toneladas y metros) los valores de las fuerzas uniformes se deben de proporcionar en ton/m, de las inercias en m^4, para el módulo elástico en ton/m^2, etc., es decir los valores deben ser consistentes.

3.3 El menú Edit

EL menú Edit (ver figura 3.4) permite desde introducir y hacer cambios a la geometría del modelo hasta suprimir algunos de sus elementos muchas de las opciones contenidas en este menú operan en conjunto con las del menú Select (ver siguiente sección), las opciones de este menú permiten:

Cut Suprimir los elementos seleccionados, guardándolos en la memoria temporal.

Copy Copiar sin borrar los elementos seleccionados a la memoria temporal

Paste Insertar los elementos contenidos en la memoria temporal especificando nuevas posiciones.

Delete Suprimir los elementos seleccionados.

Merge Joints Juntar los nudos que tengan una separación menor que un cierto valor (dejando uno solo y suprimiendo los demás es decir los nudos duplicados).

Move Mueve los nudos seleccionados especificando el incremento en sus coordenadas, moviendo también los elementos que estén conectados a esos nudos.

Replicate Realiza una copia (réplica) de los elementos seleccionados especificando el incremento en las coordenadas de sus nudos extremos.

Divide frames Divide a las barras seleccionadas en un número especificado por el usuario.

Join frames Junta varias barras seleccionadas en una sola (operación inversa de Divide frames).

Change Labels Cambia la numeración de los elementos seleccionados (renumera).

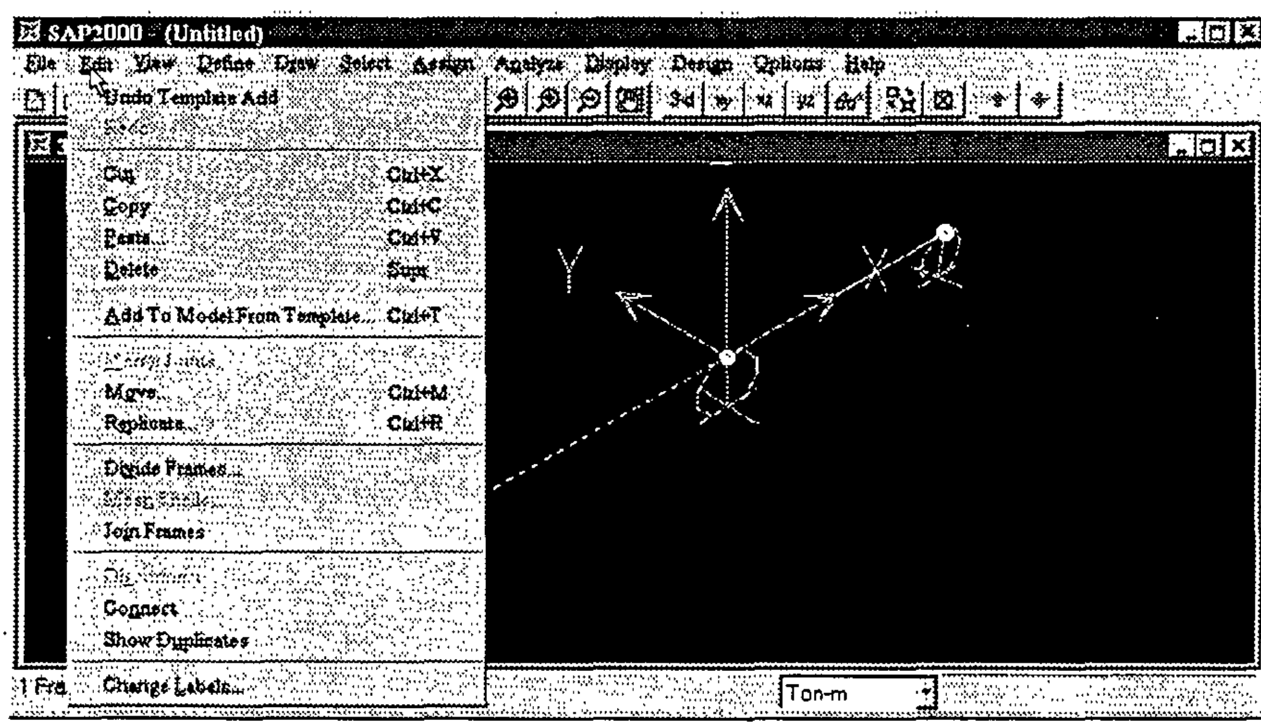


Figura 3.4. Opciones del menú **Edit**.

3.4 El menú View

EL menú View (ver figura 3.5) permite cambiar la presentación del área de dibujo de la estructura, algunas opciones que resultan de uso cotidiano son:

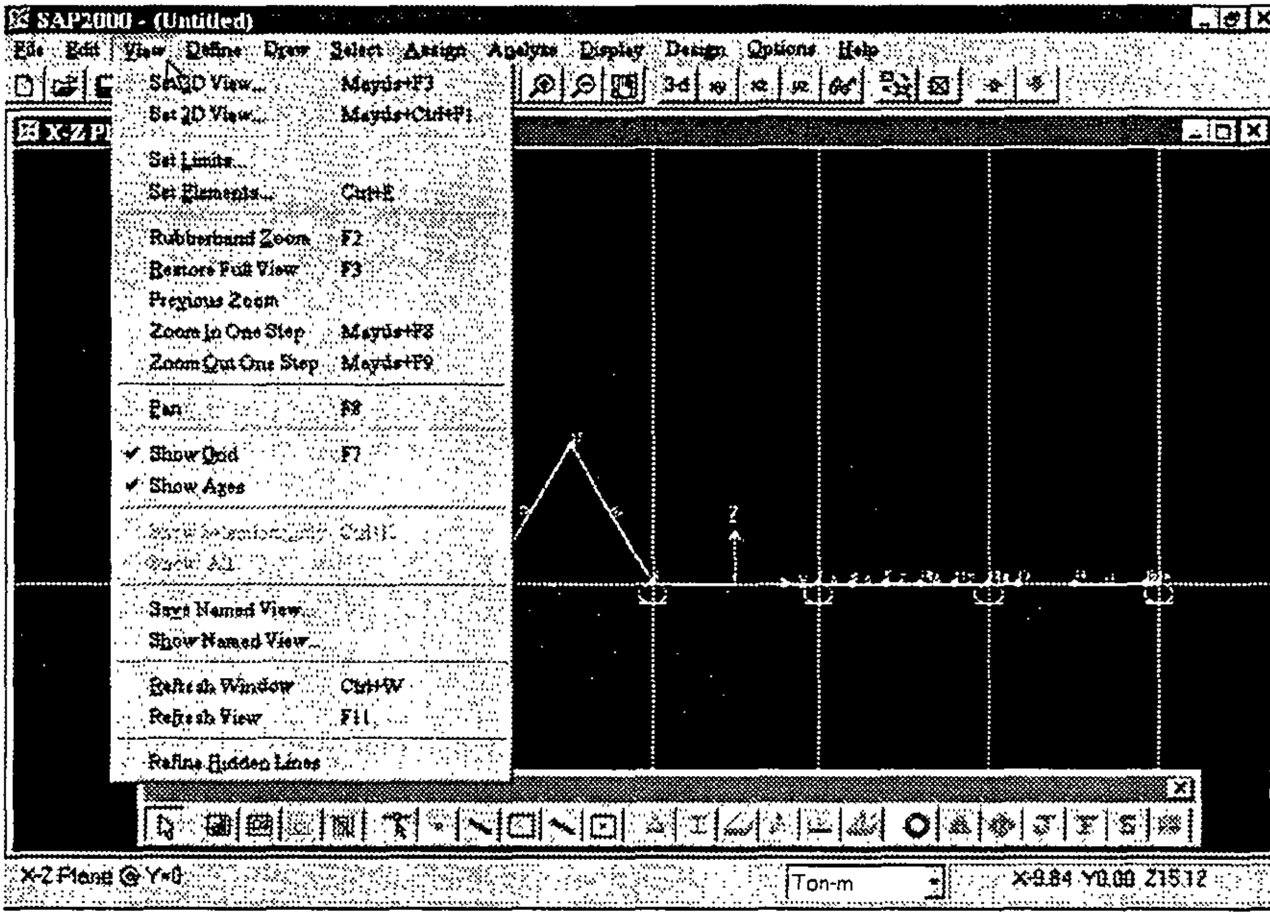


Figura 3.5. Opciones del menú View.

Set Elements Permite seleccionar la información a ser incluida dentro del área de dibujo (numeración de nudos, barras, ejes, etc.), ver figura 3.6.

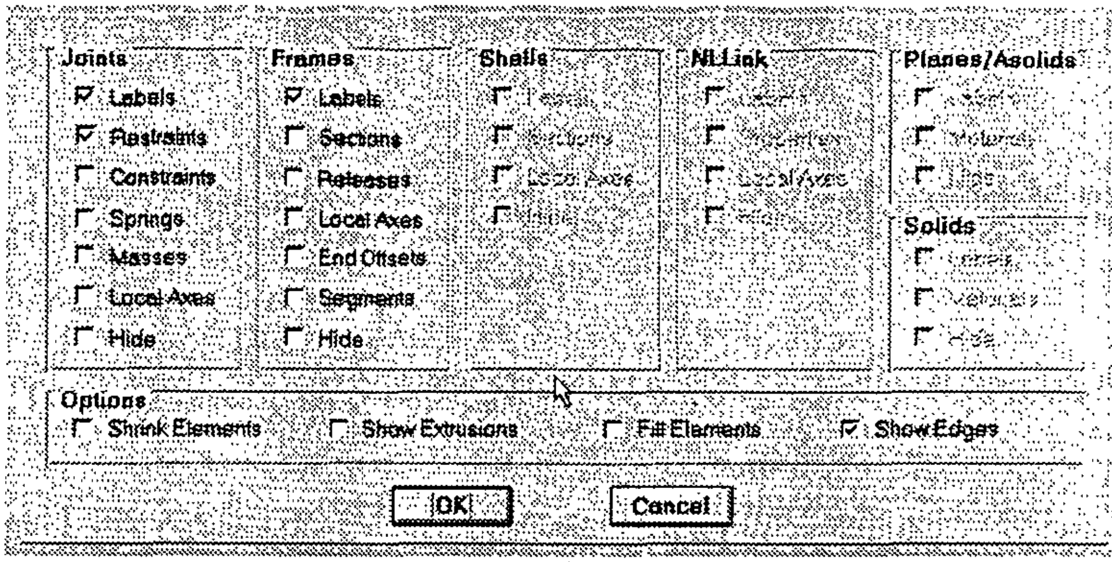


Figura 3.6. Opciones de **Set Elements**.

Show Grid Permite Activar (mostrar) o desactivar la malla auxiliar para dibujo de elementos.

Show Axes Dibuja o suprime los ejes globales de la estructura.

Se deja al lector que pruebe el efecto de las otras opciones, las características de algunas de ellas se verán posteriormente en el desarrollo paso a paso de algún ejemplo.

3.5 El menú Define

El menú Define (ver figura 3.7) permite especificar propiedades de los materiales (Materials....), características geométricas como forma, dimensiones, material, etc. para las barras del modelo (Frame Sections..) y algunas características para los elementos placa (Shell Sections...) También permite definir características generales de las condiciones de carga estática como su titulo o identificación, el tipo de carga (de acuerdo a su origen) y si se incluirá el peso propio en la condición de carga.

En este menú se podrá seleccionar o introducir un espectro de respuesta así como funciones de excitación para análisis dinámico, también se podrán definir las combinaciones de carga (Load Combinations...) seleccionando las condiciones de carga que se incluirán en cada combinación con sus respectivos factores de carga.

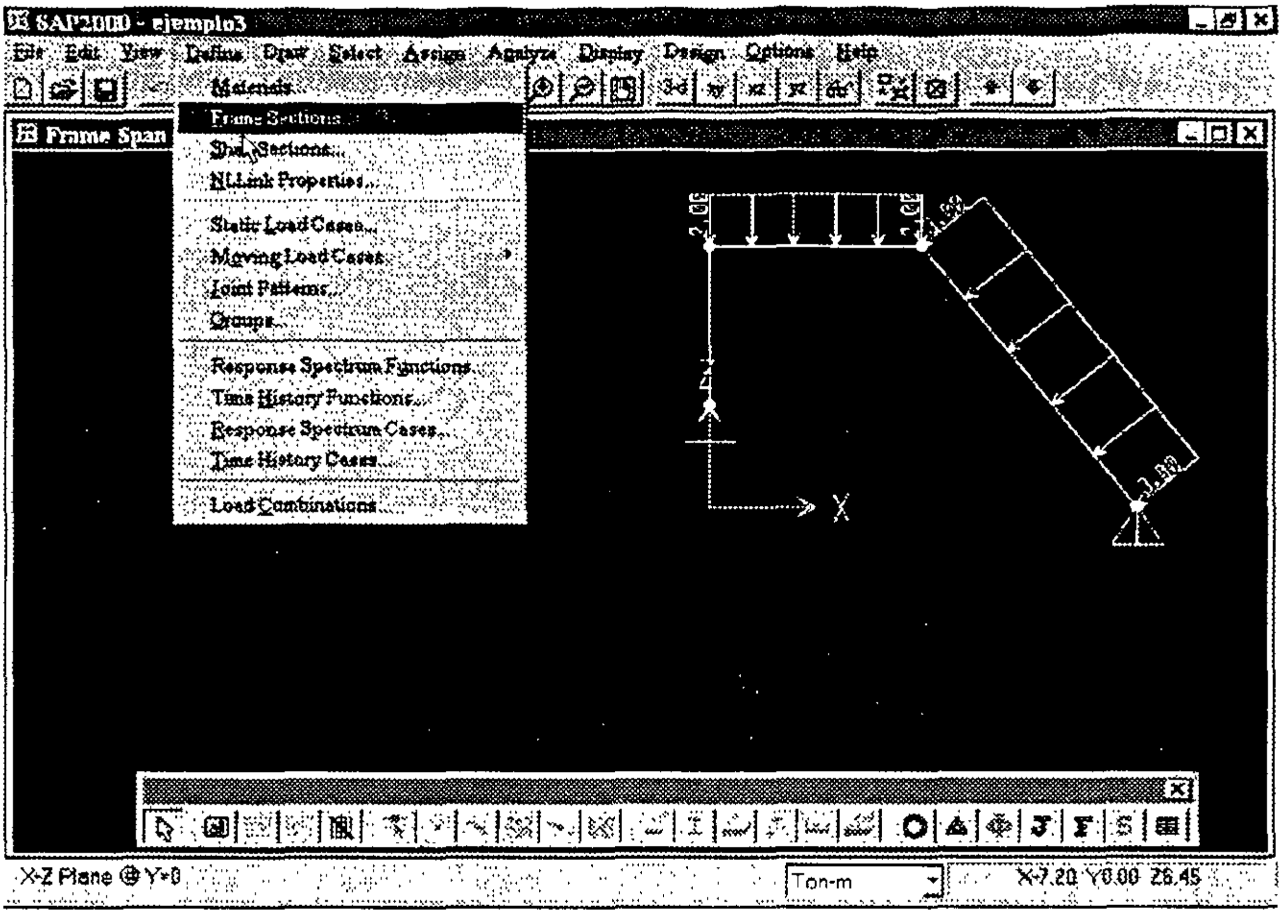


Figura 3.7. Opciones del menú Define.

3.5 El menú Draw

Algunas de las opciones del menú Draw (ver figura 3.8) permiten dibujar esquemáticamente a los elementos barra, placa, etc. con los que se irá construyendo el modelo estructural por analizar, algunas opciones de uso frecuente son:

Edit Grid — Permite adicionar, modificar, suprimir, etc las líneas que forman la malla auxiliar para dibujo de elementos.

Draw Frame Element — Permite iniciar el dibujo (con la ayuda del ratón) de elementos barra, después de seleccionar esta opción se hace clic izquierdo del ratón en el nudo donde inicia la barra (en caso de que este no halla sido creado se hace clic en sus coordenadas), luego se desplaza el puntero (sin arrastrar) hacia el nudo final de la barra haciendo clic izquierdo en el nudo con lo que queda especificada esa barra (se recomienda utilizar la malla auxiliar cambiando la separación de las líneas de la malla para que algunas de las intersecciones de esas líneas coincidan con la mayoría de los nudos de la estructura), la secuencia de dibujo de barras se puede interrumpir con un doble clic del botón derecho en cualquier parte del área de dibujo (con lo que es posible dibujar barras en otras posiciones), para terminar el dibujo de barras se hace clic en el icono de puntero de la barra flotante de iconos, posteriormente se puede dibujar más barras volviendo a seleccionar esta opción, lo anterior se puede hacer tantas veces como se requiera.

Draw Shell Element Permite iniciar el dibujo (con la ayuda del ratón) de elementos placa, funciona de manera muy similar a la opción anterior solo que en este caso se seleccionaran tres o cuatro nudos dependiendo del tipo de elemento finito que se quiera dibujar, la selección de nudos se hará en sentido horario o antihorario.

Quick Draw Frame Element y Quick Draw Shell Element permiten el dibujo de barras y placas respectivamente con un solo clic izquierdo cerca de alguna de las líneas de la malla auxiliar (para el caso de barras) y en algún punto dentro de un área delimitada por líneas de la malla auxiliar de dibujo (para el dibujo de placas), se deja al lector la práctica con estas opciones antes de abordar los ejemplos que se presentan en el capítulo correspondiente.

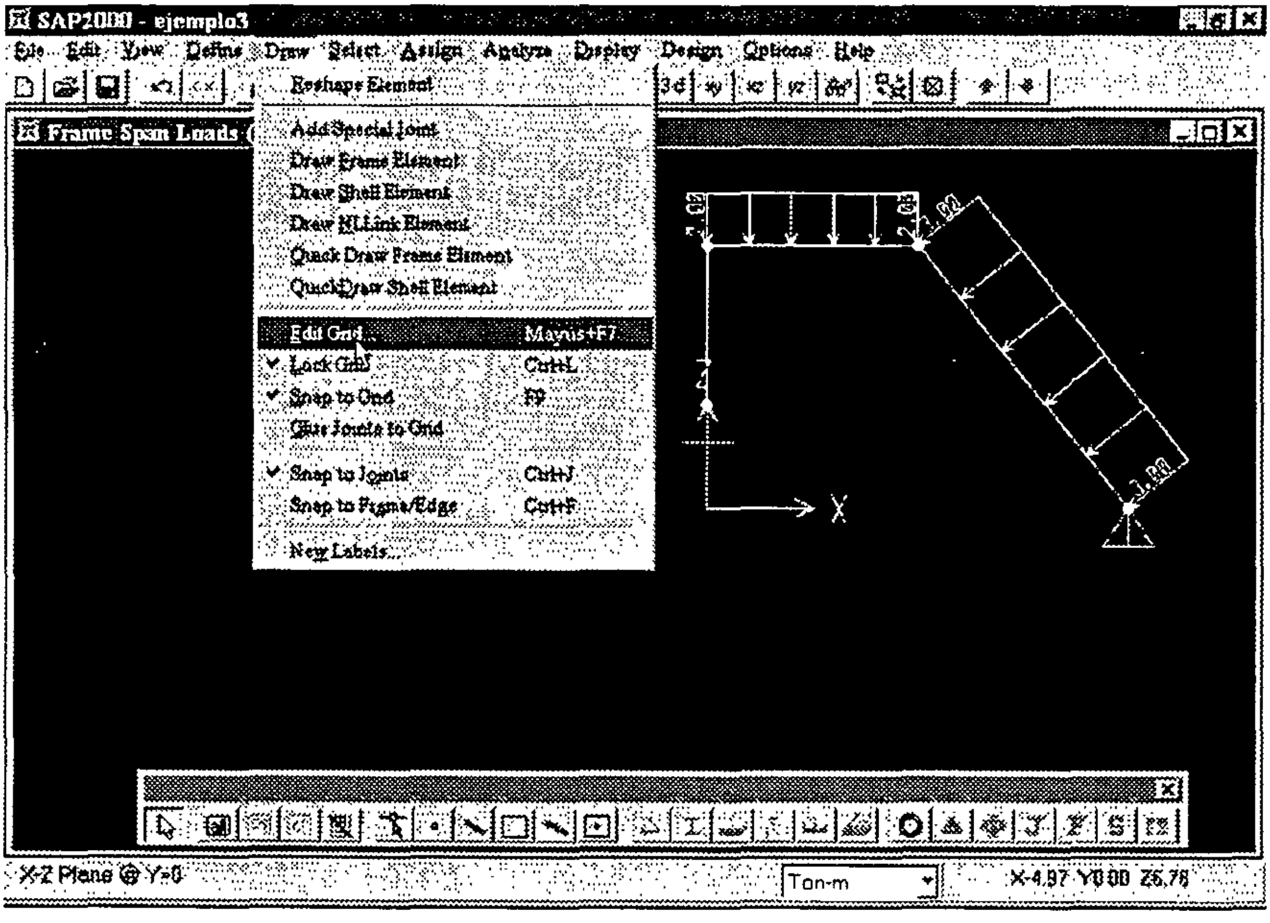


Figura 3.8. Opciones del menú **Draw.**

3.6 El menú Select

Algunas de las opciones del menú Select (ver figura 3.9) nos permitirán seleccionar elementos ya existentes dentro del modelo, la selección de elementos es necesaria para poder asignar (ver menú Assign) algunas características a los mismos, por ejemplo si se seleccionan barras se les podrá asignar secciones, cargas, etc. las siguientes son algunas opciones que resultan de uso frecuente:

Pointer/Window Permite seleccionar a los elementos que quedan contenidos dentro de un área rectangular que se define haciendo clic izquierdo en una de las esquinas del área y arrastrando el puntero del ratón hasta la esquina opuesta y soltando el botón del ratón en esa esquina, los elementos seleccionados cambian su aspecto de línea continua a línea interrumpida (punteada).

Intersecting Line Con esta opción se seleccionan a aquellos elementos que son intersectados por una línea que se define haciendo clic izquierdo en uno de los extremos de la misma y arrastrando el puntero del ratón hasta el otro extremo de la línea y soltándolo ahí mismo.

Las otras opciones de Select permiten seleccionar elementos que tienen alguna característica en común.

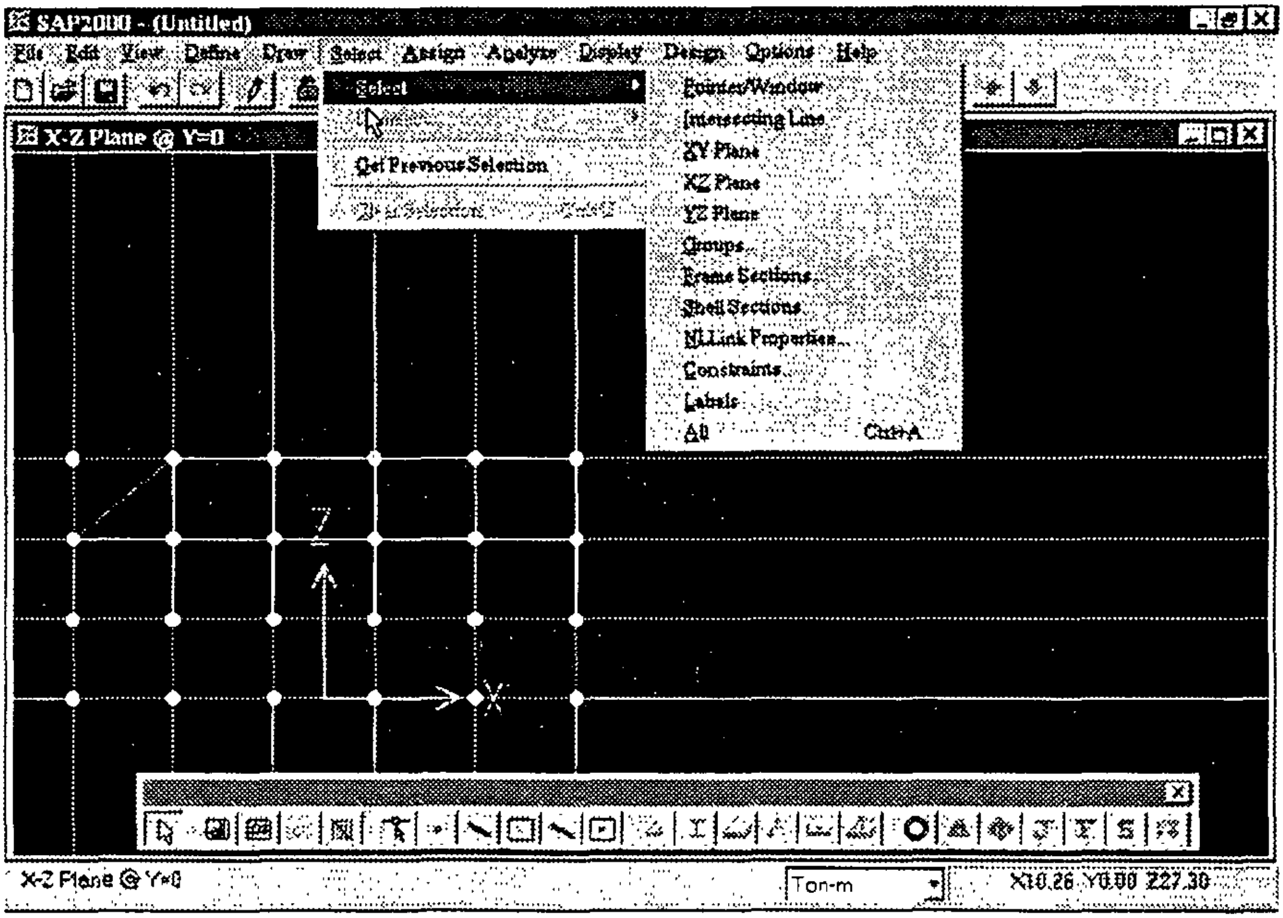


Figura 3.9. Algunas opciones del menú **Select.**

El menú Select dispone de las mismas opciones para excluir elementos ya seleccionados, lo anterior se realiza con la opción Unselect, otra manera de quitar elementos de la selección es haciendo clic en el icono de flecha de la barra flotante de iconos y luego hacer clic en cada uno de los elementos que han sido previamente seleccionados y que se quieren excluir, inclusive si se hace clic en un elemento no seleccionado este se selecciona y viceversa

3.7 El menú Assign

Una vez seleccionados algunos elementos (nudos, barras, etc.) podemos asignarles alguna característica propia del elemento (restricciones, fuerzas, secciones, etc.), el menú Assign (ver figura 3.10) junto con sus opciones nos permitirán realizar esa actividad, enseguida una breve descripción de algunas opciones del menú assign.

Joint Permite asignar a los nudos seleccionados restricciones o apoyos (restraints), asignar el mismo desplazamiento (constraints), asignar resortes (springs), etc.

Joint Static Loads Con esta opción se asignan a los nudos seleccionados fuerzas (Forces) o desplazamientos prescritos (Displacements).

Frame Permite asignar a las barras seleccionadas propiedades (Sections), liberarlas de algún elemento mecánico (Releases), especificar sus ejes locales (Local Axes), etc.

Frame Static Loads Con esta opción se asignan fuerzas estáticas de gravedad (Gravity), puntuales y/o uniformes (Point and Uniform), con variación lineal (Trapezoidal), efectos de temperatura (Temperature), y efectos de presfuerzo (Prestress).

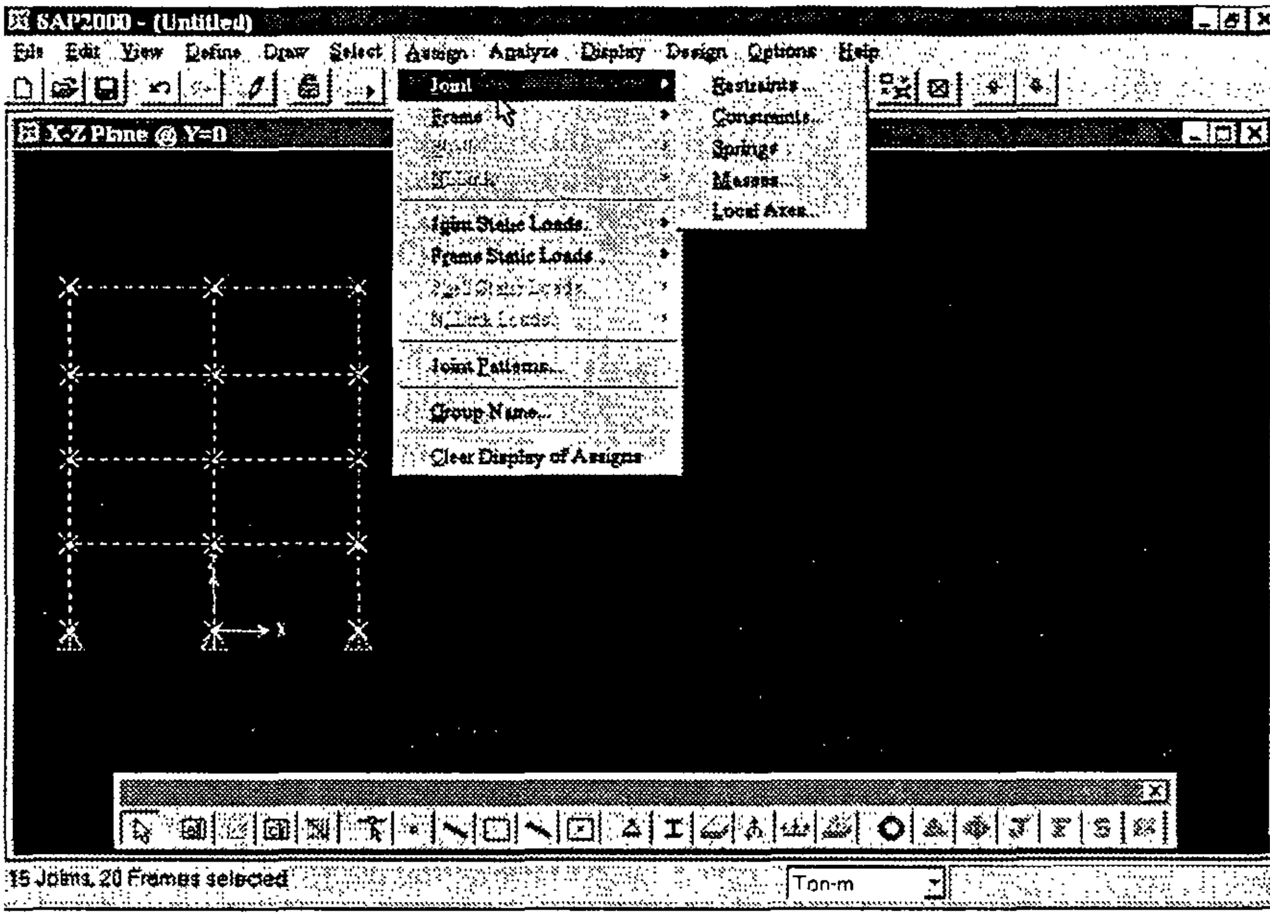


Figura 3.10. Algunas opciones del menú **Assign.**

3.8 El menú Analyze.

El menú **Analyze** (ver figura 3.11) permite seleccionar algunas opciones de análisis (**Set Options...**), o bien se puede solicitar que el programa **SAP 2000** realice el análisis (**Run**) con los resultados desplegados en una ventana normal o bien en una ventana minimizada (**Run Minimized**), se recomienda guardar el archivo de trabajo antes de solicitar el análisis (inclusive guardarlo en disco flexible y luego en el disco duro).

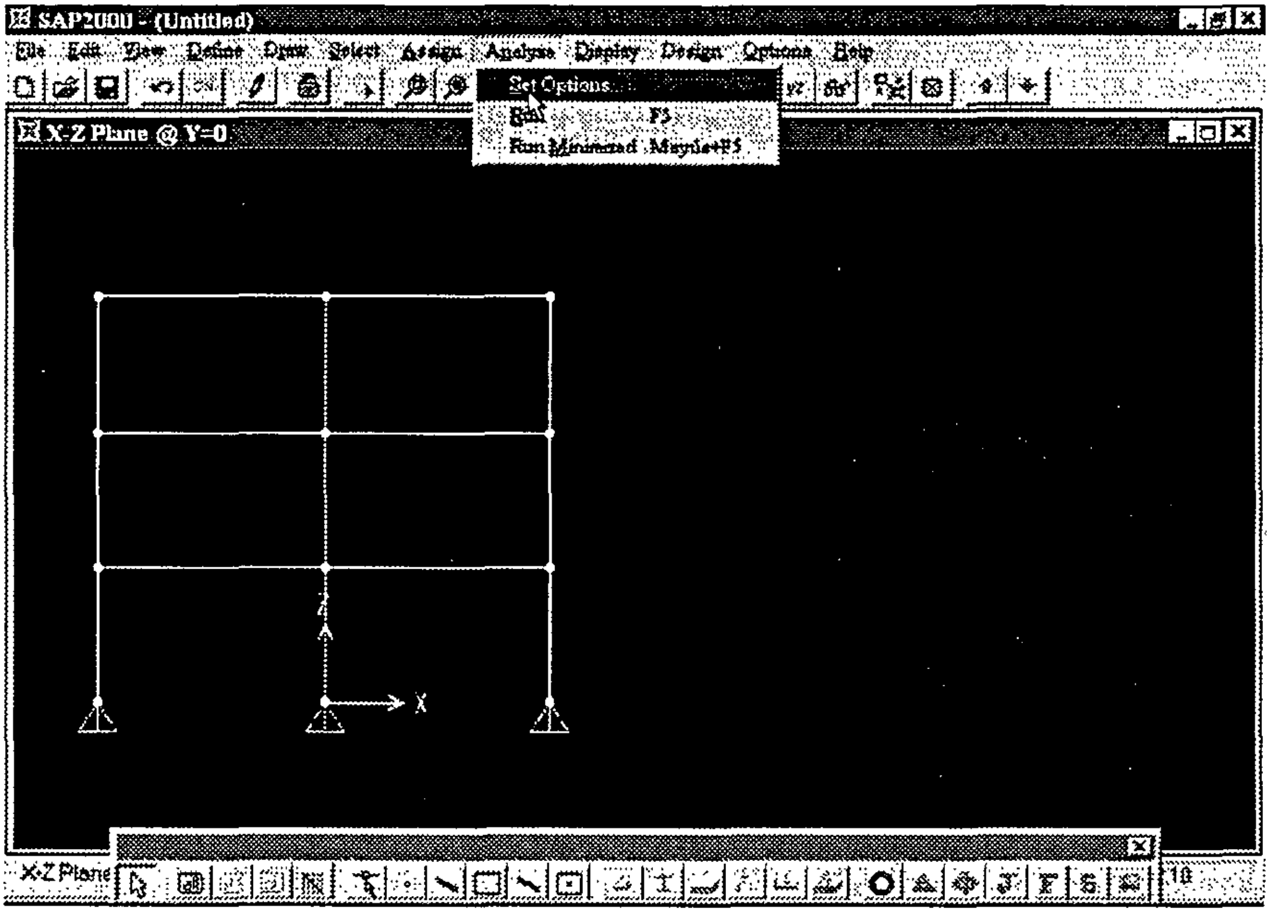


Figura 3.11 Opciones en el menú **Analyze**.

Las opciones de **Set Options...** (ver figura 3.12) permiten seleccionar los grados de libertad activos (**Available DOFs**) dependiendo del tipo de estructura que se analizará, será necesario identificar y seleccionar haciendo clic en los cuadros respectivos del área correspondiente (un cuadro en blanco significa que ese grado de libertad no esta activo), otra manera de seleccionar los grados de libertad es utilizando la opción de seleccionado rápido (**Fast DOFs**), lo anterior se realiza haciendo clic en alguna de las figuras que corresponda a nuestra estructura, la selección inadecuada de los grados de libertad puede generar resultados incorrectos o estructura inestable (división entre cero) durante la fase de análisis.

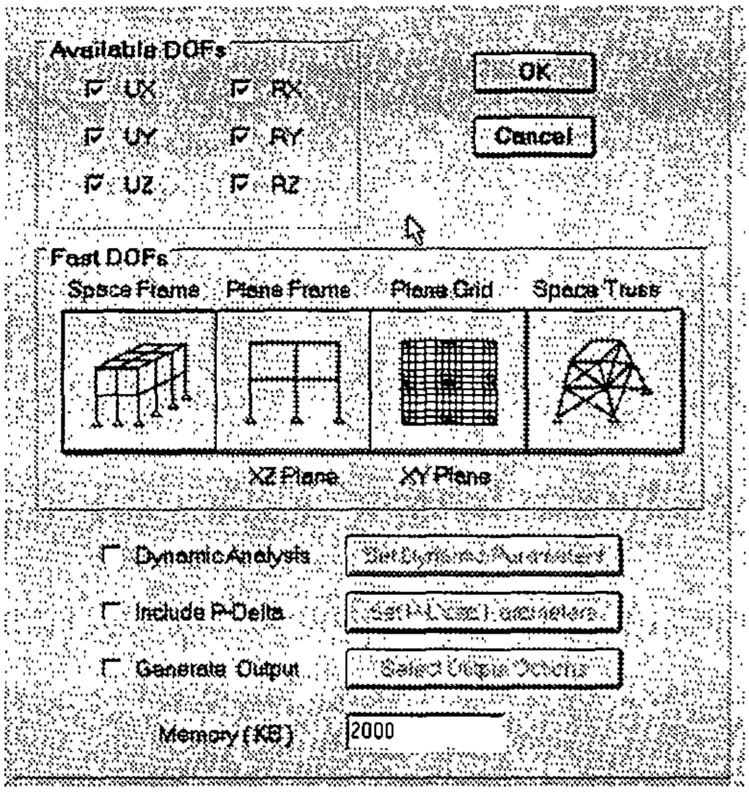


Figura 3.12. Opciones de **Set Options...** del menú **Analyze**.

Al final de la caja de selección se puede indicar que se realice un Análisis Dinámico (**Dynamic Analysis**), que se incluyan efectos P-Δ (**Include P-Delta**) y que se generen archivos de salida (**Generate Output**), para estas ultimas opciones es conveniente indicar algunos parámetros y seleccionar algunas opciones especificas.

Cuando se selecciona la opción de Análisis (**Run**), y algunos resultados del proceso se van desplegando en la pantalla (ventana) quedando al final algo similar a lo que se muestra en la figura 3.13.

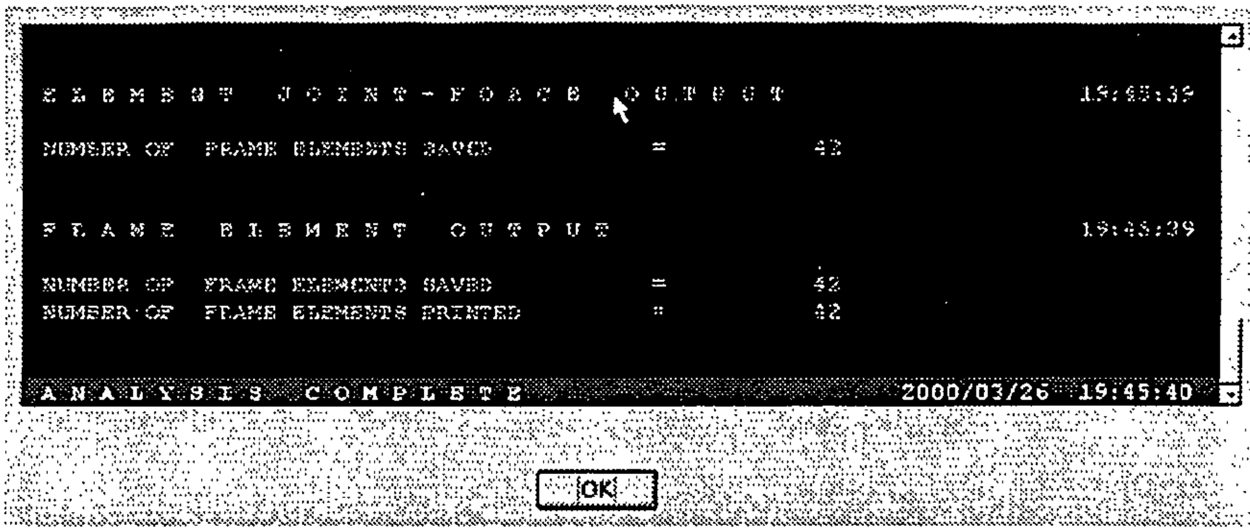


Figura 3.13. Ventana al finalizar el análisis.

Al hacer clic en el botón OK de la ventana que se muestra al final del análisis, se despliega en el área de dibujo la configuración deformada de la estructura para determinada condición de carga, en esta parte del programa se podrán seleccionar los resultados del análisis por ejemplo desplazamientos de los nudos, reacciones, elementos mecánicos, diagramas de elementos mecánicos, configuraciones deformadas, etc.

3.9 El menú Display

Este menú permite solicitarle al programa que muestre la geometría no deformada del modelo (**Show Undeformed Shape**), las cargas en los nudos (**Show Loads**), en las barras, en los elementos placa o no mostrarlas.

Mediante la opción **Show Input Tables** (ver figura 3.14) se solicita al programa que muestre en una ventana conteniendo una lista con los datos numéricos de la geometría en lo que respecta a nudos (coordenadas, restricciones, etc.), barras (incidencias, tipo de sección, etc.) y cargas (en los nudos, en las barras y en las placas), produciendo una salida parecida a la de la figura 3.15, la tabla mostrada puede imprimirse o grabarse en un archivo.

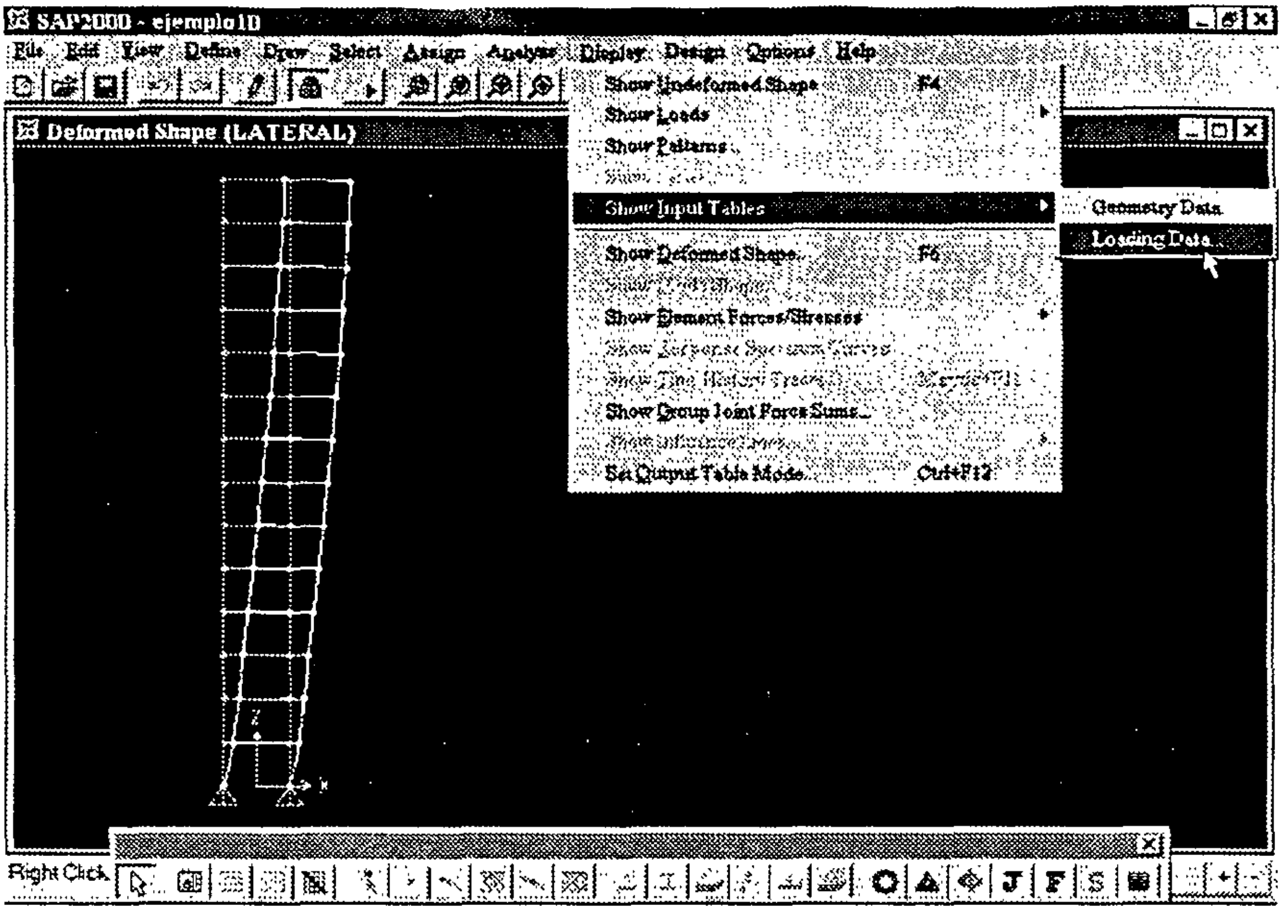


Figura 3.14. Opciones del menú **Display**.

JOINT	GLOBAL-X	GLOBAL-Y	GLOBAL-Z	RESTRAINTS	ANGLE-A
1	-3.00000	0.00000	0.00000	1 1 1 1 0 1	0.000
2	-3.00000	0.00000	4.00000	0 0 0 0 0 0	0.000
3	-3.00000	0.00000	8.00000	0 0 0 0 0 0	0.000
4	-3.00000	0.00000	12.00000	0 0 0 0 0 0	0.000
5	-3.00000	0.00000	16.00000	0 0 0 0 0 0	0.000
6	-3.00000	0.00000	20.00000	0 0 0 0 0 0	0.000
7	-3.00000	0.00000	24.00000	0 0 0 0 0 0	0.000
8	-3.00000	0.00000	28.00000	0 0 0 0 0 0	0.000
9	-3.00000	0.00000	32.00000	0 0 0 0 0 0	0.000
10	-3.00000	0.00000	36.00000	0 0 0 0 0 0	0.000

Figura 3.15. Salida típica a partir de **Show Input tables** del menú **Display**.

Mediante la opción **Show Deformed Shape** y después de seleccionar la condición de carga, SAP 2000 muestra la configuración deformada correspondiente (ver figura 3.16).

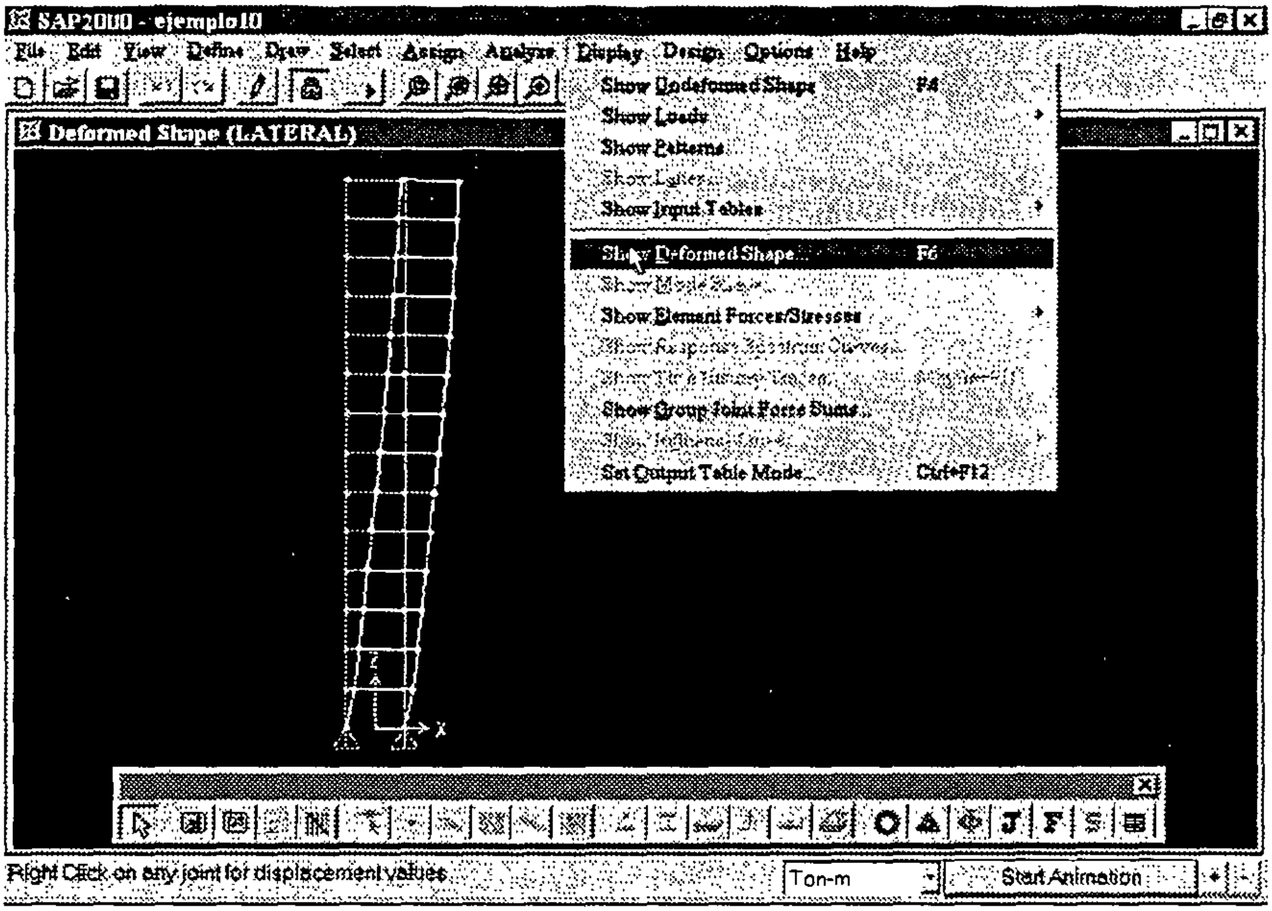


Figura 3.16. Salida típica a partir de **Show Deformed Shape** del menú **Display**

La opción **Show Element Forces/Stresses** y dependiendo de la selección que se haga **SAP 2000** puede mostrar elementos mecánicos, esfuerzos, reacciones, etc. produciendo una salida similar a la que se muestra en la figura 3.17.

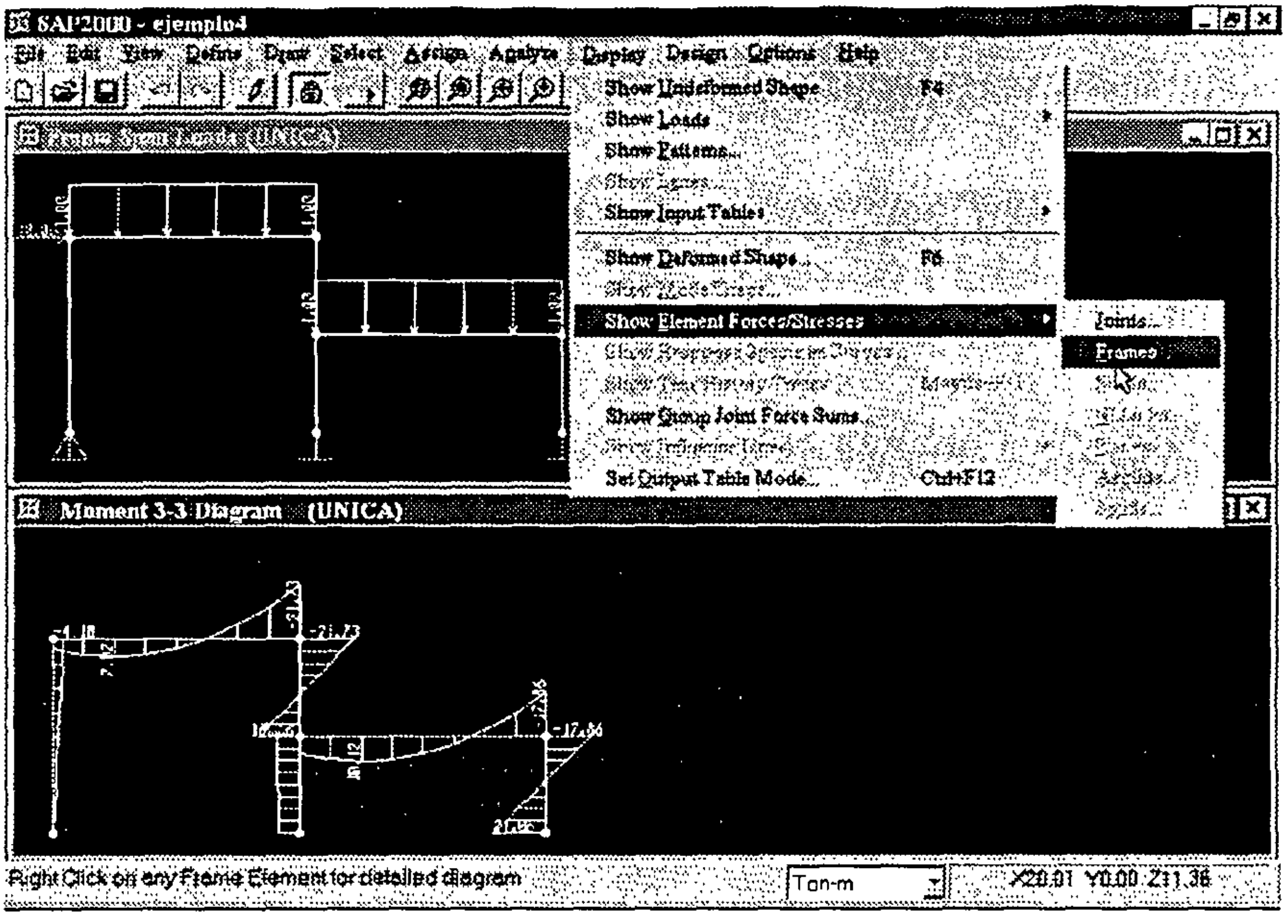


Figura 3.17. Salida obtenida con **Show Element Forces/Stresses** del menú **Display**

3.10 El menú Design

EL menú **Design** (ver figura 3.18) permite seleccionar algunas opciones de diseño, realizar el diseño (verificación) de elementos con la posibilidad de optimizar secciones, con la característica de producir salidas similares a las mostradas en las figuras 3.19 y 3.20 de entre otras.

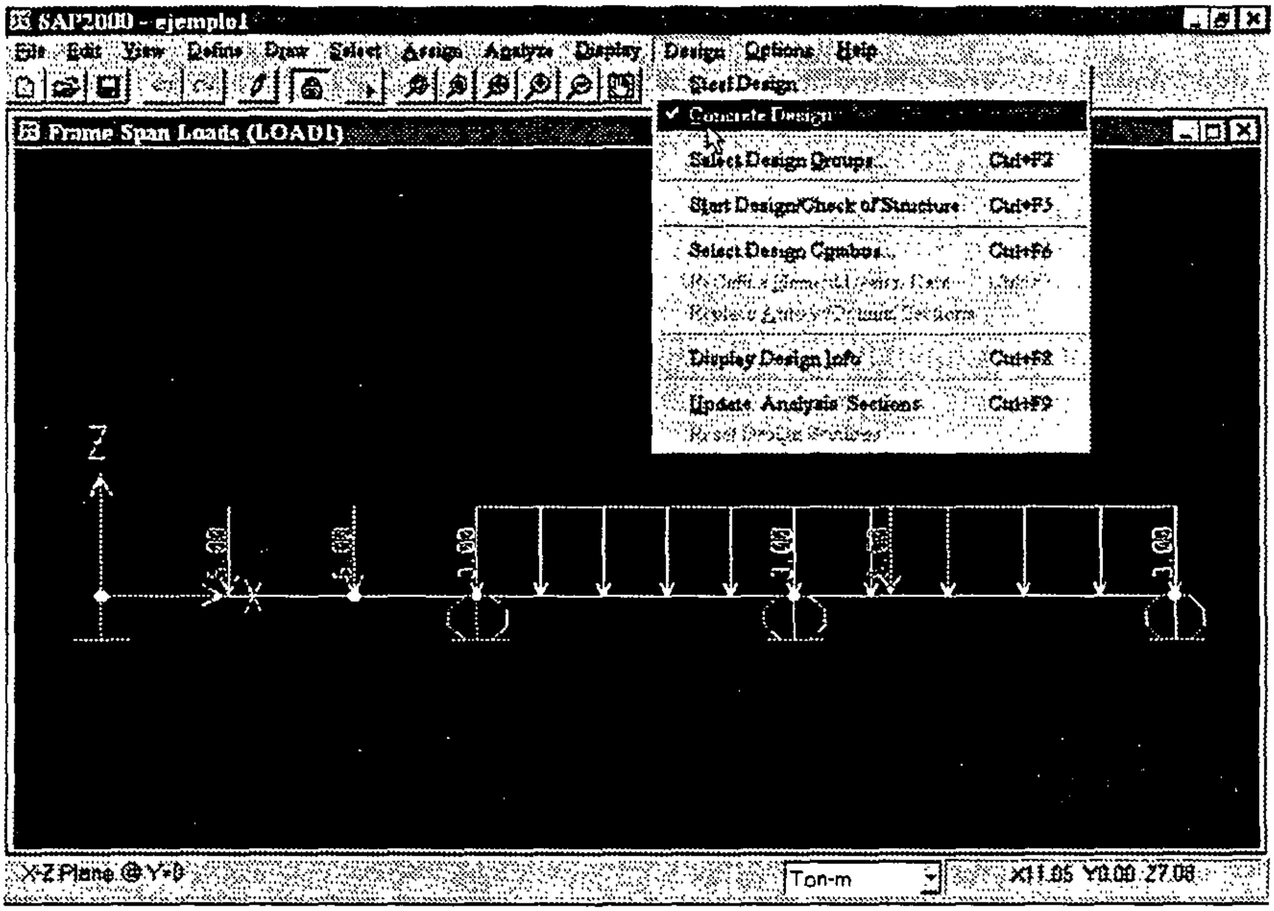


Figura 3.18. Opciones del menú **Design**.

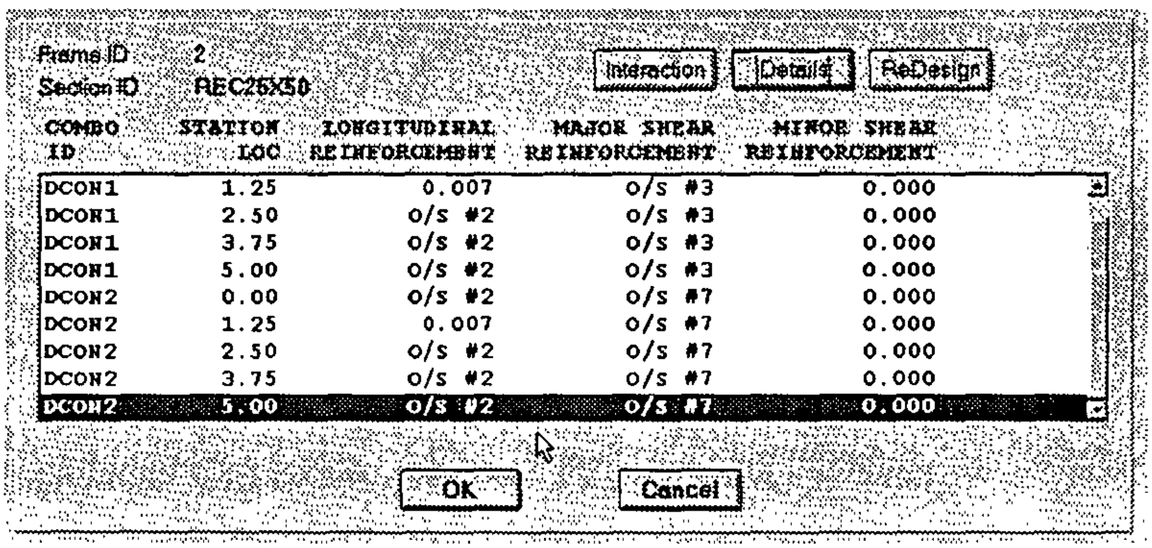


Figura 3.19. Algunos resultados del menú **Design**.

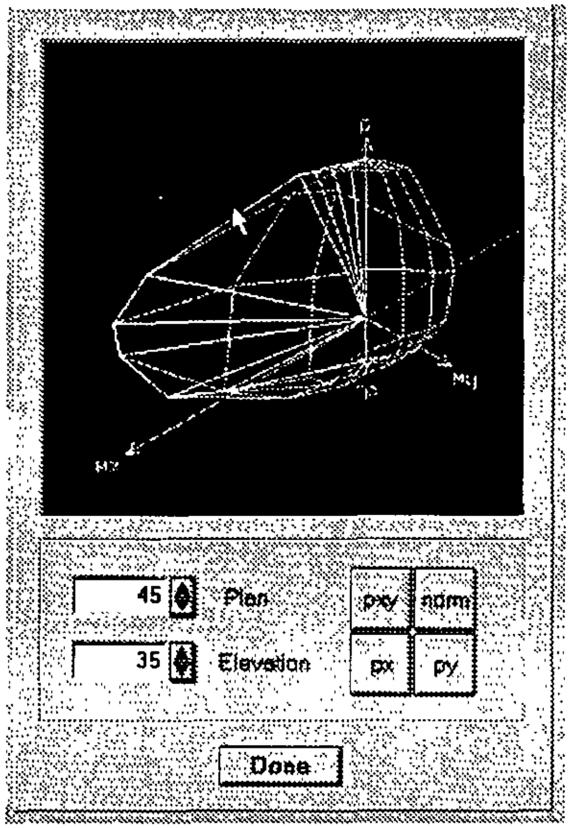


Figura 3.20. Algunos resultados del menú **Design**.

```
COLUMN SECTION DESIGN  Type: Sway Special  Units: Ton-m

Frame   ID         2
Station Loc     5.00
Section ID  REC25X50
Combo   ID     DCON2

L=5.000
B=0.250  D=0.500  dc=0.050
E=2200000.000  fy=60.000  fc=4.000  fcs=4.000

AXIAL FORCE & BIAXIAL MOMENT DESIGN FOR PU, M2, M3
                 Rebar      Design      Design      Design     Minimum
                  Area          Pu          M2          M3          M2
                O/S #2       0.000       0.000      20.793       0.000

AXIAL FORCE & BIAXIAL MOMENT FACTORS
                                Cm    Delta_ns     Delta_s           K
                            Factor      Factor      Factor      Factor
   Major Bending(M3)         1.000       1.000       1.000       1.000
   Minor Bending(M2)         1.000       1.000       1.000       1.000
```

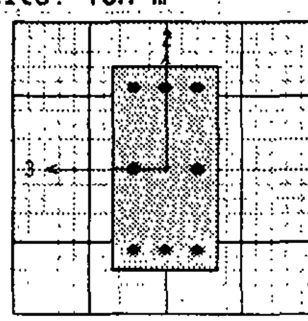


Figura 3.21. Algunos resultados del menú **Design**.

3.11 Los menús Options y Help

El menú **Options** (ver figura 3.22) permite por así decirlo controlar el tipo y características de la información que será mostrada en las diferentes áreas de presentación (colores, número de ventanas, etc.).

En este punto podemos mencionar que una vez que se realiza el análisis **SAP 2000** "bloquea" al modelo no permitiendo realizarle ninguna modificación por lo que solo es posible manejar los resultados (ver valores numéricos, gráficas, imprimirlos, etc.), para desbloquear al modelo y poder hacerle cambios se selecciona la opción **Lock Model** con esto ahora los resultados ya no están disponibles para poder tener acceso a ellos una vez realizados los cambios será necesario solicitar nuevamente la realización del análisis.

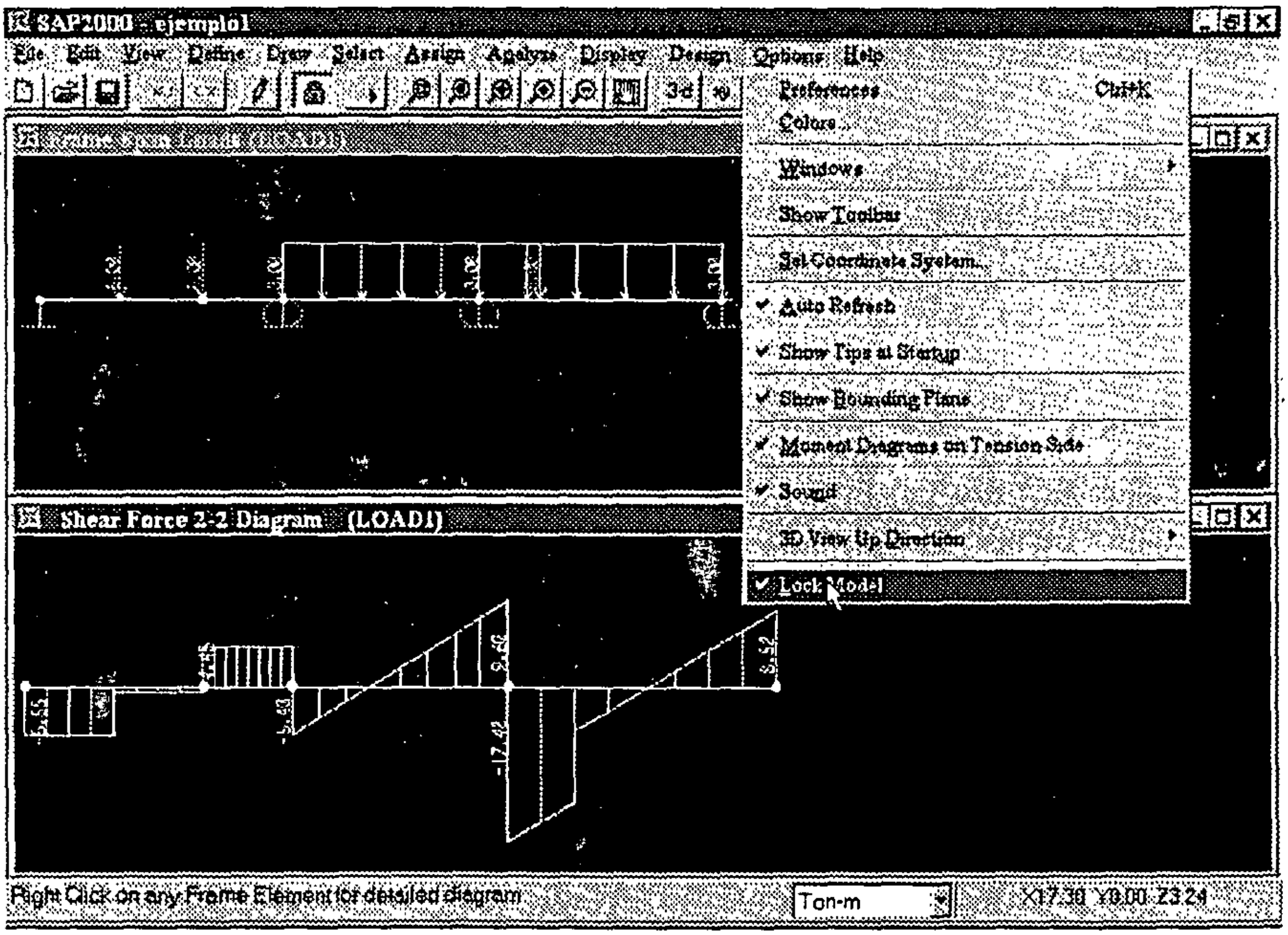


Figura 3.21. Opciones en el menú **Options** y desbloqueo del modelo.

Se deja al lector que pruebe el efecto de las otras opciones del menú **Options** así como las del menú **Help**, las características de algunas de ellas se verán posteriormente en el desarrollo de algunos ejemplos.

GENERACION DE LA ESTRUCTURA

CAPÍTULO 4

4.1 INTRODUCCIÓN

En **SAP 2000** la generación de la estructura se entiende como la ubicación con respecto a un sistema de coordenadas (global) de los elementos barra, placa y sólido, la asignación de propiedades geométricas y elásticas a los elementos ya localizados, la introducción de apoyos, la definición y asignación de fuerzas a los nudos, barras y placas, la selección del tipo de análisis y resultados, por último, el dimensionamiento o revisión de elementos.

La forma de iniciar el programa **SAP 2000** ha sido descrita con anterioridad (ver inciso 3.2 del capitulo anterior), enseguida se recomienda elegir las unidades en que se introducirán los datos haciendo clic en la pestaña que se encuentra a la derecha del cuadro de unidades y seleccionándolas de la caja que muestra el programa (ver figura 4.1).

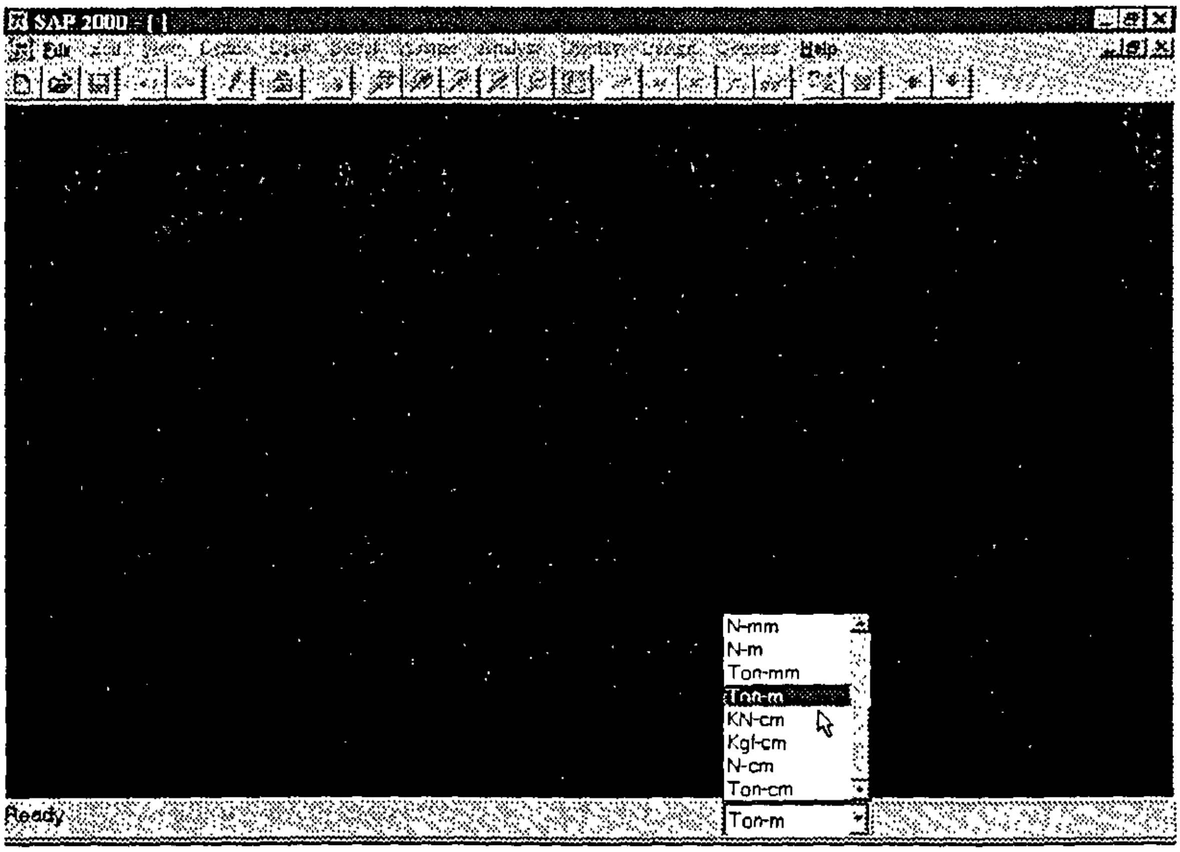


Figura 4.1 Selección de unidades.

SAP 2000 ofrece varias alternativas para introducir la topología de la estructura, aquí empezaremos por una de las más comunes que es introduciendo elemento por elemento, para ello se recomienda auxiliarnos de la malla (**grid**) que el programa nos proporciona por lo que se tendrá que ajustar la separación de las líneas que forman esa malla, seleccionemos **New Model** del menú **File**, enseguida el programa mostrará un cuadro en donde se especificarán las características de la malla como el número de espacios en cada dirección así como su separación los cuales se pueden modificar introduciendo valores particulares en los cuadros en fondo blanco haciendo clic en el que se quiera modificar (ver figura 4.2)

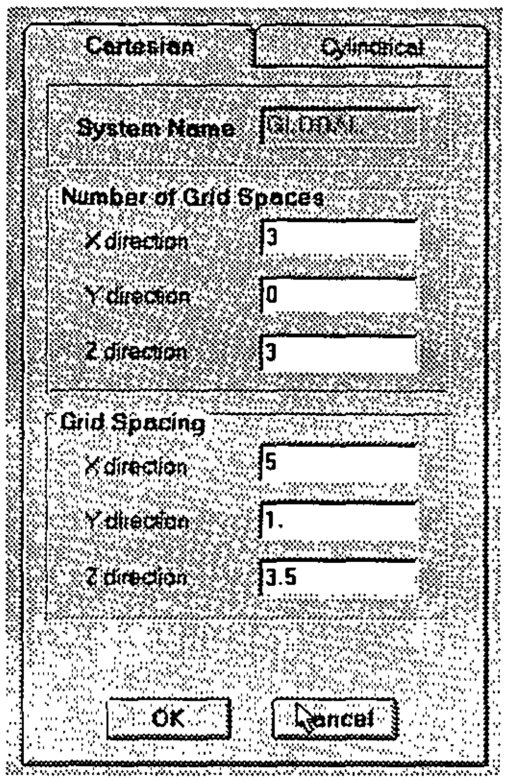


Figura 4.2 Ventana para definir las características de la malla auxiliar.

Una vez que se ha hecho clic en el botón OK el programa muestra la malla resultante en el área de dibujo (con fondo negro) dividiéndola en 2 cuadros mostrando en ellos una vista diferente de la malla (3 D y en el plano X-Y en Z=10.5), también puede observarse los ejes coordenados globales (ver figura 4.3).

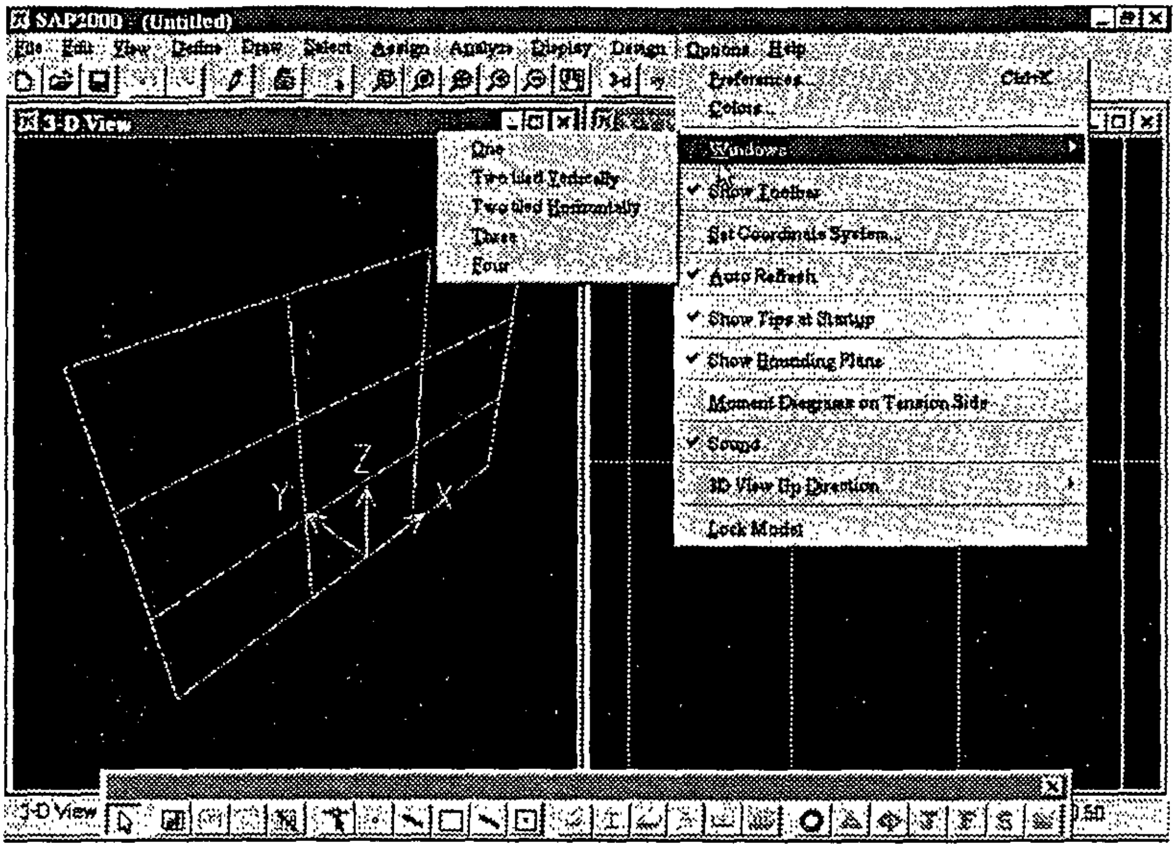


Figura 4.3 Imagen típica después de definir las características de la malla auxiliar.

Cada una de esas ventanas tiene en su extremo superior derecho los botones de minimizar, ventana completa y cerrar, el número y tipo de ventanas a mostrarse en la pantalla puede seleccionarse a través de la opción Windows del menú **Options**.

De las ventanas que se muestran, la ventana activa o en la que se muestran los resultados de los comandos que se elijan es aquella cuya barra de título está en color (generalmente diferente del gris), se activa una ventana haciendo clic en su interior.

La malla así creada tiene separación constante entre las líneas de una misma dirección, existen varias maneras de cambiar la separación entre cada línea de la malla, una de ellas es, después de seleccionar una vista en planta hacer dos clics seguidos en una de las líneas de la malla (con el botón izquierdo del ratón), enseguida se mostrará una ventana conteniendo información acerca de la posición de esas líneas con la opción de seleccionar la dirección de las líneas de la malla así como adicionar, mover y borrar líneas.

Haciendo clic en el cuadro en blanco e introduciendo el valor de la nueva posición de la línea y después de hacer clic en la opción **Add grid line** se ha introducido una nueva línea a la malla. Para modificar el valor de una línea se selecciona de la caja en gris haciendo clic izquierdo en la línea a modificar con lo cual se muestra en la caja en blanco y haciendo clic en esa caja se puede cambiar su valor, para que el cambio resulte efectivo después de modificar el valor de la caja en blanco se necesita hacer clic en el botón **Move**, las demás opciones complementan la modificación de la malla (ver figura 4.4). Desde luego que para que todos los cambios produzcan efecto es necesario hacer clic en el botón OK.

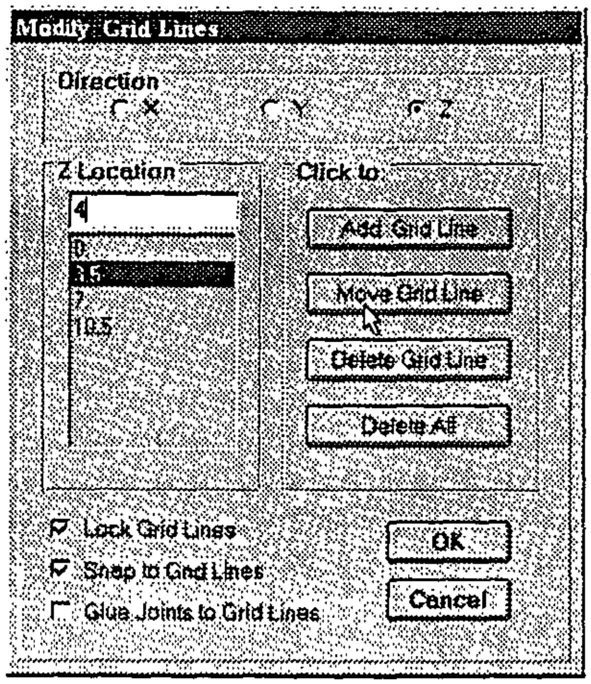


Figura 4.4 Modificación de la separación individual entre las líneas de la malla.

Otra manera para que se muestre el cuadro de la figura 4.4 es seleccionando **Edit Grid** del menú **Draw**, otro comando que resulta útil es la opción **Show Grid** del menú **View** con el cual se suprime o activa la aparición de la malla en área de dibujo.

Los datos de la estructura que se vayan introduciendo son almacenados en memoria volátil (RAM) por tal motivo se recomienda que con cierta frecuencia se graben en el disco duro (o en disco flexible), para ello se puede utilizar la opción **Save** o **Save As** del menú **File**, el programa asignará al nombre del archivo proporcionado por el usuario la extensión .SDB.

Ahora podemos introducir todos los elementos que componen a la estructura, a continuación se presenta una breve descripción lógica de las opciones de uso común así como de los comandos que nos permitirán la generación de la estructura en el orden mencionado al inicio de este capítulo, varios de los comandos fueron descritos en el capítulo anterior.

4.2 DESCRIPCIÓN GENERAL

La parte del proceso de modelación que consume más recursos (tiempo y esfuerzo) es la que concierne a la introducción de elementos (barra, placa, etc.), es por ello que el uso eficiente de los comandos del menú Draw y en combinación con algunos otros nos permitirá la generación de la topología (forma) de la estructura lo más pronto posible, como recomendaciones generales, se pueden mencionar las que se indican en los párrafos siguientes.

Procurar iniciar la geometría de la estructura a partir de alguna de las predefinidas que trae la librería del programa (vigas, marcos, etc), enseguida realizar los cambios necesarios para ajustar esa geometría a la del modelo por analizar (adicionando o borrando algunos elementos, cambiándolos de posición, copiándolos, etc.)

Para la definición de elementos (barra, placa, etc.) auxiliarse de la malla (**grid**) cambiando la separación de las líneas de la malla para que sus intersecciones definan la mayor cantidad de coordenadas de los nudos de nuestro modelo procurando que con la nueva separación de las líneas de la malla los elementos resultantes tengan las características (dimensiones e inclinación) deseadas con lo que el usos de las opciones de dibujo rápido de elementos (con un solo clic, en lugar de dos clics) traerá algún ahorro y facilidad de creación o modificación del modelo.

Las características a ser mostradas en la pantalla (numeración, ejes locales, etc.) de los elementos que se van adicionando al modelo (nudos, barras, placas, etc.) pueden ser controladas mediante la opción **Set Elements** del menú **View** (ver figura 4.5). La información mostrada puede ser de utilidad, también es conveniente recordar que las características de algún elemento (nudo, barra, placa, etc.) pueden desplegarse seleccionándolo (clic izquierdo) y luego haciendo clic derecho, algunos de los elementos en la caja mostrada pueden ser modificados (ver figura 4.6).

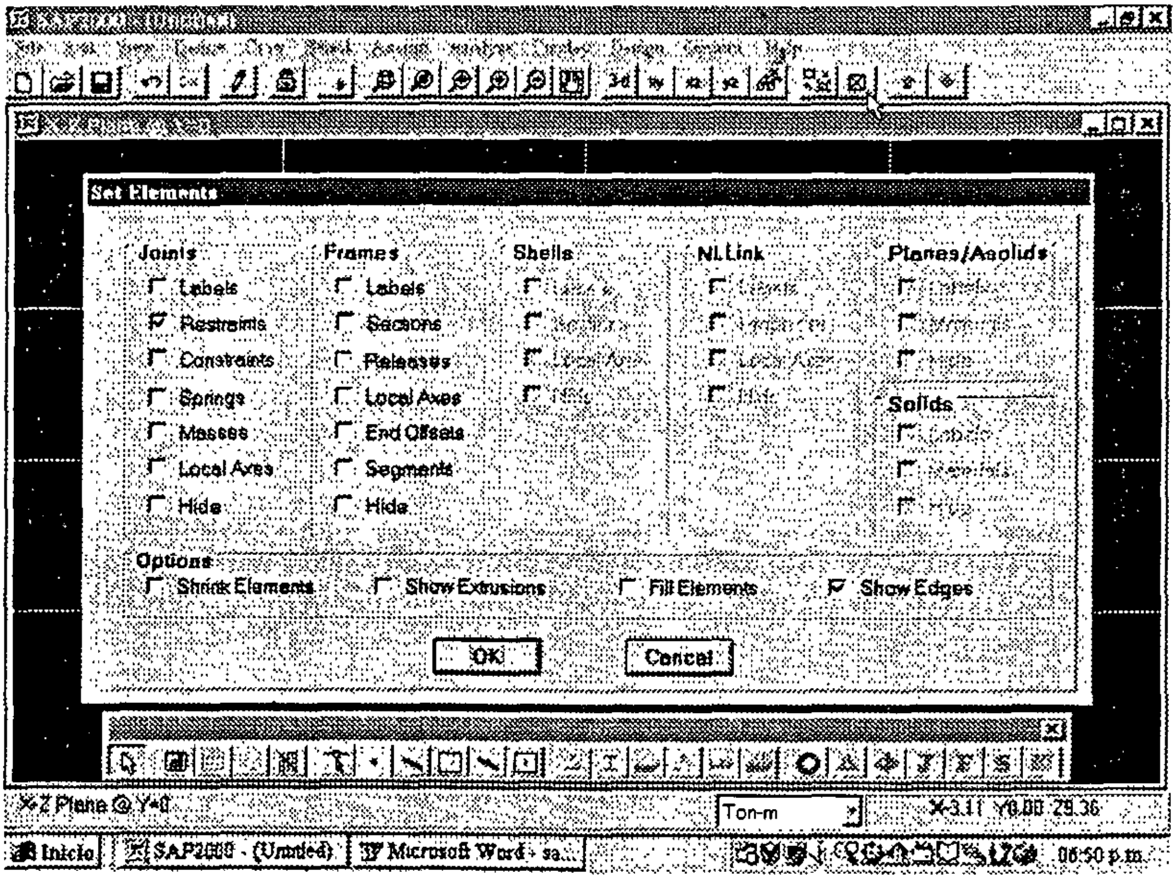


Figura 4.5 Selección de información a desplegarse en el área presentación del modelo.

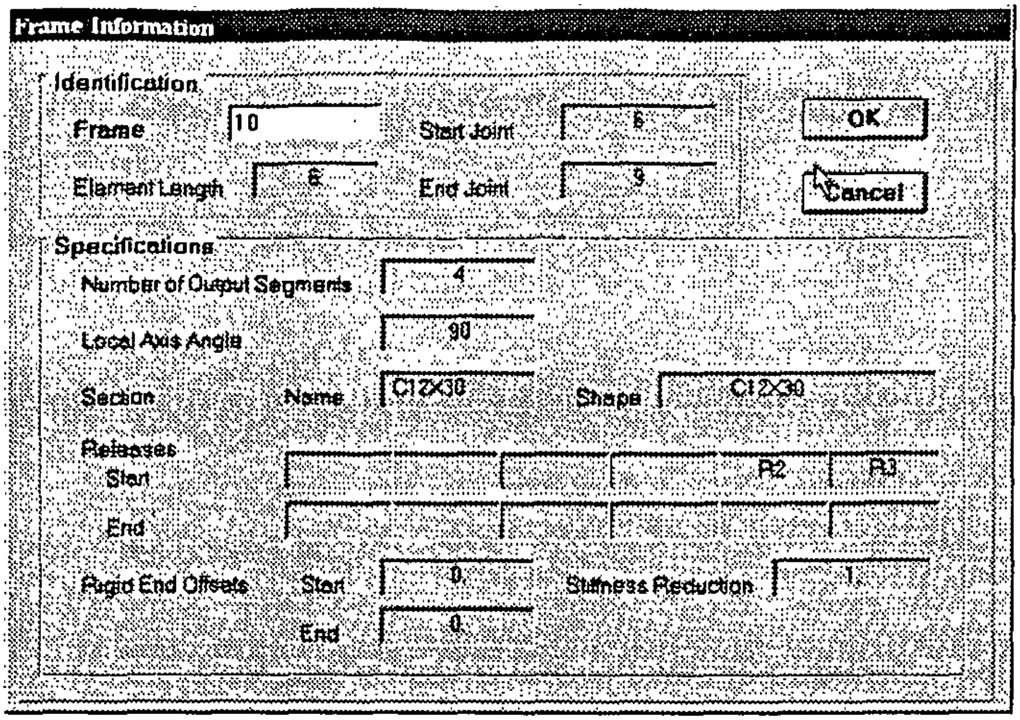


Figura 4.6 Caracteristicas del elemento barra seleccionado (la información en el cuadro en blanco puede ser modificada directamente).

4.3 GENERACIÓN DE LA GEOMETRÍA

Una de las maneras de crear el archivo de datos o de modificar su contenido, es a través del editor gráfico cuyas opciones están contenidas principalmente en el menú **Draw** (ver figura 4.7), algunas de ellas se describen a continuación.

Draw Frame Element permite adicionar un nuevo elemento barra, para ello se hace clic primero en el punto extremo de la barra y luego en el opuesto.

Draw Shell Element permite adicionar un nuevo elemento placa, haciendo clic en los puntos extremos (vértices) sucesivos del elemento empezando por cualquiera de ellos se define la geometría de este elemento.

Quick Draw Frame element permite adicionar un elemento barra haciendo un solo clic en una línea de la malla auxiliar de dibujo que este delimitada por otras dos perpendiculares a la primera, esas líneas definen los límites del elemento, si se hace clic en cualquier punto de la zona delimitada por cuatro líneas de la malla o por cuatro nudos se adicionan dos elementos barra diagonales.

Quick Draw Shell element adiciona un nuevo elemento **shell** haciendo un solo clic en cualquier punto de la zona delimitada por cuatro líneas de la malla.

Comúnmente al seleccionar alguna de las opciones anteriores la forma del cursor cambia a una flecha vertical vacía hacia arriba, para cancelar o terminar la opción se hace clic en el primer icono de la barra flotante con lo que el cursor cambia a flecha inclinada llena.

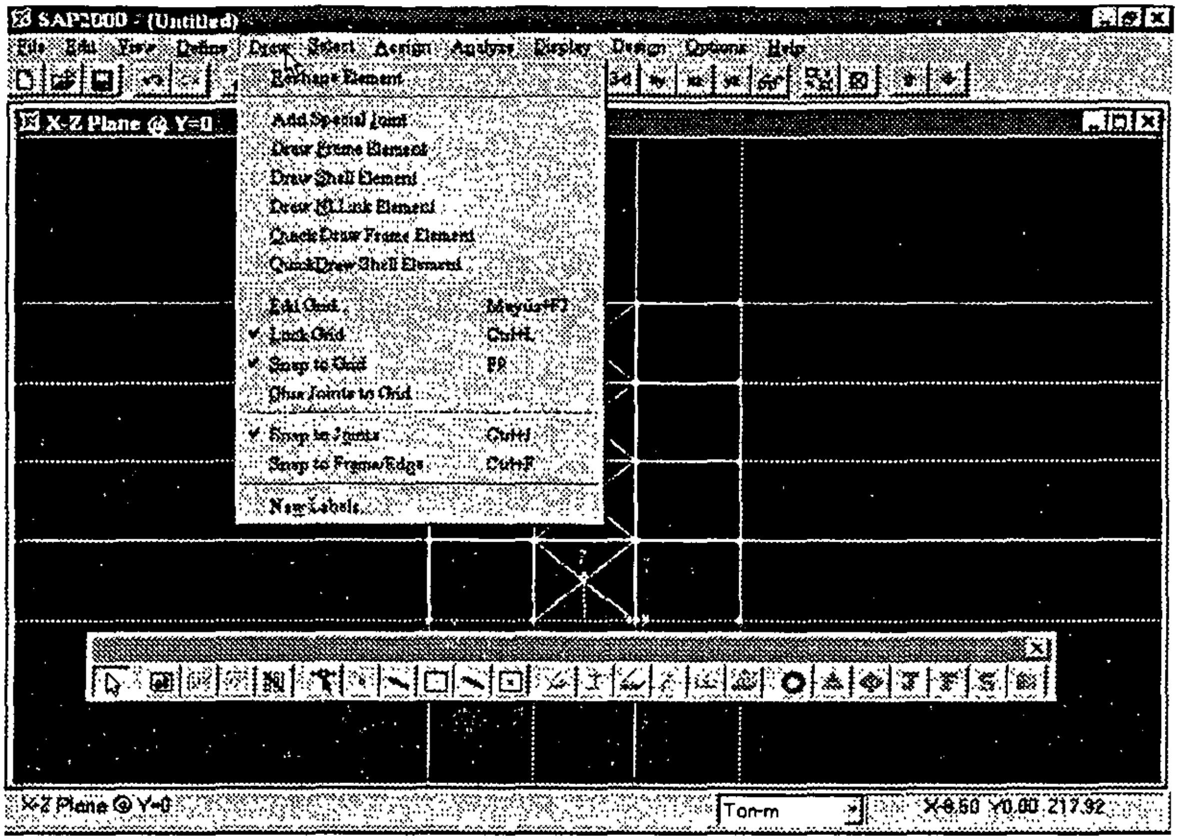


Figura 4.7 Opciones del menú **Draw**.

Con objeto de completar el modelo o realizar los cambios que se deseen, una vez que se han ubicado elementos, cuando sea posible se recomienda generar algunos otros realizando copias, giros, etc. de uno o varios de los que ya se tienen definidos.

Para tal efecto varias opciones se encuentran en el menú **Edit** (ver figura 4.8), pero para poder utilizar algunas de ellas es necesario seleccionar elementos (por ejemplo los que se van a copiar), para ello existen disponibles varias formas de seleccionar elementos, la más sencilla es hacer clic en el elemento a seleccionar (nudo o barra), el elemento seleccionado se muestra con línea discontinua, se puede anular la selección haciendo clic en un elemento seleccionado, también se pueden seleccionar elementos que queden totalmente contenidos en una ventana rectangular creada haciendo clic en uno de los vértices de la ventana y arrastrando el ratón hasta el vértice opuesto de la misma y soltando ahí, otras opciones de selección se encuentran en la opción **Select** del menú con el mismo nombre, desde luego que las acciones anteriores se pueden aplicar en repetidas ocasiones e inclusive combinar varias maneras de seleccionar y excluir (**Unselect**) elementos para lograr un resultado deseado.

Hecha la selección de algunos elementos (también puede ser uno o todos) se pueden llevar a cabo ciertas acciones con ellos dando como resultado posibles cambios a esos elementos y en general al modelo o estructura por analizar, cuando el efecto final no es el esperado se recomienda cancelar la acción, para ello se hace clic en el icono (**Un Do**) que está casi por debajo del menú **View**.

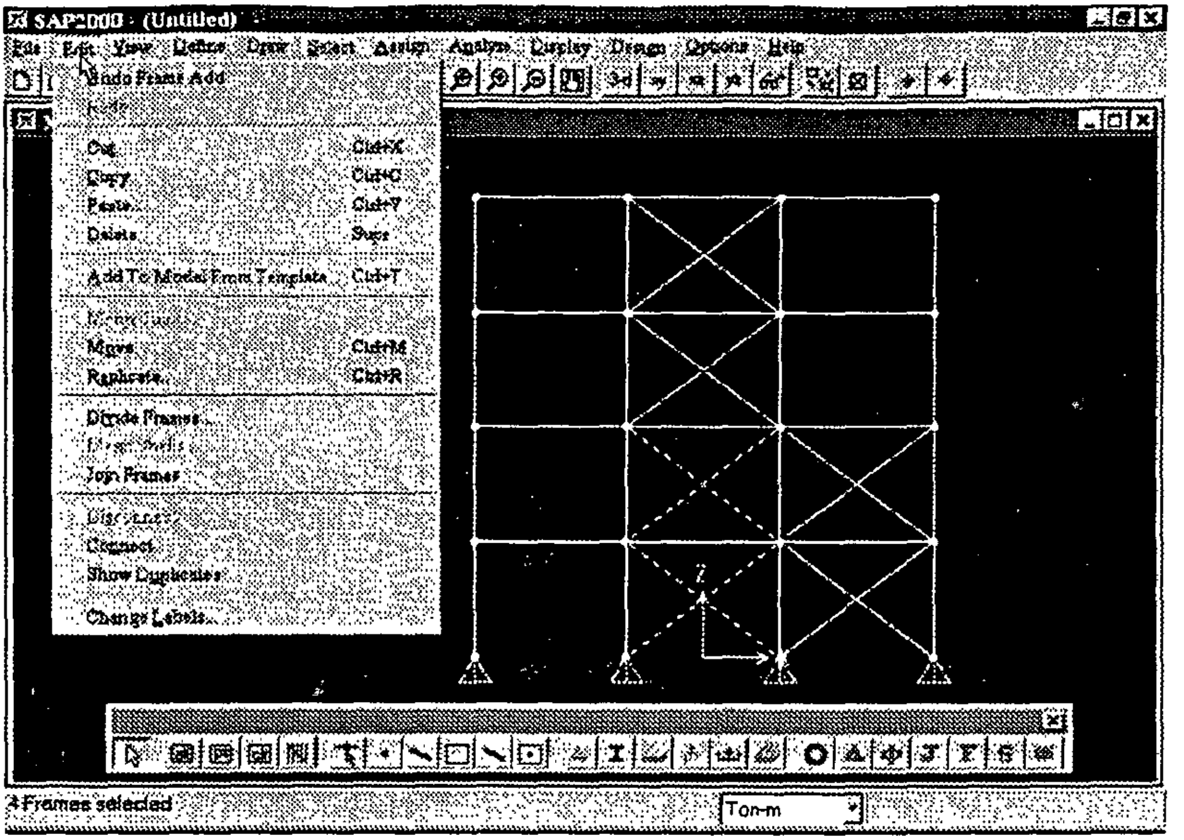


Figura 4.8 Opciones del menú **Edit**.

Cut y **Delete** Elimina los elementos seleccionados.

Copy copia los elementos seleccionados a una memoria temporal permitiendo ser insertados posteriormente, los elementos que actualmente se encuentran seleccionados no se suprimen, cuando se aplica nuevamente la opción **Copy** a una nueva selección, los elementos seleccionados anteriormente (si es que los había) se eliminan de la memoria temporal quedando los actualmente seleccionados.

Paste inserta los elementos que se almacenaron previamente en la memoria temporal mediante la opción **Copy**, al seleccionar esta opción se presenta una ventana en donde se puede especificar un incremento a todas las coordenadas de los nudos y a los nudos extremos de los elementos guardados previamente con la opción **Copy** (ver figura 4.9).

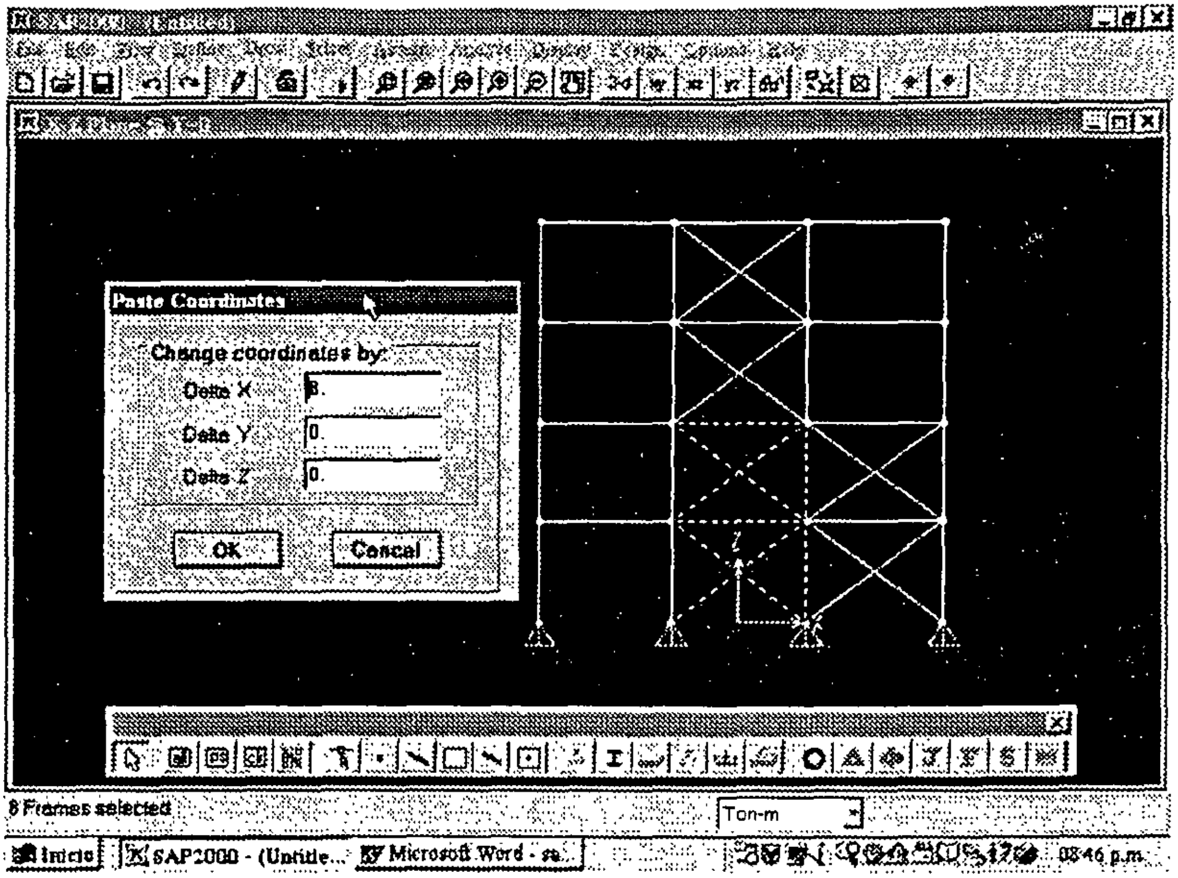


Figura 4.9 Opción **Paste** del menú **Edit**.

Move Una vez seleccionados algunos nudos este comando permite cambiar las coordenadas de los nudos seleccionados desplazándolos hacia nuevas posiciones obtenidas a partir de sus coordenadas actuales y de la información que el usuario proporcione en la ventana que se despliega cuando se elige esta opción (ver figura 4.10).

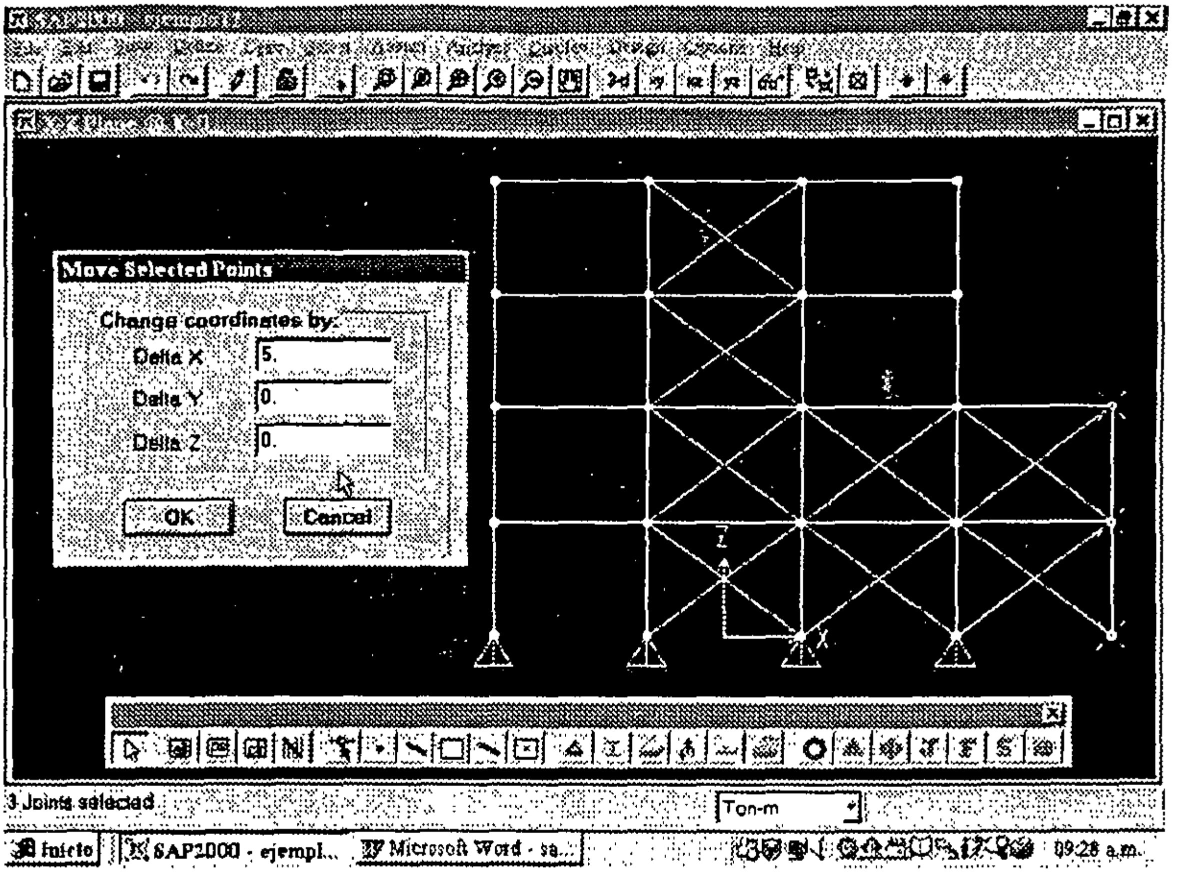


Figura 4.10 Información complementaría de la opción **Move** del menú **Edit**.

Replicate Una de las opciones más poderosas con la que se pueden realizar varios efectos es la opción **Replicate** del menú **Edit**, dentro de esta opción a su vez se encuentran disponibles otras 3, **Linear**, **Radial** y **Mirror.**

La opción **Linear** permite realizar varias copias de los elementos seleccionados, esas copias se pueden realizar en cualquiera de las direcciones x, y o z, por ejemplo si las copias se quieren realizar en dirección x se especifica un valor de x diferente de cero en la caja respectiva y cero en las demás (ver figura 4.11), desde luego se pueden especificar valores diferentes de cero con el efecto correspondiente, la opción **Radial** permite realizar copias en dirección radial (angular) especificando el eje alrededor del cual se van a hacer las copias así como el incremento en grados y el número de estas.

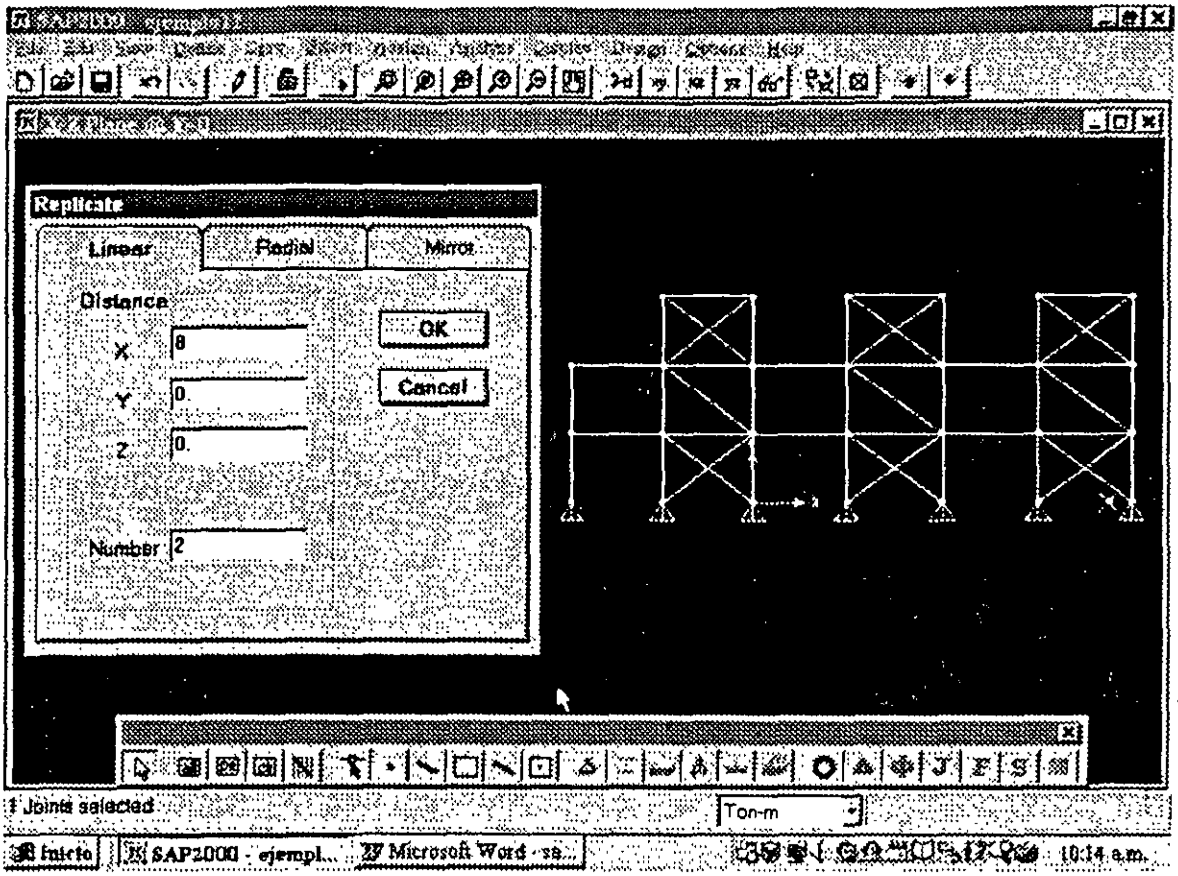


Figura 4.11 Efecto de **Linear** en la opción **Replicate** del menú **Edit**.

La opción **Mirror** permite realizar una copia tipo espejo de los elementos seleccionados especificando la posición del espejo mediante la selección de un plano (xy, yz o xz) y la distancia del origen a la posición del espejo (ver figura 4.12), esta opción resulta muy útil cuando se tiene una estructura simétrica ya que se introduce una parte de la misma y se genera la otra (parte simétrica) mediante la opción **Mirror**.

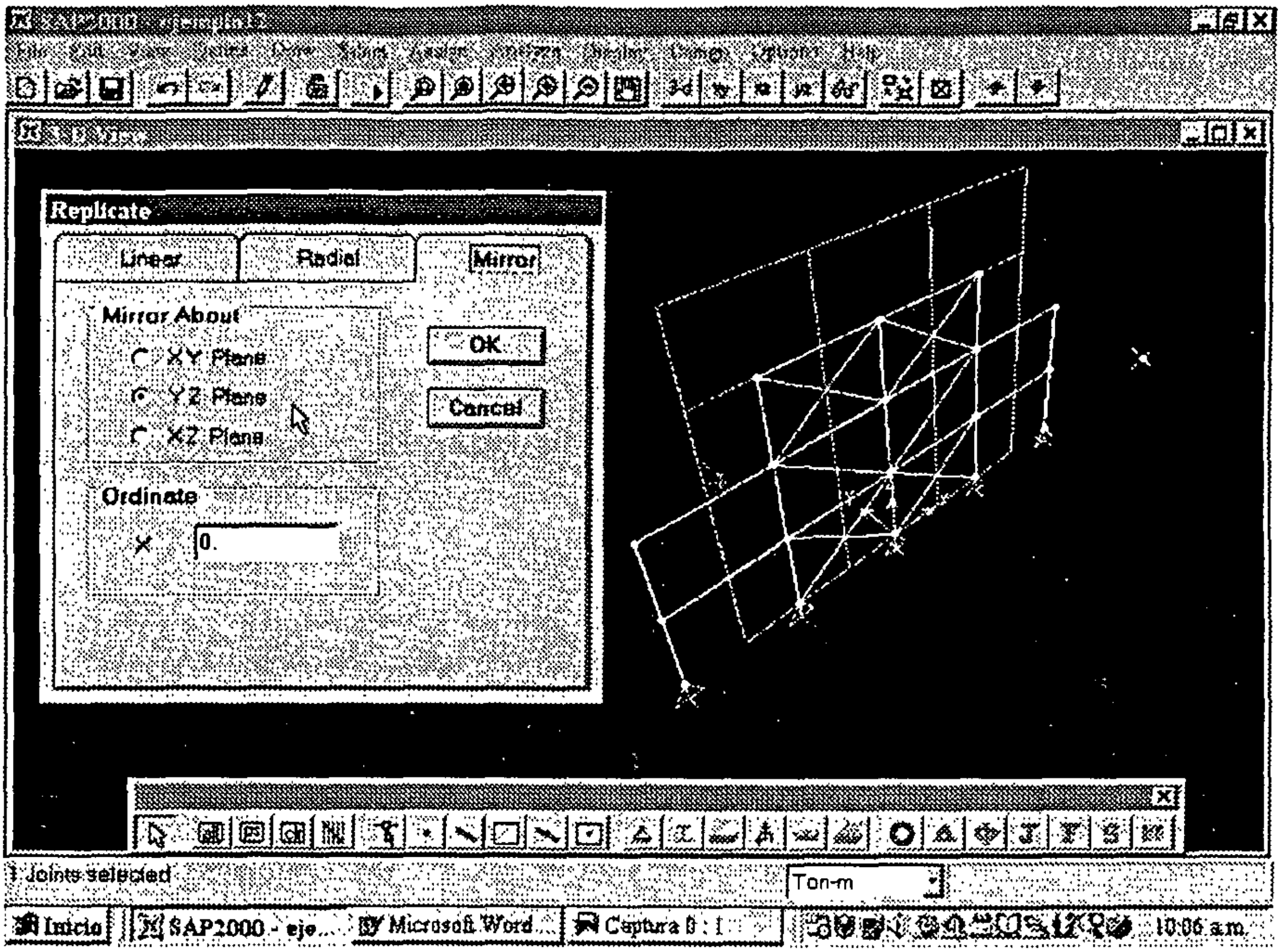


Figura 4.12 Efecto al seleccionar **Mirror** de la opción **Replicate** en el menú **Edit**.

4.4 DEFINICIÓN Y ASIGNACIÓN DE MATERIALES

En el menú Define en la opción **Materials** se podrán especificar las características de los materiales del cual estarán formados los elementos estructurales, en este menú se pueden especificar materiales tales como concreto, acero y otros; después de seleccionar esta opción aparece el cuadro que se muestra en la figura 4.13, en donde como puede observarse mediante la opción **Modify/Show Materials**, se mostrarán con la posibilidad de modificar algunas características del material que interviene para el análisis y el diseño de elementos (ver figura 4.14).

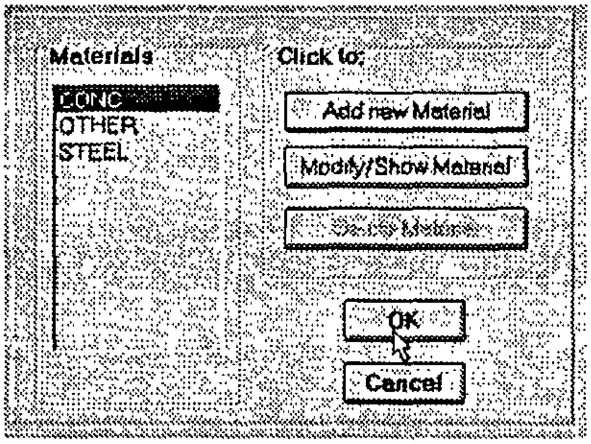


Figura 4.13 Ventana para definición de materiales.

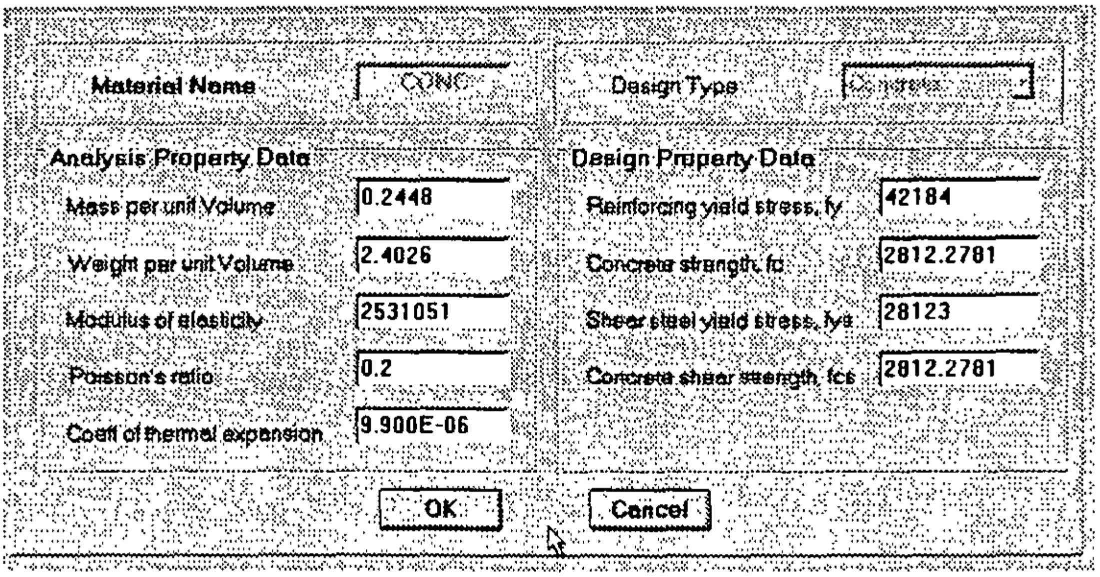


Figura 4.14 Ventana **Modifi/Show Material** de la opción **Define Materials**.

Add New material permite definir un nuevo material, se tendrá que especificar desde el nombre (**Material Name**), formado por un conjunto de hasta ocho caracteres el cual hará referencia a este material, se tendrán que proporcionar por lo menos los datos que se muestran en el cuadro análisis **Property Data**, sobre todo el módulo elástico y la relación de Poisson, en el caso de que se requiera considerar el peso propio en alguna condición de carga se tendrá que proporcionar el peso por unidad de volumen, si se va a solicitar alguna opción de análisis dinámico en donde se quiera considerar a la masa de la estructura repartida a lo largo de sus elementos será necesario proporcionar el valor de la masa por unidad de volumen, en el caso de que se requiera considerar el efecto de temperatura es necesario proporcionar el coeficiente de expansión térmica (ver figura 4.15).

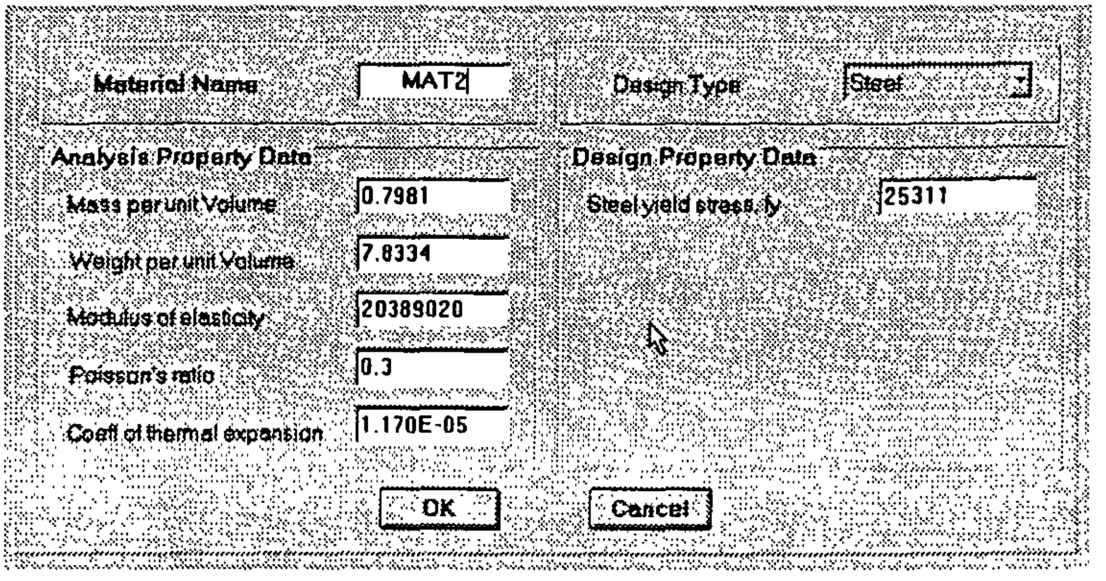


Figura 4.15 Opción **Add New** material de **Define Materials**

Se pueden definir varios materiales dependiendo de los que se requieran para especificar a los elementos en la estructura, la opción **Materials** del menú **Define** también permitirá eliminar algún material de los que se muestran en el cuadro **Materials** con excepción de los materiales **Conc** y **Steel**, para ello sólo se hace clic en el nombre del material a eliminar y luego en el botón **Delete Material**.

4.6 DEFINICION Y ASIGNACION DE PROPIEDADES GEOMETRICAS

En el menú **Define** también se encuentra presente la opción para definir características de las secciones transversales (**Frame Sections**) de los elementos que están presentes en la estructura por analizar. En la ventana correspondiente (ver figura 4.16), se tiene la opción **Import** para seleccionar las propiedades de una base de datos con extensión **PRO**, la versión educativa del programa **SAP2000** proporciona los archivos **Aisc.Pro**, **Cisc.Pro** y **Sections.Pro** de los dos primeros se pueden seleccionar algunas formas comunes, esos archivos se encuentran en la carpeta **SAP2000e**.

También se pueden definir las propiedades a partir de formas comunes mediante la opción **Add**, otras opciones que también se encuentran disponibles permiten modificar (**Modify/Show Section**) o eliminar alguna propiedad (**Delete Section**),

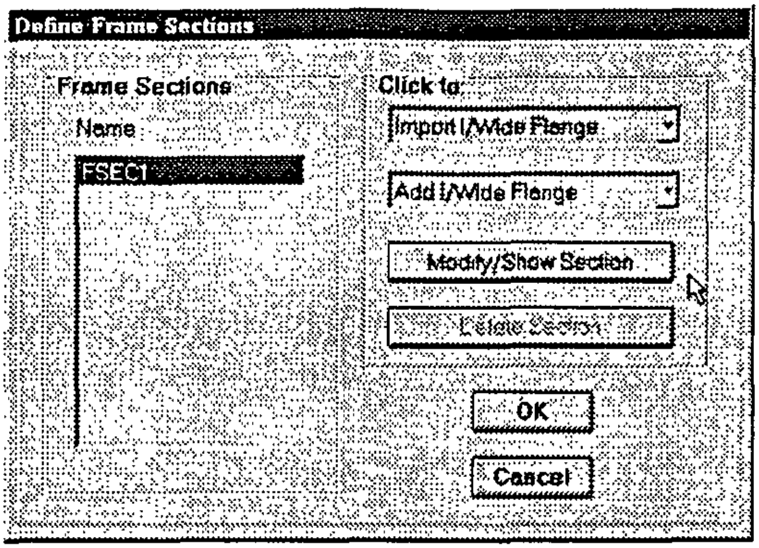


Figura 4.16 Opciones en **Define Frame Sections** del menú **Define**.

En la opción **Add** se tendrá que seleccionar la forma de la sección (rectangular, circular, tee, general, etc., ver figura 4.17).

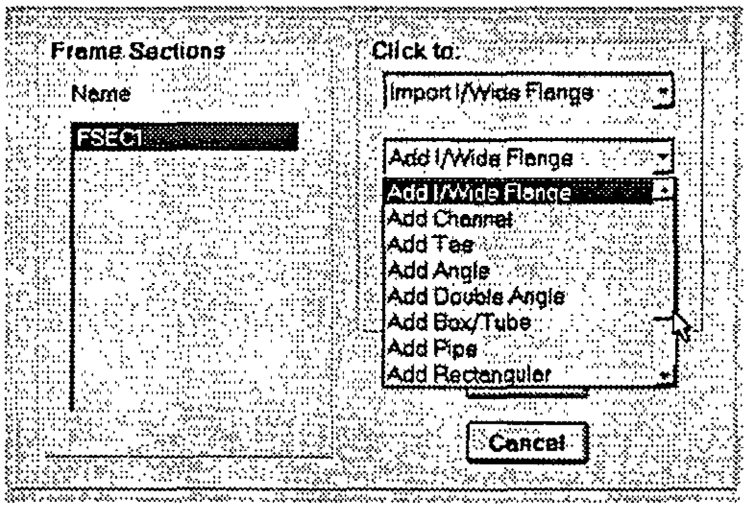


Figura 4.17 Selección de una forma predeterminada.

Una vez seleccionada la forma habrá que proporcionar algunas de las dimensiones de la misma con las cuales el programa obtiene de manera automática las propiedades geométricas de la forma definida (ver figura 4.18).

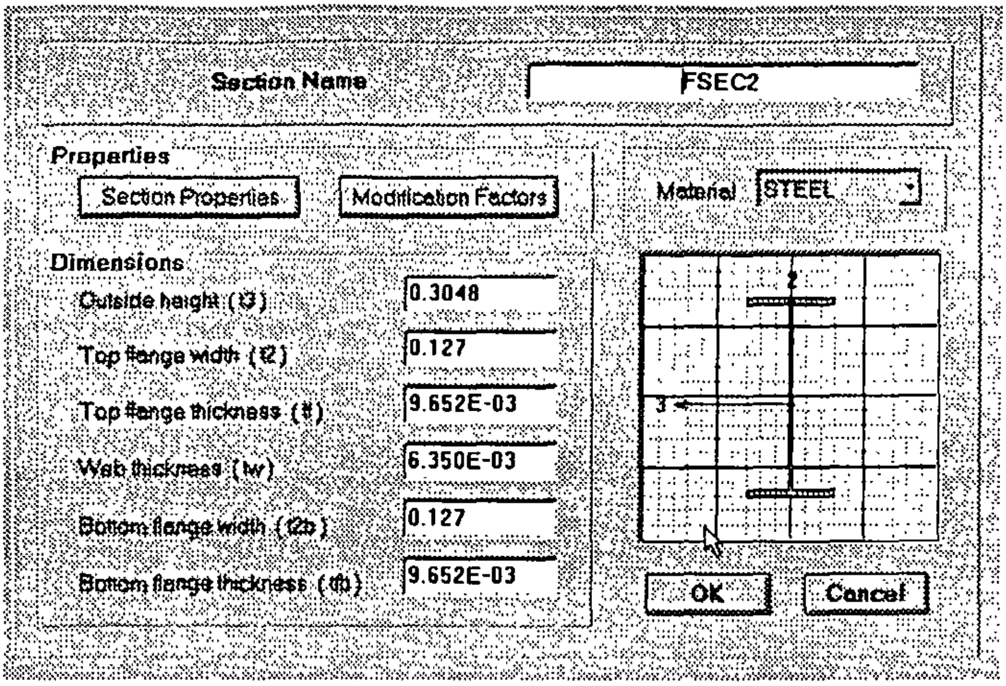


Figura 4.18 Especificación de las dimensiones de la forma de una sección transversal seleccionada.

El nombre de la sección se puede cambiar modificando el contenido del cuadro en blanco (**Section Name**), este nombre se utilizará para referencias posteriores (asignar esta sección transversal a uno o más elementos del modelo). Las características a modificar se presentan en el cuadro **Dimensions**, del cuadro **Material** se deberá seleccionar el material (los materiales se definieron previamente, ver párrafos anteriores) del cual esta o estará formada esa sección transversal. Una vez proporcionadas las dimensiones de la forma de la sección transversal se pueden mostrar sus propiedades geométricas (área, momentos de inercia, etc.) seleccionando la opción **Section Properties** del cuadro **Properties,** se pueden modificar (aumentar o disminuir en cierta proporción) algunas de esas propiedades modificando el factor correspondiente a la propiedad que se quiera modificar (el factor que se especifica es con respecto a la unidad) para ello habrá que seleccionar el botón **Modification Factors** del cuadro **Properties** y modificar el contenido del cuadro en blanco que corresponda a la propiedad que se quiere modificar.

Una vez definidas las distintas secciones de los diversos elementos estructurales habrá que indicar la sección transversal que corresponda al o a los elementos estructurales (la forma de la estructura ya se ha generado), primero se seleccionan los elementos que tiene una misma sección transversal, para ello se puede utilizar algún método de selección de la opción **Select** del menú con el mismo nombre, enseguida seleccionar **Sections** de la opción **Frame** en el menú **Assign,** con lo que aparece la ventana que se muestra en la figura 4.16, por último, en esa ventana se tendrá que seleccionar el nombre de la sección (la cual se definió previamente) del cuadro **Frame Sections**, después de hacer clic en el botón OK se asigna a los elementos seleccionados las características especificadas en la sección transversal seleccionada. La operación anterior se repetirá tantas veces como sea necesario para asignar secciones a todos los elementos que componen al modelo.

4.6 CONDICIONES DE FRONTERA, TIPOS DE APOYO

Para especificar los tipos de apoyo o condiciones de frontera de la elástica de la estructura primero se seleccionan aquellos nudos que tengan las mismas restricciones de desplazamiento, esto se hace con algunas de las opciones aplicables del menú **Select** y después seleccionar **Restraints** de la opción **Joint** del menú **Assign** (ver figura 4.19), desplegándose la ventana que se muestra en la figura 4.20, en ella se habrá de indicar el tipo de restricción que tendrán los nudos que se han seleccionado previamente, a menos que se modifiquen las direcciones 1, 2 y 3, corresponden a las direcciones globales X, Y y Z respectivamente, se puede seleccionar algún tipo de apoyo particular de uso común haciendo clic en alguno de ellos en el cuadro **Fast Restraints**. La operación anterior se puede aplicar en repetidas ocasiones para especificar completamente todos los nudos restringidos que tiene el modelo.

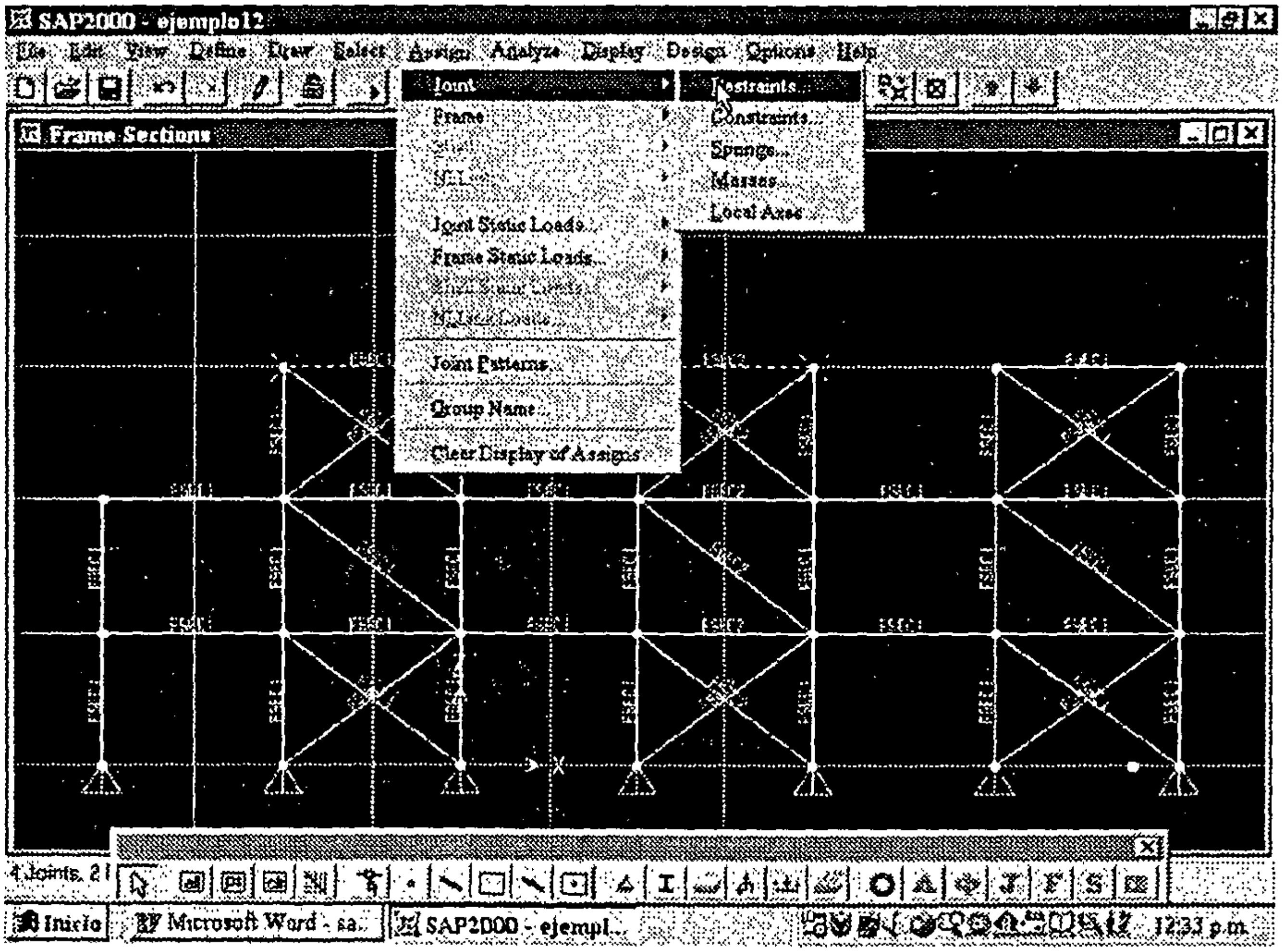


Fig. 4.19 Selección de restricciones.

Para cambiar las restricciones de algún nudo se requiere seleccionarlo y asignarle las nuevas restricciones, si se desea que ese nudo tenga posibilidad de desplazamiento lineal y angular en todas las direcciones habrá que dejar en blanco todos los cuadros del marco **Restrainsts in Local Directions** o bien hacer clic en el icono con un punto negro del marco **Fast Restraints**.

4.7 ASIGNACION DE FUERZAS Y COMBINACIONES

Para introducir diversos tipos de fuerza estática al modelo, primero habrá que definir condiciones de carga estática, para ello se selecciona la opción **Static Load Cases** del menú **Define** mostrándose la ventana de la figura 4.20, en ella se puede adicionar una nueva (**Add New Load**), modificar características de una que existe (**Change Load**), o suprimir una condición de carga (**Delete Load**), resulta lógico que al menos se debe proporcionar una condición de carga.

El nombre de la condición se especifica en el cuadro en blanco debajo de **Load** y si se quiere considerar el peso propio en esa condición de carga se debe de proporcionar el valor de 1 en el cuadro en blanco debajo de **Self Weight Multiplier**, una vez que se han introducido los datos anteriores se puede seleccionar **Add...** para definir una nueva condición de carga o bien **Change...** para cambiar los datos de la condición de carga seleccionada (con fondo oscuro) por los datos de los cuadros en blanco.

Para modificar el nombre y el multiplicador del peso propio además de introducir el nuevo valor en los cuadros en blanco habrá que seleccionar la condición que se quiera modificar haciendo clic sobre ella, con lo que el fondo de la condición seleccionada cambia a obscuro y después hacer clic en el botón **Change Load** se realizan los cambios indicados ya que hasta que se ha hecho clic en este botón quedan registrados esos cambios es decir no basta modificar el contenido de las cajas en blanco.

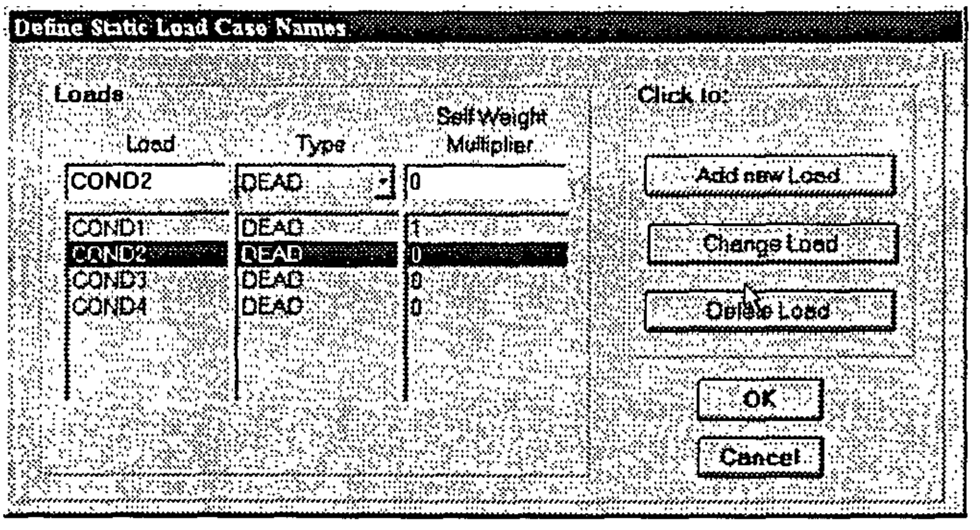


Fig. 4.20 Definición de condiciones de carga estática.

Para la asignación de fuerzas y o momentos a los nudos habrá que seleccionar aquellos nudos que tengan las mismas fuerzas y después seleccionar **Forces** de la opción **Joint Static Loads** en el menú **Asign** (ver figura 4.21).

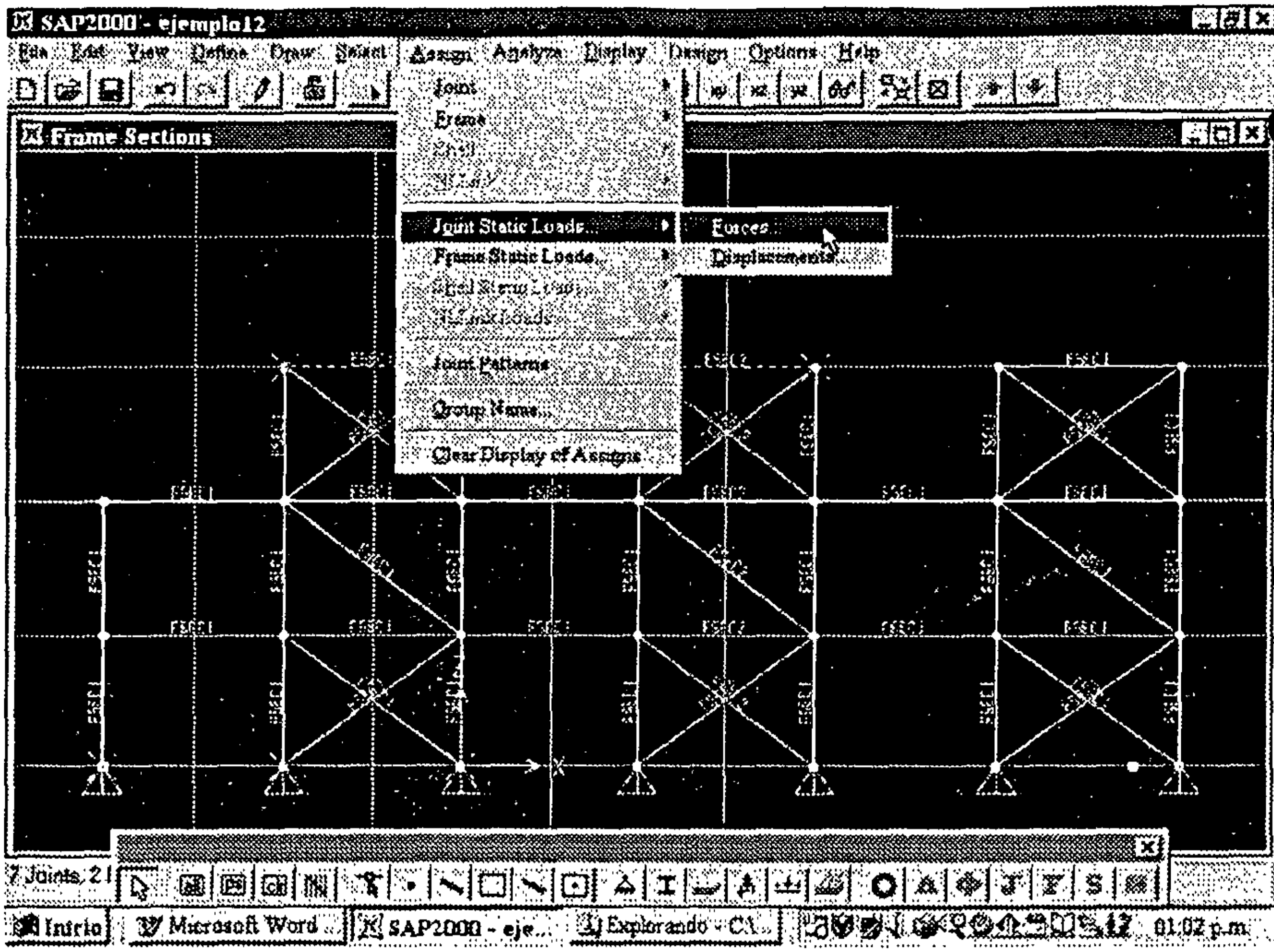


Fig. 4.21 Opción para asignar fuerzas a los nudos previamente seleccionados.

Enseguida se despliega la ventana mostrada en la figura 4.22, en ella se habrá de seleccionar del marco **Load Case Name** la condición en que se incluirán las fuerzas que se están especificando (por omisión aparece **LOAD1**), en el cuadro correspondiente a la dirección de la fuerza y o momento que actuará sobre los nudos seleccionados se introducirán los valores respectivos (en el marco **Loads**), también se encuentran disponibles las opciones:

Add To Existing Loads (seleccionada por omisión), la cual adicionará a las fuerzas existentes en los nudos seleccionados las nuevas fuerzas que se están especificando, es decir si los nudos ya tenían fuerzas se les adicionarán las nuevas fuerzas cuyos valores se han introducido en los cuadros en blanco.

Replace Existing Loads permitirá eliminar las fuerzas existentes en los nudos seleccionados remplazándolas por las que se están especificando en el marco **Loads**.

Delete Existing Loads suprimirá las fuerzas existentes en los nudos seleccionados, independientemente de los valores que se están especificando en el marco **Loads**.

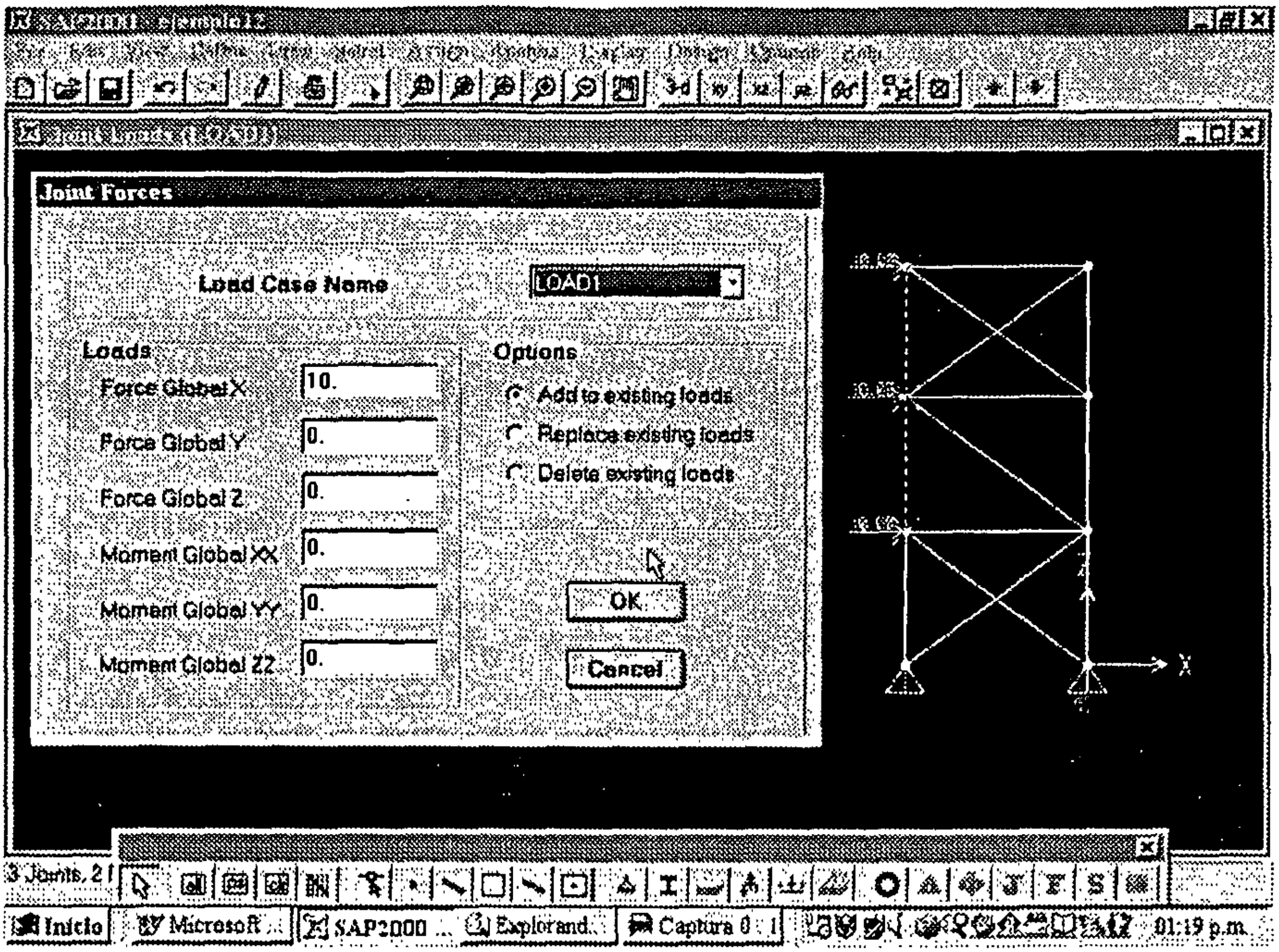


Fig. 4.22 Especificación de fuerzas en los nudos.

Para definir y asignar fuerzas a las barras primero se seleccionan las barras a las cuales se les asignarán las mismas fuerzas, después se selecciona el tipo de fuerza (uniforme, concentrada, variación lineal, etc.) de la opción **Frame Static Loads** del menú **Assign** (ver figura 4.23).

Por ejemplo para cargas puntuales y o uniformes en las barras se muestra la ventana de la figura 4.24, en donde se selecciona el nombre de la condición a donde se incluirán las fuerzas que se están especificando, así como el tipo de carga (fuerza o momento) así como la dirección en que actuarán y la opción a utilizar (**Add..., Replace... y Delete...**). En los cuadros en blanco del marco **Point Loads** se especifica el valor de las cargas concentradas así como la posición de cada una de ellas con respecto a la longitud del elemento, es decir si el valor de **Distance** es 0.5 indica que la carga está aplicada a la mitad del elemento, en el cuadro en blanco del marco **Uniform Load** se proporciona el valor correspondiente a la carga uniforme que actuará sobre el elemento. Pueden especificarse simultáneamente cargas concentradas y uniformes o sólo algún tipo de los anteriores.

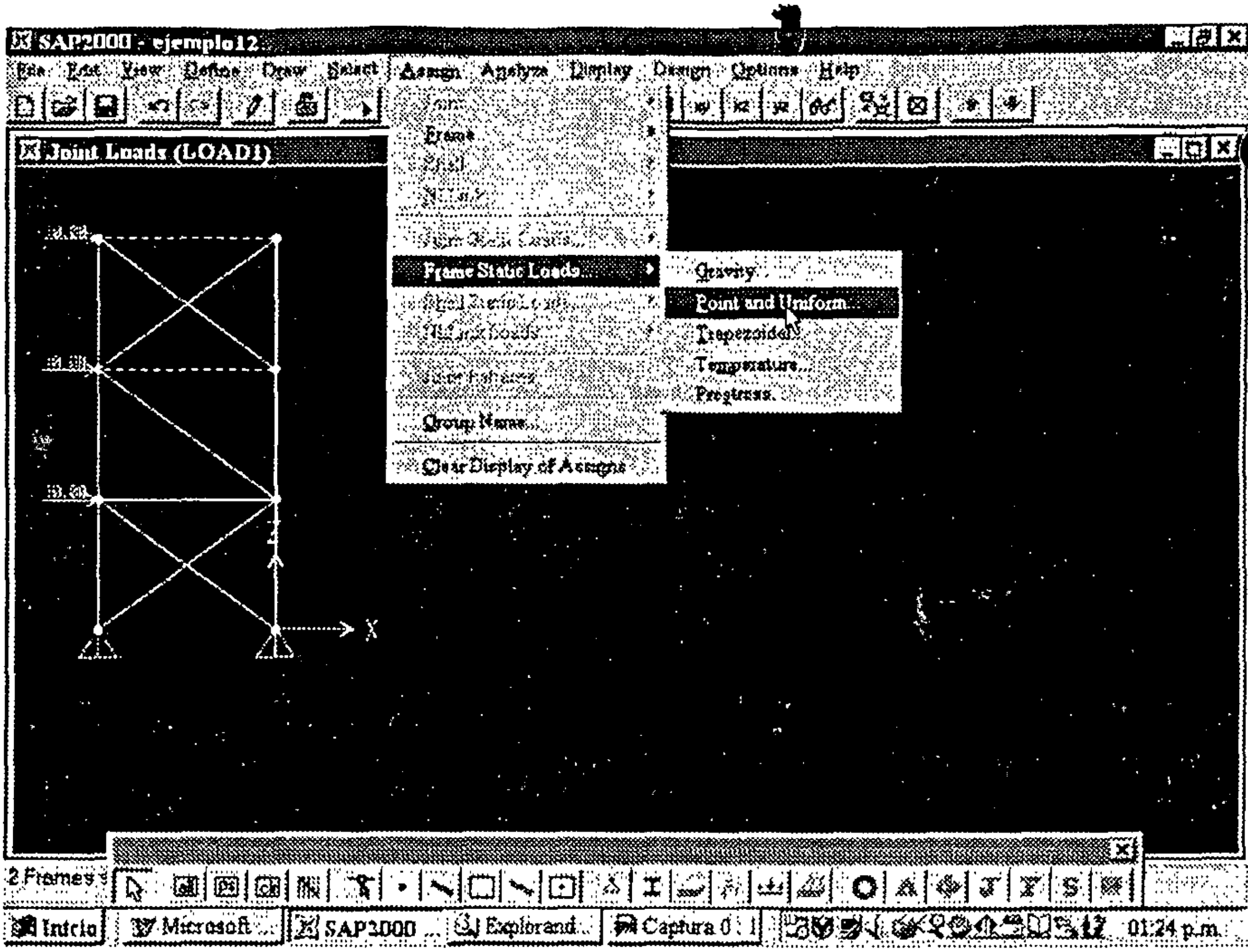


Fig. 4.23 Opción para introducir fuerzas en las barras.

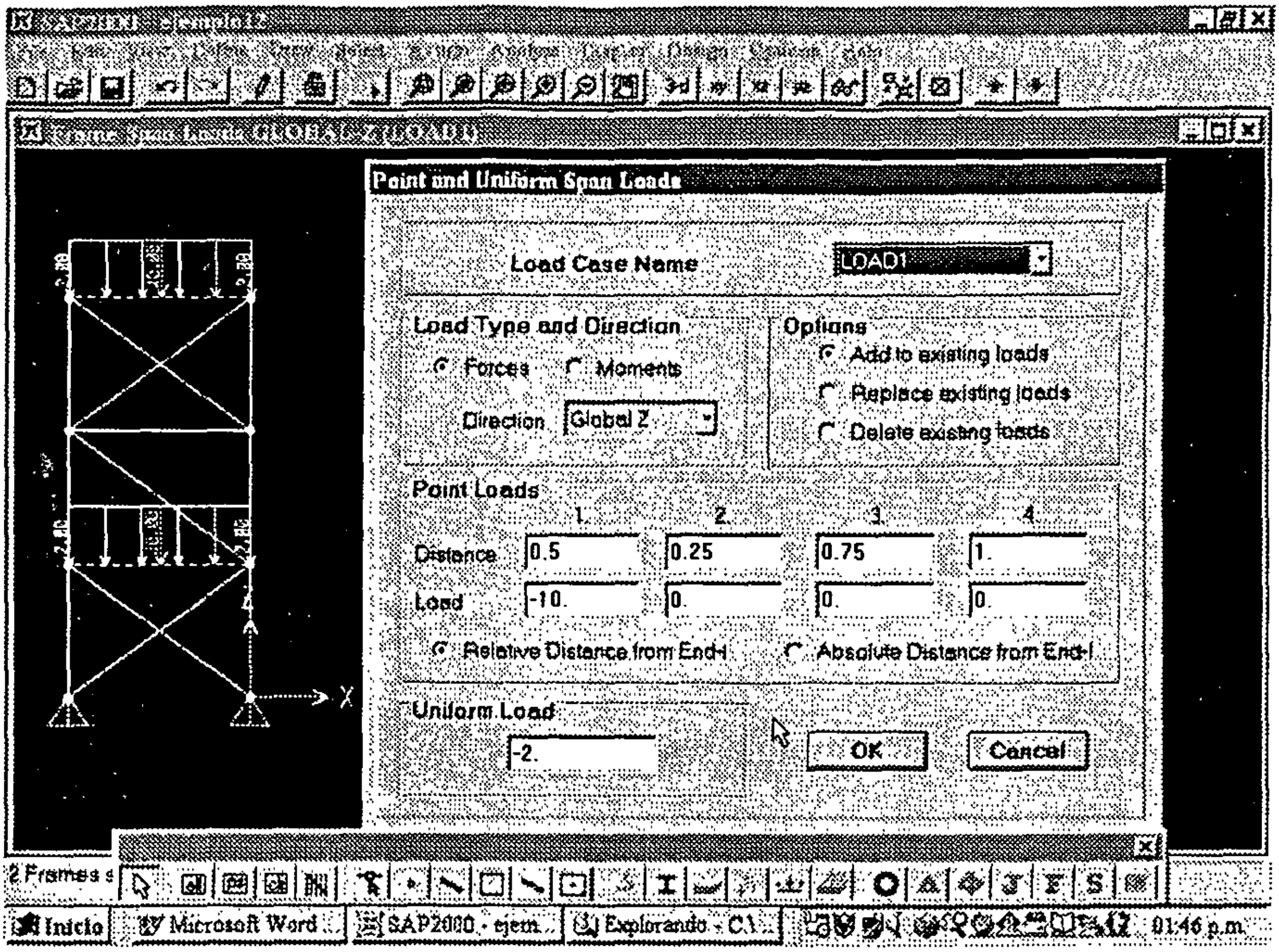


Fig. 4.24 Definición de fuerzas uniformes y/o concentradas en las barras.

Una vez que se especifican las fuerzas y se hace clic en el botón **OK** se ejecuta la opción seleccionada, en el caso de que esta sea adicionar o remplazar cargas, estas se muestran con sus caracteristicas en el área de dibujo de la pantalla.

Definidas las condiciones de carga se pueden realizar combinaciones de las anteriores, es decir condiciones de cargas dependientes, para ello se selecciona la opción **Load Combinations** del menú **Define** mostrándose la ventana de la figura 4.25, con la posibilidad de adicionar, modificar y suprimir combinaciones de carga estas opciones se muestran en el marco **Combinations** las combinaciones que se tengan definidas hasta el momento.

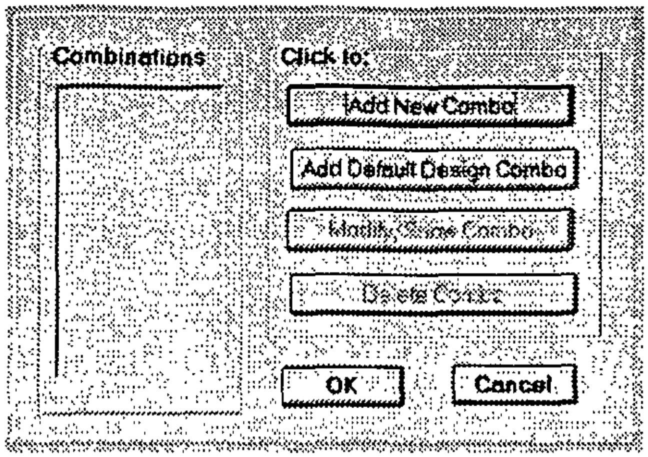


Fig. 4.25 Ventana para especificar y modificar combinaciones.

La opción para adicionar una nueva combinación despliega la ventana que se muestra en la figura 4.26, ahí se especificará el nombre, tipo y algún título para la combinación. Para definir las condiciones de carga que participarán en la combinación que se especifica, así como su respectivo factor de participación (con relación a la unidad, 1=100%) se selecciona el nombre y se modifica el valor en el cuadro en blanco debajo de **Scale Factor** en el marco **Define Combination** y después se hace clic en cualquiera de los botones **Add, Modify,** o **Delete**.

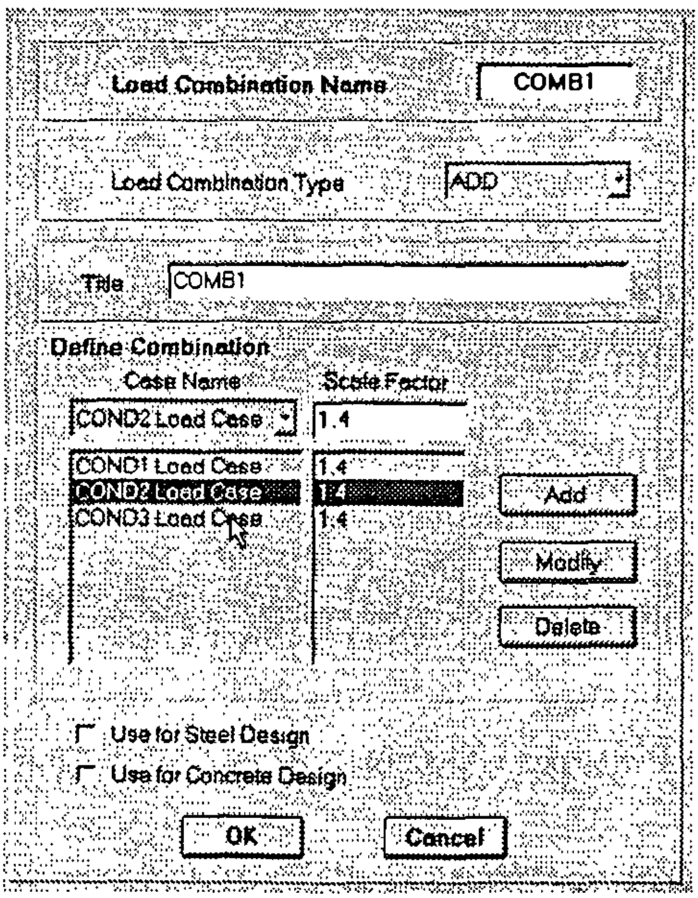


Fig. 4.26 Especificación de las características de una combinación.

Cuando se han especificado los datos de la combinación se hace clic en el botón **OK**.

Una vez que se han definido combinaciones se puede modificar sus características con la opción **Modify/Show Combo** o bien suprimir alguna combinación con la opción **Delete Combo**, cualquiera de estas opciones se selecciona haciendo clic sobre ella en el marco **Combinations.** Se pueden especificar tantas combinaciones como el problema de Análisis lo requiera.

4.8 OPCIONES DE ANALISIS SELECCIÓN DE RESULTADOS

Una vez que se han especificado completamente las características geométricas, elásticas, condiciones de frontera y fuerzas se está en posibilidades de que el programa **SAP2000** realice el Análisis Estructural del modelo, sin embargo es conveniente especificar algunas opciones de Análisis, para ello se selecciona la opción **Set Options** del menú **Analyze**, desplegándose la ventana que se muestra en la figura 4.27.

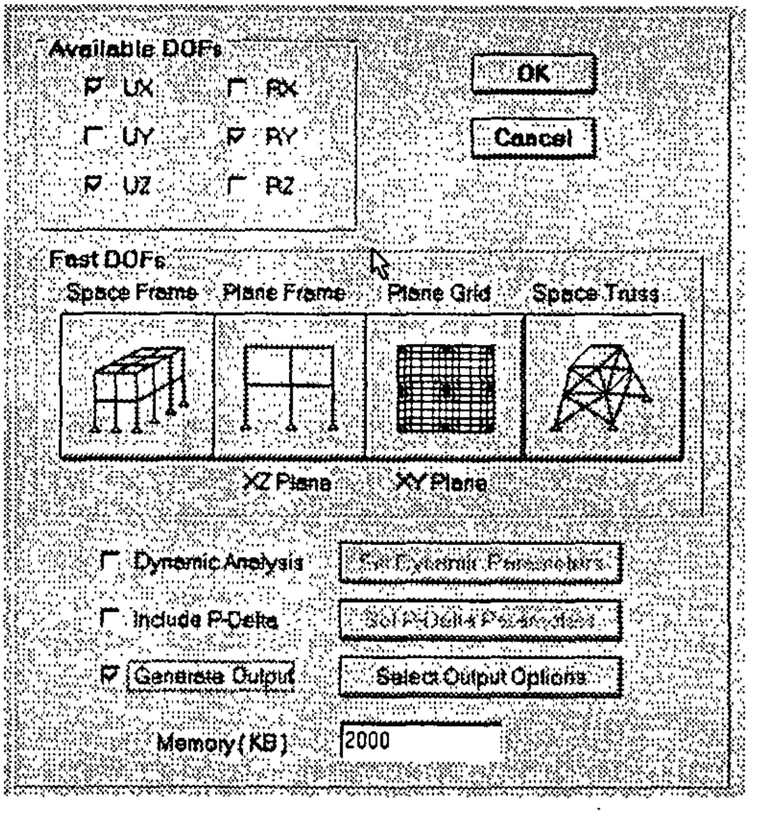


Figura 4.27 Selección de opciones de Análisis.

En ella se pueden seleccionar las componentes de desplazamiento independientes o grados de libertad que se considerarán para el análisis, **SAP2000** permite analizar estructuras en un espacio tridimensional por lo que cada nudo tiene la posibilidad de desplazarse lineal y angularmente en tres direcciones ortogonales, es decir en general posee 6 grados de libertad (a menos que se indique otra alternativa).

Si la estructura está contenida en un plano es conveniente indicar los grados de libertad que no intervienen en el Análisis con objeto de eliminar la posibilidad de inestabilidad en dirección perpendicular al plano de la estructura, disminuyendo además el tiempo de ejecución del Análisis, lo anterior se realiza desactivando grados de libertad en el marco **Available DOFs** o bien permitiendo que el programa lo realice dependiendo del tipo de estructura que se selecciona haciendo clic en alguno de los iconos que se muestran en el marco **Fast DOFs** y que corresponda con las características de la estructura que se vaya a analizar.

En la parte inferior de la ventana se muestran las opciones de Análisis Dinámico y efectos P-Delta, también se pueden seleccionar resultados que han de almacenarse en el archivo de salida (nombre.OUT), en el último renglón se muestra en un cuadro en blanco el valor de la memoria reservada para la solución del problema, este valor deberá aumentarse en caso de que no sea suficiente cuando se muestre el mensaje correspondiente durante el proceso de Análisis.

La selección de resultados del Análisis se puede realizar haciendo clic en el cuadro en blanco a la izquierda de **Generate Output** (ver figura 4.27) y después de hacer clic en el botón **Select Output Options** se muestra la ventana de la figura 4.28.

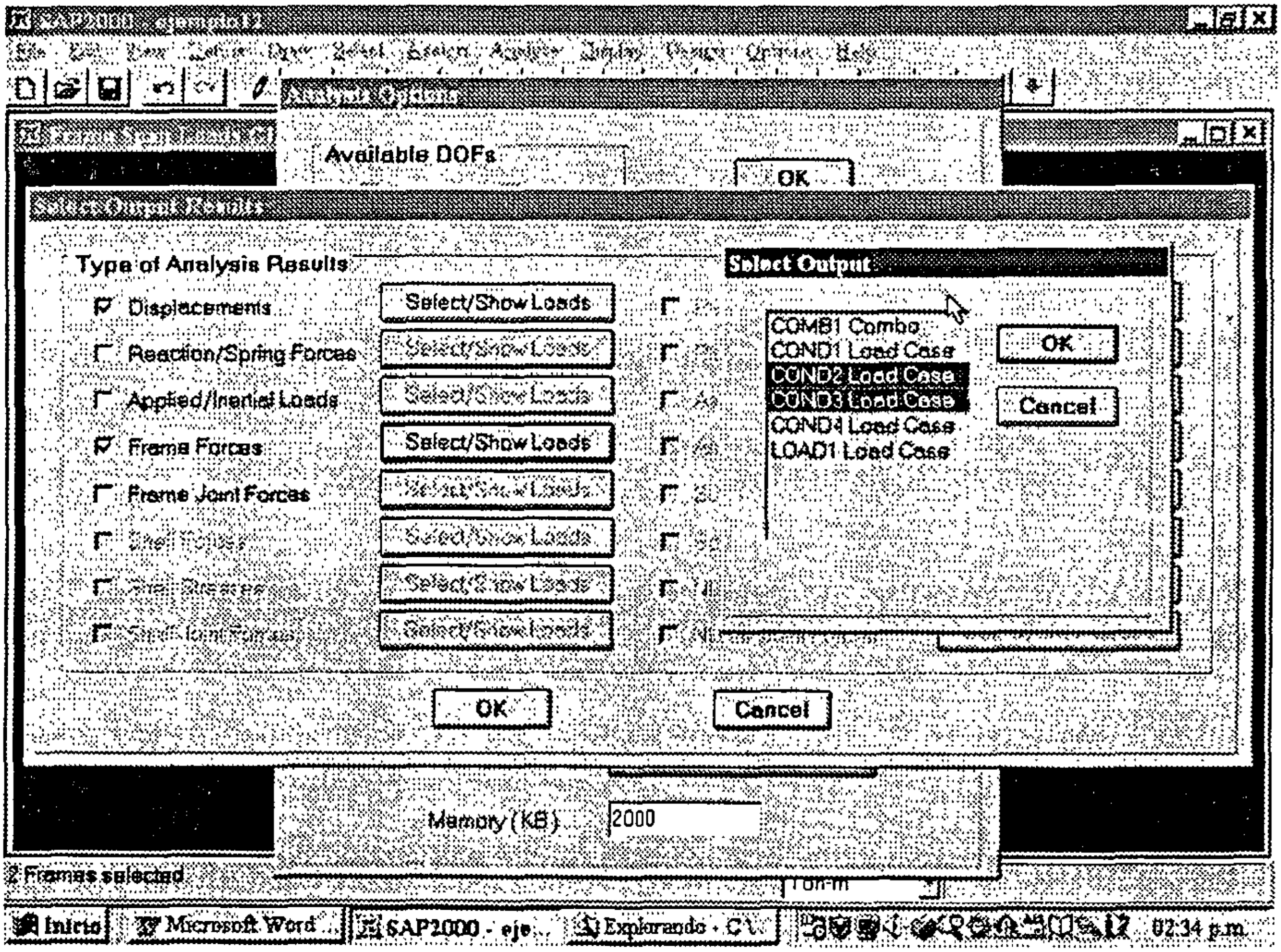


Figura 4.28 Selección de combinaciones de carga y tipos de resultados a incluirse en el archivo de salida.

En ella habrá que indicar los resultados que se incluirán en el archivo de salida haciendo clic en los cuadros en blanco y seleccionando para cada uno de esos resultados las condiciones de carga de los cuales se requieren los resultados seleccionados, lo anterior se logra haciendo clic en el botón **Select/Show Loads** correspondiente, con lo que se presentará una ventana mostrando las condiciones y combinaciones de Análisis que se han especificado para la estructura por analizar, en esa nueva ventana se deberán seleccionar las condiciones de carga para las que se requieren los resultados seleccionados, la condición o combinación de carga seleccionada se muestra con fondo oscuro, se puede seleccionar más de una combinación de carga arrastrando el ratón en el cuadro de selección.

Es necesario que se seleccione al menos una condición o combinación de carga para que los resultados se encuentren disponibles en el archivo de salida ya que de no hacerlo los resultados no se almacenarán (ver figura 4.28).

ANALISIS DE LA ESTRUCTURA

CAPÍTULO 5

5.1 ANALISIS DEL MODELO

Una vez que se han especificado las opciones de Análisis se puede solicitar la ejecución del mismo, seleccionando la opción **Run** del menú **Analyze** con lo que el análisis se efectúa y los resultados de esta fase se muestran en la ventana de la figura 5.1, en su extremo derecho se observa una barra de desplazamiento vertical que permite ver el contenido de la pantalla, al final de esta se muestra el botón **OK** haciendo clic en él se despliega en la mayoría de los casos la configuración deformada de la estructura para la primera condición de carga con lo que se está en posibilidades de tener acceso a varias opciones del menú **Display** las cuales nos mostrarán de manera gráfica y numérica algunos resultados del Análisis (desplazamientos, elementos mecánicos, etc.).

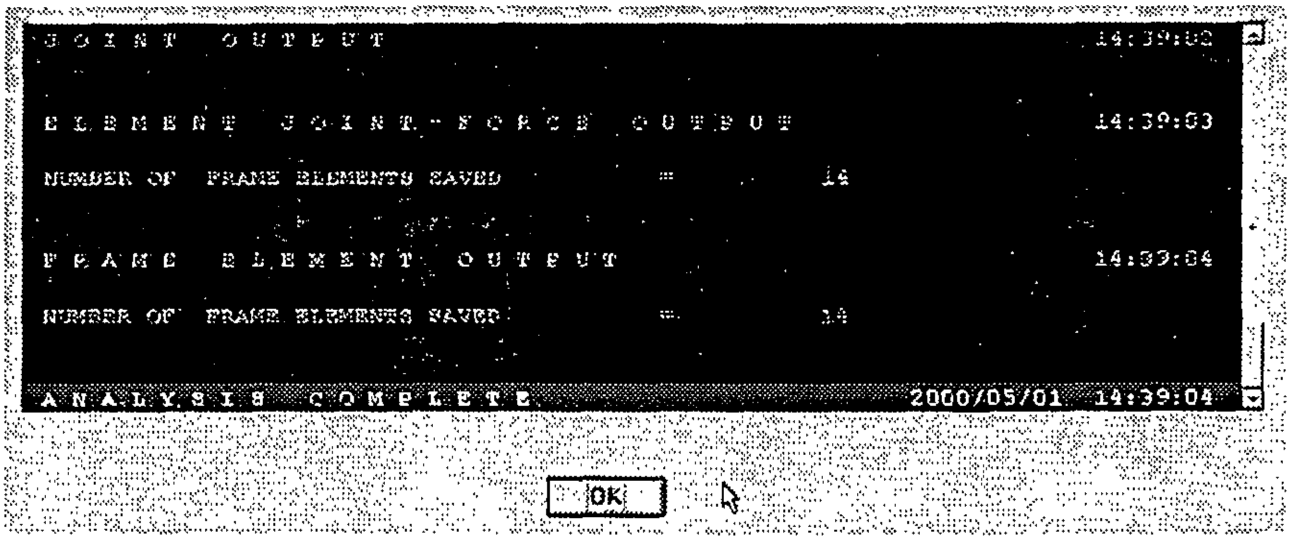


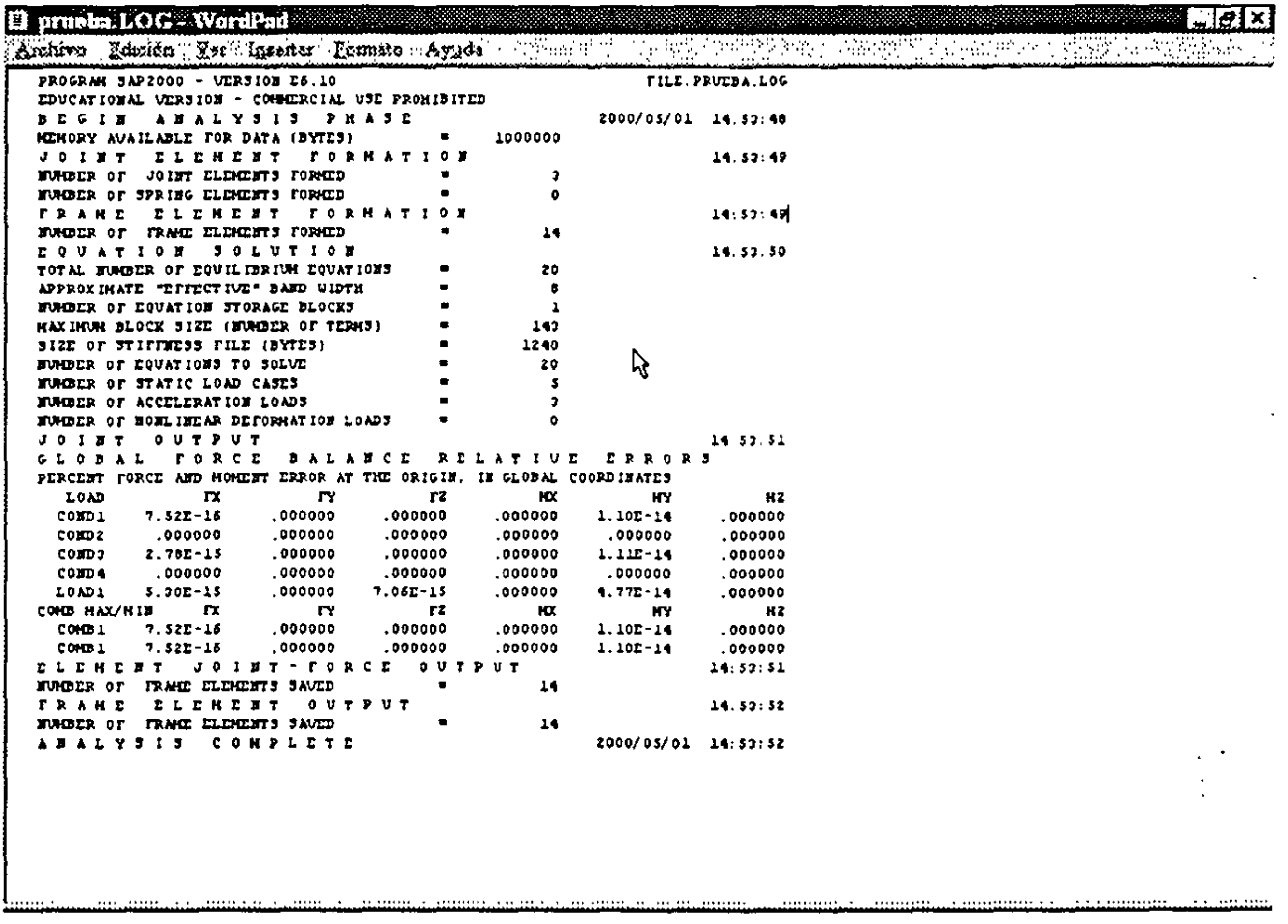
Figura 5.1 Ventana después de seleccionar la opción **Run** del menú **Analyze**

5.2 VERIFICACION DE ALGUNOS ELEMENTOS DEL PROCESO DE ANALISIS

Después de que el Análisis ha concluido se generan una serie de archivos con el mismo nombre que el archivo de datos pero con extensión diferente, algunos de los que se generan y que pueden ser de utilidad son:

El archivo nombre.LOG (ver figura 5.2), el cual contiene información de la fase de Análisi (memoria disponible, número de ecuaciones de equilibrio formadas, características de la matriz de rigideces, balance de errores relativos y diversa información de salida).

El archivo nombre.$2k contiene los datos de la estructura a analizar como son: geometría, materiales, secciones, cargas, combinaciones, etc., tal y como se generaron por ejemplo mediante el editor gráfico del programa **SAP2000**, se puede recurrir a este archivo en el caso de que el archivo nombre.SDB sufra algún cambio que lo imposibilite para ser procesado por **SAP2000**.



```
prueba.LOG - WordPad
Archivo  Edición  Ver  Insertar  Formato  Ayuda

PROGRAM SAP2000 - VERSION E6.10                                          FILE.PRUEBA.LOG
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED
B E G I N   A N A L Y S I S   P H A S E                       2000/05/01  14.53:48
MEMORY AVAILABLE FOR DATA (BYTES)               =     1000000
J O I N T   E L E M E N T   F O R M A T I O N                             14.53:49
NUMBER OF  JOINT ELEMENTS FORMED                =           3
NUMBER OF SPRING ELEMENTS FORMED                =           0
F R A M E   E L E M E N T   F O R M A T I O N                             14:53:49
NUMBER OF  FRAME ELEMENTS FORMED                =          14
E Q U A T I O N   S O L U T I O N                                         14.53.50
TOTAL NUMBER OF EQUILIBRIUM EQUATIONS           =          20
APPROXIMATE "EFFECTIVE" BAND WIDTH              =           6
NUMBER OF EQUATION STORAGE BLOCKS               =           1
MAXIMUM BLOCK SIZE (NUMBER OF TERMS)            =         143
SIZE OF STIFFNESS FILE (BYTES)                  =        1240
NUMBER OF EQUATIONS TO SOLVE                    =          20
NUMBER OF STATIC LOAD CASES                     =           5
NUMBER OF ACCELERATION LOADS                    =           3
NUMBER OF NONLINEAR DEFORMATION LOADS           =           0
J O I N T   O U T P U T                                                   14 53.51
G L O B A L   F O R C E   B A L A N C E   R E L A T I V E   E R R O R S
PERCENT FORCE AND MOMENT ERROR AT THE ORIGIN, IN GLOBAL COORDINATES
   LOAD          FX          FY          FZ          MX          MY          MZ
  COND1    7.52E-16     .000000     .000000     .000000    1.10E-14     .000000
  COND2     .000000     .000000     .000000     .000000     .000000     .000000
  COND3    2.78E-15     .000000     .000000     .000000    1.11E-14     .000000
  COND4     .000000     .000000     .000000     .000000     .000000     .000000
  LOAD1    5.30E-15     .000000    7.06E-15     .000000    4.77E-14     .000000
COMB MAX/MIN     FX          FY          FZ          MX          MY          MZ
  COMB1    7.52E-16     .000000     .000000     .000000    1.10E-14     .000000
  COMB1    7.52E-16     .000000     .000000     .000000    1.10E-14     .000000
E L E M E N T   J O I N T - F O R C E   O U T P U T                       14:53:51
NUMBER OF  FRAME ELEMENTS SAVED                 =          14
F R A M E   E L E M E N T   O U T P U T                                   14.53:52
NUMBER OF  FRAME ELEMENTS SAVED                 =          14
A N A L Y S I S   C O M P L E T E                             2000/05/01  14:53:52
```

Figura 5.2 Contenido típico del archivo nombre.LOG

El archivo nombre.EKO contiene una imagen o resultado del procesamiento de los datos contenidos en el archivo nombre.SDB generado mediante el editor gráfico del programa **SAP2000**, este archivo (nombre.EKO), contiene textos que indican las características de los datos procesados por ejemplo: hay un título y encabezado para las coordenadas de los nudos seguido de éstas, es decir se despliega información respectiva para cada bloque de datos así como los valores respectivos, únicamente se incluyen en este archivo los datos procesados.

En el caso de que se hayan seleccionado resultados para ser impresos éstos se encuentran en el archivo nombre.OUT.

Es conveniente verificar algunas características particulares del problema que se resolvió, por ejemplo que coincida el número total de grados de libertad que la estructura tiene con el número de ecuaciones de equilibrio que el programa formó y resolvió, también es conveniente verificar el número de elementos barra, placa, etc. que el programa procesó. Desde luego es conveniente verificar que todos los datos del problema fueron procesados por el módulo de Análisis, para todo lo anterior se recurre a revisar el contenido de los archivos mencionados en los párrafos anteriores para tener acceso al contenido de esos archivos se puede recurrir a varios programas o procesador de texto (por ejemplo Edit, Word Pad, Word, etc.)

SELECCION E INTERPRETACION DE RESULTADOS

CAPÍTULO 6

6.1 INTRODUCCION

Una vez que el Análisis se ejecuta sin que se hayan generado errores durante el mismo y después de hacer clic en el botón **OK** de la ventana que se muestra en la opción **Run** del menú **Analyze**, se pueden seleccionar varias opciones del menú **Display** que nos permitirán ver los diversos resultados tanto de manera gráfica como numérica, por ejemplo **Show Deformed Shape** nos mostrará la configuración deformada de la estructura para alguna condición de carga, **Show Element Forces/Stresses** nos mostrará el diagrama de elementos mecánicos, como puede verse en la figura 6.1, se encuentran disponibles algunas otras opciones para despliegue de resultados.

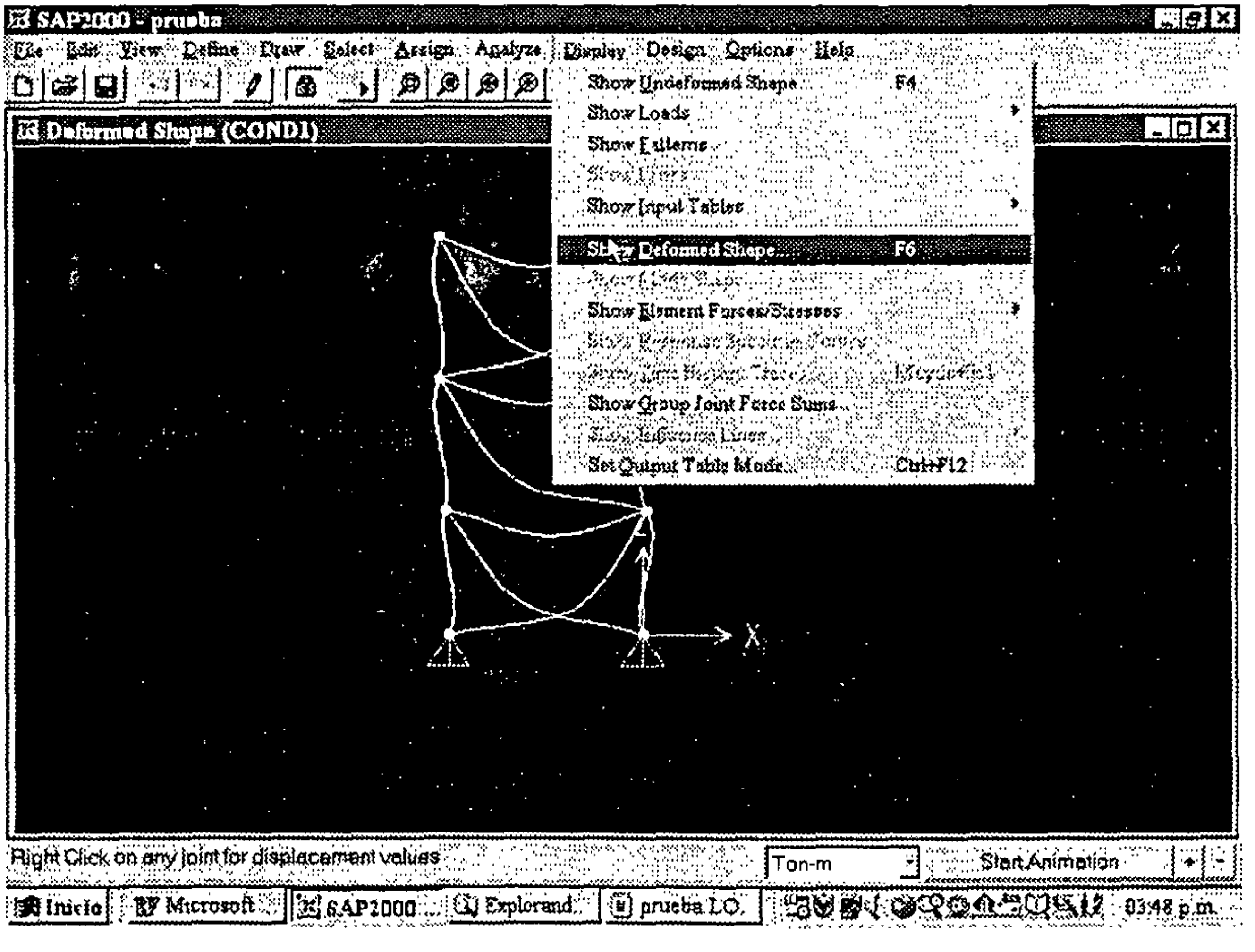


Figura 6.1 Opciones en el menú **Display**

6.2 VER LA ESTRUCTURA DEFORMADA

Para ello como se indicó en el párrafo anterior se selecciona la opción **Show Deform Shape** del menú **Display** mostrándose la ventana de la figura 6.2, en esa ventana se selecciona del marco **Load** (parte superior de la ventana) la condición de carga de la cual se quiere ver la estructura deformada; en el marco **Scaling** se presentan dos opciones para la escala que se utilizará al desplegar la configuración deformada en caso de que se seleccione **Scale Factor** se presentará el factor de escala que se utilizará para tal fin, este factor mostrado en la caja en blanco puede ser modificado por el usuario, otras dos opciones se encuentran en el extremo inferior izquierdo de esa ventana, la primera de ellas es decir **Wire Shadow** mostrará además de la configuración deformada la no deformada, la última opción que es **Cubic Curve**, en caso de estar activada mostrará la configuración deformada ajustando una curva a esa configuración, en caso contrario sólo se dibujará la configuración deformada con líneas rectas.

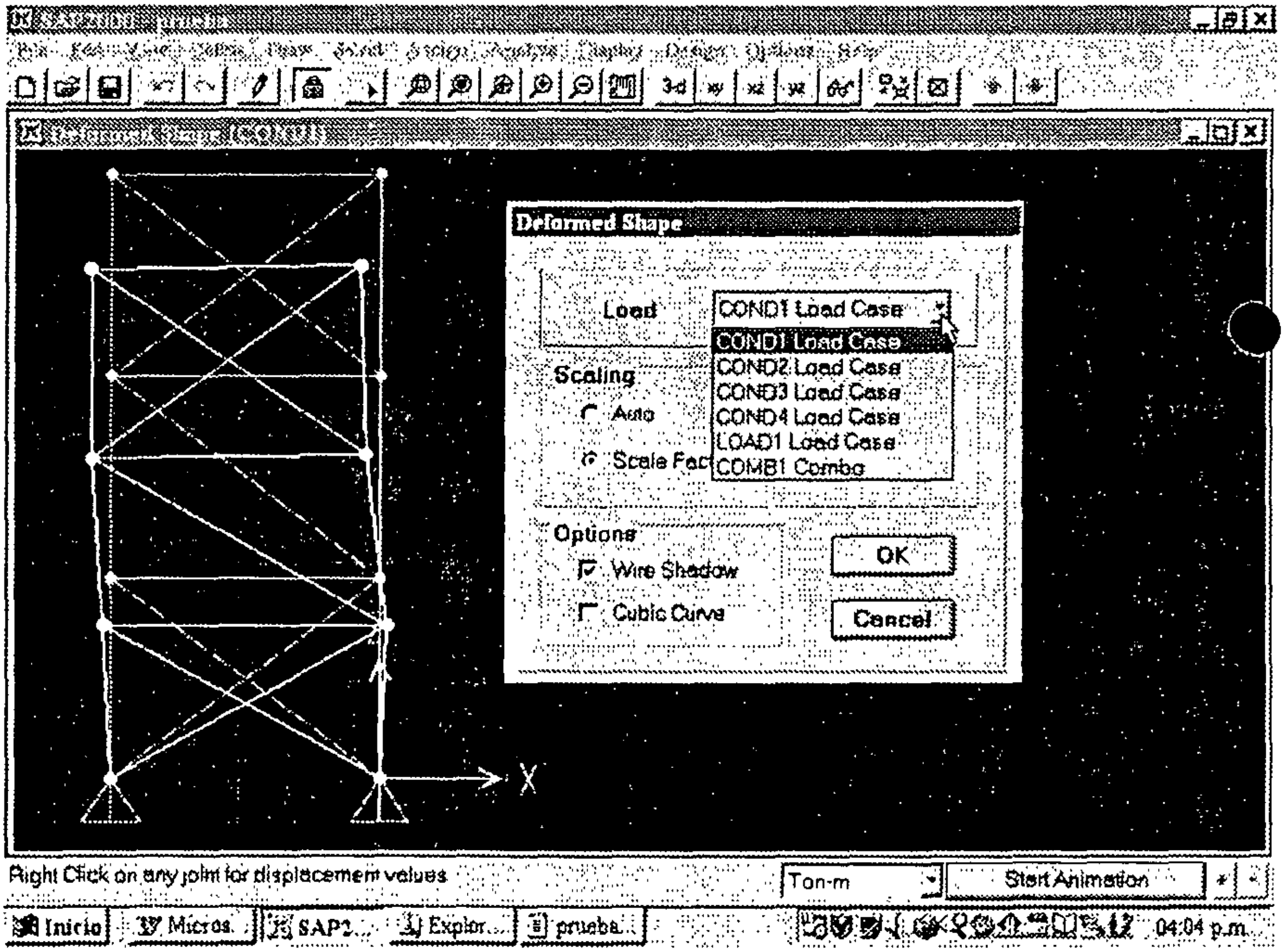


Figura 6.2 Selección de parámetros para despliegue de la configuración deformada.

Una vez mostrada la configuración deformada de la estructura se puede seleccionar algún nudo (p. ej. haciendo clic izquierdo) y después hacer clic derecho en el nudo seleccionado con lo cual se presentará una ventana conteniendo el valor de los desplazamientos de ese nudo (ver figura 6.3).

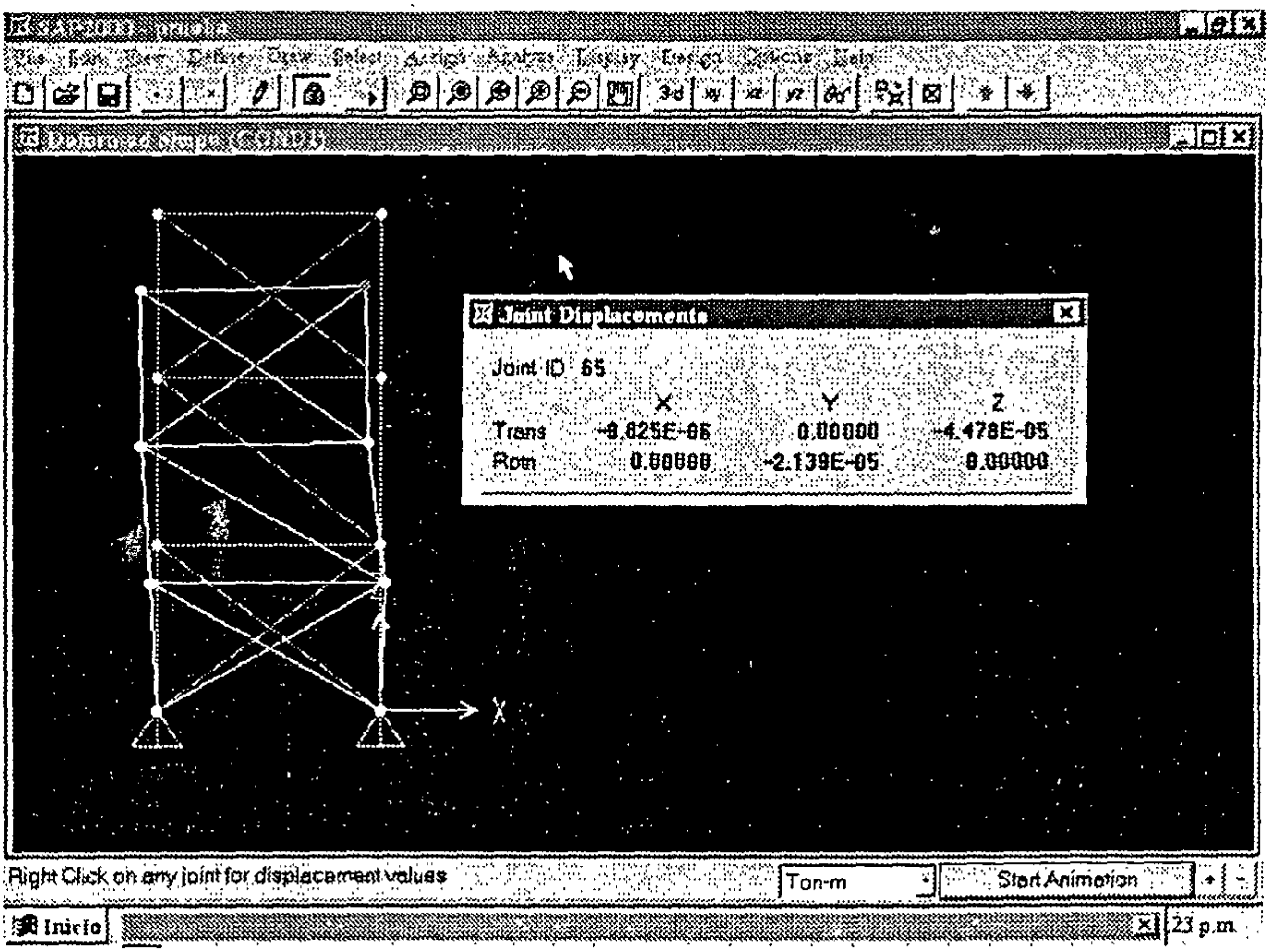


Figura 6.3 Valores del desplazamiento de un nudo seleccionado.

6.3 VER LOS DIAGRAMAS DE ELEMENTOS MECANICOS

Como se ha mencionado, **SAP2000** permite mostrar gráficamente los valores de algún elemento mecánico para determinada condición de carga, para ello se selecciona **Frames** de la opción **Show Element Forces/Stresses** en el menú **Display** presentando la ventana de la figura 6.4.

En el marco **Load** se selecciona la condición de carga y en **Component** se selecciona el tipo de elemento mecánico, las opciones en el marco **Scaling** producen el mismo efecto al caso de la configuración deformada, las opciones que se encuentran al final de la ventana nos permiten seleccionar si se desea un diagrama "lleno" y sin despliegue de valores del elemento mecánico o con valores en el diagrama.

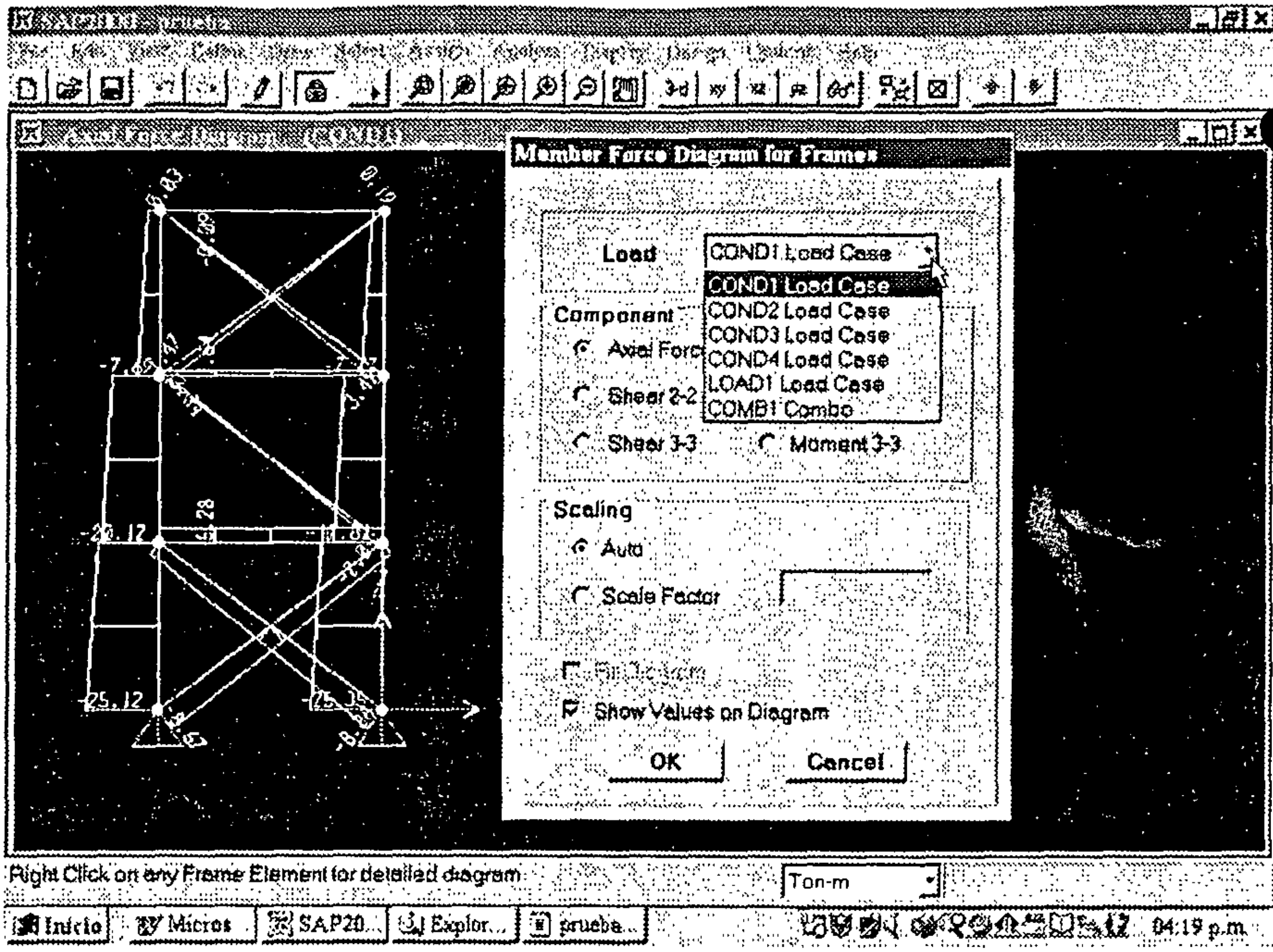


Figura 6.4 Selección de parámetros para despliegue de diagramas de elementos mecánicos.

Una vez mostrado el diagrama se puede seleccionar algún elemento barra haciendo clic sobre él y después de hacer clic derecho sobre el mismo se presenta una ventana mostrando el diagrama del elemento seleccionado, así como el valor del elemento mecánico en una sección transversal situada a la distancia que se muestra en el cuadro en blanco (ver figura 6.5), desplazando el puntero del ratón a lo largo del eje del elemento dentro de la ventana desplegada se muestra tanto la posición de la sección transversal como el valor respectivo del elemento mecánico, el contenido del cuadro puede ser modificado por el usuario desplegándose instantáneamente el valor del elemento mecánico que corresponda a la sección cuya posición se especificó en el cuadro en blanco.

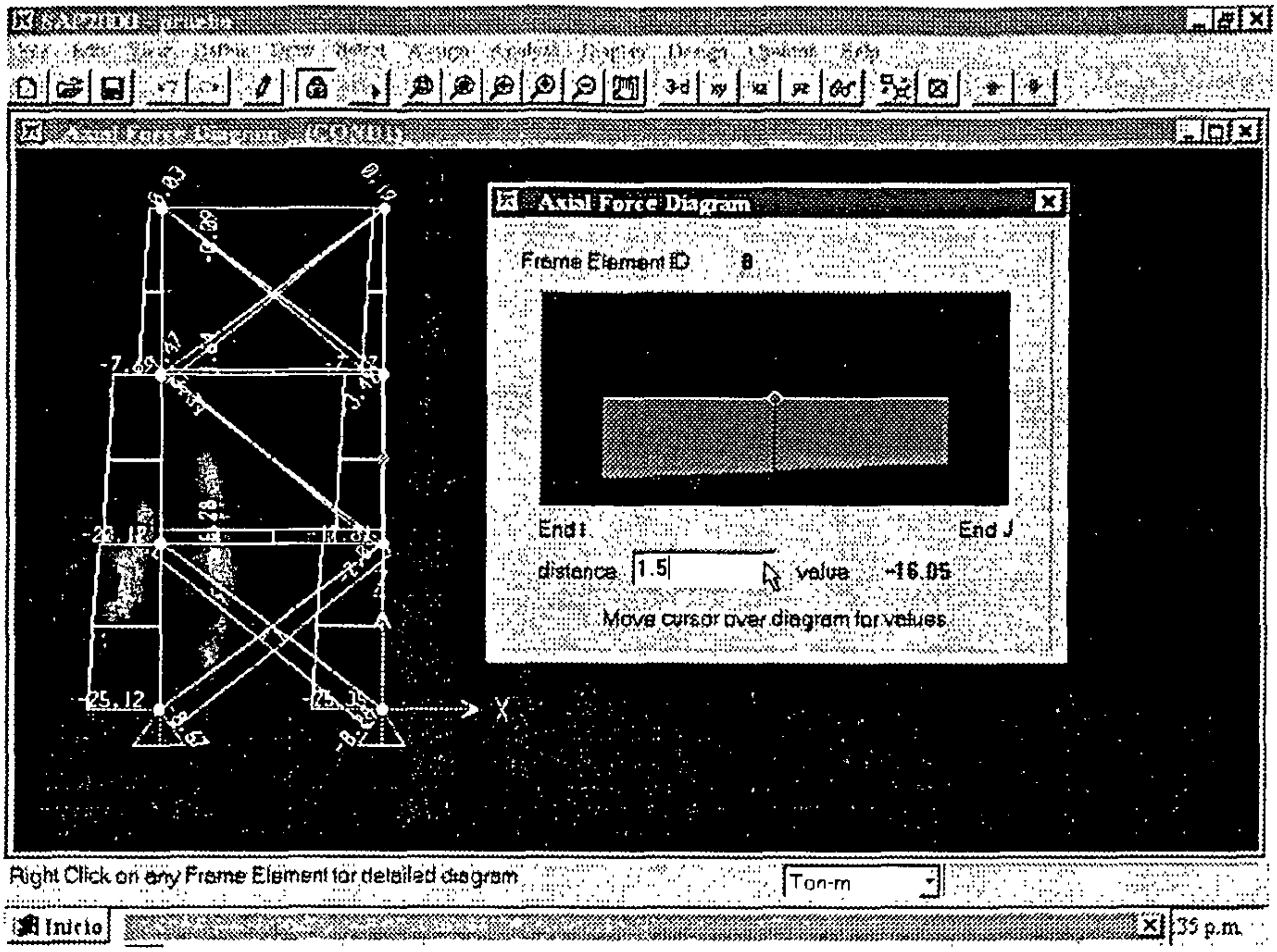


Figura 6.5 Diagrama de un elemento mecánico de una barra seleccionada.

6.4 VER LOS RESULTADOS DE DISEÑO

Algunas opciones de diseño se encuentran disponibles en el menú **Design** una vez realizado el Análisis se pueden tener acceso a ellas.

Como primer paso se seleccionará el tipo de diseño y características a utilizar, por ejemplo en el caso de diseño de concreto se tendrán que especificar algunas características de refuerzo lo cual se deberá de hacer en **Modify/Show Section** de la opción **Define Frame Sections** en el menú **Define**, seleccionando en la ventana que se despliega el botón **Reinforcement** para enseguida especificar el tipo de elemento (columna o viga), la configuración del refuerzo y las características de éste (ver figura 6.6).

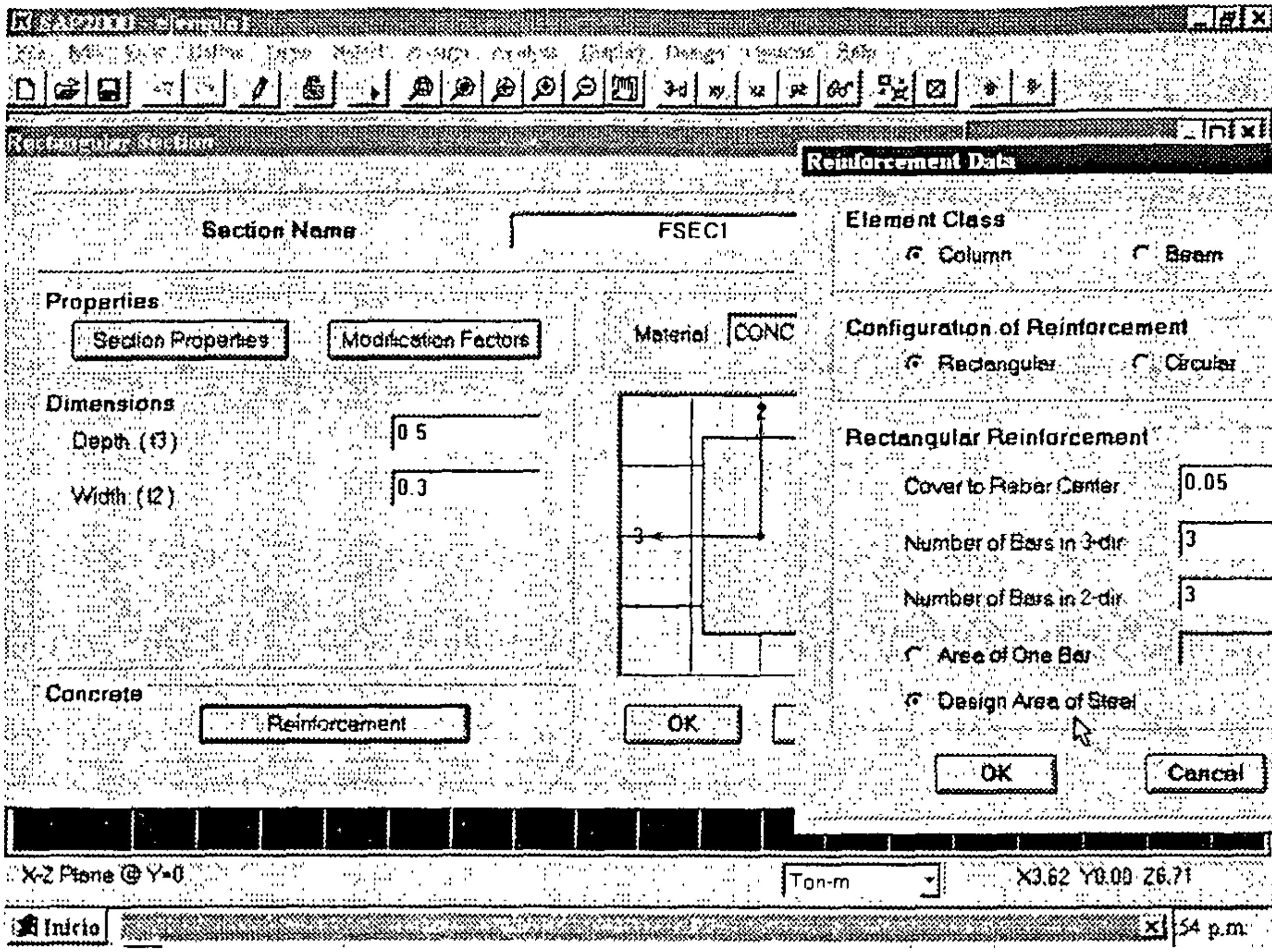


Figura 6.6 Características para diseño de un elemento.

Como segundo paso se deberá especificar las combinaciones de carga que se utilizarán para verificar el diseño activando la que se quiera para ser usada en el diseño, esto se puede hacer en **Add** o **Modify/Show Combo** en la opción **Load Combinations** del menú **Define** (ver figura 6.7).

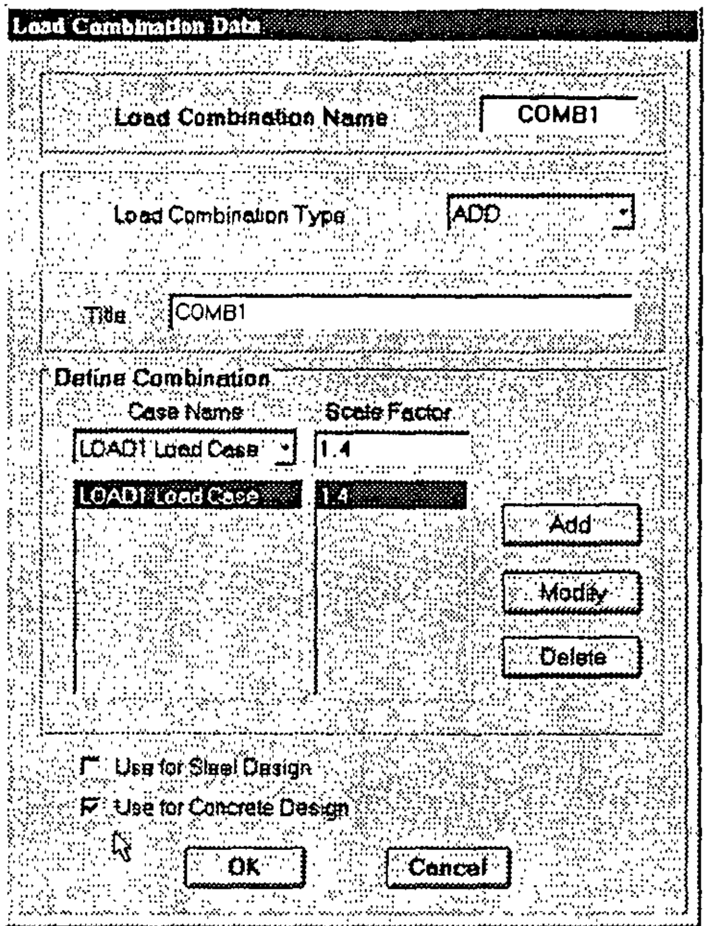


Figura 6.7 Especificación de combinaciones de carga para diseño.

Una vez realizado el Análisis, como tercer paso se seleccionarán las combinaciones de diseño para ello utiliza la opción **Select Design Combos** del menú **Design** (ver figura 6.8).

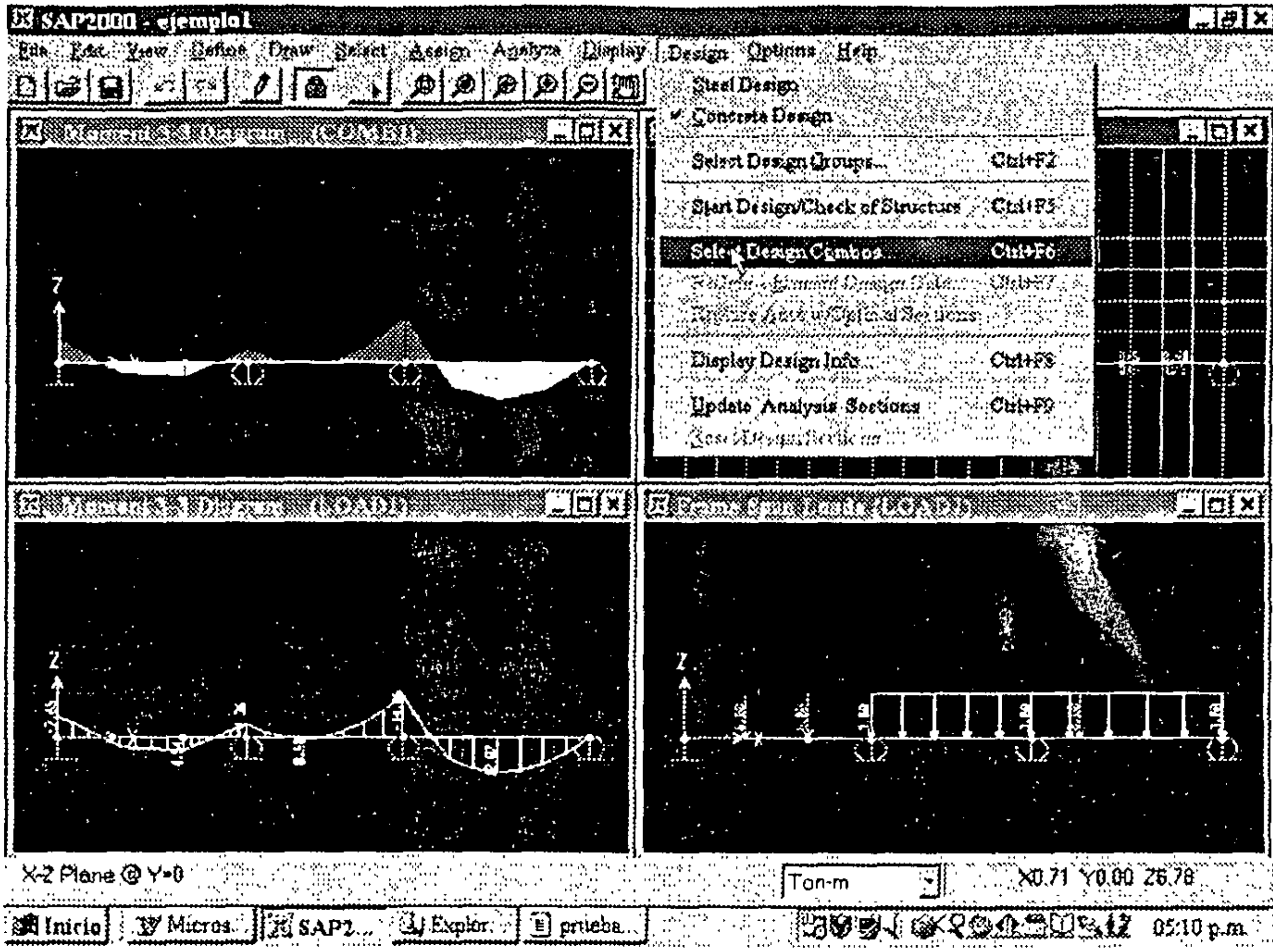


Figura 6.8 Algunas opciones del menú **Design**.

Como cuarto paso se seleccionará la opción **Start Design...** del menú **Design** (ver figura 6.8) con lo que se desplegarán algunos resultados del diseño, seleccionado una barra y después de hacer clic derecho sobre la misma se muestra una ventana similar a la de la figura 6.9.

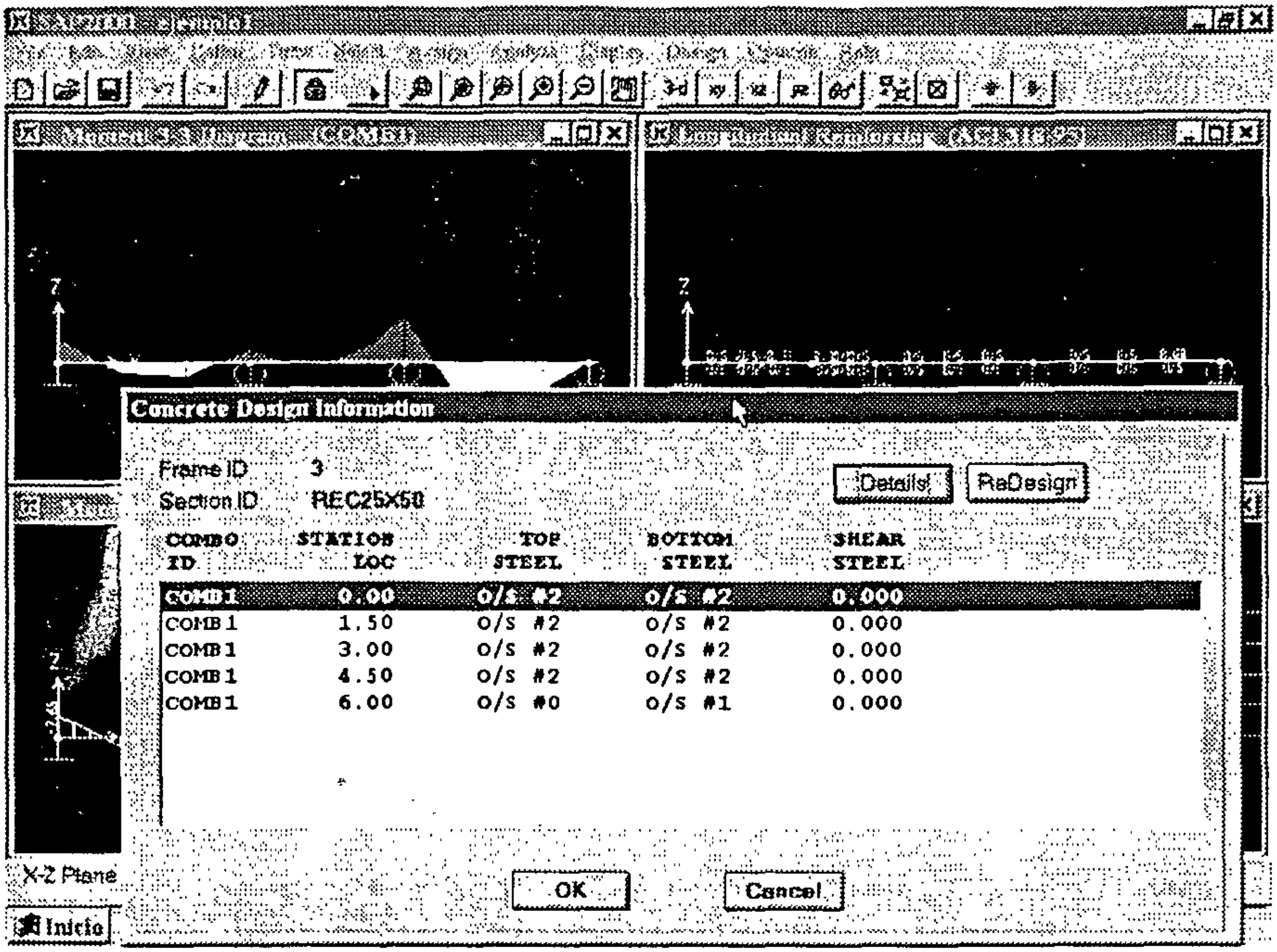


Figura 6.9 Resultados de diseño de un elemento seleccionado.

De ella se puede seleccionar el botón **Details** mostrándose información más detallada acerca de las características de diseño del elemento, se puede mostrar información diversa de la ventana arrastrando el mouse (botón izquierdo hacia alguna zona especifica de la ventana, ver figura 6.10)

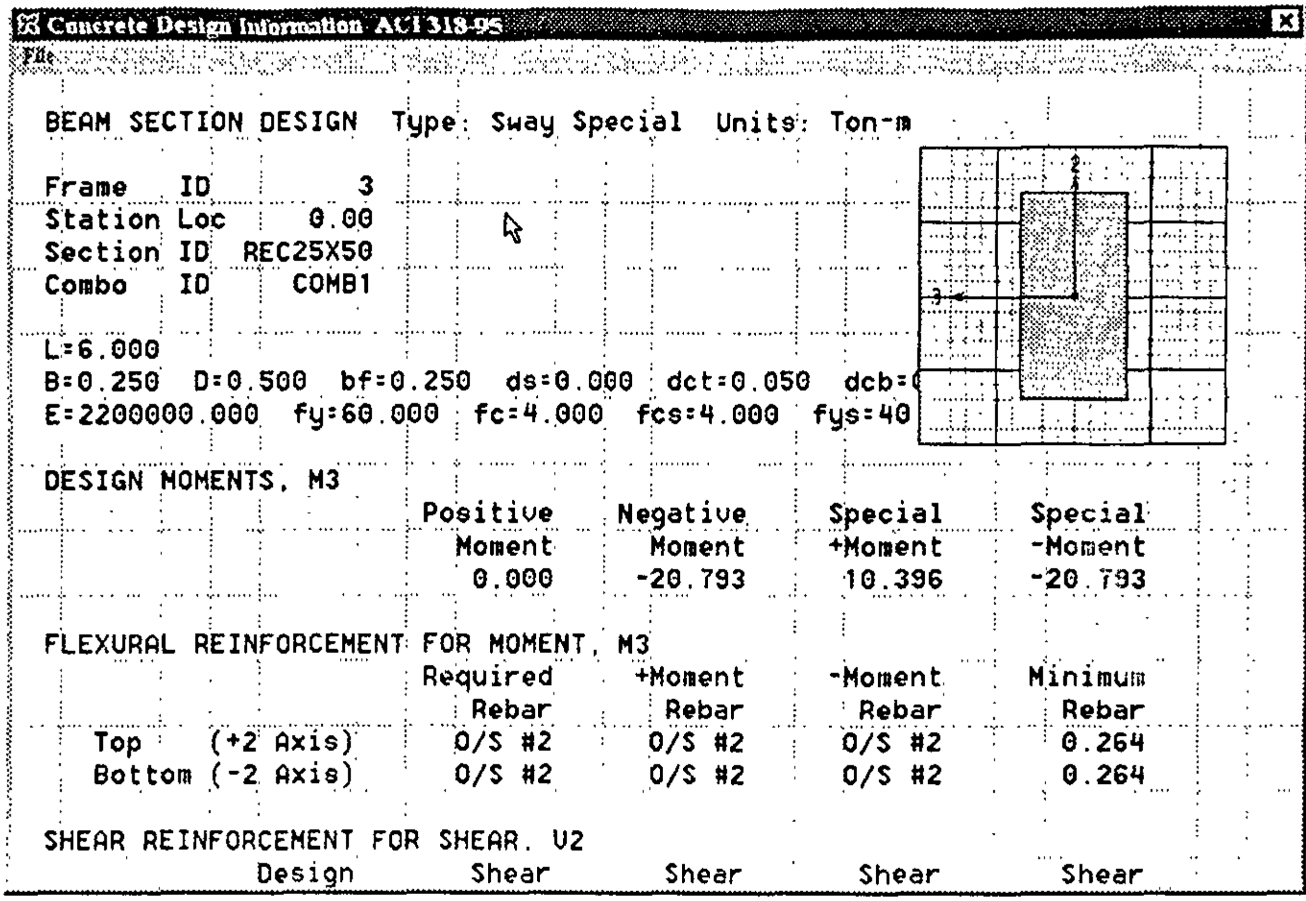


Figura 6.10 Detalle de los resultados de diseño de un elemento seleccionado.

6.4 OTRAS CARACTERISTICAS

El despliegue de reacciones puede.ser seleccionado mediante **Joints** de la opción **Show Element...** del menú **Display** mostrándose la ventana de la figura 6.11, en donde se podrá seleccionar la condición de carga, después de hacer clic en **OK** se muestran las reacciones correspondientes a la condición de carga seleccionada (ver figura 6.11).

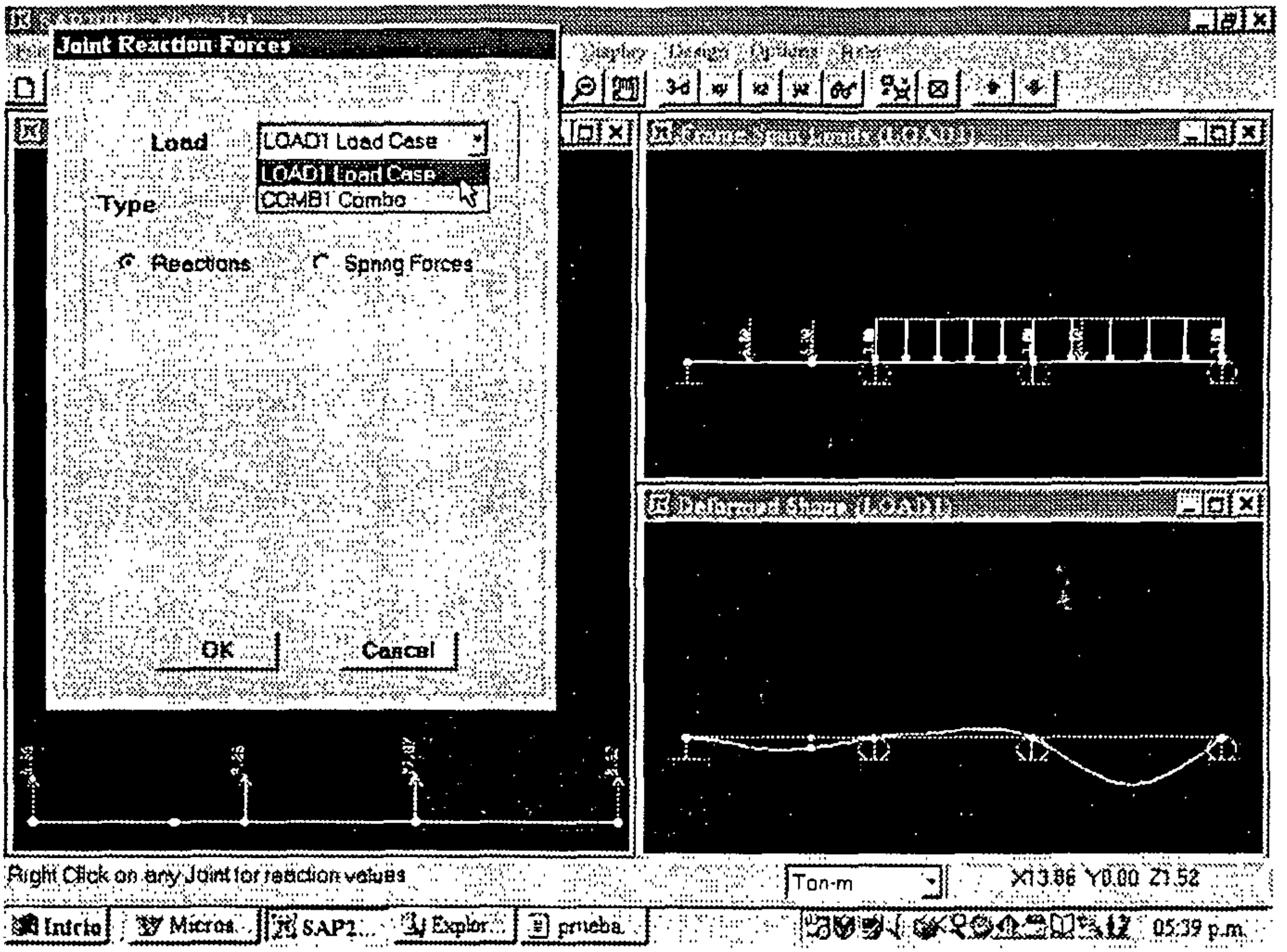


Figura 6.11 Ventana para la selección de reacciones

Están disponibles en el menú **Display** algunas otras características relacionadas con el Análisis Dinámico como el dibujo de formas modales, espectros de respuesta y análisis de la respuesta en el tiempo y otras más.

Los resultados del Análisis se pueden almacenar en un archivo a manera de tablas para ello se selecciona la opción **Set Output Table Mode** del menú **Display** mostrándose una ventana en donde se seleccionarán las condiciones de carga de los resultados que se incluirán, después de hacer clic en el botón **OK** se pasa a una ventana con de título **Analysis Output Tables** (ver figura 6.12)

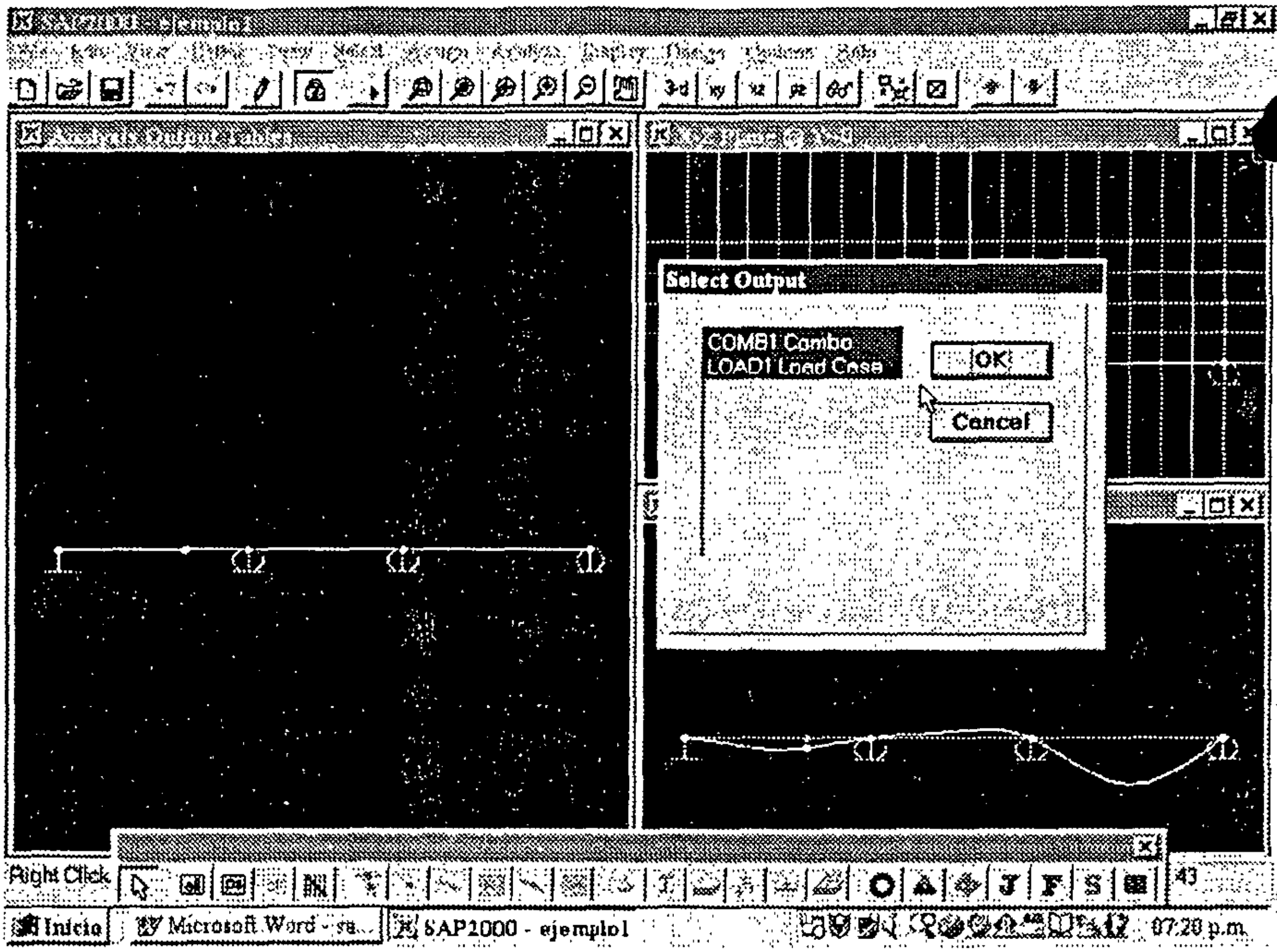


Figura 6.12 Selección de condiciones y para la generación de resultados en forma tabular.

En esa ventana se podrá seleccionar algún nudo o elemento después de hacer clic derecho en un nudo seleccionado, se desplegará una ventana conteniendo tanto los desplazamientos como las reacciones del nudo para las condiciones de carga seleccionadas (ver figura 6.13)

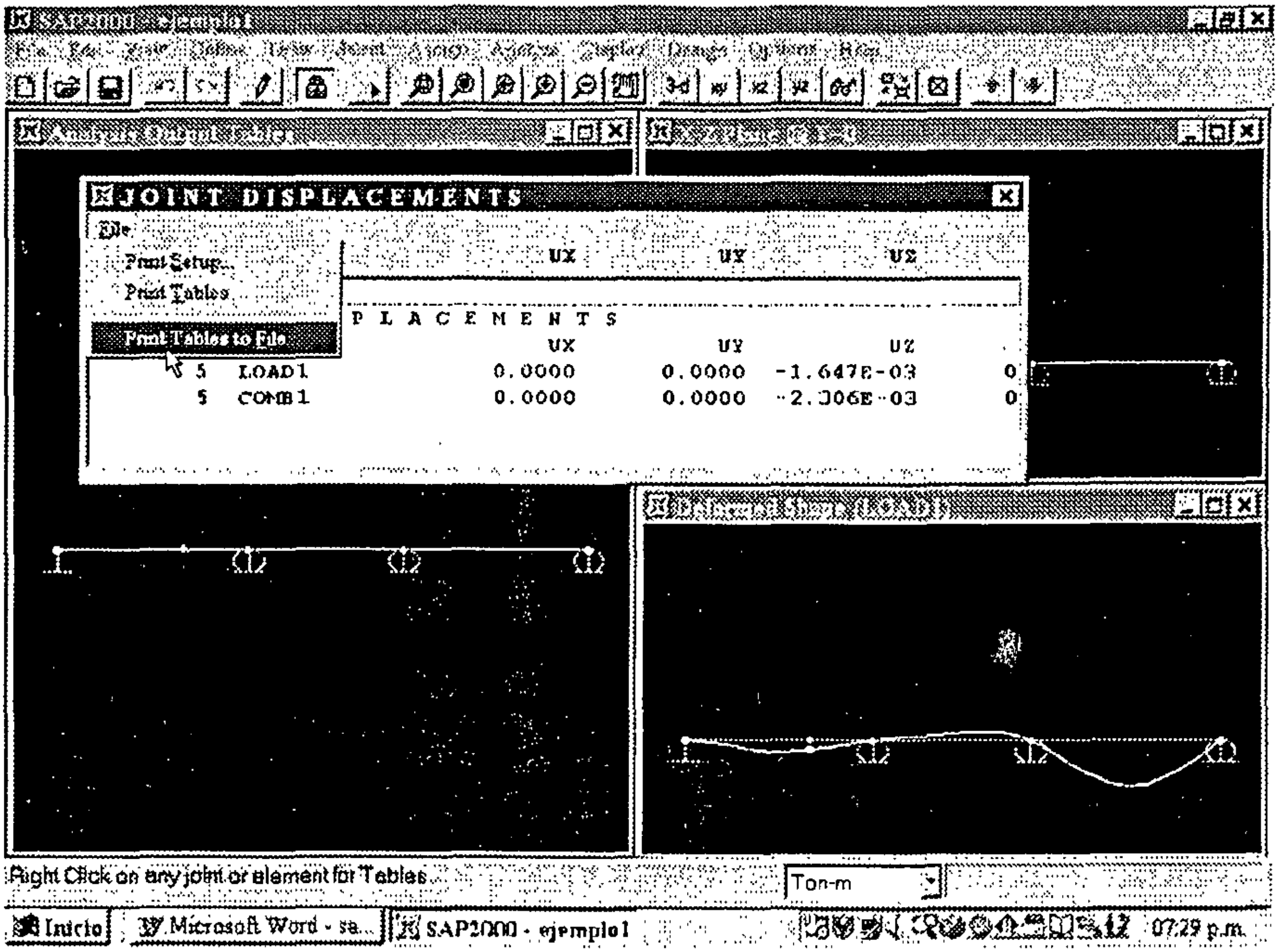


Figura 6.13 Ventana de resultados de un nudo seleccionado.

Si el elemento sobre el que se hace clic es una barra entonces la ventana que se despliega contiene los elementos mecánicos de las condiciones de carga seleccionadas (ver figura 6.14).

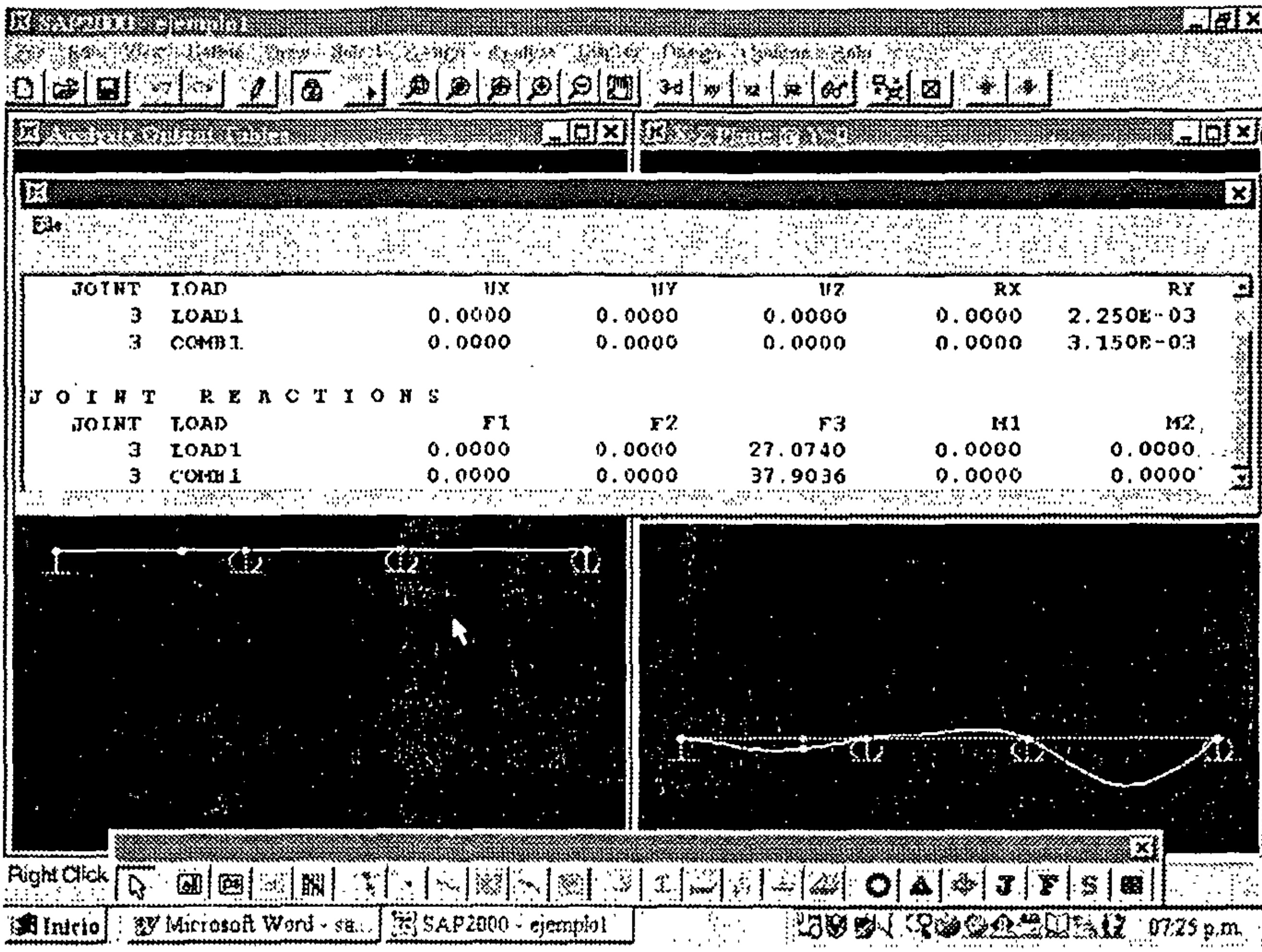


Figura 6.14 Ventana de resultados de una barra seleccionada.

Tanto en la ventana de resultados de nudos como de barras en el extremo superior izquierdo de esta se encuentra la opción **File**, que permite el almacenamiento de los resultados contenidos en la tabla mostrada en un archivo, para ello después de hace clic en **File**, habrá que proporcionar en el cuadro en blanco el nombre del archivo y hacer clic en el botón guardar (ver figura 6.15).

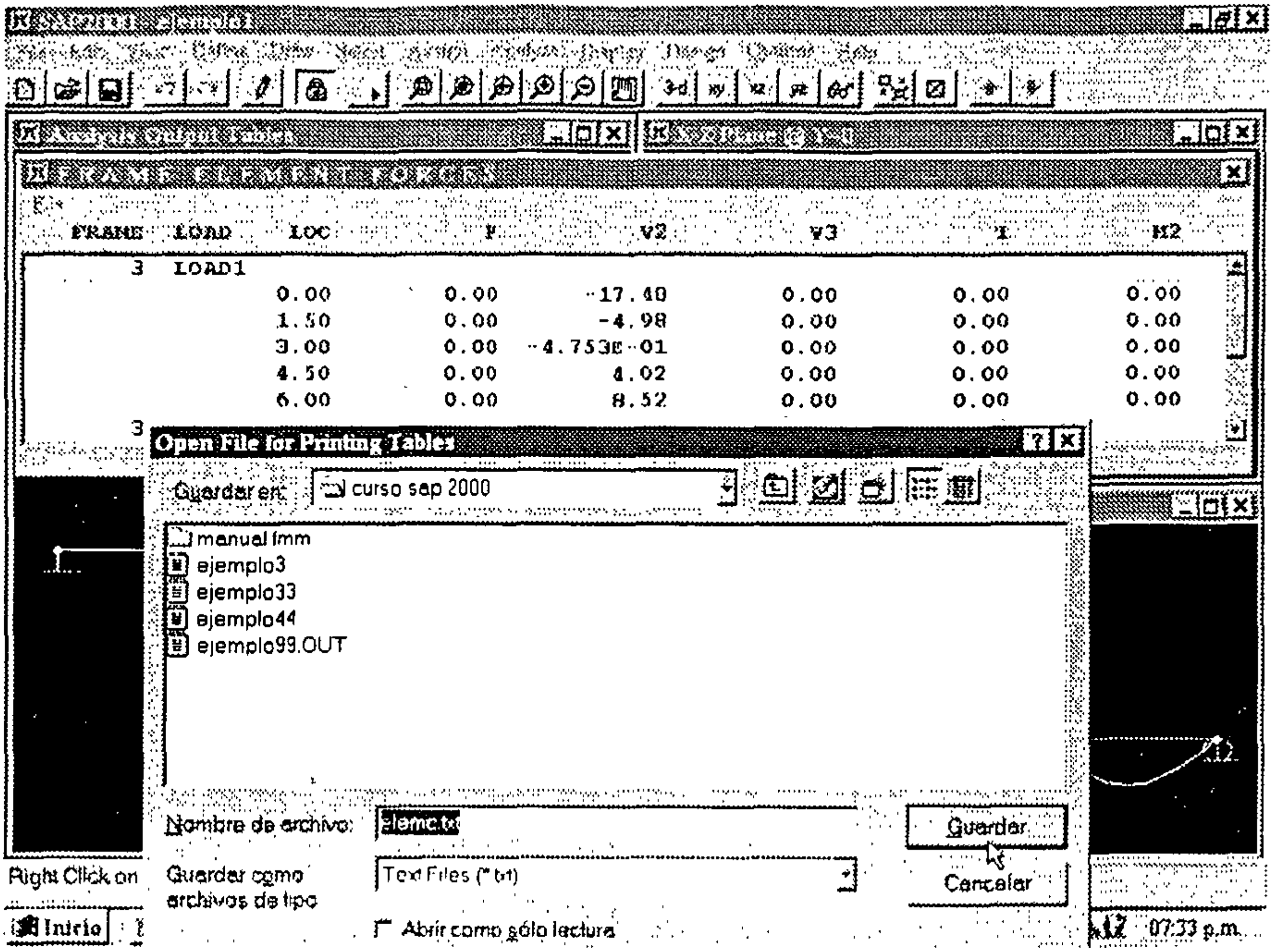


Figura 6.15 Almacenamiento de resultados en un archivo.

OPCIONES ADICIONALES

CAPÍTULO 7

7.1 INTRODUCCION

SAP2000 posee varias características, con algunas de ellas se pueden modelar por ejemplo muros y losas mediante elementos **Shell**, las opciones abarcan desde la definición de materiales dibujo de elementos, definición de características geométricas del elemento (**Shell Sections**) así como la asignación de las características anteriores además de las fuerzas (uniformes, presión, etc.) a este tipo de elementos. La estructura y secuencia es muy parecida a la utilizada para los elementos barra, se recomienda consultar la ayuda en línea, los temas relacionados en los manuales o bien ver los ejemplos en la carpeta de ejemplos o en el disco.

En cuanto al elemento finito sólido, este no se puede generar utilizando el editor gráfico de la versión estudiantil (versión 6.1 ó 6.13), por lo que su definición y demás características se tendrán que realizar mediante una serie de instrucciones que se adicionarán al archivo de datos mediante un editor, la misma recomendación hecha en el párrafo anterior es aplicable a este caso.

Una gran variedad de opciones para Análisis Dinámico está incluida en el programa **SAP2000**, para usar alguna de ellas se recomienda consultar los ejemplos que acompañan al presente instructivo o que se encuentran en el diskete, o bien los que se encuentran en el manual respectivo.

También existe la posibilidad de Análisis de estructuras de puentes obteniendo mediante el programa por ejemplo: líneas de influencia, envolventes de elementos mecánicos, etc., lo anterior para varias condiciones de carga incluyendo uno o varios carriles con cargas vehiculares tipo o definidas por el usuario, la recomendación del párrafo anterior es igualmente aplicable.

Se recomienda consultar al autor ya que se encuentra en proceso un instructivo similar al presente para los fines mencionados en los párrafos anteriores.

7.2 VER EL ARCHIVO DE ENTRADA

Durante una sesión con el programa **SAP2000** las opciones **Save** y **Save as** del menú **File** permiten almacenar en un archivo con extensión **SDB** los datos de la estructura que se han introducido, al archivo así creado sólo se podrá acceder (para fines de este programa) mediante la opción **Open** del mismo menú, sin embargo los datos pueden ser almacenados en un archivo que pueda modificarse y ser reconocido por el programa **SAP2000** para ello se selecciona **SAP2000.S2K** de la opción **Export** en el menú **File** (ver figura 7.1).

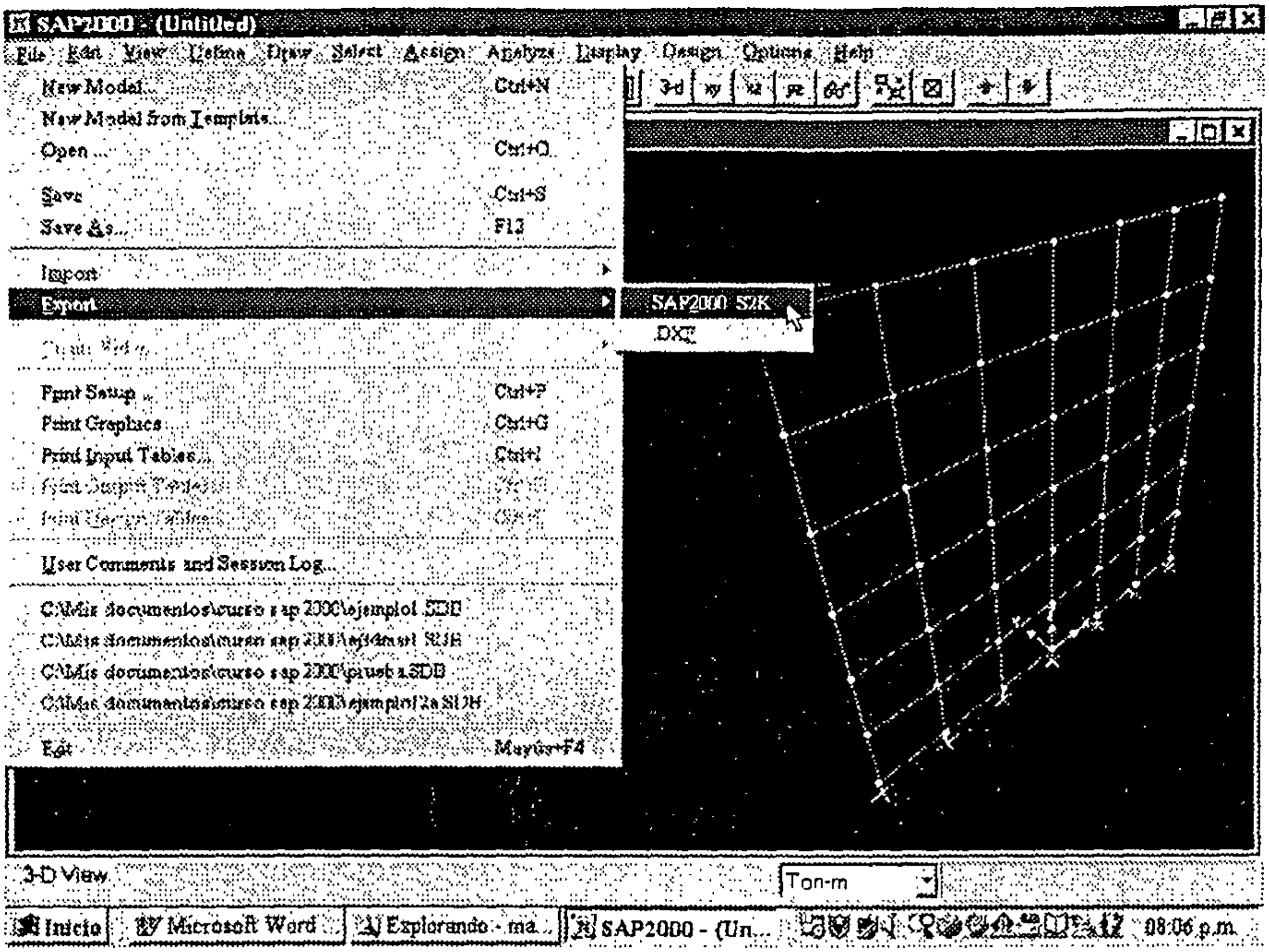


Figura 7.1 Almacenamiento de datos para poder realizar cambios al archivo.

El archivo extensión S2K puede ser modificado con la ayuda de algún editor (por ejemplo Edit, WordPad, etc.), el archivo resultante de la modificación deberá ser almacenado sin ningún caracter de control especial que se almacene en el mismo archivo, y con la misma extensión, si se usan algunos editores o procesadores de palabras se deberá tener especial cuidado de lo anterior, en caso de uso de esos procesadores se recomienda hacer varias copias de respaldo con objeto de no perder los cambios efectuados.

Una vez realizados los cambios, el contenido del archivo extensión S2K modificado podrá ser procesado por **SAP2000**, para ello se selecciona **SAP2000.S2K** de la opción **Import** en el menú **File**, para ambas opciones (**Export** e **Import**) será necesario proporcionar el nombre del archivo en el cuadro en blanco correspondiente de la ventana como la que se muestra en la figura 7.2.

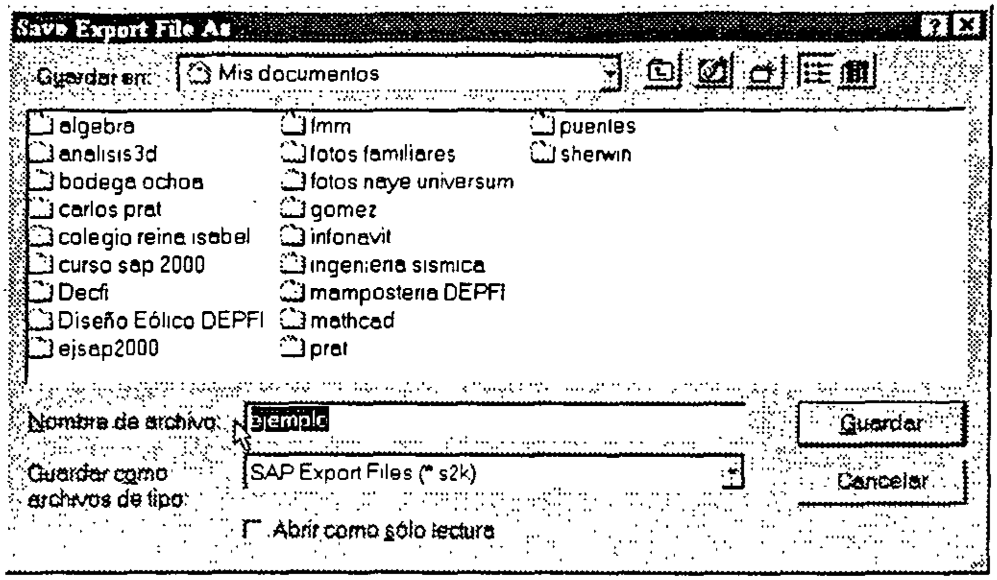


Figura 7.2 Ventana en la opción **Export** SAP2000.S2K.

7.3 VER EL ARCHIVO DE SALIDA

El contenido del archivo de resultados nombre.OUT indicado mediante **Generate Output** en la opción **Analysis Options** del menú **Analyze** se genera después de ejecutar la opción **Run** del menú del mismo nombre, el archivo así generado puede ser consultado mediante cualquier editor o procesador de palabras e inclusive por algunas hojas de cálculo, para ello se seleccionará la opción abrir (**Load** u **Open**) del programa que se vaya a utilizar con ese fin y especificar el nombre del archivo desde luego con extensión OUT (ver figura 7.3).

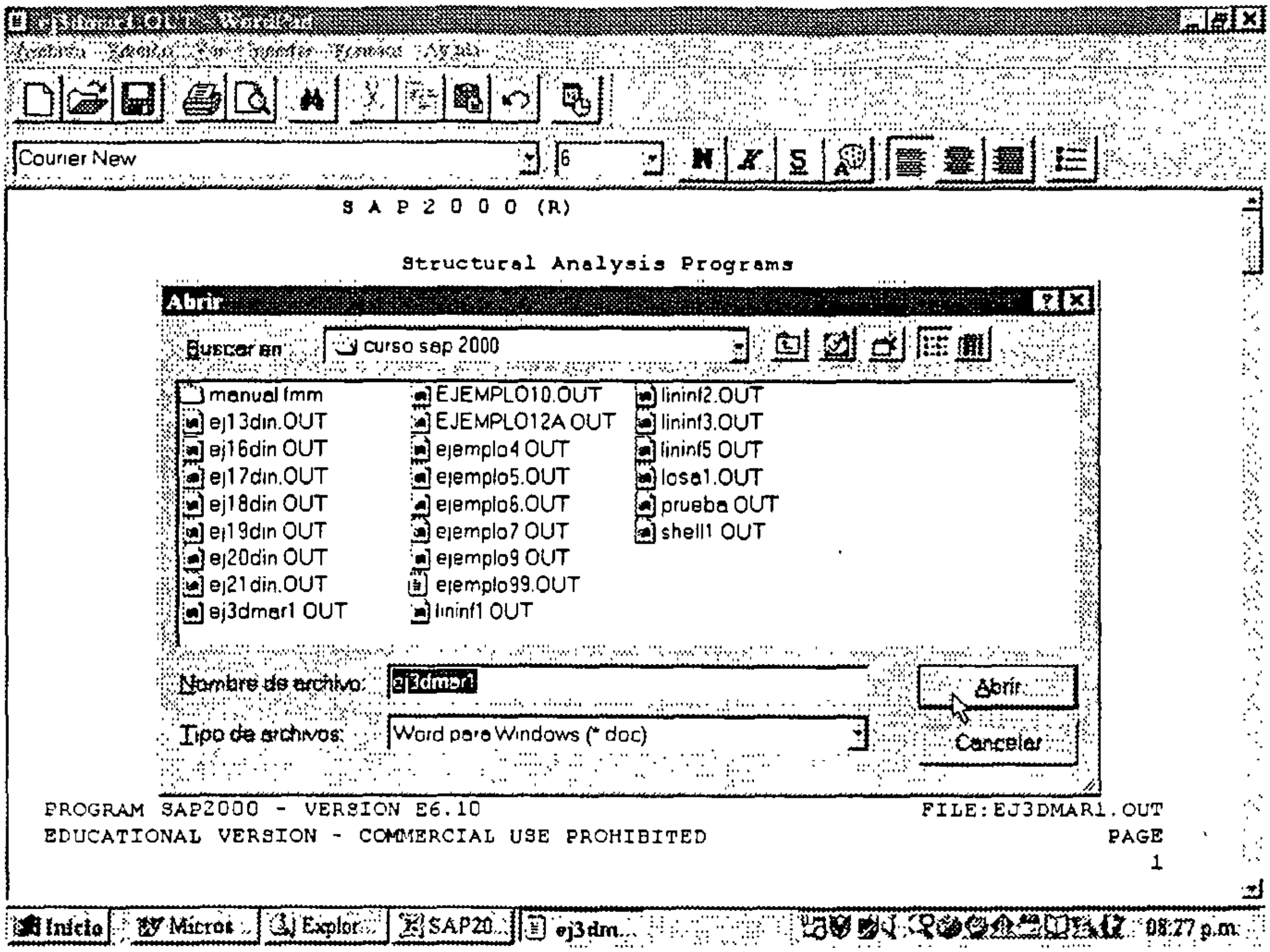


Figura 7.3 Acceso al archivo nombre.OUT mediante el programa **WordPad**.

7.4 RELACION CON AUTOCAD

La geometría de la estructura puede ser generada por AutoCAD realizando el dibujo de las barras (lineas) en una capa (**Layer**) de nombre Sap_frames (ver figura 7.4).

La geometría así generada se deberá exportar a un archivo extensión **dxf** como se muestra en la figura 7.5.

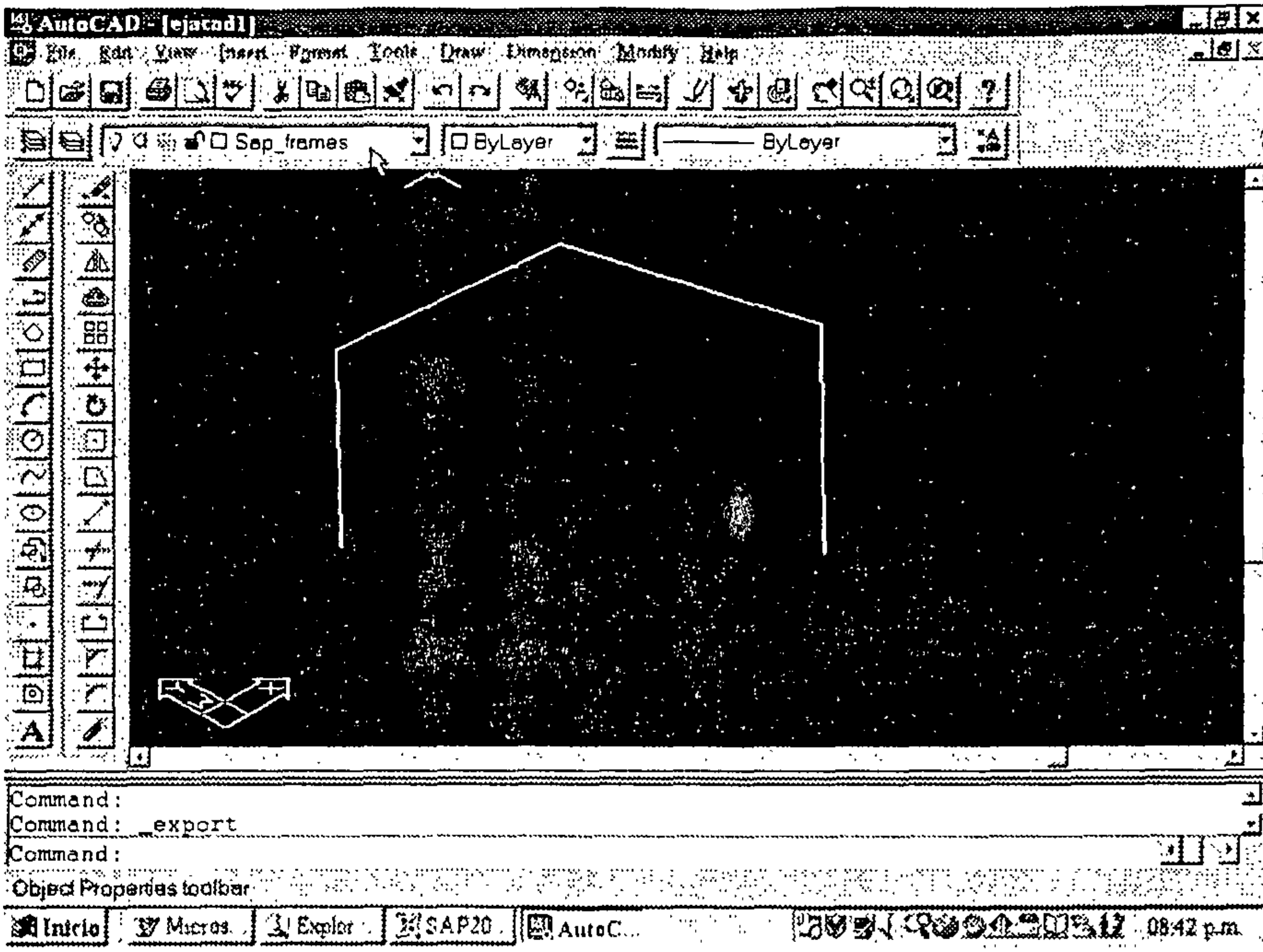


Figura 7.4 Geometria generada en AutoCAD.

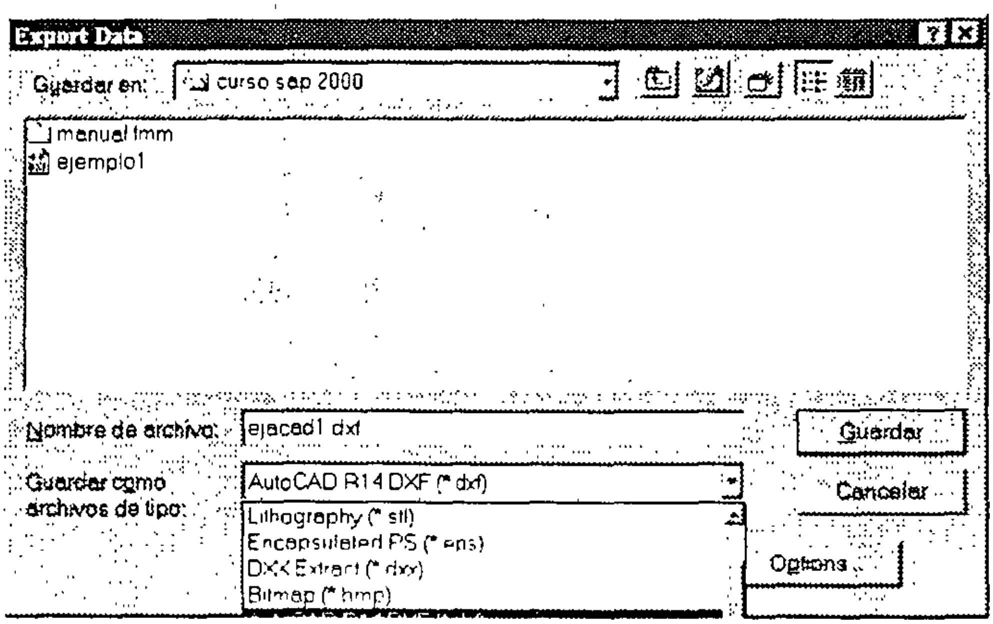


Figura 7.5 Exportando la geometria generada con AutoCAD a un archivo extensión **dxf**.

Para recuperar la información de un archivo **dxf**, se selecciona **.dxf** de la opción **Import** en el menú **File** (ver figura 7 6).

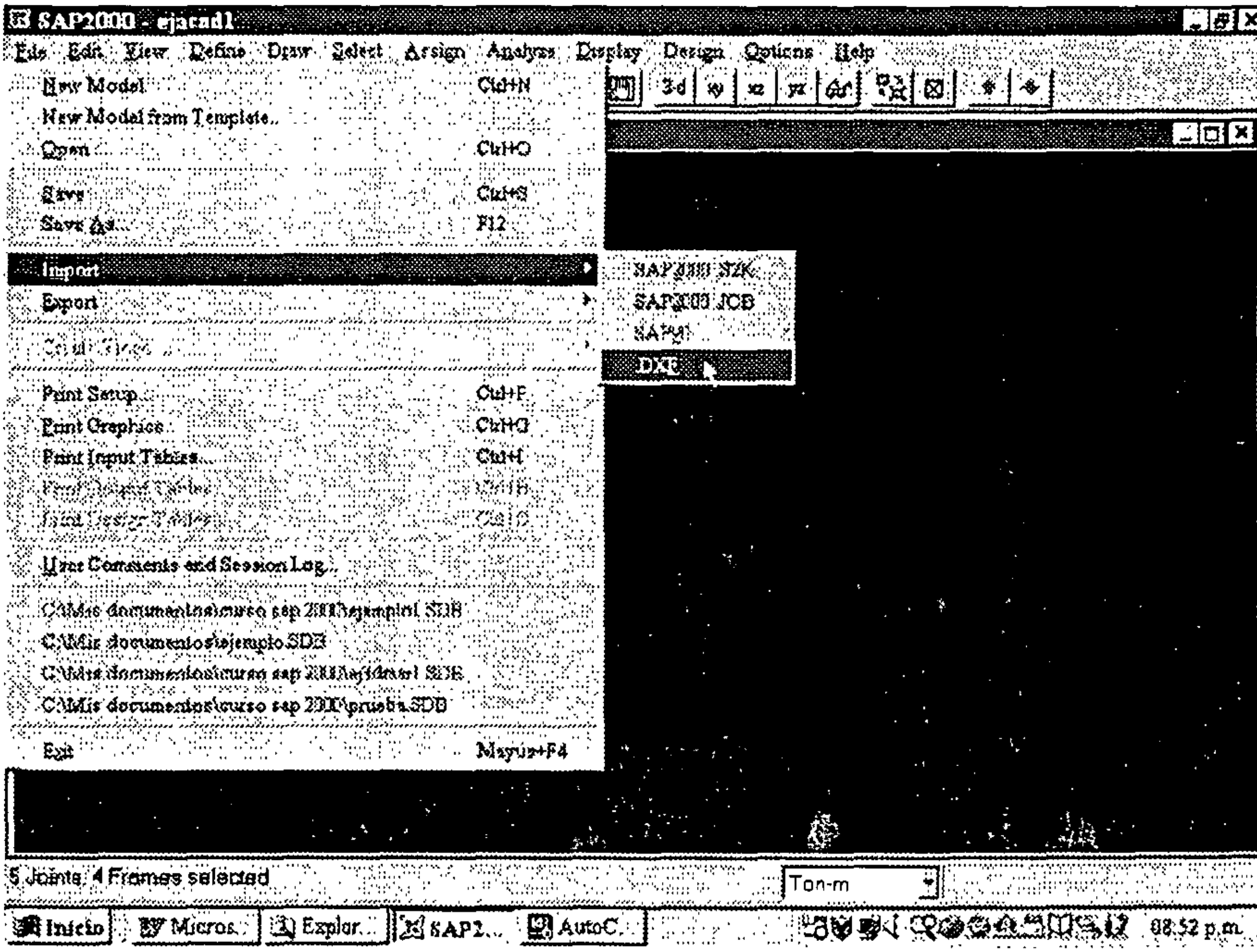


Figura 7.6 Importando datos de un archivo **.dxf**.

Desplegándose enseguida la ventana que se muestra en la figura 7.7, en donde se deberá especificar el nombre del archivo cuya extensión es **dxf** después de hacer clic en abrir se seleccionan de la ventana que se muestra en la figura 7.8 la dirección global y las unidades.

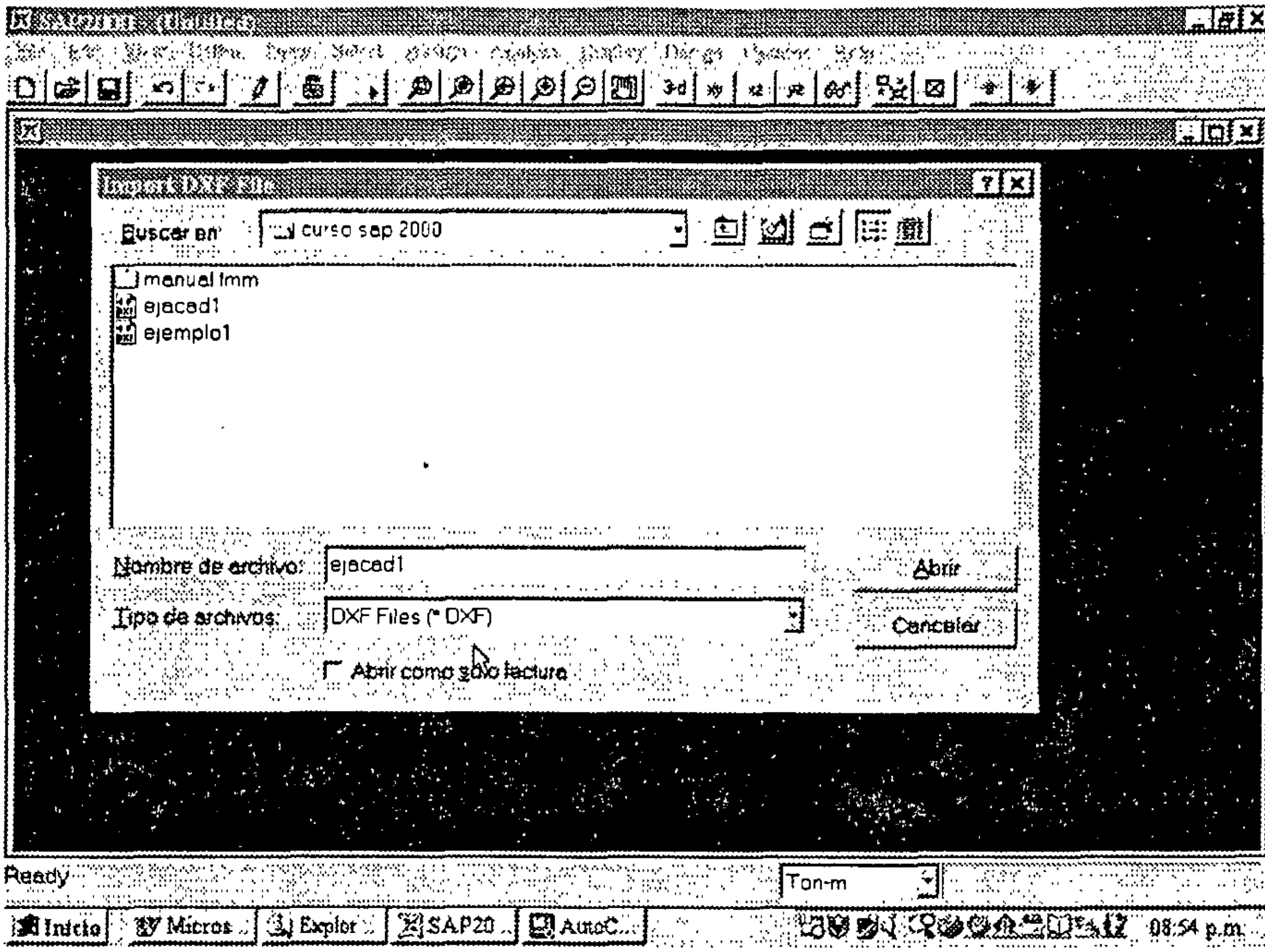


Figura 7.7 Ventana para importar datos de un archivo **dxf**.

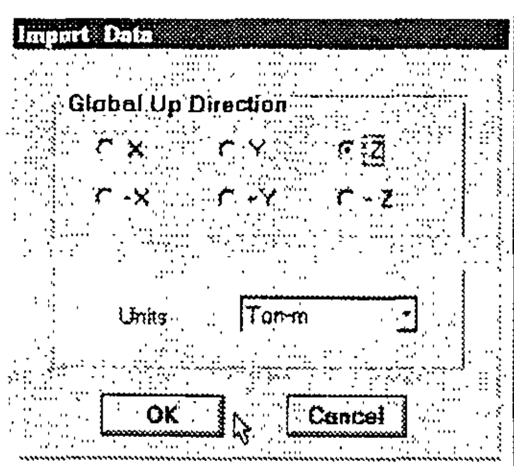


Figura 7.8 Indicando características de los datos a importar de un archivo **dxf**

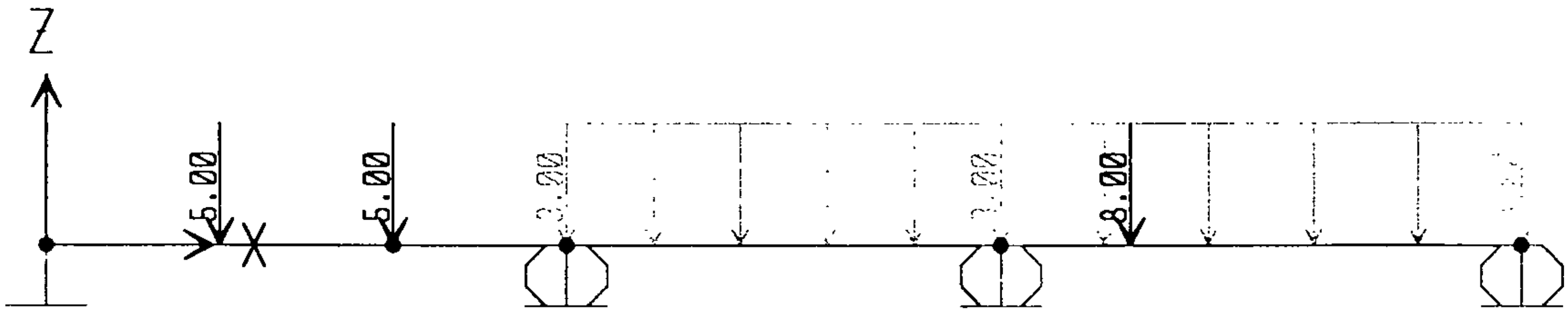
EJEMPLOS E INTERPRETACION DE RESULTADOS

CAPÍTULO 8

8.1 INTRODUCCION

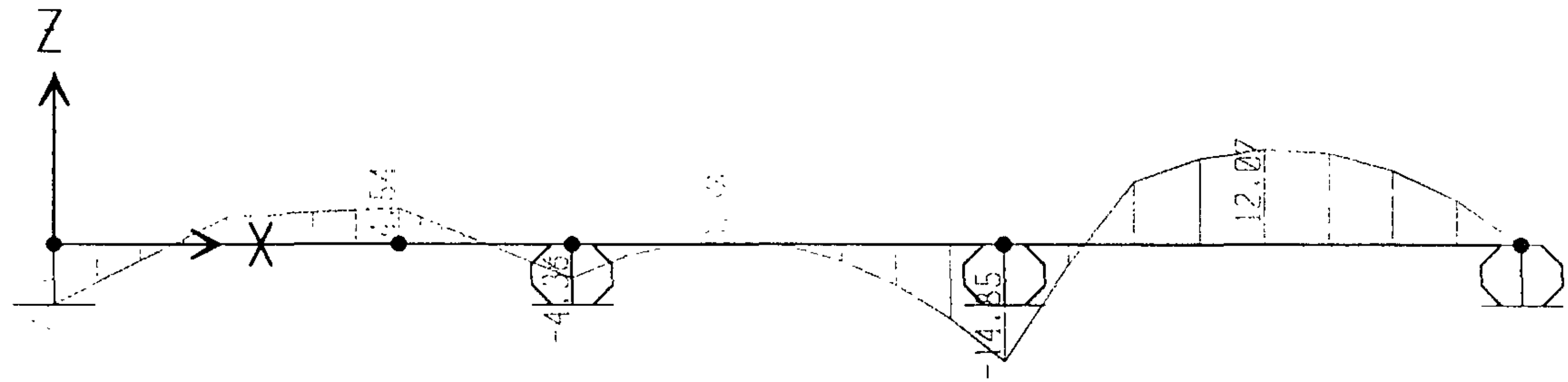
Durante la impartición del curso para uso y manejo del programa **SAP2000** tanto en la División de Ingenieria Civil, Topográfica y Geodésica como en la División de Educación Continúa de la Facultad de Ingenieria de la UNAM, se han desarrollado varios ejemplos típicos para el análisis de formas estructurales comunes (vigas continuas, marcos, armaduras, etc.) permitiendo al asistente practicar el uso de los comandos básicos tratados en los capítulos anteriores así como de algunos otros que no se han descrito o mencionado en este instructivo, por lo que sería conveniente que el lector interesado tuviera la oportunidad de asistir a alguno de esos cursos con objeto de despejar algunas dudas, desarrollar una mejor habilidad en el manejo del programa y adquirir una mejor comprensión de algunas de las opciones de Análisis así como de sus ventajas y limitaciones

A continuación se presentan los listados (datos, resultados numéricos y gráficas) de algunos de los ejemplos que se han desarrollado durante los cursos que ha impartido el autor, los datos y resultados de otros más (incluyendo los que se listan a continuación) se encuentran en el disco que acompaña al presente instructivo, se sugiere que el interesado los consulte ya sea directamente (mediante algún editor) o procese los archivos de datos a través del programa **SAP2000**

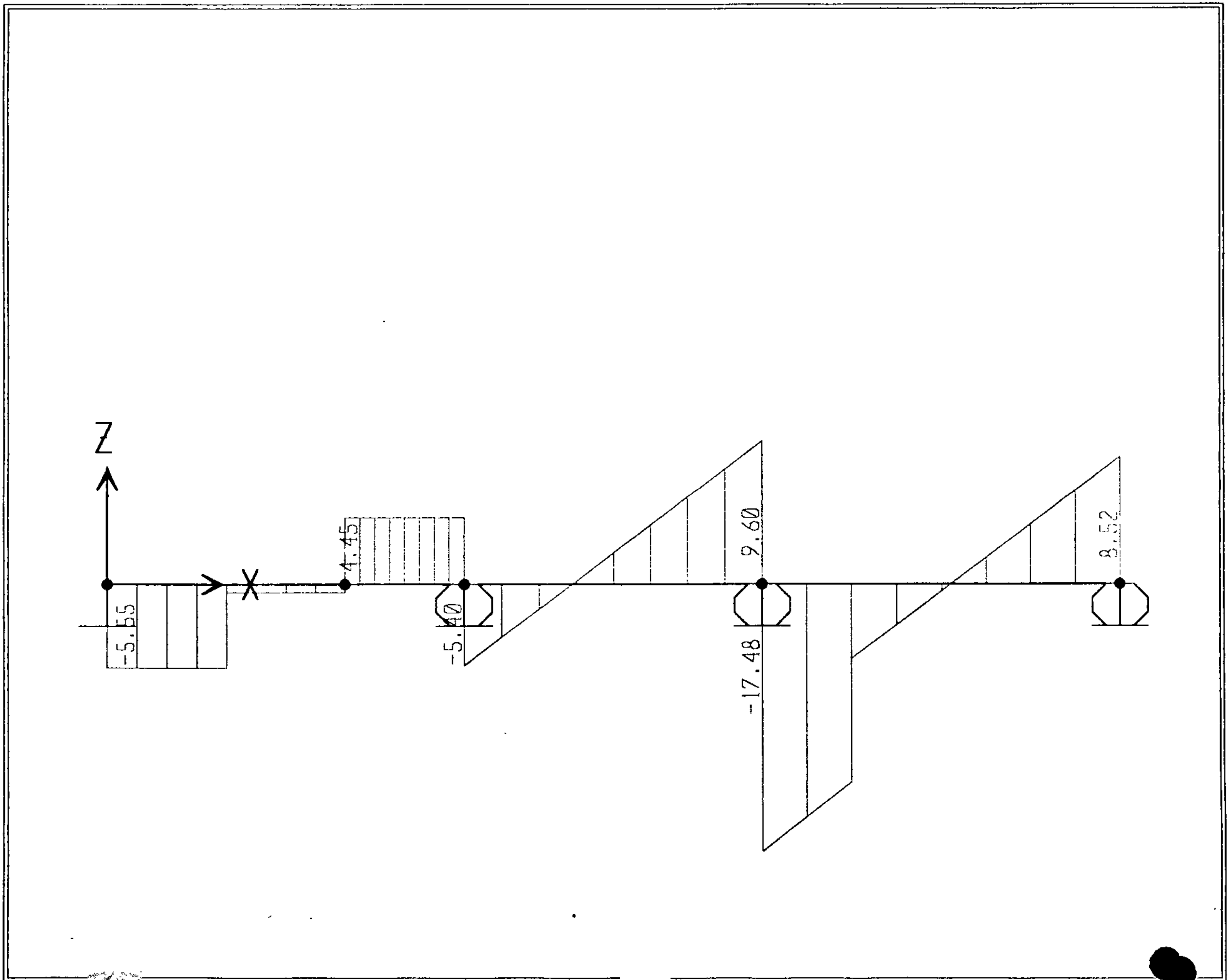

Z
X
5.00
5.00
8.00

SAP2000

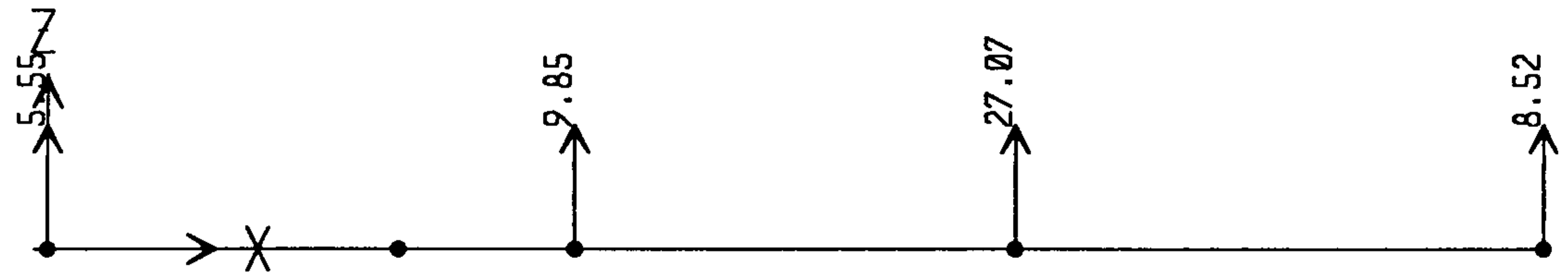
Marzo 12,2000 20:24



SAP2000 v6.10 - File:ejemplo1 - Moment 3-3 Diagram (LOAD1) - Ton-m Units



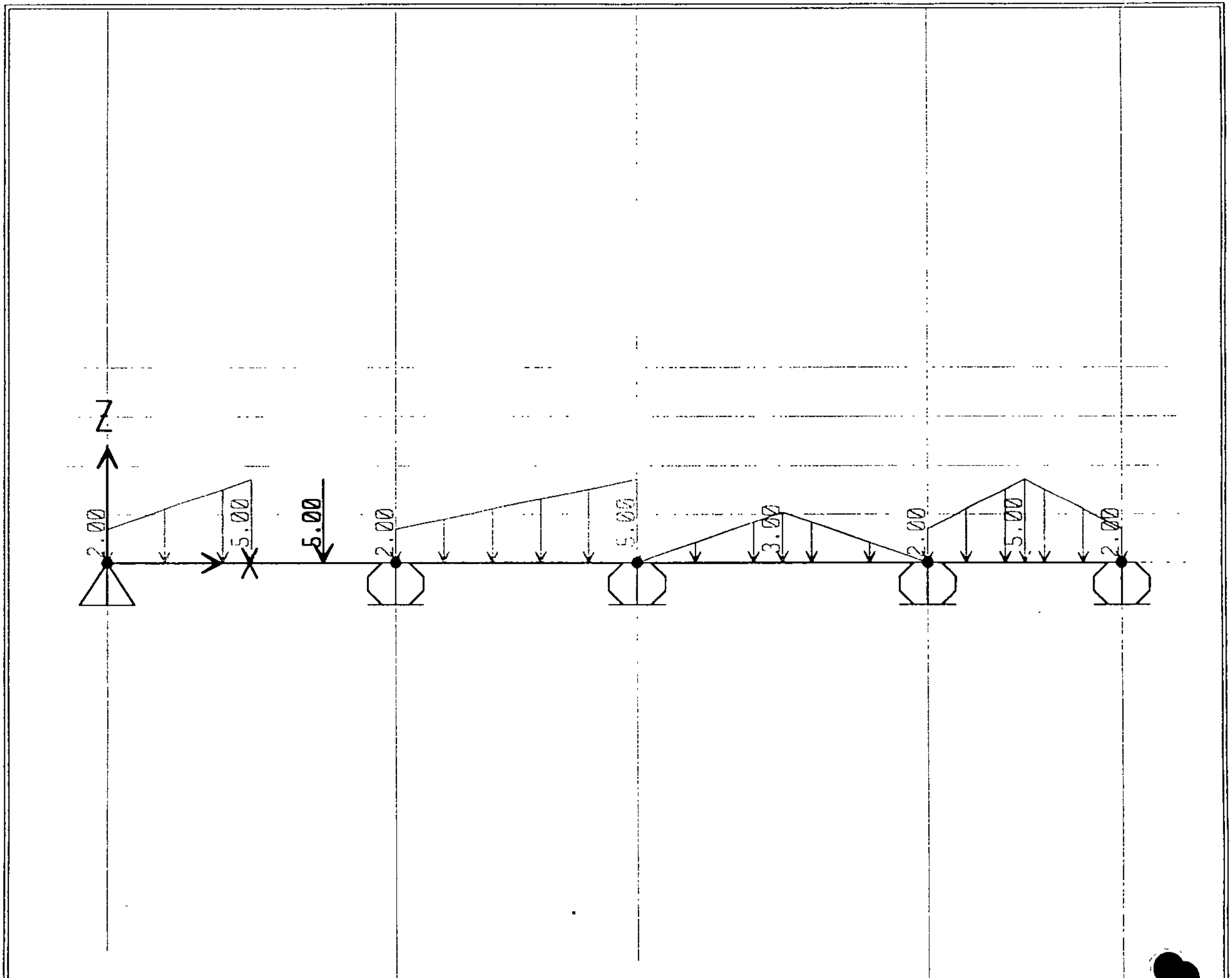
SAP2000 v... - File:ejemplo1 - Shear Force 2-2 Diagram (LOAD1) - Ton-m ...s



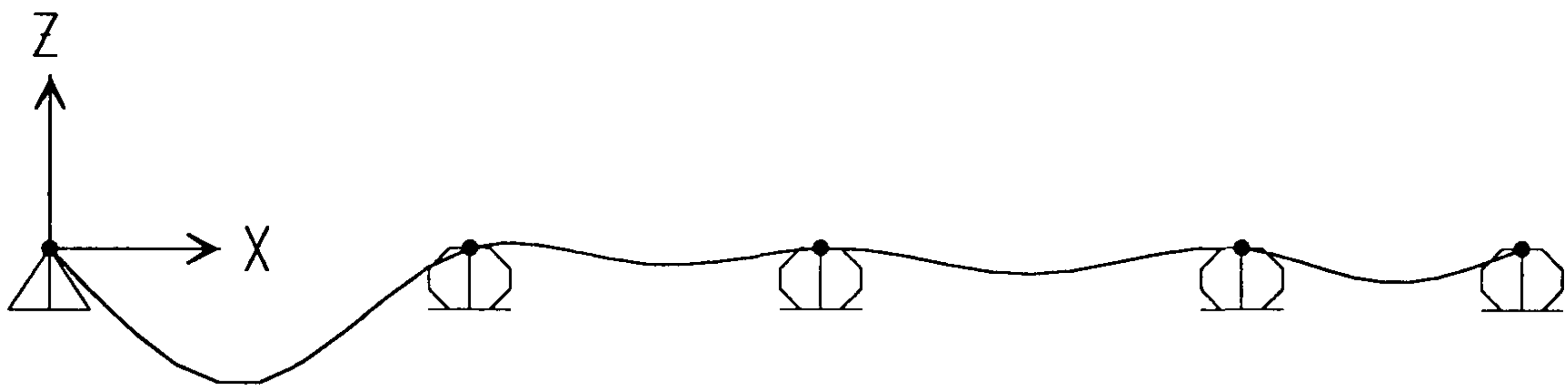
SAP2000 v6.10 - File:ejemplo1 - Restraint Reactions (LOAD1) - Ton-m Units

```
; File C:\Mis documentos\curso sap 2000\ejemplo1.s2k saved 3/12/00 20:38:00 in Ton-m
SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  PAGE=SECTIONS
JOINT
  1  X=0  Y=0  Z=0
  2  X=6  Y=0  Z=0
  3  X=11  Y=0  Z=0
  4  X=17  Y=0  Z=0
  5  X=4  Y=0  Z=0
RESTRAINT
  ADD=1  DOF=U1,U2,U3,R1,R2,R3
  ADD=2  DOF=U3
  ADD=3  DOF=U3
  ADD=4  DOF=U3
PATTERN
  NAME=DEFAULT
MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=OTHER  IDES=N  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=MAT2  IDES=N  M=.7981  W=7.8334
    T=0  E=2200000  U=.25  A=.0000117
FRAME SECTION
  NAME=FSEC1 MAT=STEEL SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125
  NAME=FS1 MAT=MAT2 SH=R T=.5,.25 A=.125 J=1.788127E-03 I=2.604167E-03,6.510417E-04 AS=.1041667,.1041667
  NAME=FS2 MAT=MAT2 SH=R T=.5,.5 A=.25 J=8.802084E-03 I=5.208333E-03,5.208333E-03 AS=.2083333,.2083333
FRAME
  2  J=2,3  SEC=FS1  NSEG=4  ANG=0
  3  J=3,4  SEC=FS1  NSEG=4  ANG=0
  4  J=1,5  SEC=FS1  NSEG=4  ANG=0
  5  J=5,2  SEC=FS2  NSEG=4  ANG=0
LOAD
  NAME=LOAD1
    TYPE=FORCE
      ADD=5  UZ=-5
    TYPE=CONCENTRATED SPAN
      ADD=4  RD=.5  UZ=-5
      ADD=3  RD=.25  UZ=-8
    TYPE=DISTRIBUTED SPAN
      ADD=2  RD=0,1  UZ=-3,-3
      ADD=3  RD=0,1  UZ=-3,-3
OUTPUT
; No Output Requested
END
; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  0
  GRID GLOBAL  X "2"  1
  GRID GLOBAL  X "3"  2
  GRID GLOBAL  X "4"  3
  GRID GLOBAL  X "5"  4
  GRID GLOBAL  X "6"  5
  GRID GLOBAL  X "7"  6
  GRID GLOBAL  X "8"  7
  GRID GLOBAL  X "9"  8
  GRID GLOBAL  X "10"  9
  GRID GLOBAL  X "11"  10
  GRID GLOBAL  X "12"  11
  GRID GLOBAL  X "13"  12
  GRID GLOBAL  X "14"  13
  GRID GLOBAL  X "15"  14
  GRID GLOBAL  X "16"  15
  GRID GLOBAL  X "17"  16
  GRID GLOBAL  X "18"  17
  GRID GLOBAL  Y "19"  0
  GRID GLOBAL  Z "20"  0
  GRID GLOBAL  Z "21"  1
  GRID GLOBAL  Z "22"  2
  GRID GLOBAL  Z "23"  3
  GRID GLOBAL  Z "24"  4
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  FRAMESECTION FS1 NAME REC25X50
  FRAMESECTION FS2 NAME REC50X50
  STATICLOAD LOAD1 TYPE DEAD
END SUPPLEMENTAL DATA
```

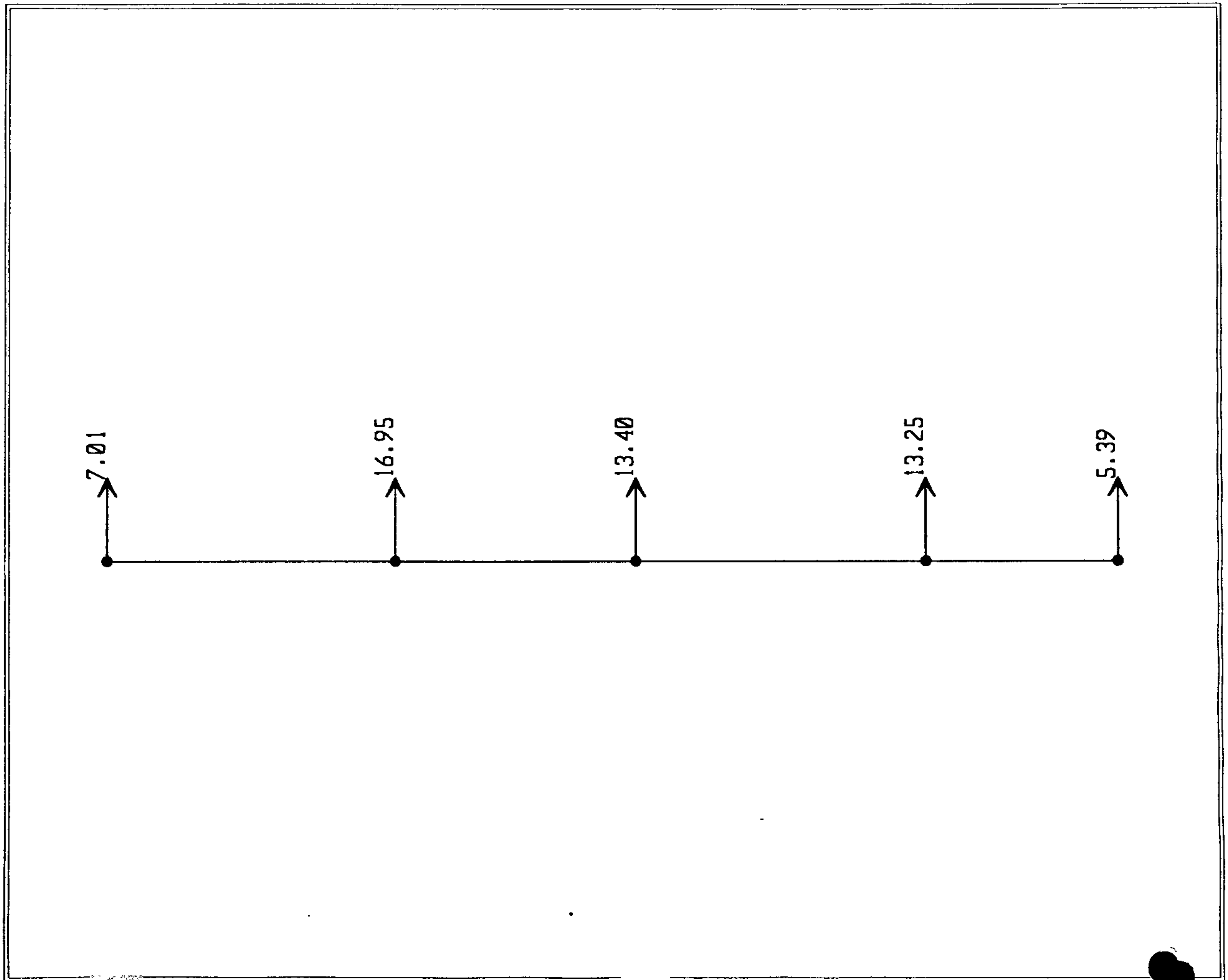
```
; File C:\Mis documentos\curso sap 2000\ejemplo1.$2k saved 3/12/00 20:38:00 in Ton-m

SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  PAGE=SECTIONS
JOINT
  1  X=0  Y=0  Z=0
  2  X=6  Y=0  Z=0
  3  X=11  Y=0  Z=0
  4  X=17  Y=0  Z=0
  5  X=4  Y=0  Z=0
RESTRAINT
  ADD=1  DOF=U1,U2,U3,R1,R2,R3
  ADD=2  DOF=U3
  ADD=3  DOF=U3
  ADD=4  DOF=U3
PATTERN
  NAME=DEFAULT
MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=OTHER  IDES=N  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=MAT2  IDES=N  M=.7981  W=7.8334
    T=0  E=2200000  U=.25  A=.0000117
FRAME SECTION
  NAME=FSEC1 MAT=STEEL SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125
  NAME=FS1 MAT=MAT2 SH=R T=.5,.25 A=.125 J=1.788127E-03 I=2.604167E-03,6.510417E-04 AS=.1041667,.1041667
  NAME=FS2 MAT=MAT2 SH=R T=.5,.5 A=.25 J=8.802084E-03 I=5.208333E-03,5.208333E-03 AS=.2083333,.2083333
FRAME
  2  J=2,3  SEC=FS1  NSEG=4  ANG=0
  3  J=3,4  SEC=FS1  NSEG=4  ANG=0
  4  J=1,5  SEC=FS1  NSEG=4  ANG=0
  5  J=5,2  SEC=FS2  NSEG=4  ANG=0
LOAD
  NAME=LOAD1
    TYPE=FORCE
      ADD=5  UZ=-5
    TYPE=CONCENTRATED SPAN
      ADD=4  RD=.5  UZ=-5
      ADD=3  RD=.25  UZ=-8
    TYPE=DISTRIBUTED SPAN
      ADD=2  RD=0,1  UZ=-3,-3
      ADD=3  RD=0,1  UZ=-3,-3
OUTPUT
; No Output Requested
END
; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  0
  GRID GLOBAL  X "2"  1
  GRID GLOBAL  X "3"  2
  GRID GLOBAL  X "4"  3
  GRID GLOBAL  X "5"  4
  GRID GLOBAL  X "6"  5
  GRID GLOBAL  X "7"  6
  GRID GLOBAL  X "8"  7
  GRID GLOBAL  X "9"  8
  GRID GLOBAL  X "10"  9
  GRID GLOBAL  X "11"  10
  GRID GLOBAL  X "12"  11
  GRID GLOBAL  X "13"  12
  GRID GLOBAL  X "14"  13
  GRID GLOBAL  X "15"  14
  GRID GLOBAL  X "16"  15
  GRID GLOBAL  X "17"  16
  GRID GLOBAL  X "18"  17
  GRID GLOBAL  Y "19"  0
  GRID GLOBAL  Z "20"  0
  GRID GLOBAL  Z "21"  1
  GRID GLOBAL  Z "22"  2
  GRID GLOBAL  Z "23"  3
  GRID GLOBAL  Z "24"  4
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  FRAMESECTION FS1 NAME REC25X50
  FRAMESECTION FS2 NAME REC50X50
  STATICLOAD LOAD1 TYPE DEAD
END SUPPLEMENTAL DATA
```



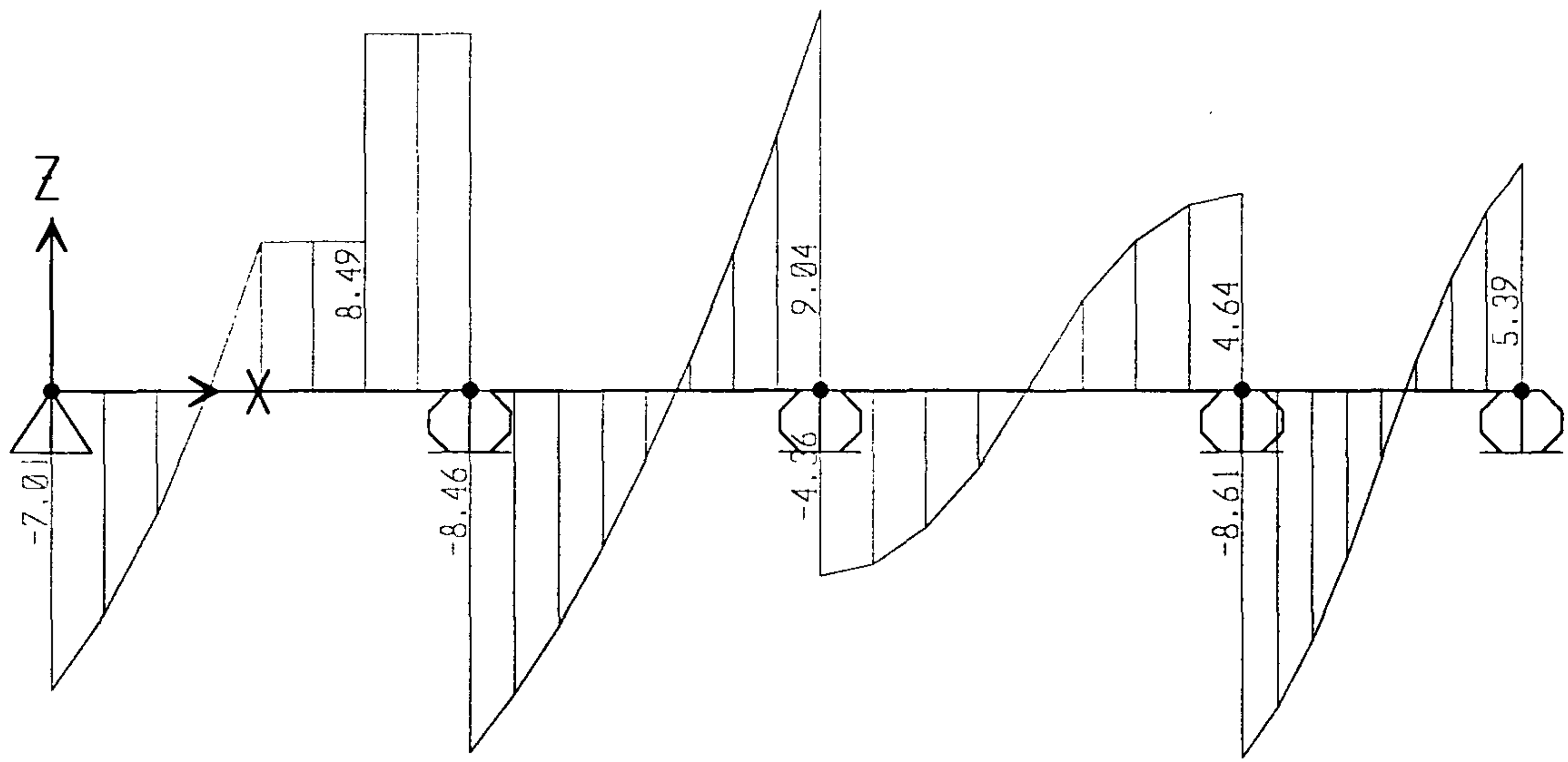
SAP2000 v... - File:ejemplo2 - Frame Span Loads (VERTICAL) - Ton-m Units



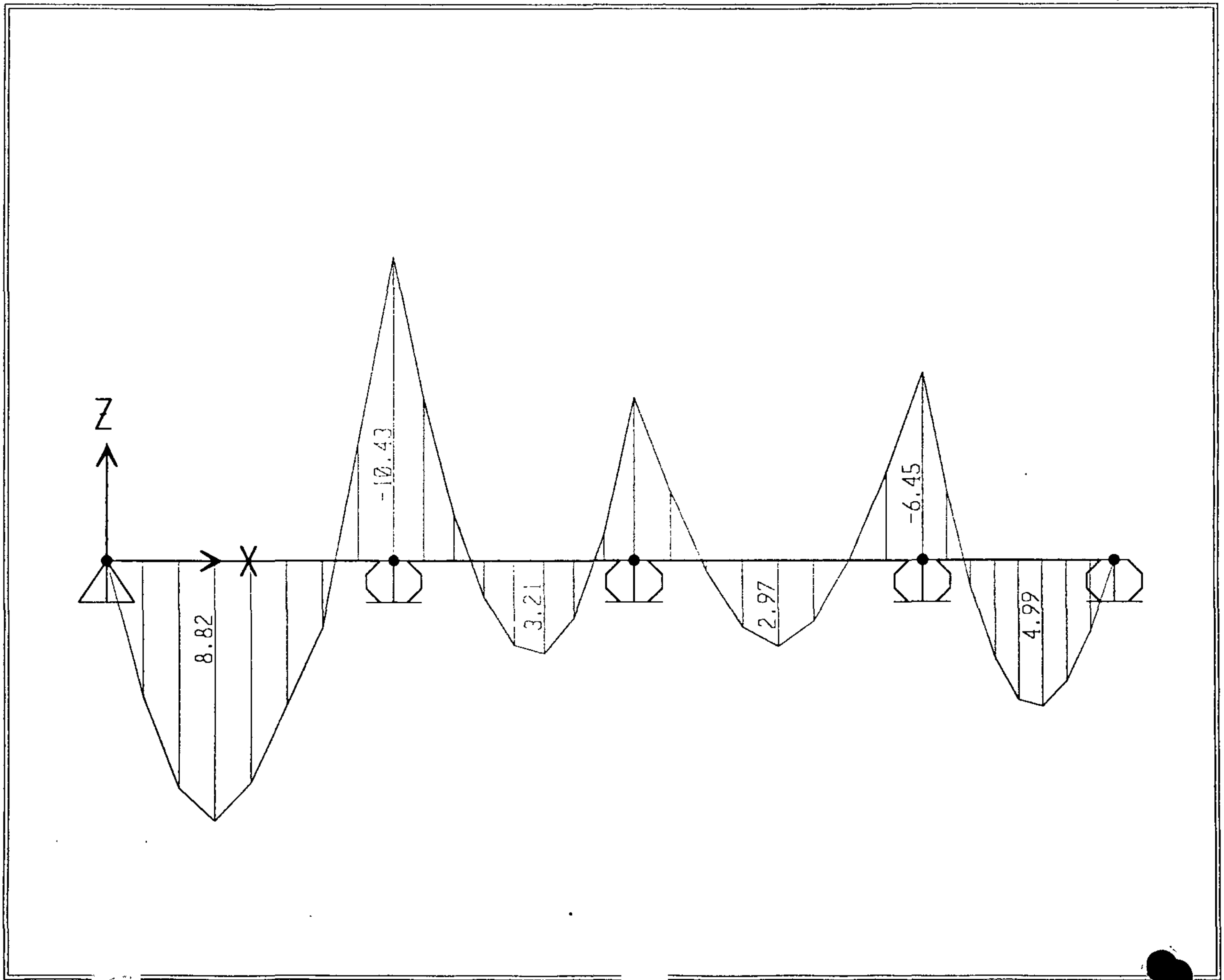
SAP2000 v6.10 - File:ejemplo2 - Deformed Shape (VERTICAL) - Ton-m Units



SAP2000 v - File:ejemplo2 - Restraint Reactions (VERTICAL) - Ton-m U.



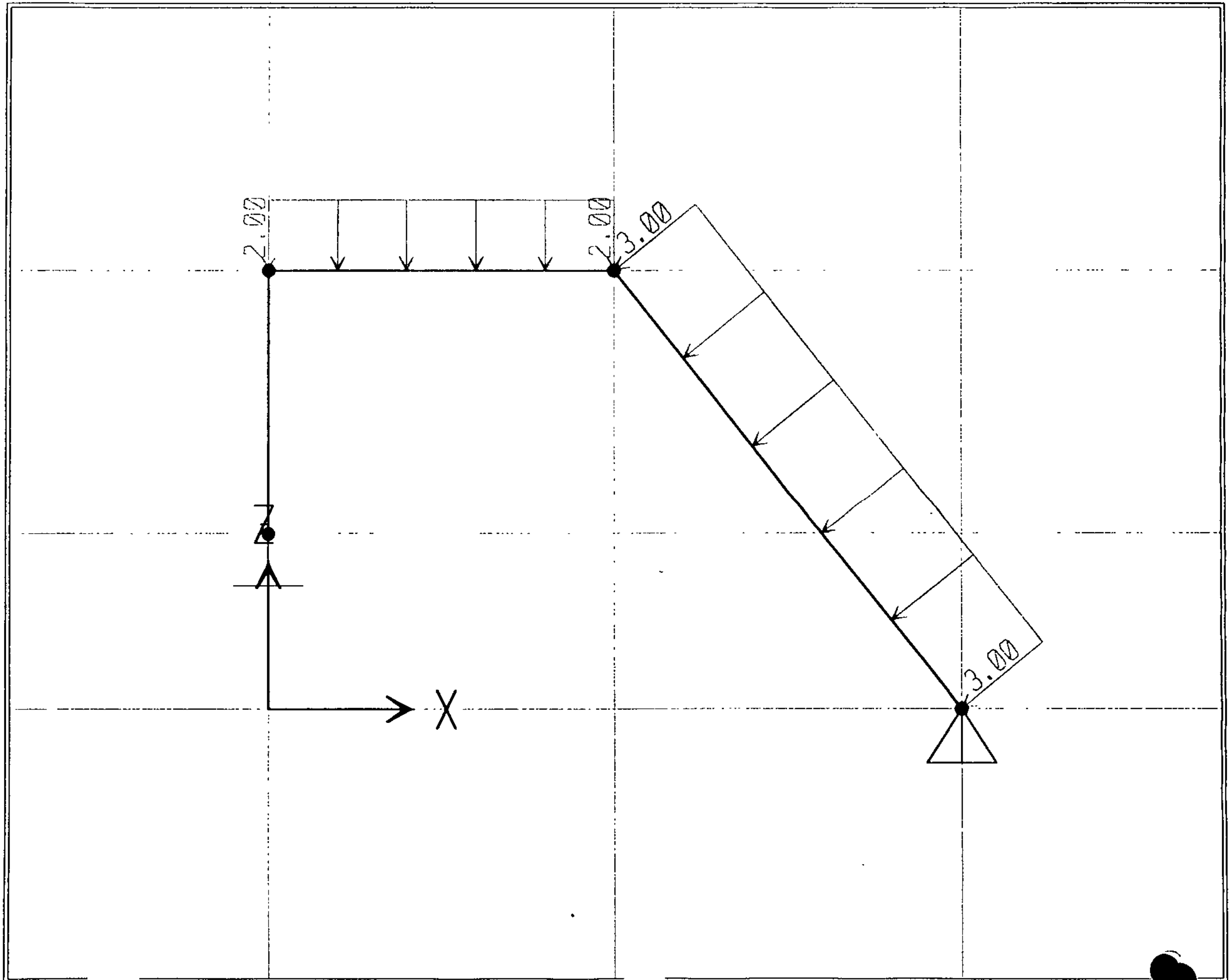
SAP2000 v6.10 - File:ejemplo2 - Shear Force 2-2 Diagram (VERTICAL) - Ton-m Units



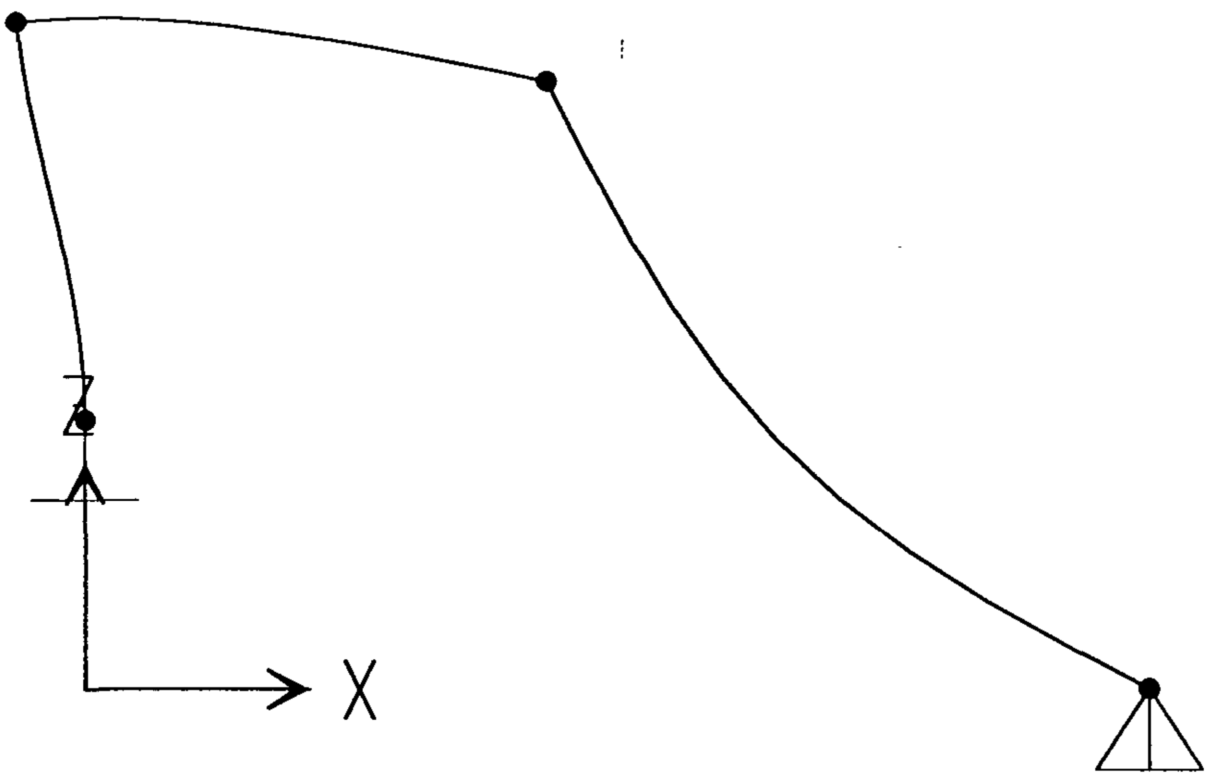
SAP2000 v - File:ejemplo2 - Moment 3-3 Diagram (VERTICAL) - Ton-m

Ejemplo 2

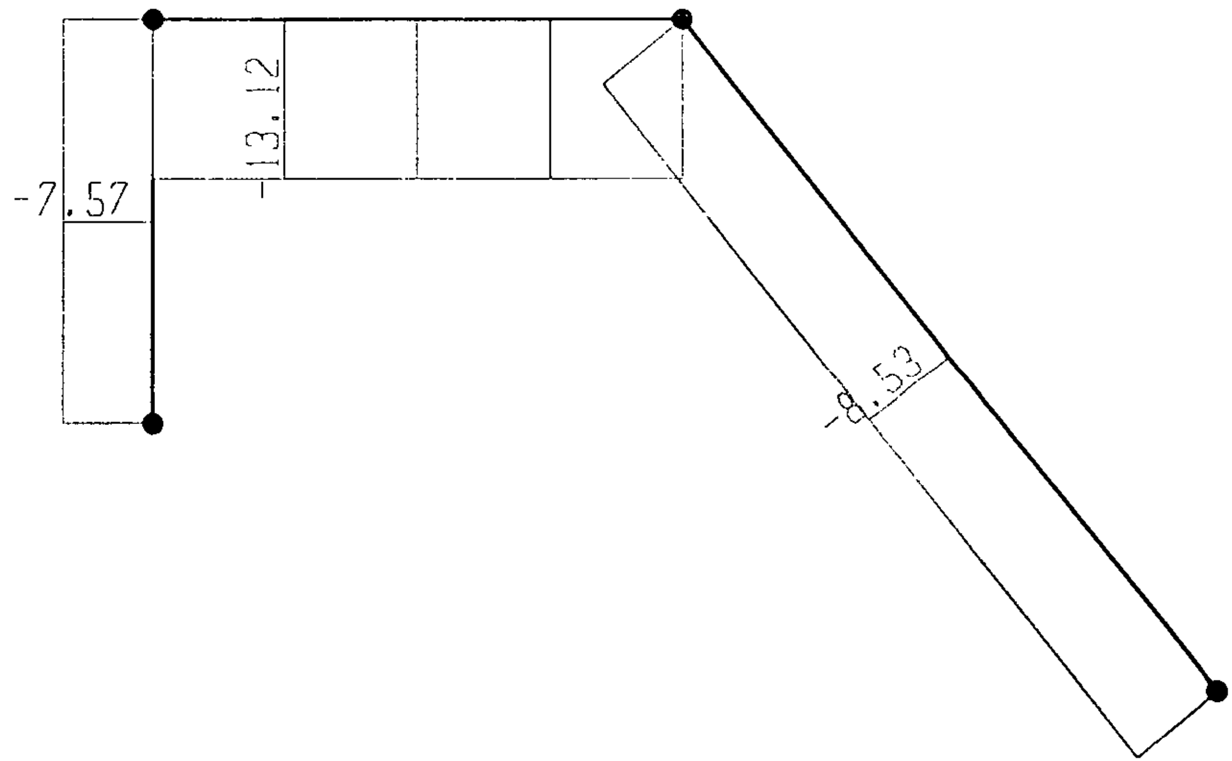
```
; File C:\Mis documentos\curso sap 2000\ejemplo2.s2k saved 3/18/00 8:59:59 in Ton-m
SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  LINES=59
JOINT
  5  X=0  Y=0  Z=0
  6  X=6  Y=0  Z=0
  7  X=11  Y=0  Z=0
  8  X=17  Y=0  Z=0
  9  X=21  Y=0  Z=0
RESTRAINT
  ADD=5  DOF=U1,U2,U3
  ADD=6  DOF=U3
  ADD=7  DOF=U3
  ADD=8  DOF=U3
  ADD=9  DOF=U3
PATTERN
  NAME=DEFAULT
MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=OTHER  IDES=N  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=MATD  IDES=C  M=.7981  W=7.8334
    T=0  E=1800000  U=.25  A=.0000117
FRAME SECTION
  NAME=FS1 MAT=MATD SH=R T=.5,.25 A=.125 J=1.788127E-03 I=2.604167E-03,6.510417E-04 AS=.1041667,.1041667
FRAME
  3  J=5,6  SEC=FS1  NSEG=4  ANG=0
  4  J=6,7  SEC=FS1  NSEG=4  ANG=0
  5  J=7,8  SEC=FS1  NSEG=4  ANG=0
  6  J=8,9  SEC=FS1  NSEG=4  ANG=0
LOAD
  NAME=VERTICAL
    TYPE=CONCENTRATED SPAN
      ADD=3  RD=.75  UZ=-5
    TYPE=DISTRIBUTED SPAN
      ADD=3  RD=0,.5  UZ=-2,-5
      ADD=4  RD=0,1  UZ=-2,-5
      ADD=5  RD=0,.5  UZ=0,-3
      ADD=5  RD=.5,1  UZ=-3,0
      ADD=6  RD=0,.5  UZ=-2,-5
      ADD=6  RD=.5,1  UZ=-5,-2
OUTPUT
; No Output Requested
END
; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  0
  GRID GLOBAL  X "2"  6
  GRID GLOBAL  X "3"  11
  GRID GLOBAL  X "4"  17
  GRID GLOBAL  X "5"  21
  GRID GLOBAL  Y "6"  0
  GRID GLOBAL  Z "7"  0
  GRID GLOBAL  Z "8"  1
  GRID GLOBAL  Z "9"  2
  GRID GLOBAL  Z "10"  3
  GRID GLOBAL  Z "11"  4
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  MATERIAL MATD FYREBAR 60 FYSHEAR 40 FC 4 FCSHEAR 4
  FRAMESECTION FS1 NAME REC25X50
  CONCRETESECTION REC25X50 BEAM  COVERTOP .05 COVERBOTTOM .05
  STATICLOAD VERTICAL TYPE DEAD
END SUPPLEMENTAL DATA
```

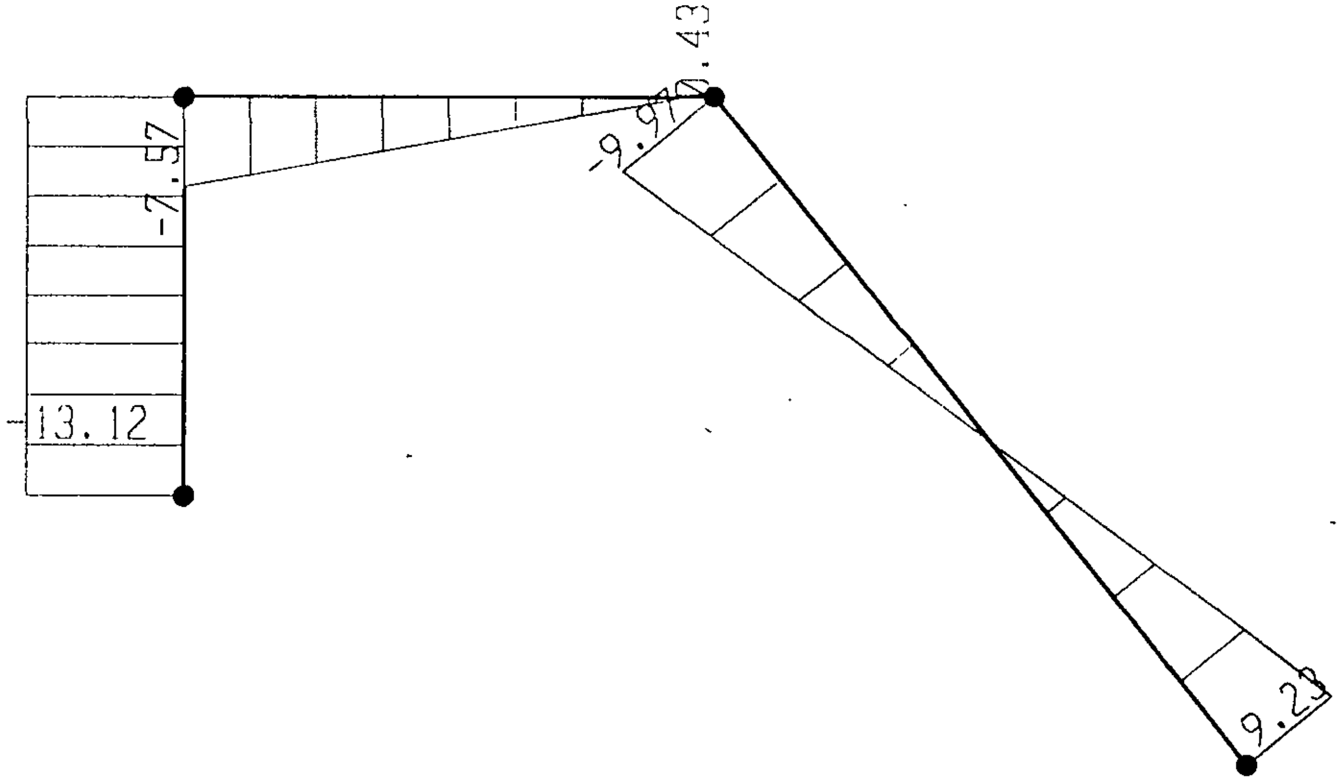


SAP2000 v - File:ejemplo3 - Frame Span Loads (UNICA) - Ton-m Units

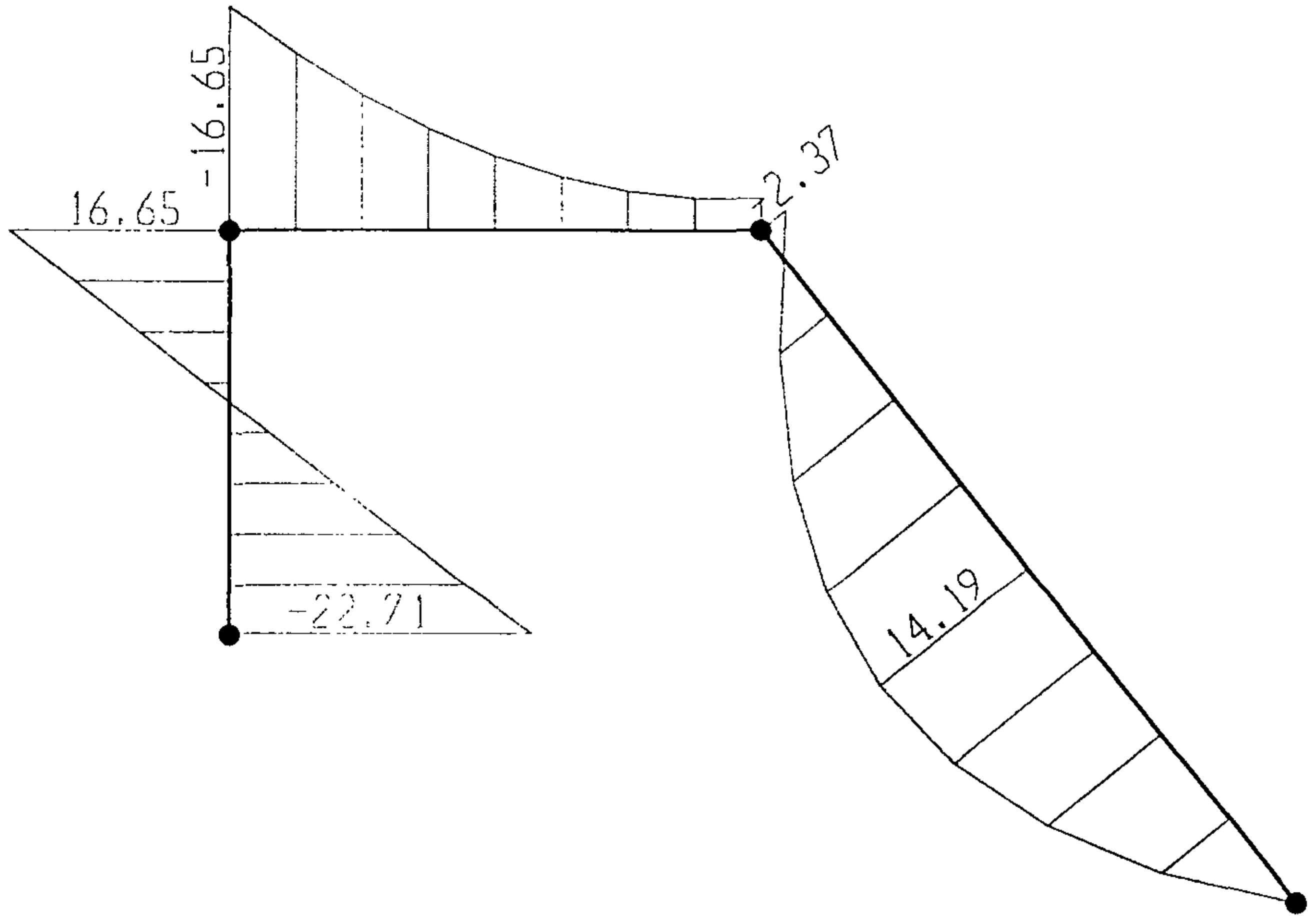


SAP2000 v6.10 - File:ejemplo3 - Deformed Shape (UNICA) - Ton-m Units

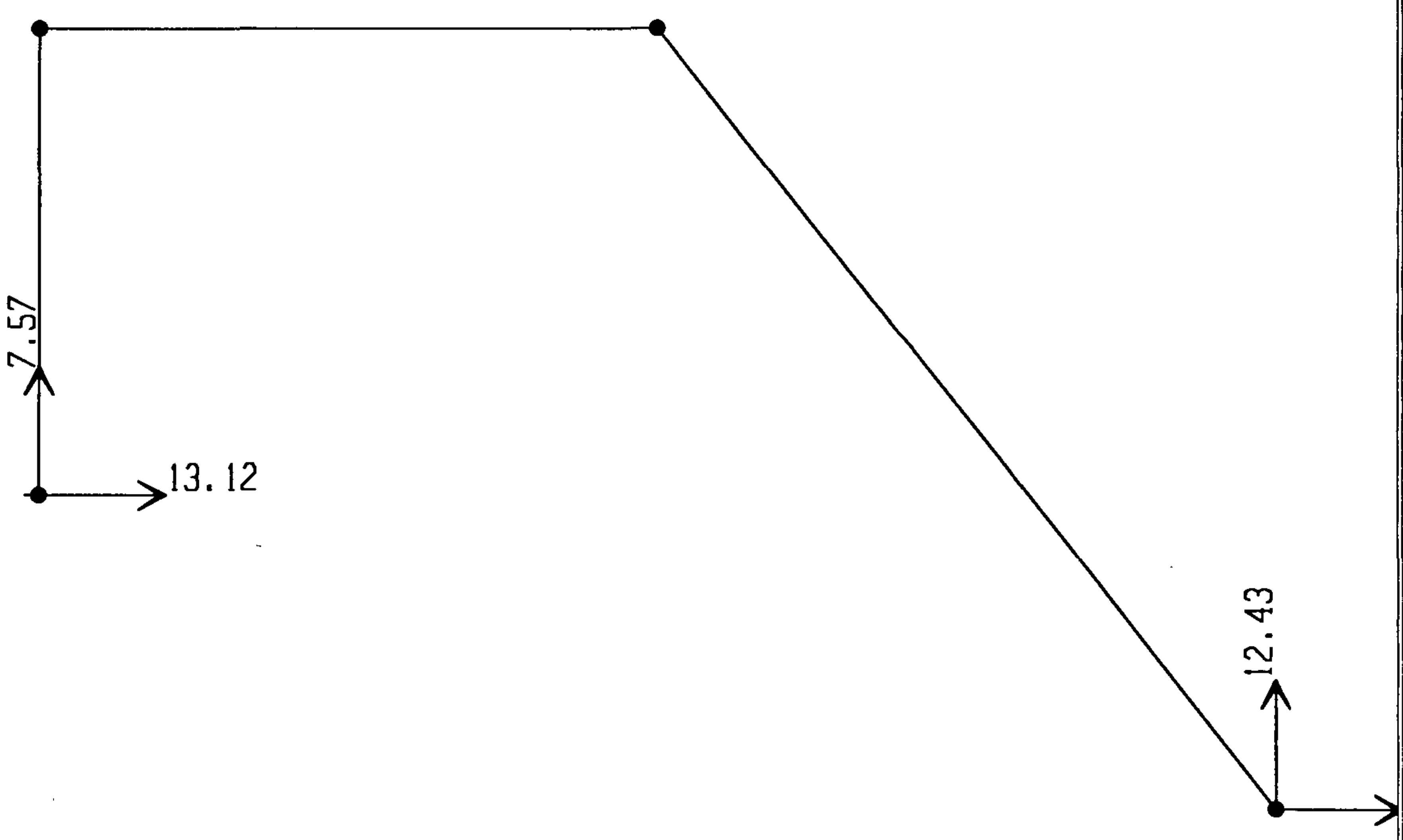

-13.12
-7.57
-8.53



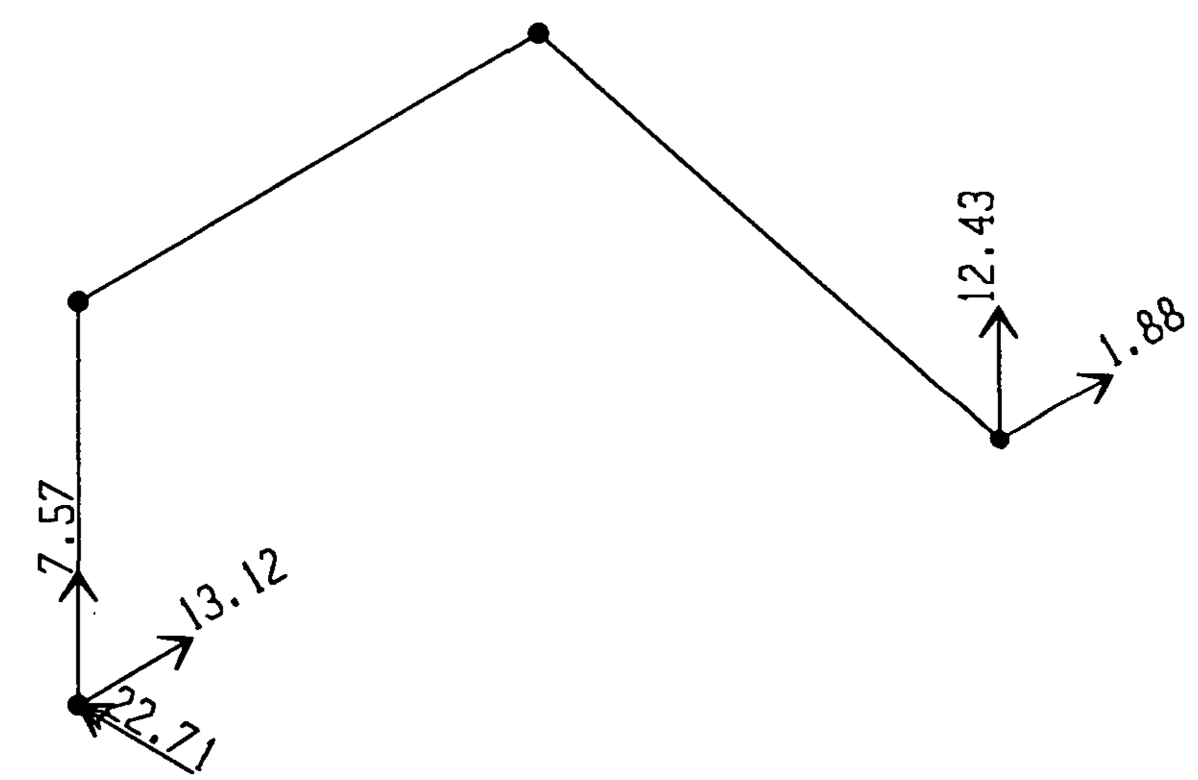
SAP2000 v6.10 - File:ejemplo3 - Shear Force 2-2 Diagram (UNICA) - Ton-m Units



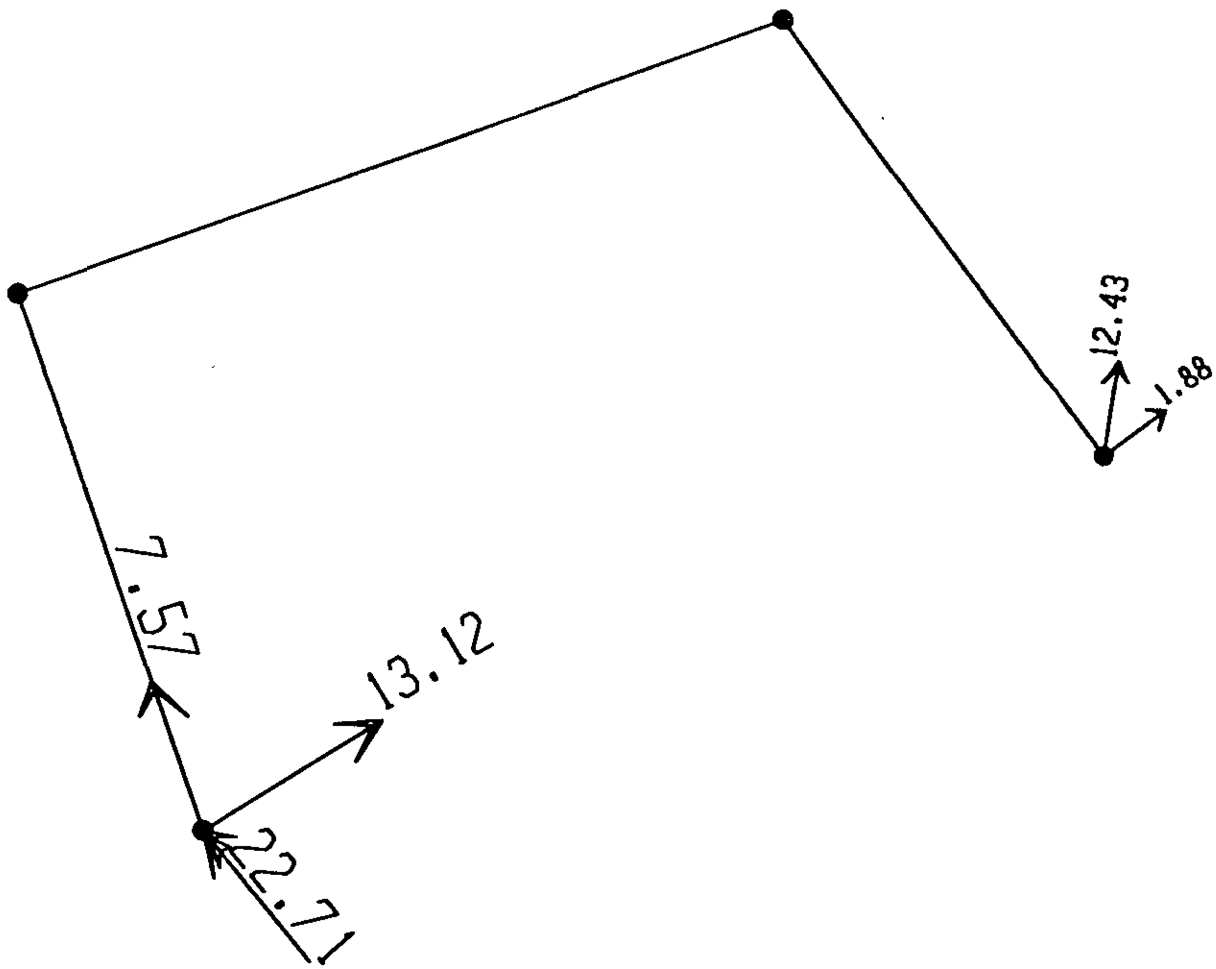
-16.65
16.65
2.37
14.19
-22.71



SAP2000 v6.10 - File:ejemplo3 - Restraint Reactions (UNICA) - Ton-m Units


12.43
1.88
7.57
13.12
22.71

SAP2000 Marzo 18,2000 9:40



SAP2000 v6.10 - File:ejemplo3 - Restraint Reactions (UNICA) - Ton-m Units

```
; File C:\Mis documentos\curso sap 2000\ejemplo3.s2k saved 3/18/00 9:28:09 in Ton-m

SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  LINES=59

JOINT
  1  X=0  Y=0  Z=2
  2  X=0  Y=0  Z=5
  3  X=4  Y=0  Z=5
  4  X=8  Y=0  Z=0

RESTRAINT
  ADD=1  DOF=U1,U2,U3,R1,R2,R3
  ADD=4  DOF=U1,U2,U3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=MAT1  IDES=N  M=.2448  W=2.4026
    T=0  E=1000000  U=.2  A=.0000099

FRAME SECTION
  NAME=FS1 MAT=MAT1 SH=R T=.5,.25 A=.125 J=1.788127E-03 I=2.604167E-03,6.510417E-04 AS=.1041667,.1041667

FRAME
  1  J=1,2  SEC=FS1  NSEG=2  ANG=0
  2  J=2,3  SEC=FS1  NSEG=4  ANG=0
  3  J=3,4  SEC=FS1  NSEG=2  ANG=0

LOAD
  NAME=UNICA
    TYPE=DISTRIBUTED SPAN
      ADD=2  RD=0,1  UZ=-2,-2
      ADD=3  RD=0,1  U2=-3,-3

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=UNICA
  ELEM=JOINT  TYPE=APPL  LOAD=UNICA
  ELEM=FRAME  TYPE=FORCE  LOAD=UNICA
  ELEM=FRAME  TYPE=JOINTF  LOAD=UNICA

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  0
  GRID GLOBAL  X "2"  4
  GRID GLOBAL  X "3"  8
  GRID GLOBAL  Y "4"  0
  GRID GLOBAL  Z "5"  0
  GRID GLOBAL  Z "6"  2
  GRID GLOBAL  Z "7"  5
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  FRAMESECTION FS1 NAME REC25X50
  STATICLOAD UNICA TYPE OTHER
END SUPPLEMENTAL DATA
```

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 1
Marzo 18, 2000 9:30

S T A T I C   L O A D   C A S E S

STATIC CASE	CASE TYPE	SELF WT FACTOR
UNICA	OTHER	0.0000

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 2
Marzo 18, 2000 9:30

M A T E R I A L   P R O P E R T Y   D A T A

MAT LABEL	MODULUS OF ELASTICITY	POISSON'S RATIO	THERMAL COEFF	WEIGHT PER UNIT VOL	MASS PER UNIT VOL
STEEL	20389020	0.300	1.170E-05	7.833	0.798
CONC	2531051	0.200	9.900E-06	2.403	0.245
MAT1	1000000.000	0.200	9.900E-06	2.403	0.245

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 3
Marzo 18, 2000 9:30

M A T E R I A L   D E S I G N   D A T A

MAT LABEL	DESIGN CODE	STEEL FY	CONCRETE FC	REBAR FY	CONCRETE FCS	REBAR FYS
STEEL	S	25310.500				
CONC	C		2812.278	42184.180	2812.278	28122.779
MAT1	N					

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 4
Marzo 18, 2000 9:30

F R A M E   S E C T I O N   P R O P E R T Y   D A T A

SECTION LABEL	MAT LABEL	SECTION TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
REC25X50	MAT1		0.500	0.250	0.000	0.000	0.000	0.000

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 5
Marzo 18, 2000 9:30

F R A M E   S E C T I O N   P R O P E R T Y   D A T A

SECTION LABEL	AREA	TORSIONAL INERTIA	MOMENTS OF INERTIA I33	I22	SHEAR AREAS A2	A3
REC25X50	0.125	1.788E-03	2.604E-03	6.510E-04	0.104	0.104

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 6
Marzo 18, 2000 9:30

F R A M E   S E C T I O N   P R O P E R T Y   D A T A

SECTION LABEL	SECTION MODULII S33	S22	PLASTIC MODULII Z33	Z22	RADII OF GYRATION R33	R22
REC25X50	1.042E-02	5.208E-03	1.563E-02	7.813E-03	0.144	7.217E-02

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 7
Marzo 18, 2000 9:30

F R A M E   S E C T I O N   P R O P E R T Y   D A T A

SECTION LABEL	TOTAL WEIGHT	TOTAL MASS
REC25X50	4.025	0.410

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 8
Marzo 18, 2000 9:30

S H E L L   S E C T I O N   P R O P E R T Y   D A T A

SECTION LABEL	TOTAL WEIGHT	TOTAL MASS
SSEC1	0.000	0.000

F R A M E   S P A N   D I S T R I B U T E D   L O A D S   Load Case UNICA

FRAME	TYPE	DIRECTION	DISTANCE-A	VALUE-A	DISTANCE-B	VALUE-B
2	FORCE	GLOBAL-Z	0.0000	-2.0000	1.0000	-2.0000
3	FORCE	LOCAL-2	0.0000	-3.0000	1.0000	-3.0000

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 1
Marzo 18, 2000 9:28

J O I N T   D I S P L A C E M E N T S

JOINT	LOAD	UX	UY	UZ	RX	RY	RZ
1	UNICA	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	UNICA	-0.0175	0.0000	-1.817E-04	0.0000	-3.490E-03	0.0000
3	UNICA	-0.0179	0.0000	-0.0149	0.0000	7.018E-03	0.0000
4	UNICA	0.0000	0.0000	0.0000	0.0000	-0.0153	0.0000

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 2
Marzo 18, 2000 9:28

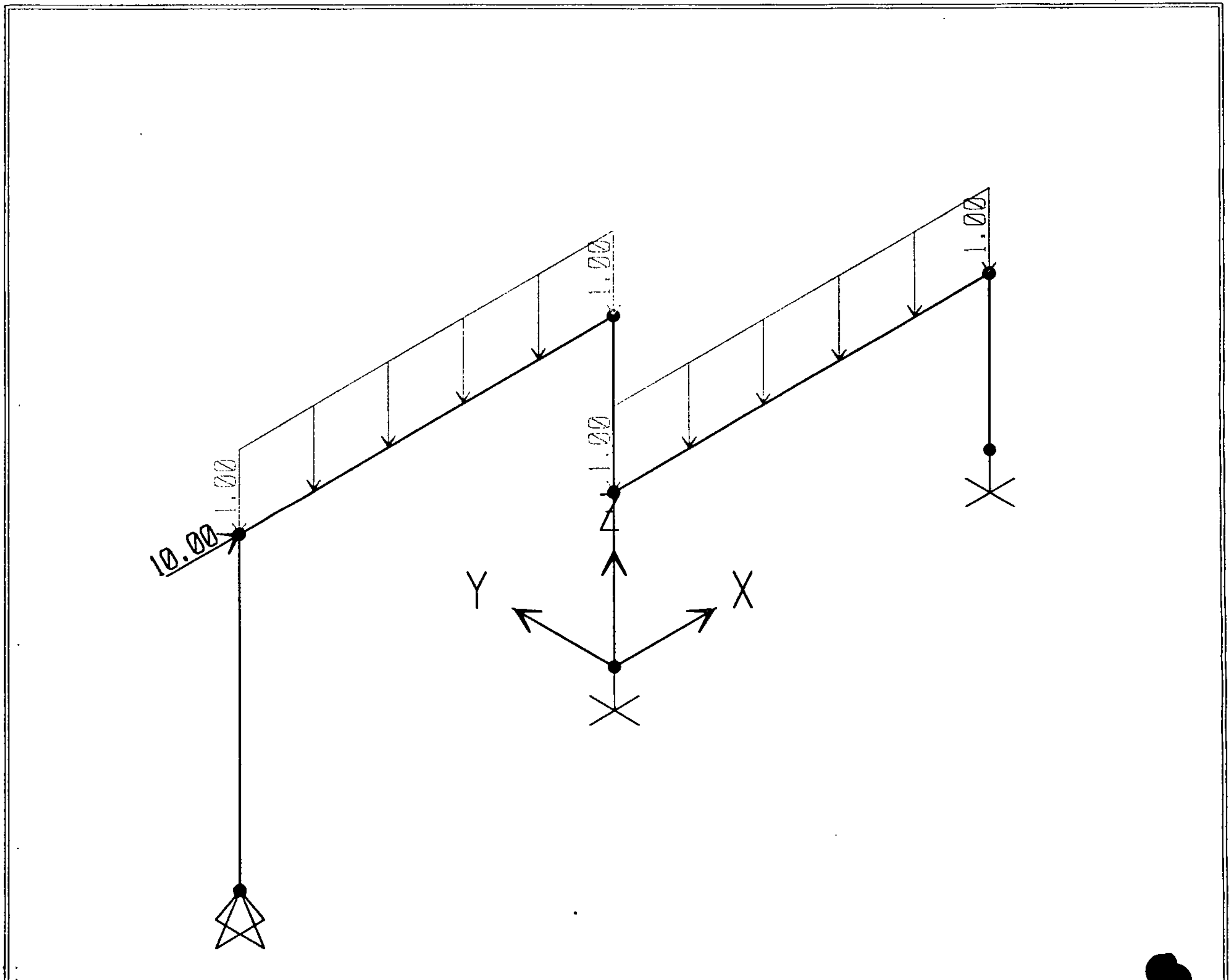
J O I N T   R E A C T I O N S

JOINT	LOAD	F1	F2	F3	M1	M2	M3
1	UNICA	13.1180	0.0000	7.5697	0.0000	22.7067	0.0000
4	UNICA	1.8820	0.0000	12.4303	0.0000	0.0000	0.0000

SAP2000 v6.10 File: EJEMPLO3 Ton-m Units PAGE 3
Marzo 18, 2000 9:28

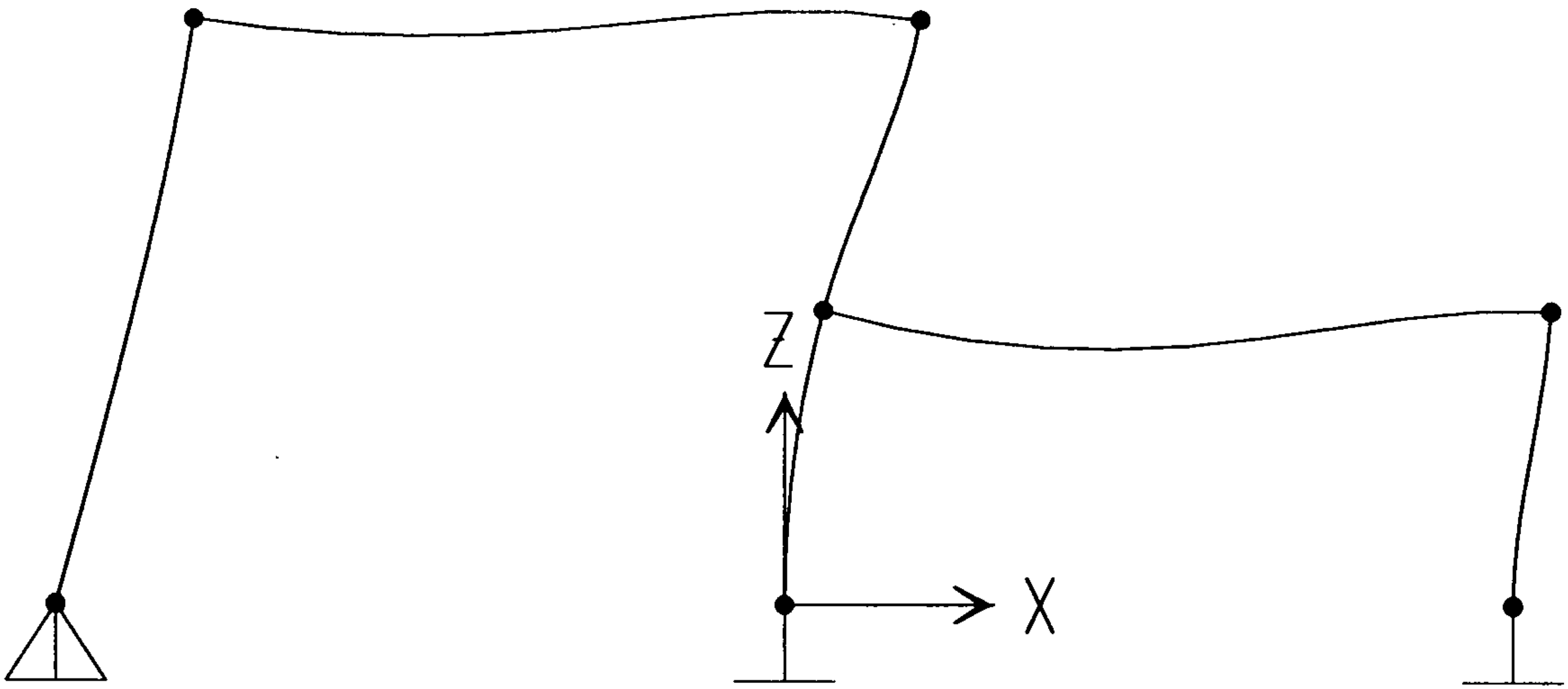
F R A M E   E L E M E N T   F O R C E S

FRAME	LOAD	LOC	P	V2	V3	T	M2	M3
1	UNICA							
		0.00	-7.57	-13.12	0.00	0.00	0.00	-22.71
		1.50	-7.57	-13.12	0.00	0.00	0.00	-3.03
		3.00	-7.57	-13.12	0.00	0.00	0.00	16.65
2	UNICA							
		0.00	-13.12	-7.57	0.00	0.00	0.00	-16.65
		1.00	-13.12	-5.57	0.00	0.00	0.00	-10.08
		2.00	-13.12	-3.57	0.00	0.00	0.00	-5.51
		3.00	-13.12	-1.57	0.00	0.00	0.00	-2.94
		4.00	-13.12	4.303E-01	0.00	0.00	0.00	-2.37
3	UNICA							
		0.00	-8.53	-9.97	0.00	0.00	0.00	-2.37
		3.20	-8.53	-3.699E-01	0.00	0.00	0.00	14.19
		6.40	-8.53	9.23	0.00	0.00	0.00	0.00

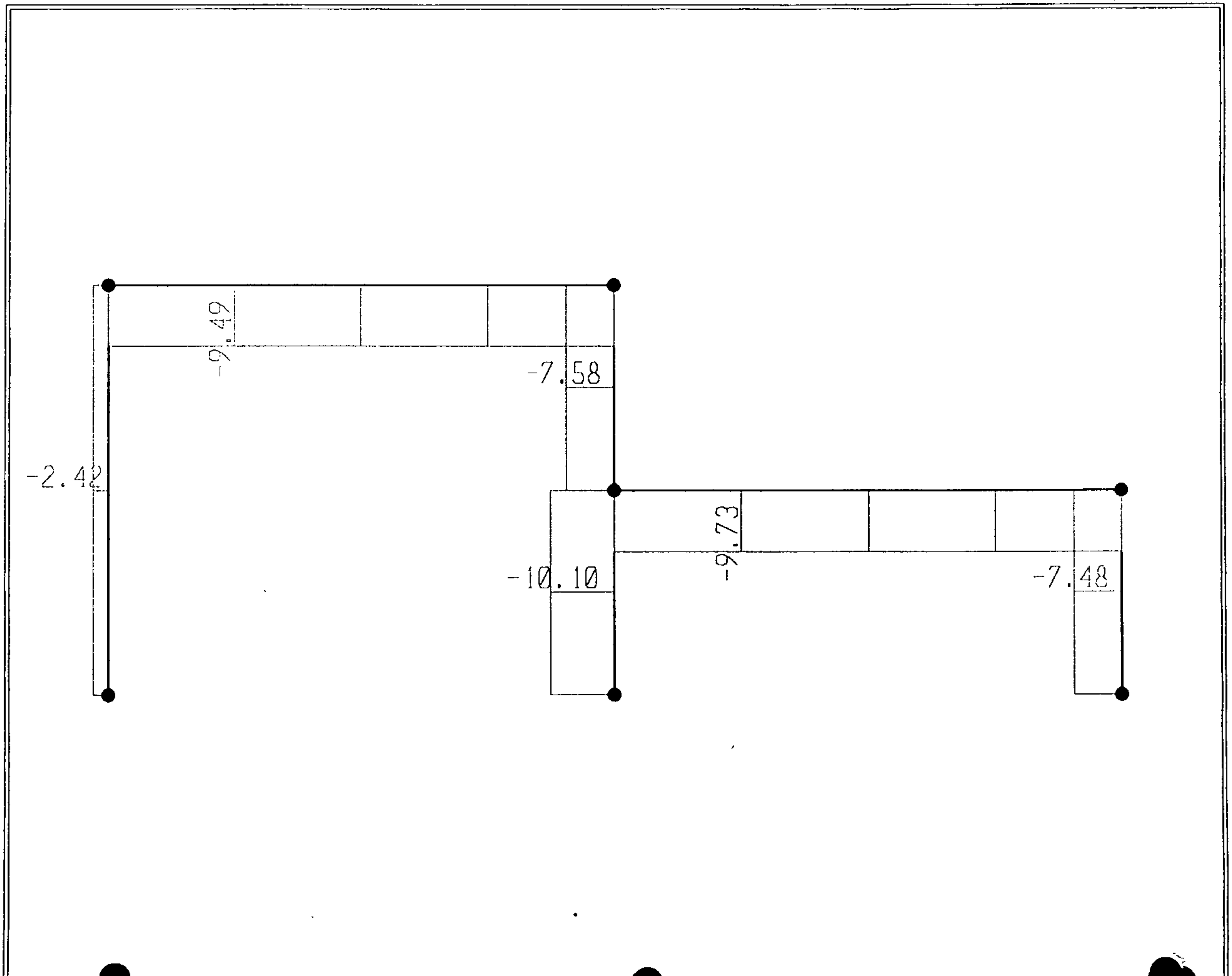

1.00
1.00
1.00
1.00
10.00
Z
Y
X

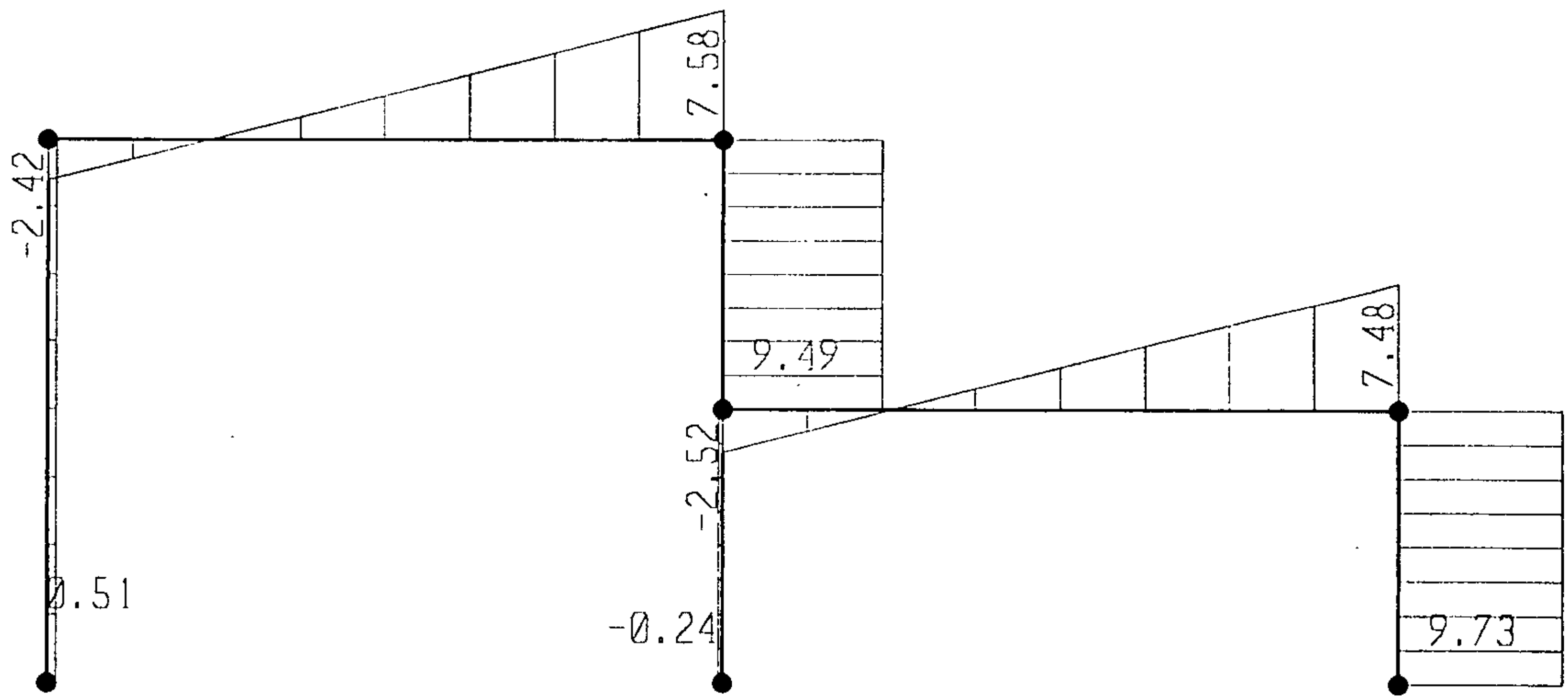
SAP2000

Marzo 18,2000 11:34

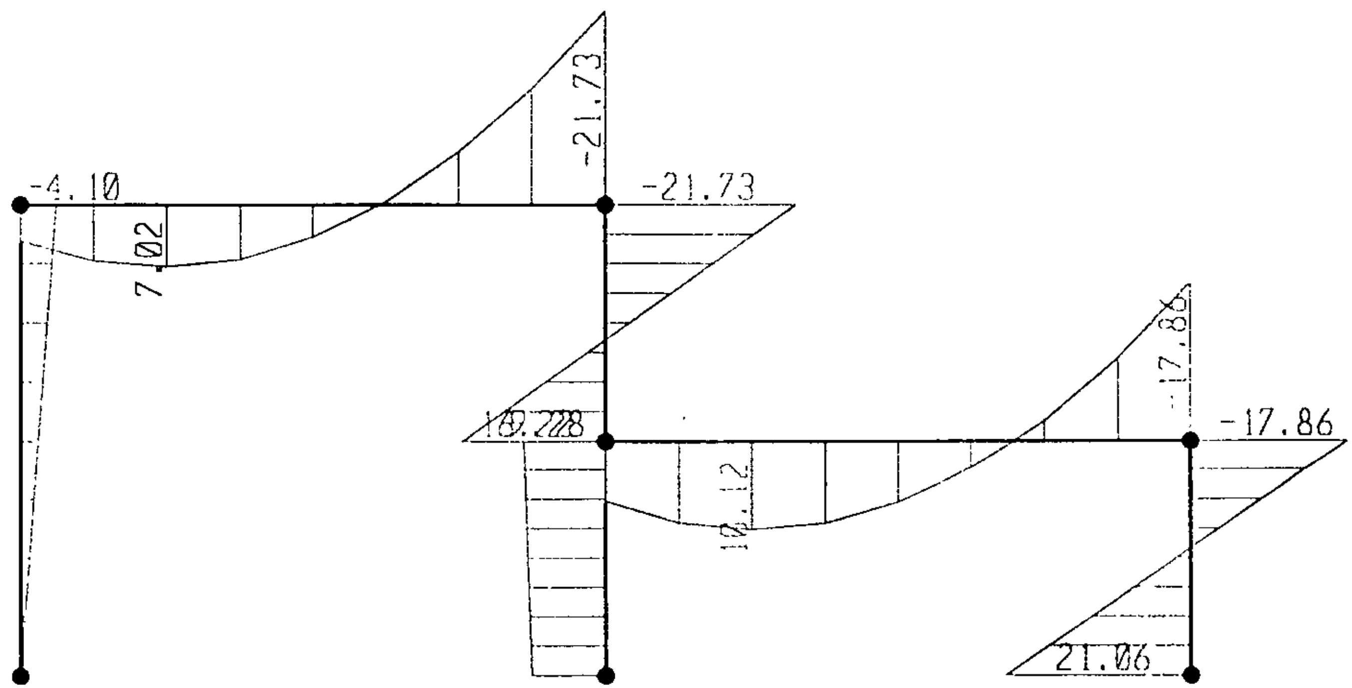


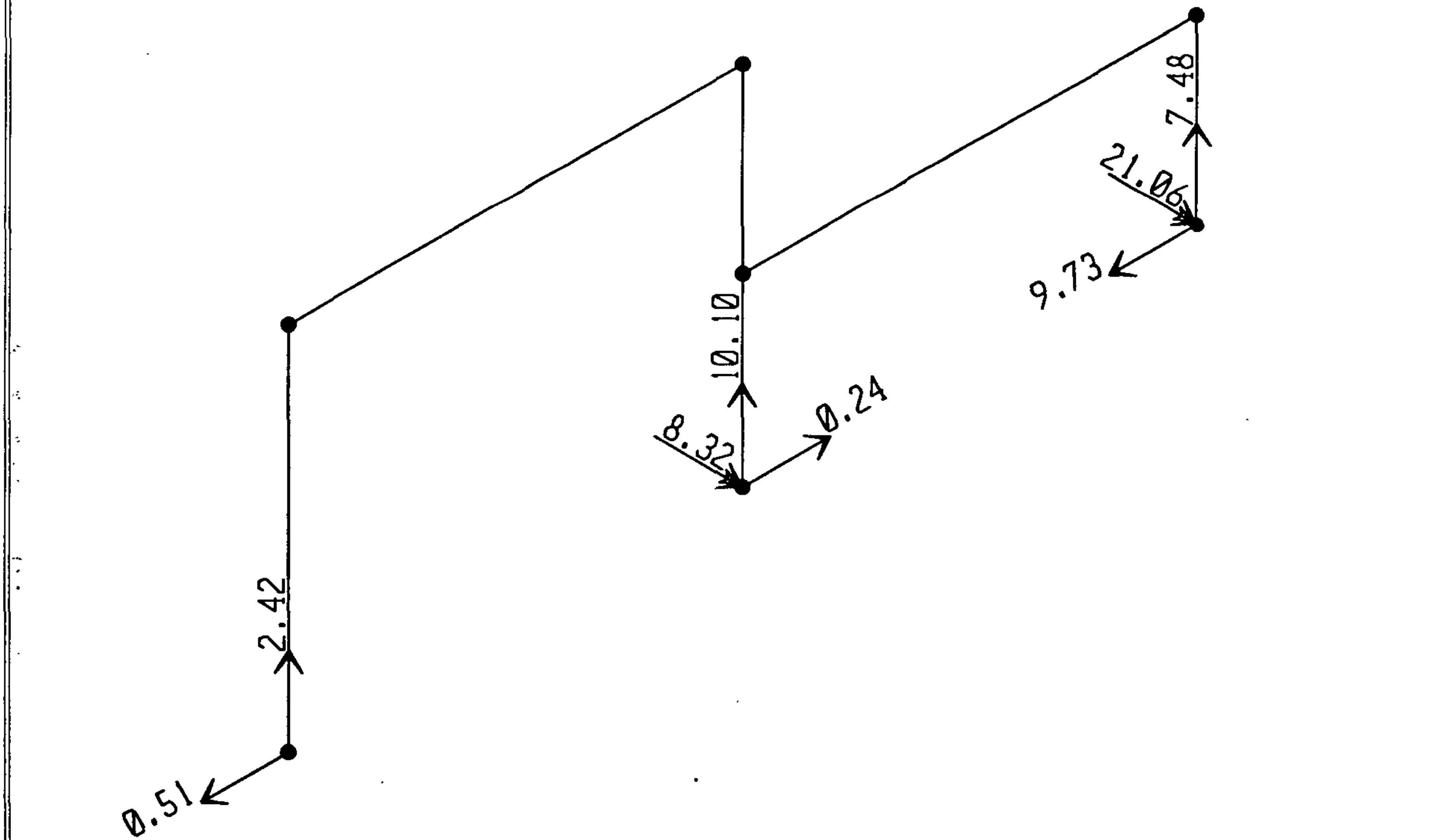
SAP2000 v6.10 - File:ejemplo4 - Deformed Shape (UNICA) - Ton-m Units


-9.49
-7.58
-2.42
-9.73
-10.10
-7.48



SAP2000 v6.10 - File:ejemplo4 - Shear Force 2-2 Diagram (UNICA) - Ton-m Units


-4.10
-21.73
-21.73
7.02
-17.86
-17.86
10.12
21.06



SAP2000 v6.10 - File:ejemplo4 - Restraint Reactions (UNICA) - Ton-m Units

```
; File C:\Mis documentos\curso sap 2000\ejemplo4.s2k saved 3/18/00 11:57:20 in Ton-m

SYSTEM
  DOF=UX,UZ,RY   LENGTH=m   FORCE=Ton   LINES=59

JOINT
  1  X=-10  Y=0  Z=0
  3  X=-10  Y=0  Z=8
  4  X=0  Y=0  Z=0
  5  X=0  Y=0  Z=4
  6  X=0  Y=0  Z=8
  7  X=10  Y=0  Z=0
  8  X=10  Y=0  Z=4

RESTRAINT
  ADD=1  DOF=U1,U2,U3
  ADD=4  DOF=U1,U2,U3,R1,R2,R3
  ADD=7  DOF=U1,U2,U3,R1,R2,R3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=OTRO  IDES=N  M=.2448  W=2.4026
    T=0  E=1000000  U=.2  A=.0000099

FRAME SECTION
  NAME=FS1 MAT=OTRO SH=R T=.5,.25 A=.125 J=1.788127E-03 I=2.604167E-03,6.510417E-04 AS=.1041667,.1041667

FRAME
  1  J=1,3  SEC=FS1  NSEG=2  ANG=0
  3  J=4,5  SEC=FS1  NSEG=2  ANG=0
  4  J=5,6  SEC=FS1  NSEG=2  ANG=0
  5  J=7,8  SEC=FS1  NSEG=2  ANG=0
  8  J=3,6  SEC=FS1  NSEG=4  ANG=0
  9  J=5,8  SEC=FS1  NSEG=4  ANG=0

LOAD
  NAME=UNICA
    TYPE=FORCE
      ADD=3  UX=10
    TYPE=DISTRIBUTED SPAN
      ADD=8  RD=0,1  UZ=-1,-1
      ADD=9  RD=0,1  UZ=-1,-1

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=UNICA
  ELEM=JOINT  TYPE=APPL  LOAD=UNICA
  ELEM=FRAME  TYPE=FORCE  LOAD=UNICA
  ELEM=FRAME  TYPE=JOINTF  LOAD=UNICA

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  -10
  GRID GLOBAL  X "2"  0
  GRID GLOBAL  X "3"  10
  GRID GLOBAL  Y "4"  0
  GRID GLOBAL  Z "5"  0
  GRID GLOBAL  Z "6"  4
  GRID GLOBAL  Z "7"  8
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  FRAMESECTION FS1 NAME REC25X50
  STATICLOAD UNICA TYPE OTHER
END SUPPLEMENTAL DATA
```

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 1
Marzo 18, 2000 11:58

S T A T I C L O A D C A S E S

STATIC CASE	CASE TYPE	SELF WT FACTOR
UNICA	OTHER	0.0000

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 2
Marzo 18, 2000 11:58

M A T E R I A L P R O P E R T Y D A T A

MAT LABEL	MODULUS OF ELASTICITY	POISSON'S RATIO	THERMAL COEFF	WEIGHT PER UNIT VOL	MASS PER UNIT VOL
STEEL	20389020	0.300	1.170E-05	7.833	0.798
CONC	2531051	0.200	9.900E-06	2.403	0.245
OTRO	1000000.000	0.200	9.900E-06	2.403	0.245

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 3
Marzo 18, 2000 11:58

M A T E R I A L D E S I G N D A T A

MAT LABEL	DESIGN CODE	STEEL FY	CONCRETE FC	REBAR FY	CONCRETE FCS	REBAR FYS
STEEL	S	25310.500				
CONC	C		2812.278	42184.180	2812.278	28122.779
OTRO	N					

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 4
Marzo 18, 2000 11:58

F R A M E S E C T I O N P R O P E R T Y D A T A

SECTION LABEL	MAT LABEL	SECTION TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
REC25X50	OTRO		0.500	0.250	0.000	0.000	0.000	0.000

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 5
Marzo 18, 2000 11:58

F R A M E S E C T I O N P R O P E R T Y D A T A

SECTION LABEL	AREA	TORSIONAL INERTIA	MOMENTS OF INERTIA I33	I22	SHEAR AREAS A2	A3
REC25X50	0.125	1.788E-03	2.604E-03	6.510E-04	0.104	0.104

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 6
Marzo 18, 2000 11:58

F R A M E S E C T I O N P R O P E R T Y D A T A

SECTION LABEL	SECTION MODULII S33	S22	PLASTIC MODULII Z33	Z22	RADII OF GYRATION R33	R22
REC25X50	1.042E-02	5.208E-03	1.563E-02	7.813E-03	0.144	7.217E-02

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 7
Marzo 18, 2000 11:58

F R A M E S E C T I O N P R O P E R T Y D A T A

SECTION LABEL	TOTAL WEIGHT	TOTAL MASS
REC25X50	12.013	1.224

SECTION LABEL	MAT LABEL	SHELL TYPE	MEMBRANE THICK	BENDING THICK	MATERIAL ANGLE
SSEC1	CONC	1	1.000	1.000	0.000

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 9
Marzo 18, 2000 11:58

S H E L L   S E C T I O N   P R O P E R T Y   D A T A

SECTION LABEL	TOTAL WEIGHT	TOTAL MASS
SSEC1	0.000	0.000

SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 10
Marzo 18, 2000 11:58

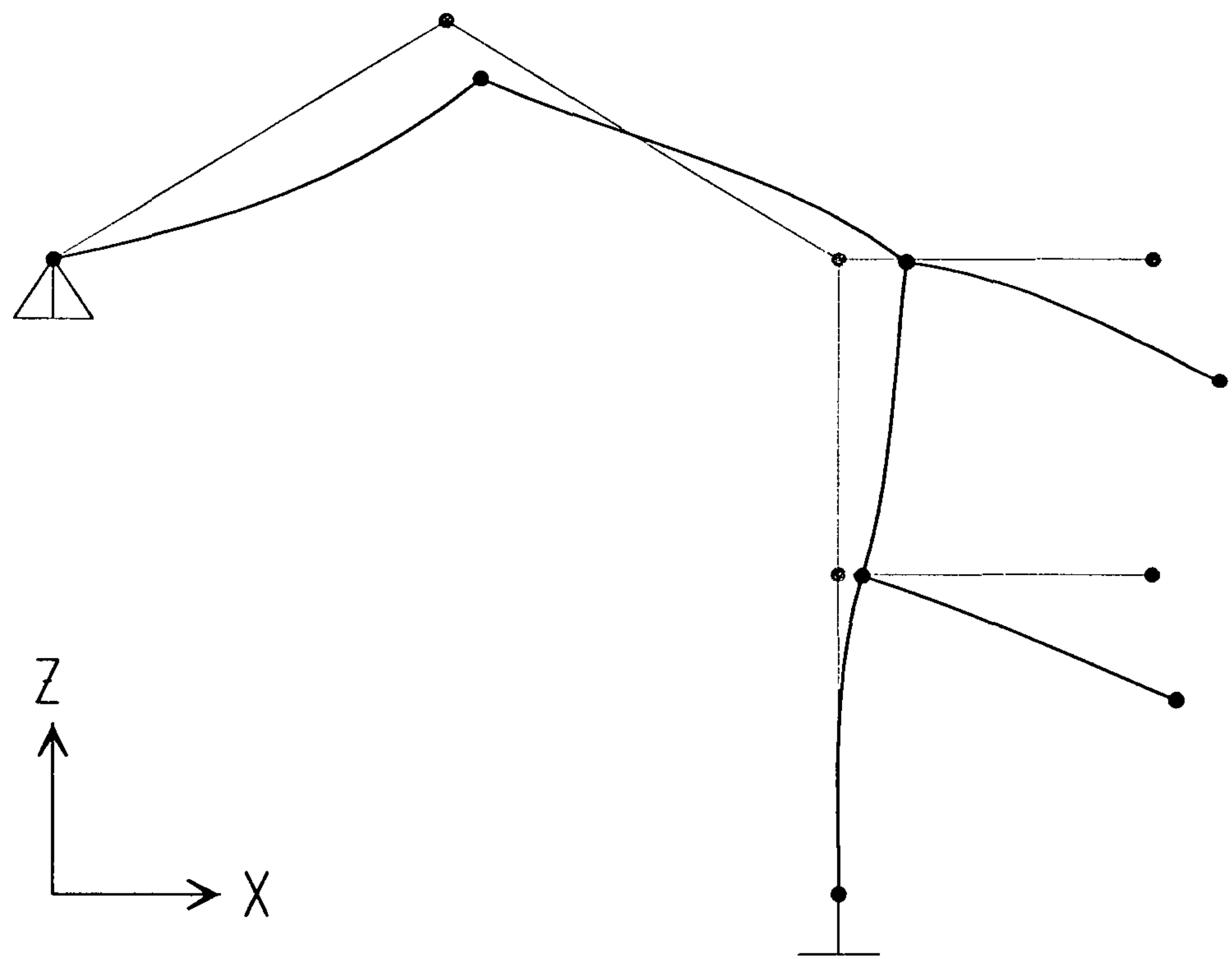
J O I N T   F O R C E S   Load Case UNICA

JOINT	GLOBAL-X	GLOBAL-Y	GLOBAL-Z	GLOBAL-XX	GLOBAL-YY	GLOBAL-ZZ
3	10.000	0.000	0.000	0.000	0.000	0.000

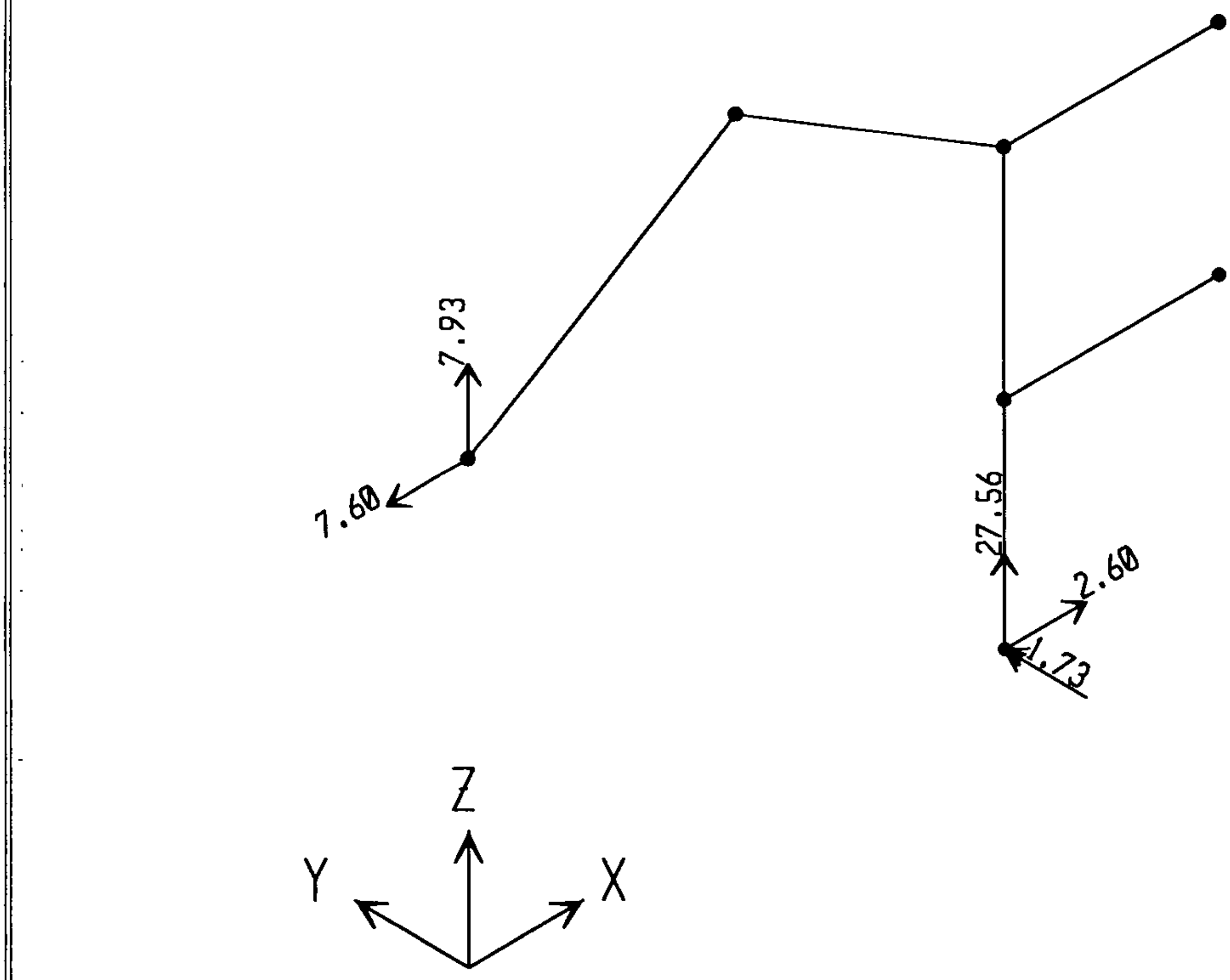
SAP2000 v6.10 File: EJEMPLO4 Ton-m Units PAGE 11
Marzo 18, 2000 11:58

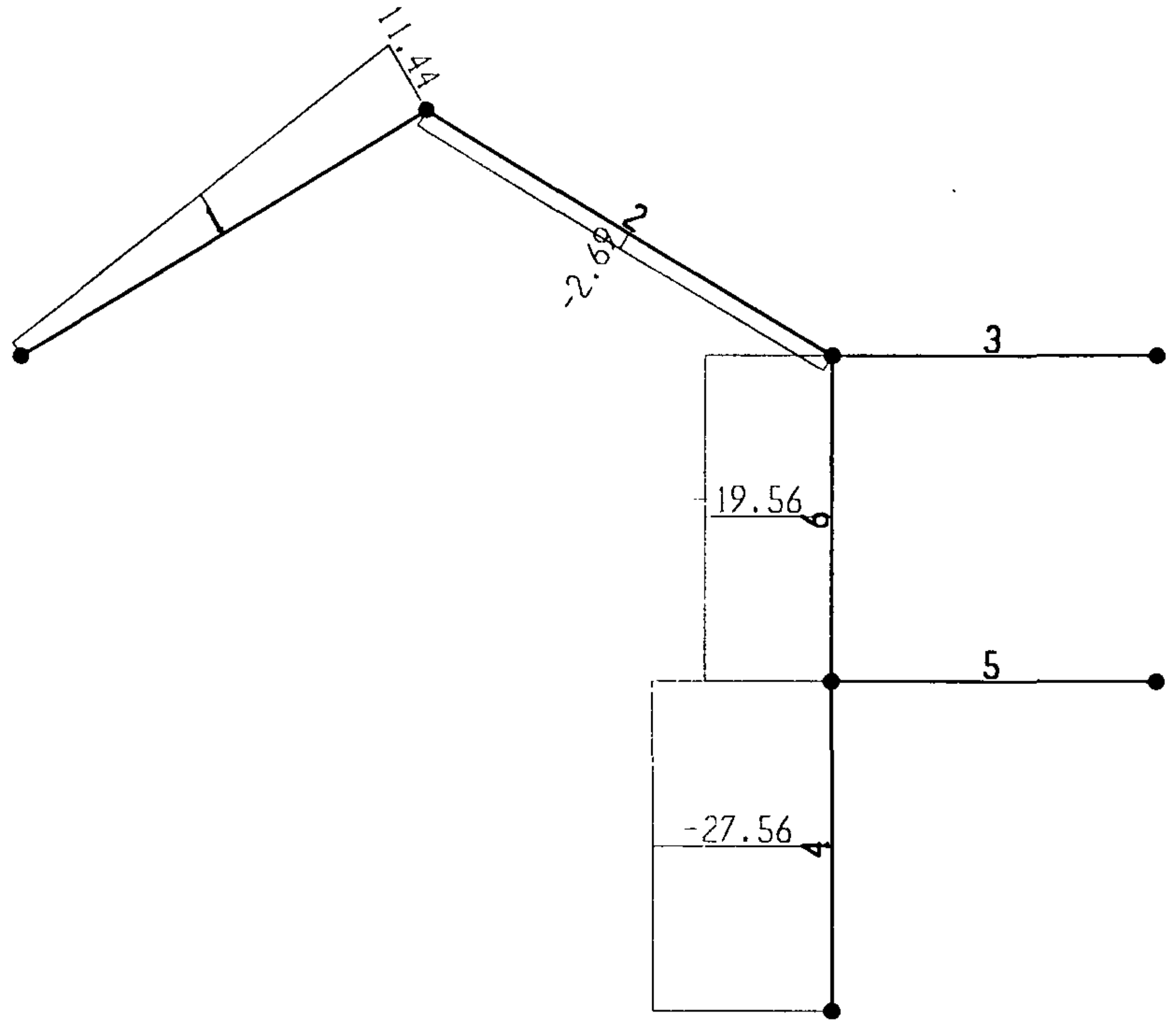
F R A M E   S P A N   D I S T R I B U T E D   L O A D S   Load Case UNICA

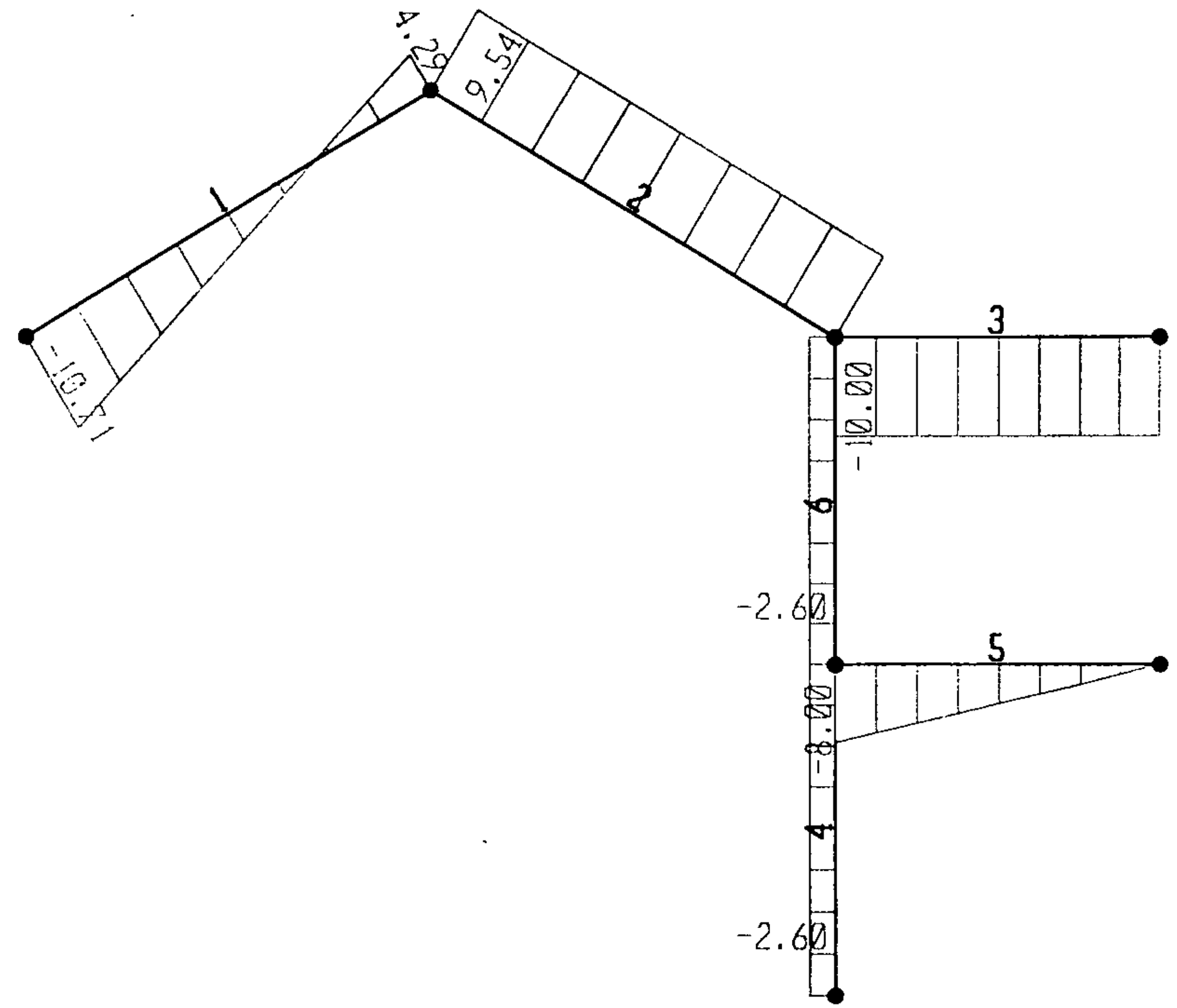
FRAME	TYPE	DIRECTION	DISTANCE-A	VALUE-A	DISTANCE-B	VALUE-B
8	FORCE	GLOBAL-Z	0.0000	-1.0000	1.0000	-1.0000
9	FORCE	GLOBAL-Z	0.0000	-1.0000	1.0000	-1.0000



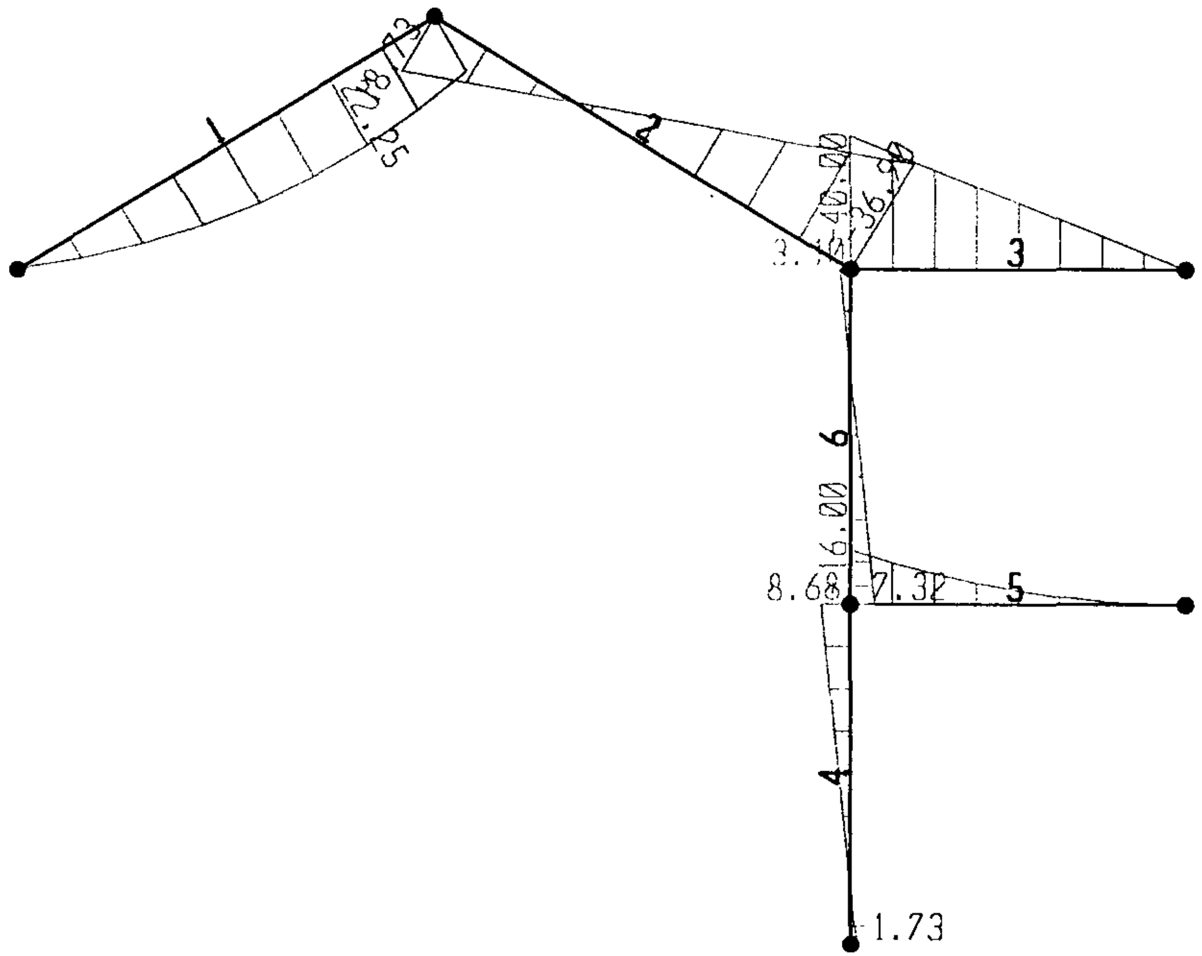
SAP2000 v6.10 - File:ejemplo5 - Deformed Shape (UNICA) - Ton-m Units


7.93
7.60
27.56
2.60
1.73
Z
Y
X


11.44
2
-2.69
3
-19.56
6
5
-27.56
4



SAP2000 v - File:ejemplo5 - Shear Force 2-2 Diagram (UNICA) - Ton-m


1
2
3
4
5
6
82.25
40.00
36.9
3.7
16.00
8.68
7.32
1.73

```
; File C:\Mis documentos\curso sap 2000\ejemplo5.s2k saved 3/18/00 13:43:48 in Ton-m

SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  LINES=59

JOINT
  1  X=0  Y=0  Z=8
  2  X=5  Y=0  Z=11
  3  X=10  Y=0  Z=8
  4  X=14  Y=0  Z=8
  5  X=10  Y=0  Z=0
  6  X=10  Y=0  Z=4
  7  X=14  Y=0  Z=4

RESTRAINT
  ADD=1  DOF=U1,U2,U3
  ADD=5  DOF=U1,U2,U3,R1,R2,R3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=OTRO  IDES=N  M=.2448  W=2.4026
    T=0  E=1000000  U=.2  A=.0000099

FRAME SECTION
  NAME=FS1 MAT=OTRO SH=R T=.3,.3 A=.09 J=1.14075E-03 I=6.750001E-04,6.750001E-04 AS=7.500001E-02,7.500001E-02
  NAME=FS2 MAT=OTRO SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125

FRAME
  1  J=1,2  SEC=FS2  NSEG=2  ANG=0
  2  J=2,3  SEC=FS2  NSEG=2  ANG=0
  3  J=3,4  SEC=FS2  NSEG=4  ANG=0
  4  J=5,6  SEC=FS1  NSEG=2  ANG=0
  5  J=6,7  SEC=FS2  NSEG=4  ANG=0
  6  J=6,3  SEC=FS1  NSEG=2  ANG=0

LOAD
  NAME=UNICA
    TYPE=FORCE
      ADD=2  UX=5
      ADD=4  UZ=-10
    TYPE=DISTRIBUTED SPAN
      ADD=1  RD=0,1  UZ=-3,-3
      ADD=5  RD=0,1  UZ=-2,-2

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=UNICA
  ELEM=JOINT  TYPE=APPL  LOAD=UNICA
  ELEM=FRAME  TYPE=FORCE  LOAD=UNICA
  ELEM=FRAME  TYPE=JOINTF  LOAD=UNICA

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  0
  GRID GLOBAL  X "2"  5
  GRID GLOBAL  X "3"  10
  GRID GLOBAL  X "4"  14
  GRID GLOBAL  Y "5"  0
  GRID GLOBAL  Z "6"  0
  GRID GLOBAL  Z "7"  4
  GRID GLOBAL  Z "8"  8
  GRID GLOBAL  Z "9"  11
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  FRAMESECTION FS1 NAME COL30X30
  FRAMESECTION FS2 NAME REC30X50
  STATICLOAD UNICA TYPE OTHER
END SUPPLEMENTAL DATA
```

S A P 2 0 0 0

Structural Analysis Programs

Version 6.10

18 Mar 2000 13:19:35

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 1
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

S Y S T E M D A T A

```
STEADY STATE LOAD FREQUENCY - - - - - - - -  0.0000E+00

LENGTH UNITS - - - - - - - - - - - - - - - -           M
FORCE  UNITS - - - - - - - - - - - - - - - -         TON

UP DIRECTION - - - - - - - - - - - - - - - -          +Z

GLOBAL DEGREES OF FREEDOM  - - - - - - - - -          UX
                           - - - - - - - - -          UZ
                           - - - - - - - - -          RY

PAGINATION BY  - - - - - - - - - - - - - - -       LINES
NUMBER OF LINES PER PAGE - - - - - - - - - -          59
INCLUDE WARNING MESSAGES IN OUTPUT FILE  - -           Y
```

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 2
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

G E N E R A T E D J O I N T C O O R D I N A T E S

JOINT	X	Y	Z
1	0.000	0.000	8.000
2	5.000	0.000	11.000
3	10.000	0.000	8.000
4	14.000	0.000	8.000
5	10.000	0.000	0.000
6	10.000	0.000	4.000
7	14.000	0.000	4.000

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 3
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

P A T T E R N S

PATTERN	JOINT	VALUE
DEFAULT		

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 4
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

R E S T R A I N T D A T A

JOINT						
1	U1	U2	U3			
5	U1	U2	U3	R1	R2	R3

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 5
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

M A T E R I A L P R O P E R T Y D A T A

MAT LABEL	NUMBER TEMPS	WEIGHT PER UNIT VOL	MASS PER UNIT VOL	DESIGN CODE
STEEL	1	0.7833E+01	0.7981E+00	S
CONC	1	0.2403E+01	0.2448E+00	C
OTRO	1	0.2403E+01	0.2448E+00	N

T E M P E R A T U R E D E P E N D E N T D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	MODULUS OF ELASTICITY E1	E2	E3	SHEAR MODULII G12	G13	G23
STEEL	0.00	0.204E+08	0.204E+08	0.204E+08	0.784E+07	0.784E+07	0.784E+07
CONC	0.00	0.253E+07	0.253E+07	0.253E+07	0.105E+07	0.105E+07	0.105E+07
OTRO	0.00	0.100E+07	0.100E+07	0.100E+07	0.417E+06	0.417E+06	0.417E+06

T E M P E R A T U R E D E P E N D E N T D A T A

THERMAL EXPANSION COEFFICIENTS

MAT LABEL	TEMP	COEFFICIENTS OF THERMAL EXPANSION A1	A2	A3	A12	A13	A23
STEEL	0.00	0.117E-04	0.117E-04	0.117E-04	0.000E+00	0.000E+00	0.000E+00
CONC	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00
OTRO	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00

T E M P E R A T U R E D E P E N D E N T D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	POISSONS RATIO U12	U13	U23	U14	U24	U34	U15	U25	U35	U45	U16	U26	U36	U46	U56
STEEL	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONC	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
OTRO	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

MATERIAL PROPERTIES

MAT LABEL	TEMP	YIELD FY
CONC	0.00	36.00

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	SHAPE TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
FS1	R	0.300	0.300				
FS2	R	0.500	0.300				

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 11
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	AXIAL AREA	TORSIONAL CONSTANT	MOMENTS OF INERTIA I33	I22	SHEAR AREAS A2	A3
FS1	0.900E-01	0.114E-02	0.675E-03	0.675E-03	0.750E-01	0.750E-01
FS2	0.150E+00	0.282E-02	0.313E-02	0.113E-02	0.125E+00	0.125E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 12
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	MAT LABEL	ADDITIONAL MASS PER LENGTH	ADDITIONAL WEIGHT PER LENGTH
FS1	OTRO	0.000E+00	0.000E+00
FS2	OTRO	0.000E+00	0.000E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 13
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

F R A M E   E L E M E N T   D A T A

ELEMENT LABEL	JOINT END-I	JOINT END-J	ELEMENT LENGTH	END-OFFSET-LENGTHS END-I	END-J	RIGID-END FACTOR	NUMBER OF SEGMENTS
1	1	2	5.831	0.000	0.000	0.0000	2
2	2	3	5.831	0.000	0.000	0.0000	2
3	3	4	4.000	0.000	0.000	0.0000	4
4	5	6	4.000	0.000	0.000	0.0000	2
5	6	7	4.000	0.000	0.000	0.0000	4
6	6	3	4.000	0.000	0.000	0.0000	2

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 14
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

F R A M E   E L E M E N T   D A T A

ELEMENT LABEL	SECTION LABEL	LOCAL PLANE	COORD SYSTEM	PLN 1ST	PLN 2ND	PLANE JOINTA	PLANE JOINTB	COORD ANGLE
1	FS2	12	0	+Z	+X	0	0	0.00
2	FS2	12	0	+Z	+X	0	0	0.00
3	FS2	12	0	+Z	+X	0	0	0.00
4	FS1	12	0	+Z	+X	0	0	0.00
5	FS2	12	0	+Z	+X	0	0	0.00
6	FS1	12	0	+Z	+X	0	0	0.00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 15
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

T O T A L   W E I G H T S   A N D   M A S S E S

SECTION LABEL	WEIGHT	MASS
FS1	1.7299	0.1763
FS2	7.0860	0.7220
TOTAL	8.8158	0.8982

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 16
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo5.EKO

L O A D   C O N D I T I O N   UNICA

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.0000E+00

JOINT FORCES IN LOCAL COORDINATES

JOINT LABEL	FORCE 1	FORCE 2	FORCE 3	MOMENT 1	MOMENT 2	MOMENT 3
2	0.500E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
4	0.000E+00	0.000E+00	-0.100E+02	0.000E+00	0.000E+00	0.000E+00

DISTRIBUTED SPAN LOADS ON FRAME ELEMENTS

ELEMENT LABEL	LOC DOF	DISTANCE AT START	DISTANCE AT END	FORCE AT START	FORCE AT END	MOMENT AT START	MOMENT AT END
1	U1	0.000E+00	0.100E+01	-0.154E+01	-0.154E+01		
1	U2	0.000E+00	0.100E+01	-0.257E+01	-0.257E+01		
5	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		

O U T P U T S E L E C T I O N

DISPLACEMENTS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
UNICA					

APPLIED AND INTERNAL LOADS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
UNICA					

INTERNAL FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
UNICA					

JOINT FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
UNICA					

I N P U T C O M P L E T E

S A P 2 0 0 0 (R)

Structural Analysis Programs

Version E6.10

It is the responsibility of the user to verify all
results produced by this program

18 Mar 2000 13:19:37

PROGRAM SAP2000 - VERSION E6.10 FILE:EJEMPLO5.OUT
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED PAGE 1

D I S P L A C E M E N T   D E G R E E S   O F   F R E E D O M

(A) = Active DOF, equilibrium equation
(-) = Restrained DOF, reaction computed
(+) = Constrained DOF
( ) = Null DOF

JOINTS	UX	UY	UZ	RX	RY	RZ
1	-		-		A	
2 TO 4	A		A		A	
5	-		-		-	
6 TO 7	A		A		A	

PROGRAM SAP2000 - VERSION E6.10 FILE:EJEMPLO5.OUT
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED PAGE 2

J O I N T   D I S P L A C E M E N T S

TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

LOAD UNICA ------------------

JOINT	UX	UZ	RY
1	.000000	.000000	0.022226
2	0.029221	-0.048177	-0.008848
3	0.056748	-0.002094	0.008103
4	0.056748	-0.103539	0.033703
5	.000000	.000000	.000000
6	0.020272	-0.001225	0.020592
7	0.020272	-0.104378	0.027418

PROGRAM SAP2000 - VERSION E6.10 FILE:EJEMPLO5.OUT
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED PAGE 3

A P P L I E D   L O A D S

FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOAD UNICA ------------------

JOINT	FX	FZ	MY
1	-8.33E-16	-8.746428	7.288690
2	5.000000	-8.746428	-7.288690
4	.000000	-10.000000	.000000
6	.000000	-4.000000	2.666667
7	.000000	-4.000000	-2.666667

PROGRAM SAP2000 - VERSION E6.10 FILE:EJEMPLO5.OUT
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED PAGE 4

G L O B A L   F O R C E   B A L A N C E

TOTAL FORCE AND MOMENT AT THE ORIGIN, IN GLOBAL COORDINATES

LOAD UNICA ------------------

	FX	FY	FZ	MX	MY	MZ
APPLIED	5.000000	.000000	-35.492856	.000000	334.732139	.000000
REACTNS	-5.000000	.000000	35.492856	.000000	-334.732139	.000000
TOTAL	2.90E-13	.000000	-1.07E-13	.000000	1.88E-12	.000000

PROGRAM SAP2000 - VERSION E6.10 FILE:EJEMPLO5.OUT
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED PAGE 5

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 1 ------------------

LOAD UNICA ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
1	-7.604415	.000000	7.929773	.000000	-3.55E-15	.000000
2	7.604415	.000000	9.563083	.000000	-18.729972	.000000

ELEM 2 ------------------

LOAD UNICA ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	-2.604415	.000000	-9.563083	.000000	18.729972	.000000
3	2.604415	.000000	9.563083	.000000	36.898686	.000000

ELEM 3 ------------------

LOAD UNICA ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	1.13E-14	.000000	10.000000	.000000	-40.000000	.000000
4	-1.13E-14	.000000	-10.000000	.000000	-6.06E-15	.000000

ELEM 4 ------------------

LOAD UNICA ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
5	2.604415	.000000	27.563083	.000000	1.734009	.000000
6	-2.604415	.000000	-27.563083	.000000	8.683652	.000000

ELEM 5 ------------------

LOAD UNICA ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
6	2.30E-13	.000000	8.000000	.000000	-16.000000	.000000
7	-2.30E-13	.000000	-6.66E-15	.000000	9.33E-15	.000000

ELEM 6 ------------------

LOAD UNICA ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
6	2.604415	.000000	19.563083	.000000	7.316348	.000000
3	-2.604415	.000000	-19.563083	.000000	3.101314	.000000

PROGRAM SAP2000 - VERSION E6.10 FILE:EJEMPLO5.OUT
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED PAGE 6

F R A M E   E L E M E N T   I N T E R N A L   F O R C E S

ELEM 1 ------------------ LENGTH = 5.830952

LOAD UNICA ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	2.440898	-10.712164	.000000	.000000	.000000	-2.45E-15
0.50000	6.940898	-3.212164	.000000	.000000	.000000	20.298021
1.00000	11.440898	4.287836	.000000	.000000	.000000	18.729972

ELEM 2 ------------------ LENGTH = 5.830952

LOAD UNICA ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-2.686898	9.540236	.000000	.000000	.000000	18.729972
0.50000	-2.686898	9.540236	.000000	.000000	.000000	-9.084357
1.00000	-2.686898	9.540236	.000000	.000000	.000000	-36.898686

ELEM 3 ------------------ LENGTH = 4.000000

LOAD UNICA ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-1.13E-14	-10.000000	.000000	.000000	.000000	-40.000000
0.25000	-1.13E-14	-10.000000	.000000	.000000	.000000	-30.000000
0.50000	-1.13E-14	-10.000000	.000000	.000000	.000000	-20.000000
0.75000	-1.13E-14	-10.000000	.000000	.000000	.000000	-10.000000
1.00000	-1.13E-14	-10.000000	.000000	.000000	.000000	3.55E-15

ELEM 4 ------------------ LENGTH = 4.000000

LOAD UNICA ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-27.563083	-2.604415	.000000	.000000	.000000	-1.734009
0.50000	-27.563083	-2.604415	.000000	.000000	.000000	3.474822
1.00000	-27.563083	-2.604415	.000000	.000000	.000000	8.683652

ELEM 5 ------------------ LENGTH = 4.000000

LOAD UNICA ------------------

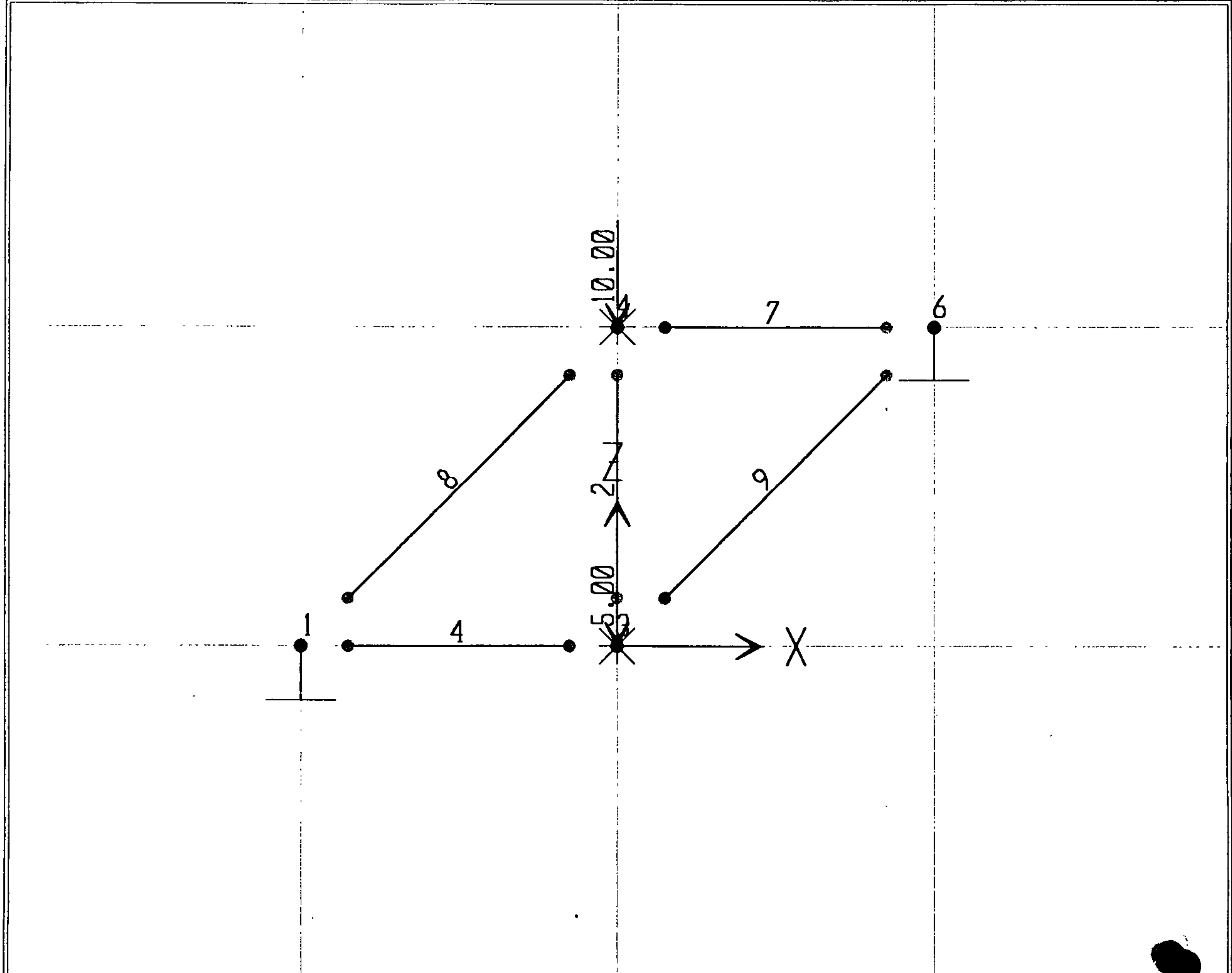
REL DIST	P	V2	V3	T	M2	M3
0.00000	-2.30E-13	-8.000000	.000000	.000000	.000000	-16.000000
0.25000	-2.30E-13	-6.000000	.000000	.000000	.000000	-9.000000
0.50000	-2.30E-13	-4.000000	.000000	.000000	.000000	-4.000000
0.75000	-2.30E-13	-2.000000	.000000	.000000	.000000	-1.000000
1.00000	-2.30E-13	-5.33E-15	.000000	.000000	.000000	-1.78E-14

F R A M E   E L E M E N T   I N T E R N A L   F O R C E S

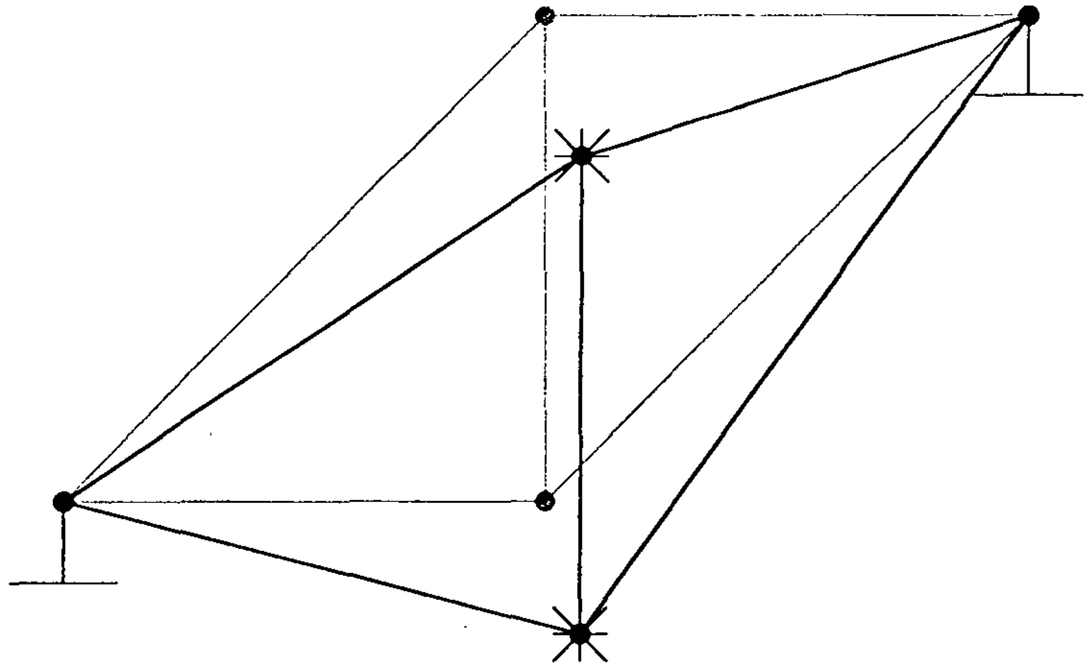
ELEM 6 ------------------ LENGTH = 4.000000

LOAD UNICA ------------------

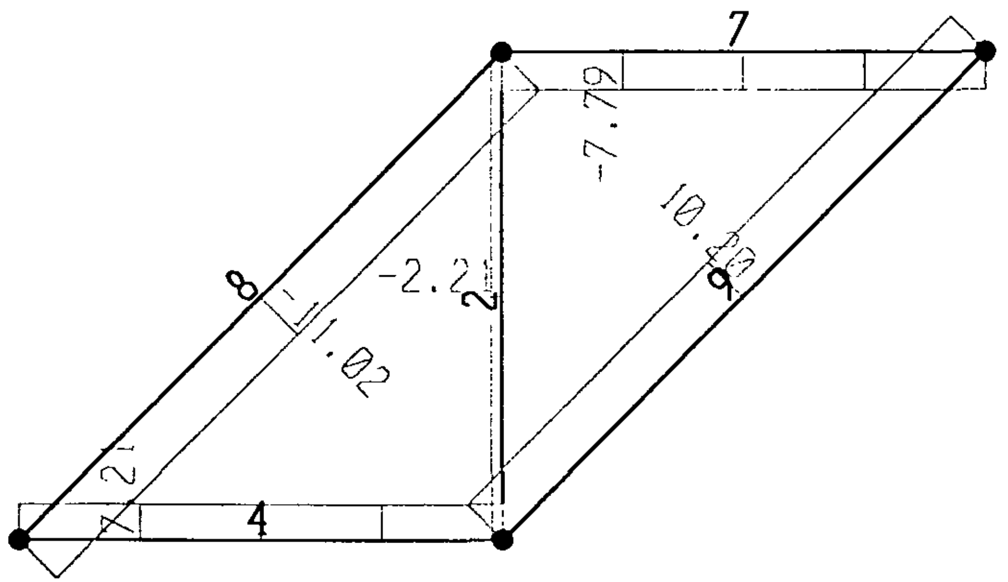
REL DIST	P	V2	V3	T	M2	M3
0.00000	-19.563083	-2.604415	.000000	.000000	.000000	-7.316348
0.50000	-19.563083	-2.604415	.000000	.000000	.000000	-2.107517
1.00000	-19.563083	-2.604415	.000000	.000000	.000000	3.101314



SAP2000 v6.10 - File:ejemplo6 - Joint Loads (VERTICAL) - Ton-m Units



SAP2000 v6.10 - File:ejemplo6 - Deformed Shape (VERTICAL) - Ton-m Units


7
-7.79
10.40
8
-2.21
2
-11.02
9
21
4

```
; File C:\Mis documentos\curso sap 2000\ejemplo6.s2k saved 3/18/00 14:39:08 in Ton-m

SYSTEM
  DOF=UX,UZ,RY   LENGTH=m   FORCE=Ton   LINES=59

JOINT
  1  X=-2  Y=0  Z=0
  3  X=0  Y=0  Z=0
  4  X=0  Y=0  Z=2
  6  X=2  Y=0  Z=2

RESTRAINT
  ADD=1  DOF=U1,U2,U3,R1,R2,R3
  ADD=6  DOF=U1,U2,U3,R1,R2,R3
  ADD=3  DOF=U2,R1,R2,R3
  ADD=4  DOF=U2,R1,R2,R3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=RARO  IDES=N  M=.2448  W=2.4026
    T=0  E=1000000  U=.2  A=.0000099

FRAME SECTION
  NAME=FSEC1 MAT=STEEL SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125

FRAME
  2  J=3,4  SEC=FSEC1  NSEG=2  ANG=0  IREL=R3  JREL=R3
  4  J=1,3  SEC=FSEC1  NSEG=4  ANG=0  IREL=R3  JREL=R3
  7  J=4,6  SEC=FSEC1  NSEG=4  ANG=0  IREL=R3  JREL=R3
  8  J=1,4  SEC=FSEC1  NSEG=2  ANG=0  IREL=R3  JREL=R3
  9  J=3,6  SEC=FSEC1  NSEG=2  ANG=0  IREL=R3  JREL=R3

LOAD
  NAME=VERTICAL
    TYPE=FORCE
      ADD=4  UZ=-10
      ADD=3  UZ=-5

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=VERTICAL
  ELEM=JOINT  TYPE=APPL  LOAD=VERTICAL
  ELEM=FRAME  TYPE=FORCE  LOAD=VERTICAL
  ELEM=FRAME  TYPE=JOINTF  LOAD=VERTICAL

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  -2
  GRID GLOBAL  X "2"  0
  GRID GLOBAL  X "3"  2
  GRID GLOBAL  Y "4"  0
  GRID GLOBAL  Z "5"  0
  GRID GLOBAL  Z "6"  2
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  STATICLOAD VERTICAL TYPE OTHER
END SUPPLEMENTAL DATA
```

S A P 2 0 0 0

Structural Analysis Programs

Version 6.10

It is the responsibility of the user to verify all
results produced by this program

18 Mar 2000 14:26:16

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 1
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

S Y S T E M D A T A

```
STEADY STATE LOAD FREQUENCY  - - - - - - - -  0.0000E+00

LENGTH UNITS - - - - - - - - - - - - - - - - -          M
FORCE  UNITS - - - - - - - - - - - - - - - - -        TON

UP DIRECTION - - - - - - - - - - - - - - - - -         +Z

GLOBAL DEGREES OF FREEDOM  - - - - - - - - - -         UX
                           - - - - - - - - - -         UZ
                           - - - - - - - - - -         RY

PAGINATION BY  - - - - - - - - - - - - - - - -      LINES
NUMBER OF LINES PER PAGE - - - - - - - - - - -         59
INCLUDE WARNING MESSAGES IN OUTPUT FILE  - -            Y
```

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 2
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

G E N E R A T E D J O I N T C O O R D I N A T E S

JOINT	X	Y	Z
1	-2.000	0.000	0.000
3	0.000	0.000	0.000
4	0.000	0.000	2.000
6	2.000	0.000	2.000

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 3
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

P A T T E R N S

PATTERN	JOINT	VALUE
DEFAULT		

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 4
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

R E S T R A I N T D A T A

JOINT						
1	U1	U2	U3	R1	R2	R3
3		U2		R1	R2	R3
4		U2		R1	R2	R3
6	U1	U2	U3	R1	R2	R3

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 5
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

M A T E R I A L P R O P E R T Y D A T A

MAT LABEL	NUMBER TEMPS	WEIGHT PER UNIT VOL	MASS PER UNIT VOL	DESIGN CODE
STEEL	1	0.7833E+01	0.7981E+00	S
CONC	1	0.2403E+01	0.2448E+00	C
RARO	1	0.2403E+01	0.2448E+00	N

T E M P E R A T U R E D E P E N D E N T D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	MODULUS OF ELASTICITY E1	E2	E3	SHEAR MODULII G12	G13	G23
STEEL	0.00	0.204E+08	0.204E+08	0.204E+08	0.784E+07	0.784E+07	0.784E+07
CONC	0.00	0.253E+07	0.253E+07	0.253E+07	0.105E+07	0.105E+07	0.105E+07
RARO	0.00	0.100E+07	0.100E+07	0.100E+07	0.417E+06	0.417E+06	0.417E+06

T E M P E R A T U R E D E P E N D E N T D A T A

THERMAL EXPANSION COEFFICIENTS

MAT LABEL	TEMP	COEFFICIENTS OF THERMAL EXPANSION A1	A2	A3	A12	A13	A23
STEEL	0.00	0.117E-04	0.117E-04	0.117E-04	0.000E+00	0.000E+00	0.000E+00
CONC	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00
RARO	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00

T E M P E R A T U R E D E P E N D E N T D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	POISSONS RATIO U12	U13	U23	U14	U24	U34	U15	U25	U35	U45	U16	U26	U36	U46	U56
STEEL	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONC	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
RARO	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

MATERIAL PROPERTIES

MAT LABEL	TEMP	YIELD FY
CONC	0.00	36.00

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	SHAPE TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
FSEC1	R	0.500	0.300				

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	AXIAL AREA	TORSIONAL CONSTANT	MOMENTS OF INERTIA I33	MOMENTS OF INERTIA I22	SHEAR AREAS A2	SHEAR AREAS A3
FSEC1	0.150E+00	0.282E-02	0.313E-02	0.113E-02	0.125E+00	0.125E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 12
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	MAT LABEL	ADDITIONAL MASS PER LENGTH	ADDITIONAL WEIGHT PER LENGTH
FSEC1	STEEL	0.000E+00	0.000E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 13
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

F R A M E   E L E M E N T   D A T A

ELEMENT LABEL	JOINT END-I	JOINT END-J	ELEMENT LENGTH	END-OFFSET-LENGTHS END-I	END-OFFSET-LENGTHS END-J	RIGID-END FACTOR	NUMBER OF SEGMENTS
2	3	4	2.000	0.000	0.000	0.0000	2
4	1	3	2.000	0.000	0.000	0.0000	4
7	4	6	2.000	0.000	0.000	0.0000	4
8	1	4	2.828	0.000	0.000	0.0000	2
9	3	6	2.828	0.000	0.000	0.0000	2

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 14
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

F R A M E   E L E M E N T   D A T A

ELEMENT LABEL	SECTION LABEL	LOCAL PLANE	COORD SYSTEM	PLN 1ST	PLN 2ND	PLANE JOINTA	PLANE JOINTB	COORD ANGLE
2	FSEC1	12	0	+Z	+X	0	0	0.00
4	FSEC1	12	0	+Z	+X	0	0	0.00
7	FSEC1	12	0	+Z	+X	0	0	0.00
8	FSEC1	12	0	+Z	+X	0	0	0.00
9	FSEC1	12	0	+Z	+X	0	0	0.00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 15
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

F R A M E   E L E M E N T   D A T A

ELEMENT LABEL	END-I RELEASE CODES	END-J RELEASE CODES
2	R3	R3
4	R3	R3
7	R3	R3
8	R3	R3
9	R3	R3

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 16
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo6.EKO

T O T A L   W E I G H T S   A N D   M A S S E S

SECTION LABEL	WEIGHT	MASS
FSEC1	13.6969	1.3956
TOTAL	13.6969	1.3956

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 17

L O A D   C O N D I T I O N   VERTICAL

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.0000E+00

JOINT FORCES IN LOCAL COORDINATES

JOINT LABEL	FORCE 1	FORCE 2	FORCE 3	MOMENT 1	MOMENT 2	MOMENT 3
3	0.000E+00	0.000E+00	-0.500E+01	0.000E+00	0.000E+00	0.000E+00
4	0.000E+00	0.000E+00	-0.100E+02	0.000E+00	0.000E+00	0.000E+00

O U T P U T   S E L E C T I O N

DISPLACEMENTS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERTICAL					

APPLIED AND INTERNAL LOADS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERTICAL					

INTERNAL FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERTICAL					

JOINT FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERTICAL					

I N P U T   C O M P L E T E

S A P 2 0 0 0 (R)

Structural Analysis Programs

Version E6.10

18 Mar 2000 14:26:17

D I S P L A C E M E N T   D E G R E E S   O F   F R E E D O M

(A) = Active DOF, equilibrium equation
(-) = Restrained DOF, reaction computed
(+) = Constrained DOF
( ) = Null DOF

JOINTS	UX	UY	UZ	RX	RY	RZ
1	-		-		-	
3 TO 4	A		A		-	
6	-		-		-	

J O I N T   D I S P L A C E M E N T S

TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

LOADVERTICAL ------------------

JOINT	UX	UZ	RY
1	.000000	.000000	.000000
3	4.72E-06	-1.81E-05	.000000
4	5.09E-06	-1.95E-05	.000000
6	.000000	.000000	.000000

A P P L I E D   L O A D S

FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOADVERTICAL ------------------

JOINT	FX	FZ	MY
3	.000000	-5.000000	.000000
4	.000000	-10.000000	.000000

G L O B A L   F O R C E   B A L A N C E

TOTAL FORCE AND MOMENT AT THE ORIGIN, IN GLOBAL COORDINATES

LOADVERTICAL ------------------

	FX	FY	FZ	MX	MY	MZ
APPLIED	.000000	.000000	-15.000000	.000000	.000000	.000000
REACTNS	1.78E-15	.000000	15.000000	.000000	1.78E-15	.000000
TOTAL	1.78E-15	.000000	.000000	.000000	1.78E-15	.000000

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 2 ------------------

LOADVERTICAL ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	.000000	.000000	2.211211	.000000	.000000	.000000
4	.000000	.000000	-2.211211	.000000	.000000	.000000

ELEM 4 -----------------

LOADVERTICAL -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
1	-7.211211	.000000	.000000	.000000	.000000	.000000
3	7.211211	.000000	.000000	.000000	.000000	.000000

ELEM 7 -----------------

LOADVERTICAL -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
4	7.788789	.000000	.000000	.000000	.000000	.000000
6	-7.788789	.000000	.000000	.000000	.000000	.000000

ELEM 8 -----------------

LOADVERTICAL -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
1	7.788789	.000000	7.788789	.000000	.000000	.000000
4	-7.788789	.000000	-7.788789	.000000	.000000	.000000

ELEM 9 -----------------

LOADVERTICAL -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	-7.211211	.000000	-7.211211	.000000	.000000	.000000
6	7.211211	.000000	7.211211	.000000	.000000	.000000

F R A M E   E L E M E N T   I N T E R N A L   F O R C E S

ELEM 2 ----------------- LENGTH = 2.000000

LOADVERTICAL -----------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-2.211211	.000000	.000000	.000000	.000000	.000000
0.50000	-2.211211	.000000	.000000	.000000	.000000	.000000
1.00000	-2.211211	.000000	.000000	.000000	.000000	.000000

ELEM 4 ----------------- LENGTH = 2.000000

LOADVERTICAL -----------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	7.211211	.000000	.000000	.000000	.000000	.000000
0.25000	7.211211	.000000	.000000	.000000	.000000	.000000
0.50000	7.211211	.000000	.000000	.000000	.000000	.000000
0.75000	7.211211	.000000	.000000	.000000	.000000	.000000
1.00000	7.211211	.000000	.000000	.000000	.000000	.000000

ELEM 7 ----------------- LENGTH = 2.000000

LOADVERTICAL -----------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-7.788789	.000000	.000000	.000000	.000000	.000000
0.25000	-7.788789	.000000	.000000	.000000	.000000	.000000
0.50000	-7.788789	.000000	.000000	.000000	.000000	.000000
0.75000	-7.788789	.000000	.000000	.000000	.000000	.000000
1.00000	-7.788789	.000000	.000000	.000000	.000000	.000000

ELEM 8 ----------------- LENGTH = 2.828427

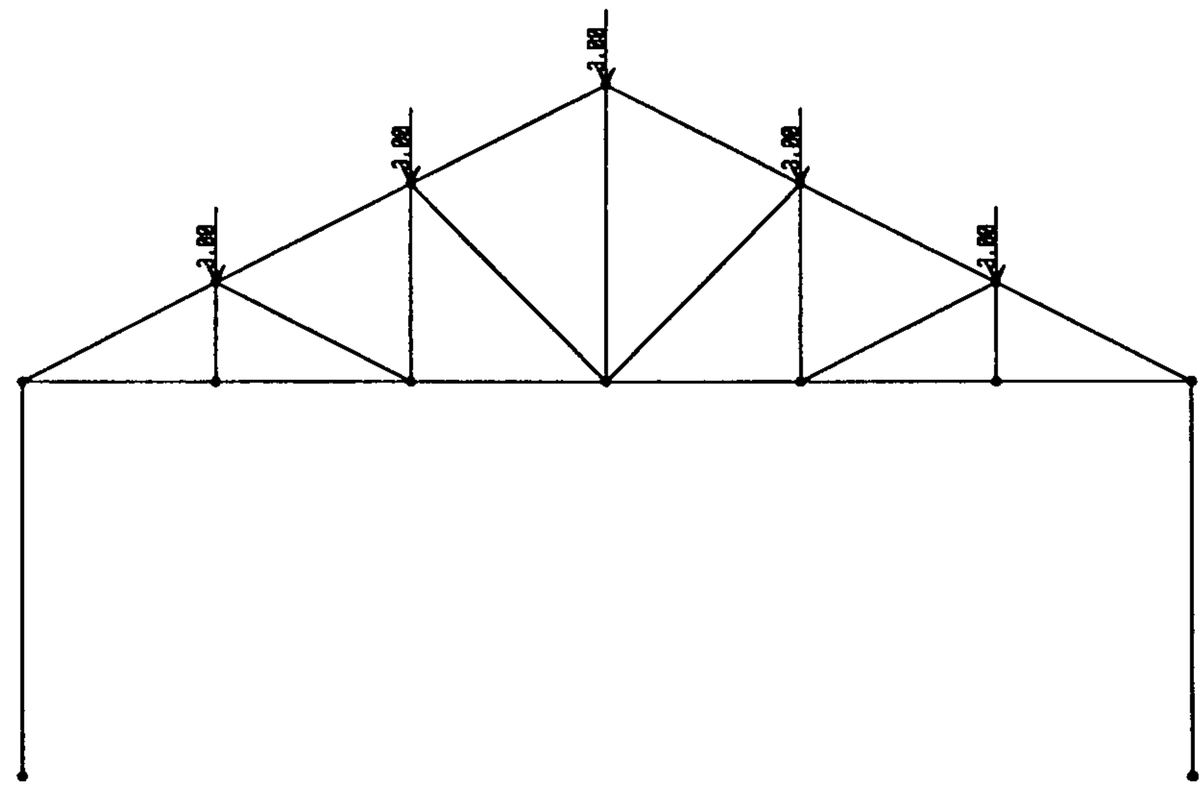
LOADVERTICAL -----------------

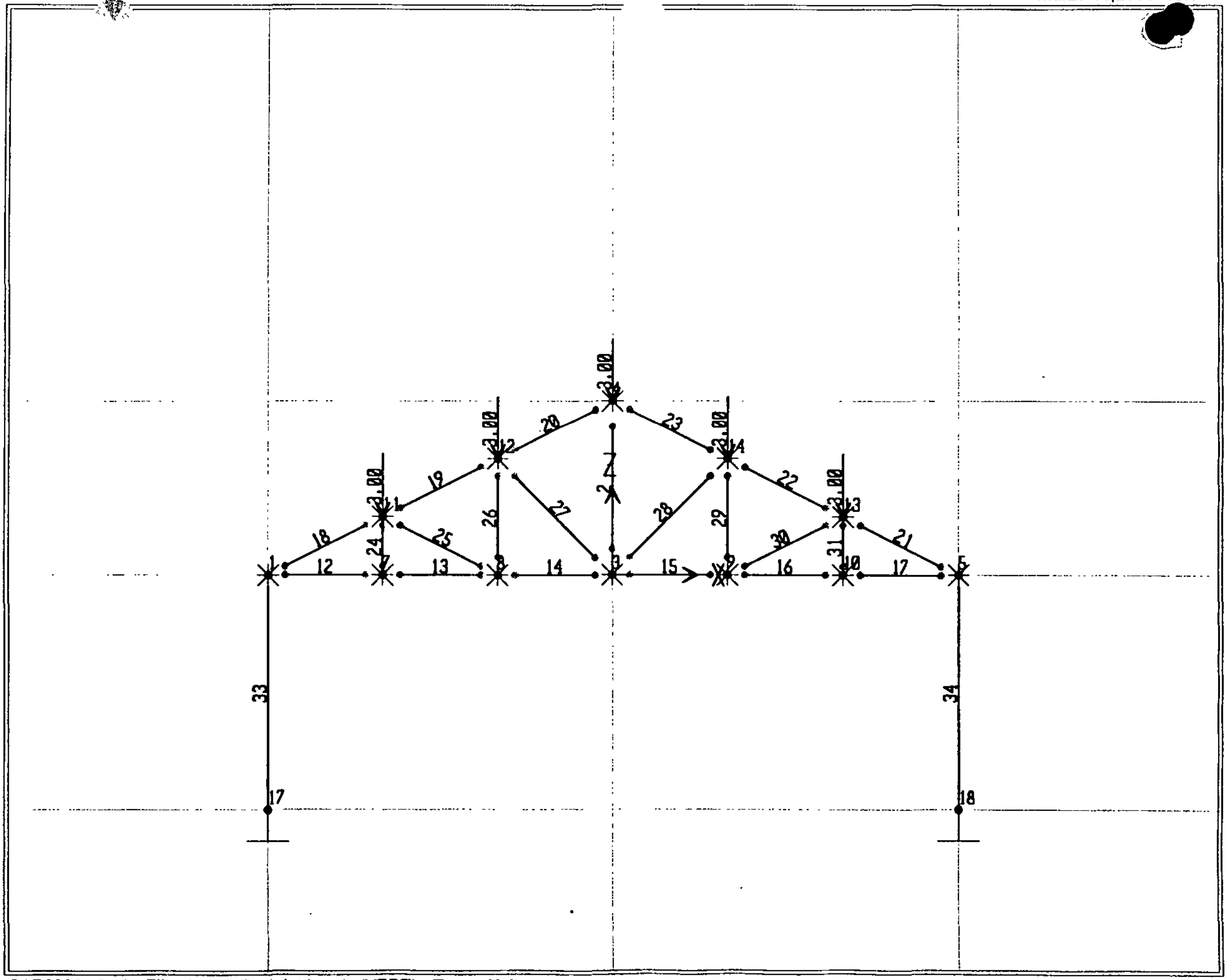
REL DIST	P	V2	V3	T	M2	M3
0.00000	-11.015010	.000000	.000000	.000000	.000000	.000000
0.50000	-11.015010	.000000	.000000	.000000	.000000	.000000
1.00000	-11.015010	.000000	.000000	.000000	.000000	.000000

ELEM 9 ----------------- LENGTH = 2.828427

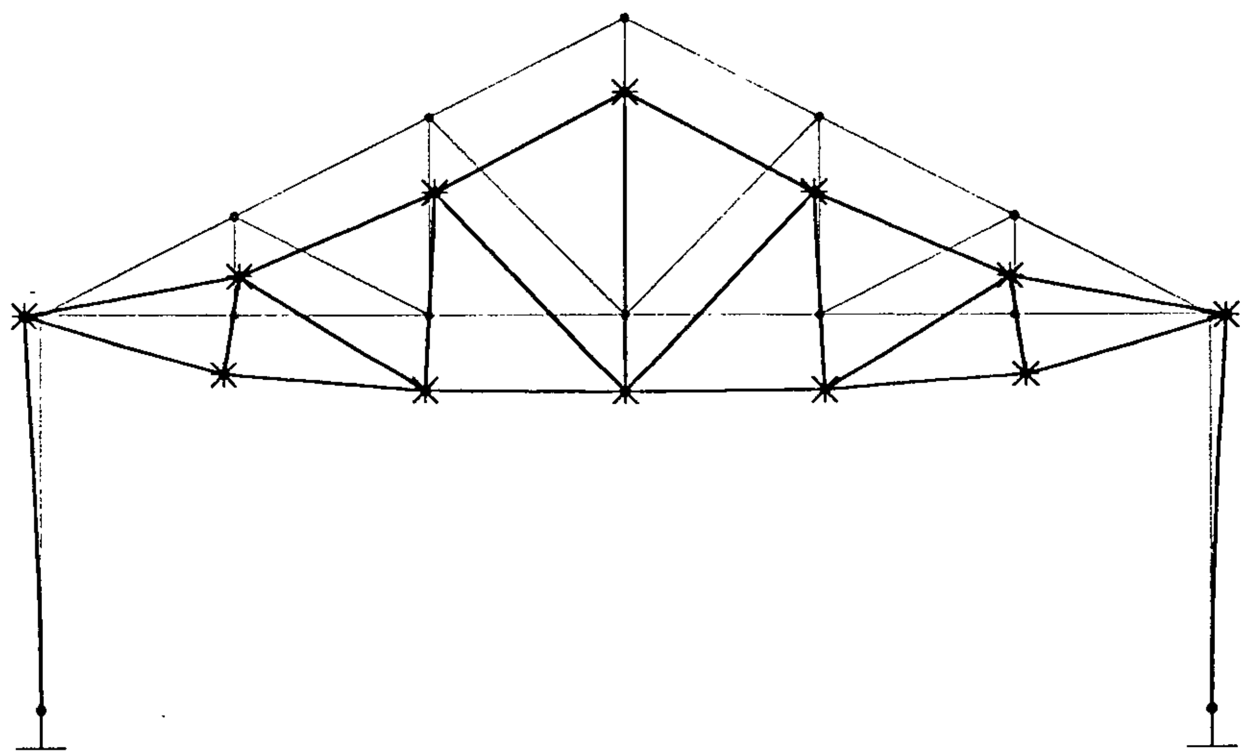
LOADVERTICAL -----------------

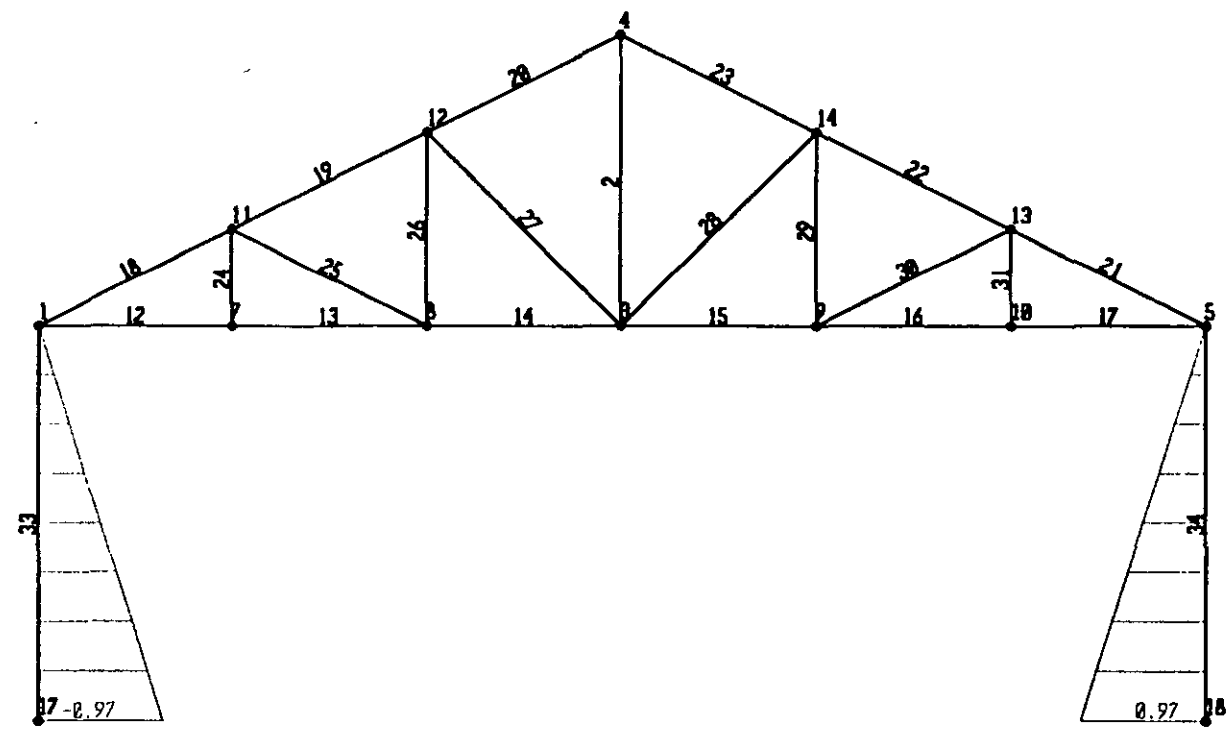
REL DIST	P	V2	V3	T	M2	M3
0.00000	10.198193	.000000	.000000	.000000	.000000	.000000
0.50000	10.198193	.000000	.000000	.000000	.000000	.000000
1.00000	10.198193	.000000	.000000	.000000	.000000	.000000



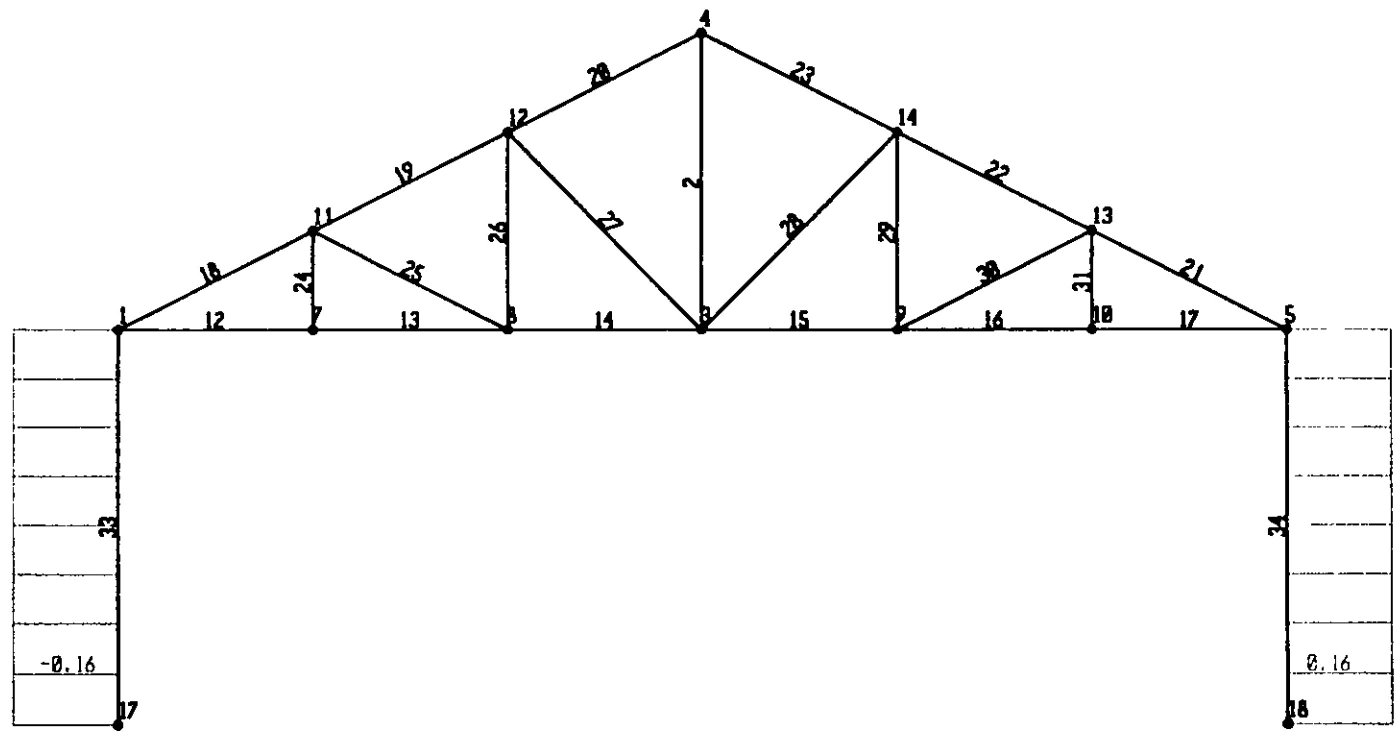


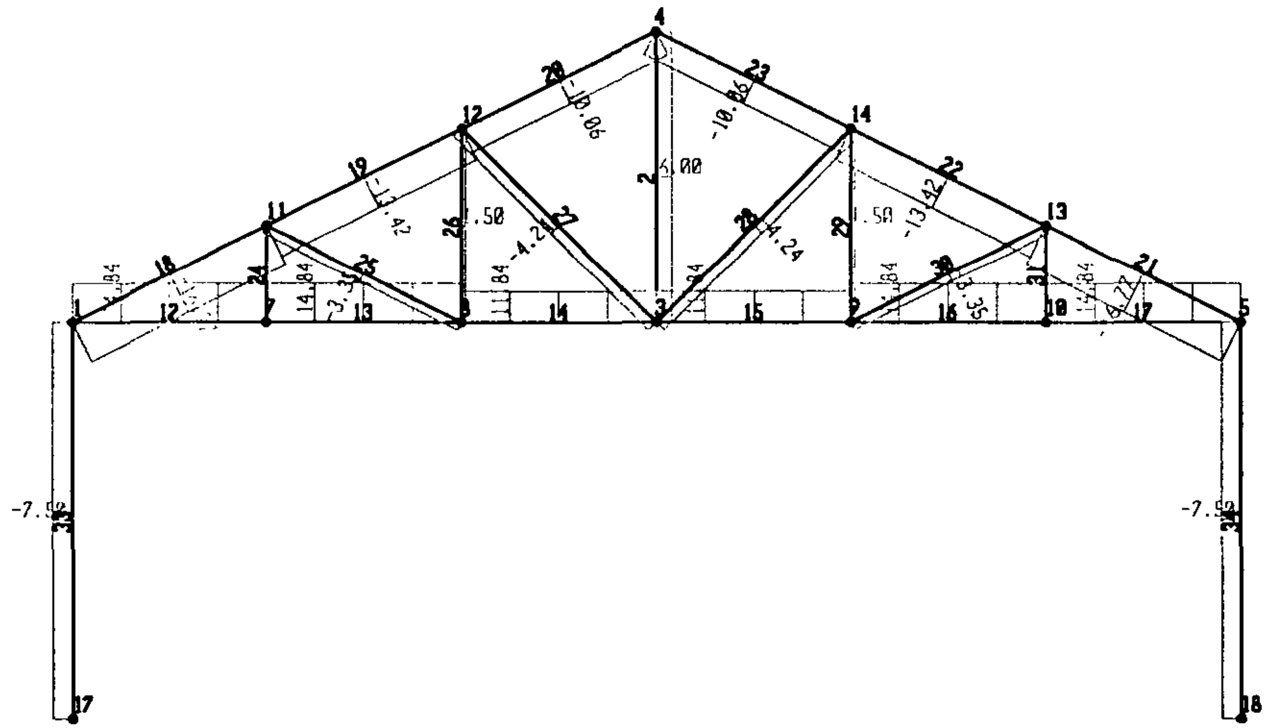
SAP2000 v6.10 - File:ejemplo7 - Joint Loads (VERT) - Ton-m Units





SAP2000 v6.10 - File:ejemplo7 - Moment 3-3 Diagram (VERT) - Ton-m Units


4
20
23
12
14
19
22
11
13
2
26
27
28
29
18
24
25
30
31
21
1
12
7
13
8
14
3
15
9
16
10
17
5
33
34
-0.16
0.16
17
18



SAP2000 v6.10 - File:ejemplo7 - Axial Force Diagram (VERT) - Ton-m Units

```
; File C:\Mis documentos\curso sap 2000\ejemplo7.s2k saved 3/18/00 15:22:55 in Ton-m

SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  LINES=59

JOINT
  1  X=-9  Y=0  Z=0
  3  X=0  Y=0  Z=0
  4  X=0  Y=0  Z=4.5
  5  X=9  Y=0  Z=0
  7  X=-6  Y=0  Z=0
  8  X=-3  Y=0  Z=0
  9  X=3  Y=0  Z=0
  10  X=6  Y=0  Z=0
  11  X=-6  Y=0  Z=1.5
  12  X=-3  Y=0  Z=3
  13  X=6  Y=0  Z=1.5
  14  X=3  Y=0  Z=3
  17  X=-9  Y=0  Z=-6
  18  X=9  Y=0  Z=-6

RESTRAINT
  ADD=17  DOF=U1,U2,U3,R1,R2,R3
  ADD=18  DOF=U1,U2,U3,R1,R2,R3
  ADD=1  DOF=U2,R1,R3
  ADD=3  DOF=U2,R1,R3
  ADD=4  DOF=U2,R1,R3
  ADD=5  DOF=U2,R1,R3
  ADD=7  DOF=U2,R1,R3
  ADD=8  DOF=U2,R1,R3
  ADD=9  DOF=U2,R1,R3
  ADD=10  DOF=U2,R1,R3
  ADD=11  DOF=U2,R1,R3
  ADD=12  DOF=U2,R1,R3
  ADD=13  DOF=U2,R1,R3
  ADD=14  DOF=U2,R1,R3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=ACERO  IDES=N  M=.2448  W=2.4026
    T=0  E=2E+07  U=.3  A=.0000099
  NAME=CONCRETO  IDES=N  M=.7981  W=7.8334
    T=0  E=1800000  U=.25  A=.0000117

FRAME SECTION
  NAME=FSEC1 MAT=STEEL SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125
  NAME=FSEC2 MAT=ACERO A=.005 J=0 I=0,0 AS=0,0 S=1,1 Z=1,1 R=1,1 T=.4572,.254
  NAME=FSEC3 MAT=CONCRETO SH=R T=.5,.5 A=.25 J=8.802084E-03 I=5.208333E-03,5.208333E-03
AS=.2083333,.2083333

FRAME
  2  J=3,4  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  12  J=1,7  SEC=FSEC2  NSEG=4  ANG=0  IREL=R3  JREL=R3
  13  J=7,8  SEC=FSEC2  NSEG=4  ANG=0  IREL=R3  JREL=R3
  14  J=8,3  SEC=FSEC2  NSEG=4  ANG=0  IREL=R3  JREL=R3
  15  J=3,9  SEC=FSEC2  NSEG=4  ANG=0  IREL=R3  JREL=R3
  16  J=9,10  SEC=FSEC2  NSEG=4  ANG=0  IREL=R3  JREL=R3
  17  J=10,5  SEC=FSEC2  NSEG=4  ANG=0  IREL=R3  JREL=R3
  18  J=1,11  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  19  J=11,12  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  20  J=12,4  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  21  J=5,13  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  22  J=13,14  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
```

```
  23  J=14,4   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  24  J=7,11   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  25  J=11,8   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  26  J=8,12   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  27  J=12,3   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  28  J=3,14   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  29  J=14,9   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  30  J=9,13   SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  31  J=10,13  SEC=FSEC2  NSEG=2  ANG=0  IREL=R3  JREL=R3
  33  J=17,1   SEC=FSEC3  NSEG=2  ANG=0
  34  J=18,5   SEC=FSEC3  NSEG=2  ANG=0

LOAD
  NAME=VERT
    TYPE=FORCE
      ADD=4  UZ=-3
      ADD=11  UZ=-3
      ADD=12  UZ=-3
      ADD=13  UZ=-3
      ADD=14  UZ=-3

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=VERT
  ELEM=JOINT  TYPE=APPL  LOAD=VERT
  ELEM=FRAME  TYPE=FORCE  LOAD=VERT
  ELEM=FRAME  TYPE=JOINTF  LOAD=VERT

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  -9
  GRID GLOBAL  X "2"  0
  GRID GLOBAL  X "3"  9
  GRID GLOBAL  Y "4"  0
  GRID GLOBAL  Z "5"  -6
  GRID GLOBAL  Z "6"  0
  GRID GLOBAL  Z "7"  4.5
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  STATICLOAD VERT TYPE OTHER
END SUPPLEMENTAL DATA
```

S A P 2 0 0 0

Structural Analysis Programs

Version 6.10

It is the responsibility of the user to verify all
results produced by this program

18 Mar 2000 15:10:44

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 1
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

S Y S T E M D A T A

```
STEADY STATE LOAD FREQUENCY  - - - - - - - -  0.0000E+00

LENGTH UNITS - - - - - - - - - - - - - - - - -        M
FORCE  UNITS - - - - - - - - - - - - - - - - -      TON

UP DIRECTION - - - - - - - - - - - - - - - - -       +Z

GLOBAL DEGREES OF FREEDOM  - - - - - - - - - -       UX
                           - - - - - - - - - -       UZ
                           - - - - - - - - - -       RY

PAGINATION BY  - - - - - - - - - - - - - - - -    LINES
NUMBER OF LINES PER PAGE - - - - - - - - - - -       59
INCLUDE WARNING MESSAGES IN OUTPUT FILE  - -          Y
```

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 2
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

G E N E R A T E D J O I N T C O O R D I N A T E S

JOINT	X	Y	Z
1	-9.000	0.000	0.000
3	0.000	0.000	0.000
4	0.000	0.000	4.500
5	9.000	0.000	0.000
7	-6.000	0.000	0.000
8	-3.000	0.000	0.000
9	3.000	0.000	0.000
10	6.000	0.000	0.000
11	-6.000	0.000	1.500
12	-3.000	0.000	3.000
13	6.000	0.000	1.500
14	3.000	0.000	3.000
17	-9.000	0.000	-6.000
18	9.000	0.000	-6.000

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 3
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

P A T T E R N S

PATTERN	JOINT	VALUE
DEFAULT		

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 4
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

R E S T R A I N T D A T A

JOINT						
1		U2		R1		R3
3		U2		R1		R3
4		U2		R1		R3
5		U2		R1		R3
7		U2		R1		R3
8		U2		R1		R3
9		U2		R1		R3
10		U2		R1		R3
11		U2		R1		R3
12		U2		R1		R3
13		U2		R1		R3
14		U2		R1		R3
17	U1	U2	U3	R1	R2	R3
18	U1	U2	U3	R1	R2	R3

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 5
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

M A T E R I A L P R O P E R T Y D A T A

MAT LABEL	NUMBER TEMPS	WEIGHT PER UNIT VOL	MASS PER UNIT VOL	DESIGN CODE
STEEL	1	0.7833E+01	0.7981E+00	S
CONC	1	0.2403E+01	0.2448E+00	C
ACERO	1	0.2403E+01	0.2448E+00	N
CONCRETO	1	0.7833E+01	0.7981E+00	N

T E M P E R A T U R E   D E P E N D E N T   D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	MODULUS OF ELASTICITY E1	E2	E3	SHEAR MODULII G12	G13	G23
STEEL	0.00	0.204E+08	0.204E+08	0.204E+08	0.784E+07	0.784E+07	0.784E+07
CONC	0.00	0.253E+07	0.253E+07	0.253E+07	0.105E+07	0.105E+07	0.105E+07
ACERO	0.00	0.200E+08	0.200E+08	0.200E+08	0.769E+07	0.769E+07	0.769E+07
CONCRETO	0.00	0.180E+07	0.180E+07	0.180E+07	0.720E+06	0.720E+06	0.720E+06

T E M P E R A T U R E   D E P E N D E N T   D A T A

THERMAL EXPANSION COEFFICIENTS

MAT LABEL	TEMP	COEFFICIENTS OF THERMAL EXPANSION A1	A2	A3	A12	A13	A23
STEEL	0.00	0.117E-04	0.117E-04	0.117E-04	0.000E+00	0.000E+00	0.000E+00
CONC	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00
ACERO	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00
CONCRETO	0.00	0.117E-04	0.117E-04	0.117E-04	0.000E+00	0.000E+00	0.000E+00

T E M P E R A T U R E   D E P E N D E N T   D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	POISSONS RATIO U12	U13	U23	U14	U24	U34	U15	U25	U35	U45	U16	U26	U36	U46	U56
STEEL	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONC	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
ACERO	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONCRETO	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

MATERIAL PROPERTIES

MAT LABEL	TEMP	YIELD FY
CONC	0.00	36.00

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	SHAPE TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
FSEC1	R	0.500	0.300				
FSEC2	G						
FSEC3	R	0.500	0.500				

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	AXIAL AREA	TORSIONAL CONSTANT	MOMENTS OF INERTIA I33	I22	SHEAR AREAS A2	A3
FSEC1	0.150E+00	0.282E-02	0.313E-02	0.113E-02	0.125E+00	0.125E+00

FSEC2	0.500E-02	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
FSEC3	0.250E+00	0.880E-02	0.521E-02	0.521E-02	0.208E+00	0.208E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 12
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	MAT LABEL	ADDITIONAL MASS PER LENGTH	ADDITIONAL WEIGHT PER LENGTH
FSEC1	STEEL	0.000E+00	0.000E+00
FSEC2	ACERO	0.000E+00	0.000E+00
FSEC3	CONCRETO	0.000E+00	0.000E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 13
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

F R A M E E L E M E N T D A T A

ELEMENT LABEL	JOINT END-I	JOINT END-J	ELEMENT LENGTH	END-OFFSET-LENGTHS END-I	END-J	RIGID-END FACTOR	NUMBER OF SEGMENTS
2	3	4	4.500	0.000	0.000	0.0000	2
12	1	7	3.000	0.000	0.000	0.0000	4
13	7	8	3.000	0.000	0.000	0.0000	4
14	8	3	3.000	0.000	0.000	0.0000	4
15	3	9	3.000	0.000	0.000	0.0000	4
16	9	10	3.000	0.000	0.000	0.0000	4
17	10	5	3.000	0.000	0.000	0.0000	4
18	1	11	3.354	0.000	0.000	0.0000	2
19	11	12	3.354	0.000	0.000	0.0000	2
20	12	4	3.354	0.000	0.000	0.0000	2
21	5	13	3.354	0.000	0.000	0.0000	2
22	13	14	3.354	0.000	0.000	0.0000	2
23	14	4	3.354	0.000	0.000	0.0000	2
24	7	11	1.500	0.000	0.000	0.0000	2
25	11	8	3.354	0.000	0.000	0.0000	2
26	8	12	3.000	0.000	0.000	0.0000	2
27	12	3	4.243	0.000	0.000	0.0000	2
28	3	14	4.243	0.000	0.000	0.0000	2
29	14	9	3.000	0.000	0.000	0.0000	2
30	9	13	3.354	0.000	0.000	0.0000	2
31	10	13	1.500	0.000	0.000	0.0000	2
33	17	1	6.000	0.000	0.000	0.0000	2
34	18	5	6.000	0.000	0.000	0.0000	2

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 14
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

F R A M E E L E M E N T D A T A

ELEMENT LABEL	SECTION LABEL	LOCAL PLANE	COORD SYSTEM	PLN 1ST	PLN 2ND	PLANE JOINTA	PLANE JOINTB	COORD ANGLE
2	FSEC2	12	0	+Z	+X	0	0	0.00
12	FSEC2	12	0	+Z	+X	0	0	0.00
13	FSEC2	12	0	+Z	+X	0	0	0.00
14	FSEC2	12	0	+Z	+X	0	0	0.00
15	FSEC2	12	0	+Z	+X	0	0	0.00
16	FSEC2	12	0	+Z	+X	0	0	0.00
17	FSEC2	12	0	+Z	+X	0	0	0.00
18	FSEC2	12	0	+Z	+X	0	0	0.00
19	FSEC2	12	0	+Z	+X	0	0	0.00
20	FSEC2	12	0	+Z	+X	0	0	0.00
21	FSEC2	12	0	+Z	+X	0	0	0.00
22	FSEC2	12	0	+Z	+X	0	0	0.00
23	FSEC2	12	0	+Z	+X	0	0	0.00
24	FSEC2	12	0	+Z	+X	0	0	0.00
25	FSEC2	12	0	+Z	+X	0	0	0.00
26	FSEC2	12	0	+Z	+X	0	0	0.00
27	FSEC2	12	0	+Z	+X	0	0	0.00
28	FSEC2	12	0	+Z	+X	0	0	0.00
29	FSEC2	12	0	+Z	+X	0	0	0.00
30	FSEC2	12	0	+Z	+X	0	0	0.00
31	FSEC2	12	0	+Z	+X	0	0	0.00
33	FSEC3	12	0	+Z	+X	0	0	0.00
34	FSEC3	12	0	+Z	+X	0	0	0.00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 15
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

F R A M E E L E M E N T D A T A

ELEMENT LABEL	END-I RELEASE CODES	END-J RELEASE CODES
2	R3	R3
12	R3	R3
13	R3	R3
14	R3	R3
15	R3	R3
16	R3	R3
17	R3	R3

18	R3	R3
19	R3	R3
20	R3	R3
21	R3	R3
22	R3	R3
23	R3	R3
24	R3	R3
25	R3	R3
26	R3	R3
27	R3	R3
28	R3	R3
29	R3	R3
30	R3	R3
31	R3	R3

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 16
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

T O T A L W E I G H T S A N D M A S S E S

SECTION LABEL	WEIGHT	MASS
FSEC2	0.8027	0.0818
FSEC3	23.5002	2.3943
TOTAL	24.3029	2.4761

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 17
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

L O A D C O N D I T I O N VERT

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.0000E+00

JOINT FORCES IN LOCAL COORDINATES

JOINT LABEL	FORCE 1	FORCE 2	FORCE 3	MOMENT 1	MOMENT 2	MOMENT 3
4	0.000E+00	0.000E+00	-0.300E+01	0.000E+00	0.000E+00	0.000E+00
11	0.000E+00	0.000E+00	-0.300E+01	0.000E+00	0.000E+00	0.000E+00
12	0.000E+00	0.000E+00	-0.300E+01	0.000E+00	0.000E+00	0.000E+00
13	0.000E+00	0.000E+00	-0.300E+01	0.000E+00	0.000E+00	0.000E+00
14	0.000E+00	0.000E+00	-0.300E+01	0.000E+00	0.000E+00	0.000E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 18
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

O U T P U T S E L E C T I O N

DISPLACEMENTS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERT					

APPLIED AND INTERNAL LOADS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERT					

INTERNAL FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERT					

JOINT FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
VERT					

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 19
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo7.EKO

I N P U T C O M P L E T E

```
PROGRAM SAP2000 - VERSION E6.10                                  FILE:EJEMPLO7.LOG
EDUCATIONAL VERSION - COMMERCIAL USE PROHIBITED


B E G I N   A N A L Y S I S   P H A S E                     2000/03/18  15:10:46

MEMORY AVAILABLE FOR DATA (BYTES)          =   1000000


J O I N T   E L E M E N T   F O R M A T I O N                           15:10:46

NUMBER OF  JOINT ELEMENTS FORMED           =         5
NUMBER OF SPRING ELEMENTS FORMED           =         0


F R A M E   E L E M E N T   F O R M A T I O N                           15:10:46

NUMBER OF  FRAME ELEMENTS FORMED           =        23


E Q U A T I O N   S O L U T I O N                                       15:10:47

TOTAL NUMBER OF EQUILIBRIUM EQUATIONS      =        26
APPROXIMATE "EFFECTIVE" BAND WIDTH         =         4

NUMBER OF EQUATION STORAGE BLOCKS          =         1
MAXIMUM BLOCK SIZE (NUMBER OF TERMS)       =       116
SIZE OF STIFFNESS FILE (BYTES)             =      1048

NUMBER OF EQUATIONS TO SOLVE               =        26
NUMBER OF STATIC LOAD CASES                =         1
NUMBER OF ACCELERATION LOADS               =         3
NUMBER OF NONLINEAR DEFORMATION LOADS      =         0


J O I N T   O U T P U T                                                 15:10:48

G L O B A L   F O R C E   B A L A N C E   R E L A T I V E   E R R O R S

PERCENT FORCE AND MOMENT ERROR AT THE ORIGIN, IN GLOBAL COORDINATES

   LOAD        FX          FY          FZ          MX          MY          MZ
   VERT    3.51E-14     .000000    1.66E-13     .000000    6.77E-14     .000000


E L E M E N T   J O I N T - F O R C E   O U T P U T                     15:10:48

NUMBER OF  FRAME ELEMENTS SAVED            =        23


F R A M E   E L E M E N T   O U T P U T                                 15:10:49

NUMBER OF  FRAME ELEMENTS SAVED            =        23
NUMBER OF  FRAME ELEMENTS PRINTED          =        23


A N A L Y S I S   C O M P L E T E                           2000/03/18  15:10:49
```

S A P 2 0 0 0 (R)

Structural Analysis Programs

Version E6.10

18 Mar 2000 15:10:46

D I S P L A C E M E N T   D E G R E E S   O F   F R E E D O M

(A) = Active DOF, equilibrium equation
(-) = Restrained DOF, reaction computed
(+) = Constrained DOF
( ) = Null DOF

JOINTS	UX	UY	UZ	RX	RY	RZ
1	A		A		A	
3 TO 4	A		A			
5	A		A		A	
7 TO 14	A		A			
17 TO 18	-		-		-	

J O I N T   D I S P L A C E M E N T S

TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

LOAD VERT ------------------

JOINT	UX	UZ	RY
1	-0.001245	-0.000100	-0.000310
3	-3.99E-17	-0.005680	.000000
4	-4.17E-17	-0.005610	.000000
5	0.001245	-0.000100	0.000310
7	-0.000800	-0.004571	.000000
8	-0.000355	-0.005753	.000000
9	0.000355	-0.005753	.000000
10	0.000800	-0.004571	.000000
11	0.000361	-0.004571	.000000
12	0.000426	-0.005708	.000000
13	-0.000361	-0.004571	.000000
14	-0.000426	-0.005708	.000000
17	.000000	.000000	.000000
18	.000000	.000000	.000000

A P P L I E D   L O A D S

FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOAD VERT ------------------

JOINT	FX	FZ	MY
4	.000000	-3.000000	.000000
11	.000000	-3.000000	.000000
12	.000000	-3.000000	.000000
13	.000000	-3.000000	.000000
14	.000000	-3.000000	.000000

G L O B A L   F O R C E   B A L A N C E

TOTAL FORCE AND MOMENT AT THE ORIGIN, IN GLOBAL COORDINATES

LOAD VERT ------------------

	FX	FY	FZ	MX	MY	MZ
APPLIED	.000000	.000000	-15.000000	.000000	.000000	.000000
REACTNS	1.05E-14	.000000	15.000000	.000000	1.04E-13	.000000
TOTAL	1.05E-14	.000000	-4.97E-14	.000000	1.04E-13	.000000

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 2 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	.000000	.000000	-6.000000	.000000	.000000	.000000
4	.000000	.000000	6.000000	.000000	.000000	.000000

ELEM 12 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
1	-14.838668	.000000	.000000	.000000	.000000	.000000
7	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 13 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
7	-14.838668	.000000	.000000	.000000	.000000	.000000
8	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 14 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
8	-11.838668	.000000	.000000	.000000	.000000	.000000
3	11.838668	.000000	.000000	.000000	.000000	.000000

ELEM 15 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	-11.838668	.000000	.000000	.000000	.000000	.000000
9	11.838668	.000000	.000000	.000000	.000000	.000000

ELEM 16 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
9	-14.838668	.000000	.000000	.000000	.000000	.000000
10	14.838668	.000000	.000000	.000000	.000000	.000000

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 17 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
10	-14.838668	.000000	.000000	.000000	.000000	.000000
5	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 18 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
1	15.000000	.000000	7.500000	.000000	.000000	.000000
11	-15.000000	.000000	-7.500000	.000000	.000000	.000000

ELEM 19 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
11	12.000000	.000000	6.000000	.000000	.000000	.000000
12	-12.000000	.000000	-6.000000	.000000	.000000	.000000

ELEM 20 -----------------

LOAD VERT -----------------

JOINT	FX	FY	FZ	MX	MY	MZ
12	9.000000	.000000	4.500000	.000000	.000000	.000000
4	-9.000000	.000000	-4.500000	.000000	.000000	.000000

ELEM 21 -----------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
5	-15.000000	.000000	7.500000	.000000	.000000	.000000
13	15.000000	.000000	-7.500000	.000000	.000000	.000000

ELEM 22 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
13	-12.000000	.000000	6.000000	.000000	.000000	.000000
14	12.000000	.000000	-6.000000	.000000	.000000	.000000

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 23 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
14	-9.000000	.000000	4.500000	.000000	.000000	.000000
4	9.000000	.000000	-4.500000	.000000	.000000	.000000

ELEM 24 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
7	.000000	.000000	5.55E-17	.000000	.000000	.000000
11	.000000	.000000	-5.55E-17	.000000	.000000	.000000

ELEM 25 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
11	3.000000	.000000	-1.500000	.000000	.000000	.000000
8	-3.000000	.000000	1.500000	.000000	.000000	.000000

ELEM 26 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
8	.000000	.000000	-1.500000	.000000	.000000	.000000
12	.000000	.000000	1.500000	.000000	.000000	.000000

ELEM 27 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
12	3.000000	.000000	-3.000000	.000000	.000000	.000000
3	-3.000000	.000000	3.000000	.000000	.000000	.000000

ELEM 28 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	3.000000	.000000	3.000000	.000000	.000000	.000000
14	-3.000000	.000000	-3.000000	.000000	.000000	.000000

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 29 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
14	.000000	.000000	1.500000	.000000	.000000	.000000
9	.000000	.000000	-1.500000	.000000	.000000	.000000

ELEM 30 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
9	3.000000	.000000	1.500000	.000000	.000000	.000000
13	-3.000000	.000000	-1.500000	.000000	.000000	.000000

ELEM 31 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
10	.000000	.000000	-3.89E-15	.000000	.000000	.000000
13	.000000	.000000	3.89E-15	.000000	.000000	.000000

ELEM 33 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
17	0.161332	.000000	7.500000	.000000	0.967990	.000000
1	-0.161332	.000000	-7.500000	.000000	-7.55E-17	.000000

ELEM 34 ------------------

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
18	-0.161332	.000000	7.500000	.000000	-0.967990	.000000
5	0.161332	.000000	-7.500000	.000000	-2.14E-17	.000000

F R A M E E L E M E N T I N T E R N A L F O R C E S

ELEM 2 ------------------ LENGTH = 4.500000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	6.000000	.000000	.000000	.000000	.000000	.000000
0.50000	6.000000	.000000	.000000	.000000	.000000	.000000
1.00000	6.000000	.000000	.000000	.000000	.000000	.000000

ELEM 12 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	14.838668	.000000	.000000	.000000	.000000	.000000
0.25000	14.838668	.000000	.000000	.000000	.000000	.000000
0.50000	14.838668	.000000	.000000	.000000	.000000	.000000
0.75000	14.838668	.000000	.000000	.000000	.000000	.000000
1.00000	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 13 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	14.838668	.000000	.000000	.000000	.000000	.000000
0.25000	14.838668	.000000	.000000	.000000	.000000	.000000
0.50000	14.838668	.000000	.000000	.000000	.000000	.000000
0.75000	14.838668	.000000	.000000	.000000	.000000	.000000
1.00000	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 14 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	11.838668	.000000	.000000	.000000	.000000	.000000
0.25000	11.838668	.000000	.000000	.000000	.000000	.000000
0.50000	11.838668	.000000	.000000	.000000	.000000	.000000
0.75000	11.838668	.000000	.000000	.000000	.000000	.000000
1.00000	11.838668	.000000	.000000	.000000	.000000	.000000

ELEM 15 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	11.838668	.000000	.000000	.000000	.000000	.000000
0.25000	11.838668	.000000	.000000	.000000	.000000	.000000
0.50000	11.838668	.000000	.000000	.000000	.000000	.000000
0.75000	11.838668	.000000	.000000	.000000	.000000	.000000
1.00000	11.838668	.000000	.000000	.000000	.000000	.000000

F R A M E E L E M E N T I N T E R N A L F O R C E S

ELEM 16 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	14.838668	.000000	.000000	.000000	.000000	.000000
0.25000	14.838668	.000000	.000000	.000000	.000000	.000000
0.50000	14.838668	.000000	.000000	.000000	.000000	.000000
0.75000	14.838668	.000000	.000000	.000000	.000000	.000000
1.00000	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 17 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	14.838668	.000000	.000000	.000000	.000000	.000000
0.25000	14.838668	.000000	.000000	.000000	.000000	.000000
0.50000	14.838668	.000000	.000000	.000000	.000000	.000000
0.75000	14.838668	.000000	.000000	.000000	.000000	.000000
1.00000	14.838668	.000000	.000000	.000000	.000000	.000000

ELEM 18 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-16.770510	.000000	.000000	.000000	.000000	.000000
0.50000	-16.770510	.000000	.000000	.000000	.000000	.000000
1.00000	-16.770510	.000000	.000000	.000000	.000000	.000000

ELEM 19 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-13.416408	.000000	.000000	.000000	.000000	.000000
0.50000	-13.416408	.000000	.000000	.000000	.000000	.000000
1.00000	-13.416408	.000000	.000000	.000000	.000000	.000000

ELEM 20 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-10.062306	.000000	.000000	.000000	.000000	.000000
0.50000	-10.062306	.000000	.000000	.000000	.000000	.000000
1.00000	-10.062306	.000000	.000000	.000000	.000000	.000000

F R A M E E L E M E N T I N T E R N A L F O R C E S

ELEM 21 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-16.770510	.000000	.000000	.000000	.000000	.000000
0.50000	-16.770510	.000000	.000000	.000000	.000000	.000000
1.00000	-16.770510	.000000	.000000	.000000	.000000	.000000

ELEM 22 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-13.416408	.000000	.000000	.000000	.000000	.000000
0.50000	-13.416408	.000000	.000000	.000000	.000000	.000000
1.00000	-13.416408	.000000	.000000	.000000	.000000	.000000

ELEM 23 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-10.062306	.000000	.000000	.000000	.000000	.000000
0.50000	-10.062306	.000000	.000000	.000000	.000000	.000000
1.00000	-10.062306	.000000	.000000	.000000	.000000	.000000

ELEM 24 ------------------ LENGTH = 1.500000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-5.55E-17	.000000	.000000	.000000	.000000	.000000
0.50000	-5.55E-17	.000000	.000000	.000000	.000000	.000000
1.00000	-5.55E-17	.000000	.000000	.000000	.000000	.000000

ELEM 25 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-3.354102	.000000	.000000	.000000	.000000	.000000
0.50000	-3.354102	.000000	.000000	.000000	.000000	.000000
1.00000	-3.354102	.000000	.000000	.000000	.000000	.000000

ELEM 26 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	1.500000	.000000	.000000	.000000	.000000	.000000
0.50000	1.500000	.000000	.000000	.000000	.000000	.000000

1.00000	1.500000	.000000	.000000	.000000	.000000	.000000

F R A M E   E L E M E N T   I N T E R N A L   F O R C E S

ELEM 27 ------------------ LENGTH = 4.242641

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-4.242641	.000000	.000000	.000000	.000000	.000000
0.50000	-4.242641	.000000	.000000	.000000	.000000	.000000
1.00000	-4.242641	.000000	.000000	.000000	.000000	.000000

ELEM 28 ------------------ LENGTH = 4.242641

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-4.242641	.000000	.000000	.000000	.000000	.000000
0.50000	-4.242641	.000000	.000000	.000000	.000000	.000000
1.00000	-4.242641	.000000	.000000	.000000	.000000	.000000

ELEM 29 ------------------ LENGTH = 3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	1.500000	.000000	.000000	.000000	.000000	.000000
0.50000	1.500000	.000000	.000000	.000000	.000000	.000000
1.00000	1.500000	.000000	.000000	.000000	.000000	.000000

ELEM 30 ------------------ LENGTH = 3.354102

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-3.354102	.000000	.000000	.000000	.000000	.000000
0.50000	-3.354102	.000000	.000000	.000000	.000000	.000000
1.00000	-3.354102	.000000	.000000	.000000	.000000	.000000

ELEM 31 ------------------ LENGTH = 1.500000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	3.89E-15	.000000	.000000	.000000	.000000	.000000
0.50000	3.89E-15	.000000	.000000	.000000	.000000	.000000
1.00000	3.89E-15	.000000	.000000	.000000	.000000	.000000

ELEM 33 ------------------ LENGTH = 6.000000

LOAD VERT ------------------

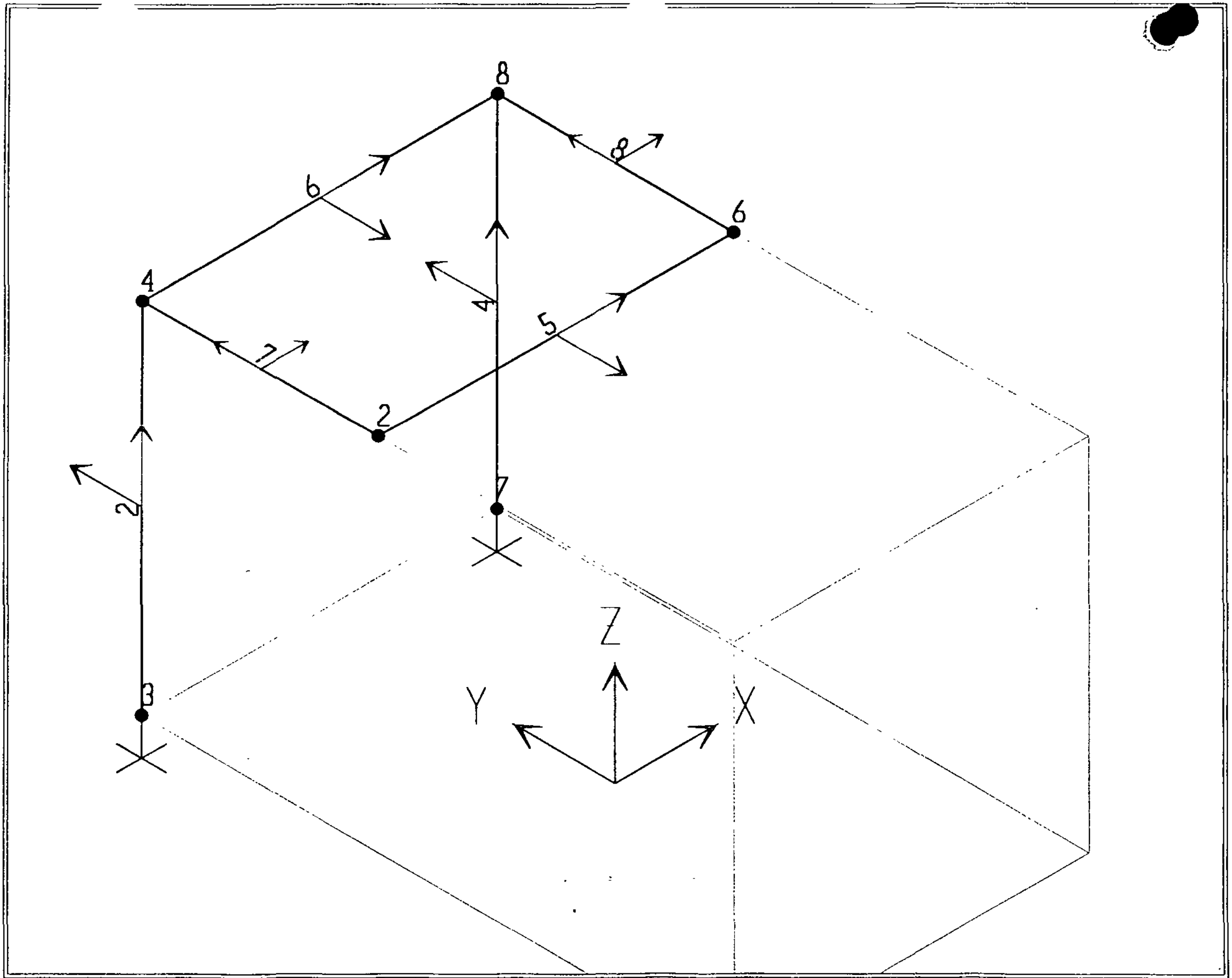
REL DIST	P	V2	V3	T	M2	M3
0.00000	-7.500000	-0.161332	.000000	.000000	.000000	-0.967990
0.50000	-7.500000	-0.161332	.000000	.000000	.000000	-0.483995
1.00000	-7.500000	-0.161332	.000000	.000000	.000000	4.44E-16

F R A M E   E L E M E N T   I N T E R N A L   F O R C E S

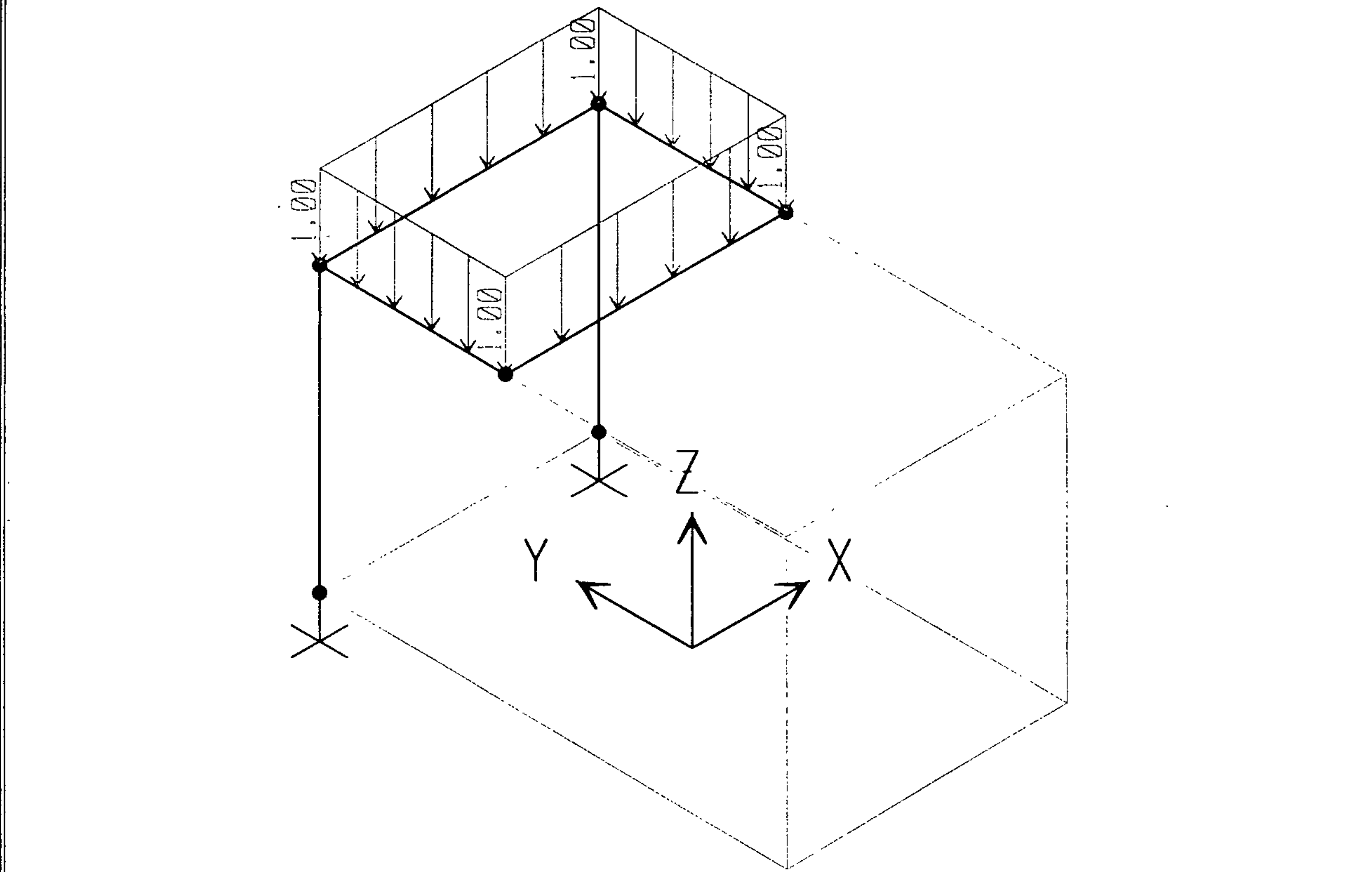
ELEM 34 ------------------ LENGTH = 6.000000

LOAD VERT ------------------

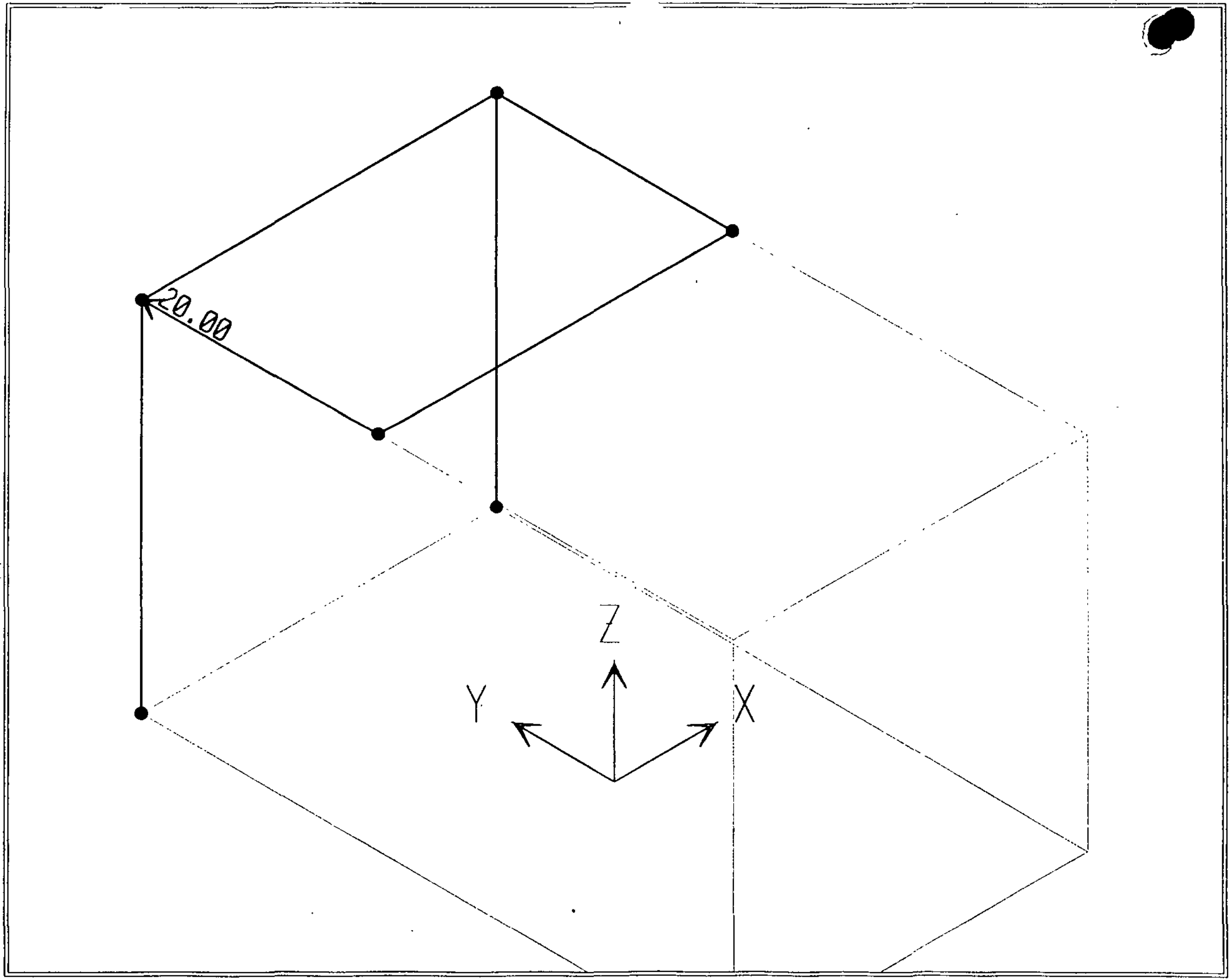
REL DIST	P	V2	V3	T	M2	M3
0.00000	-7.500000	0.161332	.000000	.000000	.000000	0.967990
0.50000	-7.500000	0.161332	.000000	.000000	.000000	0.483995
1.00000	-7.500000	0.161332	.000000	.000000	.000000	-1.39E-16



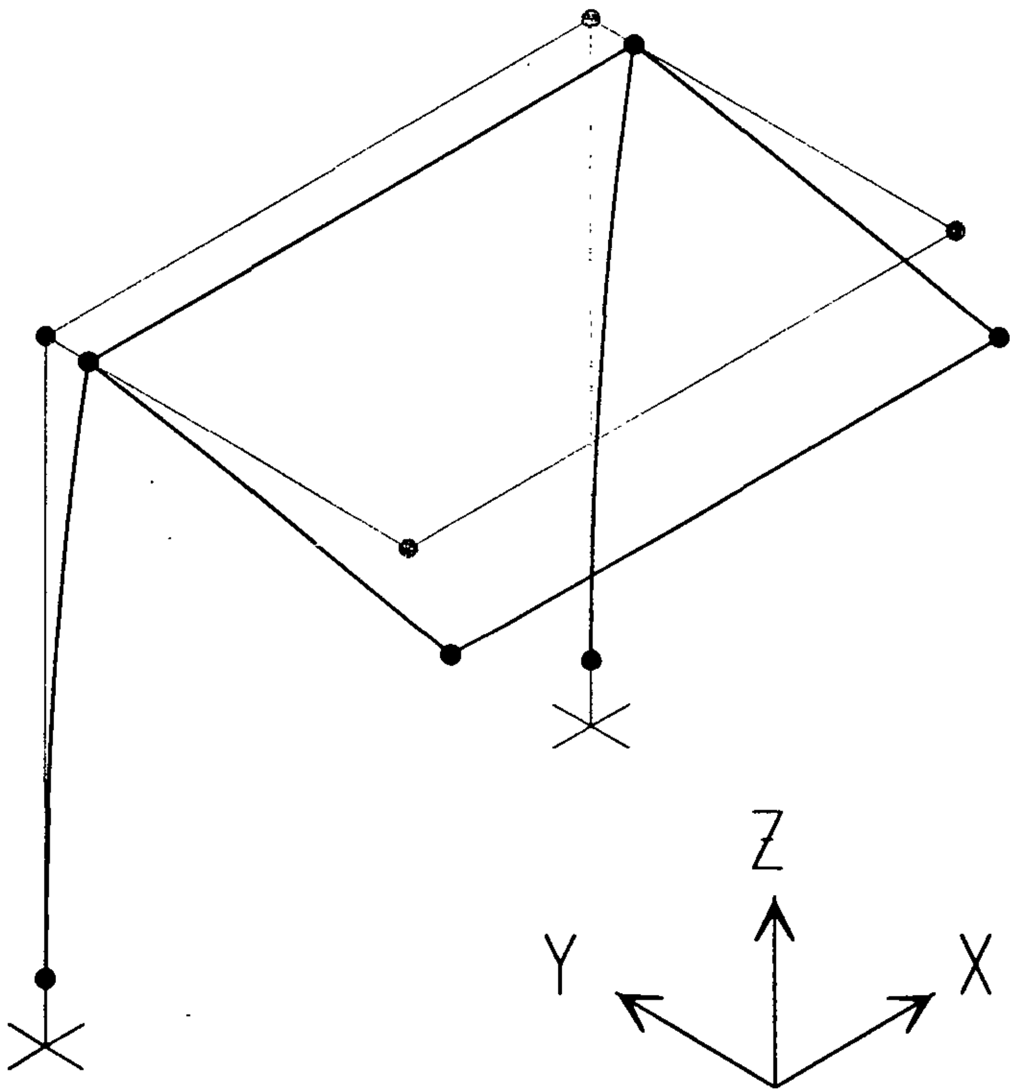
SAP2000 v6.10 - File:ejemplo8 - 3-D View - Ton-m Units



SAP2000 v6.10 - File:ejemplo8 - Frame Span Loads GLOBAL-Z (VERT) - Ton-m Units

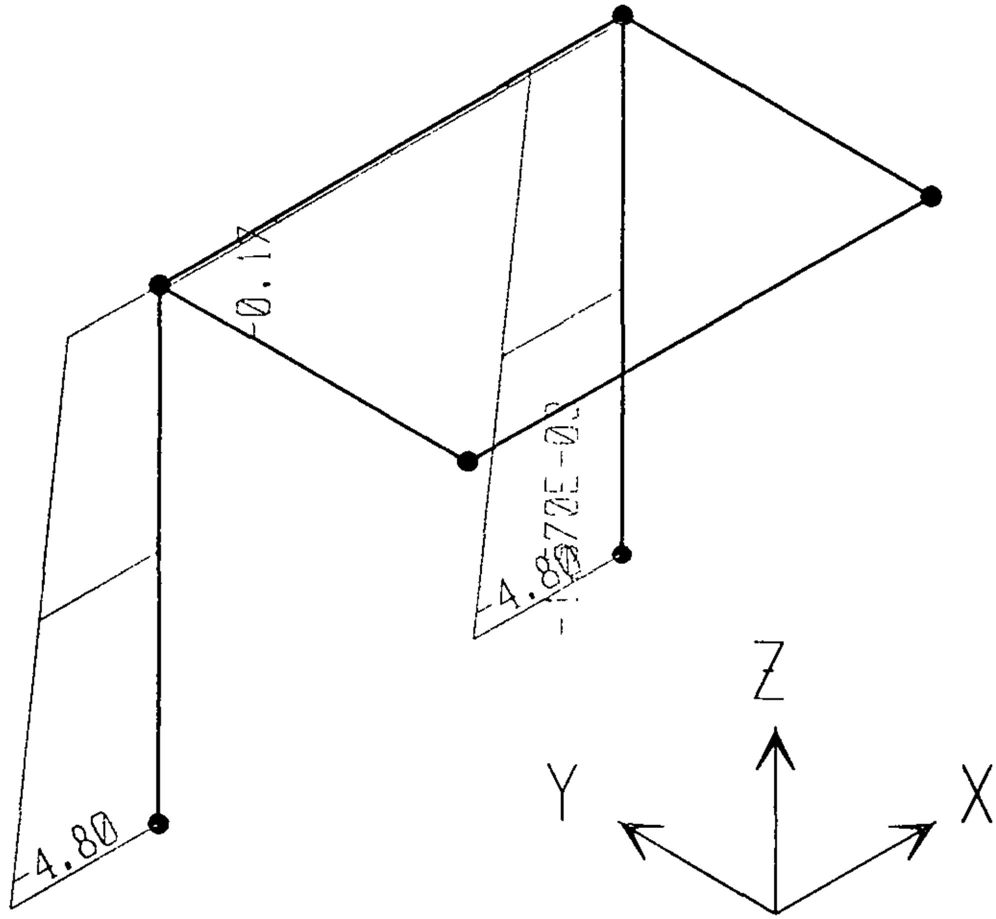


SAP2000 v6.10 - File:ejemplo8 - Joint Loads (SISMOY) - Ton-m Units

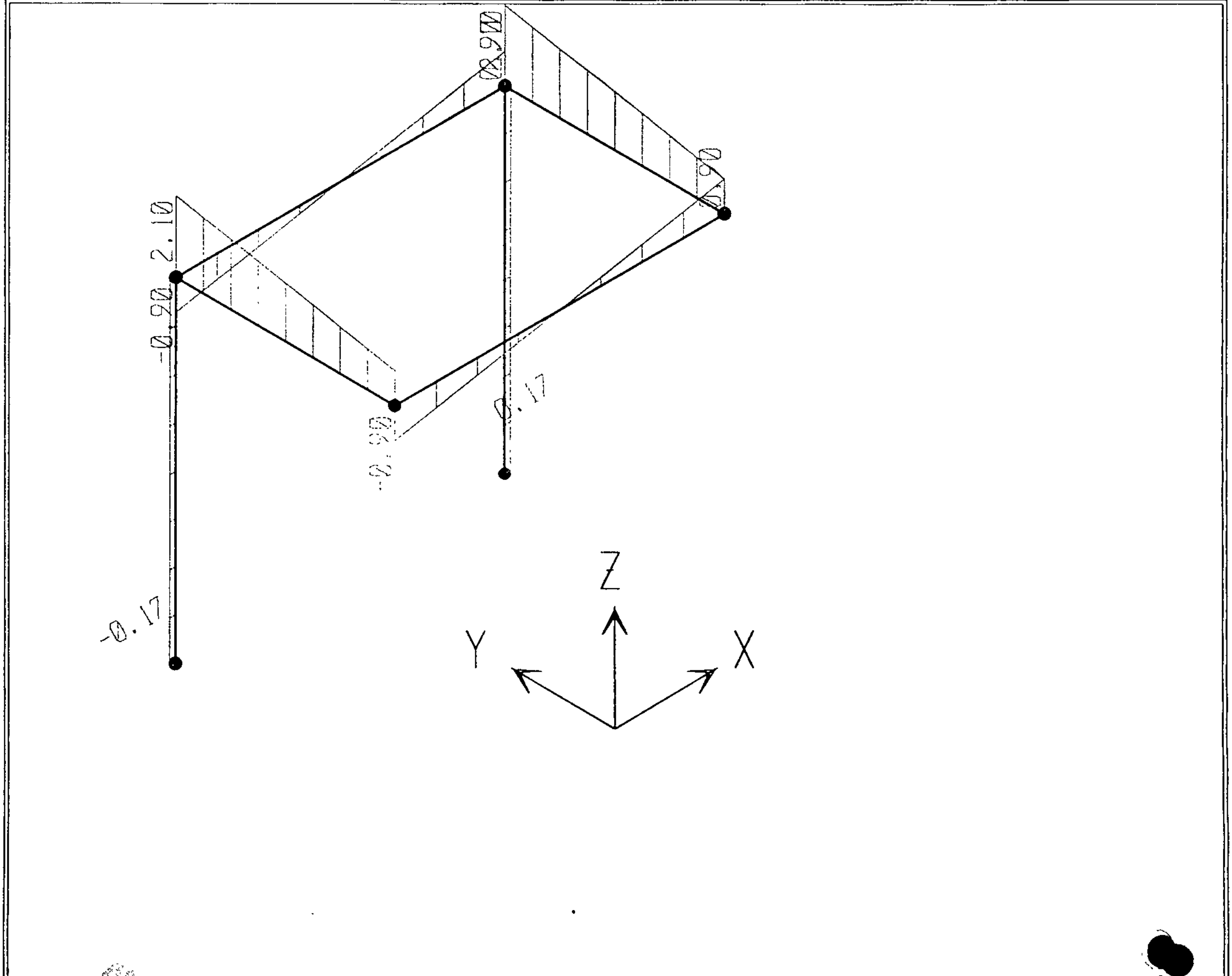


SAP2000 v6.[illegible] - File:ejemplo8 - Deformed Shape (PESOP) - Ton-m Units

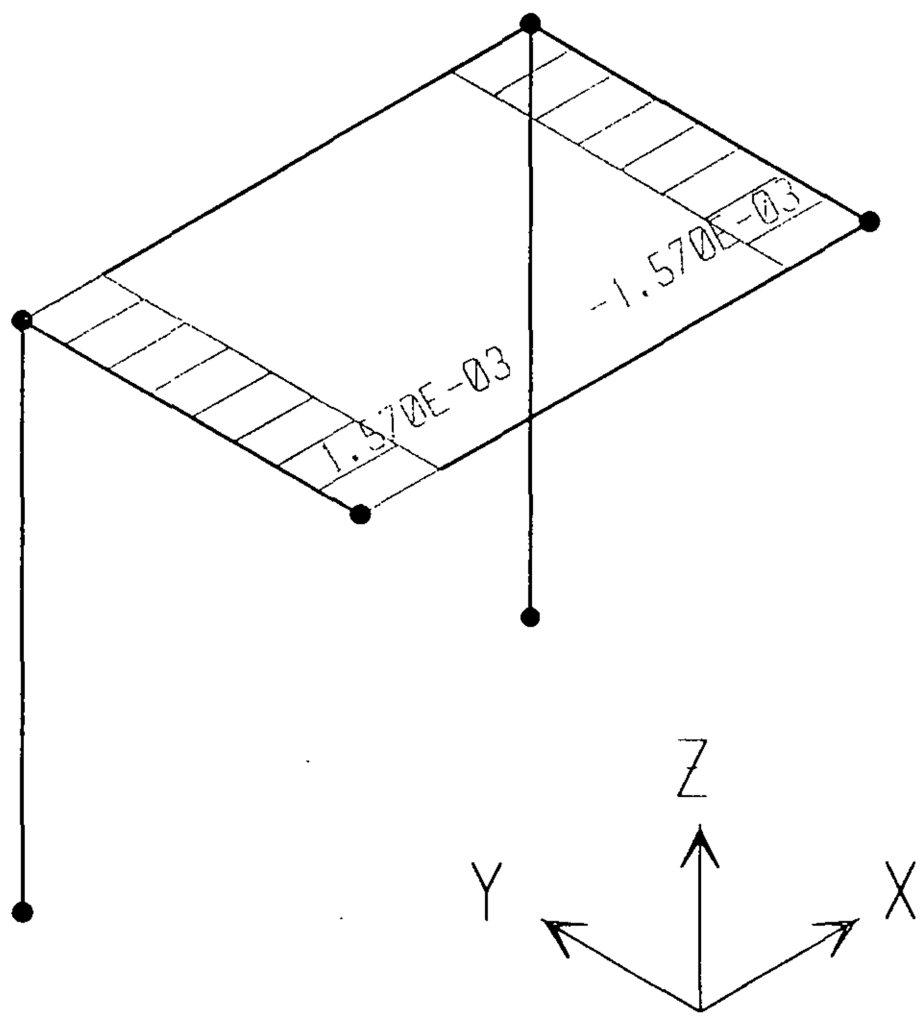
SAP2000



SAP2000 v6.10 - File:ejemplo8 - Axial Force Diagram (PESOP) - Ton-m Units

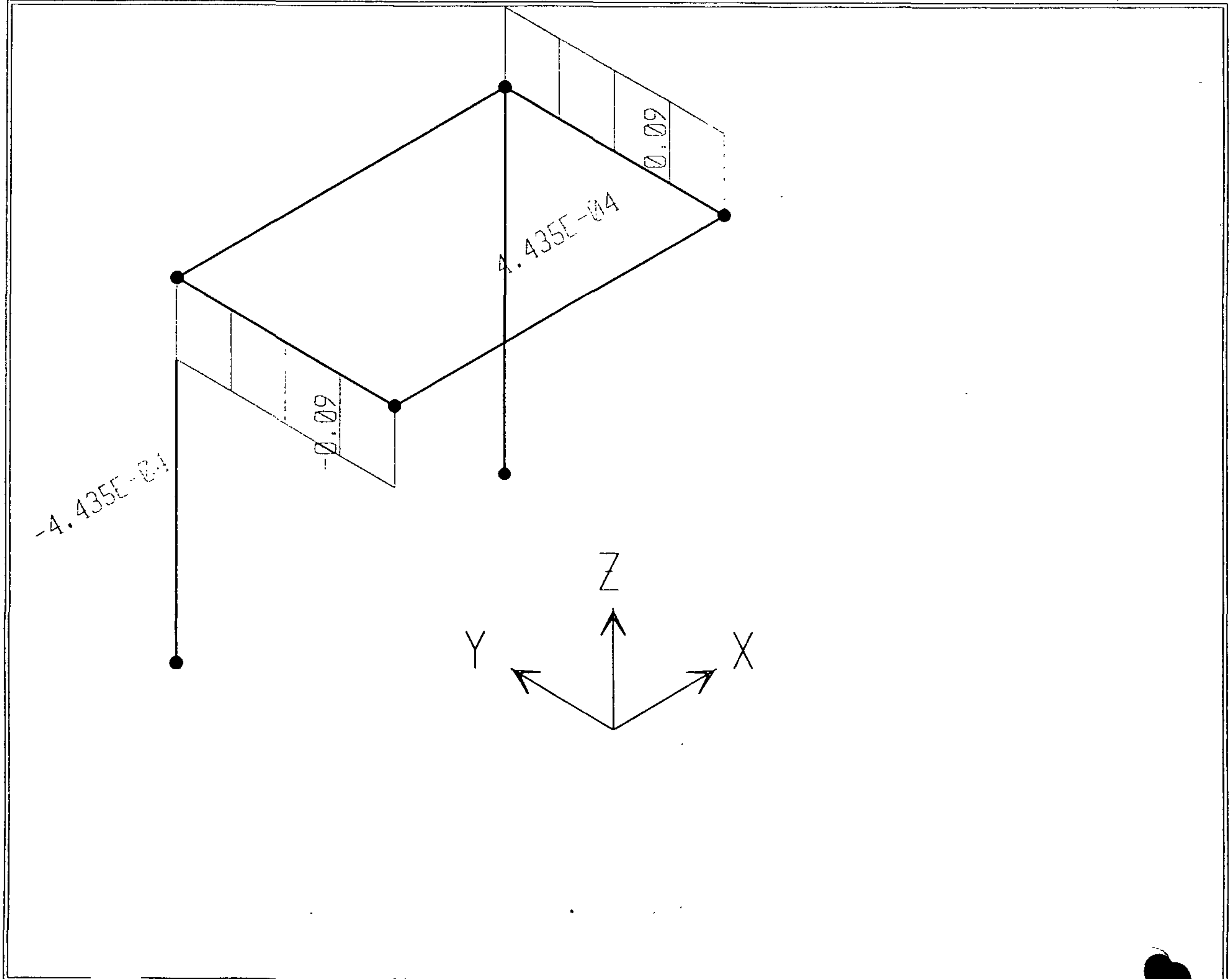


SAP2000 v6.10 - File:ejemplo8 - Shear Force 2-2 Diagram (PESOP) - Ton-m Units

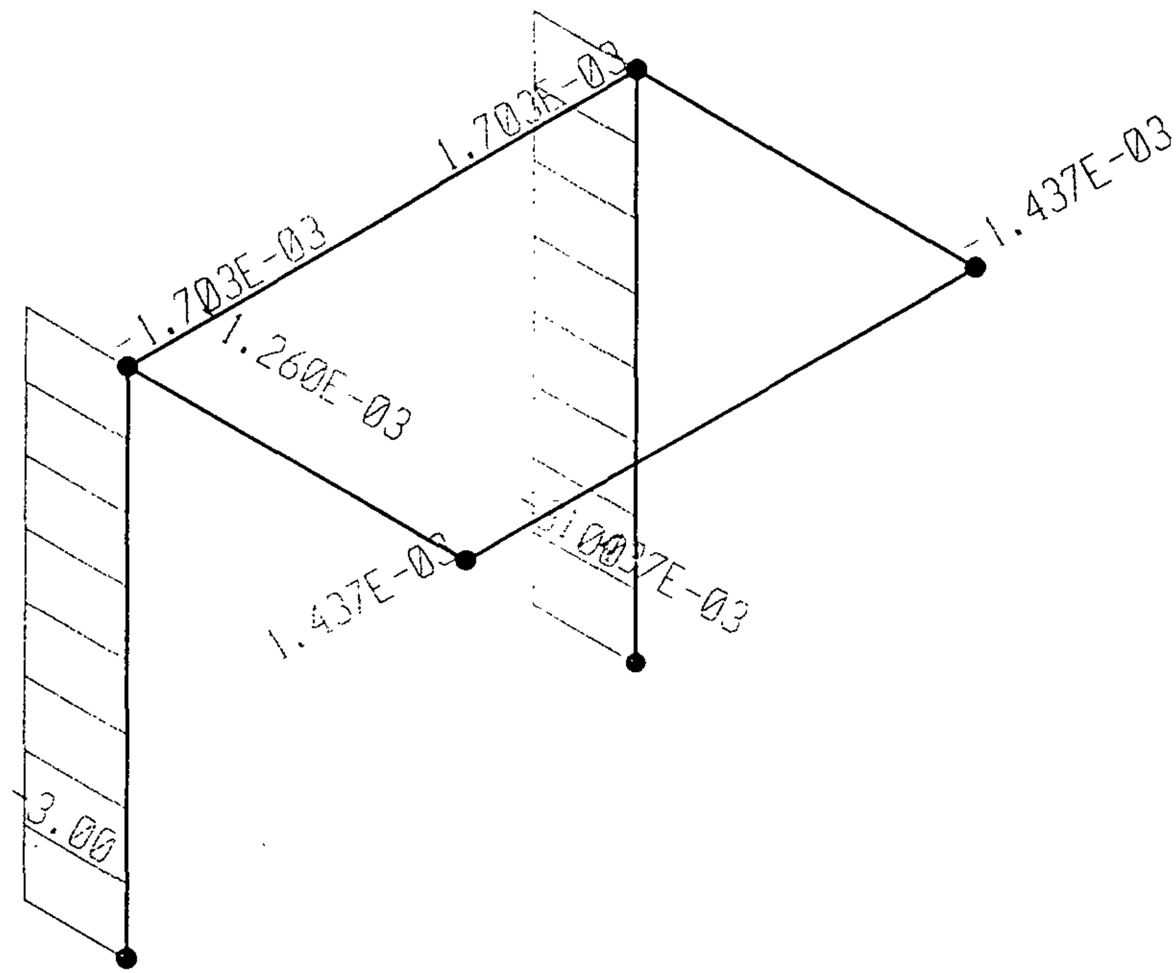


SAP2000 v6.10 - File:ejemplo8 - Shear Force 3-3 Diagram (PESOP) - Ton-m Units

SAP2000 Marzo 19,2000 14:34



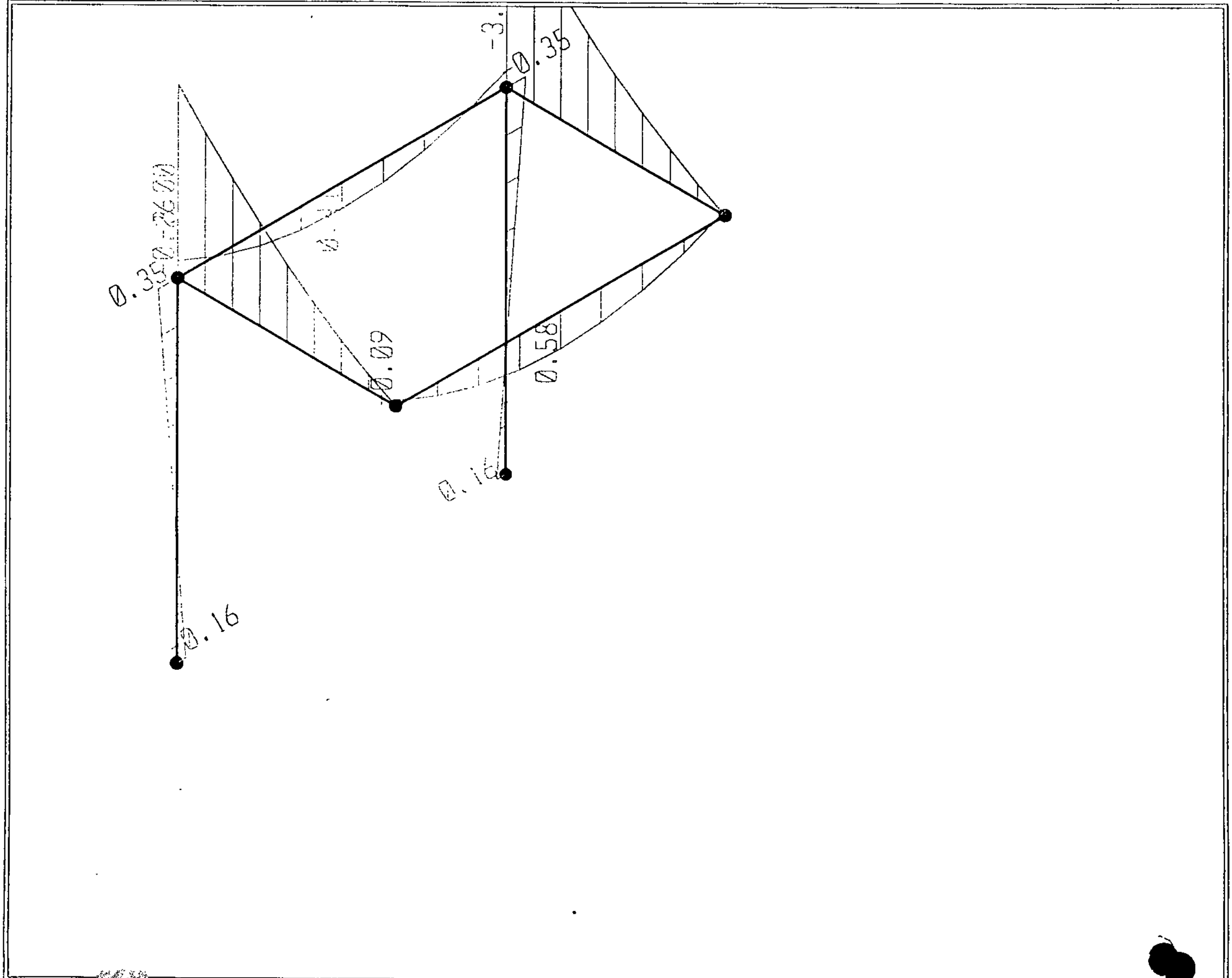
SAP2000 v. - File:ejemplo8 - Torsion Diagram (PESOP) - Ton-m Units



SAP2000 v6.10 - File:ejemplo8 - Moment 2-2 Diagram (PESOP) - Ton-m Units

SAP2000

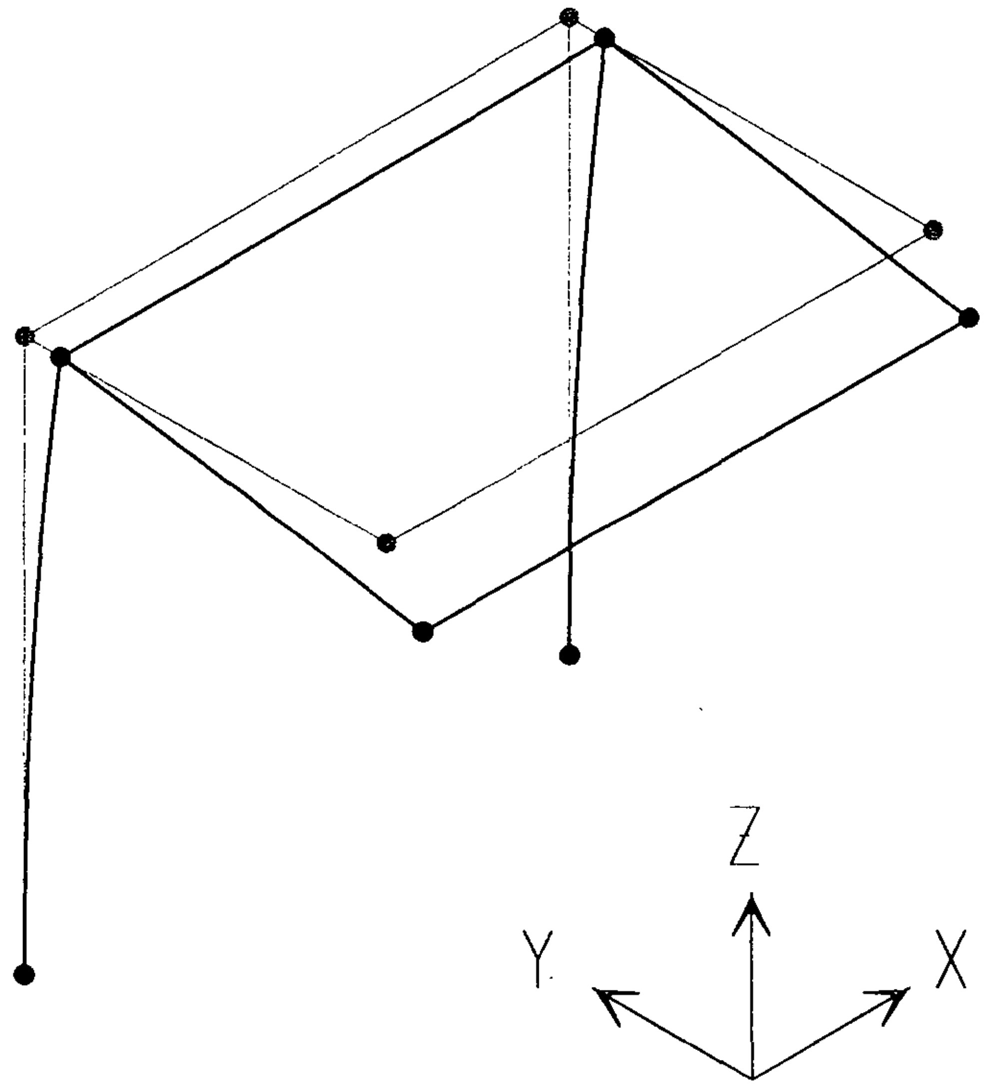
Marzo 19,2000 14:39



SAP2000 v6.1 - File:ejemplo8 - Moment 3-3 Diagram (PESOP) - Ton-m Units

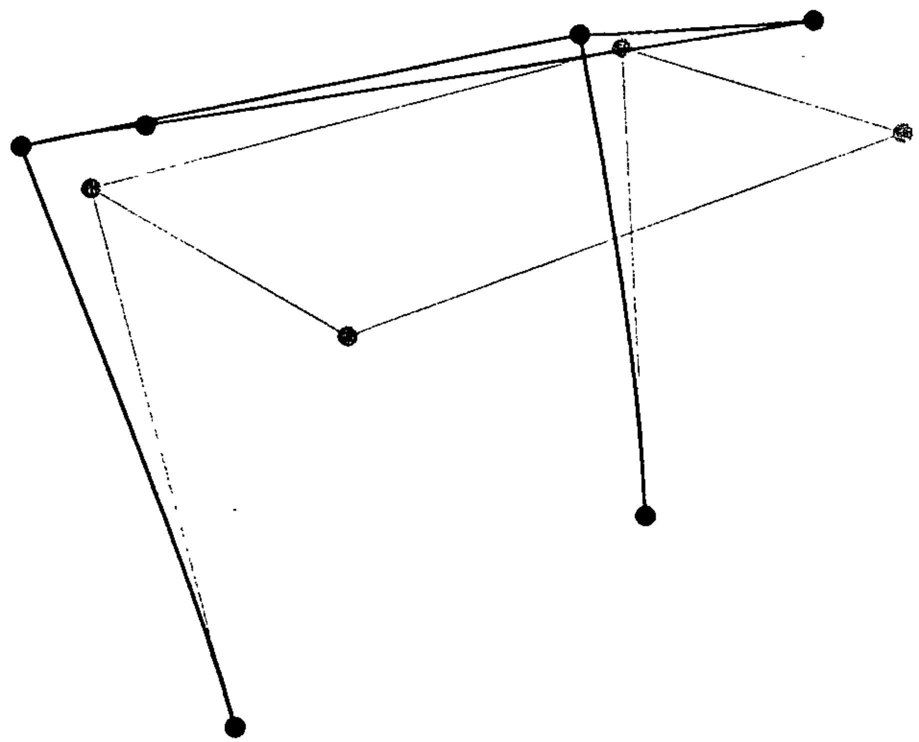
SAP2000

Marzo 19,2000 13:37



171

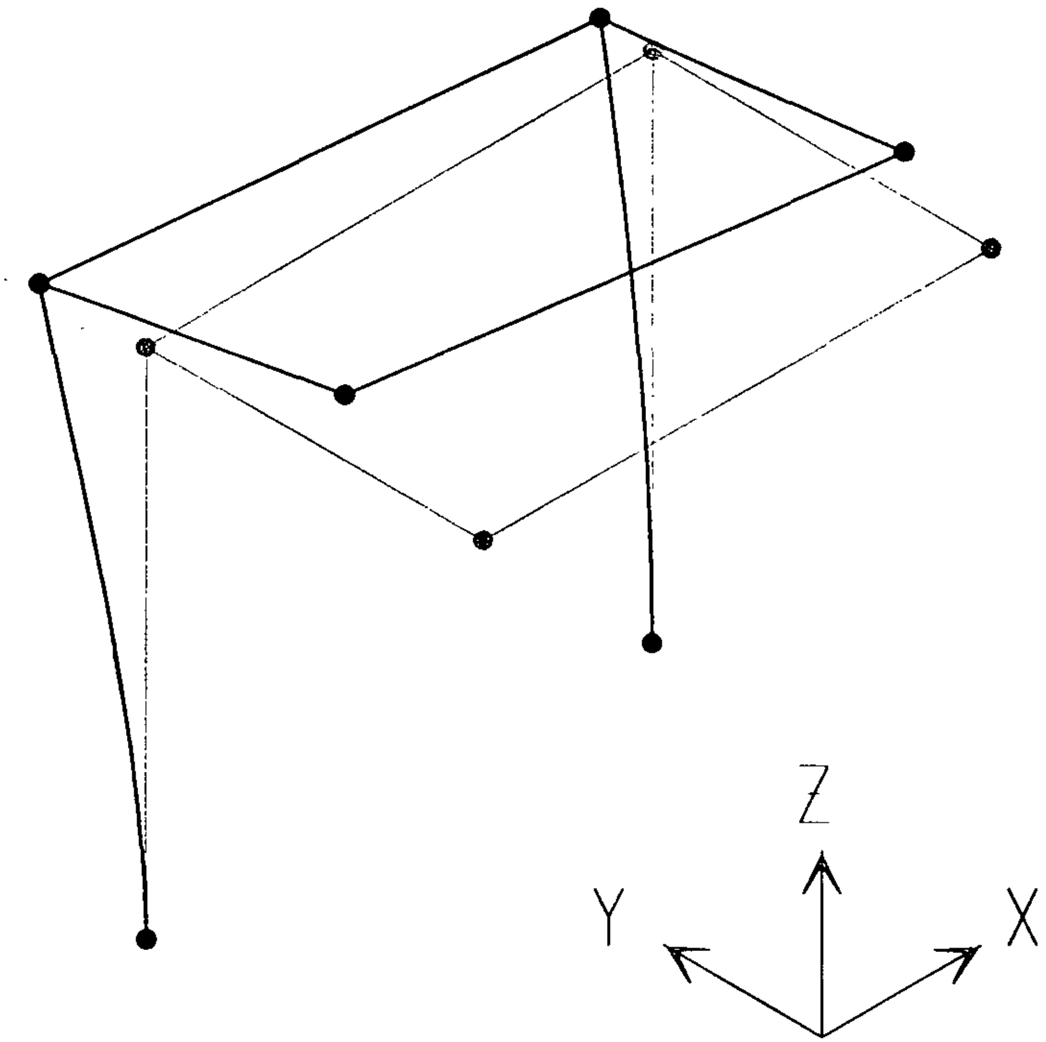
SAP2000 v6.10 - File:ejemplo8 - Deformed Shape (VERT) - Ton-m Units



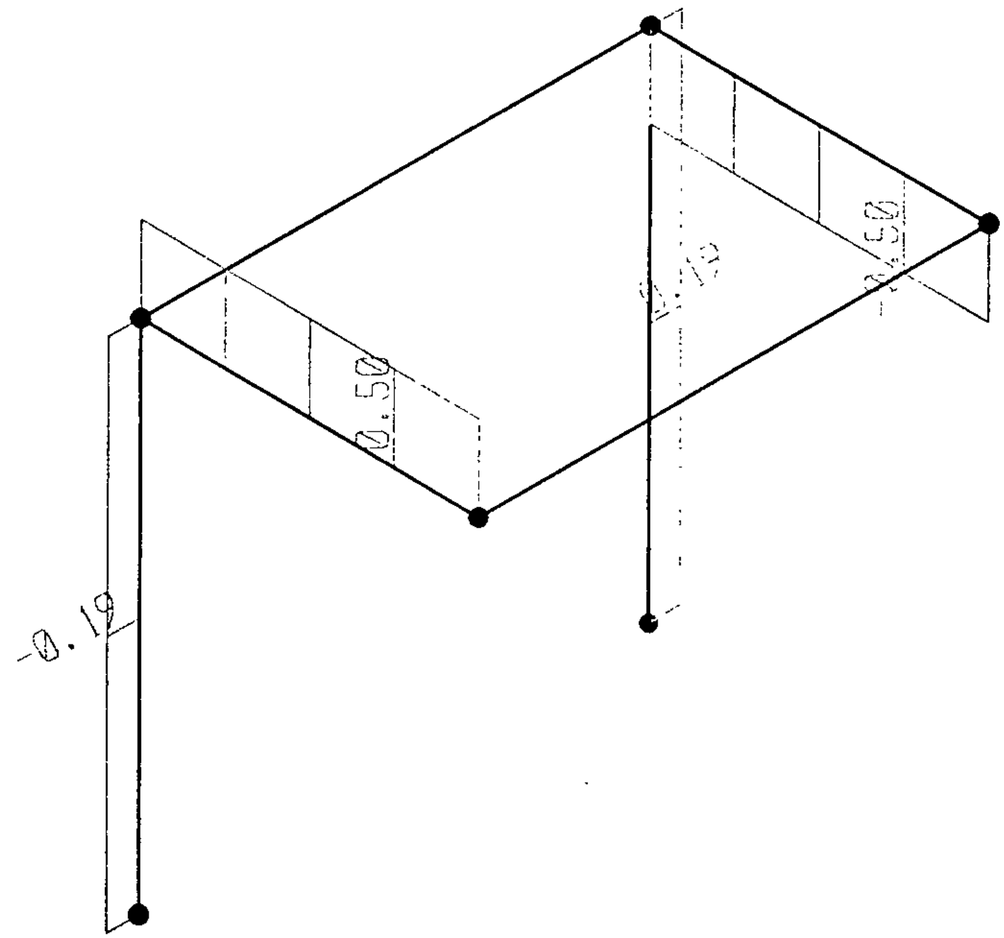
SAP2000 v[illegible] File:ejemplo8 - Deformed Shape (SISMOY) - Ton-m Units

SAP2000

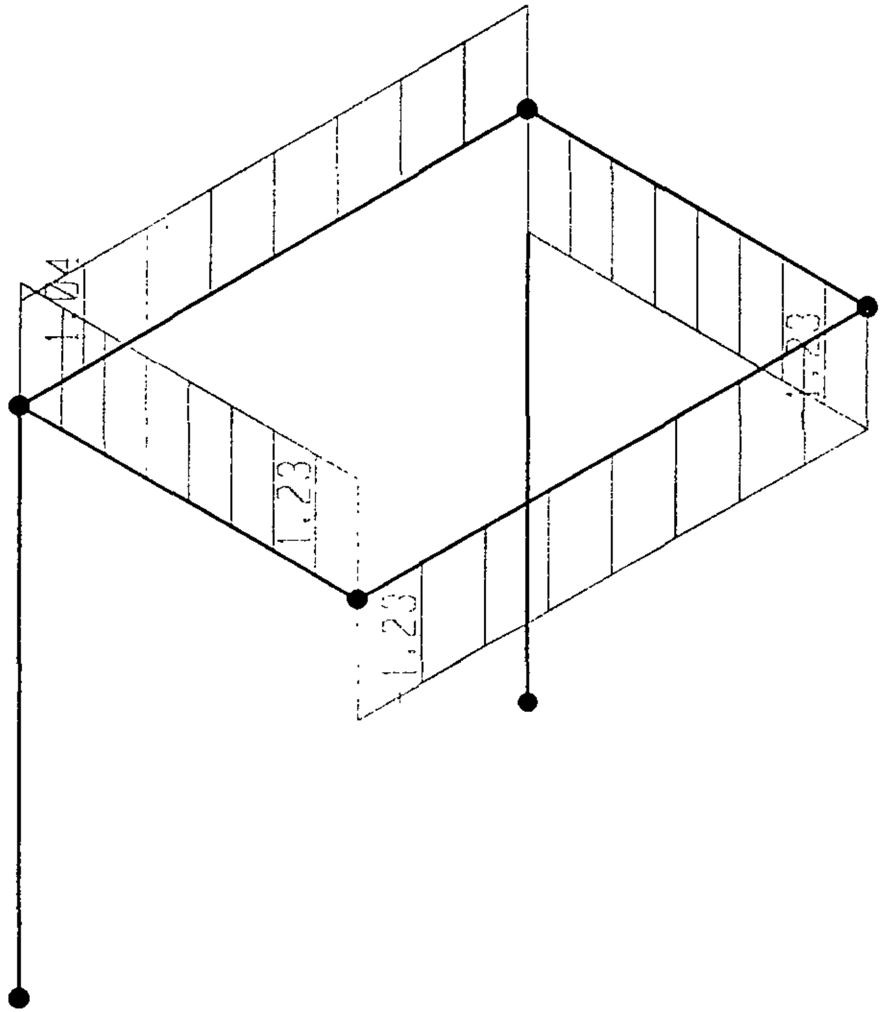
Marzo 19,2000 14:45



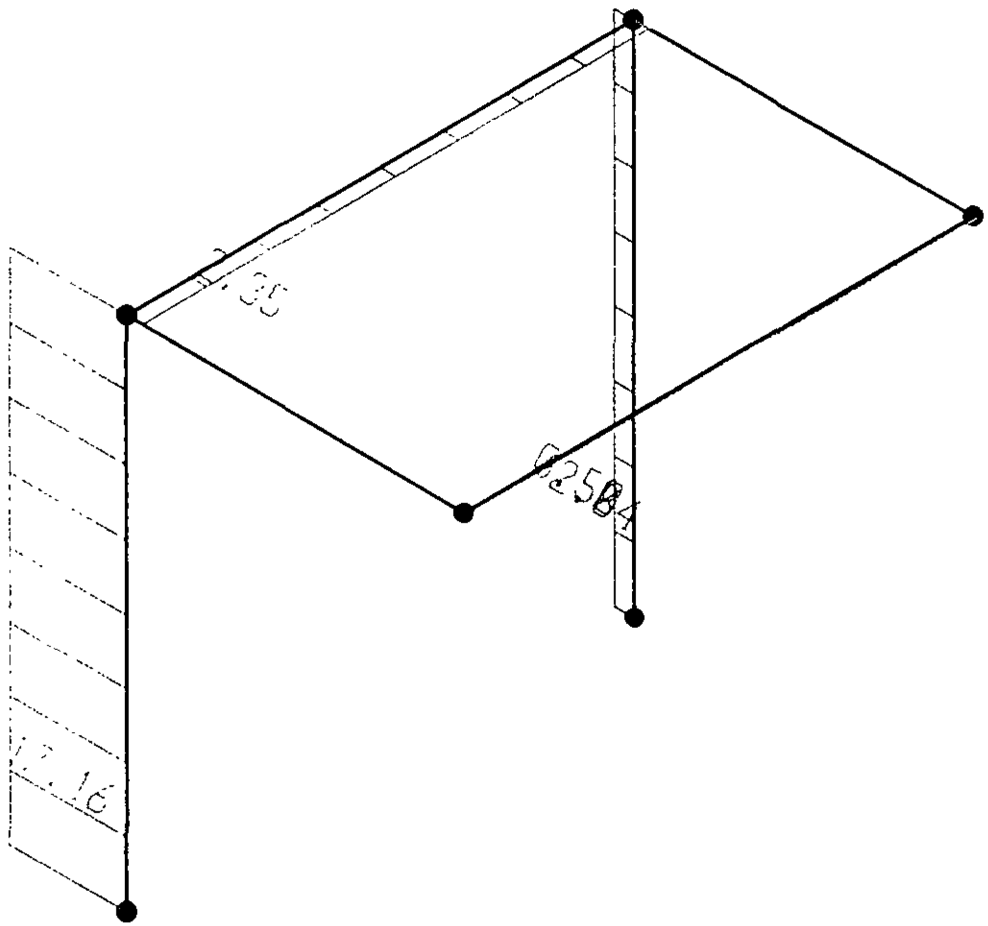
SAP2000 v6.10 - File:ejemplo8 - Deformed Shape (SISMOY) - Ton-m Units



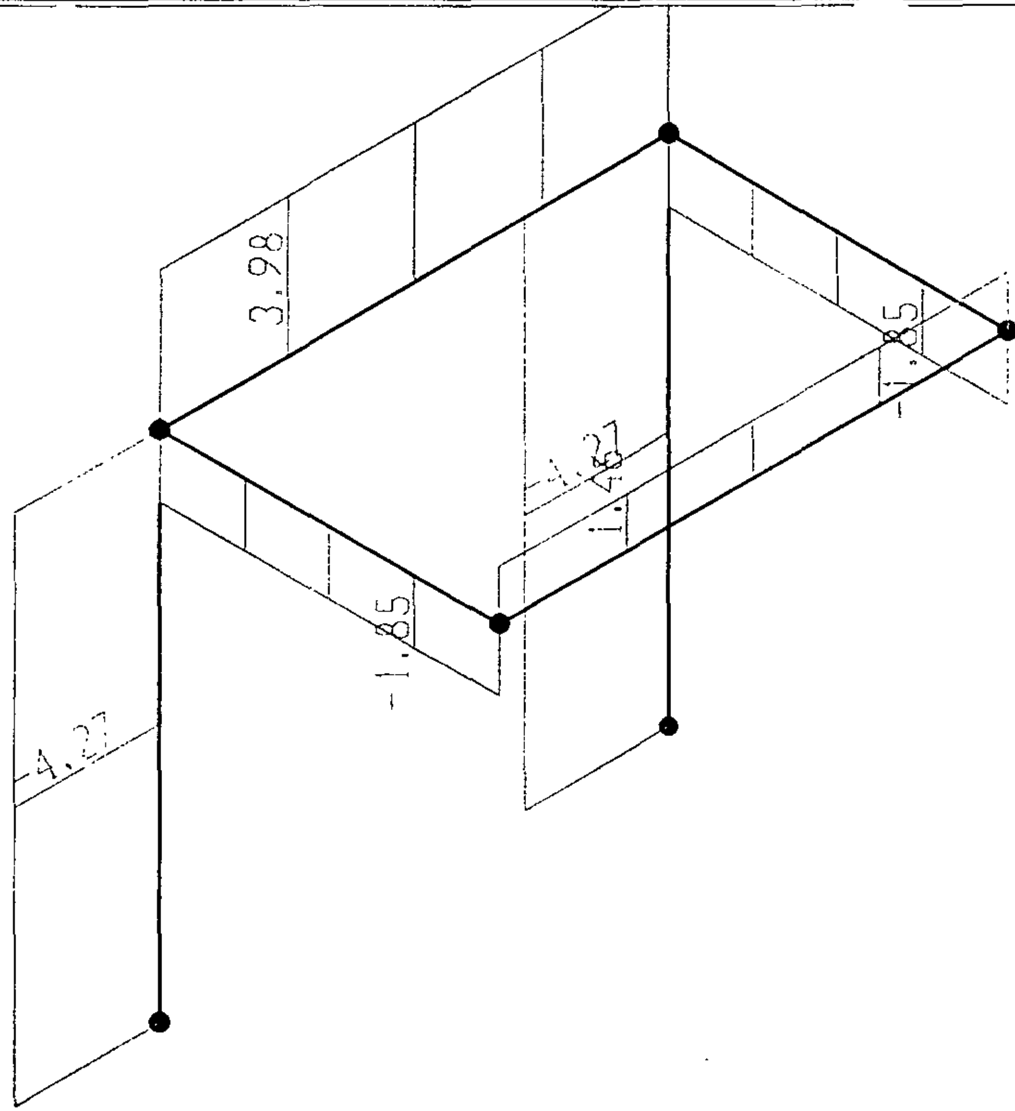
SAP2000 v6.1 - File:ejemplo8 - Axial Force Diagram (SISMOY) - Ton-m Units



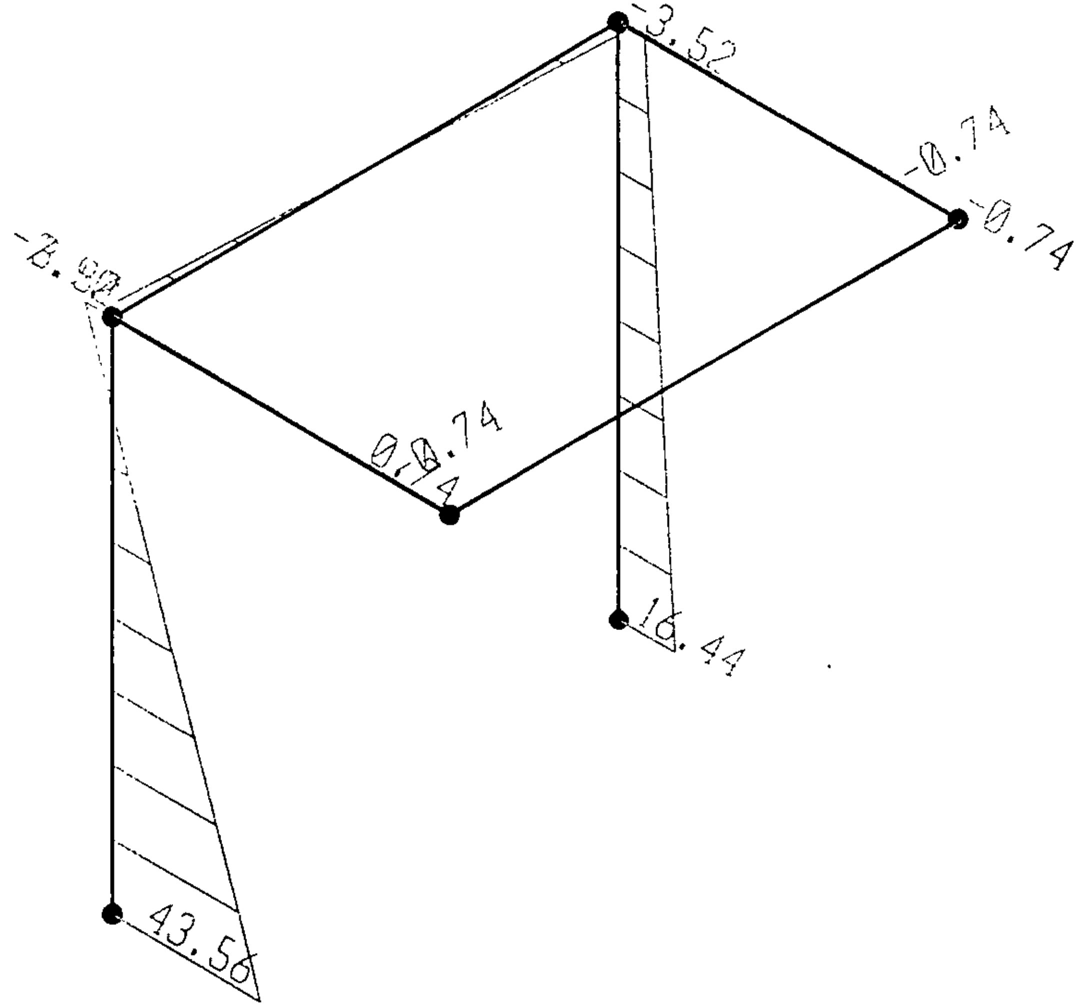
SAP2000 v6.10 - File:ejemplo8 - Shear Force 2-2 Diagram (SISMOY) - Ton-m Units

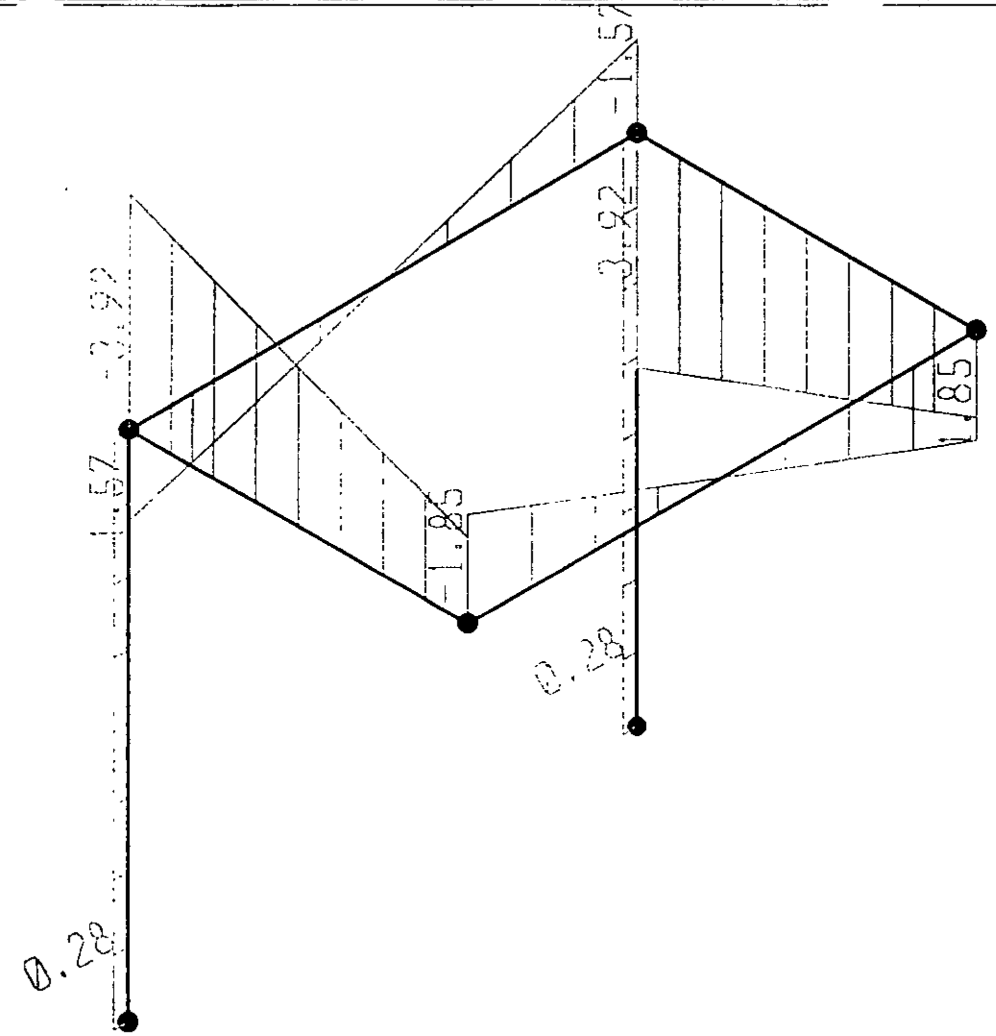


SAP2000 v - File:ejemplo8 - Shear Force 3-3 Diagram (SISMOY) - Ton-m



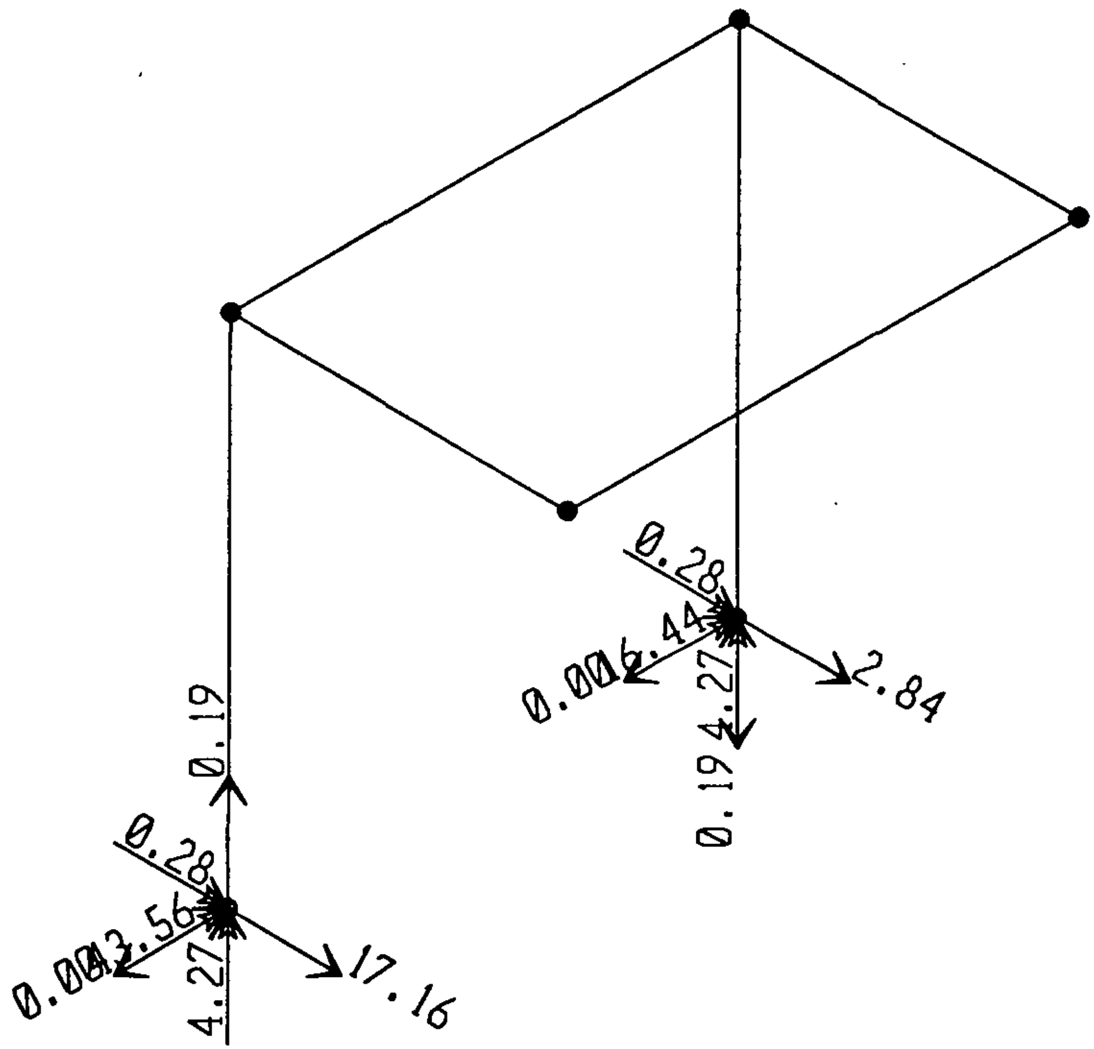
SAP2000 v6.10 - File:ejemplo8 - Torsion Diagram (SISMOY) - Ton-m Units


-3.52
-0.74
-0.74
-2.92
0.0.74
-0.74
16.44
43.56



179

SAP2000 v6.10 - File:ejemplo8 - Moment 3-3 Diagram (SISMOY) - Ton-m Units


0.28
6.44
0.00
4.27
2.84
0.19
0.19
0.28
3.56
0.00
4.27
17.16

```
; File C:\Mis documentos\curso sap 2000\ejemplo8.s2k saved 3/19/00 15:01:06 in Ton-m

SYSTEM
  DOF=UX,UY,UZ,RX,RY,RZ  LENGTH=m  FORCE=Ton  LINES=59

JOINT
  2  X=-1.5  Y=.5  Z=3
  3  X=-1.5  Y=2.5  Z=0
  4  X=-1.5  Y=2.5  Z=3
  6  X=1.5  Y=.5  Z=3
  7  X=1.5  Y=2.5  Z=0
  8  X=1.5  Y=2.5  Z=3

RESTRAINT
  ADD=3  DOF=U1,U2,U3,R1,R2,R3
  ADD=7  DOF=U1,U2,U3,R1,R2,R3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=CONC250  IDES=C  M=.244  W=2.4
    T=0  E=2213510  U=.2  A=.0000099

FRAME SECTION
  NAME=FSEC1 MAT=STEEL SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125
  NAME=FS1 MAT=CONC250 SH=R T=.5,.5 A=.25 J=8.802084E-03 I=5.208333E-03,5.208333E-03 AS=.2083333,.2083333

FRAME
  2  J=3,4  SEC=FS1  NSEG=2  ANG=0
  4  J=7,8  SEC=FS1  NSEG=2  ANG=0
  5  J=2,6  SEC=FS1  NSEG=4  ANG=0
  6  J=4,8  SEC=FS1  NSEG=4  ANG=0
  7  J=2,4  SEC=FS1  NSEG=4  ANG=0
  8  J=6,8  SEC=FS1  NSEG=4  ANG=0

LOAD
  NAME=PESOP  SW=1
  NAME=VERT
    TYPE=DISTRIBUTED SPAN
      ADD=5  RD=0,1  UZ=-1,-1
      ADD=6  RD=0,1  UZ=-1,-1
      ADD=7  RD=0,1  UZ=-1,-1
      ADD=8  RD=0,1  UZ=-1,-1
  NAME=SISMOY
    TYPE=FORCE
      ADD=4  UY=20

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=PESOP
  ELEM=JOINT  TYPE=DISP  LOAD=VERT
  ELEM=JOINT  TYPE=APPL  LOAD=PESOP
  ELEM=JOINT  TYPE=APPL  LOAD=VERT
  ELEM=FRAME  TYPE=FORCE  LOAD=PESOP
  ELEM=FRAME  TYPE=FORCE  LOAD=VERT
  ELEM=FRAME  TYPE=JOINTF  LOAD=PESOP
  ELEM=FRAME  TYPE=JOINTF  LOAD=VERT

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  -1.5
  GRID GLOBAL  X "2"  1.5
  GRID GLOBAL  Y "3"  -2.5
  GRID GLOBAL  Y "4"  2.5
  GRID GLOBAL  Z "5"  0
  GRID GLOBAL  Z "6"  3
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  MATERIAL CONC250 FYREBAR 42 FYSHEAR 21 FC 2.5 FCSHEAR 2
  FRAMESECTION FS1 NAME REC50X50
  CONCRETESECTION REC50X50 COLUMN  COVER .05 REBAR RR-3-3
  STATICLOAD PESOP TYPE DEAD
  STATICLOAD VERT TYPE LIVE
  STATICLOAD SISMOY TYPE QUAKE
END SUPPLEMENTAL DATA
```

S A P 2 0 0 0

Structural Analysis Programs

Version 6.10

It is the responsibility of the user to verify all
results produced by this program

19 Mar 2000 13:35:45

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 1
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

S Y S T E M D A T A

STEADY STATE LOAD FREQUENCY - - - - - - - - 0.0000E+00

LENGTH UNITS - - - - - - - - - - - - - - - - - M
FORCE UNITS - - - - - - - - - - - - - - - - - TON

UP DIRECTION - - - - - - - - - - - - - - - - - +Z

GLOBAL DEGREES OF FREEDOM - - - - - - - - - - ALL

PAGINATION BY - - - - - - - - - - - - - - - - LINES
NUMBER OF LINES PER PAGE - - - - - - - - - - 59
INCLUDE WARNING MESSAGES IN OUTPUT FILE - - Y

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 2
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

G E N E R A T E D J O I N T C O O R D I N A T E S

JOINT	X	Y	Z
2	-1.500	0.500	3.000
3	-1.500	2.500	0.000
4	-1.500	2.500	3.000
6	1.500	0.500	3.000
7	1.500	2.500	0.000
8	1.500	2.500	3.000

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 3
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

P A T T E R N S

PATTERN	JOINT	VALUE
DEFAULT		

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 4
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

R E S T R A I N T D A T A

JOINT						
3	U1	U2	U3	R1	R2	R3
7	U1	U2	U3	R1	R2	R3

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 5
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

M A T E R I A L P R O P E R T Y D A T A

MAT LABEL	NUMBER TEMPS	WEIGHT PER UNIT VOL	MASS PER UNIT VOL	DESIGN CODE
STEEL	1	0.7833E+01	0.7981E+00	S
CONC	1	0.2403E+01	0.2448E+00	C
CONC250	1	0.2400E+01	0.2440E+00	C

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 6
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

T E M P E R A T U R E D E P E N D E N T D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	MODULUS OF ELASTICITY E1	E2	E3	SHEAR MODULII G12	G13	G23
STEEL	0.00	0.204E+08	0.204E+08	0.204E+08	0.784E+07	0.784E+07	0.784E+07
CONC	0.00	0.253E+07	0.253E+07	0.253E+07	0.105E+07	0.105E+07	0.105E+07
CONC250	0.00	0.221E+07	0.221E+07	0.221E+07	0.922E+06	0.922E+06	0.922E+06

T E M P E R A T U R E D E P E N D E N T D A T A

THERMAL EXPANSION COEFFICIENTS

MAT LABEL	TEMP	COEFFICIENTS OF THERMAL EXPANSION A1	A2	A3	A12	A13	A23
STEEL	0.00	0.117E-04	0.117E-04	0.117E-04	0.000E+00	0.000E+00	0.000E+00
CONC	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00
CONC250	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00

T E M P E R A T U R E D E P E N D E N T D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	POISSONS RATIO U12	U13	U23	U14	U24	U34	U15	U25	U35	U45	U16	U26	U36	U46	U56
STEEL	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONC	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONC250	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

MATERIAL PROPERTIES

MAT LABEL	TEMP	YIELD FY
CONC	0.00	36.00
CONC250	0.00	36.00

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	SHAPE TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
FSEC1	R	0.500	0.300				
FS1	R	0.500	0.500				

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	AXIAL AREA	TORSIONAL CONSTANT	MOMENTS OF INERTIA I33	I22	SHEAR AREAS A2	A3
FSEC1	0.150E+00	0.282E-02	0.313E-02	0.113E-02	0.125E+00	0.125E+00
FS1	0.250E+00	0.880E-02	0.521E-02	0.521E-02	0.208E+00	0.208E+00

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	MAT LABEL	ADDITIONAL MASS PER LENGTH	ADDITIONAL WEIGHT PER LENGTH
FSEC1	STEEL	0.000E+00	0.000E+00
FS1	CONC250	0.000E+00	0.000E+00

F R A M E E L E M E N T D A T A

ELEMENT LABEL	JOINT END-I	JOINT END-J	ELEMENT LENGTH	END-OFFSET-LENGTHS END-I	END-J	RIGID-END FACTOR	NUMBER OF SEGMENTS
2	3	4	3.000	0.000	0.000	0.0000	2
4	7	8	3.000	0.000	0.000	0.0000	2
5	2	6	3.000	0.000	0.000	0.0000	4
6	4	8	3.000	0.000	0.000	0.0000	4
7	2	4	2.000	0.000	0.000	0.0000	4
8	6	8	2.000	0.000	0.000	0.0000	4

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 14
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

F R A M E E L E M E N T D A T A

ELEMENT LABEL	SECTION LABEL	LOCAL PLANE	COORD SYSTEM	PLN 1ST	PLN 2ND	PLANE JOINTA	PLANE JOINTB	COORD ANGLE
2	FS1	12	0	+Z	+X	0	0	0.00
4	FS1	12	0	+Z	+X	0	0	0.00
5	FS1	12	0	+Z	+X	0	0	0.00
6	FS1	12	0	+Z	+X	0	0	0.00
7	FS1	12	0	+Z	+X	0	0	0.00
8	FS1	12	0	+Z	+X	0	0	0.00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 15
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

T O T A L W E I G H T S A N D M A S S E S

SECTION LABEL	WEIGHT	MASS
FS1	9.6000	0.9760
TOTAL	9.6000	0.9760

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 16
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

L O A D C O N D I T I O N PESOP

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.1000E+01

L O A D C O N D I T I O N VERT

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.0000E+00

DISTRIBUTED SPAN LOADS ON FRAME ELEMENTS

ELEMENT LABEL	LOC DOF	DISTANCE AT START	DISTANCE AT END	FORCE AT START	FORCE AT END	MOMENT AT START	MOMENT AT END
5	U2	0.000E+00	0.100E+01	-0.100E+01	-0.100E+01		
6	U2	0.000E+00	0.100E+01	-0.100E+01	-0.100E+01		
7	U2	0.000E+00	0.100E+01	-0.100E+01	-0.100E+01		
8	U2	0.000E+00	0.100E+01	-0.100E+01	-0.100E+01		

L O A D C O N D I T I O N SISMOY

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.0000E+00

JOINT FORCES IN LOCAL COORDINATES

JOINT LABEL	FORCE 1	FORCE 2	FORCE 3	MOMENT 1	MOMENT 2	MOMENT 3
4	0.000E+00	0.200E+02	0.000E+00	0.000E+00	0.000E+00	0.000E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 17
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo8.EKO

O U T P U T S E L E C T I O N

DISPLACEMENTS AT JOINTS

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
PESOP					
VERT					

APPLIED AND INTERNAL LOADS AT JOINTS

LOAD	MODES	SPEC	HIST	MOVE	COMB

LABEL		LABEL	LABEL	LABEL	LABEL
PESOP VERT					

INTERNAL FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
PESOP VERT					

JOINT FORCES AT ELEMENT FRAME

LOAD LABEL	MODES	SPEC LABEL	HIST LABEL	MOVE LABEL	COMB LABEL
PESOP VERT					

I N P U T C O M P L E T E

S A P 2 0 0 0 (R)

Structural Analysis Programs

Version E6.10

19 Mar 2000 15:11:40

D I S P L A C E M E N T   D E G R E E S   O F   F R E E D O M

(A) = Active DOF, equilibrium equation
(-) = Restrained DOF, reaction computed
(+) = Constrained DOF
( ) = Null DOF

JOINTS	UX	UY	UZ	RX	RY	RZ
2	A	A	A	A	A	A
3	-	-	-	-	-	-
4 TO 6	A	A	A	A	A	A
7	-	-	-	-	-	-
8	A	A	A	A	A	A

J O I N T   D I S P L A C E M E N T S

TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

LOAD PESOP ------------------

JOINT	UX	UY	UZ	RX	RY	RZ
2	4.26E-09	-0.001171	-0.001910	0.001006	4.68E-05	-1.87E-07
3	.000000	.000000	.000000	.000000	.000000	.000000
4	4.62E-07	-0.001171	-2.11E-05	0.000781	2.45E-05	-1.64E-07
6	-4.26E-09	-0.001171	-0.001910	0.001006	-4.68E-05	1.87E-07
7	.000000	.000000	.000000	.000000	.000000	.000000
8	-4.62E-07	-0.001171	-2.11E-05	0.000781	-2.45E-05	1.64E-07

LOAD VERT ------------------

JOINT	UX	UY	UZ	RX	RY	RZ
2	7.09E-09	-0.001952	-0.003176	0.001677	7.80E-05	-3.12E-07
3	.000000	.000000	.000000	.000000	.000000	.000000
4	7.70E-07	-0.001952	-2.71E-05	0.001301	4.08E-05	-2.73E-07
6	-7.09E-09	-0.001952	-0.003176	0.001677	-7.80E-05	3.12E-07
7	.000000	.000000	.000000	.000000	.000000	.000000
8	-7.70E-07	-0.001952	-2.71E-05	0.001301	-4.08E-05	2.73E-07

LOAD SISMOY ------------------

JOINT	UX	UY	UZ	RX	RY	RZ
2	-0.003173	0.010573	0.008727	-0.004173	0.000527	-0.001706
3	.000000	.000000	.000000	.000000	.000000	.000000
4	0.000109	0.010575	-1.01E-06	-0.004639	7.27E-05	-0.001577
6	-0.003173	0.005352	0.006886	-0.003634	0.000527	-0.001706
7	.000000	.000000	.000000	.000000	.000000	.000000
8	0.000109	0.005350	1.01E-06	-0.003167	7.27E-05	-0.001577

A P P L I E D   L O A D S

FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	.000000	.000000	-1.500000	-0.200000	0.450000	.000000
3	.000000	.000000	-0.900000	.000000	.000000	.000000
4	.000000	.000000	-2.400000	0.200000	0.450000	.000000
6	.000000	.000000	-1.500000	-0.200000	-0.450000	.000000
7	.000000	.000000	-0.900000	.000000	.000000	.000000
8	.000000	.000000	-2.400000	0.200000	-0.450000	.000000

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	.000000	.000000	-2.500000	-0.333333	0.750000	.000000
4	.000000	.000000	-2.500000	0.333333	0.750000	.000000
6	.000000	.000000	-2.500000	-0.333333	-0.750000	.000000
8	.000000	.000000	-2.500000	0.333333	-0.750000	.000000

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
4	.000000	20.000000	.000000	.000000	.000000	.000000

G L O B A L   F O R C E   B A L A N C E

TOTAL FORCE AND MOMENT AT THE ORIGIN, IN GLOBAL COORDINATES

LOAD PESOP ------------------

	FX	FY	FZ	MX	MY	MZ
APPLIED	.000000	.000000	-9.600000	-18.000000	6.66E-16	.000000
REACTNS	1.39E-16	5.09E-14	9.600000	18.000000	-3.11E-15	-1.01E-13
TOTAL	1.39E-16	5.09E-14	-1.78E-15	-1.53E-13	-2.44E-15	-1.01E-13

LOAD VERT ------------------

	FX	FY	FZ	MX	MY	MZ
APPLIED	.000000	.000000	-10.000000	-15.000000	1.11E-15	.000000
REACTNS	3.33E-16	8.33E-14	10.000000	15.000000	-1.33E-15	-1.70E-13
TOTAL	3.33E-16	8.33E-14	.000000	-2.45E-13	-2.22E-16	-1.70E-13

LOAD SISMOY ------------------

	FX	FY	FZ	MX	MY	MZ
APPLIED	.000000	20.000000	.000000	-60.000000	.000000	-30.000000
REACTNS	-2.61E-14	-20.000000	4.55E-15	60.000000	-6.40E-14	30.000000
TOTAL	-2.61E-14	-4.83E-13	4.55E-15	1.44E-12	-6.40E-14	8.56E-13

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 2 ------------------

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	0.172089	4.97E-14	4.800000	-3.000000	0.164021	0.000443
4	-0.172089	-4.97E-14	-3.000000	3.000000	0.352246	-0.000443

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	0.286815	8.22E-14	5.000000	-5.000000	0.273369	0.000739
4	-0.286815	-8.22E-14	-5.000000	5.000000	0.587077	-0.000739

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
3	-1.33E-14	-17.155639	0.186242	43.562544	-0.279363	4.266542
4	1.33E-14	17.155639	-0.186242	7.904373	0.279363	-4.266542

ELEM 4 ------------------

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
7	-0.172089	1.20E-15	4.800000	-3.000000	-0.164021	-0.000443
8	0.172089	-1.20E-15	-3.000000	3.000000	-0.352246	0.000443

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
7	-0.286815	1.02E-15	5.000000	-5.000000	-0.273369	-0.000739
8	0.286815	-1.02E-15	-5.000000	5.000000	-0.587077	0.000739

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
7	-1.28E-14	-2.844361	-0.186242	16.437456	-0.279363	4.266542
8	1.28E-14	2.844361	0.186242	-7.904373	0.279363	-4.266542

ELEM 5 ------------------

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	0.001570	-2.41E-15	0.900000	4.16E-15	-0.090470	-0.001437
6	-0.001570	2.41E-15	0.900000	-4.16E-15	0.090470	0.001437

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	0.002617	-3.20E-15	1.500000	7.28E-15	-0.150784	-0.002395
6	-0.002617	3.20E-15	1.500000	-7.28E-15	0.150784	0.002395

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	-1.93E-14	0.495706	1.230498	-1.459080	-1.845747	0.743559
6	1.93E-14	-0.495706	-1.230498	1.459080	-1.845747	0.743559

ELEM 6 ------------------

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
4	0.170519	-7.39E-15	0.900000	1.26E-14	-0.261775	-0.001260
8	-0.170519	7.39E-15	0.900000	-1.26E-14	0.261775	0.001260

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
4	0.284199	-1.34E-14	1.500000	2.11E-14	-0.436292	-0.002099
8	-0.284199	1.34E-14	1.500000	-2.11E-14	0.436292	0.002099

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
4	3.82E-15	2.348655	-1.044256	-3.984296	1.566385	3.522982
8	-3.82E-15	-2.348655	1.044256	3.984296	1.566385	3.522982

ELEM 7 ------------------

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	-0.001570	-1.94E-16	-0.900000	-7.52E-15	0.090470	0.001437
4	0.001570	1.94E-16	2.100000	-3.000000	-0.090470	0.001703

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	-0.002617	4.07E-14	-1.500000	-8.60E-15	0.150784	0.002395
4	0.002617	-4.07E-14	3.500000	-5.000000	-0.150784	0.002838

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
2	1.29E-14	-0.495706	-1.230498	1.459080	1.845747	-0.743559
4	-1.29E-14	0.495706	1.230498	-3.920077	-1.845747	0.743559

F R A M E   E L E M E N T   J O I N T   F O R C E S

FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES

ELEM 8 ------------------

LOAD PESOP ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
6	0.001570	3.49E-14	-0.900000	5.22E-15	-0.090470	-0.001437
8	-0.001570	-3.49E-14	2.100000	-3.000000	0.090470	-0.001703

LOAD VERT ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
6	0.002617	7.92E-14	-1.500000	1.08E-14	-0.150784	-0.002395
8	-0.002617	-7.92E-14	3.500000	-5.000000	0.150784	-0.002838

LOAD SISMOY ------------------

JOINT	FX	FY	FZ	MX	MY	MZ
6	1.33E-14	0.495706	1.230498	-1.459080	1.845747	-0.743559
8	-1.33E-14	-0.495706	-1.230498	3.920077	-1.845747	0.743559

FRAME ELEMENT INTERNAL FORCES

ELEM 2 ------------------ LENGTH = 3.000000

LOAD PESOP ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-4.800000	-0.172089	-4.97E-14	-0.000443	-3.000000	-0.164021
0.50000	-3.900000	-0.172089	-4.97E-14	-0.000443	-3.000000	0.094112
1.00000	-3.000000	-0.172089	-4.97E-14	-0.000443	-3.000000	0.352246

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-5.000000	-0.286815	-8.22E-14	-0.000739	-5.000000	-0.273369
0.50000	-5.000000	-0.286815	-8.22E-14	-0.000739	-5.000000	0.156854
1.00000	-5.000000	-0.286815	-8.22E-14	-0.000739	-5.000000	0.587077

LOAD SISMOY ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-0.186242	1.33E-14	17.155639	-4.266542	43.562544	0.279363
0.50000	-0.186242	1.33E-14	17.155639	-4.266542	17.829086	0.279363
1.00000	-0.186242	1.33E-14	17.155639	-4.266542	-7.904373	0.279363

ELEM 4 ------------------ LENGTH = 3.000000

LOAD PESOP ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-4.800000	0.172089	-1.20E-15	0.000443	-3.000000	0.164021
0.50000	-3.900000	0.172089	-1.20E-15	0.000443	-3.000000	-0.094112
1.00000	-3.000000	0.172089	-1.20E-15	0.000443	-3.000000	-0.352246

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-5.000000	0.286815	-1.02E-15	0.000739	-5.000000	0.273369
0.50000	-5.000000	0.286815	-1.02E-15	0.000739	-5.000000	-0.156854
1.00000	-5.000000	0.286815	-1.02E-15	0.000739	-5.000000	-0.587077

LOAD SISMOY ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	0.186242	1.28E-14	2.844361	-4.266542	16.437456	0.279363
0.50000	0.186242	1.28E-14	2.844361	-4.266542	12.170914	0.279363
1.00000	0.186242	1.28E-14	2.844361	-4.266542	7.904373	0.279363

ELEM 5 ------------------ LENGTH = 3.000000

LOAD PESOP ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-0.001570	-0.900000	-2.41E-15	-4.16E-15	-0.001437	-0.090470
0.25000	-0.001570	-0.450000	-2.41E-15	-4.16E-15	-0.001437	0.415780

FRAME ELEMENT INTERNAL FORCES

REL DIST	P	V2	V3	T	M2	M3
0.50000	-0.001570	5.80E-15	-2.41E-15	-4.16E-15	-0.001437	0.584530
0.75000	-0.001570	0.450000	-2.41E-15	-4.16E-15	-0.001437	0.415780
1.00000	-0.001570	0.900000	-2.41E-15	-4.16E-15	-0.001437	-0.090470

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-0.002617	-1.500000	-3.20E-15	-7.28E-15	-0.002395	-0.150784
0.25000	-0.002617	-0.750000	-3.20E-15	-7.28E-15	-0.002395	0.692966
0.50000	-0.002617	8.66E-15	-3.20E-15	-7.28E-15	-0.002395	0.974216
0.75000	-0.002617	0.750000	-3.20E-15	-7.28E-15	-0.002395	0.692966
1.00000	-0.002617	1.500000	-3.20E-15	-7.28E-15	-0.002395	-0.150784

LOAD SISMOY ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	1.93E-14	-1.230498	0.495706	1.459080	0.743559	-1.845747
0.25000	1.93E-14	-1.230498	0.495706	1.459080	0.371780	-0.922874
C.50000	1.93E-14	-1.230498	0.495706	1.459080	-4.30E-15	3.63E-15
0.75000	1.93E-14	-1.230498	0.495706	1.459080	-0.371780	0.922874
1.00000	1.93E-14	-1.230498	0.495706	1.459080	-0.743559	1.845747

ELEM 6 ------------------ LENGTH = 3.000000

LOAD PESOP ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-0.170519	-0.900000	-7.39E-15	-1.26E-14	-0.001260	-0.261775
0.25000	-0.170519	-0.450000	-7.39E-15	-1.26E-14	-0.001260	0.244475
0.50000	-0.170519	-2.97E-15	-7.39E-15	-1.26E-14	-0.001260	0.413225

0.75000	-0.170519	0.450000	-7.39E-15	-1.26E-14	-0.001260	0.244475
1.00000	-0.170519	0.900000	-7.39E-15	-1.26E-14	-0.001260	-0.261775

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-0.284199	-1.500000	-1.34E-14	-2.11E-14	-0.002099	-0.436292
0.25000	-0.284199	-0.750000	-1.34E-14	-2.11E-14	-0.002099	0.407458
0.50000	-0.284199	-5.11E-15	-1.34E-14	-2.11E-14	-0.002099	0.688708
0.75000	-0.284199	0.750000	-1.34E-14	-2.11E-14	-0.002099	0.407458
1.00000	-0.284199	1.500000	-1.34E-14	-2.11E-14	-0.002099	-0.436292

LOAD SISMOY ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-3.82E-15	1.044256	2.348655	3.984296	3.522982	1.566385
0.25000	-3.82E-15	1.044256	2.348655	3.984296	1.761491	0.783192
0.50000	-3.82E-15	1.044256	2.348655	3.984296	-4.44E-16	-4.44E-16
0.75000	-3.82E-15	1.044256	2.348655	3.984296	-1.761491	-0.783192
1.00000	-3.82E-15	1.044256	2.348655	3.984296	-3.522982	-1.566385

F R A M E E L E M E N T I N T E R N A L F O R C E S

ELEM 7 ------------------ LENGTH = 2.000000

LOAD PESOP ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	1.94E-16	0.900000	0.001570	-0.090470	0.001437	4.82E-15
0.25000	1.94E-16	1.200000	0.001570	-0.090470	0.000652	-0.525000
0.50000	1.94E-16	1.500000	0.001570	-0.090470	-0.000133	-1.200000
0.75000	1.94E-16	1.800000	0.001570	-0.090470	-0.000918	-2.025000
1.00000	1.94E-16	2.100000	0.001570	-0.090470	-0.001703	-3.000000

LOAD VERT ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-4.07E-14	1.500000	0.002617	-0.150784	0.002395	6.40E-15
0.25000	-4.07E-14	2.000000	0.002617	-0.150784	0.001087	-0.875000
0.50000	-4.07E-14	2.500000	0.002617	-0.150784	-0.000222	-2.000000
0.75000	-4.07E-14	3.000000	0.002617	-0.150784	-0.001530	-3.375000
1.00000	-4.07E-14	3.500000	0.002617	-0.150784	-0.002838	-5.000000

LOAD SISMOY ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	0.495706	1.230498	-1.29E-14	-1.845747	-0.743559	-1.459080
0.25000	0.495706	1.230498	-1.29E-14	-1.845747	-0.743559	-2.074329
0.50000	0.495706	1.230498	-1.29E-14	-1.845747	-0.743559	-2.689578
0.75000	0.495706	1.230498	-1.29E-14	-1.845747	-0.743559	-3.304828
1.00000	0.495706	1.230498	-1.29E-14	-1.845747	-0.743559	-3.920077

ELEM 8 ------------------ LENGTH = 2.000000

LOAD PESOP ------------------

REL DIST	P	V2	V3	T	M2	M3
0.00000	-3.49E-14	0.900000	-0.001570	0.090470	-0.001437	-4.36E-15
0.25000	-3.49E-14	1.200000	-0.001570	0.090470	-0.000652	-0.525000
0.50000	-3.49E-14	1.500000	-0.001570	0.090470	0.000133	-1.200000
0.75000	-3.49E-14	1.800000	-0.001570	0.090470	0.000918	-2.025000
1.00000	-3.49E-14	2.100000	-0.001570	0.090470	0.001703	-3.000000

LOAD VERT ------------------

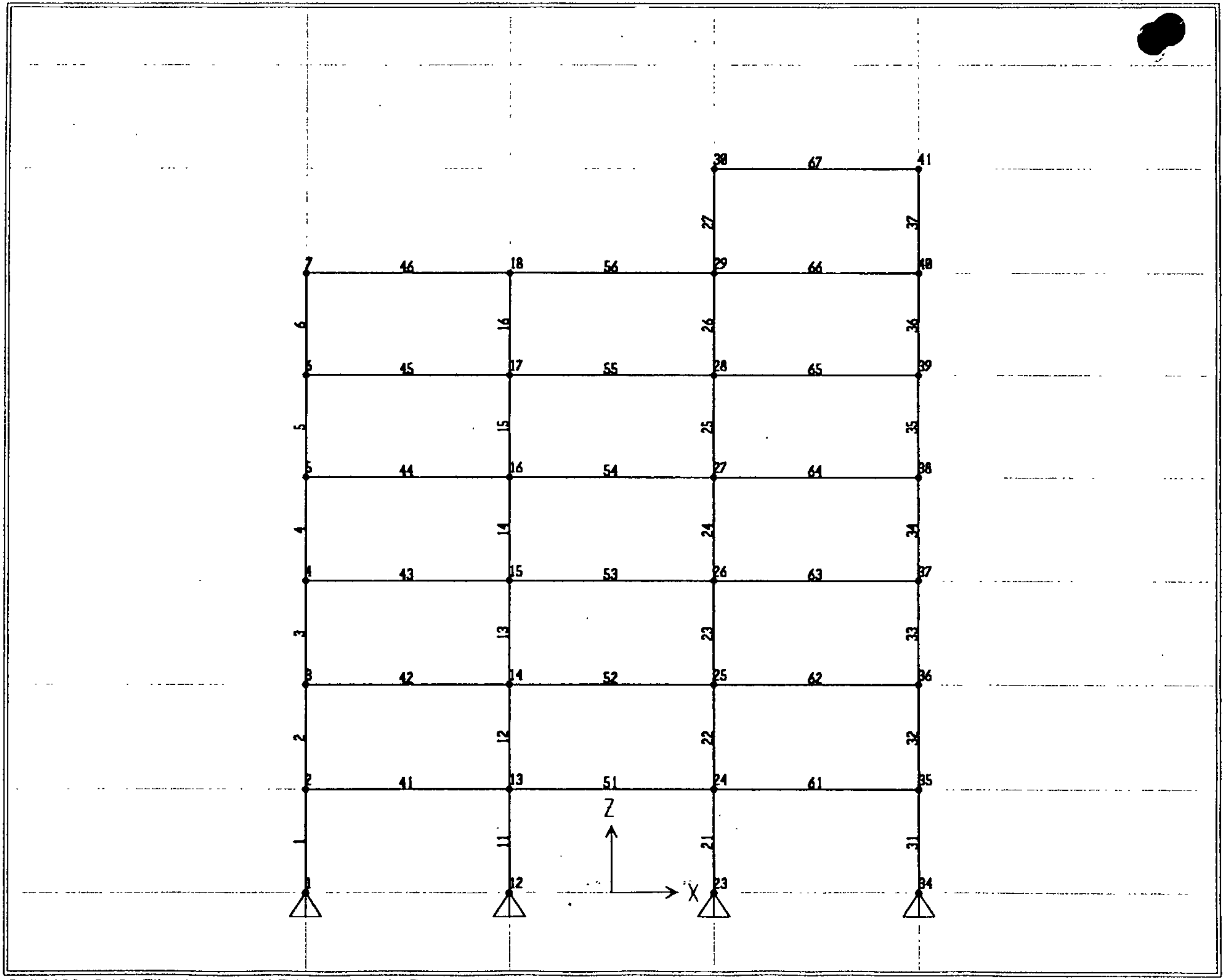
REL DIST	P	V2	V3	T	M2	M3
0.00000	-7.92E-14	1.500000	-0.002617	0.150784	-0.002395	-1.30E-14
0.25000	-7.92E-14	2.000000	-0.002617	0.150784	-0.001087	-0.875000
0.50000	-7.92E-14	2.500000	-0.002617	0.150784	0.000222	-2.000000
0.75000	-7.92E-14	3.000000	-0.002617	0.150784	0.001530	-3.375000
1.00000	-7.92E-14	3.500000	-0.002617	0.150784	0.002838	-5.000000

LOAD SISMOY ------------------

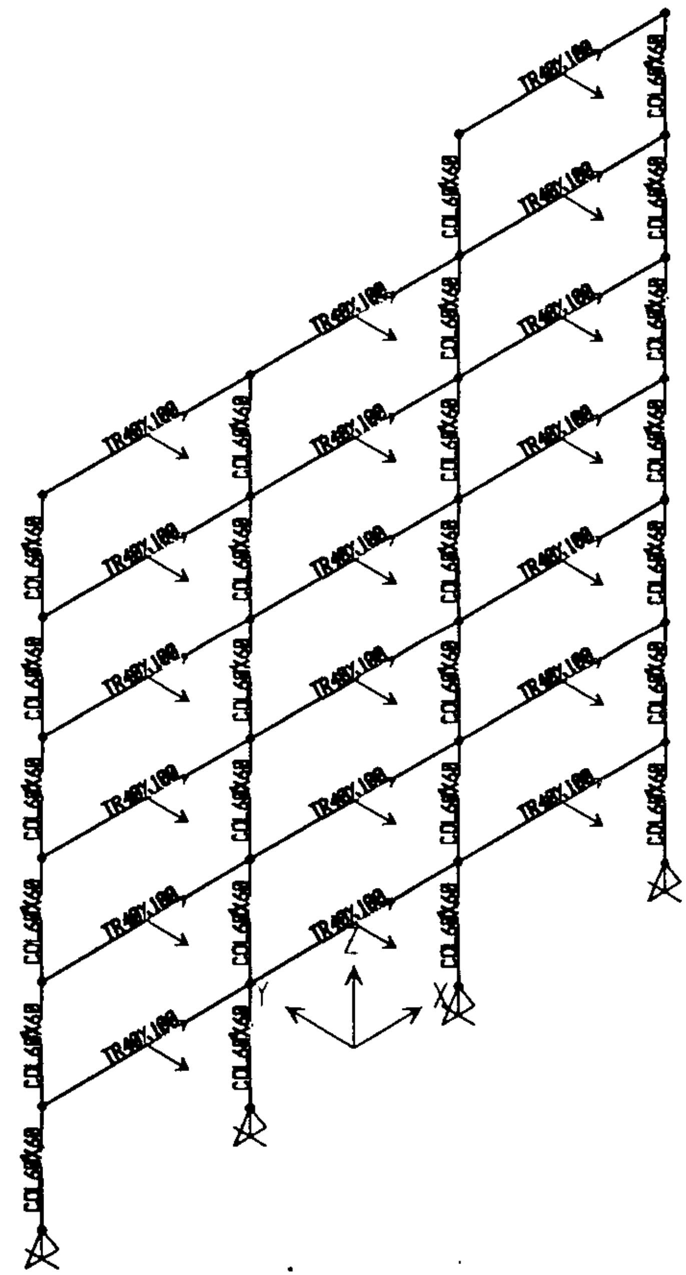
REL DIST	P	V2	V3	T	M2	M3
0.00000	-0.495706	-1.230498	-1.33E-14	-1.845747	-0.743559	1.459080

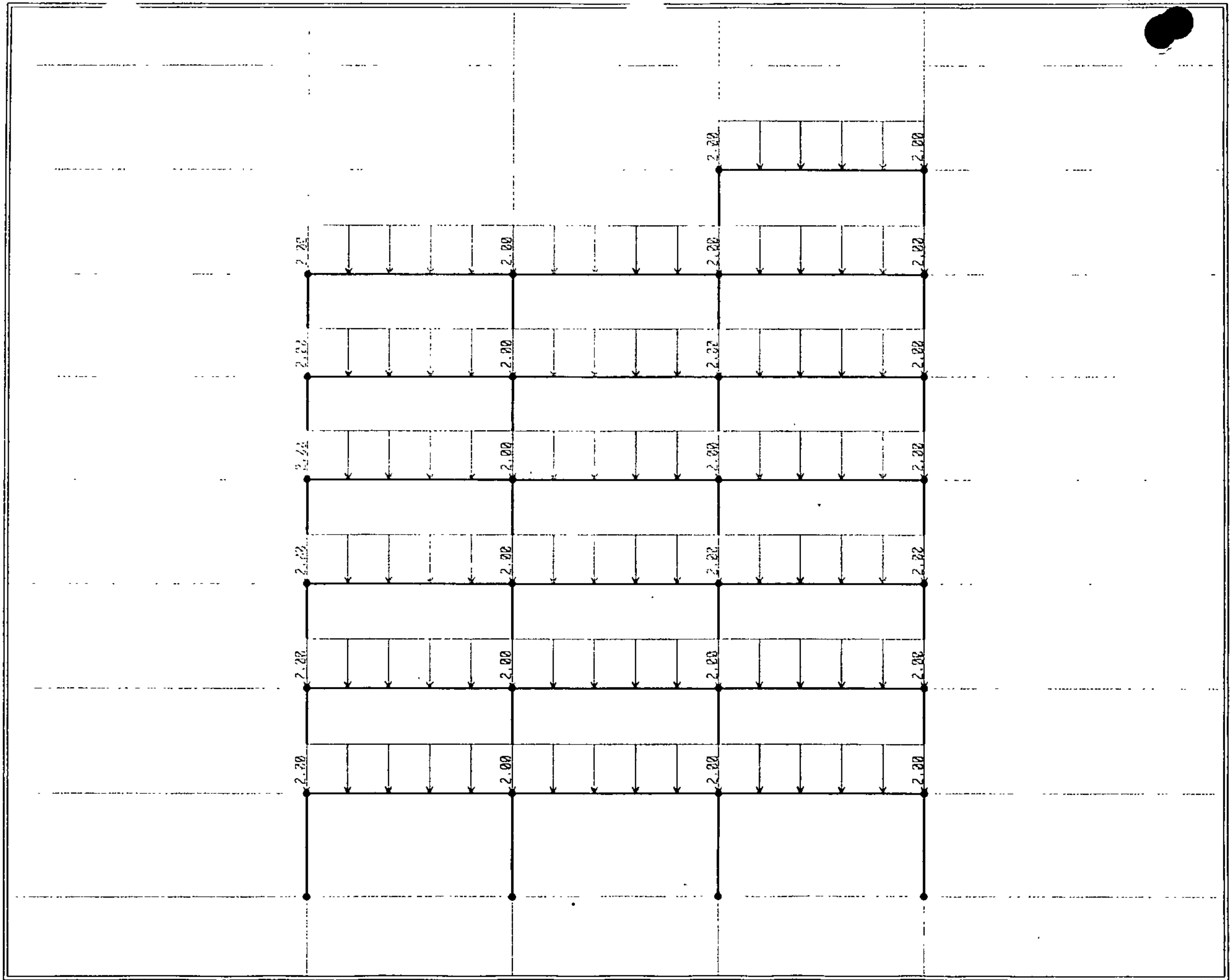
F R A M E E L E M E N T I N T E R N A L F O R C E S

REL DIST	P	V2	V3	T	M2	M3
0.25000	-0.495706	-1.230498	-1.33E-14	-1.845747	-0.743559	2.074329
0.50000	-0.495706	-1.230498	-1.33E-14	-1.845747	-0.743559	2.689578
0.75000	-0.495706	-1.230498	-1.33E-14	-1.845747	-0.743559	3.304828
1.00000	-0.495706	-1.230498	-1.33E-14	-1.845747	-0.743559	3.920077

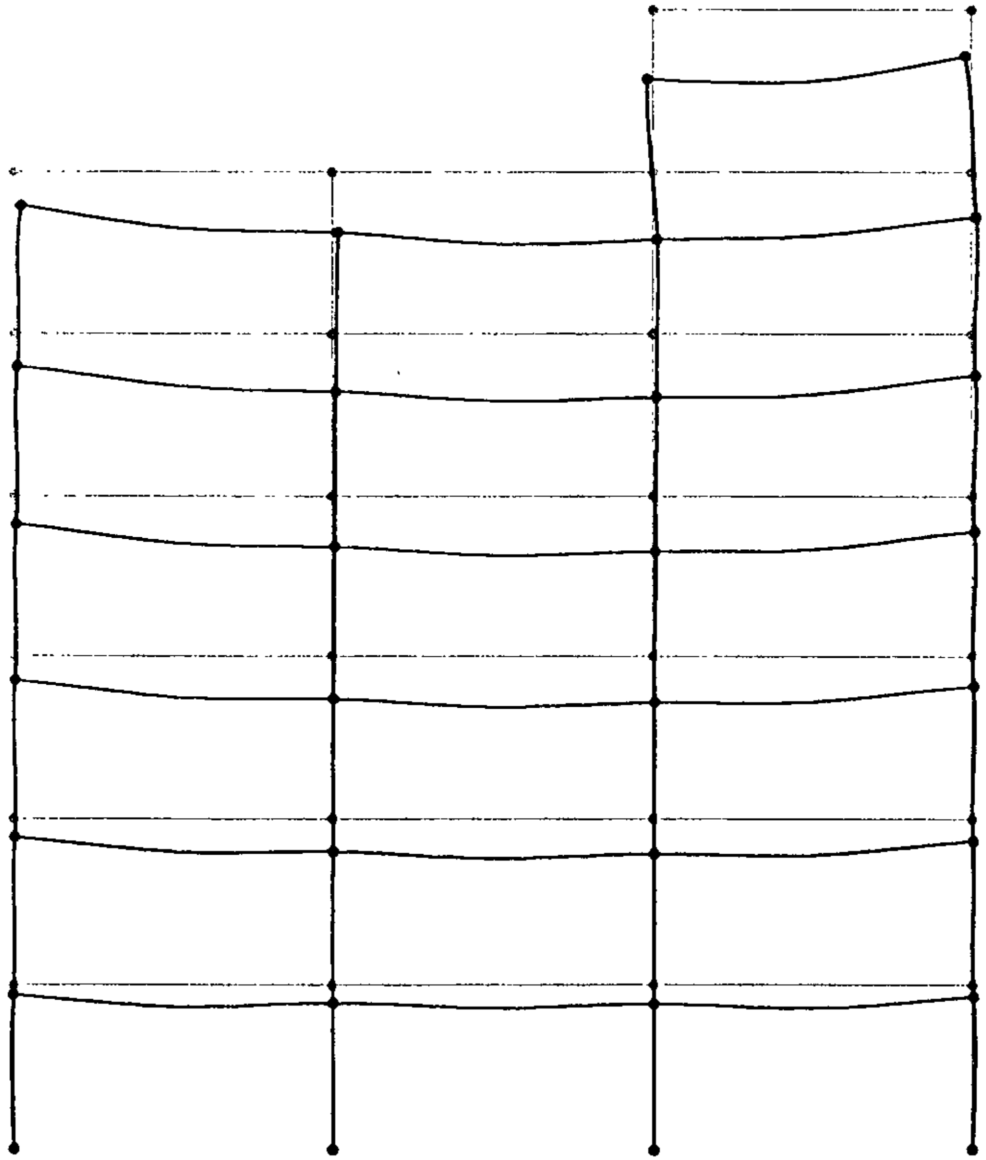


SAP2000 v6.10 - File:ejemplo9 - X-Z Plane @ Y=0 - Ton-m Units

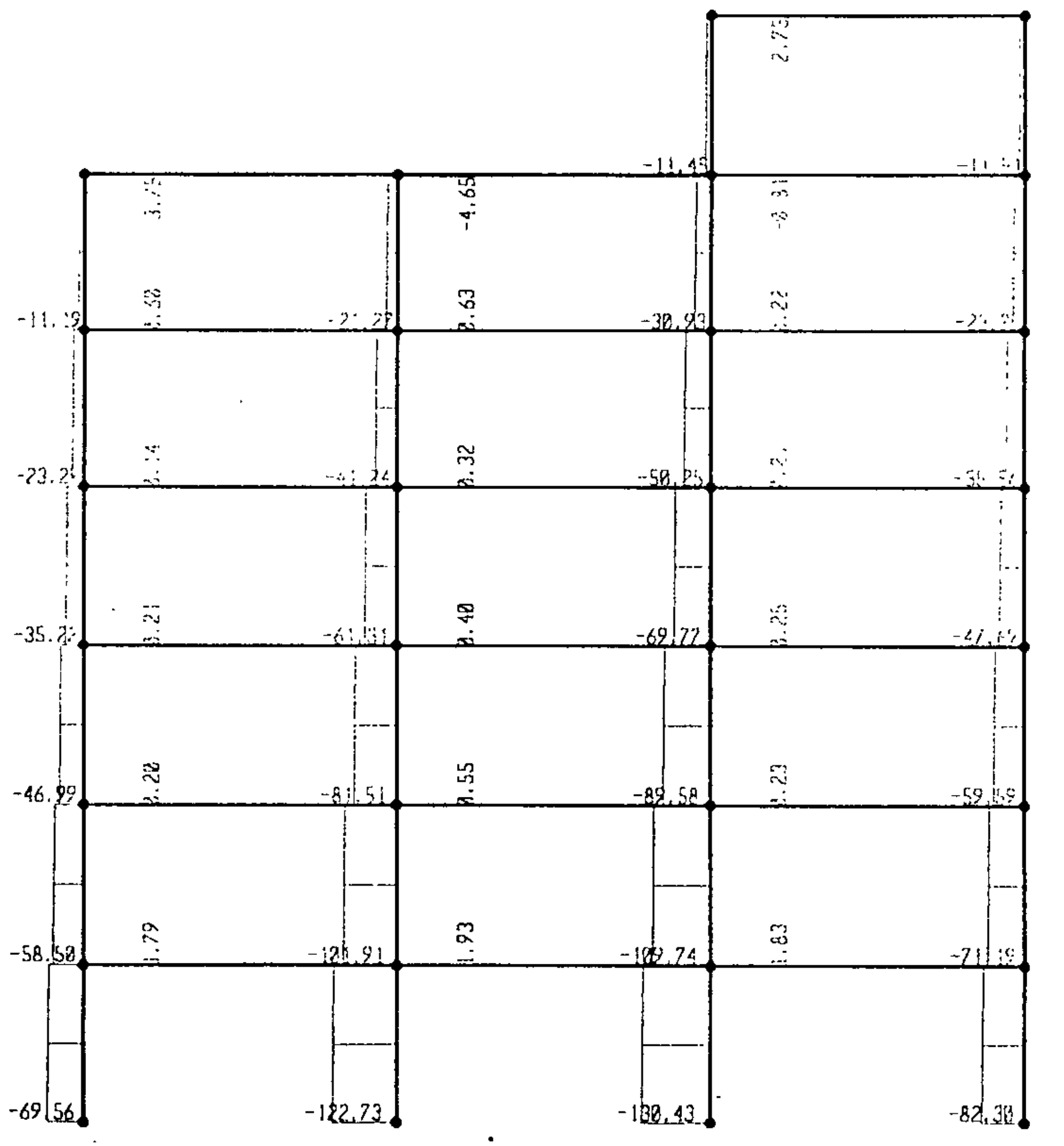

TR40X100
COL60X60
Z
Y
X



SAP2000 v6.10 - File:ejemplo9 - Frame Span Loads (VERT) - Ton-m Units

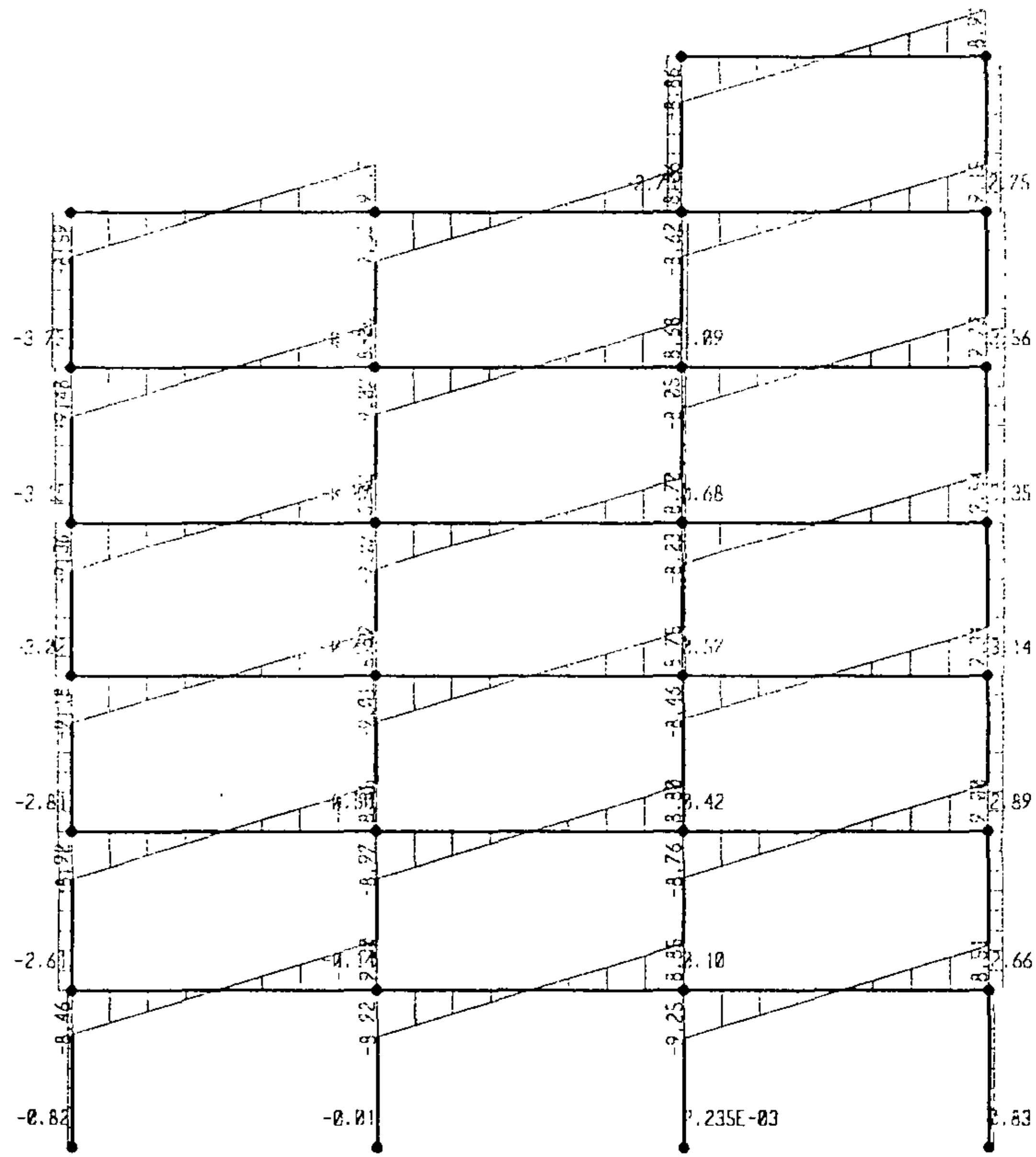


SAP2000 v6.10 - File:ejemplo9 - Deformed Shape (VERT) - Ton-m Units

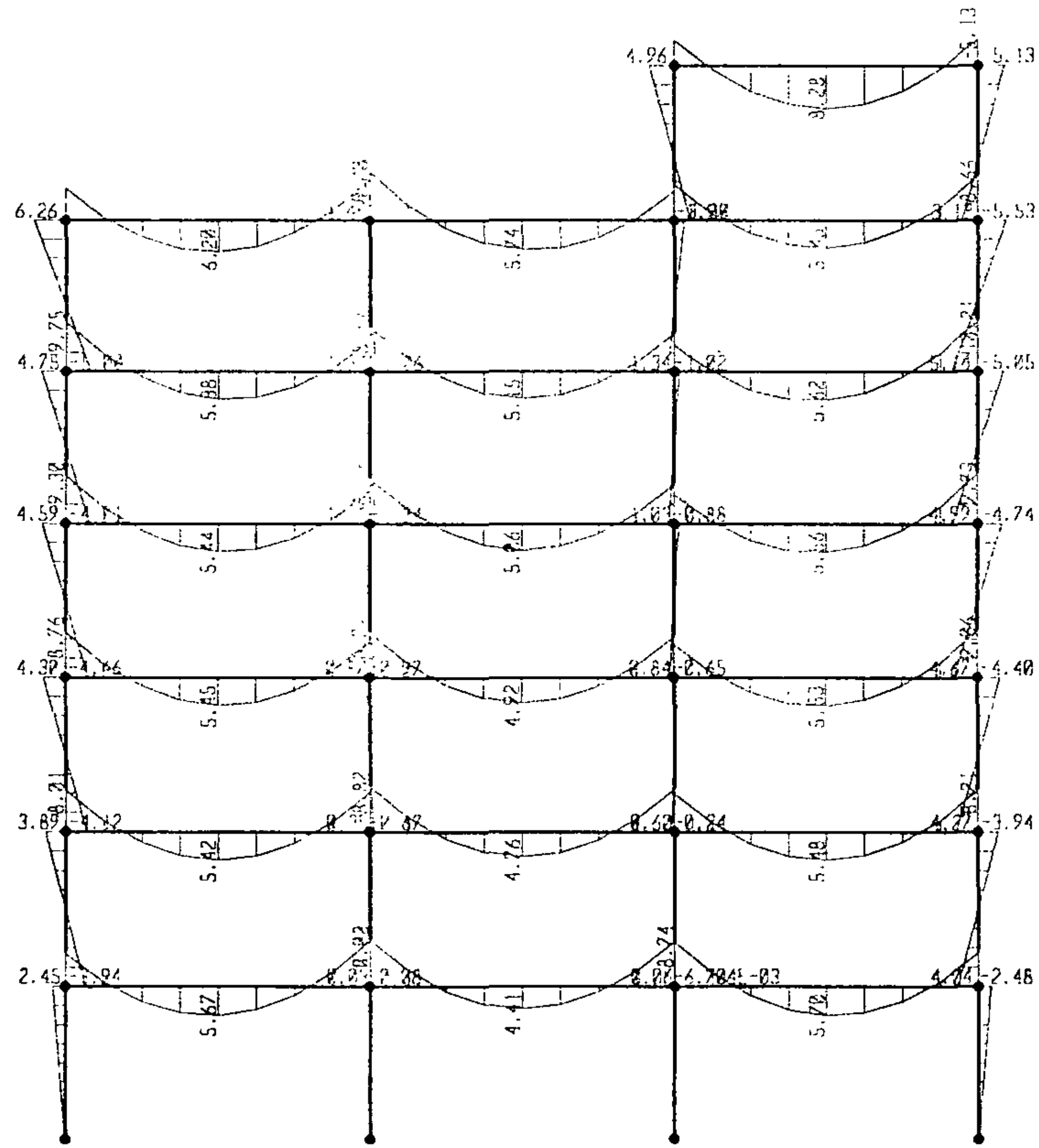


SAP2000 v6.10 - File:ejemplo9 - Axial Force Diagram (VERT) - Ton-m Units

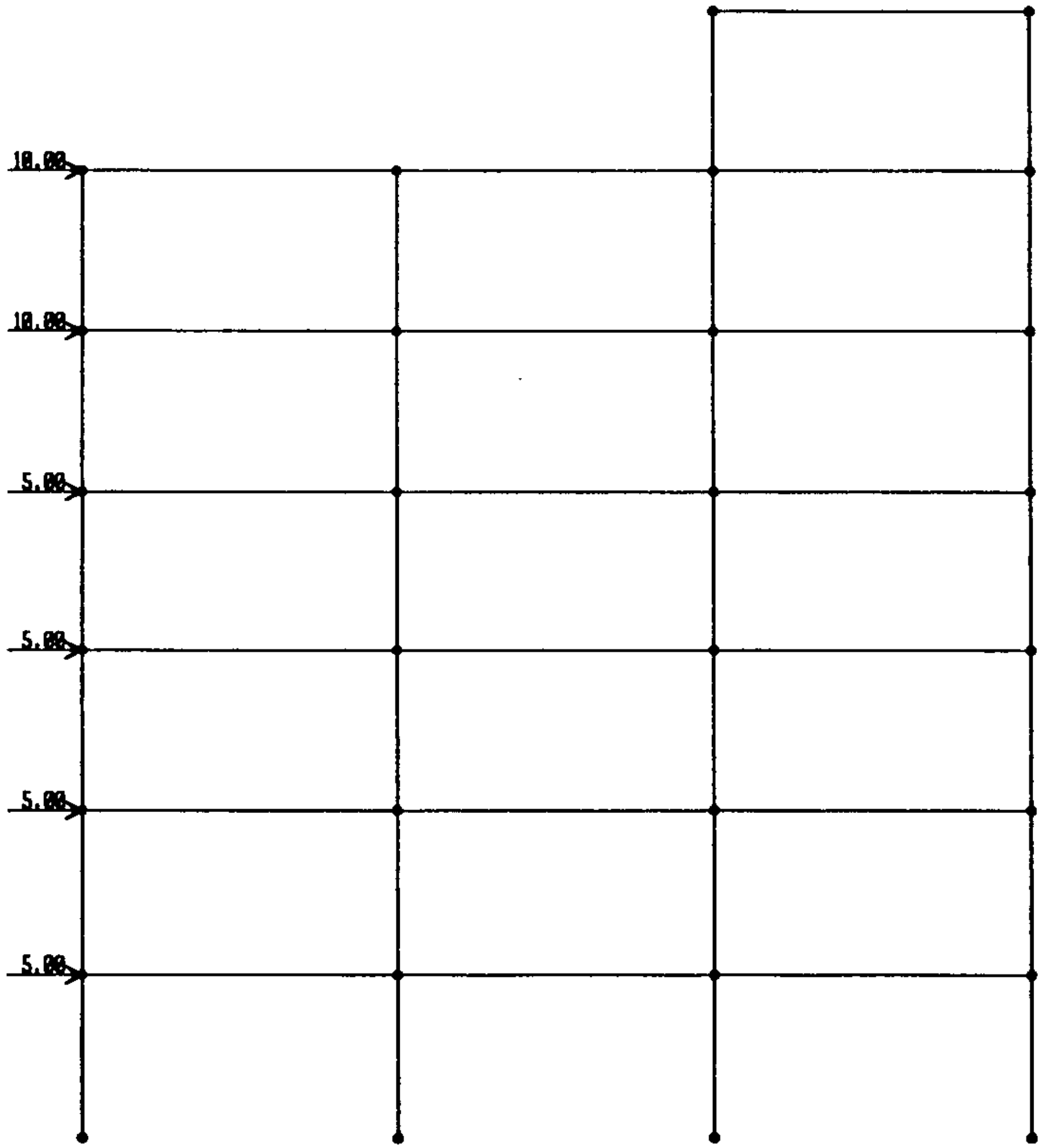
SAP2000 Marzo 21,2000 9:47



SAP2000 v6.10 - File:ejemplo9 - Shear Force 2-2 Diagram (VERT) - Ton-m Units

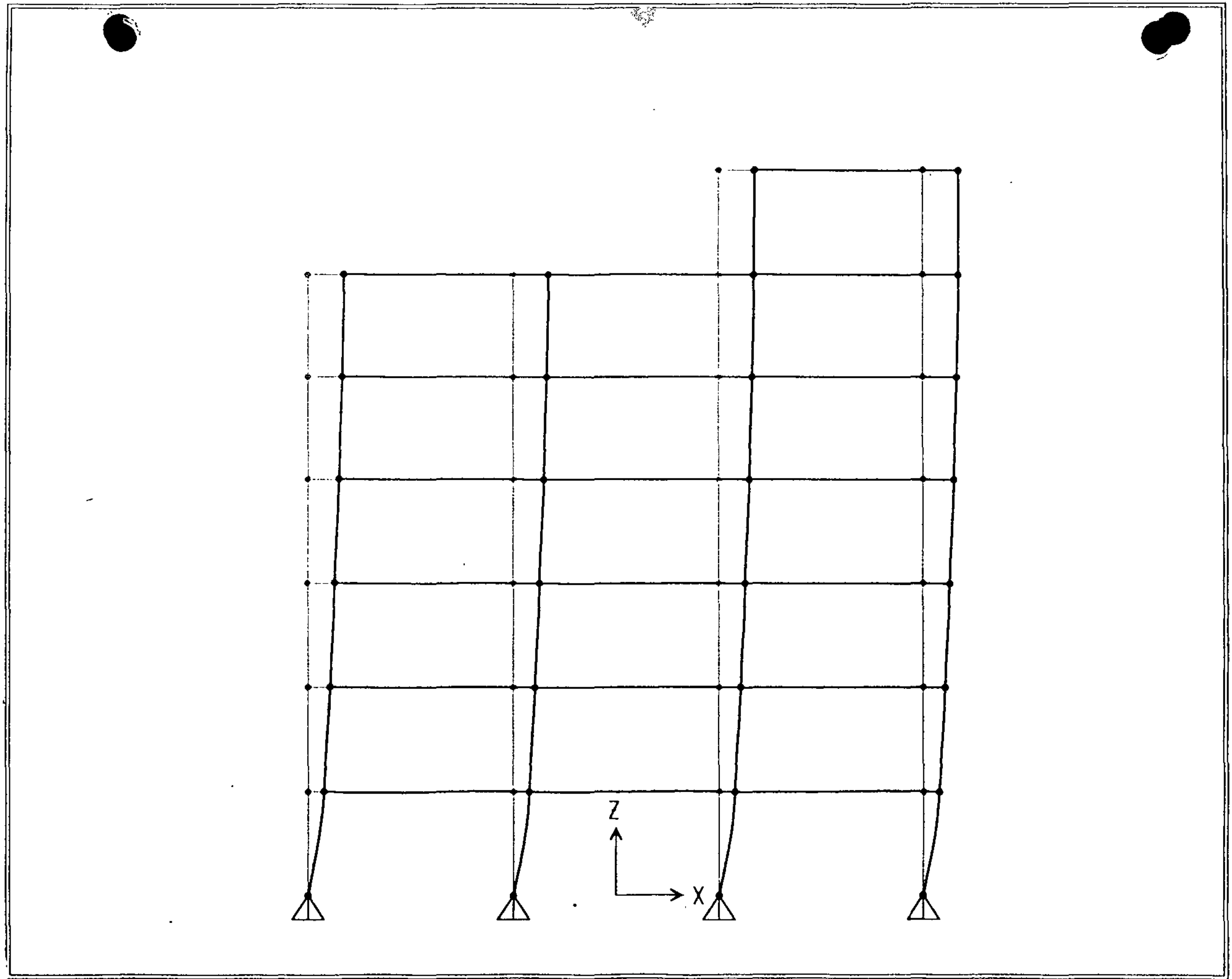


SAP2000 v6.10 - File:ejemplo9 - Moment 3-3 Diagram (VERT) - Ton-m Units

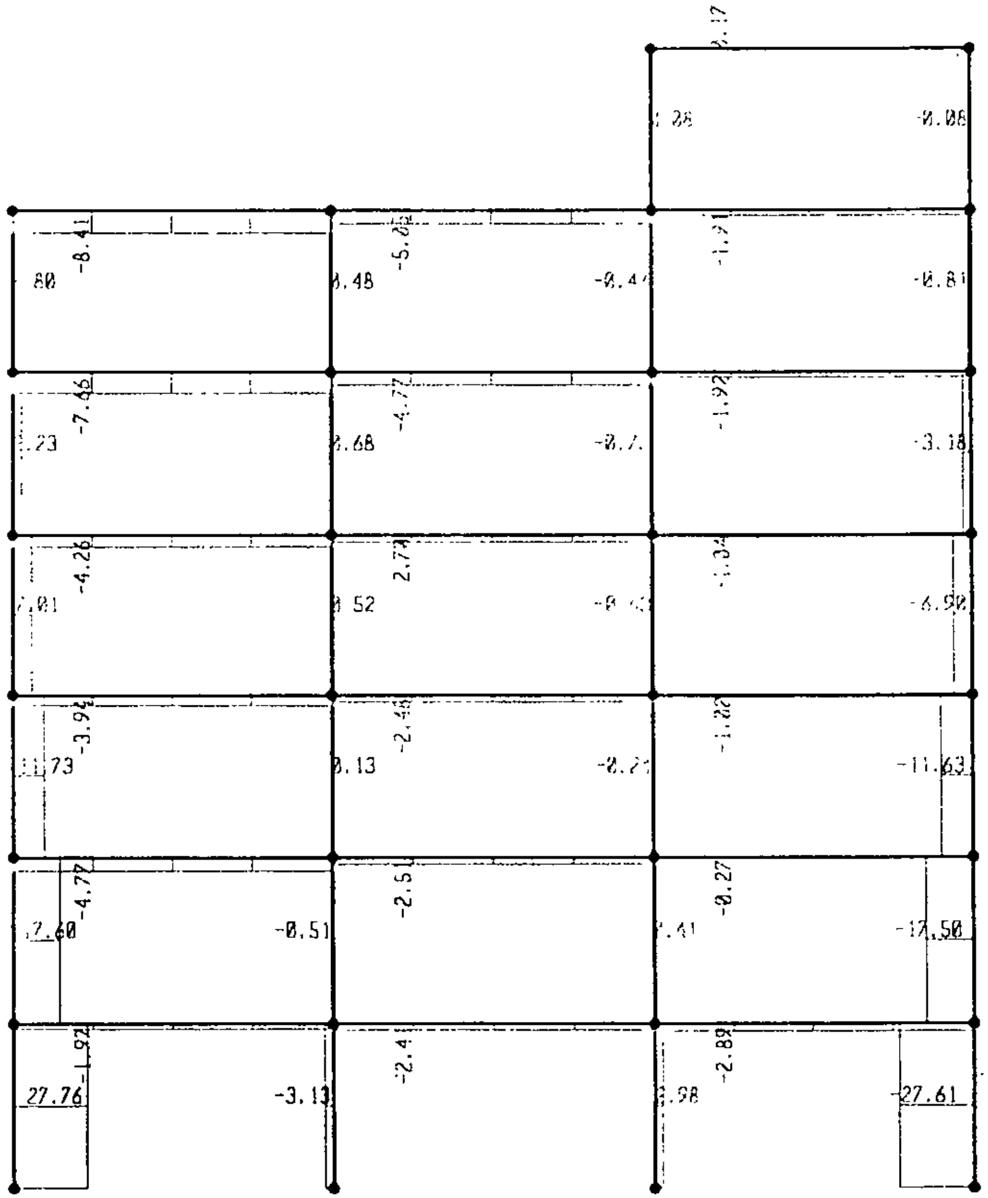


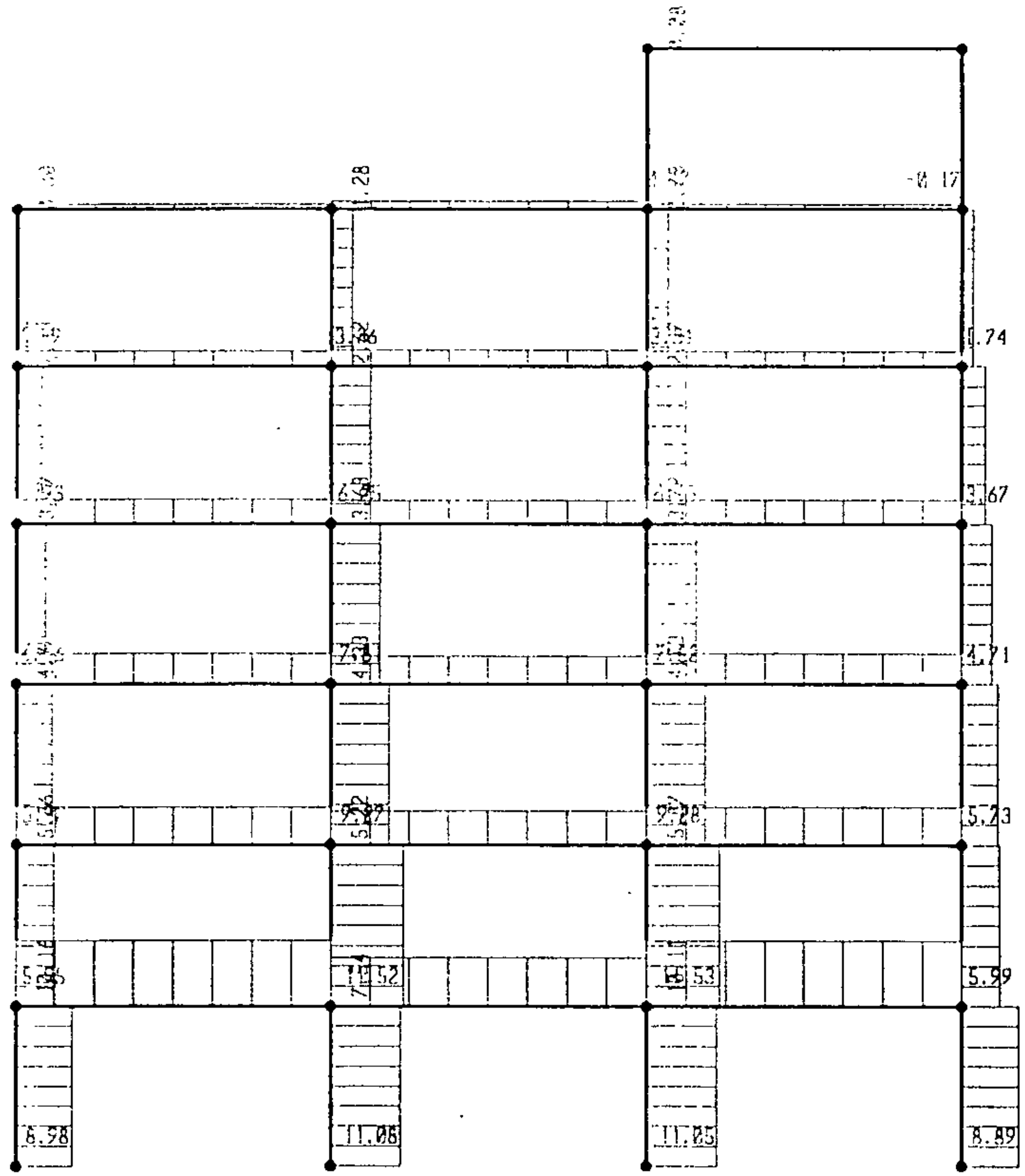
198

SAP2000 v6.. - File:ejemplo9 - Joint Loads (LATERAL) - Ton-m Units

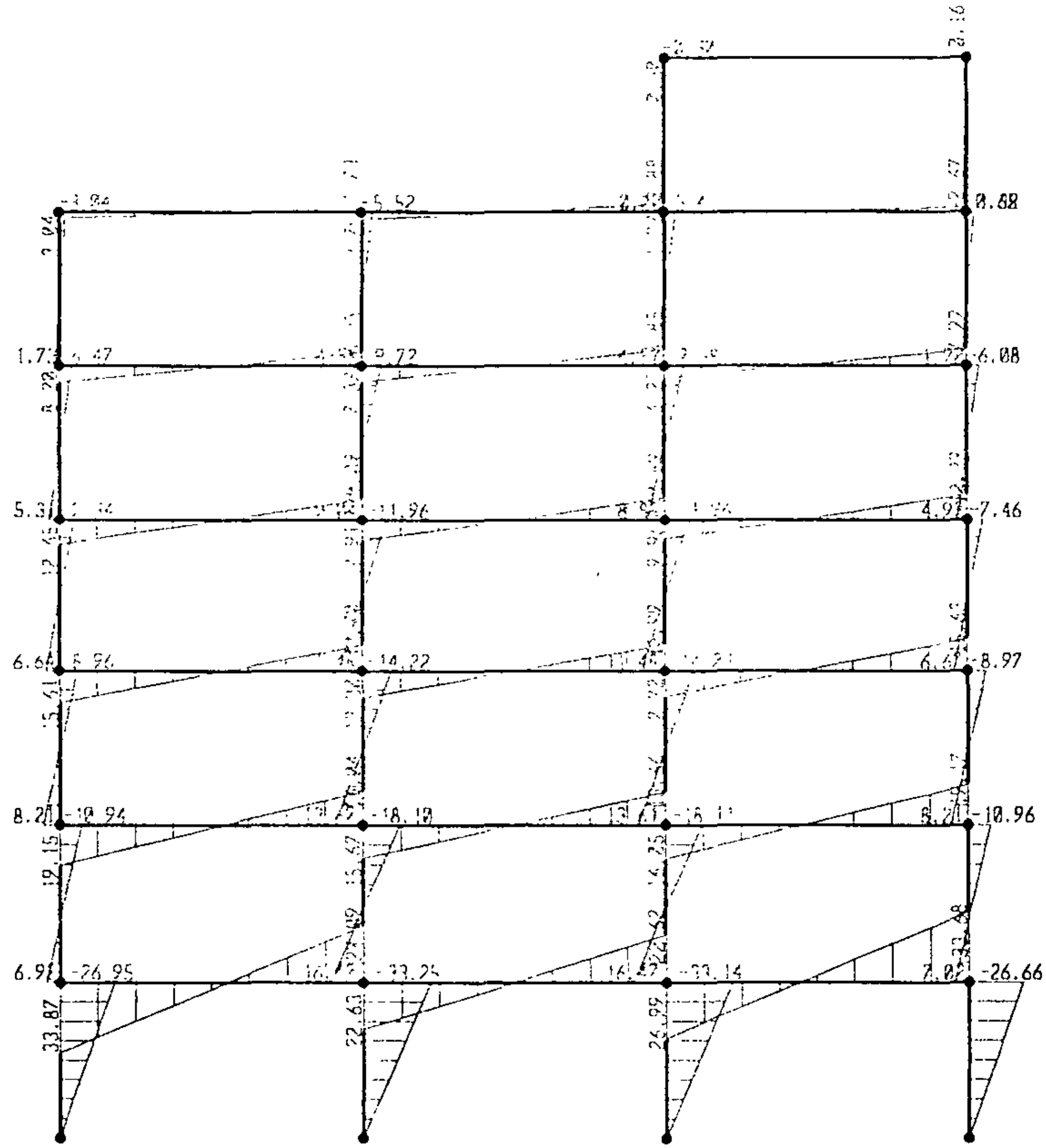


SAP2000 v6.10 - File:ejemplo9 - Deformed Shape (LATERAL) - Ton-m Units


-0.08
-8.41
-0.81
-7.46
-4.77
-1.92
-3.18
-4.26
2.73
-6.90
-3.94
-11.63
-4.77
-2.51
-0.27
-0.51
-17.50
-1.92
-2.41
-2.89
27.76
-3.13
-27.61



SAP2000 v6.10 - File:ejemplo9 - Shear Force 2-2 Diagram (LATERAL) - Ton-m Units



SAP2000 v6.10 - File:ejemplo9 - Moment 3-3 Diagram (LATERAL) - Ton-m Units

```
; File C:\Mis documentos\curso sap 2000\ejemplo9.s2k saved 3/21/00 10:12:33 in Ton-m

SYSTEM
  DOF=UX,UZ,RY  LENGTH=m  FORCE=Ton  LINES=59

JOINT
  1  X=-9  Y=0  Z=0
  2  X=-9  Y=0  Z=3
  3  X=-9  Y=0  Z=6
  4  X=-9  Y=0  Z=9
  5  X=-9  Y=0  Z=12
  6  X=-9  Y=0  Z=15
  7  X=-9  Y=0  Z=18
  12  X=-3  Y=0  Z=0
  13  X=-3  Y=0  Z=3
  14  X=-3  Y=0  Z=6
  15  X=-3  Y=0  Z=9
  16  X=-3  Y=0  Z=12
  17  X=-3  Y=0  Z=15
  18  X=-3  Y=0  Z=18
  23  X=3  Y=0  Z=0
  24  X=3  Y=0  Z=3
  25  X=3  Y=0  Z=6
  26  X=3  Y=0  Z=9
  27  X=3  Y=0  Z=12
  28  X=3  Y=0  Z=15
  29  X=3  Y=0  Z=18
  30  X=3  Y=0  Z=21
  34  X=9  Y=0  Z=0
  35  X=9  Y=0  Z=3
  36  X=9  Y=0  Z=6
  37  X=9  Y=0  Z=9
  38  X=9  Y=0  Z=12
  39  X=9  Y=0  Z=15
  40  X=9  Y=0  Z=18
  41  X=9  Y=0  Z=21

RESTRAINT
  ADD=1  DOF=U1,U2,U3,R1,R3
  ADD=12  DOF=U1,U2,U3,R1,R3
  ADD=23  DOF=U1,U2,U3,R1,R3
  ADD=34  DOF=U1,U2,U3,R1,R3

PATTERN
  NAME=DEFAULT

MATERIAL
  NAME=STEEL  IDES=S  M=.798142  W=7.833413
    T=0  E=2.038902E+07  U=.3  A=.0000117
  NAME=CONC  IDES=C  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099
  NAME=OTHER  IDES=N  M=.2448012  W=2.402616
    T=0  E=2531051  U=.2  A=.0000099

FRAME SECTION
  NAME=FSEC1 MAT=STEEL SH=R T=.5,.3 A=.15 J=2.817371E-03 I=.003125,.001125 AS=.125,.125
  NAME=FS1 MAT=CONC SH=R T=.6,.6 A=.36 J=1.825201E-02 I=.0108,.0108 AS=.3,.3
  NAME=FS2 MAT=CONC SH=R T=1,.4 A=.4 J=.0159688 I=3.333334E-02,5.333334E-03 AS=.3333333,.3333333

FRAME
  1  J=1,2  SEC=FS1  NSEG=2  ANG=0
  2  J=2,3  SEC=FS1  NSEG=2  ANG=0
  3  J=3,4  SEC=FS1  NSEG=2  ANG=0
  4  J=4,5  SEC=FS1  NSEG=2  ANG=0
  5  J=5,6  SEC=FS1  NSEG=2  ANG=0
  6  J=6,7  SEC=FS1  NSEG=2  ANG=0
  11  J=12,13  SEC=FS1  NSEG=2  ANG=0
  12  J=13,14  SEC=FS1  NSEG=2  ANG=0
  13  J=14,15  SEC=FS1  NSEG=2  ANG=0
  14  J=15,16  SEC=FS1  NSEG=2  ANG=0
  15  J=16,17  SEC=FS1  NSEG=2  ANG=0
  16  J=17,18  SEC=FS1  NSEG=2  ANG=0
  21  J=23,24  SEC=FS1  NSEG=2  ANG=0
  22  J=24,25  SEC=FS1  NSEG=2  ANG=0
  23  J=25,26  SEC=FS1  NSEG=2  ANG=0
  24  J=26,27  SEC=FS1  NSEG=2  ANG=0
  25  J=27,28  SEC=FS1  NSEG=2  ANG=0
  26  J=28,29  SEC=FS1  NSEG=2  ANG=0
  27  J=29,30  SEC=FS1  NSEG=2  ANG=0
  31  J=34,35  SEC=FS1  NSEG=2  ANG=0
  32  J=35,36  SEC=FS1  NSEG=2  ANG=0
  33  J=36,37  SEC=FS1  NSEG=2  ANG=0
  34  J=37,38  SEC=FS1  NSEG=2  ANG=0
  35  J=38,39  SEC=FS1  NSEG=2  ANG=0
  36  J=39,40  SEC=FS1  NSEG=2  ANG=0
  37  J=40,41  SEC=FS1  NSEG=2  ANG=0
  41  J=2,13  SEC=FS2  NSEG=4  ANG=0
  42  J=3,14  SEC=FS2  NSEG=4  ANG=0
  43  J=4,15  SEC=FS2  NSEG=4  ANG=0
  44  J=5,16  SEC=FS2  NSEG=4  ANG=0
  45  J=6,17  SEC=FS2  NSEG=4  ANG=0
  46  J=7,18  SEC=FS2  NSEG=4  ANG=0
  51  J=13,24  SEC=FS2  NSEG=4  ANG=0
```

```
  52  J=14,25  SEC=FS2  NSEG=4  ANG=0
  53  J=15,26  SEC=FS2  NSEG=4  ANG=0
  54  J=16,27  SEC=FS2  NSEG=4  ANG=0
  55  J=17,28  SEC=FS2  NSEG=4  ANG=0
  56  J=18,29  SEC=FS2  NSEG=4  ANG=0
  61  J=24,35  SEC=FS2  NSEG=4  ANG=0
  62  J=25,36  SEC=FS2  NSEG=4  ANG=0
  63  J=26,37  SEC=FS2  NSEG=4  ANG=0
  64  J=27,38  SEC=FS2  NSEG=4  ANG=0
  65  J=28,39  SEC=FS2  NSEG=4  ANG=0
  66  J=29,40  SEC=FS2  NSEG=4  ANG=0
  67  J=30,41  SEC=FS2  NSEG=4  ANG=0

LOAD
  NAME=VERT  SW=1
    TYPE=DISTRIBUTED SPAN
      ADD=41  RD=0,1  UZ=-2,-2
      ADD=42  RD=0,1  UZ=-2,-2
      ADD=43  RD=0,1  UZ=-2,-2
      ADD=44  RD=0,1  UZ=-2,-2
      ADD=45  RD=0,1  UZ=-2,-2
      ADD=46  RD=0,1  UZ=-2,-2
      ADD=51  RD=0,1  UZ=-2,-2
      ADD=52  RD=0,1  UZ=-2,-2
      ADD=53  RD=0,1  UZ=-2,-2
      ADD=54  RD=0,1  UZ=-2,-2
      ADD=55  RD=0,1  UZ=-2,-2
      ADD=56  RD=0,1  UZ=-2,-2
      ADD=61  RD=0,1  UZ=-2,-2
      ADD=62  RD=0,1  UZ=-2,-2
      ADD=63  RD=0,1  UZ=-2,-2
      ADD=64  RD=0,1  UZ=-2,-2
      ADD=65  RD=0,1  UZ=-2,-2
      ADD=66  RD=0,1  UZ=-2,-2
      ADD=67  RD=0,1  UZ=-2,-2
  NAME=LATERAL
    TYPE=FORCE
      ADD=2  UX=5
      ADD=3  UX=5
      ADD=4  UX=5
      ADD=5  UX=5
      ADD=6  UX=10
      ADD=7  UX=10

COMBO
  NAME=DCON1
    LOAD=VERT  SF=1.4
  NAME=DCON2
    LOAD=VERT  SF=1.4
  NAME=DCON3
    LOAD=VERT  SF=1.05
    LOAD=LATERAL  SF=1.4025
  NAME=DCON4
    LOAD=VERT  SF=1.05
    LOAD=LATERAL  SF=-1.4025
  NAME=DCON5
    LOAD=VERT  SF=.9
    LOAD=LATERAL  SF=1.43
  NAME=DCON6
    LOAD=VERT  SF=.9
    LOAD=LATERAL  SF=-1.43

OUTPUT
  ELEM=JOINT  TYPE=DISP  LOAD=VERT
  ELEM=JOINT  TYPE=DISP  LOAD=LATERAL
  ELEM=JOINT  TYPE=APPL  LOAD=VERT
  ELEM=JOINT  TYPE=APPL  LOAD=LATERAL
  ELEM=FRAME  TYPE=FORCE  LOAD=VERT
  ELEM=FRAME  TYPE=FORCE  LOAD=LATERAL
  ELEM=FRAME  TYPE=JOINTF  LOAD=VERT
  ELEM=FRAME  TYPE=JOINTF  LOAD=LATERAL

END

; The following data is not required for analysis. It is written here as a backup.
; This data will be used for graphics and design if this file is imported.
; If changes are made to the analysis data above, then the following data
; should be checked for consistency.
; Any errors in importing the following data are ignored without warning.
SAP2000 V6.10 SUPPLEMENTAL DATA
  GRID GLOBAL  X "1"  -9
  GRID GLOBAL  X "2"  -3
  GRID GLOBAL  X "3"  3
  GRID GLOBAL  X "4"  9
  GRID GLOBAL  Y "5"  0
  GRID GLOBAL  Z "6"  0
  GRID GLOBAL  Z "7"  3
  GRID GLOBAL  Z "8"  6
  GRID GLOBAL  Z "9"  9
  GRID GLOBAL  Z "10"  12
  GRID GLOBAL  Z "11"  15
  GRID GLOBAL  Z "12"  18
  GRID GLOBAL  Z "13"  21
  GRID GLOBAL  Z "14"  24
  GRID GLOBAL  Z "15"  27
```

```
GRID GLOBAL  Z "16"  30
CONCRETEGROUP "COLS"
CONCRETEGROUP "TRABES"
GROUP "COLS"  JOINT 1
GROUP "COLS"  JOINT 2
GROUP "COLS"  JOINT 3
GROUP "COLS"  JOINT 4
GROUP "COLS"  JOINT 5
GROUP "COLS"  JOINT 6
GROUP "COLS"  JOINT 7
GROUP "COLS"  JOINT 12
GROUP "COLS"  JOINT 13
GROUP "COLS"  JOINT 14
GROUP "COLS"  JOINT 15
GROUP "COLS"  JOINT 16
GROUP "COLS"  JOINT 17
GROUP "COLS"  JOINT 18
GROUP "COLS"  JOINT 23
GROUP "COLS"  JOINT 24
GROUP "COLS"  JOINT 25
GROUP "COLS"  JOINT 26
GROUP "COLS"  JOINT 27
GROUP "COLS"  JOINT 28
GROUP "COLS"  JOINT 29
GROUP "COLS"  JOINT 30
GROUP "COLS"  JOINT 34
GROUP "COLS"  JOINT 35
GROUP "COLS"  JOINT 36
GROUP "COLS"  JOINT 37
GROUP "COLS"  JOINT 38
GROUP "COLS"  JOINT 39
GROUP "COLS"  JOINT 40
GROUP "COLS"  JOINT 41
GROUP "COLS"  FRAME 1
GROUP "COLS"  FRAME 2
GROUP "COLS"  FRAME 3
GROUP "COLS"  FRAME 4
GROUP "COLS"  FRAME 5
GROUP "COLS"  FRAME 6
GROUP "COLS"  FRAME 11
GROUP "COLS"  FRAME 12
GROUP "COLS"  FRAME 13
GROUP "COLS"  FRAME 14
GROUP "COLS"  FRAME 15
GROUP "COLS"  FRAME 16
GROUP "COLS"  FRAME 21
GROUP "COLS"  FRAME 22
GROUP "COLS"  FRAME 23
GROUP "COLS"  FRAME 24
GROUP "COLS"  FRAME 25
GROUP "COLS"  FRAME 26
GROUP "COLS"  FRAME 27
GROUP "COLS"  FRAME 31
GROUP "COLS"  FRAME 32
GROUP "COLS"  FRAME 33
GROUP "COLS"  FRAME 34
GROUP "COLS"  FRAME 35
GROUP "COLS"  FRAME 36
GROUP "COLS"  FRAME 37
GROUP "TRABES"  JOINT 2
GROUP "TRABES"  JOINT 3
GROUP "TRABES"  JOINT 4
GROUP "TRABES"  JOINT 5
GROUP "TRABES"  JOINT 6
GROUP "TRABES"  JOINT 7
GROUP "TRABES"  JOINT 13
GROUP "TRABES"  JOINT 14
GROUP "TRABES"  JOINT 15
GROUP "TRABES"  JOINT 16
GROUP "TRABES"  JOINT 17
GROUP "TRABES"  JOINT 18
GROUP "TRABES"  JOINT 24
GROUP "TRABES"  JOINT 25
GROUP "TRABES"  JOINT 26
GROUP "TRABES"  JOINT 27
GROUP "TRABES"  JOINT 28
GROUP "TRABES"  JOINT 29
GROUP "TRABES"  JOINT 30
GROUP "TRABES"  JOINT 35
GROUP "TRABES"  JOINT 36
GROUP "TRABES"  JOINT 37
GROUP "TRABES"  JOINT 38
GROUP "TRABES"  JOINT 39
GROUP "TRABES"  JOINT 40
GROUP "TRABES"  JOINT 41
GROUP "TRABES"  FRAME 41
GROUP "TRABES"  FRAME 42
GROUP "TRABES"  FRAME 43
GROUP "TRABES"  FRAME 44
GROUP "TRABES"  FRAME 45
GROUP "TRABES"  FRAME 46
GROUP "TRABES"  FRAME 51
GROUP "TRABES"  FRAME 52
GROUP "TRABES"  FRAME 53
GROUP "TRABES"  FRAME 54
```

```
  GROUP "TRABES"  FRAME 55
  GROUP "TRABES"  FRAME 56
  GROUP "TRABES"  FRAME 61
  GROUP "TRABES"  FRAME 62
  GROUP "TRABES"  FRAME 63
  GROUP "TRABES"  FRAME 64
  GROUP "TRABES"  FRAME 65
  GROUP "TRABES"  FRAME 66
  GROUP "TRABES"  FRAME 67
  MATERIAL STEEL FY 25310.5
  MATERIAL CONC FYREBAR 42184.18 FYSHEAR 28122.78 FC 2812.278 FCSHEAR 2812.278
  FRAMESECTION FS1 NAME COL60X60
  FRAMESECTION FS2 NAME TR40X100
  CONCRETESECTION COL60X60 COLUMN  COVER .06 REBAR RR-3-3
  CONCRETESECTION TR40X100 COLUMN  COVER .04572 REBAR RR-3-3
  STATICLOAD VERT TYPE DEAD
  STATICLOAD LATERAL TYPE QUAKE
  COMBO DCON1 DESIGN CONCRETE
  COMBO DCON2 DESIGN CONCRETE
  COMBO DCON3 DESIGN CONCRETE
  COMBO DCON4 DESIGN CONCRETE
  COMBO DCON5 DESIGN CONCRETE
  COMBO DCON6 DESIGN CONCRETE
  CONCRETEDESIGN "ACI 318-95"
END SUPPLEMENTAL DATA
```

S A P 2 0 0 0
Structural Analysis Programs
Version 6.10

21 Mar 2000 09:11:27

S Y S T E M D A T A

```
STEADY STATE LOAD FREQUENCY - - - - - - - - 0.0000E+00

LENGTH UNITS - - - - - - - - - - - - - - - -          M
FORCE  UNITS - - - - - - - - - - - - - - - -        TON

UP DIRECTION - - - - - - - - - - - - - - - -         +Z

GLOBAL DEGREES OF FREEDOM  - - - - - - - - -         UX
                           - - - - - - - - -         UZ
                           - - - - - - - - -         RY

PAGINATION BY  - - - - - - - - - - - - - - -      LINES
NUMBER OF LINES PER PAGE - - - - - - - - - -         59
INCLUDE WARNING MESSAGES IN OUTPUT FILE  - -          Y
```


G E N E R A T E D J O I N T C O O R D I N A T E S

JOINT	X	Y	Z
1	-9.000	0.000	0.000
2	-9.000	0.000	3.000
3	-9.000	0.000	6.000
4	-9.000	0.000	9.000
5	-9.000	0.000	12.000
6	-9.000	0.000	15.000
7	-9.000	0.000	18.000
12	-3.000	0.000	0.000
13	-3.000	0.000	3.000
14	-3.000	0.000	6.000
15	-3.000	0.000	9.000
16	-3.000	0.000	12.000
17	-3.000	0.000	15.000
18	-3.000	0.000	18.000
23	3.000	0.000	0.000
24	3.000	0.000	3.000
25	3.000	0.000	6.000
26	3.000	0.000	9.000
27	3.000	0.000	12.000
28	3.000	0.000	15.000
29	3.000	0.000	18.000
30	3.000	0.000	21.000
34	9.000	0.000	0.000
35	9.000	0.000	3.000
36	9.000	0.000	6.000
37	9.000	0.000	9.000
38	9.000	0.000	12.000
39	9.000	0.000	15.000
40	9.000	0.000	18.000
41	9.000	0.000	21.000

P A T T E R N S

PATTERN	JOINT	VALUE
DEFAULT		

R E S T R A I N T D A T A

JOINT						
1	U1	U2	U3	R1		R3
12	U1	U2	U3	R1		R3
23	U1	U2	U3	R1		R3
34	U1	U2	U3	R1		R3

PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

M A T E R I A L   P R O P E R T Y   D A T A

MAT LABEL	NUMBER TEMPS	WEIGHT PER UNIT VOL	MASS PER UNIT VOL	DESIGN CODE
STEEL	1	0.7833E+01	0.7981E+00	S
CONC	1	0.2403E+01	0.2448E+00	C
OTHER	1	0.2403E+01	0.2448E+00	N

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 6
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

T E M P E R A T U R E   D E P E N D E N T   D A T A

MATERIAL PROPERTIES

MAT LABEL	TEMP	MODULUS OF ELASTICITY E1	E2	E3	SHEAR MODULII G12	G13	G23
STEEL	0.00	0.204E+08	0.204E+08	0.204E+08	0.784E+07	0.784E+07	0.784E+07
CONC	0.00	0.253E+07	0.253E+07	0.253E+07	0.105E+07	0.105E+07	0.105E+07
OTHER	0.00	0.253E+07	0.253E+07	0.253E+07	0.105E+07	0.105E+07	0.105E+07

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 7
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

T E M P E R A T U R E   D E P E N D E N T   D A T A

THERMAL EXPANSION COEFFICIENTS

MAT LABEL	TEMP	COEFFICIENTS OF THERMAL EXPANSION A1	A2	A3	A12	A13	A23
STEEL	0.00	0.117E-04	0.117E-04	0.117E-04	0.000E+00	0.000E+00	0.000E+00
CONC	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00
OTHER	0.00	0.990E-05	0.990E-05	0.990E-05	0.000E+00	0.000E+00	0.000E+00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 8
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

T E M P E R A T U R E   D E P E N D E N T   D A T A

MATERIAL PROPERTIES

POISSONS RATIO

MAT LABEL	TEMP	U12	U13	U23	U14	U24	U34	U15	U25	U35	U45	U16	U26	U36	U46	U56
STEEL	0.00	0.3	0.3	0.3	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
CONC	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0
OTHER	0.00	0.2	0.2	0.2	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 9
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

MATERIAL PROPERTIES

MAT LABEL	TEMP	YIELD FY
CONC	0.00	36.00

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 10
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION LABEL	SHAPE TYPE	DEPTH	FLANGE WIDTH TOP	FLANGE THICK TOP	WEB THICK	FLANGE WIDTH BOTTOM	FLANGE THICK BOTTOM
FSEC1	R	0.500	0.300				
FS1	R	0.600	0.600				
FS2	R	1.000	0.400				

C S I / S A P 2 0 0 0 - FINITE ELEMENT ANALYSIS OF STRUCTURES PAGE 11
PROGRAM:SAP2000/FILE:\Misdocumentos\cursosap2000\ejemplo9.EKO

F R A M E   S E C T I O N   P R O P E R T Y   D A T A - P R I S M A T I C

SECTION   AXIAL   TORSIONAL   MOMENTS OF INERTIA   SHEAR AREAS

LABEL	AREA	CONSTANT	I33	I22	A2	A3
FSEC1	0.150E+00	0.282E-02	0.313E-02	0.113E-02	0.125E+00	0.125E+00
FS1	0.360E+00	0.183E-01	0.108E-01	0.108E-01	0.300E+00	0.300E+00
FS2	0.400E+00	0.160E-01	0.333E-01	0.533E-02	0.333E+00	0.333E+00

F R A M E S E C T I O N P R O P E R T Y D A T A - P R I S M A T I C

SECTION LABEL	MAT LABEL	ADDITIONAL MASS PER LENGTH	ADDITIONAL WEIGHT PER LENGTH
FSEC1	STEEL	0.000E+00	0.000E+00
FS1	CONC	0.000E+00	0.000E+00
FS2	CONC	0.000E+00	0.000E+00

F R A M E E L E M E N T D A T A

ELEMENT LABEL	JOINT END-I	JOINT END-J	ELEMENT LENGTH	END-OFFSET-LENGTHS END-I	END-J	RIGID-END FACTOR	NUMBER OF SEGMENTS
1	1	2	3.000	0.000	0.000	0.0000	2
2	2	3	3.000	0.000	0.000	0.0000	2
3	3	4	3.000	0.000	0.000	0.0000	2
4	4	5	3.000	0.000	0.000	0.0000	2
5	5	6	3.000	0.000	0.000	0.0000	2
6	6	7	3.000	0.000	0.000	0.0000	2
11	12	13	3.000	0.000	0.000	0.0000	2
12	13	14	3.000	0.000	0.000	0.0000	2
13	14	15	3.000	0.000	0.000	0.0000	2
14	15	16	3.000	0.000	0.000	0.0000	2
15	16	17	3.000	0.000	0.000	0.0000	2
16	17	18	3.000	0.000	0.000	0.0000	2
21	23	24	3.000	0.000	0.000	0.0000	2
22	24	25	3.000	0.000	0.000	0.0000	2
23	25	26	3.000	0.000	0.000	0.0000	2
24	26	27	3.000	0.000	0.000	0.0000	2
25	27	28	3.000	0.000	0.000	0.0000	2
26	28	29	3.000	0.000	0.000	0.0000	2
27	29	30	3.000	0.000	0.000	0.0000	2
31	34	35	3.000	0.000	0.000	0.0000	2
32	35	36	3.000	0.000	0.000	0.0000	2
33	36	37	3.000	0.000	0.000	0.0000	2
34	37	38	3.000	0.000	0.000	0.0000	2
35	38	39	3.000	0.000	0.000	0.0000	2
36	39	40	3.000	0.000	0.000	0.0000	2
37	40	41	3.000	0.000	0.000	0.0000	2
41	2	13	6.000	0.000	0.000	0.0000	4
42	3	14	6.000	0.000	0.000	0.0000	4
43	4	15	6.000	0.000	0.000	0.0000	4
44	5	16	6.000	0.000	0.000	0.0000	4
45	6	17	6.000	0.000	0.000	0.0000	4
46	7	18	6.000	0.000	0.000	0.0000	4
51	13	24	6.000	0.000	0.000	0.0000	4
52	14	25	6.000	0.000	0.000	0.0000	4
53	15	26	6.000	0.000	0.000	0.0000	4
54	16	27	6.000	0.000	0.000	0.0000	4
55	17	28	6.000	0.000	0.000	0.0000	4
56	18	29	6.000	0.000	0.000	0.0000	4
61	24	35	6.000	0.000	0.000	0.0000	4
62	25	36	6.000	0.000	0.000	0.0000	4
63	26	37	6.000	0.000	0.000	0.0000	4
64	27	38	6.000	0.000	0.000	0.0000	4
65	28	39	6.000	0.000	0.000	0.0000	4
66	29	40	6.000	0.000	0.000	0.0000	4
67	30	41	6.000	0.000	0.000	0.0000	4

F R A M E E L E M E N T D A T A

ELEMENT LABEL	SECTION LABEL	LOCAL PLANE	COORD SYSTEM	PLN 1ST	PLN 2ND	PLANE JOINTA	PLANE JOINTB	COORD ANGLE
1	FS1	12	0	+Z	+X	0	0	0.00
2	FS1	12	0	+Z	+X	0	0	0.00
3	FS1	12	0	+Z	+X	0	0	0.00
4	FS1	12	0	+Z	+X	0	0	0.00
5	FS1	12	0	+Z	+X	0	0	0.00
6	FS1	12	0	+Z	+X	0	0	0.00
11	FS1	12	0	+Z	+X	0	0	0.00
12	FS1	12	0	+Z	+X	0	0	0.00
13	FS1	12	0	+Z	+X	0	0	0.00
14	FS1	12	0	+Z	+X	0	0	0.00
15	FS1	12	0	+Z	+X	0	0	0.00

16	FS1	12	0	+Z	+X	0	0	0.00
21	FS1	12	0	+Z	+X	0	0	0.00
22	FS1	12	0	+Z	+X	0	0	0.00
23	FS1	12	0	+Z	+X	0	0	0.00
24	FS1	12	0	+Z	+X	0	0	0.00
25	FS1	12	0	+Z	+X	0	0	0.00
26	FS1	12	0	+Z	+X	0	0	0.00
27	FS1	12	0	+Z	+X	0	0	0.00
31	FS1	12	0	+Z	+X	0	0	0.00
32	FS1	12	0	+Z	+X	0	0	0.00
33	FS1	12	0	+Z	+X	0	0	0.00
34	FS1	12	0	+Z	+X	0	0	0.00
35	FS1	12	0	+Z	+X	0	0	0.00
36	FS1	12	0	+Z	+X	0	0	0.00
37	FS1	12	0	+Z	+X	0	0	0.00
41	FS2	12	0	+Z	+X	0	0	0.00
42	FS2	12	0	+Z	+X	0	0	0.00
43	FS2	12	0	+Z	+X	0	0	0.00
44	FS2	12	0	+Z	+X	0	0	0.00
45	FS2	12	0	+Z	+X	0	0	0.00
46	FS2	12	0	+Z	+X	0	0	0.00
51	FS2	12	0	+Z	+X	0	0	0.00
52	FS2	12	0	+Z	+X	0	0	0.00
53	FS2	12	0	+Z	+X	0	0	0.00
54	FS2	12	0	+Z	+X	0	0	0.00
55	FS2	12	0	+Z	+X	0	0	0.00
56	FS2	12	0	+Z	+X	0	0	0.00
61	FS2	12	0	+Z	+X	0	0	0.00
62	FS2	12	0	+Z	+X	0	0	0.00
63	FS2	12	0	+Z	+X	0	0	0.00
64	FS2	12	0	+Z	+X	0	0	0.00
65	FS2	12	0	+Z	+X	0	0	0.00
66	FS2	12	0	+Z	+X	0	0	0.00
67	FS2	12	0	+Z	+X	0	0	0.00

T O T A L W E I G H T S A N D M A S S E S

SECTION LABEL	WEIGHT	MASS
FS1	67.4655	6.8740
FS2	109.5593	11.1629
TOTAL	177.0247	18.0370

L O A D C O N D I T I O N VERT

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.1000E+01

DISTRIBUTED SPAN LOADS ON FRAME ELEMENTS

ELEMENT LABEL	LOC DOF	DISTANCE AT START	DISTANCE AT END	FORCE AT START	FORCE AT END	MOMENT AT START	MOMENT AT END
41	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
42	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
43	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
44	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
45	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
46	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
51	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
52	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
53	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
54	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
55	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
56	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
61	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
62	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
63	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
64	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
65	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
66	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		
67	U2	0.000E+00	0.100E+01	-0.200E+01	-0.200E+01		

L O A D C O N D I T I O N LATERAL

SELF-WEIGHT MULTIPLIER FOR ENTIRE STRUCTURE = 0.0000E+00

JOINT FORCES IN LOCAL COORDINATES

JOINT LABEL	FORCE 1	FORCE 2	FORCE 3	MOMENT 1	MOMENT 2	MOMENT 3
2	0.500E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
3	0.500E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
4	0.500E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00
5	0.500E+01	0.000E+00	0.000E+00	0.000E+00	0.000E+00	0.000E+00

```
      6  0.100E+02  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
      7  0.100E+02  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
```



```
 O U T P U T   S E L E C T I O N

 DISPLACEMENTS AT JOINTS

      LOAD      MODES      SPEC      HIST      MOVE      COMB
     LABEL                LABEL     LABEL     LABEL     LABEL

      VERT
   LATERAL

 APPLIED AND INTERNAL LOADS AT JOINTS

      LOAD      MODES      SPEC      HIST      MOVE      COMB
     LABEL                LABEL     LABEL     LABEL     LABEL

      VERT
   LATERAL

 INTERNAL FORCES AT ELEMENT FRAME

      LOAD      MODES      SPEC      HIST      MOVE      COMB
     LABEL                LABEL     LABEL     LABEL     LABEL

      VERT
   LATERAL

 JOINT FORCES AT ELEMENT FRAME

      LOAD      MODES      SPEC      HIST      MOVE      COMB
     LABEL                LABEL     LABEL     LABEL     LABEL

      VERT
   LATERAL
```



```
 I N P U T   C O M P L E T E
```

S A P 2 0 0 0 (R)
Structural Analysis Programs
Version E6.10

21 Mar 2000 09:11:29

D I S P L A C E M E N T D E G R E E S O F F R E E D O M

(A) = Active DOF, equilibrium equation
(-) = Restrained DOF, reaction computed
(+) = Constrained DOF
( ) = Null DOF

JOINTS			UX	UY	UZ	RX	RY	RZ
1			-		-		A	
2	TO	7	A		A		A	
12			-		-		A	
13	TO	18	A		A		A	
23			-		-		A	
24	TO	30	A		A		A	
34			-		-		A	
35	TO	41	A		A		A	

J O I N T D I S P L A C E M E N T S
TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

LOAD VERT ------------------

JOINT	UX	UZ	RY
1	.000000	.000000	-4.54E-05
2	-9.41E-06	-0.000225	8.91E-05
3	1.43E-05	-0.000413	8.65E-05
4	3.07E-05	-0.000564	9.63E-05
5	5.38E-05	-0.000675	0.000104
6	7.93E-05	-0.000748	0.000106
7	0.000157	-0.000780	0.000175
12	.000000	.000000	-1.45E-07
13	1.22E-06	-0.000400	1.61E-06
14	1.54E-05	-0.000731	1.66E-05
15	3.19E-05	-0.000995	2.50E-05
16	5.46E-05	-0.001193	4.25E-05
17	8.28E-05	-0.001324	3.98E-05
18	0.000135	-0.001390	5.03E-05
23	.000000	.000000	4.33E-06
24	1.27E-05	-0.000425	3.97E-06
25	1.86E-05	-0.000782	-6.00E-06
26	3.43E-05	-0.001073	-9.02E-06
27	5.65E-05	-0.001298	-1.13E-05
28	8.65E-05	-0.001460	-1.04E-05
29	0.000107	-0.001557	-4.24E-05
30	-0.000135	-0.001599	4.91E-05
34	.000000	.000000	5.05E-05
35	1.35E-05	-0.000267	-8.53E-05
36	1.99E-05	-0.000497	-7.98E-05
37	3.57E-05	-0.000689	-8.65E-05
38	5.77E-05	-0.000841	-9.07E-05
39	8.78E-05	-0.000954	-9.40E-05
40	0.000103	-0.001027	-0.000114
41	-0.000152	-0.001060	-0.000224

LOAD LATERAL ------------------

JOINT	UX	UZ	RY
1	.000000	.000000	0.002006
2	0.004626	9.14E-05	0.000528
3	0.006425	0.000149	0.000307
4	0.007911	0.000186	0.000266
5	0.008979	0.000211	0.000228
6	0.009927	0.000222	0.000164
7	0.010457	0.000224	9.21E-05
12	.000000	.000000	0.002111
13	0.004614	-1.03E-05	0.000287
14	0.006397	-1.20E-05	0.000197
15	0.007787	-1.15E-05	0.000163
16	0.008954	-9.82E-06	0.000137
17	0.009882	-7.59E-06	9.92E-05
18	0.010407	-6.00E-06	4.57E-05

J O I N T D I S P L A C E M E N T S
TRANSLATIONS AND ROTATIONS, IN GLOBAL COORDINATES

JOINT	UX	UZ	RY
23	.000000	.000000	0.002104
24	0.004600	9.82E-06	0.000286
25	0.006382	1.12E-05	0.000197
26	0.007773	1.04E-05	0.000163
27	0.008938	8.33E-06	0.000136
28	0.009854	5.95E-06	9.78E-05
29	0.010378	4.39E-06	4.91E-05
30	0.010531	4.64E-06	4.44E-05
34	.000000	.000000	0.001987
35	0.004583	-9.09E-05	0.000524
36	0.006380	-0.000149	0.000308
37	0.007767	-0.000187	0.000266
38	0.008932	-0.000210	0.000223
39	0.009842	-0.000220	0.000160
40	0.010366	-0.000223	8.27E-05
41	0.010532	-0.000223	3.83E-05

A P P L I E D L O A D S
FORCES AND MOMENTS ACTING ON JOINTS, IN GLOBAL COORDINATES

LOAD VERT ------------------

JOINT	FX	FZ	MY
1	.000000	-1.297413	.000000

```
     2     .000000  -11.477964     8.883139
     3     .000000  -11.477964     8.883139
     4     .000000  -11.477964     8.883139
     5     .000000  -11.477964     8.883139
     6     .000000  -11.477964     8.883139
     7     .000000  -10.180552     8.883139
    12     .000000   -1.297413      .000000
    13     .000000  -20.361104     1.60E-14
    14     .000000  -20.361104     1.60E-14
    15     .000000  -20.361104     1.60E-14
    16     .000000  -20.361104     1.60E-14
    17     .000000  -20.361104     1.60E-14
    18     .000000  -19.063691     1.60E-14
    23     .000000   -1.297413      .000000
    24     .000000  -20.361104     1.60E-14
    25     .000000  -20.361104     1.60E-14
    26     .000000  -20.361104     1.60E-14
    27     .000000  -20.361104     1.60E-14
    28     .000000  -20.361104     1.60E-14
    29     .000000  -20.361104     1.60E-14
    30     .000000  -10.180552     8.883139
    34     .000000   -1.297413      .000000
    35     .000000  -11.477964    -8.883139
    36     .000000  -11.477964    -8.883139
    37     .000000  -11.477964    -8.883139
    38     .000000  -11.477964    -8.883139
    39     .000000  -11.477964    -8.883139
    40     .000000  -11.477964    -8.883139
    41     .000000  -10.180552    -8.883139
LOAD LATERAL ------------------
  JOINT           FX          FZ          MY
     2     5.000000     .000000     .000000
     3     5.000000     .000000     .000000
     4     5.000000     .000000     .000000
     5     5.000000     .000000     .000000
     6    10.000000     .000000     .000000
     7    10.000000     .000000     .000000
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 5
G L O B A L   F O R C E   B A L A N C E
TOTAL FORCE AND MOMENT AT THE ORIGIN, IN GLOBAL COORDINATES
LOAD    VERT ------------------
                  FX          FY           FZ           MX           MY           MZ
APPLIED      .000000     .000000  -405.024747      .000000   137.735574      .000000
REACTNS     1.09E-14     .000000   405.024747      .000000  -137.735574      .000000
  TOTAL     1.09E-14     .000000    -5.68E-13      .000000     3.41E-13      .000000
LOAD LATERAL ------------------
                  FX          FY           FZ           MX           MY           MZ
APPLIED    40.000000     .000000      .000000      .000000   480.000000      .000000
REACTNS   -40.000000     .000000     1.07E-14      .000000  -480.000000      .000000
  TOTAL    -6.39E-14     .000000     1.07E-14      .000000     2.27E-12      .000000
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 6
F R A M E   E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM         1 ------------------
LOAD    VERT ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     1     0.816605     .000000    69.561426      .000000     1.38E-16      .000000
     2    -0.816605     .000000   -66.966601      .000000     2.449816      .000000
LOAD LATERAL ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     1    -8.982460     .000000   -27.756703      .000000     6.93E-15      .000000
     2     8.982460     .000000    27.756703      .000000   -26.947381      .000000
ELEM         2 ------------------
LOAD    VERT ------------------
  JOINT           FY          FY           FZ           MX           MY           MZ
     2     2.611554     .000000    58.504322      .000000     3.941074      .000000
     3    -2.611554     .000000   -55.909496      .000000     3.893588      .000000
LOAD LATERAL ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     2    -5.954896     .000000   -17.596670      .000000    -6.924680      .000000
     3     5.954896     .000000    17.596670      .000000   -10.940006      .000000
ELEM         3 ------------------
LOAD    VERT ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     3     2.807400     .000000    46.988612      .000000     4.120981      .000000
     4    -2.807400     .000000   -44.393787      .000000     4.301216      .000000
LOAD LATERAL ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     3    -5.722883     .000000   -11.732831      .000000    -8.207464      .000000
     4     5.722883     .000000    11.732831      .000000    -8.961184      .000000
ELEM         4 ------------------
LOAD    VERT ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     4     3.017396     .000000    35.217087      .000000     4.459042      .000000
     5    -3.017396     .000000   -32.622262      .000000     4.593146      .000000
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 7
F R A M E   E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
LOAD LATERAL ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     4    -4.661364     .000000    -7.014936      .000000    -6.643920      .000000
     5     4.661364     .000000     7.014936      .000000    -7.340172      .000000
ELEM         5 ------------------
LOAD    VERT ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     5     3.153406     .000000    23.266381      .000000     4.706837      .000000
     6    -3.153406     .000000   -20.671555      .000000     4.753381      .000000
LOAD LATERAL ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
     5    -3.926147     .000000    -3.225879      .000000    -5.306865      .000000
     6     3.926147     .000000     3.225879      .000000    -6.471576      .000000
ELEM         6 ------------------
LOAD    VERT ------------------
  JOINT           FX          FY           FZ           MX           MY           MZ
```

```
      6    3.750902     .000000   11.187923       .000000    4.997366       .000000
      7   -3.750902     .000000   -8.593098       .000000    6.255321       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
      6   -1.589671     .000000   -0.795336       .000000   -1.730379       .000000
      7    1.589671     .000000    0.795336       .000000   -3.038635       .000000
ELEM       11 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     12    0.010682     .000000  122.731645       .000000   -3.06E-18       .000000
     13   -0.010682     .000000 -120.136820       .000000    0.032046       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     12  -11.084411     .000000    3.126082       .000000   -1.66E-16       .000000
     13   11.084411     .000000   -3.126082       .000000  -33.253232       .000000
PROGRAM SAP2000 - VERSION E6.10                                  FILE:EJEMPLO9.OUT PAGE 8
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM       12 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     13    0.144793     .000000  101.912283       .000000    0.080316       .000000
     14   -0.144793     .000000  -99.317457       .000000    0.354062       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     13  -11.524494     .000000    0.508852       .000000  -16.468811       .000000
     14   11.524494     .000000   -0.508852       .000000  -18.104670       .000000
ELEM       13 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     14    0.499248     .000000   81.505551       .000000    0.672851       .000000
     15   -0.499248     .000000  -78.910726       .000000    0.824892       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     14   -9.271377     .000000   -0.133337       .000000  -13.597942       .000000
     15    9.271377     .000000    0.133337       .000000  -14.216188       .000000
ELEM       14 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     15    0.692183     .000000   61.308506       .000000    0.969808       .000000
     16   -0.692183     .000000  -58.713681       .000000    1.106735       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     15   -7.814389     .000000   -0.519534       .000000  -11.484626       .000000
     16    7.814389     .000000    0.519534       .000000  -11.958541       .000000
ELEM       15 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     16    0.874630     .000000   41.237390       .000000    1.244912       .000000
     17   -0.874630     .000000  -38.642565       .000000    1.378978       .000000
PROGRAM SAP2000 - VERSION E6.10                                  FILE:EJEMPLO9.OUT PAGE 9
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     16   -6.251986     .000000   -0.676443       .000000   -9.035087       .000000
     17    6.251986     .000000    0.676443       .000000   -9.720872       .000000
ELEM       16 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     17    0.903738     .000000   21.273240       .000000    1.260490       .000000
     18   -0.903738     .000000  -18.678415       .000000    1.450725       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     17   -3.357797     .000000   -0.483850       .000000   -4.549555       .000000
     18    3.357797     .000000    0.483850       .000000   -5.523837       .000000
ELEM       21 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     23   -0.002235     .000000  130.433623       .000000    1.46E-17       .000000
     24    0.002235     .000000 -127.838797       .000000   -0.006704       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     23  -11.045663     .000000   -2.982055       .000000    1.27E-14       .000000
     24   11.045663     .000000    2.982055       .000000  -33.116986       .000000
ELEM       22 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     24   -0.097589     .000000  109.738587       .000000   -0.055600       .000000
     25    0.097589     .000000 -107.143762       .000000   -0.237168       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     24  -11.526709     .000000   -0.413356       .000000  -16.474336       .000000
     25   11.526709     .000000    0.413356       .000000  -18.105790       .000000
PROGRAM SAP2000 - VERSION E6.10                                  FILE:EJEMPLO9.OUT PAGE 10
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM       23 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     25   -0.416628     .000000   89.582675       .000000   -0.597409       .000000
     26    0.416628     .000000  -86.967850       .000000   -0.652474       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     25   -9.279987     .000000    0.236201       .000000  -13.609975       .000000
     26    9.279987     .000000   -0.236201       .000000  -14.229987       .000000
ELEM       24 -----------------
LOAD    VERT -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     26   -0.574047     .000000   69.774059       .000000   -0.839995       .000000
     27    0.574047     .000000  -67.179234       .000000   -0.882146       .000000
LOAD LATERAL -----------------
  JOINT          FX          FY          FZ            MX          MY            MZ
     26   -7.815629     .000000    0.630088       .000000  -11.482325       .000000
```

```
    27     7.815629      .000000    -0.630088       .000000  -11.964563       .000000
ELEM       25 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     27    -0.682415      .000000    50.249964      .000000    -1.032169      .000000
     28     0.682415      .000000   -47.655139      .000000    -1.015076      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     27    -6.152927      .000000     0.722728      .000000    -8.880128      .000000
     28     6.152927      .000000    -0.722728      .000000    -9.578652      .000000
ELEM       26 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     28    -1.090707      .000000    30.926253      .000000    -1.344552      .000000
     29     1.090707      .000000   -28.331427      .000000    -1.927570      .000000
PROGRAM SAP2000 - VERSION E6.10                                FILE:EJEMPLO9.OUT PAGE 11
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     28    -3.307681      .000000     0.472495      .000000    -4.517675      .000000
     29     3.307681      .000000    -0.472495      .000000    -5.405369      .000000
ELEM       27 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     29     2.753659      .000000    11.450543      .000000     3.296919      .000000
     30    -2.753659      .000000    -8.855718      .000000     4.964059      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     29    -0.166062      .000000    -0.076089      .000000    -0.197643      .000000
     30     0.166062      .000000     0.076089      .000000    -0.300542      .000000
ELEM       31 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     34    -0.825053      .000000    82.298053      .000000     4.40E-17      .000000
     35     0.825053      .000000   -79.703228      .000000    -2.475158      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     34    -8.887467      .000000    27.612676      .000000    -4.50E-15      .000000
     35     8.887467      .000000   -27.612676      .000000   -26.662400      .000000
ELEM       32 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     35    -2.658758      .000000    71.191419      .000000    -4.038574      .000000
     36     2.658758      .000000   -66.596594      .000000    -3.937698      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     35    -5.993902      .000000    17.501174      .000000    -7.018198      .000000
     36     5.993902      .000000   -17.501174      .000000   -10.963508      .000000
PROGRAM SAP2000 - VERSION E6.10                                FILE:EJEMPLO9.OUT PAGE 12
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM       33 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     36    -2.890020      .000000    59.591635      .000000    -4.273420      .000000
     37     2.890020      .000000   -56.996810      .000000    -4.396639      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     36    -5.725753      .000000    11.629967      .000000    -8.210822      .000000
     37     5.725753      .000000   -11.629967      .000000    -8.966439      .000000
ELEM       34 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     37    -3.135530      .000000    47.690685      .000000    -4.665389      .000000
     38     3.135530      .000000   -45.095860      .000000    -4.741202      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     37    -4.708618      .000000     6.904383      .000000    -6.666391      .000000
     38     4.708618      .000000    -6.904383      .000000    -7.459462      .000000
ELEM       35 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     38    -3.345621      .000000    35.558466      .000000    -4.988659      .000000
     39     3.345621      .000000   -32.963641      .000000    -5.048203      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     38    -3.668940      .000000     3.179595      .000000    -4.931138      .000000
     39     3.668940      .000000    -3.179595      .000000    -6.075682      .000000
ELEM       36 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     39    -3.563933      .000000    23.246650      .000000    -5.161317      .000000
     40     3.563933      .000000   -20.651825      .000000    -5.930482      .000000
PROGRAM SAP2000 - VERSION E6.10                                FILE:EJEMPLO9.OUT PAGE 13
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     39    -1.744850      .000000     0.806691      .000000    -1.915115      .000000
     40     1.744850      .000000    -0.806691      .000000    -3.319435      .000000
ELEM       37 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     40    -2.753659      .000000    11.505386      .000000    -3.132390      .000000
     41     2.753659      .000000    -8.910561      .000000    -5.128588      .000000
LOAD LATERAL ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
     40     0.166062      .000000     0.076089      .000000     0.654175      .000000
     41    -0.166062      .000000    -0.076089      .000000    -0.155990      .000000
ELEM       41 ------------------
LOAD    VERT ------------------
  JOINT           FX           FY           FZ           MX           MY           MZ
      2    -1.794949      .000000     8.462279      .000000    -6.390890      .000000
```

```
    13    1.794949     .000000     9.303999      .000000     8.916051      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     2    1.972435     .000000   -10.160033      .000000    33.872061      .000000
    13   -1.972435     .000000    10.160033      .000000    27.088139      .000000
ELEM       42 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     3   -0.195846     .000000     8.920884      .000000    -8.014569      .000000
    14    0.195846     .000000     8.845394      .000000     7.788101      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     3    4.767987     .000000    -5.863839      .000000    19.147470      .000000
    14   -4.767987     .000000     5.863839      .000000    16.035564      .000000
PROGRAM SAP2000 - VERSION E6.10                                    FILE:EJEMPLO9.OUT PAGE 14
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM       43 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     4   -0.209996     .000000     9.176700      .000000    -8.760260      .000000
    15    0.209996     .000000     8.589579      .000000     6.998897      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     4    3.938481     .000000    -4.717894      .000000    15.605104      .000000
    15   -3.938481     .000000     4.717894      .000000    12.702262      .000000
ELEM       44 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     5   -0.136010     .000000     9.355882      .000000    -9.299983      .000000
    16    0.136010     .000000     8.410397      .000000     6.463529      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     5    4.264783     .000000    -3.789057      .000000    12.647037      .000000
    16   -4.264783     .000000     3.789057      .000000    10.087305      .000000
ELEM       45 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     6   -0.597496     .000000     9.483632      .000000    -9.750767      .000000
    17    0.597496     .000000     8.282646      .000000     6.147808      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     6    7.663524     .000000    -2.430544      .000000     8.201955      .000000
    17   -7.663524     .000000     2.430544      .000000     6.381307      .000000
ELEM       46 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     7    3.750902     .000000     8.593098      .000000    -6.255321      .000000
    18   -3.750902     .000000     9.173191      .000000     7.995570      .000000
PROGRAM SAP2000 - VERSION E6.10                                    FILE:EJEMPLO9.OUT PAGE 15
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
     7    8.410329     .000000    -0.795336      .000000     3.038635      .000000
    18   -8.410329     .000000     0.795336      .000000     1.733381      .000000
ELEM       51 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    13   -1.929060     .000000     8.920536      .000000    -9.028414      .000000
    24    1.929060     .000000     8.845740      .000000     8.604020      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    13    2.412518     .000000    -7.542803      .000000    22.633904      .000000
    24   -2.412518     .000000     7.542803      .000000    22.622912      .000000
ELEM       52 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    14   -0.550300     .000000     8.966512      .000000    -8.815013      .000000
    25    0.550300     .000000     8.799767      .000000     8.314778      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    14    2.514870     .000000    -5.221650      .000000    15.667048      .000000
    25   -2.514870     .000000     5.221650      .000000    15.662855      .000000
ELEM       53 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    15   -0.402930     .000000     9.012641      .000000    -8.793597      .000000
    26    0.402930     .000000     8.753637      .000000     8.016584      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    15    2.481494     .000000    -4.331698      .000000    12.998552      .000000
    26   -2.481494     .000000     4.331698      .000000    12.991634      .000000
PROGRAM SAP2000 - VERSION E6.10                                    FILE:EJEMPLO9.OUT PAGE 16
F R A M E     E L E M E N T   J O I N T   F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM       54 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    16   -0.318459     .000000     9.065894      .000000    -8.815176      .000000
    27    0.318459     .000000     8.700385      .000000     7.718650      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    16    2.702381     .000000    -3.632148      .000000    10.906323      .000000
    27   -2.702381     .000000     3.632148      .000000    10.886563      .000000
ELEM       55 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    17   -0.626605     .000000     9.086679      .000000    -8.787276      .000000
    28    0.626605     .000000     8.679599      .000000     7.566037      .000000
LOAD LATERAL ------------------
  JOINT          FX          FY           FZ           MX           MY           MZ
    17    4.769335     .000000    -2.623137      .000000     7.889121      .000000
    28   -4.769335     .000000     2.623137      .000000     7.849701      .000000
```

```
ELEM      56 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     18    4.654641     .000000    9.505234     .000000   -9.446295     .000000
     29   -4.654641     .000000    8.261045     .000000    5.713727     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     18    5.052531     .000000   -1.279186     .000000    4.790456     .000000
     29   -5.052531     .000000    1.279186     .000000    3.884659     .000000
ELEM      61 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     24   -1.833705     .000000    9.254470     .000000   -8.741716     .000000
     35    1.833705     .000000    8.511809     .000000    6.513733     .000000
PROGRAM SAP2000 - VERSION E6.10                          FILE:EJEMPLO9.OUT PAGE 17
F R A M E    E L E M E N T    J O I N T    F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     24    2.893564     .000000  -10.111502     .000000   26.988412     .000000
     35   -2.893564     .000000   10.111502     .000000   33.680598     .000000
ELEM      62 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     25   -0.231262     .000000    8.761320     .000000   -7.480201     .000000
     36    0.231262     .000000    9.004959     .000000    8.211118     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     25    0.268149     .000000   -5.871207     .000000   16.052910     .000000
     36   -0.268149     .000000    5.871207     .000000   19.174330     .000000
ELEM      63 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     26   -0.245511     .000000    8.460154     .000000   -6.524115     .000000
     37    0.245511     .000000    9.306125     .000000    9.062028     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     26    1.017136     .000000   -4.725585     .000000   12.720677     .000000
     37   -1.017136     .000000    4.725585     .000000   15.632830     .000000
ELEM      64 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     27   -0.210091     .000000    8.228885     .000000   -5.804335     .000000
     38    0.210091     .000000    9.537394     .000000    9.729861     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     27    1.039678     .000000   -3.724788     .000000    9.956128     .000000
     38   -1.039678     .000000    3.724788     .000000   12.390600     .000000
PROGRAM SAP2000 - VERSION E6.10                          FILE:EJEMPLO9.OUT PAGE 18
F R A M E    E L E M E N T    J O I N T    F O R C E S
FORCES AND MOMENTS ACTING ON ELEMENTS, IN GLOBAL COORDINATES
ELEM      65 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     28   -0.218312     .000000    8.049287     .000000   -5.206409     .000000
     39    0.218312     .000000    9.716991     .000000   10.209520     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     28    1.924090     .000000   -2.372904     .000000    6.246627     .000000
     39   -1.924090     .000000    2.372904     .000000    7.990797     .000000
ELEM      66 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     29    0.810274     .000000    8.639840     .000000   -7.084076     .000000
     40   -0.810274     .000000    9.146439     .000000    8.662873     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     29    1.910912     .000000   -0.730602     .000000    1.728352     .000000
     40   -1.910912     .000000    0.730602     .000000    2.665260     .000000
ELEM      67 ------------------
LOAD    VERT ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     30    2.753659     .000000    8.855718     .000000   -4.964059     .000000
     41   -2.753659     .000000    8.910561     .000000    5.128588     .000000
LOAD LATERAL ------------------
  JOINT          FX          FY          FZ          MX          MY          MZ
     30   -0.166062     .000000   -0.076089     .000000    0.300542     .000000
     41    0.166062     .000000    0.076089     .000000    0.155990     .000000
PROGRAM SAP2000 - VERSION E6.10                          FILE:EJEMPLO9.OUT PAGE 19
F R A M E    E L E M E N T    I N T E R N A L    F O R C E S
ELEM       1 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -69.56142[illegible]   -0.816605     .000000     .000000     .000000   -1.38E-16
0.50000  -68.264013   -0.816605     .000000     .000000     .000000    1.224908
1.00000  -66.966601   -0.816605     .000000     .000000     .000000    2.449816
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   27.756703    8.982460     .000000     .000000     .000000   -6.93E-15
0.50000   27.756703    8.982460     .000000     .000000     .000000  -13.473690
1.00000   27.756703    8.982460     .000000     .000000     .000000  -26.947381
ELEM       2 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -58.504322   -2.611554     .000000     .000000     .000000   -3.941074
0.50000  -57.206909   -2.611554     .000000     .000000     .000000   -0.023743
1.00000  -55.909496   -2.611554     .000000     .000000     .000000    3.893588
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   17.596670    5.954896     .000000     .000000     .000000    6.924680
0.50000   17.596670    5.954896     .000000     .000000     .000000   -2.007663
1.00000   17.596670    5.954896     .000000     .000000     .000000  -10.940006
ELEM       3 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
```

```
REL DIST          P          V2          V3           T          M2          M3
0.00000  -46.988612   -2.807400     .000000     .000000     .000000   -4.120981
0.50000  -45.691200   -2.807400     .000000     .000000     .000000    0.090119
1.00000  -44.393787   -2.807400     .000000     .000000     .000000    4.301218
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   11.732831    5.722883     .000000     .000000     .000000    8.207464
0.50000   11.732831    5.722883     .000000     .000000     .000000   -0.376860
1.00000   11.732831    5.722883     .000000     .000000     .000000   -8.961184
PROGRAM SAP2000 - VERSION E6.10                            FILE:EJEMPLO9.OUT PAGE 20
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      4 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -35.217087   -3.017396     .000000     .000000     .000000   -4.459042
0.50000  -33.919675   -3.017396     .000000     .000000     .000000    0.067052
1.00000  -32.622262   -3.017396     .000000     .000000     .000000    4.593146
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    7.014936    4.661364     .000000     .000000     .000000    6.643920
0.50000    7.014936    4.661364     .000000     .000000     .000000   -0.348126
1.00000    7.014936    4.661364     .000000     .000000     .000000   -7.340172
ELEM      5 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -23.266381   -3.153406     .000000     .000000     .000000   -4.706837
0.50000  -21.968968   -3.153406     .000000     .000000     .000000    0.023272
1.00000  -20.671555   -3.153406     .000000     .000000     .000000    4.753381
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    3.225879    3.926147     .000000     .000000     .000000    5.306865
0.50000    3.225879    3.926147     .000000     .000000     .000000   -0.582355
1.00000    3.225879    3.926147     .000000     .000000     .000000   -6.471576
ELEM      6 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -11.187923   -3.750902     .000000     .000000     .000000   -4.997386
0.50000   -9.890510   -3.750902     .000000     .000000     .000000    0.628967
1.00000   -8.593098   -3.750902     .000000     .000000     .000000    6.255321
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.795336    1.589671     .000000     .000000     .000000    1.730379
0.50000    0.795336    1.589671     .000000     .000000     .000000   -0.654128
1.00000    0.795336    1.589671     .000000     .000000     .000000   -3.038635
PROGRAM SAP2000 - VERSION E6.10                            FILE:EJEMPLO9.OUT PAGE 21
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM     11 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000 -122.731645   -0.010682     .000000     .000000     .000000    3.06E-18
0.50000 -121.434233   -0.010682     .000000     .000000     .000000    0.016023
1.00000 -120.136820   -0.010682     .000000     .000000     .000000    0.032046
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -3.126082   11.084411     .000000     .000000     .000000    1.66E-16
0.50000   -3.126082   11.084411     .000000     .000000     .000000  -16.626616
1.00000   -3.126082   11.084411     .000000     .000000     .000000  -33.253232
ELEM     12 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000 -101.912283   -0.144793     .000000     .000000     .000000   -0.080316
0.50000 -100.614870   -0.144793     .000000     .000000     .000000    0.136873
1.00000  -99.317457   -0.144793     .000000     .000000     .000000    0.354062
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -0.508852   11.524494     .000000     .000000     .000000   16.468811
0.50000   -0.508852   11.524494     .000000     .000000     .000000   -0.817930
1.00000   -0.508852   11.524494     .000000     .000000     .000000  -18.104670
ELEM     13 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -81.505551   -0.499248     .000000     .000000     .000000   -0.672851
0.50000  -80.208139   -0.499248     .000000     .000000     .000000    0.076021
1.00000  -78.910726   -0.499248     .000000     .000000     .000000    0.824892
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.133337    9.271377     .000000     .000000     .000000   13.597942
0.50000    0.133337    9.271377     .000000     .000000     .000000   -0.309123
1.00000    0.133337    9.271377     .000000     .000000     .000000  -14.216188
PROGRAM SAP2000 - VERSION E6.10                            FILE:EJEMPLO9.OUT PAGE 22
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM     14 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -61.308506   -0.692181     .000000     .000000     .000000   -0.969808
0.50000  -60.011093   -0.692181     .000000     .000000     .000000    0.068464
1.00000  -58.713681   -0.692181     .000000     .000000     .000000    1.106735
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.519534    7.814389     .000000     .000000     .000000   11.484626
0.50000    0.519534    7.814389     .000000     .000000     .000000   -0.236958
1.00000    0.519534    7.814389     .000000     .000000     .000000  -11.958541
ELEM     15 ------------------ LENGTH =    3.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000  -41.237390   -0.874630     .000000     .000000     .000000   -1.244912
0.50000  -39.939978   -0.874630     .000000     .000000     .000000    0.067033
1.00000  -38.642565   -0.874630     .000000     .000000     .000000    1.378978
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.676443    6.251986     .000000     .000000     .000000    9.035087
0.50000    0.676443    6.251986     .000000     .000000     .000000   -0.342892
1.00000    0.676443    6.251986     .000000     .000000     .000000   -9.720872
ELEM     16 ------------------ LENGTH =    3.000000
```

```
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -21.273240   -0.903738     .000000     .000000     .000000   -1.260490
0.50000  -19.975827   -0.903738     .000000     .000000     .000000    0.095118
1.00000  -18.678415   -0.903738     .000000     .000000     .000000    1.450725
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    0.483850    3.357797     .000000     .000000     .000000    4.549555
0.50000    0.483850    3.357797     .000000     .000000     .000000   -0.487141
1.00000    0.483850    3.357797     .000000     .000000     .000000   -5.523837
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 23
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      21 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000 -130.433623    0.002235     .000000     .000000     .000000   -1.46E-17
0.50000 -129.136210    0.002235     .000000     .000000     .000000   -0.003352
1.00000 -127.838797    0.002235     .000000     .000000     .000000   -0.006704
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    2.982055   11.045663     .000000     .000000     .000000   -1.27E-14
0.50000    2.982055   11.045663     .000000     .000000     .000000  -16.568494
1.00000    2.982055   11.045663     .000000     .000000     .000000  -33.136988
ELEM      22 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000 -109.738587    0.097589     .000000     .000000     .000000    0.055600
0.50000 -108.441175    0.097589     .000000     .000000     .000000   -0.090784
1.00000 -107.143762    0.097589     .000000     .000000     .000000   -0.237168
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    0.413356   11.526709     .000000     .000000     .000000   16.474336
0.50000    0.413356   11.526709     .000000     .000000     .000000   -0.815727
1.00000    0.413356   11.526709     .000000     .000000     .000000  -18.105790
ELEM      23 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -89.582675    0.416628     .000000     .000000     .000000    0.597409
0.50000  -88.285263    0.416628     .000000     .000000     .000000   -0.027533
1.00000  -86.987850    0.416628     .000000     .000000     .000000   -0.652474
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000   -0.236201    9.279987     .000000     .000000     .000000   13.609975
0.50000   -0.236201    9.279987     .000000     .000000     .000000   -0.310006
1.00000   -0.236201    9.279987     .000000     .000000     .000000  -14.229987
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 24
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      24 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -69.774059    0.574047     .000000     .000000     .000000    0.839995
0.50000  -68.476647    0.574047     .000000     .000000     .000000   -0.021076
1.00000  -67.179234    0.574047     .000000     .000000     .000000   -0.882146
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000   -0.630088    7.815629     .000000     .000000     .000000   11.482325
0.50000   -0.630088    7.815629     .000000     .000000     .000000   -0.241119
1.00000   -0.630088    7.815629     .000000     .000000     .000000  -11.964563
ELEM      25 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -50.249964    0.682415     .000000     .000000     .000000    1.032169
0.50000  -48.952552    0.682415     .000000     .000000     .000000    0.008546
1.00000  -47.655139    0.682415     .000000     .000000     .000000   -1.015076
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000   -0.722728    6.152927     .000000     .000000     .000000    8.880128
0.50000   -0.722728    6.152927     .000000     .000000     .000000   -0.349262
1.00000   -0.722728    6.152927     .000000     .000000     .000000   -9.578652
ELEM      26 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -30.926253    1.090707     .000000     .000000     .000000    1.344552
0.50000  -29.628840    1.090707     .000000     .000000     .000000   -0.291509
1.00000  -28.331427    1.090707     .000000     .000000     .000000   -1.927570
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000   -0.472495    3.307681     .000000     .000000     .000000    4.517675
0.50000   -0.472495    3.307681     .000000     .000000     .000000   -0.443847
1.00000   -0.472495    3.307681     .000000     .000000     .000000   -5.405369
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 25
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      27 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -11.450543   -2.753659     .000000     .000000     .000000   -3.296919
0.50000  -10.153130   -2.753659     .000000     .000000     .000000    0.833570
1.00000   -8.855718   -2.753659     .000000     .000000     .000000    4.964059
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    0.076089    0.166062     .000000     .000000     .000000    0.197643
0.50000    0.076089    0.166062     .000000     .000000     .000000   -0.051450
1.00000    0.076089    0.166062     .000000     .000000     .000000   -0.300542
ELEM      31 ----------------- LENGTH =     3.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -82.298053    0.825053     .000000     .000000     .000000   -4.40E-17
0.50000  -81.000640    0.825053     .000000     .000000     .000000   -1.237579
1.00000  -79.703228    0.825053     .000000     .000000     .000000   -2.475158
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000  -27.612676    8.887467     .000000     .000000     .000000    4.50E-15
0.50000  -27.612676    8.887467     .000000     .000000     .000000  -13.331200
1.00000  -27.612676    8.887467     .000000     .000000     .000000  -26.662400
```

```
ELEM      32 ------------------ LENGTH =     3.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -71.191419     2.658758      .000000       .000000       .000000     4.038574
0.50000  -69.894006     2.658758      .000000       .000000       .000000     0.050438
1.00000  -68.596594     2.658758      .000000       .000000       .000000    -3.937698
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -17.501174     5.993902      .000000       .000000       .000000     7.018198
0.50000  -17.501174     5.993902      .000000       .000000       .000000    -1.972655
1.00000  -17.501174     5.993902      .000000       .000000       .000000   -10.963508
PROGRAM SAP2000 - VERSION E6.10                                       FILE:EJEMPLO9.OUT PAGE 26
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      33 ------------------ LENGTH =     3.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -59.591635     2.890020      .000000       .000000       .000000     4.273420
0.50000  -58.294223     2.890020      .000000       .000000       .000000    -0.061610
1.00000  -56.996810     2.890020      .000000       .000000       .000000    -4.396639
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -11.629967     5.725753      .000000       .000000       .000000     8.210822
0.50000  -11.629967     5.725753      .000000       .000000       .000000    -0.377809
1.00000  -11.629967     5.725753      .000000       .000000       .000000    -8.966439
ELEM      34 ------------------ LENGTH =     3.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -47.690685     3.135530      .000000       .000000       .000000     4.665389
0.50000  -46.393273     3.135530      .000000       .000000       .000000    -0.037907
1.00000  -45.095860     3.135530      .000000       .000000       .000000    -4.741202
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000   -6.904383     4.708618      .000000       .000000       .000000     6.666391
0.50000   -6.904383     4.708618      .000000       .000000       .000000    -0.396535
1.00000   -6.904383     4.708618      .000000       .000000       .000000    -7.459462
ELEM      35 ------------------ LENGTH =     3.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -35.558466     3.345621      .000000       .000000       .000000     4.988659
0.50000  -34.261054     3.345621      .000000       .000000       .000000    -0.029772
1.00000  -32.963641     3.345621      .000000       .000000       .000000    -5.048203
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000   -3.179595     3.668940      .000000       .000000       .000000     4.931138
0.50000   -3.179595     3.668940      .000000       .000000       .000000    -0.572272
1.00000   -3.179595     3.668940      .000000       .000000       .000000    -6.075682
PROGRAM SAP2000 - VERSION E6.10                                       FILE:EJEMPLO9.OUT PAGE 27
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      36 ------------------ LENGTH =     3.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -23.246650     3.563933      .000000       .000000       .000000     5.161317
0.50000  -21.949237     3.563933      .000000       .000000       .000000    -0.184583
1.00000  -20.651825     3.563933      .000000       .000000       .000000    -5.530482
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000   -0.806691     1.744850      .000000       .000000       .000000     1.915115
0.50000   -0.806691     1.744850      .000000       .000000       .000000    -0.702160
1.00000   -0.806691     1.744850      .000000       .000000       .000000    -3.319435
ELEM      37 ------------------ LENGTH =     3.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000  -11.505386     2.753659      .000000       .000000       .000000     3.132390
0.50000  -10.207973     2.753659      .000000       .000000       .000000    -0.998099
1.00000   -8.910561     2.753659      .000000       .000000       .000000    -5.128588
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000   -0.076089    -0.166062      .000000       .000000       .000000    -0.654175
0.50000   -0.076089    -0.166062      .000000       .000000       .000000    -0.405083
1.00000   -0.076089    -0.166062      .000000       .000000       .000000    -0.155990
ELEM      41 ------------------ LENGTH =     6.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000    1.794949    -8.462279      .000000       .000000       .000000    -6.390890
0.25000    1.794949    -4.020709      .000000       .000000       .000000     2.971351
0.50000    1.794949     0.420860      .000000       .000000       .000000     5.671238
0.75000    1.794949     4.862430      .000000       .000000       .000000     1.708770
1.00000    1.794949     9.303999      .000000       .000000       .000000    -8.916051
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000   -1.972435    10.160033      .000000       .000000       .000000    33.872061
0.25000   -1.972435    10.160033      .000000       .000000       .000000    18.632011
0.50000   -1.972435    10.160033      .000000       .000000       .000000     3.391961
0.75000   -1.972435    10.160033      .000000       .000000       .000000   -11.848089
1.00000   -1.972435    10.160033      .000000       .000000       .000000   -27.088139
PROGRAM SAP2000 - VERSION E6.10                                       FILE:EJEMPLO9.OUT PAGE 28
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      42 ------------------ LENGTH =     6.000000
LOAD    VERT ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000    0.195846    -8.920084      .000000       .000000       .000000    -8.014569
0.25000    0.195846    -4.479314      .000000       .000000       .000000     2.035580
0.50000    0.195846    -0.037745      .000000       .000000       .000000     5.423374
0.75000    0.195846     4.403825      .000000       .000000       .000000     2.148814
1.00000    0.195846     8.845394      .000000       .000000       .000000    -7.788101
LOAD LATERAL ------------------
REL DIST           P           V2            V3             T            M2           M3
0.00000   -4.767987     5.863839      .000000       .000000       .000000    19.147470
0.25000   -4.767987     5.863839      .000000       .000000       .000000    10.351712
0.50000   -4.767987     5.863839      .000000       .000000       .000000     1.555953
0.75000   -4.767987     5.863839      .000000       .000000       .000000    -7.239806
1.00000   -4.767987     5.863839      .000000       .000000       .000000   -16.035564
ELEM      43 ------------------ LENGTH =     6.000000
LOAD    VERT ------------------
```

```
REL DIST           P          V2          V3           T          M2          M3
0.00000     0.209996   -9.176700     .000000     .000000     .000000   -6.760260
0.25000     0.209996   -4.735130     .000000     .000000     .000000    1.673612
0.50000     0.209996   -0.293561     .000000     .000000     .000000    5.445130
0.75000     0.209996    4.148009     .000000     .000000     .000000    2.554294
1.00000     0.209996    8.589579     .000000     .000000     .000000   -6.998697
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -3.938481    4.717894     .000000     .000000     .000000   15.605104
0.25000    -3.938481    4.717894     .000000     .000000     .000000    8.528263
0.50000    -3.938481    4.717894     .000000     .000000     .000000    1.451421
0.75000    -3.938481    4.717894     .000000     .000000     .000000   -5.625420
1.00000    -3.938481    4.717894     .000000     .000000     .000000  -12.702262
ELEM       44 ------------------ LENGTH =     6.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000     0.136010   -9.355882     .000000     .000000     .000000   -9.299983
0.25000     0.136010   -4.914312     .000000     .000000     .000000    1.402662
0.50000     0.136010   -0.472742     .000000     .000000     .000000    5.442953
0.75000     0.136010    3.968827     .000000     .000000     .000000    2.820889
1.00000     0.136010    8.410397     .000000     .000000     .000000   -6.463529
PROGRAM SAP2000 - VERSION E6.10                                 FILE:EJEMPLO9.OUT PAGE 29
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -4.264783    3.789057     .000000     .000000     .000000   12.647037
0.25000    -4.264783    3.789057     .000000     .000000     .000000    6.963451
0.50000    -4.264783    3.789057     .000000     .000000     .000000    1.279866
0.75000    -4.264783    3.789057     .000000     .000000     .000000   -4.403720
1.00000    -4.264783    3.789057     .000000     .000000     .000000  -10.087305
ELEM       45 ------------------ LENGTH =     6.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000     0.597496   -9.483632     .000000     .000000     .000000   -9.750767
0.25000     0.597496   -5.042063     .000000     .000000     .000000    1.143504
0.50000     0.597496   -0.600493     .000000     .000000     .000000    5.375421
0.75000     0.597496    3.841076     .000000     .000000     .000000    2.944984
1.00000     0.597496    8.282646     .000000     .000000     .000000   -6.147808
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -7.663524    2.430544     .000000     .000000     .000000    8.201955
0.25000    -7.663524    2.430544     .000000     .000000     .000000    4.556139
0.50000    -7.663524    2.430544     .000000     .000000     .000000    0.910324
0.75000    -7.663524    2.430544     .000000     .000000     .000000   -2.735491
1.00000    -7.663524    2.430544     .000000     .000000     .000000   -6.381307
ELEM       46 ------------------ LENGTH =     6.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -3.750902   -8.593098     .000000     .000000     .000000   -6.255321
0.25000    -3.750902   -4.151528     .000000     .000000     .000000    3.303149
0.50000    -3.750902    0.290042     .000000     .000000     .000000    4.199263
0.75000    -3.750902    4.731612     .000000     .000000     .000000    2.433024
1.00000    -3.750902    9.173181     .000000     .000000     .000000   -7.995570
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -8.410329    0.795336     .000000     .000000     .000000    3.038635
0.25000    -8.410329    0.795336     .000000     .000000     .000000    1.845631
0.50000    -8.410329    0.795336     .000000     .000000     .000000    0.652627
0.75000    -8.410329    0.795336     .000000     .000000     .000000   -0.540377
1.00000    -8.410329    0.795336     .000000     .000000     .000000   -1.733381
PROGRAM SAP2000 - VERSION E6.10                                 FILE:EJEMPLO9.OUT PAGE 30
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM       51 ------------------ LENGTH =     6.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000     1.929060   -8.920538     .000000     .000000     .000000   -9.028414
0.25000     1.929060   -4.478969     .000000     .000000     .000000    1.021216
0.50000     1.929060   -0.037399     .000000     .000000     .000000    4.408492
0.75000     1.929060    4.404171     .000000     .000000     .000000    1.133413
1.00000     1.929060    8.845740     .000000     .000000     .000000   -8.804020
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -2.412518    7.542803     .000000     .000000     .000000   22.633904
0.25000    -2.412518    7.542803     .000000     .000000     .000000   11.319700
0.50000    -2.412518    7.542803     .000000     .000000     .000000    0.005496
0.75000    -2.412518    7.542803     .000000     .000000     .000000  -11.308706
1.00000    -2.412518    7.542803     .000000     .000000     .000000  -22.622912
ELEM       52 ------------------ LENGTH =     6.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000     0.550300   -8.966512     .000000     .000000     .000000   -8.815013
0.25000     0.550300   -4.524942     .000000     .000000     .000000    1.303578
0.50000     0.550300   -0.083373     .000000     .000000     .000000    4.759813
0.75000     0.550300    4.358197     .000000     .000000     .000000    1.553695
1.00000     0.550300    8.799767     .000000     .000000     .000000   -8.314778
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000    -2.514870    5.221650     .000000     .000000     .000000   15.667046
0.25000    -2.514870    5.221650     .000000     .000000     .000000    7.834571
0.50000    -2.514870    5.221650     .000000     .000000     .000000    0.002096
0.75000    -2.514870    5.221650     .000000     .000000     .000000   -7.830379
1.00000    -2.514870    5.221650     .000000     .000000     .000000  -15.662855
ELEM       53 ------------------ LENGTH =     6.000000
LOAD    VERT -----------------
REL DIST           P          V2          V3           T          M2          M3
0.00000     0.402930   -9.012641     .000000     .000000     .000000   -8.793597
0.25000     0.402930   -4.571072     .000000     .000000     .000000    1.394188
0.50000     0.402930   -0.129502     .000000     .000000     .000000    4.919618
0.75000     0.402930    4.312067     .000000     .000000     .000000    1.782694
1.00000     0.402930    8.753637     .000000     .000000     .000000   -8.016584
PROGRAM SAP2000 - VERSION E6.10                                 FILE:EJEMPLO9.OUT PAGE 31
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
LOAD LATERAL -----------------
REL DIST           P          V2          V3           T          M2          M3
```

```
0.00000   -2.481494    4.331698     .000000     .000000     .000000   12.998552
0.25000   -2.481494    4.331698     .000000     .000000     .000000    6.501005
0.50000   -2.481494    4.331698     .000000     .000000     .000000    0.003459
0.75000   -2.481494    4.331698     .000000     .000000     .000000   -6.494088
1.00000   -2.481494    4.331698     .000000     .000000     .000000  -12.991634
ELEM      54 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.318459   -9.065894     .000000     .000000     .000000   -8.815176
0.25000    0.318459   -4.624324     .000000     .000000     .000000    1.452487
0.50000    0.318459   -0.182754     .000000     .000000     .000000    5.057796
0.75000    0.318459    4.258815     .000000     .000000     .000000    2.000750
1.00000    0.318459    8.700385     .000000     .000000     .000000   -7.718650
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -2.702381    3.632148     .000000     .000000     .000000   10.906323
0.25000   -2.702381    3.632148     .000000     .000000     .000000    5.458102
0.50000   -2.702381    3.632148     .000000     .000000     .000000    0.009880
0.75000   -2.702381    3.632148     .000000     .000000     .000000   -5.438341
1.00000   -2.702381    3.632148     .000000     .000000     .000000  -10.886563
ELEM      55 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.626605   -9.086679     .000000     .000000     .000000   -8.787276
0.25000    0.626605   -4.645109     .000000     .000000     .000000    1.511565
0.50000    0.626605   -0.203540     .000000     .000000     .000000    5.148052
0.75000    0.626605    4.238030     .000000     .000000     .000000    2.122185
1.00000    0.626605    8.679599     .000000     .000000     .000000   -7.566037
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -4.769335    2.623137     .000000     .000000     .000000    7.889121
0.25000   -4.769335    2.623137     .000000     .000000     .000000    3.954415
0.50000   -4.769335    2.623137     .000000     .000000     .000000    0.019710
0.75000   -4.769335    2.623137     .000000     .000000     .000000   -3.914995
1.00000   -4.769335    2.623137     .000000     .000000     .000000   -7.849701
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 32
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      56 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -4.654641   -9.505234     .000000     .000000     .000000   -9.446295
0.25000   -4.654641   -5.063664     .000000     .000000     .000000    1.480379
0.50000   -4.654641   -0.622095     .000000     .000000     .000000    5.744698
0.75000   -4.654641    3.819475     .000000     .000000     .000000    3.346663
1.00000   -4.654641    8.261045     .000000     .000000     .000000   -5.713727
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -5.052531    1.279186     .000000     .000000     .000000    3.790456
0.25000   -5.052531    1.279186     .000000     .000000     .000000    1.871677
0.50000   -5.052531    1.279186     .000000     .000000     .000000   -0.047102
0.75000   -5.052531    1.279186     .000000     .000000     .000000   -1.965880
1.00000   -5.052531    1.279186     .000000     .000000     .000000   -3.884655
ELEM      61 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    1.833705   -9.254470     .000000     .000000     .000000   -8.741716
0.25000    1.833705   -4.812900     .000000     .000000     .000000    1.808811
0.50000    1.833705   -0.371331     .000000     .000000     .000000    5.696984
0.75000    1.833705    4.070239     .000000     .000000     .000000    2.922803
1.00000    1.833705    8.511809     .000000     .000000     .000000   -6.513733
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -2.893564   10.111502     .000000     .000000     .000000   26.988412
0.25000   -2.893564   10.111502     .000000     .000000     .000000   11.821159
0.50000   -2.893564   10.111502     .000000     .000000     .000000   -3.346093
0.75000   -2.893564   10.111502     .000000     .000000     .000000  -18.513345
1.00000   -2.893564   10.111502     .000000     .000000     .000000  -33.680598
ELEM      62 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.231262   -8.761320     .000000     .000000     .000000   -7.480201
0.25000    0.231262   -4.319750     .000000     .000000     .000000    2.330601
0.50000    0.231262    0.121819     .000000     .000000     .000000    5.479049
0.75000    0.231262    4.563389     .000000     .000000     .000000    1.965143
1.00000    0.231262    9.004959     .000000     .000000     .000000   -8.211116
PROGRAM SAP2000 - VERSION E6.10                              FILE:EJEMPLO9.OUT PAGE 33
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -0.268149    5.871207     .000000     .000000     .000000   16.052910
0.25000   -0.268149    5.871207     .000000     .000000     .000000    7.246100
0.50000   -0.268149    5.871207     .000000     .000000     .000000   -1.560710
0.75000   -0.268149    5.871207     .000000     .000000     .000000  -10.367520
1.00000   -0.268149    5.871207     .000000     .000000     .000000  -19.174330
ELEM      63 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.245511   -8.460154     .000000     .000000     .000000   -6.524115
0.25000    0.245511   -4.018584     .000000     .000000     .000000    2.834938
0.50000    0.245511    0.422985     .000000     .000000     .000000    5.531637
0.75000    0.245511    4.864555     .000000     .000000     .000000    1.565982
1.00000    0.245511    9.306125     .000000     .000000     .000000   -9.062028
LOAD LATERAL ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -1.017136    4.725585     .000000     .000000     .000000   12.720677
0.25000   -1.017136    4.725585     .000000     .000000     .000000    5.632301
0.50000   -1.017136    4.725585     .000000     .000000     .000000   -1.456076
0.75000   -1.017136    4.725585     .000000     .000000     .000000   -8.544453
1.00000   -1.017136    4.725585     .000000     .000000     .000000  -15.632830
ELEM      64 ------------------ LENGTH =    6.000000
LOAD    VERT ------------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.210091   -8.228885     .000000     .000000     .000000   -5.804335
0.25000    0.210091   -3.787315     .000000     .000000     .000000    3.207815
```

```
0.50000    0.210091    0.654254     .000000     .000000     .000000    5.557611
0.75000    0.210091    5.095824     .000000     .000000     .000000    1.245052
1.00000    0.210091    9.537394     .000000     .000000     .000000   -9.729861
LOAD LATERAL -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -1.039678    3.724788     .000000     .000000     .000000    9.958128
0.25000   -1.039678    3.724788     .000000     .000000     .000000    4.370946
0.50000   -1.039678    3.724788     .000000     .000000     .000000   -1.216236
0.75000   -1.039678    3.724788     .000000     .000000     .000000   -6.803418
1.00000   -1.039678    3.724788     .000000     .000000     .000000  -12.390600
PROGRAM SAP2000 - VERSION E6.10                           FILE:EJEMPLO9.OUT PAGE 34
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
ELEM      65 ----------------- LENGTH =    6.000000
LOAD    VERT -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.218312   -6.049287     .000000     .000000     .000000   -5.206409
0.25000    0.218312   -3.607718     .000000     .000000     .000000    3.536345
0.50000    0.218312    0.833852     .000000     .000000     .000000    5.616744
0.75000    0.218312    5.275422     .000000     .000000     .000000    1.034789
1.00000    0.218312    9.716991     .000000     .000000     .000000  -10.209520
LOAD LATERAL -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -1.924090    2.372904     .000000     .000000     .000000    6.246627
0.25000   -1.924090    2.372904     .000000     .000000     .000000    2.687271
0.50000   -1.924090    2.372904     .000000     .000000     .000000   -0.872085
0.75000   -1.924090    2.372904     .000000     .000000     .000000   -4.431441
1.00000   -1.924090    2.372904     .000000     .000000     .000000   -7.990797
ELEM      66 ----------------- LENGTH =    6.000000
LOAD    VERT -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -0.810274   -8.619840     .000000     .000000     .000000   -7.083076
0.25000   -0.810274   -4.178270     .000000     .000000     .000000    2.515507
0.50000   -0.810274    0.263300     .000000     .000000     .000000    5.451735
0.75000   -0.810274    4.704869     .000000     .000000     .000000    1.725608
1.00000   -0.810274    9.146439     .000000     .000000     .000000   -8.662873
LOAD LATERAL -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -1.910912    0.730602     .000000     .000000     .000000    1.718352
0.25000   -1.910912    0.730602     .000000     .000000     .000000    0.622449
0.50000   -1.910912    0.730602     .000000     .000000     .000000   -0.473454
0.75000   -1.910912    0.730602     .000000     .000000     .000000   -1.569357
1.00000   -1.910912    0.730602     .000000     .000000     .000000   -2.665260
ELEM      67 ----------------- LENGTH =    6.000000
LOAD    VERT -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000   -2.753659   -8.855718     .000000     .000000     .000000   -4.964059
0.25000   -2.753659   -4.414148     .000000     .000000     .000000    4.988340
0.50000   -2.753659    0.027422     .000000     .000000     .000000    8.278385
0.75000   -2.753659    4.468991     .000000     .000000     .000000    4.906076
1.00000   -2.753659    8.910561     .000000     .000000     .000000   -5.128588
PROGRAM SAP2000 - VERSION E6.10                           FILE:EJEMPLO9.OUT PAGE 35
F R A M E   E L E M E N T   I N T E R N A L   F O R C E S
LOAD LATERAL -----------------
REL DIST          P          V2          V3           T          M2          M3
0.00000    0.166062    0.076089     .000000     .000000     .000000    0.300542
0.25000    0.166062    0.076089     .000000     .000000     .000000    0.186409
0.50000    0.166062    0.076089     .000000     .000000     .000000    0.072276
0.75000    0.166062    0.076089     .000000     .000000     .000000   -0.041857
1.00000    0.166062    0.076089     .000000     .000000     .000000   -0.155990
```

SAP2000®

Análisis y Diseño Integrado de Estructuras por el Método de Elementos Finitos

EJEMPLOS DE APLICACION

COMPUTERS & STRUCTURES INC.

Computers and Structures, Inc.
Berkeley, California, USA

Version 6.1
September 1997

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMP-TIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Indice

Ejemplo 1 Pórtico Bidimensional bajo Carga Estática............ 1

Ejemplo 2 Pórtico Bidimensional con Carga de un Espectro de Respuesta 13

Ejemplo 3 Pórtico Bidimensional Análisis de Historia en el Tiempo.. 20

Ejemplo 4 Diseño en Acero de un Pórtico Bidimensional.......... 32

Apéndice A Descripción de los Iconos de la Barra de Herramientas.. A1

Apéndice B Descripción de los Iconos de la Barra Flotante.......... B1

4. Edite la geometría de la malla y presione el botón OK para cerrar la plantilla.

   Sugerencia: *Finalizada la edición de la malla, se puede hacer click con el botón derecho del mouse sobre las columnas para verificar si éstas tienen la longitud apropiada. Esta es una manera muy práctica de obtener información sobre cualquier nudo o elemento de la estructura.*

Edición de Apoyos

El siguiente paso es el cambio de los apoyos de la estructura de la opción por defecto que corresponde a nudos articulados, a la opción de nudos rígidos que tenemos en este caso.

1. Seleccione el icono **Pointer Tool** de la barra de herramientas flotante.
2. Marque un área rectangular que abarque los tres nudos en la base de la estructura.

   Sugerencia: *Se puede observar la barra de estado para ver el número y tipo de elementos que han sido seleccionados.*

3. Seleccione el icono **Assign Joint Restraints** de la barra de herramientas flotante para asignar empotramiento en los apoyos de la estructura. Se pueden asignar también otras características de los nudos desde el menú **Assign**.

Definición de la Sección Transversal de los Elementos

1. Seleccione primeramente todas las secciones transversales que van a emplearse en el pórtico. Desde el menú **Define** seleccione la opción **Frame Sections**. Luego importe los perfiles de acero mostrados en la Figura 1-1.

   Nota: *Se puede seleccionar más de una sección a la vez de la lista Section Selection. Para ello presione la tecla Ctrl mientras se efectúa la selección.*

2. Bajo el menú **Select** encontrará varias formas de seleccionar nudos y elementos. Para este problema son útiles los modos de selección **Pointer/Window** e **Intersecting Line**.
3. Una vez seleccionados los elementos del pórtico deseados, se podrán asignar las secciones de acero correspondientes a través del botón **Assign Frame Sections** que está ubicado en la barra de herramientas flotante.

Asignación de Cargas

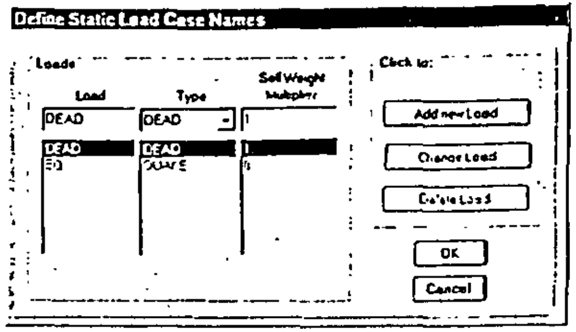


Figura 1-2 Plantilla con los nombres de las condiciones de carga estática

1. El primer paso al ingresar las cargas es definir las condiciones de carga estática. Para ello ingrese al menú **Define** y seleccione la opción **Static Load Cases**.

- DEAD puede usarse para las cargas verticales por peso propio de las vigas, manteniendo el indicador **Self Weight Multiplier** con el valor 1. SAP2000 agrega el peso propio de las vigas.

2. Defina una condición de carga lateral estática llamada EQ para la carga de sismo. Asigne esta carga lateral como una carga del tipo QUAKE. Esto permitirá al programa efectuar automáticamente las combinaciones de carga a ser empleadas por el módulo de diseño del SAP2000. Además asigne al parámetro **Self Weight Multiplier** el valor cero.
3. Las cargas verticales mostradas en la Figura 1-1 pueden asignarse a las vigas seleccionando todas las vigas utilizando el botón **Assign Frame Span Loads** de la barra de herramientas flotante.
4. Las cargas laterales estáticas necesitan ingresarse seleccionando individualmente cada nudo y empleando el botón **Assign Joint Loads**.

   Importante: *Asegúrese de que esté añadiendo las cargas a la condición de carga correspondiente.*

Creació. Diafragmas de Piso

Crear diafragmas de piso y especificar la masa del piso sólo en la dirección X son técnicas comúnmente usadas para reducir el tamaño del problema. Por otro lado, al añadir diafragmas el comportamiento del modelo se asemeja al de un edificio con diafragmas rígidos.

1. Repita los siguientes pasos para cada piso:
   - Seleccione todos los nudos del piso.
   - Entre al menú **Assign** y seleccione la opción **Joint ... Constraints**.
   - Seleccione **Add Diaphragm** del la caja de opciones.
     - En la plantilla **Diaphragm Constraint** ingrese un nombre para el diafragma del primer piso. En este caso usaremos el nombre DIA1.
     - Seleccione la opción **Z-axis constraint**. Esta opción define un diafragma perpendicular al eje-Z.
     - Presione el botón OK.
   - Presione el botón OK para finalizar la operación.
   - Repita estos pasos para los demás pisos usando diferentes nombres en cada uno.
2. La masa de todos los pisos es la misma. Luego seleccione un nudo en cada piso.
3. Cambie las unidades en que se van a ingresar los datos a Kip-in, puesto que la masa indicada en la Figura 1-1 está dada en esas unidades.
4. Del menú **Assign** seleccione la opción **Joint ... Masses**
   - Ingrese la masa de cada piso en la dirección del eje coordenado local 1 (que en éste caso coincide con la dirección del Eje Global X).
   - Todos los demás valores son cero.
5. Retorne las unidades a Kip-ft.

Propiedades de los Materiales

Por último, antes de efectuar el análisis de la estructura, deberemos verificar que la asignación de las propiedades de los materiales es la correcta.

1. Desde el menú **Define** seleccione la opción **Materials**.
2. En la plantilla **Materials** seleccione STEEL y presione el botón MODIFY/SHOW MATERIAL

- En la plantilla **Material Property Data** verifique que las p[illegible]es del material sean las correctas. Recuerde que los valores son reportados [illegible] unidades con las que se este trabajando en este momento.

Efectuando el Análisis

Una vez que los datos han sido ingresados, es tiempo para correr el modelo y revisar los resultados.

1. Grabe el modelo.
2. Especifique los parámetros para el análisis seleccionando la opción **Analyze** del menú **Set Options**.
   - En la plantilla **Analysis Options** seleccione **Plane Frame Analysis** para reducir el tamaño del problema y por tanto reducir el tiempo de cálculo.
   - Presione el botón OK para aceptar los cambios realizados.
3. Seleccione la opción **Run** del menú **Analyze** para proceder al análisis la estructura.

   Nota: *Una vez concluido el análisis Ud. podrá revisar los resultados completos en la pantalla antes de presionar el botón OK. Esta será su primera verificación para ver si existe algún problema en el modelo.*

Usando los Resultados

Verificación de los Resultados

Una vez que se ha analizado el modelo se debe verificar si los resultados son correctos y que sus valores son del orden y magnitud a los esperados.

Verificación del Modelo:

1. Verifique que el cortante total en la base es igual a la carga lateral total para la condición de carga EQ.
   - Seleccione el grupo de elementos del pórtico que están ubicados en el primer nivel así como los nudos en la base de la estructura.
   - Desde el menú **Assign** seleccione **Group Names**.
   - Asigne a este grupo de elementos un nombre representativo por ejemplo BASE SHEAR.
   - Seleccione el botón ADD NEW GROUP NAME y presione el botón OK.
   - En el menú **Display** seleccione la opción **Show Group Joint Force Sums** y elija el grupo previamente creado.

2. Observe la deformada de la estructura y cree una animación de la misma bajo cargas verticales y laterales para asegurarse de que el comportamiento del modelo es el esperado.

   - Ingrese al menú **Display** y seleccione **Show Deformed Shape** y seleccione la condición de carga en la que este interesado. También seleccione la opción **Wire Shadow**, así podrá ver la geometría no-deformada de la estructura al mismo tiempo. Vea las Figuras 1-3 y 1-4 para las formas deformadas de la estructura. Haga click con el botón derecho del mouse sobre cualquier nudo para observar los desplazamientos y rotaciones correspondientes.

   - Genere una animación de la deformada presionando el botón START ANIMATION ubicado en la parte inferior de la barra de estado (Para esto se necesita que se encuentre activa una ventana conteniendo la deformada de la estructura). La animación así creada puede salvarse como una archivo *.AVI para verse después desde el menú **File**. (Vea la Ayuda En-línea bajo el ítem "Export an AVI file".)

     Intente esto : ***Presione los botones + y – ubicados junto al botón Animate y vea lo que le sucede a la deformada de le estructura.***

   - Presione el botón STOP ANIMATION cuando haya terminado de observar la animación.

Si los procedimientos antes descritos muestran que la información ingresada aparenta ser correcta, podemos entonces avanzar hacia procedimientos más avanzados de revisión de los resultados.

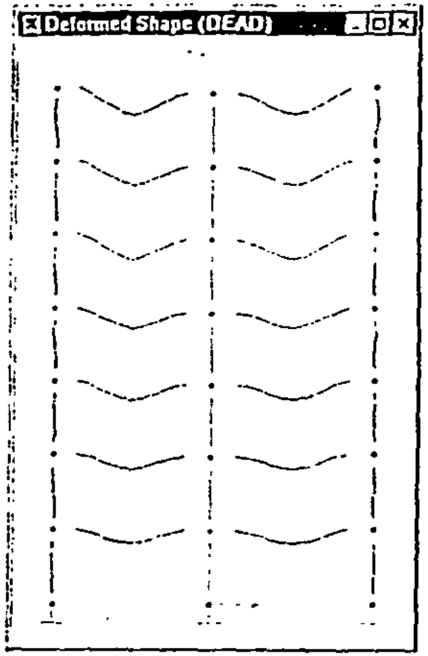


Figura 1-3 Deformada de la Estructura para Cargas Verticales

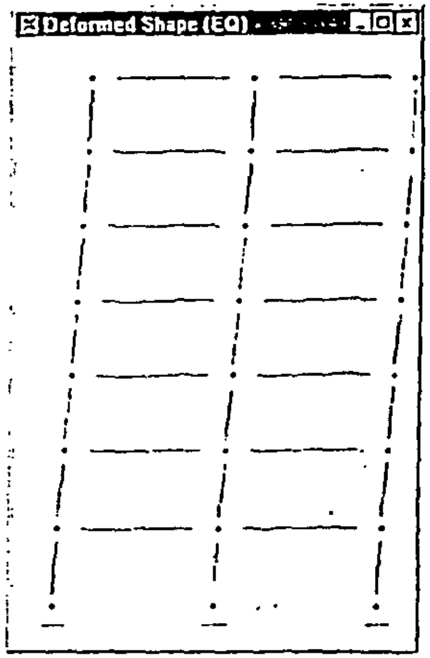


Figura 1-4 Deformada de la Estructura para Cargas Laterales

Comportamiento Estructural

En muchas ocasiones se desea verificar si la estructura se encuentra dentro de determinados límites de comportamiento, tales como rangos de esfuerzo especificados por algún código de diseño. SAP2000 hace todas estas verificaciones automáticamente cuando los elementos son diseñados. (Las opciones de diseño del SAP2000 serán discutidas con mayor detalle en los siguientes ejemplos).

1. Los elementos estructurales pueden diseñarse desde al menú **Design** y seleccionando la opción **Start Design/Check of Structure**.

   - Los elementos del pórtico mostrarán en este momento colores que representan el nivel del esfuerzo en cada elemento. Asimismo en la parte inferior de cada elemento se muestra un valor numérico representativo del nivel de esfuerzo presente en el elemento. Un valor 1 por ejemplo significa 100% esforzado.

- Para tener más información sobre el diseño de los elementos entre al menú **Design** y seleccione la opción **Display Design Info**.

2. Se puede también ver la información del diseño de cada elemento, o inclusive asignar secciones transversales alternativas, haciendo click con el botón derecho del mouse sobre un elemento.

   - De la ventana que se muestre se puede seleccionar el botón DETAILS para apreciar información detallada de la sección bajo cada una de las combinaciones de carga empleadas en el diseño.

   - También se puede rediseñar el elemento después de cambiar sus parámetros de diseño, longitud efectiva, factor K o propiedades a la sección, presionando el botón REDESIGN.

3. Si se ha seleccionado una nueva sección la cual se quiere utilizar en el diseño final de la estructura, únicamente ingrese al menú Design y seleccione la opción **Update Analysis Sections** para reanalizar la estructura con las nuevas secciones seleccionadas.

Nota: Puede ser necesario el uso del botón Refresh Window de la barra de herramientas para actualizar la información en la ventana activa luego de haber efectuado cambios en los parámetros de diseño.

Observando e Imprimiendo Resultados

A menudo se necesita disponer de una copia impresa de los resultados de los análisis obtenidos con el SAP2000. Existen diferentes formas de obtenerlos:

1. Se pueden elegir los resultados que nos interesan con la opción **Generate Output** ubicada en la plantilla **Analysis Options**. El botón **Select Output Options** que aparece permitirá seleccionar cuantos y cuales de los resultados queremos imprimir. Estos resultados son escritos en un archivo de texto con el mismo nombre de nuestro archivo de datos, pero con la extensión *.OUT.

2. Los datos ingresados así como la mayor parte de los resultados generados también pueden verse a partir del menú **Display**.

3. Desde el menú **File** se puede optar por imprimir ya sea **Gráficos**, **Tablas con los Datos Ingresados** ó **Tablas con los Resultados del Análisis y Diseño de elementos**.

Sugerencia: Si existen elementos ó nudos seleccionados al momento de generar la impresión de resultados, únicamente se imprimirá la información correspondiente a dichos elementos. De lo contrario, la impresión se generará para todos los elementos y nudos del modelo.

4. El análisis efectuado por el SAP2000 genera dos archivos de salida. El archivo *filename*.EKO que incluye toda la información empleada en el análisis; y el archivo *filename*.OUT, que

229

contiene los resultados del análisis así como los resultados específica[illegible]leccionados en el menú **Analyze ...Set Options**.

Recuerde: ***Es un buen hábito generar la salida primeramente a un archivo de texto antes de enviarlo directamente a la impresora. Esto nos permite revisar previamente la información usando cualquier editor de textos, sin tener que hacer frente a enormes pilas de papel.***

Comentarios Finales

Como habrá podido observar, SAP2000 es una poderosa herramienta para el análisis estructural que puede usarse en una gran variedad de problemas. Sin embargo, es muy importante entender los principios de ingeniería sobre los cuales este programa ha sido creado.

La mayoría de los trabajos en ingeniería se inician con sencillos anteproyectos para posteriormente madurar en complejos proyectos de análisis/diseño. Esto hace que sea muy importante decidir desde un inicio las herramientas apropiadas de forma tal que no sea necesario cambiar de programas a la mitad de un proyecto. SAP2000 trata de satisfacer la mayor parte de las necesidades que un diseñador puede tener durante el desarrollo de un proyecto.

Las características que SAP2000 ofrece en el proceso de diseño incluyen:

- La capacidad de diseñar pequeños ó grandes proyectos sin necesidad de aprender a usar un nuevo programa.
- La capacidad de diseñar elementos de concreto y acero en un mismo programa.
- Algoritmos de cálculo rápidos que permiten al usuario dedicar mayor tiempo en la modelación del problema y optimización del diseño de elementos estructurales.
- La habilidad para modificar y mejorar el diseño fácilmente.
- Existen probablemente tantas formas de modelar una estructura como ingenieros existen. Sin embargo, puede encontrar útiles algunas de las siguientes ideas:
- Comience con un modelo básico de la estructura y trate de entenderlo antes de añadir más detalles. Será más sencillo corregir problemas en el sistema estructural adoptado cuando el modelo es aún simple.
- Asegúrese de que la estructura pueda construirse y que se comportará en la manera en que la hemos modelado. Si no puede ser construida en esa forma, es necesario entender el efecto del proceso constructivo en el comportamiento final de la estructura.
- Documente detalladamente su diseño incluyendo información sobre las consideraciones asumidas, áreas que deban revisarse e incluso sobre información que aún es requerida. Para ello use el editor de textos **User Comments and Session Log** que se encuentra dentro del menú **File**. Este editor de textos incorporado en el programa, le permitirá que dichas anotaciones y comentarios formen parte del modelo.

- Experimente con sistemas estructurales alternativos. SAP2000 ha sido diseñado para efectuar cálculos numéricos rápidamente, permitiendo utilizar mayor tiempo en el mejoramiento de nuestros diseños.
- Así como hay un tiempo asignado para la revisión general al final de un proyecto, no hay razón por la que no deba haber un tiempo para revisar el proyecto desde sus inicios.

EJEMPLO 2

Pórtico Bidimensional con Carga de un Espectro de Respuesta

Descripción

Este ejemplo es una continuación del Ejemplo 1. En esta sección mostraremos como incorporar un Espectro de Respuesta en el análisis de un pórtico bidimensional. La base para definir el Espectro de Respuesta será el espectro del código UBC94S2 el cual esta incluido en el SAP2000.

Aspectos Significativos del Modelo y del SAP2000

- Uso del comando Help para obtener instrucciones sobre las opciones del SAP2000.
- Incorporación de una carga proveniente de un Espectro de Respuesta.
- Adecuar la escala del Espectro de Respuesta para su uso en el diseño.

Definiendo el Espectro de Respuesta

Un Espectro de Respuesta es la máxima respuesta de un sistema excitado en su base por una función aceleración-tiempo. Esta función se expresa en términos de la frecuencia natural de la estructura y del amortiguamiento del sistema. El Espectro de Respuesta del código UBC94S2 que vamos a emplear en este ejemplo es suministrado con SAP2000 y no es necesario definirlo por separado. Si hubiese la necesidad de definir un Espectro de Respuesta distinto, se puede usar la ayuda en línea para obtener instrucciones que indican paso a paso como efectuar esta tarea.

Ayuda En-línea

Recuerde: ***Podrá utilizar alguno de los métodos siguientes para obtener información sobre cualquiera de las funciones del SAP2000***

1. Del menú Help seleccione Search for Help on.
2. Con el plantilla Index seleccionada:
    - En el Area 1 escriba 'Define'. Ud. vera en el Area 2 una lista de todos los tópicos disponibles que comiencen con la palabra 'define'. Uno de esos tópicos es 'Define Response Spectrum Functions', que es el tópico del cual necesitamos obtener ayuda. Haga doble click en la línea con la frase 'Define Response Spectrum Functions' para que el programa muestre la información correspondiente.
3. Alternativamente, seleccione el indicador Find para buscar una palabra clave en cualesquiera de los tópicos disponibles en la Ayuda En-línea.
    - Si es la primera vez que usa la opción Find de la Ayuda En-línea del SAP2000, aparecerá una plantilla denominada Find Setup Wizard.
        - Presione el botón NEXT para aceptar el criterio para construir la base de datos de búsqueda.
        - Presione el botón FINISH para construir la base de datos.
    - En el Area 1 escriba 'Response Spectrum'
    - En el Area 3 encontrará nuevamente la opción 'Define Response Spectrum Functions' la cual puede seleccionarse para obtener la información de ayuda correspondiente.

    Nota: ***Se puede encontrar mayor información sobre el uso de la Ayuda En-línea, en la documentación de Windows. También puede ejecutar el archivo WINHELP32.HLP ubicado en C:\WINDOWS\HELP.***

Definiendo el Espectro de Respuesta

1. Si el modelo esta protegido (locked), use el botón Lock/Unlock Model para remover la protección y poder efectuar cambios en el modelo.

2. Ajuste las unidades a Kip-ft.
3. Ingrese al menú Define y seleccione la opción Response Spectrum.
4. Presione el botón ADD NEW SPECTRA en la plantilla Response Spectra.
5. En la plantilla Response Spectrum Case Data:
    - Especifique el amortiguamiento asociado al Espectro de Respuesta colocando en la casilla Damping el valor, para nuestro caso: 0.05 (5 %)
    - Seleccione UBC94S2 para la dirección U1, así como un factor de escala 32.2ft/sec^2 en la casilla Scale Factor. Este factor de escala es usado por el Espectro de Respuesta debido a que el espectro UBC94S2 esta normalizado al valor de la aceleración de la gravedad g
    - El resto de los valores por defecto son aceptables.
    - Presione el botón OK para aceptar los cambios hechos en ambas plantillas.

Efectuando el Análisis

Una vez que se han realizado las modificaciones, es tiempo de analizar el modelo y echar una mirada a los resultados del Espectro de Respuesta.

1. Grabe el modelo.
2. En el menú Analyze seleccione la opción Set Options.
    - Marque la casilla Dynamic Analysis.
    - Presione el botón SET DYNAMIC PARAMETERS y modifique el número de modos de vibración a ser considerados en el análisis en la opción Number of Modes. Para nuestro caso 7. El resto de valores por defecto son aceptables.
    - Presione el botón OK en ambas plantillas para aceptar los cambios.

    Nota: ***Se debe decidir cuantos modos de vibración deben considerarse en el análisis para obtener resultados adecuados. Para ello hay muchos criterios a tomar en cuenta, pero para una estructura sencilla como la que estamos analizando puede considerarse satisfactorio un numero de modos igual al numero de pisos.***
3. Seleccione la opción Run Minimized del menú Analyze para analizar la estructura.

    Nota: ***La opción Run Minimized es sumamente útil cuando se tienen modelos grandes que requieren mayor tiempo para ser analizados. Esta opción permite al SAP2000 correr en un segundo plano, permitiendo continuar trabajando con otros programas. Otra ventaja de esta opción es que nos brinda un botón para cancelar la ejecución del análisis en caso necesario.***

Verificación de los Resultados

1. Verifique si las formas modales y períodos de vibración son los esperados.

   - Del menú **Display** seleccione la opción **Show Mode Shape** y elija el modo en que esta interesado. Pude también seleccionar la opción **Wire Shadow** para ver al mismo tiempo la forma no-deformada de la estructura. Observe las Figuras 2-1 a la 2-4 y note que el número de modo y período de vibración correspondiente están indicados en el título de la ventana.

   Nota: *Se puede apreciar los modos de vibración subsecuentes presionando los botones + y –, próximos al botón START ANIMATION.*

2. Es útil ver el cortante en la base producido por el análisis del Espectro de Respuesta.

   - Usando el grupo BASE SHEAR que fue definido en el Ejemplo 1 observe el cortante en la base de la estructura debido al Espectro de Respuesta. Se puede apreciar que este cortante es considerablemente mayor que el debido a la condición de carga estática.

3. Se puede verificar el desplazamiento de un nudo debido al Espectro de Respuesta.

   - Del menú **Display** seleccione **Show Deformed Shape**.
     - En la plantilla **Deformed Shape** seleccione la condición de carga para el análisis espectral.
     - Presione el botón OK.
   - Haga click con el botón derecho del mouse sobre un nudo del nivel superior de la estructura para ver su correspondiente desplazamiento en la dirección global X.

4. Verifique la participación de la masa de la estructura para ver si se ha incluido el en la solución del problema el número de modos suficiente. Para ello se debe revisar el archivo de texto filename.OUT empleando un editor de textos como el WordPad de Windows.

   - Minimice el programa SAP2000.
   - Inicie el programa WordPad o cualquier otro editor de textos.
   - En WordPad abra el archivo *filename*.OUT. Donde *filename* es el nombre del archivo usado al grabar este ejemplo.
     - Busque la sección titulada MODAL PARTICIPATING MASS RATIOS como se muestra en la Figura 2-5.
     - Bajo la columna CUMULATIVE SUM encontrará que los Modos de Vibración 1 al 7 incluyen el 100% de la participación de masa. Lo cual significa que los 7 modos empleados en el análisis fueron suficientes.

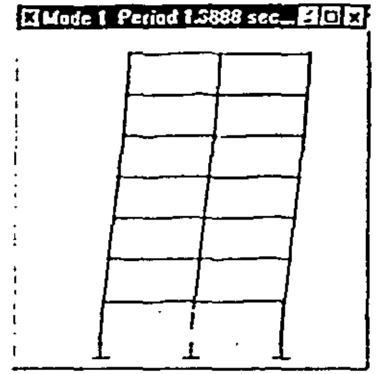


Figura 2-1 Forma Modal y Período de Vibración 1

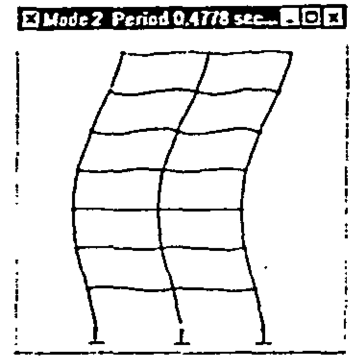


Figura 2-2 Forma Modal y Período de Vibración 2

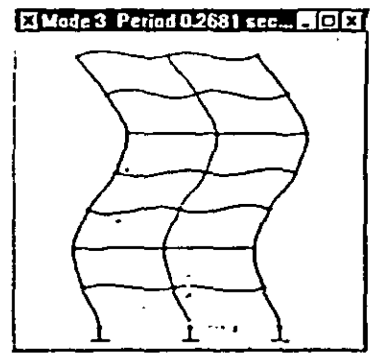


Figura 2-3 Forma Modal y Período de Vibración 3

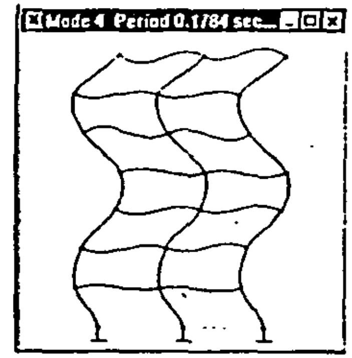


Figura 2-4 Forma Modal y Período de Vibración 4

232

MODAL PARTICIPATING MASS RATIOS

MODE	PERIOD	INDIVIDUAL MODE (PERCENT) UX	UY	UZ	CUMULATIVE SUM (PERCENT) UX	UY	UZ
1	1.388750	79.6359	0.0000	0.0000	79.6359	0.0000	0.0000
2	0.477833	11.5761	0.0000	0.0000	91.2120	0.0000	0.0000
3	0.268126	4.3023	0.0000	0.0000	95.5144	0.0000	0.0000
4	0.178439	2.1229	0.0000	0.0000	97.6373	0.0000	0.0000
5	0.133678	1.4077	0.0000	0.0000	99.0450	0.0000	0.0000
6	0.107637	0.6592	0.0000	0.0000	99.7043	0.0000	0.0000
7	0.090778	0.2957	0.0000	0.0000	100.0000	0.0000	0.0000

Figura 2-5 Bloque de Participación de la Masa del archivo de salida

Modificando la Escala del Espectro de Respuesta

Algunos códigos de diseño permiten modificar la escala del Espectro de Respuesta de manera que el cortante en la base del análisis espectral sea igual al cortante en la base del análisis empleando cargas sísmicas estáticas.

En este sentido para obtener el nuevo factor de escala para el Espectro de Respuesta tenemos que:

1. Dividir el Cortante en la Base producido por la Carga Sísmica Estática por el Cortante en la Base obtenido del Análisis Espectral, y multiplicar dicho número por 32.2 ft/seg^2 para obtener el nuevo factor de escala del Espectro de Respuesta.
2. Substituir el nuevo factor de escala en el Espectro de Respuesta.
3. Efectuar nuevamente el análisis para obtener las nuevas fuerzas en los elementos bajo la acción del Espectro de Respuesta escalado.

Comentarios Finales

Un análisis empleando el Espectro de Respuesta introduce un nivel de complejidad mayor, que requiere que el ingeniero verifique cuidadosamente los resultados, y tenga muy presente las consideraciones hechas al crear el modelo. Algunos de los aspectos a considerar durante un análisis espectral son:

- Entender completamente el comportamiento estático del modelo antes de efectuar un análisis dinámico.
- Tener un conocimiento cabal y racional de los aspectos involucrados al escalar los resultados del análisis dinámico para obtener un cortante en la base similar al que se obtiene al efectuar un análisis por cargas sísmicas estáticas.
- La ventaja de la mayor rapidez del Análisis Espectral en comparación con el Análisis de Historia en el Tiempo es en muchos casos sustancial. En el diseño, el espectro de respuesta puede incluso proveer aun mayores ventajas debido a que no se deben efectuar verificaciones para diferentes intervalos de tiempo. Sin embargo, es necesario tener presente las limitaciones del Análisis Espectral frente al mayor refinamiento que se obtiene al efectuar un Análisis de Historia en el Tiempo.

EJEMPLO 3

Pórtico Bidimensional
Análisis de Historia en el Tiempo

Descripción

Este ejemplo continua el análisis del pórtico bidimensional visto en los Ejemplos 1 y 2, añadiendo en este caso una carga de sismo especificada con un acelerograma en la base de la estructura. El registro de aceleraciones a utilizarse se muestra en la Figura 3-1 y corresponde a la componente N-S del sismo ocurrido en El Centro en 1940. Los resultados del análisis de historia en el tiempo se utilizarán para generar un Espectro de Respuesta que luego se empleará para reanalizar la estructura a manera de comparación.

Aspectos Significativos del Modelo y del SAP2000

- Respuesta de historia en el tiempo de una excitación en la base.
- Gráficas de los resultados del análisis de Historia en el Tiempo.
- Gráficas de un Espectro de Respuesta a partir de los resultados de la Historia en el Tiempo.
- Importación del Espectro de respuesta para su uso en el análisis.

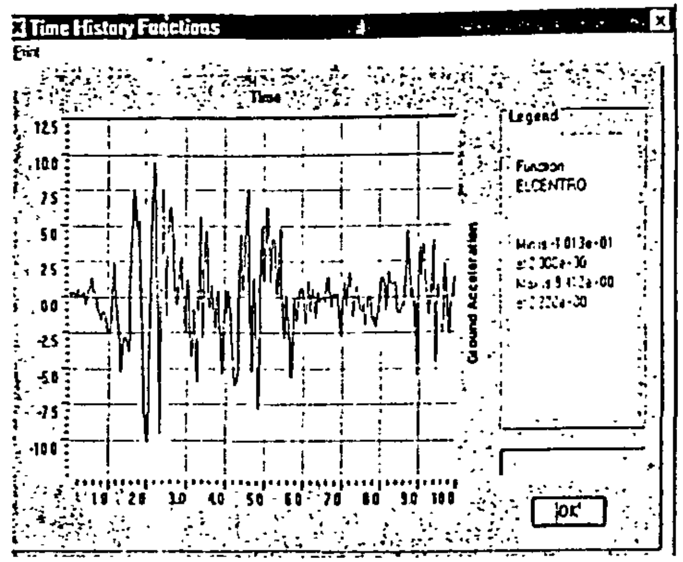


Figura 3-1 Acelerograma de entrada del Sismo de El Centro 1940 (ft/sec^2)

Definición del Concepto Historia en el Tiempo

El término Historia en el Tiempo define un registro de aceleraciones del terreno a determinados intervalos de tiempo para una excitación sísmica específica en una dirección determinada. El registro es usualmente normalizado y en consecuencia necesita multiplicarse por la aceleración de la gravedad ó por algún otro factor correspondiente.

1. Desde el menú **Define** seleccione **Time History Functions**.
2. Seleccione el botón ADD FUNCTION FROM FILE.
   - Presione el botón **Open File** y seleccione el archivo ELCENTRO ubicado en el subdirectorio EXAMPLES del directorio SAP2000.
   - Cambie el nombre de la función a ELCENTRO para que sea fácil de reconocer.
   - El formato de este archivo es de tres pares de columnas de datos por renglón. El primer par de datos la primera columna de cada par es el tiempo y la segunda es la aceleración.
     - Ingrese 3 Puntos por Línea.

- Seleccione la opción **Time and Function Values** (en archivo de datos).
- Presione el botón OK.

- Presione el botón OK para aceptar los datos ingresados.

3. Del menú **Define** seleccione la opción **Time History Cases** para definir los parámetros específicos para el análisis de Historia en el Tiempo de nuestro modelo.
   - Seleccione el botón ADD NEW HISTORY.
   - Para adicionar amortiguamiento al sistema presione el botón MODIFY/SHOW MODAL DAMPING, ingrese 0.05 (5%) para todos los modos y presione OK.
   - Ingrese en el ítem **Number of Output Time Steps** el valor 500.
   - Ingrese 0.02 (sec) en el ítem **Output Time Step Sizes.** Estos parámetros nos darán 10 segundos del registro sísmico para el análisis de historia en el tiempo.
   - De la Lista **Analysis Type** seleccione la opción **Linear**.
   - En el área **Load Assignment** :
     - Seleccione ACC DIR1 para el parámetro **Load**
     - Seleccione la opción ELCENTRO para el parámetro **Function**
     - Para el parámetro **Scale Function** ingrese la aceleración de la gravedad que es de 386.4 in/sec^2 si trabaja en Kip-in o 32.2 ft/sec^2 si trabaja en Kip-ft.
     - Para los parámetros **Arrival Time** y **Angle** asigne valor cero.
     - Presione el botón **ADD** para agregar esta carga al modelo, y presione el botón OK para aceptar los datos que acaba de ingresar.
   - Presione el botón OK en ambas plantillas para aceptar las adiciones al modelo.

De esta manera hemos ingresado toda la información que necesitamos para efectuar el Análisis de Historia en el Tiempo.

Nota: *Por lo general es una buena idea correr el modelo cada vez que se hace un cambio ó adición importante al modelo. Esto permite detectar errores y ahorra tiempo en el diseño final.*

Efectuando el Análisis

1. Grabe el modelo.
2. Ajuste los parámetros para el análisis seleccionando **Analyze** del menú **Set Options**.
   - Verifique que los parámetros en **Dynamic Analysis** son los mismos del Ejemplo 2.
3. Seleccione en el menú **Analyze** la opción **Run** para analizar la estructura.

Usando los Resultados

Verificación de los Resultados

Una vez que ha corrido el modelo se deben verificar que los resultados obtenidos sean del orden y magnitud a lo esperado.

1. Verifique el Cortante en la Base producido en el Análisis de Historia en el Tiempo.
   - Desde el menú **Display** seleccione la opción **Show Time History Traces.**
     - De la plantilla **Time History Display Definition** presione el botón DEFINE FUNCTIONS.
     - En la plantilla **Time History Functions** seleccione Add Base Functions y marque solamente la opción Base Shear X.
     - Presione OK para regresar a la plantilla **Time History Display Definition.**
     - Adicione la función Base Shear X a la lista **Plot Functions.**
     - Presione el botón DISPLAY para ver una gráfica del cortante en la base en la dirección global X como función del tiempo. Vea la Figura 3-2.

   Nota: *También se puede generar la gráfica del cortante en la base seleccionando Add Group Summation Forces en lugar de Add Base Functions y seleccionando el grupo de elementos BASE SHEAR definido en el Ejemplo 1.*

2. También se puede verificar el desplazamiento de un nudo ante una excitación de Historia en el Tiempo, para ello:
   - Escoja un nudo y desde el menú **Display** seleccione la opción **Show Time History Traces.**

- Presione el botón DEFINE FUNCTIONS de la plantilla **Time History Functions** Seleccione el nombre del nudo de la lista y presione el botón MODIFY/SHOW TH FUNCTION.
- En la plantilla Time History Joint Function seleccione DISPL para el parámetro **Vector Type** y UX para el parámetro **Vector Direction**.
- Presione el botón OK para aceptar los cambios.
- Presione los botones OK para regresar a la plantilla **Time History Display Definition**.
- Añada el nudo de la lista **List of Functions** a la lista **Plot Functions** y remueva de esta última la función Base Shear X.
- Presione el botón DISPLAY para ver el desplazamiento del nudo con respecto al tiempo. Vea Figura 3-3.

• Se puede también definir una función nodal directamente en la plantilla **Time History Display Definition** sin haber seleccionado previamente el nudo.

- En la plantilla Time History Display Definition presione el botón DEFINE FUNCTIONS y en la plantilla Time History Functions seleccione la opción **Add Joint Disps/Forces**.
- En la plantilla Time History Joint Function ingrese el nombre del nudo (ID).
- Seleccione **Vector Type** y **Vector Direction**.
- Presione los botones OK para regresar a la plantilla **Time History Display Definition** en donde encontrará la nueva función nodal en el recuadro **List of Functions**.

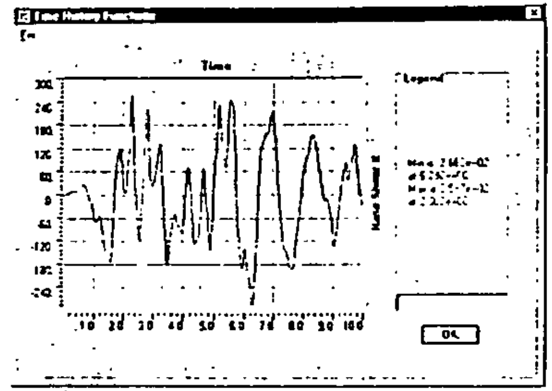


Figura 3-2 Cortante en la Base – Kips

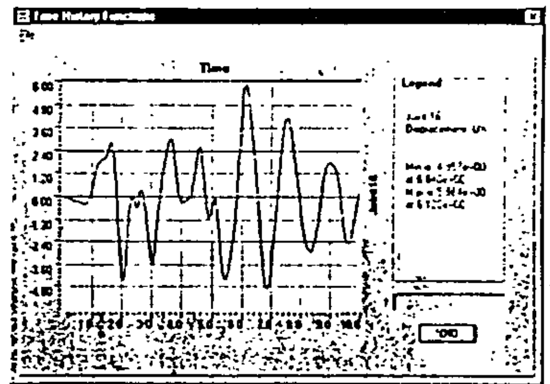


Figura 3-3 Historia en el Tiempo del Desplazamiento del Nivel Superior – In

Creación de un Espectro de Respuesta

Lo primero que se tiene que hacer en este punto es crear un Espectro de Respuesta a partir de los resultados del Análisis de Historia en el Tiempo. Los datos generados de esta forma deben imprimirse en archivo de texto para poder editarse en un formato que pueda ser leído por el SAP2000.

Gráficas del Espectro de Respuesta

1. Seleccione un nudo en la base de la estructura.
2. Del menú **Display** seleccione la opción **Show Response Spectrum Curves**. Esta opción aparece solamente cuando se ha seleccionado un nudo.
3. En la plantilla **Response Spectrum Generation** encontrará el nombre del nudo que fue seleccionado.
   - Bajo el ítem **Define** asigne el valor X para el parámetro **Vector Direction**.
   - Bajo el ítem **Axes** seleccione **Period** para el parámetro **Abscissa** y **PSA** (Seudoaceleración Espectral) para el parámetro **Ordinate**.
   - Bajo el ítem **Options** seleccione **Arithmetic** tanto para **Abscissa** como para **Ordinate**. Para el parámetro **Ordinate** asigne el factor de escala 1/g ($g=32.2$ ft/sec^2) es decir, 0.03106 sec^2/ft si las unidades en que se trabaja son kip-ft.

     Nota: *El factor de escala es usado para normalizar el Espectro de Respuesta. El registro para el análisis de historia en el tiempo que se utilizó para generar el Espectro de Respuesta estaba normalizado a la aceleración de la gravedad (g) por lo que necesitamos dividir el espectro por la misma cantidad para obtener valores normalizados.*
   - Bajo el ítem **Period** seleccione para las frecuencias los parámetros **Default** y **Structural**. Estos parámetros son usados en la generación del Espectro de Respuesta, las frecuencias del tipo **Default** son una serie de frecuencias predeterminadas que son típicamente de interés en las estructuras; las frecuencias del tipo **Structural** son las frecuencias naturales de la estructura.
   - Bajo el ítem **Damping** mantenga el valor del amortiguamiento de 0.05 para el parámetro **Damping Value**. Como se ha asumido que la estructura tiene 5% de amortiguamiento no será necesario emplear otros valores para el amortiguamiento del sistema.
   - Presione el botón DISPLAY cuando se halla terminado.
4. Ahora Ud. podrá apreciar la gráfica del Espectro de Respuesta para el sismo de El Centro para un amortiguamiento de 5%. Ver Figura 3-4.

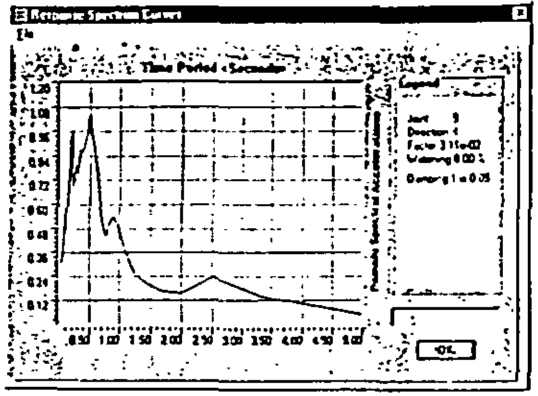


Figura 3-4 Espectro de Respuesta del Análisis de Historia en el Tiempo

5. En la plantilla **Response Spectrum Curves** seleccione la opción **Print Tables to File**. Esta opción generará un archivo que tiene dos columnas de datos. La primera de ellas es el período, y la segunda es su correspondiente Seudoacelaración (PSA).
   - Grabe el archivo con el nombre RS-ELCEN.TXT

Edición de Tablas.

El siguiente paso consiste en hacer algunas pequeñas modificaciones al archivo de texto RS-ELCEN.TXT para que tenga un formato que pueda ser leído por el SAP2000. Esto se debe a que cuando el archivo original es creado se le agrega información aclaratoria que permite al usuario interpretar y entender fácilmente su contenido. Esta información extra debe removerse.

1. Con un editor de textos como WORDPAD o NOTEPAD abra el archivo RS-ELCEN.TXT.
   - Seleccione todo el texto que se muestra resaltado en la **Figura 3-5** y remuévalo.
   - Grabe el archivo RS-ELCEN.TXT como un archivo de texto con el mismo nombre.
2. Ahora que el archivo tiene tan solo las columnas con los períodos y sus correspondientes Seudoacceleraciones, podrá ser leído directamente por el SAP2000

```
SAP2000 v6.06 File: TUTOPIAL1  Kip-ft Units  PAGE 1
May 19, 1997 17:26

S P E C T R U M   D A T A

Joint        9
Direction    XT
Factor       0.03
Widening     0 %

Pseudo Spectral Accelerations vs Time Period <Seconds>

                 DAMPING
                  0.0500
      0.0303  3.0656E-01
      0.0357  3.2657E-01
      0.0400  3.1139E-01
      0.0455  3.2217E-01
      0.0500  3.2205E-01
      0.0556  3.3093E-01
      0.0625  3.4265E-01
      0.0667  3.7431E-01
```

Figura 3-5 Archivo de salida con el Espectro de Respuesta generado

Lectura de los Datos del Espectro

Ahora que se tienen datos en un formato que el SAP2000 puede leer, necesitamos indicar al programa la ubicación del archivo así como la forma en que éste contiene la información.

1. Si el modelo esta protegido presione el botón **Lock/ Unlock Model** en la barra de herramientas. Al hacer esto se remueve la protección sobre el modelo y nos permitirá realizar las modificaciones.
2. Del menú Define seleccione **Response Spectrum Functions**.
3. En la plantilla Response Spectrum Functions presione el botón **Add Function from File.**
   - Asigne al espectro el nombre RSELCEN
   - Presione el botón **Open File** y seleccione el archivo RS-ELCEN.TXT en **Pick File.**
   - Mantenga el parámetro **Number Of Points Per Line** en el valor 1 puesto que únicamente hay un par de datos por renglón para definir el Espectro de Respuesta.
   - Seleccione la opción **Period and Acceleration Values.**
   - Presione el botón OK para cerrar las plantillas.

4. Del menú Define seleccione **Response Spectrum Cases.**
5. En la plantilla Response Spectrum presione el botón ADD NEW SPECTRA.
   - Asigne al parámetro **Modal Damping** el valor 0.05.
   - En el área **Input Response Spectra** seleccione RSELCEN para la dirección U1 y asígnele el factor de escala 32.2 ft/sec^2.
   - Los demás valores por defecto son aceptables.
   - Presione el botón OK para cerrar las plantillas.

Efectuando el Análisis

Una vez que se han hecho las modificaciones es tiempo de correr el modelo y revisar los resultados obtenidos.

1. Grabe el modelo.
2. Seleccione **Run Minimized** del menú **Analyze** para analizar la estructura.

Revisando los Resultados

Lo primero que debe hacerse es la revisión de la máxima deflexión en la parte superior de la estructura así como el cortante en la base tanto para el análisis espectral como para el análisis de Historia en el Tiempo. La comparación de estos resultados nos permitirá ver que tan bién funciona la metodología previamente descrita. Al final de esta sección encontrará los resultados del análisis por cargas sísmicas estáticas, del análisis espectral y del análisis de historia en el tiempo..

Deflexiones de acuerdo al Análisis Espectral

1. Del menú Display seleccione Display Deformed Shape.
   - En la plantilla Deformed Shape seleccione la condición de carga para el análisis espectral.
   - Presione el botón OK.
2. Haga click con el botón derecho del mouse sobre un nudo ubicado en el nivel superior para ver su desplazamiento en la dirección del eje global X.

Cortante en la Base en el Análisis Espectral

Usando el grupo BASE SHEAR que fue creado en el Ejemplo 1, observe los valores del cortante en la base de la estructura debido al Análisis Espectral.

Deflexión y Cortante en la Base en el Análisis de Historia en el Tiempo

1. Usando el método descrito en la primera parte de esta guía grafique la deflexión en el nivel superior de la estructura.
2. Ahora remueva ese nudo de la lista Plot Functions y en su lugar grafique la función Base Shear X ubicada en List of Functions de la plantilla History Display Definitions.

	Carga Lateral Estática	Espectro de Respuesta	Historia en el Tiempo
Max Deflexión	1.65 in	5.5 in	5.8 in
Max Cortante en la Base	72.5 Kips	302 Kips	325 Kips

Tabla 3-1 Comparación de los Resultados del Análisis de Cargas Laterales

Comentarios Finales

Como habrá podido apreciar el Análisis de Historia en el Tiempo involucra un mayor tiempo de cómputo que el Análisis con Espectro de Respuesta. Debe observarse sin embargo que ambos métodos de análisis brindan resultados similares. En este sentido es sumamente importante que el ingeniero entienda las ventajas y limitaciones de cada método para poder utilizarlos de la manera más adecuada y efectiva.

EJEMPLO 4

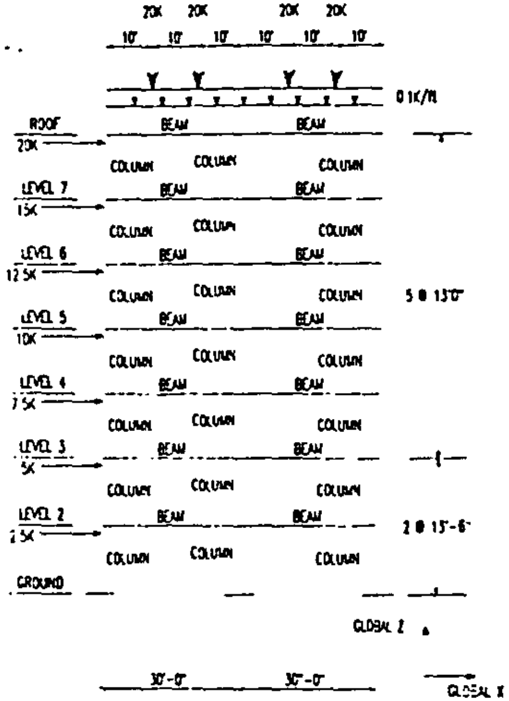
Diseño en Acero de un Pórtico Bidimensional

Descripción

Este ejemplo es un introducción al uso de las poderosas herramientas que posee el SAP2000 para el diseño de una estructura una vez concluido su análisis estructural. Se dará énfasis en esta parte a módulos de diseño en acero empleando como ejemplo la estructura analizada en el Ejemplo 1.

Aspectos Importante del Modelo y del SAP2000

- Creación de zonas rígidas en los elementos.
- Selección Automática de grupos
- Cambio de propiedades en elementos
- Designación de elementos por grupos
- Inclusión del efecto P-Delta en el análisis
- Visualización de los resultados
- Auto Selección de secciones



ALL COLUMNS ARE W14'S
ALL BEAMS ARE W24'S
TYPICAL STORY MASS = 0.49 kip-sec-sec/in
MODULUS OF ELASTICITY = 29500 ksi
STEEL STRENGTH (F_y) = 36ksi

Figura 4-1 Pórtico Bidimensional a diseñarse

Creando el Modelo en el SAP2000

Es posible emplear el modelo desarrollado en el Ejemplo 1 efectuando pequeñas modificaciones.

Materiales

Lo primero que se debe hacer es especificar las propiedades de los materiales.

1. Verifique que las unidades estén en Kip-in.
2. Entre al menú Define y seleccione la opción **Materials**.
3. Elija STEEL para el parámetro Material y presione el botón MODIFY/SHOW MATERIAL
4. Especifique el Esfuerzo de Fluencia ajustando el parámetro Steel Yield Stress fy en 36 Ksi.
5. Especifique el Módulo de Elasticidad ajustando el parámetro Modulus of Elasticity E en 29,500 Ksi.
6. Presione los botones OK para aceptar los cambios y cerrar las plantillas.

Cargas

1. En el ejemplo 1 se asignó el peso propio de la estructura así como una serie de cargas concentradas y distribuidas a la condición de carga DEAD. (Vea Figura 4-2 para la nueva lista Static Loads Case). En este ejemplo asignaremos una condición de carga para la carga viva y otra para el peso propio de los elementos. Es una buena práctica incluir una condición de carga para el peso propio de la estructura con el fin de seguir de cerca el proceso de optimización estructural. Las cargas son separadas en carga muerta, carga viva y carga transversal de sismo de tal manera que el SAP2000 pueda generar automáticamente las combinaciones de carga.
   - Para el caso de carga DEAD especifique el valor del multiplicador **Self Weight Multiplier** en cero.
   - Agregar una condición de carga para el peso propio y nómbrela SELF, asígnele el tipo DEAD como parámetro a Type y ajuste el multiplicador Self Weight Multiplier en 1.
   - Agregar otra condición de carga estática llamada LIVE y asígnele el tipo LIVE como parámetro a Type.
2. Añada las mismas cargas correspondientes a la carga DEAD a la condición de carga LIVE. Esto significa que cada viga de la estructura tiene cargas idénticas para carga muerta y viva. (Puede revisar el ejemplo 1 para ver las instrucciones de como ingresar las una cargas).

241

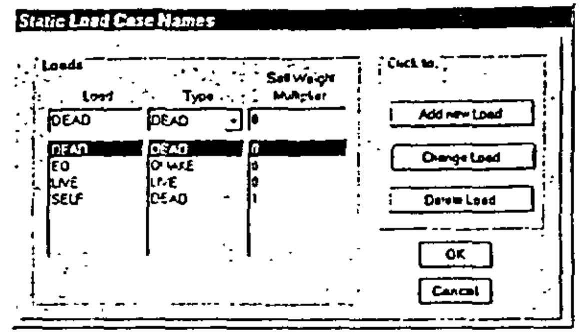


Figura 4-2 - Condiciones de Carga Estática

Definiendo un Grupo de Auto-Selección

La opción de Auto-Selección del SAP2000 es una forma muy efectiva de diseñar estructuras. Al definir un grupo de secciones denominado grupo de Auto-Selección, el programa puede diseñar cada elemento del pórtico escogiendo de entre las secciones especificadas en ese grupo. Por ejemplo se puede definir un grupo de Auto-Selección denominado COLUMNS con únicamente perfiles W14, y otro grupo llamado BEAM con perfiles W24. El programa de esta manera diseñará las secciones tipo COLUMNS empleando solamente secciones W14 y las secciones tipo BEAM empleando únicamente secciones tipo W24.

Lo primero que debemos hacer es definir un grupo de Auto-Selección que incluya únicamente secciones tipo columna. Esencialmente lo que estamos haciendo en esta etapa es darle al programa una lista de secciones de entre las cuales puede elegir al momento de diseñar los elementos del pórtico. El programa por su parte seleccionará la sección mas eficiente de entre ese grupo.

Una vez se haya concluido el diseño preliminar es momento de refinarlo, para ello los grupos BEAM y COLUMN serán reemplazados por secciones optimizadas elegidas de entre el grupo de Auto-Selección. Este proceso asignará a los elementos secciones que serán empleadas tanto en el análisis como en el diseño, esto hará mucho más fácil el cambio de secciones que necesiten ser modificadas.

Nota: ***La opción de Auto-Selección funciona solo en pórticos de acero.***

1. Desde el menú Define y seleccione la opción **Frame Sections**.
2. Importe a la plantilla **Frame Sections** todas las secciones de acero comprendidas entre W14x61 y W14x283.
   - Seleccione en la caja de diálogo la opción IMPORT I/WIDE FLANGE.

- Busque y seleccione la sección W14x283.
- Manteniendo presionada la tecla SHIFT haga click con el botón izquierdo del mouse sobre W14x61 y presione el botón OK. Esta operación permite seleccionar todas las secciones entre la sección W14x283 y la sección W14x61.

3. Borre de plantilla Frame Sections cualquier sección que pudiera estar duplicada.

    Recuerde: *No es posible borrar una sección que este en uso. De esta manera el programa asegura que todos los elementos tienen asignada secciones existentes.*

4. Desde la plantilla Frame Section añada una sección Auto Select. Esta sección se ubicará en la parte inferior de la lista Add.
    - Cambie el nombre en Auto Section Name a COLUMN.
    - De la lista Auto Selections elija y remueva usando el botón Remove todas las secciones exceptuando los perfiles W14. Esto significa que todos los elementos que tengan una sección tipo COLUMN serán diseñados empleando alguno de los perfiles W14 de entre la lista Auto Selections (Vea Figura 4-3).

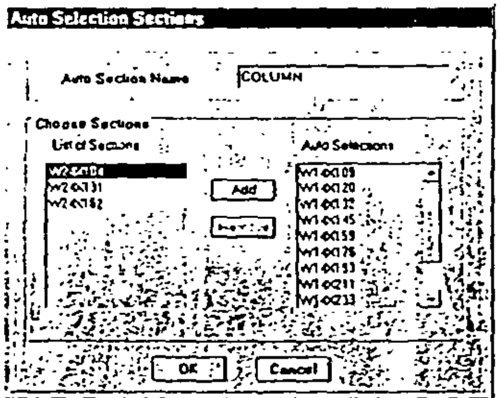


Figura 4-3 Definiendo el grupo de Auto-Selección COLUMN

5. Siguiendo las instrucciones dadas en los pasos 2 al 4:
    - Importe todas las secciones entre la W24x55 y la W24x162.
    - Asigne un grupo de Auto-Selección denominado BEAM con perfiles W24 únicamente.

6. Finalmente, selecciones todos los elementos verticales del pórtico y asígnele la sección tipo COLUMN. Luego seleccione todos los elementos horizontales y asígnele la sección tipo BEAM. (Vea el ejemplo 1 para las instrucciones de como asignar secciones a los elementos del pórtico).

    Nota: *Se puede por supuesto seleccionar una sección específica tanto para el diseño como para el análisis en lugar de emplear la opción de Auto-Selección. Para ello simplemente se necesita asignar a los elementos del pórtico una sección de acero adecuada y diseñarla de acuerdo a lo descrito en el ejemplo 1. Esta sección puede ser ya sea una sección definida por el usuario o bien una sección elegida de entre las secciones predeterminadas.*

Efectuando el Análisis

Una vez que se han ingresado los datos es tiempo de correr el modelo y revisar los resultados.

1. Grabe el modelo.
2. Asigne los parámetros para el diseño entrando al menú **Analyze** y seleccionando la opción Set Options.
    - En la plantilla **Analysis Options** seleccione el tipo de análisis **Plane Frame** para reducir el tamaño de la solución y en consecuencia reducir el tiempo de análisis.
    - Marque la opción **Include P-Delta**.
    - Presione el botón SET P-DELTA PARAMETERS para ajustar los parámetros del análisis.
        - Asigne a **Maximum Iterations** el valor 10.
        - Incluya las condiciones de carga muerta DEAD y SELF, en la combinación P-Delta, ambas con factores de carga igual a 1.
        - Incluya la condición de carga LIVE con un factor de carga 1.

        Nota: *Los factores de carga a emplearse deben ser los correspondientes a las combinaciones de carga que se usen en el diseño de la estructura y que produzcan los máximos efectos en la misma. Deben incluirse además el efecto de las cargas laterales.*

        - El resto de valores por defecto es aceptable.
        - Presione los botones OK para aceptar los cambios y cerrar las plantillas.
3. Entre al menú Analyze y seleccione la opción Run para analizar la estructura.

Nota: *Debido a que hemos asignado grupos de secciones y no secciones específicas para nuestro diseño, el SAP2000 elegirá las propiedades de las secciones más convenientes para generar la matriz de rigidez y efectuar el análisis estructural. Una vez que se ha efectuado el primer análisis y diseño, se puede instruir al programa para realizar el análisis con las propiedades de las secciones designadas.*

Diseño de Secciones

Una vez que se ha efectuado el análisis y revisado sus resultados, se podrán establecer los parámetros necesarios para el diseño en acero de la estructura.

Selección del Código de Diseño

La información resultante del análisis es empleada para efectuar una verificación de las secciones empleando un código de diseño.

1. Desde el menú Options seleccione **Preferences**.
2. En la plantilla Preferences, bajo el ítem **Steel** seleccione el código de diseño que desee emplear. En este caso se empleará el código AISC-ASD-89.
   - Emplee el mismo archivo **Section Properties file** que fue usado para importar las secciones de acero.

Combinaciones de Carga y Diseño

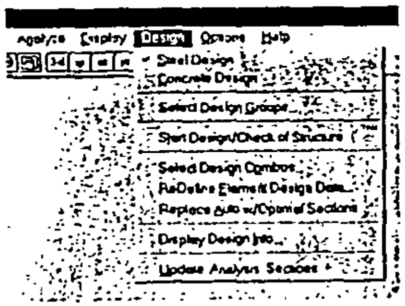


Figura 4-5 Opciones en el Menú de Diseño

Una vez que se ha seleccionado el código a utilizar en el diseño de los elementos se deben verificar las combinaciones de carga a emplearse

1. Primeramente, en el menú **Design** asegúrese que exista la contraseña en el ítem **Steel Design**. Esto le indica al SAP2000 que debe efectuarse un diseño de secciones en acero.
2. En el menú **Design** elija la opción **Select Design Combos**.
   - Revise las combinaciones de carga generadas bajo la lista **Design Combos**, seleccionando las combinaciones y presionando el botón SHOW.
   - Si existen otras combinaciones de carga que desee incluir en el diseño, puede agregarlas usando para ello la opción Select Design Combos del menú Define.
3. Inicie el Diseño/Revisión de la estructura desde el menú Design seleccionando el ítem Start Design/Check of Structure.
   - Cada uno de los elementos será diseñado empleando la sección más eficiente de entre las correspondiente a su grupo de Auto-Selección
   - El SAP2000 mostrará automáticamente el porcentaje del nivel de esfuerzos existente en cada elemento con relación al máximo esfuerzo admisible.
   - Para mayor comodidad el programa asignará colores a cada uno de los elementos, los cuales muestran el nivel de esfuerzo presente en cada miembro usando una escala gráfica de colores/esfuerzos ubicada en la parte inferior de la ventana.

   Nota: *Si desea verificar el diseño de un número determinado de elementos, podrá realizarlo simplemente con seleccionarlos y ejecutar la opción Start Design/Check of Structure.*

Revisión de los Resultados y Rediseño

Una vez efectuado el Diseño/Revisión es tiempo de verificar si los resultados son correctos. SAP2000 brinda al usuario una serie de herramientas para ello.

1. Haga click con el botón derecho del mouse sobre cualquier elemento para ver los resultados de su diseño. El elemento que haya sido seleccionado parpadeará para su fácil identificación.
2. En la plantilla **Steel Stress Check Information** encontrará una lista de las combinaciones de carga empleadas para verificar la sección en varios puntos a lo largo del elemento. (Vea Figura 4-5)
   - Una de las combinaciones de carga estará resaltada cuando abra esta plantilla. Esta es la combinación que controla el diseño del elemento.
   - Junto a cada combinación de carga hay un indicador de la ubicación a lo largo del elemento donde fue efectuada la verificación, seguida por la relación de esfuerzo para la interacción de momento y cortante.

243

Sugerencia : *Se puede cambiar el número de puntos a lo largo del elemento en los cuales las fuerzas de sección son reportadas. Para ello seleccione los elementos y desde el menú Assign elija la opción Frame Output Segments para cambiar el número de segmentos. Es necesario ejecutar nuevamente el análisis del modelo para obtener los resultados.*

3. Seleccionando cualesquiera de las combinaciones de carga y presionando el botón **Details** se mostrarán los resultados del análisis para ese elemento así como las ecuaciones que gobiernan su diseño de acuerdo al código empleado. (Ver Figura 4-6)

4. Al presionar el botón **ReDesign** se presentará la plantilla **Element Overwrite Assignments**. En esta plantilla se puede elegir de entre varias opciones:

   Nota: *Si se efectúan cambios en la plantilla Element Overwrite Assignments empleando el botón ReDesign, será necesario presionar el botón Refresh Window de la barra de herramientas para ver los resultados del diseño actualizados en la ventana activa .*

   - Seleccionar otra sección para ver el cambio en los esfuerzos en el elemento.

     Nota: *En el modo Auto-Selección, esta sección puede emplearse para ensamblar una nueva matriz de rigidez si se elige la opción "Update Analysis Sections". Esto último se llevará acabo una vez que se ejecute nuevamente el Diseño/Revisión.*

   - Clasificar los elementos por tipo **Moment Resisting Element** o **Brace**.
   - Sobrescribir los factores de diseño tales como longitud efectiva y la relación de longitud no arriostrada.
   - Elija la opción **Overwrite Allowable Stresses** para sobrescribir los esfuerzos admisibles empleados en el diseño de la sección.
   - Cuando halla terminado de modificar los parámetros de diseño presione el botón **OK**.

     Nota: ***Al cambiar la información de la plantilla ReDesign, el SAP2000 automáticamente recalculará los esfuerzos de diseño de acuerdo a la nueva información y actualizará la información en la plantilla Steel Stress Check Information. Para más instrucciones sobre como actualizar las secciones para el análisis refiérase a la sección "Re-Analizando".***

5. Para usar la sección elegida en el Re-Diseño en el siguiente análisis estructural, es necesario entrar al menú **Design** y seleccionar la opción **Update Analysis Sections**. Esta opción reemplaza las secciones empleadas inicialmente para formar la matriz de rigidez de la estructura, por las nuevas secciones dándonos mayor precisión en los cálculos.

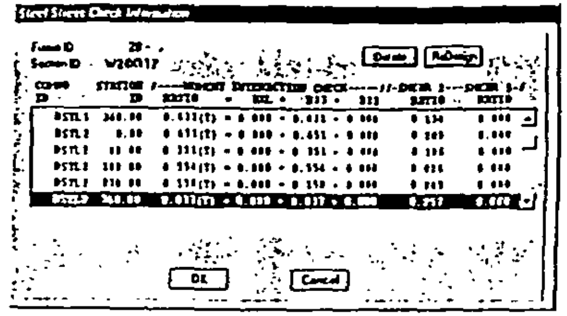


Figura 4-5 Verificación de Esfuerzos en las secciones de Acero para las Combinaciones de Carga Especificadas

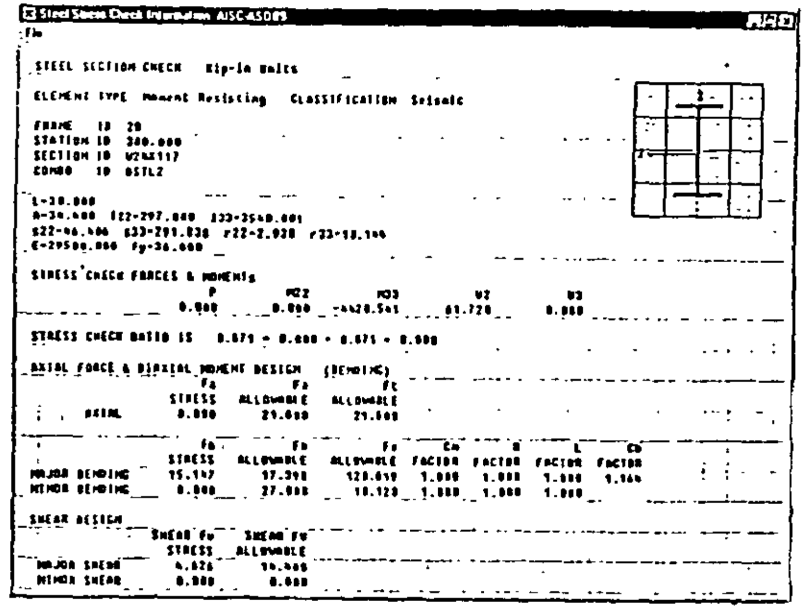


Figura 4-6 Información detallada del diseño en acero de un elemento tipo viga

6. También se pueden observar los resultado del diseño de manera gráfica en la pantalla. Para ello ingrese al menú **Design** y seleccione la opción **Display Design Info** Los resultados se mostrarán en forma gráfica en la parte inferior derecha de los elementos del pórtico.

   Nota: ***Las secciones empleadas en el análisis estructural son mostradas en la parte superior izquierda de cada elemento. Por otra parte, toda la información correspondiente al diseño de los elementos es mostrada en la parte inferior derecha de los mismos.***

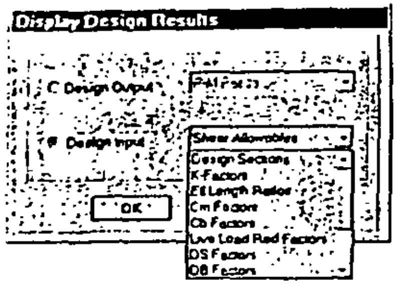


Figura 4-7 Plantilla de salida resultados

7. Se pueden imprimir los resultados del diseño ingresando al menú **File** y seleccionando la opción **Print Design Tables**. Para imprimir los resultados de un número limitado de elementos, primeramente selecciónelos y enseguida efectúe la opción **Print Design Tables**.

Edición de las Propiedades de la Sección

En la Figura 4-1 podrá observar que las vigas del pórtico tienen cargas concentradas ubicadas a los tercio del claro. Si asumimos que dichas cargas son transmitidas por otros miembros, entonces podemos considerar que las vigas tienen soporte lateral en los puntos de carga. El modelo tal como ha sido creado no toma en cuenta este hecho, por lo que las vigas están sobredimensionadas. Para diseñar las vigas más eficientemente se deben editar sus propiedades de diseño.

1. Seleccione todas las vigas de la estructura. Para ello puede usar el modo de selección **Intersecting Line Select Mode**.
2. Desde el menú **Design** escoja la opción **ReDefine Element Design Data**.
   - En la plantilla **Element Overwrite Assignments** seleccione el ítem **Unbraced Length Ratio, L22** y asígnele el valor 0.33. Esto introduce un soporte lateral a las vigas a cada 1/3 del claro (arriostramiento contra el pandeo en el eje local 1-2), en

   lugar de la opción por defecto que corresponde a elementos arriostrados solo en los extremos. (Vea Figura 4-8)
   - Presione el botón **OK** luego de ingresar la nueva información. El SAP2000 automáticamente efectuará el Diseño/Revisión del modelo actualizado.
3. Podrá apreciar ahora que las dimensiones de las secciones de las vigas son menores.

   Sugerencia: ***Se puede conocer el peso de la estructura obtenlendo la suma de cargas nodales de los elementos del grupo BASE SHEAR. Esta es una manera rápida de ver si los cambios efectuados producen una estructura más eficiente. También se puede obtener el peso total de la estructura empleando el archivo filename.EKO.***

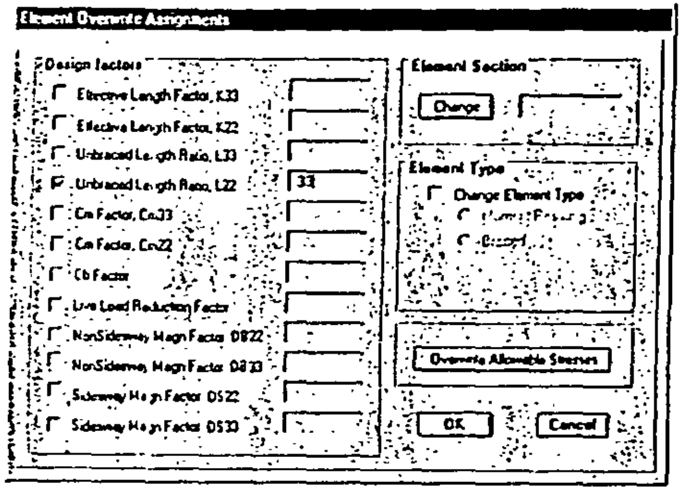


Figura 4-8 Plantilla de sobre-escritura de parámetros de diseño de elementos tipo viga

Re-Analizando

El primer análisis empleó propiedades de sección aproximadas para genera la matriz de rigidez de la estructura. Es por ello que el modelo se debe reanalizar a manera de un proceso iterativo para asegurarnos que el análisis toma en cuenta las propiedades y secciones actualizadas de los elementos.

245

1. Una vez que se halla terminado de modificar las secciones estructurales que se van a emplear, ingrese al menú **Design** y elija la opción **Update Analysis Sections**. Luego efectúe nuevamente el análisis empleando las últimas propiedades de sección.
2. Lleve acabo nuevamente el diseño de los elementos para ver si hay cambios.
3. Una vez que se encuentre satisfecho con las secciones elegidas, ingrese al menú **Design** y elija la opción **Replace Auto w/ Optimal Sections**. Esta opción asigna las secciones definitivas ya sean las óptimas o bien aquellas seleccionadas por el usuario tanto para el análisis como para el diseño, y reemplaza las propiedades de sección preliminares tomadas de los grupos de auto selección BEAM y COLUMN.

Diseño de acuerdo al LRFD

La metodología empleada por el LRFD es esencialmente la misma que usa el ASD. Sin embargo las combinaciones de carga así como la ecuaciones de verificación de los elementos son efectuadas empleando el código LRFD, por lo que los resultados e información resultante es distinta. Para efectuar el Diseño/Revisión de acuerdo al código LRFD, es necesario cambiar algunos parámetros de entrada.

1. Ingrese los nuevos factores de carga para el análisis P-Delta.
2. Seleccione de la plantilla **Preferences** el código de diseño en acero AISC-LRFD93.
3. Rediseñe las secciones de acero.

Opciones Avanzadas

Definición de Grupos de Elementos para el Diseño

Algunas veces puede encontrará útil esta opción al diseñar elementos en estructuras aporticadas. Esta opción permite diseñar todos los elementos de un grupo usando únicamente una sección. La ventaja de este método de diseño es que reduce el número de secciones diferentes a emplearse. Por ejemplo se pueden agrupar las columnas o las vigas de dos o tres pisos del pórtico dentro de un mismo grupo de diseño, permitiéndonos usar una sola sección para dicho conjunto de elementos.

1. **Reasigne las secciones del tipo Auto-Selección a los elementos del pórtico.**
2. Agrupe los elementos del 3er piso hacia abajo en un grupo denominado BOTTOM.
3. Agrupe los elementos entre los pisos 3 y 5 en un grupo llamado MIDDLE.
4. Asigne los elementos restante al grupo TOP.

5. Efectúe nuevamente el análisis del modelo.
6. En el menú **Design** seleccione la opción **Select Design Group**. En esta parte se le indica al programa que se diseñe un grupo de elementos empleando la sección más ligera que satisfaga los requerimientos y esfuerzos admisibles en todos los elementos.
   - Incluya en la lista **Design Groups** los grupos de elementos TOP, MIDDLE y BOTTOM. Al hacer esto indicamos que estos grupos serán diseñados con la sección más eficiente de las secciones del grupo Auto-Selección.

     Nota: *Si no hay grupos en la lista "Design Group list", cada uno de los elementos de la estructura serán diseñados individualmente.*
   - Cuando presione el botón **OK**, El SAP2000 automáticamente diseñará las secciones de acero y mostrará los resultados en la ventana activa
7. Compare los resultados del diseño anteriormente efectuado con el diseño por grupos para ver como afecta este hecho en las secciones seleccionadas.

Zonas Rígidas

La estructura que hemos venido estudiando, ha sido analizada y diseñada considerando que los elementos se extienden completamente de nudo a nudo, sin tomar en cuenta las dimensiones propias de las secciones transversales de los elementos. Si bien es cierto que ésta no es una mala consideración, el SAP2000 permite efectuar análisis aun más precisos mediante la introducción de zonas rígidas en el modelo. Las zonas rígidas definen una región en la conexión entre viga y columna, en la cual los elementos no sufren deformaciones por flexión. Se genera así esencialmente una zona rígida en la conexión. Esta área puede ser tan grande como el usuario especifique, pero usualmente se considera igual al peralte del miembro (o una fracción del mismo) al que se esta llegando en ese nudo.

1. Seleccione todos los elementos del pórtico.
2. Del menú **Assign** seleccione la opción **Frame... End Offsets**.
   - En la plantilla **End Offset** seleccione la opción **Update Lengths From Current Connectivity**. Esta opción hace que el programa calcule automáticamente las dimensiones de las zonas rígidas a considerarse en cada nudo.
   - Ingrese el valor 1 para **Rigid Zone Factor**. Esto significa que el 100% de la "longitud potencial" de zona rígida deberá considerarse en el análisis.
   - Presione el botón **OK**.
3. Si se especifica la opción **Element Shrink** de la plantilla **Set Elements**, y se observa en la pantalla activa, se podrá apreciar una serie de líneas en cada nudo que indican la asignación de zonas rígidas en los elementos.

245

Recuerde : ***Se necesita restablecer la opción End Offsets cada vez que las secciones de los elementos sean modificadas.***

Nota: ***Los momentos y cortantes en las vigas y columnas van a ser ligeramente diferentes en aquellos casos en los que no se toman en cuenta las zonas rígidas. Esto se debe a que la introducción de las mismas reduce la longitud flexible de los elementos.***

Comentarios Finales

Las herramientas de diseño del SAP2000 son muy útiles en el diseño de estructuras aporticadas. Sin embargo hay algunos puntos que se deben tener presentes:

1. Asegúrese que toda la información de diseño sea correcta. Los valores por defecto que usa el programa no son necesariamente los correctos (p.ej. K y Longitudes no Arriostradas de los elementos). Se puede usar la plantilla Display Design Results para ver esta información en los elementos del pórtico. De manera conveniente, es posible apreciar el análisis de las secciones al mismo tiempo que la información del diseño.
2. Verifique que las combinaciones de carga de diseño que el programa ha proporcionado sean las correctas y adecuadas para el tipo de estructura en particular que se este analizando. Sin no lo son, añada las combinaciones de carga que desea utilizar en el diseño.
3. Verifique los resultados del diseño en puntos claves, para asegurarse que los resultados del diseño guardan relación con los resultados esperados.
4. Verifique que los factores de carga en el análisis P-Delta son los correctos.
5. Rediseñe la estructura toda vez que efectúe cambios en el modelo. Esto permite ver si las secciones empleadas son aún aceptables.
6. Emplee grupos de elementos para determinar el peso total de la estructura. (Revise el Ejemplo 1 para tener instrucciones de como efectuar este paso.)
7. Emplee grupos de elementos en el diseño para reducir el número de las diferentes secciones a utilizarse en la estructura.
8. El archivo *filename*.EKO contiene la información del peso total de cada uno de las secciones (perfiles) empleadas en el diseño. Esta información nos permite estimar costos de una manera preliminar.

Apéndice A – Descripción de los Iconos de la Barra de Herramientas

Icono	Nombre del Control	Permite
	New Model	Iniciar un nuevo modelo.
	Open *.SDB file	Abrir un archivo existente del SAP2000.
	Save Model	Grabar el modelo activo.
	Undo	Deshacer el último cambio.
	Redo	Revierte el último Deshacer.
	Refresh Window	Regenera la ventana activa con información actualizada
	Lock/Unlock Model	Protege el modelo contra cambios de datos.
	Run Analysis	Efectúa el Análisis.
	Zoom	Zoom en la estructura del área determinada con el mouse.
	Restore Full View	Restaura la vista total del modelo.
	Restore Previous View	Restaura la vista anterior del modelo.
	Zoom In	Zoom in en el modelo. (Acercamiento)
	Zoom Out	Zoom out en el modelo. (Alejamiento)
	Pan	Mueve dinámicamente la estructura en cualquier dirección.
	Show 3-d view	Muestra vista 3-d del modelo.
	Show 2-d View of X-Y/r-Θ Plane	Vista 2-d del modelo paralela al plano X-Y/r-Θ.
	Show 2-d View of X-Z/r-Z Plane	Vista 2-d del modelo paralela al plano X-Z/r-Z .
	Show 2-d View of Y-Z/Θ-Z Plane	Vista 2-d del modelo paralela al plano X-Z o una vista desarrollada del plano r-Z plane.
	Perspective Toggle	Muestra vista 3-d en perspectiva.
	Shrink Elements	Contrae los elementos para facilitar la visualización de la conectividad.
	Set Element	Ajusta la visibilidad de los elementos y sus propiedades.
	Up One Gridline	Visualiza el siguiente nivel superior en una malla en la vista en planta 2-d .
	Down One Gridline	Visualiza el siguiente nivel inferior en una malla en la vista en planta 2-d.

Apéndice B – Descripción de Iconos de la Barra de Herramientas Flotante

Icono	Nombre del Control	Permite
	Pointer Tool	Selecciona elementos individualmente o en cajas.
	Select All	Selecciona todos los elementos en un gráfico.
	Restore Previous Selection	Restaura elementos previamente seleccionados.
	Clear Selection	Libera todos los elementos seleccionados.
	Set Intersecting Line Select Mode	Selecciona los elementos interceptados por una línea.
	Reshape Element	Mueve elementos tomándolos en su parte central y redimensionalos seleccionando sus extremos
	Add Special Joint	Añade manualmente un nudo.
	Draw Frame Element	Dibuja un elemento tipo frame al definir la ubicación de sus nudos extremos.
	Draw Shell Element	Dibuja un elemento tipo shell al definir la ubicación de sus esquinas.
	Quick Draw Frame Element	Dibuja un elemento tipo frame usando una malla.
	Quick Draw Shell Element	Dibuja un elemento tipo shell usando una malla.
	Assign Joint Restraints	Asigna restricciones en los nudos
	Assign Frame Sections	Asigna secciones y materiales a elementos frame
	Assign Shell Sections	Asigna secciones y materiales a elementos shell.
	Assign Joint Load	Asigna cargas concentradas nodales.
	Assign Frame Span Loading	Asigna cargas en elementos tipo frame.
	Assign Shell Uniform Loading	Asigna cargas en elementos tipo shell.
	Show Undeformed Shape	Muestra la geometría original del modelo.
	Display Static Deformed Shape	Muestra la geometría deformada de la estructura.
	Display Mode Shapes	Muestra formas de modo y periodos de vibración.
	Display Element Forces/Stresses	Muestra resultados del análisis – Fuerzas y Esfuerzos.
	Set Output Table Mode	Muestra tablas con los resultados del análisis.

COMENTARIOS FINALES

CAPÍTULO 9

9.1 COMENTARIOS FINALES

Después de dar una primera visión hasta cierto punto con poca profundidad en algunos aspectos ya que se pretende que el presente instructivo inicie al lector interesado de una manera clara, rápida y sencilla en el uso de **SAP2000** pero a su vez lo motive para que explore y profundice en otras opciones que están disponibles en el programa, los comentarios finales tienen por objeto eliminar algunas posibilidades de error en los datos proporcionados al programa así como mejorar la interpretación de resultados, estos son:

- Verificar la geometría y características de los materiales, para ello se sugiere almacenar los datos en un archivo con extensión S2K y revisarlos con algún editor con objeto de detectar posibles errores en las características de los materiales y dimensiones de los elementos.

- Verificar las condiciones y combinaciones de carga bajo las cuales se realizará el análisis del modelo, lo anterior con objeto de detectar posibles omisiones o duplicidad de cargas (por ejemplo peso propio).

- Verificar los grados de libertad y las condiciones de apoyo o restricción de los nudos especificados en el archivo de datos los cuales deberán ser acordes con el número de ecuaciones que se forman y resuelven.

- Una vez realizado el Análisis se deberán de verificar e interpretar los resultados, el equilibrio se deberá satisfacer en todo momento, se recomienda que manualmente se verifique éste, por lo menos de manera global (suma de fuerzas externas y reacciones), sin embargo no está por demás verificar el equilibrio de algunos elementos de la estructura de manera aislada (nudos y barras), por ejemplo verificando el equilibrio de algún entrepiso (suma de cortantes en las columnas de entrepiso con el cortante externo en la misma dirección).

- Se deberá verificar la forma de los diagramas de elementos mecánicos la cual tendrá que corresponder con el tipo de cargas, por ejemplo, si en una barra existe carga uniforme el diagrama de cortantes deberá presentar variación lineal y el de momentos variación parabólica.

- La configuración deformada que la estructura presente para alguna condición de carga deberá ser consistente con las condiciones de apoyo del modelo analizado así como con las características de las fuerzas contenidas en esa condición.

- Cuando sea posible se tratará de relacionar los resultados obtenidos con el programa con los que resulten de la aplicación de algún método aproximado, por ejemplo para cargas laterales en un marco se puede utilizar alguno de los métodos aproximados como el del factor o el de Bowman.

FACULTAD DE INGENIERIA U.N.A.M.
DIVISION DE EDUCACION CONTINUA

"Tres décadas de orgullosa excelencia" 1971 - 2001

CURSOS ABIERTOS

SAP – 2000 PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

TEMA

ANÁLISIS Y DISEÑO INTEGRADO DE ESTRUCTURAS POR EL MÉTODO DE ELEMENTOS FINITOS

EJEMPLO DE APLICACIÓN

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERIA
OCTUBRE DEL 2001

Palacio de Minería   Calle de Tacuba 5   Primer piso   Deleg. Cuauhtemoc 06000   México, D.F.   APDO Postal M-2285
Telefonos 5512-8955   5512-5121   5521-7335   5521-1987   Fax 5510-0573   5521-4021 AL 25

DISCLAIMER

Indice

Ejemplo 1	Pórtico Bidimensional bajo Carga Estática	1
Ejemplo 2	Pórtico Bidimensional con Carga de un Espectro de Respuesta	13
Ejemplo 3	Pórtico Bidimensional Análisis de Historia en el Tiempo	20
Ejemplo 4	Diseño en Acero de un Pórtico Bidimensional	32
Apéndice A	Descripción de los Iconos de la Barra de Herramientas	A1
Apéndice B	Descripción de los Iconos de la Barra Flotante	B1

4 Edite la geometría de la malla y presione el botón OK para cerrar la plantilla.

> Sugerencia: *Finalizada la edición de la malla, se puede hacer click con el botón derecho del mouse sobre las columnas para verificar si éstas tienen la longitud apropiada. Esta es una manera muy práctica de obtener información sobre cualquier nudo o elemento de la estructura.*

Edición de Apoyos

El siguiente paso es el cambio de los apoyos de la estructura de la opción por defecto que corresponde a nudos articulados, a la opción de nudos rígidos que tenemos en este caso.

1 Seleccione el icono **Pointer Tool** de la barra de herramientas flotante.

2 Marque un área rectangular que abarque los tres nudos en la base de la estructura.

> Sugerencia: *Se puede observar la barra de estado para ver el número y tipo de elementos que han sido seleccionados.*

3 Seleccione el icono **Assign Joint Restraints** de la barra de herramientas flotante para asignar empotramiento en los apoyos de la estructura. Se pueden asignar también otras características de los nudos desde el menú **Assign**.

Definición de la Sección Transversal de los Elementos

1. Seleccione primeramente todas las secciones transversales que van a emplearse en el pórtico. Desde el menú **Define** seleccione la opción **Frame Sections**. Luego importe los perfiles de acero mostrados en la Figura 1-1.

> Nota: ***Se puede seleccionar más de una sección a la vez de la lista Section Selection. Para ello presione la tecla Ctrl mientras se efectúa la selección.***

2 Bajo el menú **Select** encontrará varias formas de seleccionar nudos y elementos. Para este problema son útiles los modos de selección **Pointer/Window** e **Intersecting Line**.

3. Una vez seleccionados los elementos del pórtico deseados, se podrán asignar las secciones de acero correspondientes a través del botón **Assign Frame Sections** que está ubicado en la barra de herramientas flotante.

Ejemplo 1 Pórtico Bidimensional con Carga Estática

Asignación de Cargas

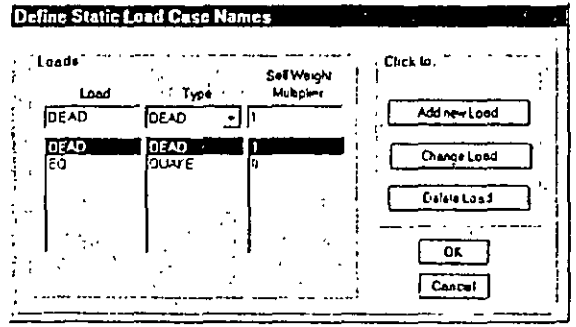


Figura 1-2 Plantilla con los nombres de las condiciones de carga estática

1. El primer paso al ingresar las cargas es definir las condiciones de carga estática. Para ello ingrese al menú **Define** y seleccione la opción **Static Load Cases**.

- **DEAD** puede usarse para las cargas verticales por peso propio de las vigas, manteniendo el indicador **Self Weight Multiplier** con el valor 1, SAP2000 agregará el peso propio de las vigas.

2. Defina una condición de carga lateral estática llamada **EQ** para la carga de sismo. Asigne esta carga lateral como una carga del tipo **QUAKE**. Esto permitirá al programa efectuar automáticamente las combinaciones de carga a ser empleadas por el módulo de diseño del SAP2000. Además asigne al parámetro **Self Weight Multiplier** el valor cero.

3. Las cargas verticales mostradas en la Figura 1-1 pueden asignarse a las vigas seleccionando todas las vigas utilizando el botón **Assign Frame Span Loads** de la barra de herramientas flotante.

4. Las cargas laterales estáticas necesitan ingresarse seleccionando individualmente cada nudo y empleando el botón **Assign Joint Loads**.

> Importante: ***Asegúrese de que esté añadiendo las cargas a la condición de carga correspondiente.***

Creación de Diafragmas de Piso

Crear diafragmas de piso y especificar la masa del piso sólo en la dirección X son técnicas comúnmente usadas para reducir el tamaño del problema. Por otro lado, al añadir diafragmas el comportamiento del modelo se asemeja al de un edificio con diafragmas rígidos.

1. Repita los siguientes pasos para cada piso:
   - Seleccione todos los nudos del piso.
   - Entre al menú **Assign** y seleccione la opción **Joint ... Constraints**.
   - Seleccione **Add Diaphragm** del la caja de opciones.
     - En la plantilla **Diaphragm Constraint** ingrese un nombre para el diafragma del primer piso En este caso usaremos el nombre DIA1.
     - Seleccione la opción **Z-axis constraint**. Esta opción define un diafragma perpendicular al eje-Z.
     - Presione el botón OK.
   - Presione el botón OK para finalizar la operación.
   - Repita estos pasos para los demás pisos usando diferentes nombres en cada uno.
2. La masa de todos los pisos es la misma. Luego seleccione un nudo en cada piso.
3. Cambie las unidades en que se van a ingresar los datos a Kip-in, puesto que la masa indicada en la Figura 1-1 está dada en esas unidades
4. Del menú **Assign** seleccione la opción **Joint ... Masses**
   - Ingrese la masa de cada piso en la dirección del eje coordenado local 1 (que en éste caso coincide con la dirección del Eje Global X).
   - Todos los demás valores son cero.
5. Retorne las unidades a Kip-ft.

Propiedades de los Materiales

Por último, antes de efectuar el análisis de la estructura, deberemos verificar que la asignación de las propiedades de los materiales es la correcta.

1. Desde el menú **Define** seleccione la opción **Materials**.
2. En la plantilla **Materials** seleccione STEEL y presione el botón MODIFY/SHOW MATERIAL
   - En la plantilla **Material Property Data** verifique que las propiedades del material sean las correctas. Recuerde que los valores son reportados en las unidades con las que se este trabajando en este momento.

Efectuandó el Análisis

Una vez que los datos han sido ingresados, es tiempo para correr el modelo y revisar los resultados.

1. Grabe el modelo.
2. Especifique los parámetros para el análisis seleccionando la opción **Analyze** del menú **Set Options**.
   - En la plantilla **Analysis Options** seleccione **Plane Frame Analysis** para reducir el tamaño del problema y por tanto reducir el tiempo de cálculo.
   - Presione el botón OK para aceptar los cambios realizados.
3. Seleccione la opción **Run** del menú **Analyze** para proceder al análisis la estructura.

   Nota: ***Una vez concluido el análisis Ud. podrá revisar los resultados completos en la pantalla antes de presionar el botón OK. Esta será su primera verificación para ver si existe algún problema en el modelo.***

Usando los Resultados

Verificación de los Resultados

Una vez que se ha analizado el modelo se debe verificar si los resultados son correctos y que sus valores son del orden y magnitud a los esperados.

Verificación del Modelo:

1. Verifique que el cortante total en la base es igual a la carga lateral total para la condición de carga EQ.
   - Seleccione el grupo de elementos del pórtico que están ubicados en el primer nivel así como los nudos en la base de la estructura
   - Desde el menú **Assign** seleccione **Group Names**.
   - Asigne a este grupo de elementos un nombre representativo por ejemplo BASE SHEAR.
   - Seleccione el botón ADD NEW GROUP NAME y presione el botón OK.
   - En el menú **Display** seleccione la opción **Show Group Joint Force Sums** y elija el grupo previamente creado.

2 Observe la deformada de la estructura y cree una animación de la misma bajo cargas verticales y laterales para asegurarse de que el comportamiento del modelo es el esperado.

- Ingrese al menú **Display** y seleccione **Show Deformed Shape** y seleccione la condición de carga en la que este interesado. También seleccione la opción **Wire Shadow**, así podrá ver la geometría no-deformada de la estructura al mismo tiempo. Vea las Figuras 1-3 y 1-4 para las formas deformadas de la estructura. Haga click con el botón derecho del mouse sobre cualquier nudo para observar los desplazamientos y rotaciones correspondientes.

- Genere una animación de la deformada presionando el botón START ANIMATION ubicado en la parte inferior de la barra de estado (Para esto se necesita que se encuentre activa una ventana conteniendo la deformada de la estructura). La animación así creada puede salvarse como una archivo *.AVI para verse después desde el menú **File**. (Vea la Ayuda En-línea bajo el ítem "Export an AVI file".)

  Intente esto *: **Presione los botones + y – ubicados junto al botón Animate y vea lo que le sucede a la deformada de le estructura.***

- Presione el botón STOP ANIMATION cuando haya terminado de observar la animación.

Si los procedimientos antes descritos muestran que la información ingresada aparenta ser correcta, podemos entonces avanzar hacia procedimientos más avanzados de revisión de los resultados.

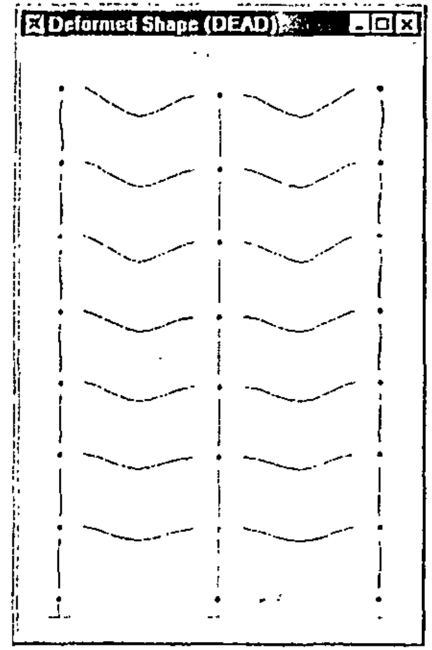


Figura 1-3 Deformada de la Estructura para Cargas Verticales

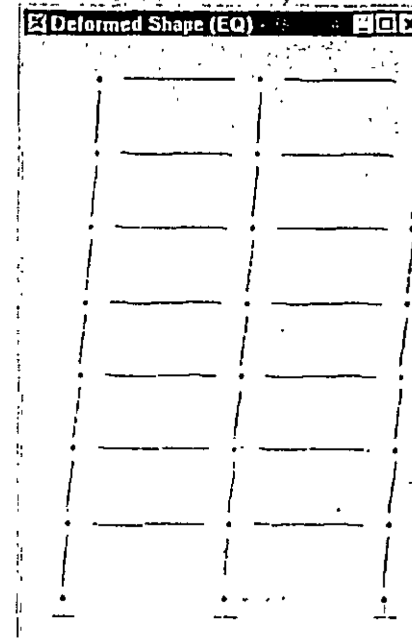


Figura 1-4 Deformada de la Estructura para Cargas Laterales

Comportamiento Estructural

En muchas ocasiones se desea verificar si la estructura se encuentra dentro de determinados límites de comportamiento, tales como rangos de esfuerzo especificados por algún código de diseño. SAP2000 hace todas estas verificaciones automáticamente cuando los elementos son diseñados. (Las opciones de diseño del SAP2000 serán discutidas con mayor detalle en los siguientes ejemplos).

1. Los elementos estructurales pueden diseñarse desde al menú **Design** y seleccionando la opción **Start Design/Check of Structure**.

   - Los elementos del pórtico mostrarán en este momento colores que representan el nivel del esfuerzo en cada elemento. Asimismo en la parte inferior de cada elemento se muestra un valor numérico representativo del nivel de esfuerzo presente en el elemento. Un valor 1 por ejemplo significa 100% esforzado.

- Para tener más información sobre el diseño de los elementos entre al menú **Design** y seleccione la opción **Display Design Info.**

2 Se puede también ver la información del diseño de cada elemento, o inclusive asignar secciones transversales alternativas, haciendo click con el botón derecho del mouse sobre un elemento.

- De la ventana que se muestre se puede seleccionar el botón DETAILS para apreciar información detallada de la sección bajo cada una de las combinaciones de carga empleadas en el diseño.
- También se puede rediseñar el elemento después de cambiar sus parámetros de diseño, longitud efectiva, factor K o propiedades a la sección, presionando el botón REDESIGN.

3. Si se ha seleccionado una nueva sección la cual se quiere utilizar en el diseño final de la estructura, únicamente ingrese al menú **Design** y seleccione la opción **Update Analysis Sections** para reanalizar la estructura con las nuevas secciones seleccionadas.

Nota: Puede ser necesario el uso del botón Refresh Window de la barra de herramientas para actualizar la información en la ventana activa luego de haber efectuado cambios en los parámetros de diseño.

Observando e Imprimiendo Resultados

A menudo se necesita disponer de una copia impresa de los resultados de los análisis obtenidos con el SAP2000. Existen diferentes formas de obtenerlos:

1. Se pueden elegir los resultados que nos interesan con la opción **Generate Output** ubicada en la plantilla **Analysis Options**. El botón **Select Output Options** que aparece permitirá seleccionar cuantos y cuales de los resultados queremos imprimir. Estos resultados son escritos en un archivo de texto con el mismo nombre de nuestro archivo de datos, pero con la extensión *.OUT.

2 Los datos ingresados así como la mayor parte de los resultados generados también pueden verse a partir del menú **Display**.

3. Desde el menú **File** se puede optar por imprimir ya sea **Gráficos**, **Tablas con los Datos Ingresados** ó **Tablas con los Resultados del Análisis y Diseño de elementos.**

Sugerencia: Si existen elementos ó nudos seleccionados al momento de generar la impresión de resultados, únicamente se imprimirá la información correspondiente a dichos elementos. De lo contrario, la impresión se generará para todos los elementos y nudos del modelo.

4 El análisis efectuado por el SAP2000 genera dos archivos de salida. El archivo *filename*.EKO que incluye toda la información empleada en el análisis; y el archivo *filename*.OUT, que

contiene los resultados del análisis así como los resultados específicamente seleccionados en el menú **Analyze ...Set Options.**

Recuerde: ***Es un buen hábito generar la salida primeramente a un archivo de texto antes de enviarlo directamente a la impresora. Esto nos permite revisar previamente la información usando cualquier editor de textos, sin tener que hacer frente a enormes pilas de papel.***

Comentarios Finales

Como habrá podido observar, SAP2000 es una poderosa herramienta para el análisis estructural que puede usarse en una gran variedad de problemas. Sin embargo, es muy importante entender los principios de ingeniería sobre los cuales este programa ha sido creado.

La mayoría de los trabajos en ingeniería se inician con sencillos anteproyectos para posteriormente madurar en complejos proyectos de análisis/diseño. Esto hace que sea muy importante decidir desde un inicio las herramientas apropiadas de forma tal que no sea necesario cambiar de programas a la mitad de un proyecto. SAP2000 trata de satisfacer la mayor parte de las necesidades que un diseñador puede tener durante el desarrollo de un proyecto.

Las características que SAP2000 ofrece en el proceso de diseño incluyen:

- La capacidad de diseñar pequeños ó grandes proyectos sin necesidad de aprender a usar un nuevo programa.
- La capacidad de diseñar elementos de concreto y acero en un mismo programa.
- Algoritmos de cálculo rápidos que permiten al usuario dedicar mayor tiempo en la modelación del problema y optimización del diseño de elementos estructurales.
- La habilidad para modificar y mejorar el diseño fácilmente.
- Existen probablemente tantas formas de modelar una estructura como ingenieros existen. Sin embargo, puede encontrar útiles algunas de las siguientes ideas:
- Comience con un modelo básico de la estructura y trate de entenderlo antes de añadir más detalles. Será más sencillo corregir problemas en el sistema estructural adoptado cuando el modelo es aún simple.
- Asegúrese de que la estructura pueda construirse y que se comportará en la manera en que la hemos modelado. Si no puede ser construida en esa forma, es necesario entender el efecto del proceso constructivo en el comportamiento final de la estructura.
- Documente detalladamente su diseño incluyendo información sobre las consideraciones asumidas, áreas que deban revisarse e incluso sobre información que aún es requerida. Para ello use el editor de textos **User Comments and Session Log** que se encuentra dentro del menú **File**. Este editor de textos incorporado en el programa, le permitirá que dichas anotaciones y comentarios formen parte del modelo.

- Experimente con sistemas estructurales alternativos. SAP2000 ha sido diseñado para efectuar cálculos numéricos rápidamente, permitiendo utilizar mayor tiempo en el mejoramiento de nuestros diseños.
- Así como hay un tiempo asignado para la revisión general al final de un proyecto, no hay razón por la que no deba haber un tiempo para revisar el proyecto desde sus inicios.

EJEMPLO 2

Pórtico Bidimensional con Carga de un Espectro de Respuesta

Descripción

Este ejemplo es una continuación del Ejemplo 1. En esta sección mostraremos como incorporar un Espectro de Respuesta en el análisis de un pórtico bidimensional. La base para definir el Espectro de Respuesta será el espectro del código UBC94S2 el cual esta incluido en el SAP2000.

Aspectos Significativos del Modelo y del SAP2000

- Uso del comando Help para obtener instrucciones sobre las opciones del SAP2000.
- Incorporación de una carga proveniente de un Espectro de Respuesta..
- Adecuar la escala del Espectro de Respuesta para su uso en el diseño.

Definiendo el Espectro de Respuesta

Un Espectro de Respuesta es la máxima respuesta de un sistema excitado en su base por una función aceleración-tiempo. Esta función se expresa en términos de la frecuencia natural de la estructura y del amortiguamiento del sistema. El Espectro de Respuesta del código UBC94S2 que vamos a emplear en este ejemplo es suministrado con SAP2000 y no es necesario definirlo por separado. Si hubiese la necesidad de definir un Espectro de Respuesta distinto, se puede usar la ayuda en línea para obtener instrucciones que indican paso a paso como efectuar esta tarea.

Ayuda En-línea

Recuerde: *Podrá utilizar alguno de los métodos siguientes para obtener información sobre cualquiera de las funciones del SAP2000*

1. Del menú **Help** seleccione **Search for Help on.**
2. Con el plantilla **Index** seleccionada:
   - En el Area 1 escriba 'Define'. Ud. vera en el Area 2 una lista de todos los tópicos disponibles que comiencen con la palabra 'define'. Uno de esos tópicos es 'Define Response Spectrum Functions', que es el tópico del cual necesitamos obtener ayuda. Haga doble click en la línea con la frase 'Define Response Spectrum Functions' para que el programa muestre la información correspondiente.
3. Alternativamente, seleccione el indicador **Find** para buscar una palabra clave en cualesquiera de los tópicos disponibles en la Ayuda En-línea.
   - Si es la primera vez que usa la opción **Find** de la Ayuda En-línea del SAP2000, aparecerá una plantilla denominada **Find Setup Wizard.**
     - Presione el botón NEXT para aceptar el criterio para construir la base de datos de búsqueda.
     - Presione el botón FINISH para construir la base de datos.
   - En el Area 1 escriba 'Response Spectrum'
   - En el Area 3 encontrará nuevamente la opción 'Define Response Spectrum Functions' la cual puede seleccionarse para obtener la información de ayuda correspondiente.

   Nota: *Se puede encontrar mayor información sobre el uso de la Ayuda En-línea, en la documentación de Windows. También puede ejecutar el archivo WINHELP32.HLP ubicado en C:\WINDOWS\HELP.*

Definiendo el Espectro de Respuesta

1. Si el modelo esta protegido (locked), use el botón **Lock/Unlock Model** para remover la protección y poder efectuar cambios en el modelo.

2. Ajuste las unidades a Kip-ft.
3. Ingrese al menú Define y seleccione la opción Response Spectrum Case.
4. Presione el botón ADD NEW SPECTRA en la plantilla Response Spectra.
5. En la plantilla Response Spectrum Case Data:
   - Especifique el amortiguamiento asociado al Espectro de Respuesta colocando en la casilla Damping el valor, para nuestro caso: 0.05 (5 %)
   - Seleccione **UBC94S2** para la dirección **U1**, así como un factor de escala 32.2ft/sec^2 en la casilla Scale Factor. Este factor de escala es usado por el Espectro de Respuesta debido a que el espectro UBC94S2 esta normalizado al valor de la aceleración de la gravedad g.
   - El resto de los valores por defecto son aceptables.
   - Presione el botón OK para aceptar los cambios hechos en ambas plantillas.

Efectuando el Análisis

Una vez que se han realizado las modificaciones, es tiempo de analizar el modelo y echar una mirada a los resultados del Espectro de Respuesta.

1. Grabe el modelo.
2. En el menú **Analyze** seleccione la opción **Set Options.**
   - Marque la casilla **Dynamic Analysis.**
   - Presione el botón SET DYNAMIC PARAMETERS y modifique el número de modos de vibración a ser considerados en el análisis en la opción Number of Modes. Para nuestro caso 7. El resto de valores por defecto son aceptables
   - Presione el botón OK en ambas plantillas para aceptar los cambios.

   Nota: *Se debe decidir cuantos modos de vibración deben considerarse en el análisis para obtener resultados adecuados. Para ello hay muchos criterios a tomar en cuenta, pero para una estructura sencilla como la que estamos analizando puede considerarse satisfactorio un numero de modos igual al numero de pisos.*
3. Seleccione la opción **Run Minimized** del menú **Analyze** para analizar la estructura.

   Nota: *La opción Run Minimized es sumamente útil cuando se tienen modelos grandes que requieren mayor tiempo para ser analizados. Esta opción permite al SAP2000 correr en un segundo plano, permitiendo continuar trabajando con otros programas. Otra ventaja de esta opción es que nos brinda un botón para cancelar la ejecución del análisis en caso necesario.*

Verificación de los Resultados

1. Verifique si las formas modales y períodos de vibración son los esperados.

   - Del menú **Display** seleccione la opción **Show Mode Shape** y elija el modo en que esta interesado. Pude también seleccionar la opción **Wire Shadow** para ver al mismo tiempo la forma no-deformada de la estructura Observe las Figuras 2-1 a la 2-4 y note que el número de modo y período de vibración correspondiente están indicados en el título de la ventana.

     Nota: ***Se puede apreciar los modos de vibración subsecuentes presionando los botones + y −, próximos al botón START ANIMATION.***

2. Es útil ver el cortante en la base producido por el análisis del Espectro de Respuesta.

   - Usando el grupo BASE SHEAR que fue definido en el Ejemplo 1 observe el cortante en la base de la estructura debido al Espectro de Respuesta. Se puede apreciar que este cortante es considerablemente mayor que el debido a la condición de carga estática.

3. Se puede verificar el desplazamiento de un nudo debido al Espectro de Respuesta.

   - Del menú **Display** seleccione **Show Deformed Shape.**
     - En la plantilla **Deformed Shape** seleccione la condición de carga para el análisis espectral.
     - Presione el botón OK.
   - Haga click con el botón derecho del mouse sobre un nudo del nivel superior de la estructura para ver su correspondiente desplazamiento en la dirección global X.

4. Verifique la participación de la masa de la estructura para ver si se ha incluido el en la solución del problema el número de modos suficiente. Para ello se debe revisar el archivo de texto filename.OUT empleando un editor de textos como el WordPad de Windows.

   - Minimice el programa SAP2000.
   - Inicie el programa WordPad o cualquier otro editor de textos.
   - En WordPad abra el archivo *filename*.OUT. Donde *filename* es el nombre del archivo usado al grabar este ejemplo.
     - Busque la sección titulada MODAL PARTICIPATING MASS RATIOS como se muestra en la Figura 2-5.
     - Bajo la columna CUMULATIVE SUM encontrará que los Modos de Vibración 1 al 7 incluyen el 100% de la participación de masa. Lo cual significa que los 7 modos empleados en el análisis fueron suficientes.

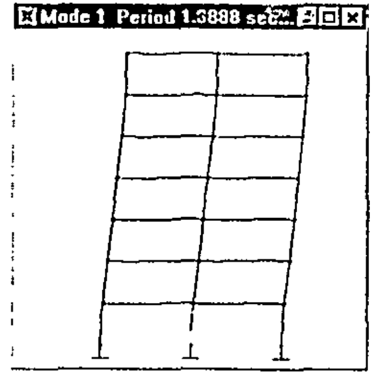


Figura 2-1 Forma Modal y Período de Vibración 1

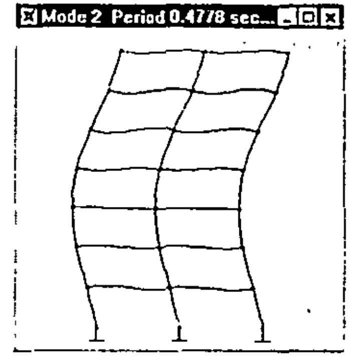


Figura 2-2 Forma Modal y Período de Vibración 2

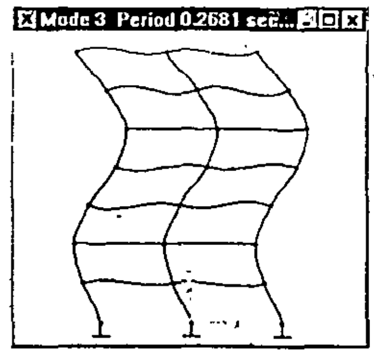


Figura 2-3 Forma Modal y Período de Vibración 3

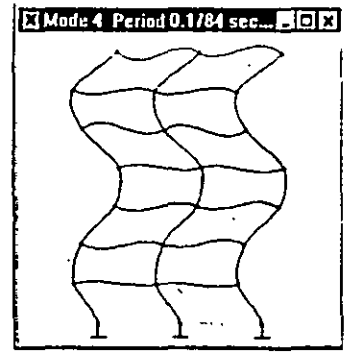


Figura 2-4 Forma Modal y Período de Vibración 4

M O D A L   P A R T I C I P A T I N G   M A S S   R A T I O S

MODE	PERIOD	INDIVIDUAL MODE (PERCENT)			CUMULATIVE SUM (PERCENT)		
		UX	UY	UZ	UX	UY	UZ
1	1.388750	79.6359	0.0000	0.0000	79.6359	0.0000	0.0000
2	0.477933	11.5761	0.0000	0.0000	91.2120	0.0000	0.0000
3	0.268126	4.3023	0.0000	0.0000	95.5144	0.0000	0.0000
4	0.178439	2.1229	0.0000	0.0000	97.6373	0.0000	0.0000
5	0.133678	1.4077	0.0000	0.0000	99.0450	0.0000	0.0000
6	0.107637	0.6592	0.0000	0.0000	99.7043	0.0000	0.0000
7	0.090778	0.2957	0.0000	0.0000	100.0000	0.0000	0.0000

Figura 2-5 Bloque de Participación de la Masa del archivo de salida

Modificando la Escala del Espectro de Respuesta

Algunos códigos de diseño permiten modificar la escala del Espectro de Respuesta de manera que el cortante en la base del análisis espectral sea igual al cortante en la base del análisis empleando cargas sísmicas estáticas.

En este sentido para obtener el nuevo factor de escala para el Espectro de Respuesta tenemos que :

1. Dividir el Cortante en la Base producido por la Carga Sísmica Estática por el Cortante en la Base obtenido del Análisis Espectral, y multiplicar dicho número por 32.2 ft/seg^2 para obtener el nuevo factor de escala del Espectro de Respuesta.
2. Substituir el nuevo factor de escala en el Espectro de Respuesta.
3. Efectuar nuevamente el análisis para obtener las nuevas fuerzas en los elementos bajo la acción del Espectro de Respuesta escalado.

Comentarios Finales

Un análisis empleando el Espectro de Respuesta introduce un nivel de complejidad mayor, que requiere que el ingeniero verifique cuidadosamente los resultados, y tenga muy presente las consideraciones hechas al crear el modelo. Algunos de los aspectos a considerar durante un análisis espectral son:

- Entender completamente el comportamiento estático del modelo antes de efectuar un análisis dinámico.
- Tener un conocimiento cabal y racional de los aspectos involucrados al escalar los resultados del análisis dinámico para obtener un cortante en la base similar al que se obtiene al efectuar un análisis por cargas sísmicas estáticas.
- La ventaja de la mayor rapidez del Análisis Espectral en comparación con el Análisis de Historia en el Tiempo es en muchos casos sustancial. En el diseño, el espectro de respuesta puede incluso proveer aun mayores ventajas debido a que no se deben efectuar verificaciones para diferentes intervalos de tiempo. Sin embargo, es necesario tener presente las limitaciones del Análisis Espectral frente al mayor refinamiento que se obtiene al efectuar un Análisis de Historia en el Tiempo.

EJEMPLO 3

Pórtico Bidimensional
Análisis de Historia en el Tiempo

Descripción

Este ejemplo continua el análisis del pórtico bidimensional visto en los Ejemplos 1 y 2, añadiendo en este caso una carga de sismo especificada con un acelerograma en la base de la estructura. El registro de aceleraciones a utilizarse se muestra en la Figura 3-1 y corresponde a la componente N-S del sismo ocurrido en El Centro en 1940. Los resultados del análisis de historia en el tiempo se utilizarán para generar un Espectro de Respuesta que luego se empleará para reanalizar la estructura a manera de comparación.

Aspectos Significativos del Modelo y del SAP2000

- Respuesta de historia en el tiempo de una excitación en la base.
- Gráficas de los resultados del análisis de Historia en el Tiempo.
- Gráficas de un Espectro de Respuesta a partir de los resultados de la Historia en el Tiempo.
- Importación del Espectro de respuesta para su uso en el análisis.

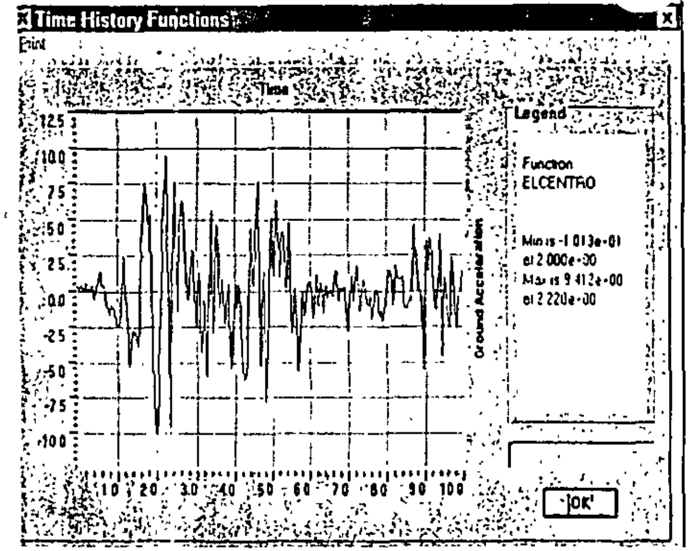


Figura 3-1 Acelerograma de entrada del Sismo de El Centro 1940 (ft/sec²)

Definición del Concepto Historia en el Tiempo

El término Historia en el Tiempo define un registro de aceleraciones del terreno a determinados intervalos de tiempo para una excitación sísmica específica en una dirección determinada. El registro es usualmente normalizado y en consecuencia necesita multiplicarse por la aceleración de la gravedad ó por algún otro factor correspondiente.

1. Desde el menú **Define** seleccione **Time History Functions**.
2. Seleccione el botón ADD FUNCTION FROM FILE.
   - Presione el botón **Open File** y seleccione el archivo ELCENTRO ubicado en el subdirectorio EXAMPLES del directorio SAP2000.
   - Cambie el nombre de la función a ELCENTRO para que sea fácil de reconocer.
   - El formato de este archivo es de tres pares de columnas de datos por renglón. El primer par de datos la primera columna de cada par es el tiempo y la segunda es la aceleración.
     - Ingrese 3 Puntos por Línea.

- Seleccione la opción **Time and Function Values** (en archivo de datos).
- Presione el botón OK.

- Presione el botón OK para aceptar los datos ingresados.

3 Del menú **Define** seleccione la opción **Time History Cases** para definir los parámetros específicos para el análisis de Historia en el Tiempo de nuestro modelo.

- Seleccione el botón ADD NEW HISTORY.
- Para adicionar amortiguamiento al sistema presione el botón MODIFY/SHOW MODAL DAMPING, ingrese 0.05 (5%) para todos los modos y presione OK.
- Ingrese en el ítem **Number of Output Time Steps** el valor 500.
- Ingrese 0.02 (sec) en el ítem **Output Time Step Sizes**. Estos parámetros nos darán 10 segundos del registro sísmico para el análisis de historia en el tiempo.
- De la Lista **Analysis Type** seleccione la opción **Linear**.
- En el área **Load Assignment** :
  - Seleccione ACC DIR1 para el parámetro **Load**
  - Seleccione la opción ELCENTRO para el parámetro **Function**
  - Para el parámetro **Scale Function** ingrese la aceleración de la gravedad que es de 386.4 in/sec^2 si trabaja en Kip-in o 32.2 ft/sec^2 si trabaja en Kip-ft.
  - Para los parámetros **Arrival Time** y **Angle** asigne valor cero.
  - Presione el botón **ADD** para agregar esta carga al modelo, y presione el botón OK para aceptar los datos que acaba de ingresar.
- Presione el botón OK en ambas plantillas para aceptar las adiciones al modelo.

De esta manera hemos ingresado toda la información que necesitamos para efectuar el Análisis de Historia en el Tiempo.

Nota: *Por lo general es una buena idea correr el modelo cada vez que se hace un cambio ó adición importante al modelo. Esto permite detectar errores y ahorra tiempo en el diseño final.*

Efectuando el Análisis

1. Grabe el modelo.
2. Ajuste los parámetros para el análisis seleccionando **Analyze** del menú **Set Options**.
   - Verifique que los parámetros en **Dynamic Analysis** son los mismos del Ejemplo 2.
3. Seleccione en el menú **Analyze** la opción **Run** para analizar la estructura.

Usando los Resultados

Verificación de los Resultados

Una vez que ha corrido el modelo se deben verificar que los resultados obtenidos sean del orden y magnitud a lo esperado.

1. Verifique el Cortante en la Base producido en el Análisis de Historia en el Tiempo.
   - Desde el menú **Display** seleccione la opción **Show Time History Traces**.
     - De la plantilla **Time History Display Definition** presione el botón DEFINE FUNCTIONS.
     - En la plantilla **Time History Functions** seleccione Add Base Functions y marque solamente la opción Base Shear X.
     - Presione OK para regresar a la plantilla **Time History Display Definition**.
     - Adicione la función Base Shear X a la lista **Plot Functions**.
     - Presione el botón DISPLAY para ver una gráfica del cortante en la base en la dirección global X como función del tiempo. Vea la Figura 3-2.

   Nota: ***También se puede generar la gráfica del cortante en la base seleccionando Add Group Summation Forces en lugar de Add Base Functions y seleccionando el grupo de elementos BASE SHEAR definido en el Ejemplo 1.***
2. También se puede verificar el desplazamiento de un nudo ante una excitación de Historia en el Tiempo, para ello:
   - Escoja un nudo y desde el menú **Display** seleccione la opción **Show Time History Traces**.

- Presione el botón DEFINE FUNCTIONS de la plantilla **Time History Functions** Seleccione el nombre del nudo de la lista y presione el botón MODIFY/SHOW TH FUNCTION
- En la plantilla Time History Joint Function seleccione DISPL para el parámetro **Vector Type** y UX para el parámetro **Vector Direction**.
- Presione el botón OK para aceptar los cambios.
- Presione los botones OK para regresar a la plantilla **Time History Display Definition**.
- Añada el nudo de la lista **List of Functions** a la lista **Plot Functions** y remueva de esta última la función Base Shear X
- Presione el botón DISPLAY para ver el desplazamiento del nudo con respecto al tiempo. Vea Figura 3-3

- Se puede también definir una función nodal directamente en la plantilla **Time History Display Definition** sin haber seleccionado previamente el nudo.
  - En la plantilla **Time History Display Definition** presione el botón DEFINE FUNCTIONS y en la plantilla **Time History Functions** seleccione la opción Add Joint Disps/Forces.
  - En la plantilla Time History Joint Function ingrese el nombre del nudo (ID).
  - Seleccione **Vector Type** y **Vector Direction**.
  - Presione los botones OK para regresar a la plantilla **Time History Display Definition** en donde encontrará la nueva función nodal en el recuadro **List of Functions**.

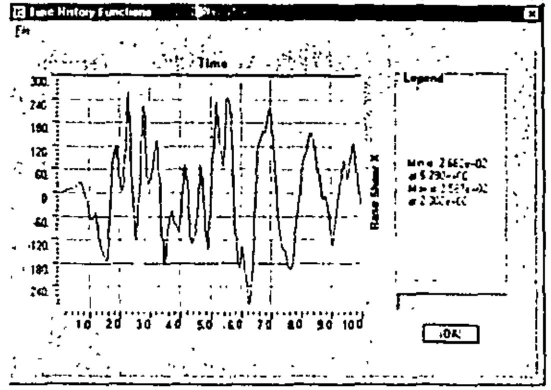


Figura 3-2 Cortante en la Base – Kips

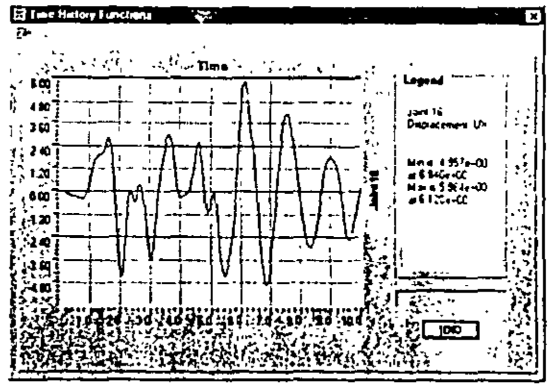


Figura 3-3 Historia en el Tiempo del Desplazamiento del Nivel Superior – In

Creación de un Espectro de Respuesta

Lo primero que se tiene que hacer en este punto es crear un Espectro de Respuesta a partir de los resultados del Análisis de Historia en el Tiempo Los datos generados de esta forma deben imprimirse en archivo de texto para poder editadarse en un formato que pueda ser leído por el SAP2000.

Gráficas del Espectro de Respuesta

1. Seleccione un nudo en la base de la estructura.
2. Del menú **Display** seleccione la opción **Show Response Spectrum Curves**. Esta opción aparece solamente cuando se ha seleccionado un nudo.
3. En la plantilla **Response Spectrum Generation** encontrará el nombre del nudo que fue seleccionado.
   - Bajo el ítem **Define** asigne el valor X para el parámetro **Vector Direction**.
   - Bajo el ítem **Axes** seleccione **Period** para el parámetro **Abscissa** y **PSA** (Seudoaceleración Espectral) para el parámetro **Ordinate**.
   - Bajo el ítem **Options** seleccione **Arithmetic** tanto para **Abscissa** como para **Ordinate**. Para el parámetro **Ordinate** asigne el factor de escala 1/g (g=32.2 ft/sec^2) es decir, 0.03106 sec^2/ft si las unidades en que se trabaja son kip-ft.

     Nota: *El factor de escala es usado para normalizar el Espectro de Respuesta. El registro para el análisis de historia en el tiempo que se utilizó para generar el Espectro de Respuesta estaba normalizado a la aceleración de la gravedad (g) por lo que necesitamos dividir el espectro por la misma cantidad para obtener valores normalizados.*
   - Bajo el ítem **Period** seleccione para las frecuencias los parámetros **Default** y **Structural**. Estos parámetros son usados en la generación del Espectro de Respuesta, las frecuencias del tipo **Default** son una serie de frecuencias predeterminadas que son típicamente de interés en las estructuras; las frecuencias del tipo **Structural** son las frecuencias naturales de la estructura.
   - Bajo el ítem **Damping** mantenga el valor del amortiguamiento de 0.05 para el parámetro **Damping Value**. Como se ha asumido que la estructura tiene 5% de amortiguamiento no será necesario emplear otros valores para el amortiguamiento del sistema.
   - Presione el botón DISPLAY cuando se halla terminado.
4. Ahora Ud. podrá apreciar la gráfica del Espectro de Respuesta para el sismo de El Centro para un amortiguamiento de 5%. Ver Figura 3-4.

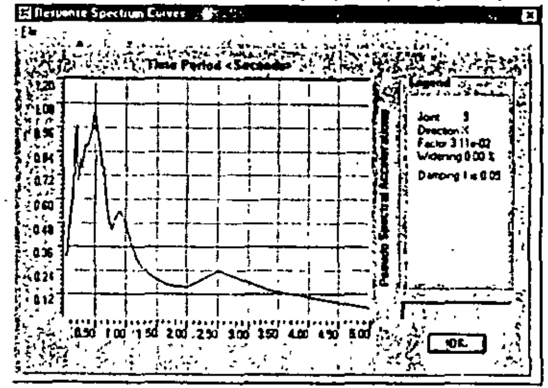


Figura 3-4 Espectro de Respuesta del Análisis de Historia en el Tiempo

5. En la plantilla **Response Spectrum Curves** seleccione la opción **Print Tables to File**. Esta opción generará un archivo que tiene dos columnas de datos. La primera de ellas es el período, y la segunda es su correspondiente Seudoacelaración (PSA).
   - Grabe el archivo con el nombre RS-ELCEN.TXT

Edición de Tablas.

El siguiente paso consiste en hacer algunas pequeñas modificaciones al archivo de texto RS-ELCEN.TXT para que tenga un formato que pueda ser leído por el SAP2000. Esto se debe a que cuando el archivo original es creado se le agrega información aclaratoria que permite al usuario interpretar y entender fácilmente su contenido. Esta información extra debe removerse.

1. Con un editor de textos como WORDPAD o NOTEPAD abra el archivo RS-ELCEN.TXT.
   - Seleccione todo el texto que se muestra resaltado en la **Figura 3-5** y remuévalo.
   - Grabe el archivo RS-ELCEN.TXT como un archivo de texto con el mismo nombre.
2. Ahora que el archivo tiene tan solo las columnas con los períodos y sus correspondientes Seudoacceleraciones, podrá ser leído directamente por el SAP2000

```
SAP2000 v6.06 File: TUTORIAL1  Kip-ft Units  PAGE 1
May 19, 1997 17:26

S P E C T R U M   D A T A

Joint       9
Direction   XT
Factor      0.03
Widening    0.5

Pseudo Spectral Accelerations vs Time Period <Seconds>

                DAMPING
                 0.0500
      0.0303   3.0866E-01
      0.0357   3.2657E-01
      0.0400   3.1139E-01
      0.0455   3.2217E-01
      0.0500   3.2205E-01
      0.0555   3.3096E-01
      0.0606   3.4265E-01
      0.0667   3.7431E-01
```

Figura 3-5 Archivo de salida con el Espectro de Respuesta generado

Lectura de los Datos del Espectro

Ahora que se tienen datos en un formato que el SAP2000 puede leer, necesitamos indicar al programa la ubicación del archivo así como la forma en que éste contiene la información.

1. Si el modelo esta protegido presione el botón **Lock/ Unlock Model** en la barra de herramientas. Al hacer esto se remueve la protección sobre el modelo y nos permitirá realizar las modificaciones.
2. Del menú **Define** seleccione **Response Spectrum Functions.**
3. En la plantilla **Response Spectrum Functions** presione el botón **Add Function from File.**
   - Asigne al espectro el nombre RSELCEN
   - Presione el botón **Open File** y seleccione el archivo RS-ELCEN.TXT en **Pick File.**
   - Mantenga el parámetro **Number Of Points Per Line** en el valor 1 puesto que únicamente hay un par de datos por renglón para definir el Espectro de Respuesta.
   - Seleccione la opción **Period and Acceleration Values.**
   - Presione el botón OK para cerrar las plantillas.

4. Del menú **Define** seleccione **Response Spectrum Cases.**
5. En la plantilla Response Spectrum presione el botón ADD NEW SPECTRA.
   - Asigne al parámetro **Modal Damping** el valor 0.05.
   - En el área **Input Response Spectra** seleccione RSELCEN **para la dirección U1** y asígnele el factor de escala 32.2 ft/sec^2.
   - Los demás valores por defecto son aceptables.
   - Presione el botón OK para cerrar las plantillas.

Efectuando el Análisis

Una vez que se han hecho las modificaciones es tiempo de correr el modelo y revisar los resultados obtenidos.

1. Grabe el modelo.
2. Seleccione **Run Minimized** del menú **Analyze** para analizar la estructura.

Revisando los Resultados

Lo primero que debe hacerse es la revisión de la máxima deflexión en la parte superior de la estructura así como el cortante en la base tanto para el análisis espectral como para el análisis de Historia en el Tiempo. La comparación de estos resultados nos permitirá ver que tan bién funciona la metodología previamente descrita. Al final de esta sección encontrará los resultados del análisis por cargas sísmicas estáticas, del análisis espectral y del análisis de historia en el tiempo..

Deflexiones de acuerdo al Análisis Espectral

1 Del menú Display seleccione **Display Deformed Shape**.

- En la plantilla **Deformed Shape** seleccione la condición de carga para el análisis espectral.
- Presione el botón OK.

2 Haga click con el botón derecho del mouse sobre un nudo ubicado en el nivel superior para ver su desplazamiento en la dirección del eje global X.

Cortante en la Base en el Análisis Espectral

Usando el grupo BASE SHEAR que fue creado en el Ejemplo 1, observe los valores del cortante en la base de la estructura debido al Análisis Espectral.

Deflexión y Cortante en la Base en el Análisis de Historia en el Tiempo

1 Usando el método descrito en la primera parte de esta guía grafique la deflexión en el nivel superior de la estructura.

2. Ahora remueva ese nudo de la lista **Plot Functions** y en su lugar grafique la función **Base Shear X** ubicada en **List of Functions** de la plantilla **History Display Definitions**.

	Carga Lateral Estática	Espectro de Respuesta	Historia en el Tiempo
Max Deflexión	1.65 in	5.5 in	5.8 in
Max Cortante en la Base	72.5 Kips	302 Kips	325 Kips

Tabla 3-1 Comparación de los Resultados del Análisis de Cargas Laterales

Comentarios Finales

Como habrá podido apreciar el Análisis de Historia en el Tiempo involucra un mayor tiempo de cómputo que el Análisis con Espectro de Respuesta Debe observarse sin embargo que ambos métodos de análisis brindan resultados similares. En este sentido es sumamente importante que el ingeniero entienda las ventajas y limitaciones de cada método para poder utilizarlos de la manera más adecuada y efectiva.

EJEMPLO 4

Diseño en Acero de un Pórtico Bidimensional

Descripción

Este ejemplo es un introducción al uso de las poderosas herramientas que posee el SAP2000 para el diseño de una estructura una vez concluido su análisis estructural. Se dará énfasis en esta parte a módulos de diseño en acero empleando como ejemplo la estructura analizada en el Ejemplo 1.

Aspectos Importante del Modelo y del SAP2000

- Creación de zonas rígidas en los elementos.
- Selección Automática de grupos
- Cambio de propiedades en elementos
- Designación de elementos por grupos
- Inclusión del efecto P-Delta en el análisis
- Visualización de los resultados
- Auto Selección de secciones

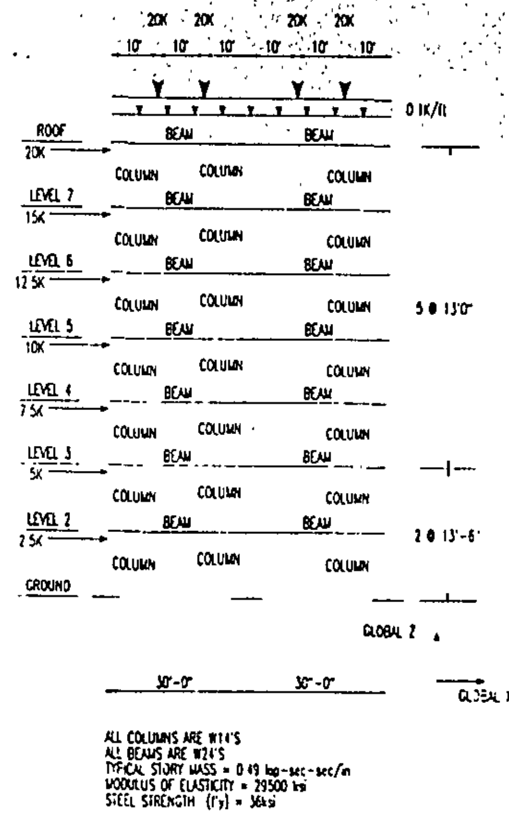


Figura 4-1 Pórtico Bidimensional a diseñarse

Creando el Modelo en el SAP2000

Es posible emplear el modelo desarrollado en el Ejemplo 1 efectuando pequeñas modificaciones.

Materiales

Lo primero que se debe hacer es especificar las propiedades de los materiales.

1. Verifique que las unidades estén en Kip-in.
2. Entre al menú **Define** y seleccione la opción **Materials**.
3. Elija STEEL para el parámetro **Material** y presione el botón MODIFY/SHOW MATERIAL.
4. Especifique el Esfuerzo de Fluencia ajustando el parámetro **Steel Yield Stress** fy en 36 Ksi.
5. Especifique el Módulo de Elasticidad ajustando el parámetro **Modulus of Elasticity** E en 29,500 Ksi.
6. Presione los botones OK para aceptar los cambios y cerrar las plantillas.

Cargas

1. En el ejemplo 1 se asignó el peso propio de la estructura así como una serie de cargas concentradas y distribuidas a la condición de carga DEAD. (Vea Figura 4-2 para la nueva lista Static Loads Case). En este ejemplo asignaremos una condición de carga para la carga viva y otra para el peso propio de los elementos. Es una buena práctica incluir una condición de carga para el peso propio de la estructura con el fin de seguir de cerca el proceso de optimización estructural. Las cargas son separadas en carga muerta, carga viva y carga transversal de sismo de tal manera que el SAP2000 pueda generar automáticamente las combinaciones de carga.
   - Para el caso de carga DEAD especifique el valor del multiplicador **Self Weight Multiplier** en cero.
   - Agregar una condición de carga para el peso propio y nómbrela SELF, asígnele el tipo DEAD como parámetro a **Type** y ajuste el multiplicador **Self Weight Multiplier** en 1.
   - Agregar otra condición de carga estática llamada LIVE y asígnele el tipo LIVE como parámetro a **Type**.
2. Añada las mismas cargas correspondientes a la carga DEAD a la condición de carga LIVE. Esto significa que cada viga de la estructura tiene cargas idénticas para carga muerta y viva. (Puede revisar el ejemplo 1 para ver las instrucciones de como ingresar las una cargas).

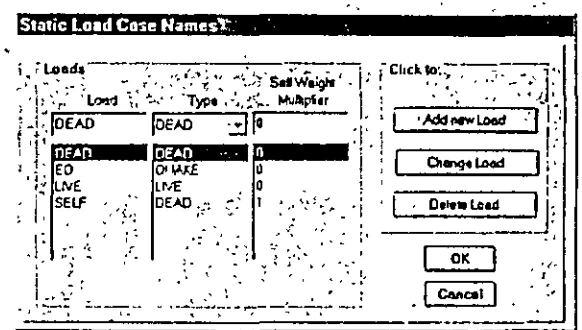


Figura 4-2 - Condiciones de Carga Estática

Definiendo un Grupo de Auto-Selección

La opción de Auto-Selección del SAP2000 es una forma muy efectiva de diseñar estructuras. Al definir un grupo de secciones denominado grupo de Auto-Selección, el programa puede diseñar cada elemento del pórtico escogiendo de entre las secciones especificadas en ese grupo. Por ejemplo se puede definir un grupo de Auto-Selección denominado COLUMNS con únicamente perfiles W14, y otro grupo llamado BEAM con perfiles W24. El programa de esta manera diseñará las secciones tipo COLUMNS empleando solamente secciones W14 y las secciones tipo BEAM empleando únicamente secciones tipo W24.

Lo primero que debemos hacer es definir un grupo de Auto-Selección que incluya únicamente secciones tipo columna. Esencialmente lo que estamos haciendo en esta etapa es darle al programa una lista de secciones de entre las cuales puede elegir al momento de diseñar los elementos del pórtico. El programa por su parte seleccionará la sección mas eficiente de entre ese grupo.

Una vez se haya concluido el diseño preliminar es momento de refinarlo, para ello los grupos BEAM y COLUMN serán reemplazados por secciones optimizadas elegidas de entre el grupo de Auto-Selección. Este proceso asignará a los elementos secciones que serán empleadas tanto en el análisis como en el diseño, esto hará mucho más fácil el cambio de secciones que necesiten ser modificadas.

Nota: ***La opción de Auto-Selección funciona solo en pórticos de acero.***

1. Desde el menú **Define** y seleccione la opción **Frame Sections**.
2. Importe a la plantilla **Frame Sections** todas las secciones de acero comprendidas entre W14x61 y W14x283.
   - Seleccione en la caja de diálogo la opción IMPORT I/WIDE FLANGE.

- Busque y seleccione la sección W14x283.
- Manteniendo presionada la tecla SHIFT haga click con el botón izquierdo del mouse sobre W14x61 y presione el botón OK. Esta operación permite seleccionar todas las secciones entre la sección W14x283 y la sección W14x61.

3. Borre de plantilla **Frame Sections** cualquier sección que pudiera estar duplicada.

   Recuerde: *No es posible borrar una sección que este en uso. De esta manera el programa asegura que todos los elementos tienen asignada secciones existentes.*

4. Desde la plantilla **Frame Section** añada una sección **Auto Select**. Esta sección se ubicará en la parte inferior de la lista **Add**.
   - Cambie el nombre en **Auto Section Name** a COLUMN.
   - De la lista **Auto Selections** elija y remueva usando el botón **Remove** todas las secciones exceptuando los perfiles W14. Esto significa que todos los elementos que tengan una sección tipo COLUMN serán diseñados empleando alguno de los perfiles W14 de entre la lista **Auto Selections** (Vea Figura 4-3).

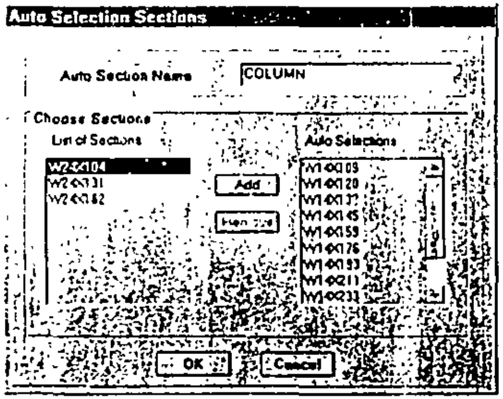


Figura 4-3 Definiendo el grupo de Auto-Selección COLUMN

5. Siguiendo las instrucciones dadas en los pasos 2 al 4:
   - Importe todas las secciones entre la W24x55 y la W24x162.
   - Asigne un grupo de Auto-Selección denominado BEAM con perfiles W24 únicamente.

6. Finalmente, selecciones todos los elementos verticales del pórtico y asígnele la sección tipo COLUMN. Luego seleccione todos los elementos horizontales y asígnele la sección tipo BEAM (Vea el ejemplo 1 para las instrucciones de como asignar secciones a los elementos del pórtico).

   Nota: *Se puede por supuesto seleccionar una sección específica tanto para el diseño como para el análisis en lugar de emplear la opción de Auto-Selección. Para ello simplemente se necesita asignar a los elementos del pórtico una sección de acero adecuada y diseñarla de acuerdo a lo descrito en el ejemplo 1. Esta sección puede ser ya sea una sección definida por el usuario o bien una sección elegida de entre las secciones predeterminadas.*

Efectuando el Análisis

Una vez que se han ingresado los datos es tiempo de correr el modelo y revisar los resultados.

1. Grabe el modelo.
2. Asigne los parámetros para el diseño entrando al menú **Analyze** y seleccionando la opción **Set Options**.
   - En la plantilla **Analysis Options** seleccione el tipo de análisis **Plane Frame** para reducir el tamaño de la solución y en consecuencia reducir el tiempo de análisis.
   - Marque la opción **Include P-Delta**.
   - Presione el botón SET P-DELTA PARAMETERS para ajustar los parámetros del análisis.
     - Asigne a **Maximum Iterations** el valor 10.
     - Incluya las condiciones de carga muerta DEAD y SELF, en la combinación P-Delta, ambas con factores de carga igual a 1.
     - Incluya la condición de carga LIVE con un factor de carga 1.

     Nota: *Los factores de carga a emplearse deben ser los correspondientes a las combinaciones de carga que se usen en el diseño de la estructura y que produzcan los máximos efectos en la misma. Deben incluirse además el efecto de las cargas laterales.*

     - El resto de valores por defecto es aceptable.
     - Presione los botones OK para aceptar los cambios y cerrar las plantillas.
3. Entre al menú **Analyze** y seleccione la opción Run para analizar la estructura.

Sugerencia *: Se puede cambiar el número de puntos a lo largo del elemento en los cuales las fuerzas de sección son reportadas. Para ello seleccione los elementos y desde el menú Assign elija la opción Frame Output Segments para cambiar el número de segmentos. Es necesario ejecutar nuevamente el análisis del modelo para obtener los resultados.*

3 Seleccionando cualesquiera de las combinaciones de carga y presionando el botón **Details** se mostrarán los resultados del análisis para ese elemento así como las ecuaciones que gobiernan su diseño de acuerdo al código empleado. (Ver Figura 4-6)

4. Al presionar el botón **ReDesign** se presentará la plantilla **Element Overwrite Assignments.** En esta plantilla se puede elegir de entre varias opciones:

Nota: *Si se efectúan cambios en la plantilla Element Overwrite Assignments empleando el botón ReDesign, será necesario presionar el botón Refresh Window de la barra de herramientas para ver los resultados del diseño actualizados en la ventana activa .*

- Seleccionar otra sección para ver el cambio en los esfuerzos en el elemento.

Nota: En el modo Auto-Selección, esta sección puede emplearse para ensamblar una nueva matriz de rigidez si se elige la opción "Update Analysis Sections". Esto último se llevará acabo una vez que se ejecute nuevamente el Diseño/Revisión.

- Clasificar los elementos por tipo **Moment Resisting Element** o **Brace.**
- Sobrescribir los factores de diseño tales como longitud efectiva y la relación de longitud no arriostrada.
- Elija la opción **Overwrite Allowable Stresses** para sobrescribir los esfuerzos admisibles empleados en el diseño de la sección.
- Cuando halla terminado de modificar los parámetros de diseño presione el botón **OK.**

Nota: ***Al cambiar la información de la plantilla ReDesign, el SAP2000 automáticamente recalculará los esfuerzos de diseño de acuerdo a la nueva información y actualizará la información en la plantilla Steel Stress Check Information. Para más instrucciones sobre como actualizar las secciones para el análisis refiérase a la sección "Re-Analizando".***

5. Para usar la sección elegida en el Re-Diseño en el siguiente análisis estructural, es necesario entrar al menú **Design** y seleccionar la opción **Update Analysis Sections.** Esta opción reemplaza las secciones empleadas inicialmente para formar la matriz de rigidez de la estructura, por las nuevas secciones dándonos mayor precisión en los cálculos.

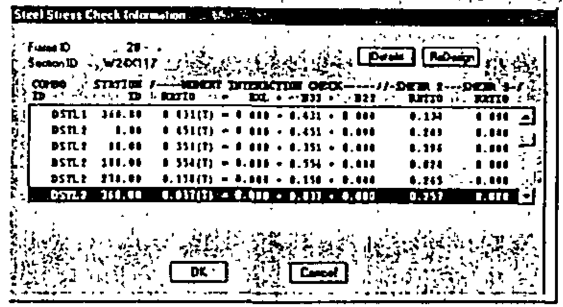


Figura 4-5 Verificación de Esfuerzos en las secciones de Acero para las Combinaciones de Carga Especificadas

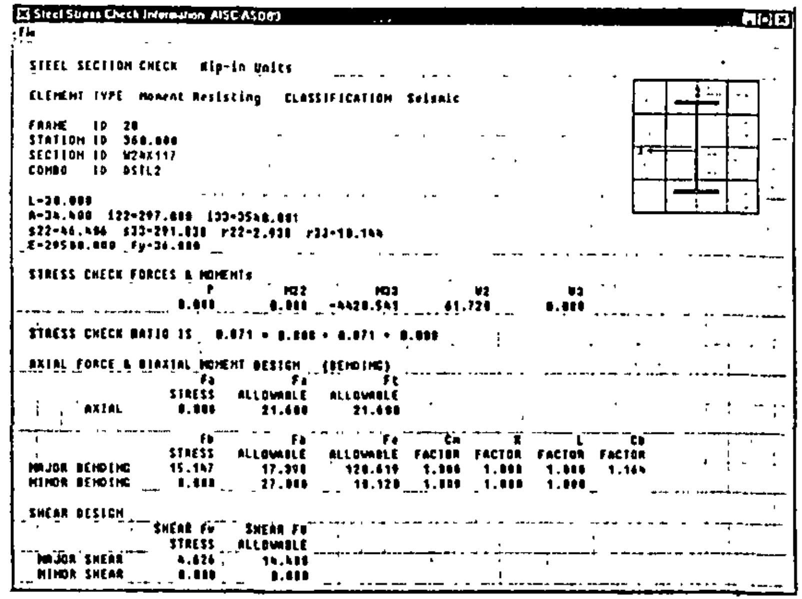


Figura 4-6 Información detallada del diseño en acero de un elemento tipo viga

1 Una vez que se halla terminado de modificar las secciones estructurales que se van a emplear, ingrese al menú Design y elija la opción **Update Analysis Sections**. Luego efectúe nuevamente el análisis empleando las últimas propiedades de sección.

2 Lleve acabo nuevamente el diseño de los elementos para ver si hay cambios.

3. Una vez que se encuentre satisfecho con las secciones elegidas, ingrese al menú **Design** y elija la opción **Replace Auto w/ Optimal Sections**. Esta opción asigna las secciones definitivas ya sean las óptimas o bien aquellas seleccionadas por el usuario tanto para el análisis como para el diseño, y reemplaza las propiedades de sección preliminares tomadas de los grupos de auto selección BEAM y COLUMN.

Diseño de acuerdo al LRFD

La metodología empleada por el LRFD es esencialmente la misma que usa el ASD. Sin embargo las combinaciones de carga así como la ecuaciones de verificación de los elementos son efectuadas empleando el código LRFD, por lo que los resultados e información resultante es distinta. Para efectuar el Diseño/Revisión de acuerdo al código LRFD, es necesario cambiar algunos parámetros de entrada.

1. Ingrese los nuevos factores de carga para el análisis P-Delta.
2. Seleccione de la plantilla Preferences el código de diseño en acero AISC-LRFD93.
3. Rediseñe las secciones de acero.

Opciones Avanzadas

Definición de Grupos de Elementos para el Diseño

Algunas veces puede encontrará útil esta opción al diseñar elementos en estructuras aporticadas. Esta opción permite diseñar todos los elementos de un grupo usando únicamente una sección. La ventaja de este método de diseño es que reduce el número de secciones diferentes a emplearse. Por ejemplo se pueden agrupar las columnas o las vigas de dos o tres pisos del pórtico dentro de un mismo grupo de diseño, permitiéndonos usar una sola sección para dicho conjunto de elementos.

1. Reasigne las secciones del tipo Auto-Selección a los elementos del pórtico.
2. Agrupe los elementos del 3er piso hacia abajo en un grupo denominado BOTTOM.
3. Agrupe los elementos entre los pisos 3 y 5 en un grupo llamado MIDDLE.
4. Asigne los elementos restante al grupo TOP.

5. Efectúe nuevamente el análisis del modelo.
6. En el menú Design seleccione la opción **Select Design Group**. En esta parte se le indica al programa que se diseñe un grupo de elementos empleando la sección más ligera que satisfaga los requerimientos y esfuerzos admisibles en todos los elementos.
   - Incluya en la lista **Design Groups** los grupos de elementos TOP, MIDDLE y BOTTOM. Al hacer esto indicamos que estos grupos serán diseñados con la sección más eficiente de las secciones del grupo Auto-Selección.

     Nota: ***Si no hay grupos en la lista "Design Group list", cada uno de los elementos de la estructura serán diseñados individualmente.***
   - Cuando presione el botón OK, El SAP2000 automáticamente diseñará las secciones de acero y mostrará los resultados en la ventana activa
7. Compare los resultados del diseño anteriormente efectuado con el diseño por grupos para ver como afecta este hecho en las secciones seleccionadas.

Zonas Rígidas

La estructura que hemos venido estudiando, ha sido analizada y diseñada considerando que los elementos se extienden completamente de nudo a nudo, sin tomar en cuenta las dimensiones propias de las secciones transversales de los elementos. Si bien es cierto que ésta no es una mala consideración, el SAP2000 permite efectuar análisis aun más precisos mediante la introducción de zonas rígidas en el modelo. Las zonas rígidas definen una región en la conexión entre viga y columna, en la cual los elementos no sufren deformaciones por flexión. Se genera así esencialmente una zona rígida en la conexión. Esta área puede ser tan grande como el usuario especifique, pero usualmente se considera igual al peralte del miembro (o una fracción del mismo) al que se está llegando en ese nudo

1. Seleccione todos los elementos del pórtico.
2. Del menú **Assign** seleccione la opción **Frame... End Offsets**.
   - En la plantilla **End Offset** seleccione la opción **Update Lengths From Current Connectivity**. Esta opción hace que el programa calcule automáticamente las dimensiones de las zonas rígidas a considerarse en cada nudo
   - Ingrese el valor 1 para **Rigid Zone Factor**. Esto significa que el 100% de la "longitud potencial" de zona rígida deberá considerarse en el análisis.
   - Presione el botón OK.
3. Si se especifica la opción **Element Shrink** de la plantilla **Set Elements**, y se observa en la pantalla activa, se podrá apreciar una serie de líneas en cada nudo que indican la asignación de zonas rígidas en los elementos.

Recuerde : *Se necesita restablecer la opción End Offsets cada vez que las secciones de los elementos sean modificadas.*

Nota: *Los momentos y cortantes en las vigas y columnas van a ser ligeramente diferentes en aquellos casos en los que no se toman en cuenta las zonas rígidas. Esto se debe a que la introducción de las mismas reduce la longitud flexible de los elementos.*

Comentarios Finales

Las herramientas de diseño del SAP2000 son muy útiles en el diseño de estructuras aporticadas. Sin embargo hay algunos puntos que se deben tener presentes:

1. Asegúrese que toda la información de diseño sea correcta. Los valores por defecto que usa el programa no son necesariamente los correctos (p.ej. K y Longitudes no Arriostradas de los elementos). Se puede usar la plantilla **Display Design Results** para ver esta información en los elementos del pórtico. De manera conveniente, es posible apreciar el análisis de las secciones al mismo tiempo que la información del diseño.
2. Verifique que las combinaciones de carga de diseño que el programa ha proporcionado sean las correctas y adecuadas para el tipo de estructura en particular que se este analizando. Sin no lo son, añada las combinaciones de carga que desea utilizar en el diseño
3. Verifique los resultados del diseño en puntos claves, para asegurarse que los resultados del diseño guardan relación con los resultados esperados.
4. Verifique que los factores de carga en el análisis P-Delta son los correctos.
5. Rediseñe la estructura toda vez que efectúe cambios en el modelo. Esto permite ver si las secciones empleadas son aún aceptables.
6. Emplee grupos de elementos para determinar el peso total de la estructura. (Revise el Ejemplo 1 para tener instrucciones de como efectuar este paso.)
7. Emplee grupos de elementos en el diseño para reducir el número de las diferentes secciones a utilizarse en la estructura.
8. El archivo *filename*.EKO contiene la información del peso total de cada uno de las secciones (perfiles) empleadas en el diseño. Esta información nos permite estimar costos de una manera preliminar.

Apéndice A – Descripción de los Iconos de la Barra de Herramientas

Icono	Nombre del Control	Permite
	New Model	Iniciar un nuevo modelo.
	Open *.SDB file	Abrir un archivo existente del SAP2000.
	Save Model	Grabar el modelo activo.
	Undo	Deshacer el último cambio.
	Redo	Revierte el último Deshacer.
	Refresh Window	Regenera la ventana activa con información actualizada.
	Lock/Unlock Model	Protege el modelo contra cambios de datos.
	Run Analysis	Efectúa el Análisis.
	Zoom	Zoom en la estructura del área determinada con el mouse.
	Restore Full View	Restaura la vista total del modelo.
	Restore Previous View	Restaura la vista anterior del modelo.
	Zoom In	Zoom in en el modelo (Acercamiento)
	Zoom Out	Zoom out en el modelo. (Alejamiento)
	Pan	Mueve dinámicamente la estructura en cualquier dirección.
	Show 3-d view	Muestra vista 3-d del modelo.
	Show 2-d View of X-Y/r-Θ Plane	Vista 2-d del modelo paralela al plano X-Y/r-Θ.
	Show 2-d View of X-Z/r-Z Plane	Vista 2-d del modelo paralela al plano X-Z/r-Z.
	Show 2-d View of Y-Z/Θ-Z Plane	Vista 2-d del modelo paralela al plano X-Z o una vista desarrollada del plano r-Z plane.
	Perspective Toggle	Muestra vista 3-d en perspectiva.
	Shrink Elements	Contrae los elementos para facilitar la visualización de la conectividad.
	Set Element	Ajusta la visibilidad de los elementos y sus propiedades.
	Up One Gridline	Visualiza el siguiente nivel superior en una malla en la vista en planta 2-d .
	Down One Gridline	Visualiza el siguiente nivel inferior en una malla en la vista en planta 2-d.

Apéndice B – Descripción de Iconos de la Barra de Herramientas Flotante

Icono	Nombre del Control	Permite
	Pointer Tool	Selecciona elementos individualmente o en cajas.
	Select All	Selecciona todos los elementos en un gráfico.
	Restore Previous Selection	Restaura elementos previamente seleccionados.
	Clear Selection	Libera todos los elementos seleccionados.
	Set Intersecting Line Select Mode	Selecciona los elementos interceptados por una línea.
	Reshape Element	Mueve elementos tomándolos en su parte central y redimensionalos seleccionando sus extremos
	Add Special Joint	Añade manualmente un nudo.
	Draw Frame Element	Dibuja un elemento tipo frame al definir la ubicación de sus nudos extremos.
	Draw Shell Element	Dibuja un elemento tipo shell al definir la ubicación de sus esquinas.
	Quick Draw Frame Element	Dibuja un elemento tipo frame usando una malla.
	Quick Draw Shell Element	Dibuja un elemento tipo shell usando una malla.
	Assign Joint Restraints	Asigna restricciones en los nudos
	Assign Frame Sections	Asigna secciones y materiales a elementos frame
	Assign Shell Sections	Asigna secciones y materiales a elementos shell.
	Assign Joint Load	Asigna cargas concentradas nodales.
	Assign Frame Span Loading	Asigna cargas en elementos tipo frame.
	Assign Shell Uniform Loading	Asigna cargas en elementos tipo shell.
	Show Undeformed Shape	Muestra la geometría original del modelo.
	Display Static Deformed Shape	Muestra la geometría deformada de la estructura.
	Display Mode Shapes	Muestra formas de modo y periodos de vibración.
	Display Element Forces/Stresses	Muestra resultados del análisis – Fuerzas y Esfuerzos.
	Set Output Table Mode	Muestra tablas con los resultados del análisis.