



**FACULTAD DE INGENIERÍA UNAM  
DIVISIÓN DE EDUCACIÓN CONTINUA**



**CA 354  
STAAD PRO PARA ANÁLISIS Y DISEÑO  
ESTRUCTURAL**

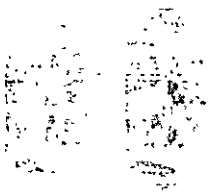
**TEMA:**

**INSTRUCTIVO PARA EL USO DEL PROGRAMA**

**COORDINADOR: ING. JOSÉ LUIS ESQUIVEL ÁVILA**

**DEL 22 AL 29 DE NOVIEMBRE DE 2004**

**PALACIO DE MINERÍA**



DIVISIÓN DE  
EDUCACIÓN  
CONTINUA



# PRÓLOGO

El programa **STAD/Pro** recientemente es uno de los programas más conocidos en el campo de la Ingeniería Estructural a nivel mundial, ha sido utilizado 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 con los manuales respectivos para uso del programa y de los módulos que lo componen.

Por lo anterior el Departamento de Estructuras de la División de Ingeniería Civil, Topográfica y Geodésica de la Facultad de Ingeniería de la UNAM, consideró conveniente iniciar una serie de cursos para enseñar a los alumnos de la carrera de Ingeniero Civil a utilizar varios programas incluyendo el módulo **STAAD** del programa **STAAD/Pro**, para ello, el contar con un instructivo que permita introducir al usuario de una manera sencilla, al programa, facilitará el objetivo anterior.

En este instructivo se pretende describir algunos de los principales elementos que intervienen en el uso del programa de computadora para Análisis y Diseño Estructural **STAAD**, cuya principal utilización será para los alumnos de la materia "Diseño Estructural" de la carrera de Ingeniero Civil, de 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 **STAAD (STAAD/Pro)**.

Se recomienda que si algunos de los elementos no son descritos ampliamente se consulten los manuales respectivos o la ayuda en línea incluida dentro del programa y se observen los ejemplos que se desarrollan al final del instructivo. Se supone que el usuario está 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.

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

# **CONTENIDO**

## **PRÓLOGO**

## **CAPÍTULO 1 EL PROGRAMA STAAD/Pro**

- 1.1 Introducción al programa **STAAD/Pro**.
- 1.2 Introducción al programa **STAAD**

## **CAPITULO 2 RECOMENDACIONES PARA USO DEL PROGRAMA**

- 2.1 Paso 1. Ejecución del programa
- 2.2 Paso 2. Tipo de estructura y datos generales
- 2.3 Paso 3. Definición de la geometría
- 2.4 Paso 4. Definición de las propiedades geométricas de los elementos
- 2.5 Paso 5. Definición las propiedades elásticas de los materiales
- 2.6 Paso 6 Tipos de fuerzas y combinaciones de carga
- 2.7 Paso 7 Elección del tipo de análisis y los resultados
- 2.8 Paso 8 Diseño de elementos

## **CAPÍTULO 3 MÓDULOS DEL PROGRAMA DESCRIPCIÓN GENERAL**

- 3.1 Ejecución del programa, módulos que lo componen.
- 3.2 Descripción general.

## **CAPÍTULO 4 GENERACIÓN DE LA ESTRUCTURA**

- 4.1 Generación de la geometría
- 4.2 Asignación de propiedades geométricas

4.3 Definición y asignación materiales

4.4 Tipos de apoyo

4.5 Asignación de fuerzas, condiciones y combinaciones de carga

4.6 Opciones de análisis, selección de resultados

## **CAPÍTULO 5 ANÁLISIS DE LA ESTRUCTURA**

## **CAPÍTULO 6 VER RESULTADOS**

6.1 Introducción

6.2 Ver estructura deformada

6.3 Ver diagramas de elementos mecánicos

## **CAPÍTULO 7 EJEMPLOS E INTERPRETACIÓN DE RESULTADOS**

Ejemplo No. 1

Ejemplo No. 2

Ejemplo No. 3

Ejemplo No. 4

Ejemplo No. 5

## **CAPÍTULO 8 COMENTARIOS FINALES**

# EL PROGRAMA STAAD/Pro

# CAPÍTULO 1

## 1.1 Introducción al programa STAAD/Pro

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 más eficientemente y entre otras cosas facilitarnos la posibilidad de explorar varias alternativas de solución de problemas estructurales o bien considerar más variables en el modelo de las estructuras con el objeto de lograr un mejor entendimiento comportamiento de la estructura.

Tomando en cuenta lo anterior, **STAAD/Pro** 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 al programa que, junto con la relativa sencillez y facilidad de uso son algunas de sus principales características.

**STAAD/Pro** consta básicamente de una serie de módulos (véase figura 1), de ellos, en este instructivo se describirá sólo el módulo **STAAD**, en éste, el usuario puede seleccionar diversas opciones para poder 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.

**STAAD/Pro**, la siguiente generación del programa **STAAD-III**, es el principal software para Análisis y Diseño Estructural de Research Engineers. En **STAAD/Pro**, el enfoque principal está en la productividad. **STAAD/Pro** dirige el proceso completo de la Ingeniería Estructural, desde el desarrollo del modelo hasta el análisis, diseño, bosquejo y detallado de componentes estructurales, **STAAD/Pro** se diseñó para trabajar de manera similar a como se hace en un despacho de Proyecto Estructural.

**STAAD/Pro** es el ambiente de funcionamiento nativo con una ventana para la selección de los componentes que lo constituyen, permitiendo la construcción del modelo así como la visualización y comprobación de resultados. **STAAD/Pro** es el paquete principal con varios componentes optativos, que consisten en lo siguiente:

**STAAD** proporciona el análisis estructural y el diseño integrado de acero, concreto y madera

**STARDYNE** proporciona características avanzadas de análisis. Construido alrededor de una biblioteca de elemento finito, **STARDYNE** proporciona poderosas opciones de análisis Dinámico, Sísmico, No-lineal, por temperatura, pandeo y otras capacidades avanzadas de análisis.

El ambiente **FEMkit** ofrece modelación de Elemento Finito orientada gráficamente, se complementa con tecnologías para generación de mallas 2D y 3D y herramientas poderosas para la comprobación del modelo.

**Visual DRAW** permite la generación de planos, elevaciones, secciones y dibujos de detalle. Totalmente integrado en el ambiente **STAAD/Pro**, **Visual DRAW** proporciona la generación de dibujos, con capacidades de edición y ploteo.

Los módulos siguientes también están disponibles como componentes de **STAAD/Pro**

**STAAD.etc** es una colección de módulos de diseño de componentes estructurales, le permite al ingeniero completar el proyecto diseñando cimentaciones, muros de retención, mampostería, conexiones y otros componentes estructurales de utilidad.

**FabriCAD** es una herramienta integrada que realiza el detallado de acero, cálculos de fabricación y montaje, así como la generación de dibujos.

El componente **ADLPIPE** ofrece un sistema confiable para modelado y análisis. Este componente ofrece una solución completa para diseño de plantas industriales.

Poderoso y comprensivo, **STAAD/Pro** está basado en un diseño orientado a objetos que utiliza la tecnología MFC (Microsoft Foundation Class), aprovechando la computación de 32 bits. Una base de datos relacional, con enlaces OLE y DDE, permite intercambio de información entre múltiples aplicaciones integradas con todo el software basado en Windows.

## 1.2 Introducción al programa STAAD

El Sistema **STAAD/Pro** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis y Diseño de Estructuras bajo uno o más sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura proporcionando, después del análisis, los desplazamientos de los nudos, elementos mecánicos, reacciones, formas modales y resultado del diseño.

STAAD fue desarrollado básicamente 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 estructuras utilizando elementos placa y sólido (elemento finito).

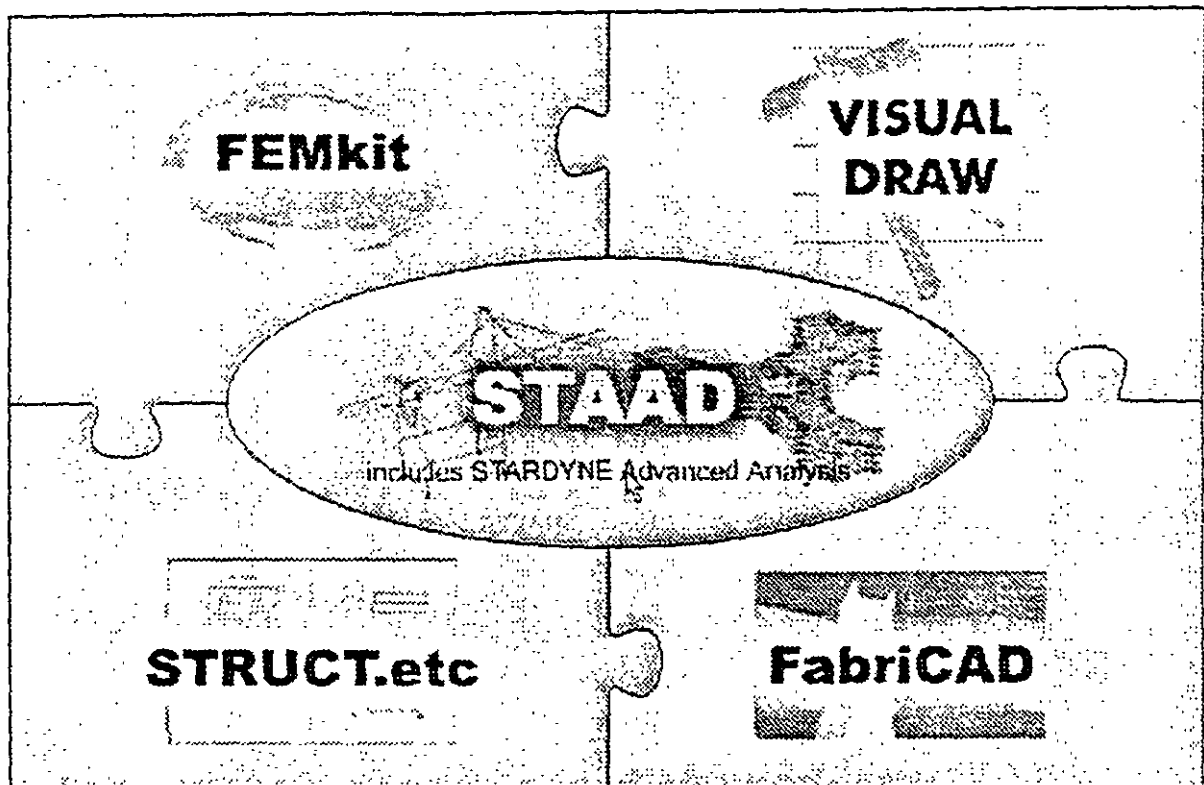


Figura 1.1 STAAD/Pro, programa principal y sistemas que lo integran

Una de las principales características del programa es la interacción que se puede establecer entre éste y el usuario, sin embargo, 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. STAAD es un programa orientado a eventos (seleccionar un elemento con el mouse, 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, 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 o generar la estructura, en el capítulo

**STAAD** proporciona el análisis estructural y el diseño integrado de acero, concreto y madera.

**STARDYNE** proporciona características avanzadas de análisis. Construido alrededor de una biblioteca de elemento finito, **STARDYNE** proporciona poderosas opciones de análisis Dinámico, Sísmico, No-lineal, por temperatura, pandeo y otras capacidades avanzadas de análisis.

El ambiente **FEMkit** ofrece modelación de Elemento Finito orientada gráficamente, se complementa con tecnologías para generación de mallas 2D y 3D y herramientas poderosas para la comprobación del modelo.

**Visual DRAW** permite la generación de planos, elevaciones, secciones y dibujos de detalle. Totalmente integrado en el ambiente **STAAD/Pro**, **Visual DRAW** proporciona la generación de dibujos, con capacidades de edición y ploteo.

Los módulos siguientes también están disponibles como componentes de **STAAD/Pro**

**STAAD.etc** es una colección de módulos de diseño de componentes estructurales, le permite al ingeniero completar el proyecto diseñando cimentaciones, muros de retención, mampostería, conexiones y otros componentes estructurales de utilidad.

**FabriCAD** es una herramienta integrada que realiza el detallado de acero, cálculos de fabricación y montaje, así como la generación de dibujos.

El componente **ADLPIPE** ofrece un sistema confiable para modelado y análisis. Este componente ofrece una solución completa para diseño de plantas industriales.

Poderoso y comprensivo, **STAAD/Pro** está basado en un diseño orientado a objetos que utiliza la tecnología MFC (Microsoft Foundation Class), aprovechando la computación de 32 bits. Una base de datos relacional, con enlaces OLE y DDE, permite intercambio de información entre múltiples aplicaciones integradas con todo el software basado en Windows.

## 1.2 Introducción al programa STAAD

El Sistema **STAAD/Pro** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis y Diseño de Estructuras bajo uno o más sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura proporcionando, después del análisis, los desplazamientos de los nudos, elementos mecánicos, reacciones, formas modales y resultado del diseño.



**STAAD** proporciona el análisis estructural y el diseño integrado de acero, concreto y madera.

**STARDYNE** proporciona características avanzadas de análisis. Construido alrededor de una biblioteca de elemento finito, **STARDYNE** proporciona poderosas opciones de análisis Dinámico, Sísmico, No-lineal, por temperatura, pandeo y otras capacidades avanzadas de análisis.

El ambiente **FEMkit** ofrece modelación de Elemento Finito orientada gráficamente, se complementa con tecnologías para generación de mallas 2D y 3D y herramientas poderosas para la comprobación del modelo.

**Visual DRAW** permite la generación de planos, elevaciones, secciones y dibujos de detalle. Totalmente integrado en el ambiente **STAAD/Pro**, **Visual DRAW** proporciona la generación de dibujos, con capacidades de edición y ploteo.

Los módulos siguientes también están disponibles como componentes de **STAAD/Pro**.

**STAAD.etc** es una colección de módulos de diseño de componentes estructurales, le permite al ingeniero completar el proyecto diseñando cimentaciones, muros de retención, mampostería, conexiones y otros componentes estructurales de utilidad.

**FabriCAD** es una herramienta integrada que realiza el detallado de acero, cálculos de fabricación y montaje, así como la generación de dibujos.

El componente **ADLPIPE** ofrece un sistema confiable para modelado y análisis. Este componente ofrece una solución completa para diseño de plantas industriales.

Poderoso y comprensivo, **STAAD/Pro** está basado en un diseño orientado a objetos que utiliza la tecnología MFC (Microsoft Foundation Class), aprovechando la computación de 32 bits. Una base de datos relacional, con enlaces OLE y DDE, permite intercambio de información entre múltiples aplicaciones integradas con todo el software basado en Windows.

## 1.2 Introducción al programa STAAD

El Sistema **STAAD/Pro** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis y Diseño de Estructuras bajo uno o más sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura proporcionando, después del análisis, los desplazamientos de los nudos, elementos mecánicos, reacciones, formas modales y resultado del diseño.

**STAAD** fue desarrollado básicamente 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 estructuras utilizando elementos placa y sólido (elemento finito).

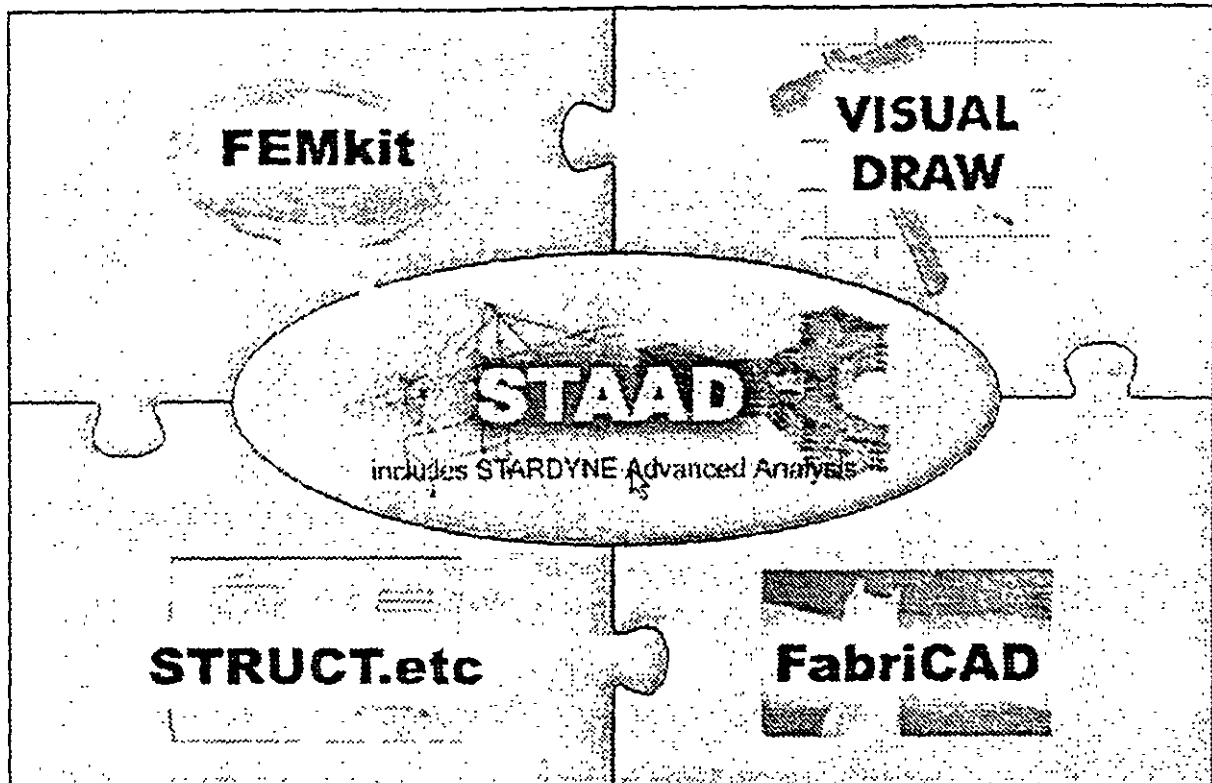


Figura 1.1 STAAD/Pro, programa principal y sistemas que lo integran.

Una de las principales características del programa es la interacción que se puede establecer entre éste y el usuario, sin embargo, 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. **STAAD** es un programa orientado a eventos (seleccionar un elemento con el mouse, 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, 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 o generar la estructura, en el capítulo

5 se presentan las opciones de análisis, en el capítulo 6 se muestran las opciones para ver resultados del Análisis, el capítulo 7 contiene algunos ejemplos con la correspondiente interpretación de los resultados obtenidos por el programa **STAAD**, por último, en el capítulo 8 se incluyen algunos comentarios y sugerencias finales

# RECOMENDACIONES PARA EL USO DEL PROGRAMA

# CAPITULO 2

## 2.1 Ejecución del programa

Una vez instalado, para iniciar la ejecución del programa **STAAD**, se puede hacer clic en inicio luego deslizar el puntero del ratón hasta programas, enseguida desplazarlo a la derecha y hacia abajo hasta la carpeta **STAAD/Pro** y por último a la derecha y hacia arriba (en la computadora donde se preparó este instructivo), para finalmente hacer clic en **STAAD** (véase figura 2.1), con lo cual aparece la ventana de la figura 1.1, después de hacer clic en su zona central (**STAAD**) se muestra la ventana de la figura 2.2.

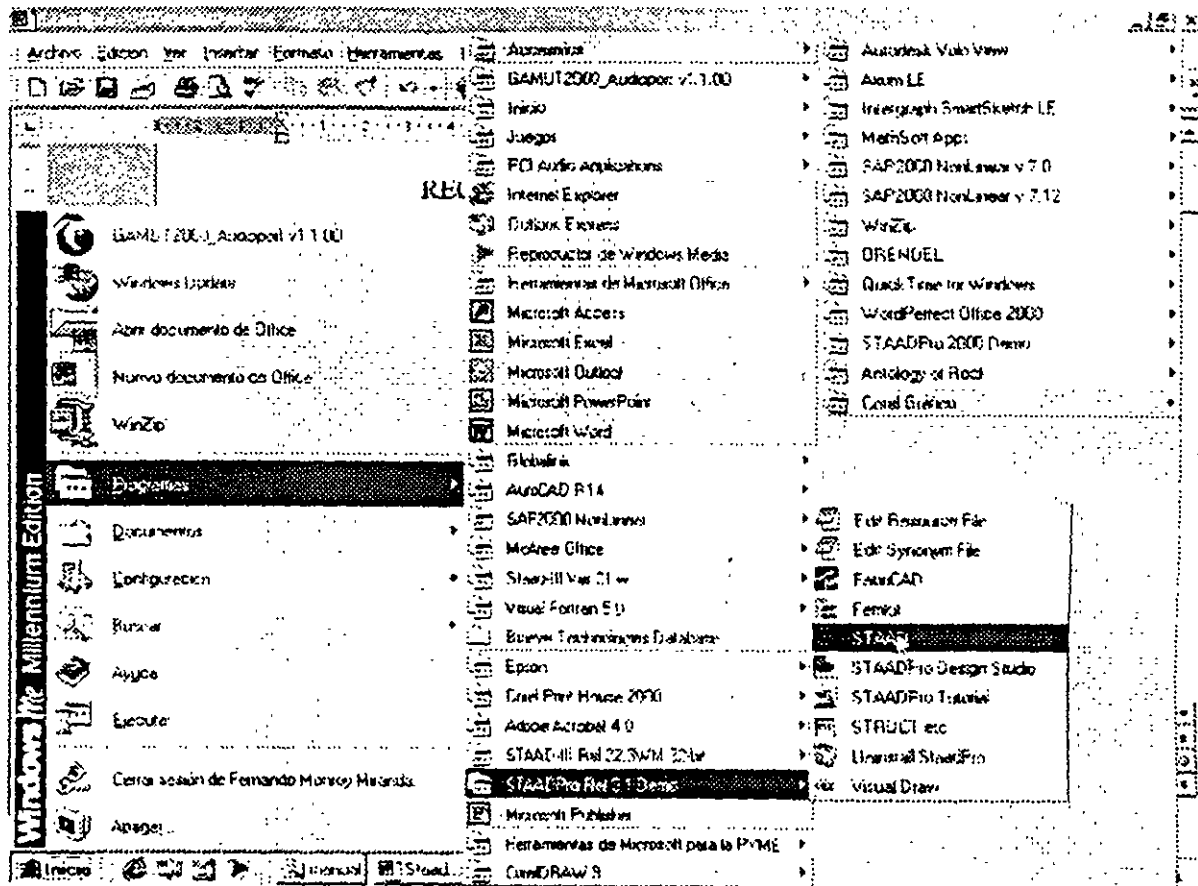


Figura 2.1 Ejecución del programa STAAD/Pro

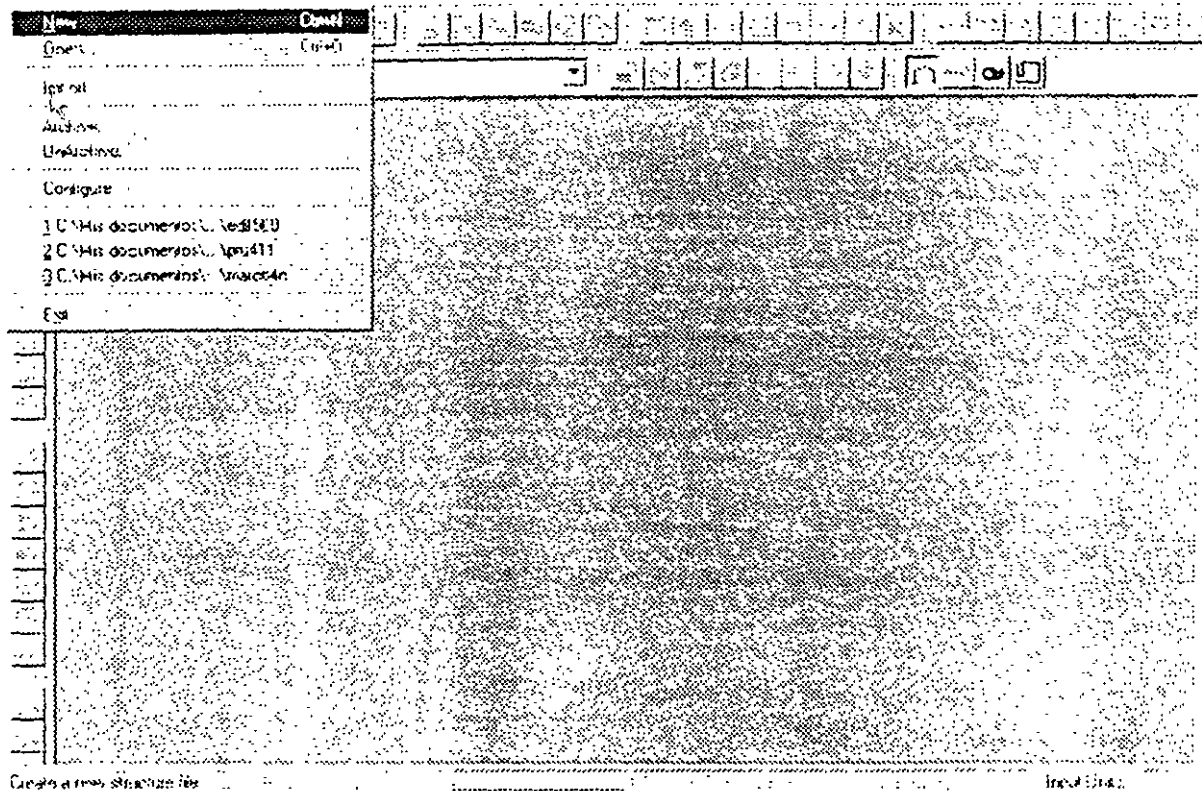


Figura 2.2 Inicio del programa STAAD.

## 2.2 Tipo de estructura y datos generales

Para iniciar la introducción de datos generales y el tipo de estructura por analizar se utiliza la opción **N**ew del menú **F**ile (véase figura 2.2) mostrándose la ventana de la figura 2.3

**STAAD** permite manejar la estructura a analizar como una de las siguientes:

- Truss**
- Plane**
- Floor**
- Space**

Para el caso de la estructura tipo **Truss** (armadura) esta puede ser plana o en 3 dimensiones (3D) en ambos casos en el análisis sólo se considerará el efecto axial

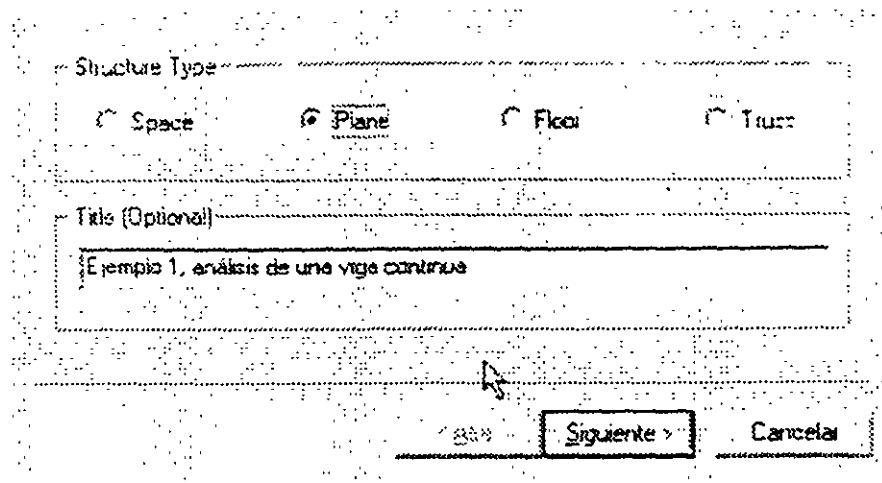


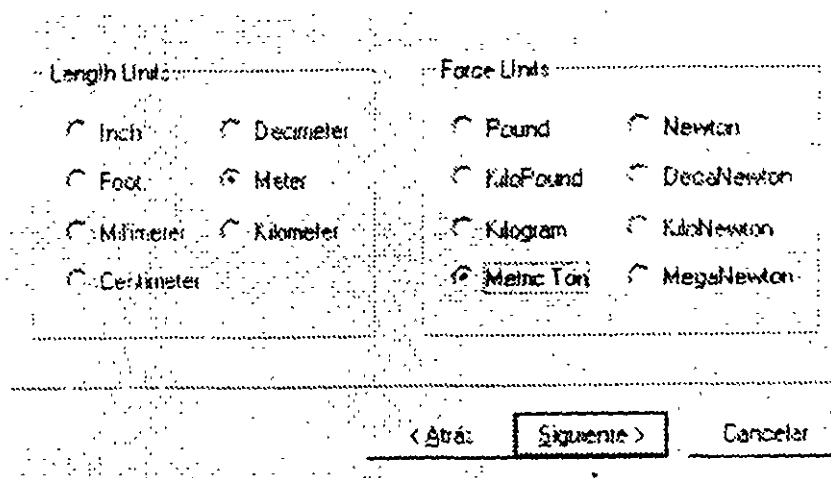
Figura 2.3 Datos generales al inicio del programa STAAD.

En la estructura tipo **Plane** se consideran cortante y axial en el plano de la estructura y flexión perpendicular a ese plano.

El tipo **Floor** permite analizar estructuras con acciones perpendiculares a su plano (retículas) considerando flexión en el plano, torsión, y cortante

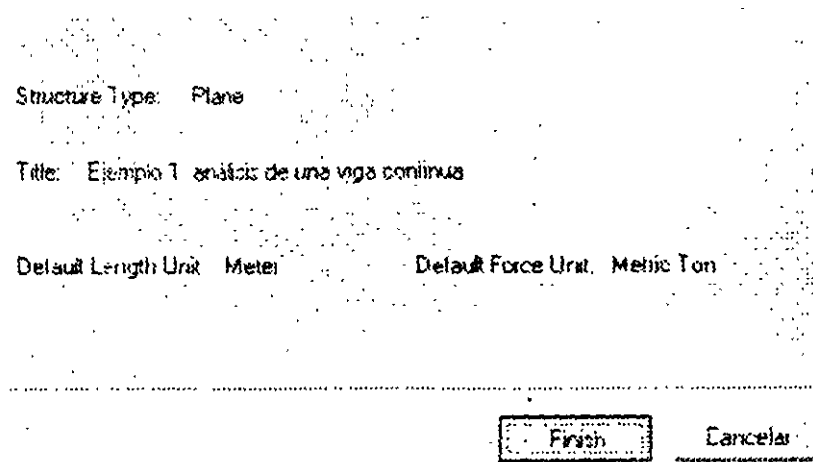
El caso general lo constituye el tipo **Space** en donde se consideran flexión y cortante en dos direcciones, torsión y axial, y seis grados de libertad por nudo, desde luego que se pueden liberar extremos de las barras a algún elemento mecánico y suprimir o ligar grados de libertad (por ejemplo diafragma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

La opción que corresponda a la estructura por analizar, y la introducción de un título (opcional) como identificación que se incluirá dentro del archivo de datos, se realiza en la ventana de la figura 2.3, una vez introducidos los datos y seleccionado el tipo de estructura y después de hacer clic en el cuadro **Siguiente** se muestra la ventana de la figura 2.4, en donde han de seleccionarse las unidades para las fuerzas y longitudes de los datos de la estructura que se introducirán posteriormente (geometría, propiedades, cargas, etc.)



**Figura 2.4** Datos de unidades al inicio del programa STAAD

Una vez seleccionadas las unidades se hace clic en Siguiente para que se despliegue la ventana de la figura 2.5, finalmente, Finish conduce a la ventana de la figura 2.6 que es la ventana o módulo principal de STAAD.



**Figura 2.5** Datos seleccionados por el usuario al inicio del programa STAAD.

Obsérvese que en esta ventana (figura 2.6), en general, están contenidos algunos elementos típicos de varios programas desarrollados para ambiente o plataforma **Windows**, es decir, una barra de título (extremo superior de la ventana), una de menús desplegables (File, Edit, View, etc.), barras de iconos (algunos típicos de varios programas, y otros propios STAAD), una barra de estado en el extremo inferior de la ventana (for help press.....). En el extremo izquierdo se muestran algunos iconos y varias opciones agrupadas por categorías (Job, Setup, Geometry, etc.), seguidas por un área con fondo blanco que se utilizará para desplegar gráficamente la geometría y algunas características de la estructura

(apoyos, cargas, etc.), el área restante (a la derecha de la anterior) la utiliza el programa STAAD para mostrar, generalmente, información de los elementos de la estructura en forma numérica (coordenadas de los nudos, incidencias de las barras, fuerzas, etc.).

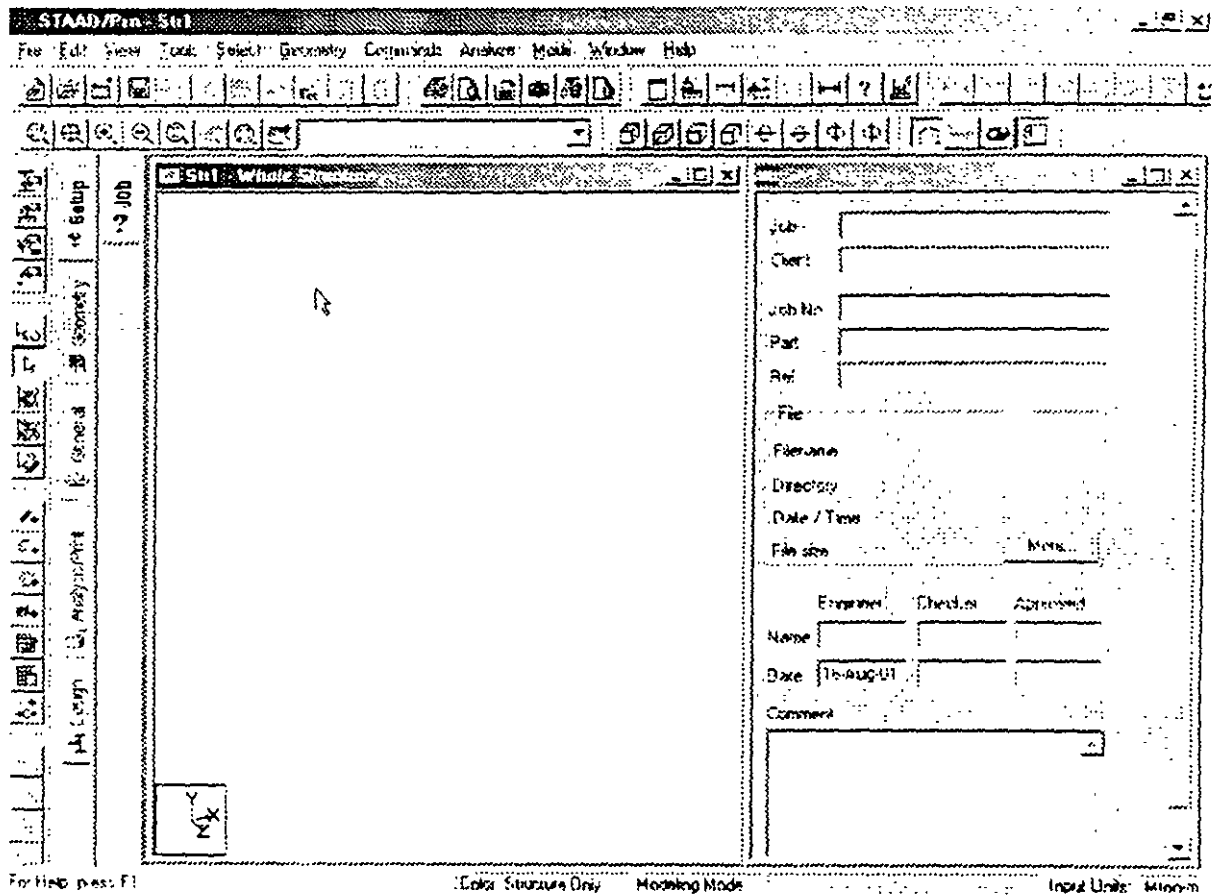


Figura 2.6 Ventana completa del programa STAAD

### 2.3 Definición de la geometría

Antes de iniciar la ejecución del programa **STAAD** es conveniente tener completamente bien definida la geometría del modelo. La estructura por analizar se idealizará mediante una serie de elementos estructurales conectados entre si, los cuales, de acuerdo a sus características o con fines de análisis se podrán modelar como elementos barra (trabes, columnas, diagonales, etc.), elementos finitos placa (losas, muros) o elementos finitos sólidos (elementos tridimensionales), 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 o ubicación de los elementos (barra, placa, sólido) se logra localizando sus nudos extremos, por ejemplo, en un sistema coordenado cartesiano. Proporcionando las coordenadas de esos nudos (o su longitud si es que el elemento barra es paralelo a alguno de los ejes de referencia) así como los nudos extremos (incidencias) de la barra queda definida su posición.

No es necesario numerar los nudos que forman parte de la estructura ya que el programa lo hace de manera automática. Es conveniente localizar nudos en donde se tenga cambio de propiedades geométricas o elásticas, recordando que el elemento barra requiere de dos nudos para posicionarlo, el elemento placa 3 ó 4 y el sólido desde 4 hasta 8 nudos (véase figura 2.7).

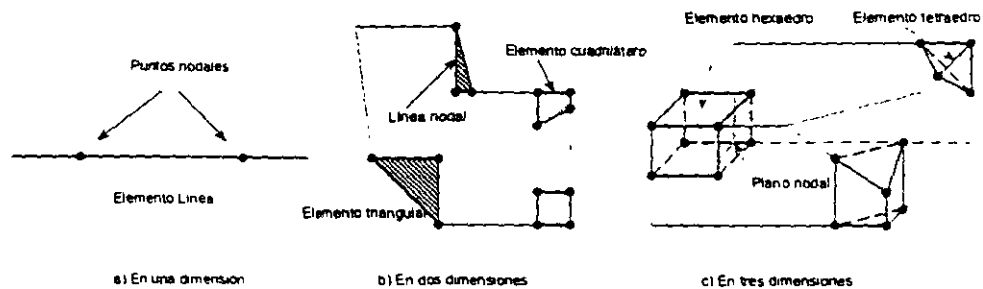


Figura 2.7 Tipos de elementos y nudos que los definen

## 2.4 Definición de las propiedades geométricas de los elementos

Los siguientes son algunos de los tipos de elementos barra que permite manejar STAAD.

- Prismáticos (rectangular, circular, etc.).
- Elementos estándar de acero
- Elementos de acero definidos por el usuario.
- Sección I de peralte variable
- Asignarles una forma específica.

Para elementos barra prismáticos de forma arbitraria se requiere proporcionar las siguientes propiedades referidas a ejes locales y centroidales de la barra.

- AX = Área de la sección transversal.
- IX = Constante de torsión.
- IY = Momento de inercia al rededor del eje y.
- Iz = Momento de inercia al rededor del eje z

$AY$  = Área de cortante en dirección y.  
 $AZ$  = Área de cortante en dirección z.  
 $YD$  = Dimensión de la sección en dirección y.  
 $ZD$  = Dimensión de la sección en dirección z.

Para barras de sección trapezoidal o T el significado de  $YB$  y  $ZB$  se muestra en la figura 2.8.

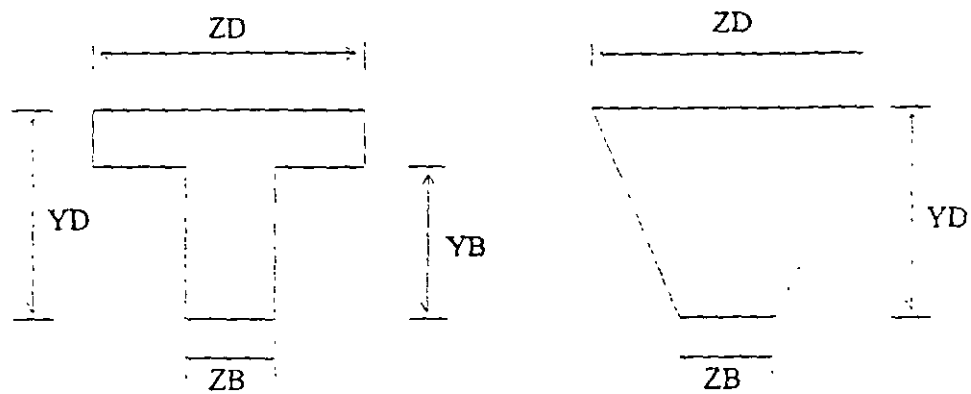


Figura 2.8 Características de secciones T y trapezoidal

Si al programa se le solicita el cálculo de esfuerzos o el diseño (revisión) en concreto o acero será necesario proporcionar los valores de  $YD$  y  $ZD$  en caso contrario se pueden omitir.

Si no se proporcionan las áreas de cortante el programa no considera ese efecto en el análisis, esto sólo es posible definiendo a las barras de tipo "General" e introduciendo los valores de sus propiedades.

Para secciones específicas (rectangular, circular, etc.) las propiedades son obtenidas por el programa sólo con proporcionar las dimensiones características según la forma de la sección transversal de la barra (p.ej.  $B$  y  $D$  para la sección rectangular,  $D$  para la circular, etc.) en este caso serán considerados los efectos de deformación por cortante.

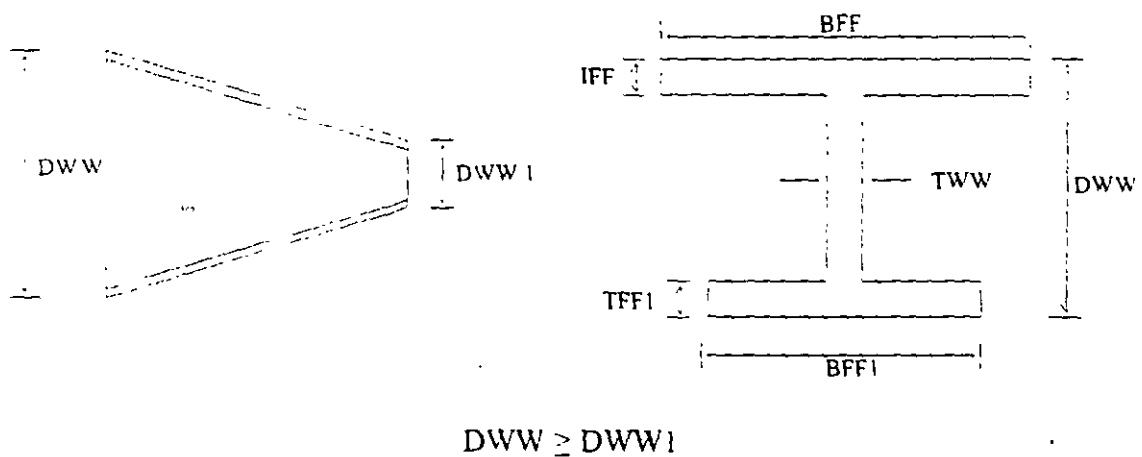
Dependiendo del tipo de estructura, en la tabla 2.1 se muestran las propiedades geométricas mínimas que es necesario proporcionar para que el análisis se pueda realizar.

Tipo de estructura	Propiedad geométrica 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 para el análisis.

El programa **STAAD** permite asignar las propiedades geométricas de los elementos barra de acuerdo a una tabla de perfiles de acero estándar ( P.ej tabla AISC) o tomarlas de una tabla definida por el usuario.

En el caso de secciones I de peralte variable los datos son los que se muestran en la figura 2.9.



**Figura 2.9** Características de la sección I de peralte variable.

Al programa se le pueden dar instrucciones para que, de manera automática, maneje a los elementos con secciones de formas específicas (sección T, o formada por uno o dos ángulos, etc.).

Para el caso de los elementos placa será necesario proporcionar el espesor de la placa en cada esquina, para el sólido no es necesario proporcionar propiedades geométricas sólo constantes elásticas

## 2.5 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 y sólido) como son E (Módulo elástico), y  $\mu$  (relación de Poisson) y, mediante la siguiente expresión se obtiene el modelo de rigidez a cortante

$$G = \frac{E}{2(1 + \mu)}$$

Para incluir el peso propio es necesario proporcionar el peso volumétrico, si se consideran efectos de temperatura será necesario especificar el coeficiente lineal de dilatación térmica

## 2.6 Tipos de fuerzas y combinaciones de carga

Es necesario tener completamente identificados 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, contar con las características de las fuerzas (tipo, magnitud, dirección, sentido y punto de aplicación) que componen cada sistema de fuerzas (condición de carga).

Por ejemplo, una condición de carga puede ser la carga muerta, que puede estar formada por fuerzas uniformes en algunas barras simulando el peso, por ejemplo, de los muros divisorios, o fuerzas concentradas que representan, por ejemplo, el peso de tanques, etc. Otra condición de carga, el sismo, puede ser representado por una serie de fuerzas estáticas (sismo estático) o dinámicas aplicadas a 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, pasillos, escaleras, etc.).

Los sistemas de carga independientes o primarios (como los llama el programa) pueden ser utilizados para formar sistemas de carga dependientes de los anteriores, 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 incluyan así como su participación respectiva (factor de carga). Por ejemplo, teniendo como marco de referencia al Reglamento de Construcciones para el D.F. pensando en una estructura del grupo A, localizada en el D.F. una combinación será 1.5 de la carga muerta + 1.5 de la carga viva máxima, por lo que el factor de carga o participación de las condiciones anteriores 1 y 2 es 1.5, siendo 1 y 2 las condiciones de carga respectivas (1 la carga muerta y 2 la viva).

## 2.7 Elección del tipo de análisis y los resultados

**STAAD** permite realizar un análisis elástico lineal de 1<sup>er</sup> orden y también de 2<sup>do</sup> orden, en el segundo caso se consideran efectos P- $\Delta$ , o un análisis no lineal por geometría en cuanto a considerar la geometría deformada de la estructura, por lo anterior habrá que decidir 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 resultados de diseño (revisión), y de que elementos se requieren, por ejemplo: algunos o todos los nudos, algunos o todos los elementos (barras, placas, etc.). Gráficas de la deformada, de algún marco o de toda la estructura, etc. Lo anterior se tendrá que especificar para una, algunas o todas las condiciones de carga y/o combinaciones. Si el usuario no selecciona o define los elementos (nudos, barras, etc.) y las condiciones y/o combinaciones, la impresión la realiza para todos los elementos y todos los sistemas de fuerza existentes

## 2.8 Diseño de elementos

**STAAD** permite diseñar o revisar elementos de acero, concreto y madera por lo que será necesario especificar un código aplicable a utilizar (ACI, AISC, LRFD, ASSTHO, etc.), así como proporcionar los valores de los parámetros a utilizar ( $f'_c$ ,  $f_y$ , etc.), e indicar los elementos que se diseñarán y el criterio a seguir para su diseño (viga, columna, etc.).

## MÓDULOS DEL PROGRAMA DESCRIPCIÓN GENERAL

# CAPÍTULO 3

### 3.1 Introducción

Para poder introducir y/o hacer cambios a los datos o características de la estructura el programa **STAAD**, además de contar con un editor en línea modo texto, principalmente tiene un editor gráfico integrado desde donde también se puede invocar al editor modo texto. Casi con cualquiera de los dos editores se puede:

- Manejar (Definir, mover, copiar, borrar, etc.) elementos estructurales (nudos, barras, placas sólidos)
- Especificar tipos de apoyo (fijo o con grados de libertad, resortes, apoyos inclinados, tipo "Foundation", etc.).
- Asignar propiedades geométricas de los elementos barra de acuerdo a: una tabla de perfiles estándar (AISC por ejemplo), una tabla previamente definida por el usuario, secciones prismáticas (circular rectangular, Te, trapezoidal, I de peralte constante o con variación lineal etc.), o introducir sus características particulares (propiedades geométricas, orientación de su sección transversal, etc.).
- Especificar espesores de los elementos placa.
- 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. 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, concreto, etc.) o se pueden introducir valores particulares.
- Especificar que elementos desempeñarán sólo una función estructural específica por ejemplo cable, barra en compresión, en tensión, armadura (tensión o compresión), con articulación o liberación a algún elemento mecánico en un extremo, ignorarlos y otras opciones. También se puede definir diafragmas rígidos.
- Definir cargas variables (móviles) pudiendo ser definidas por el usuario (tren de cargas concentradas), de acuerdo a AASHTO(HS20, HS15, H20, H15) o bien tomadas de un archivo externo.

- Especificar fuerzas definidas en el tiempo (fuerza-tiempo o aceleración-tiempo) tomando los valores de un archivo existente o introduciéndolos de acuerdo a una función (seno o coseno) proporcionando características dinámicas (amplitud y frecuencia), definiendo el lapso de tiempo de actuación de la fuerza así como también considerar el amortiguamiento.
- Definir características para generar cargas definidas por el UBC (Uniform Building Code).
- Definir cargas de viento especificando (hasta cinco) intensidades (presiones) actuando respectivamente en n alturas.
- Especificar fuerzas estáticas aplicadas a los nudos, desplazamientos prescritos de los apoyos, peso propio, etc. Para barras: fuerzas y/o momentos uniformes, fuerzas y/o momentos concentrados, fuerzas con variación lineal, presión hidrostática. Para los elementos placa: presión uniforme, lineal, hidrostática.
- Asignar carga uniforme por unidad de área en un nivel específico y en cierta área.
- Incluir en las barras, presfuerzo, incrementos de temperatura y ajustes en la longitud inicial de los elementos.
- Seleccionar el tipo de análisis como puede ser: elástico-lineal de primer orden, análisis no lineal P- $\Delta$ , análisis de segundo orden (especificando el número de iteraciones) y análisis dinámico
- Y otras opciones más.

### 3.2 Descripción general

En la figura 3.1 se muestra la ventana deslizable correspondiente a la opción o menú File

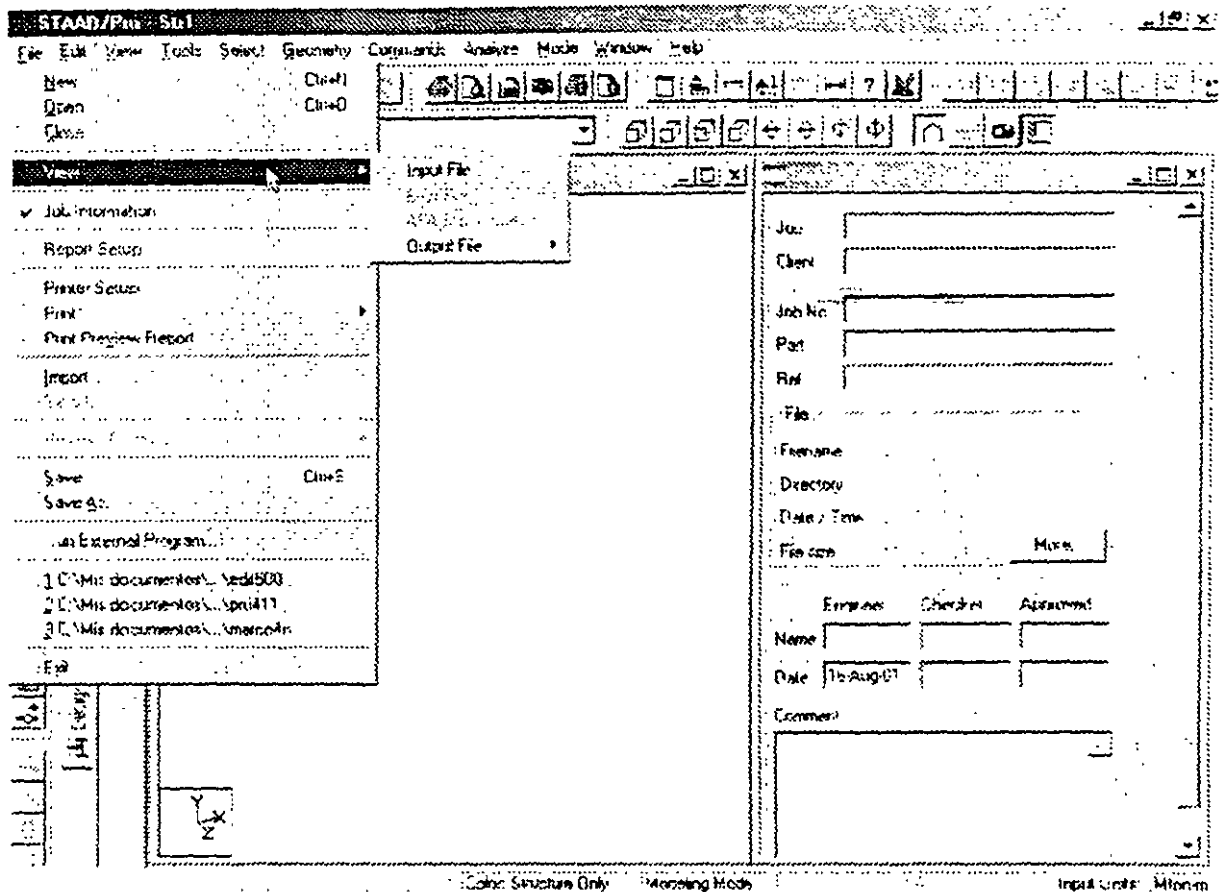


Figura 3.1 Menú File de STAAD.

Algunas de las opciones del menú File permiten

- New                      Iniciar un problema nuevo
- Open                    Abrir un archivo existente con datos de alguna estructura.
- View                    Ver el contenido del archivo de datos: (**Input File**) o el archivo de resultados (**Output File**)
- Printer Setup**        Seleccionar una impresora o bien modificar sus propiedades.
- Print Input File**     Imprimir el contenido de un archivo de datos.
- Preview Print Input** Ver el contenido del archivo de datos antes de imprimir.
- Save, Save As**       Permiten guardar el archivo de datos.



**Exit** Cerrar el programa

Existen, dentro del menú anterior, otras opciones que pueden ser de uso no muy frecuente.

Ahora en la figura 3.2 se presentan las opciones del menú **E**dit

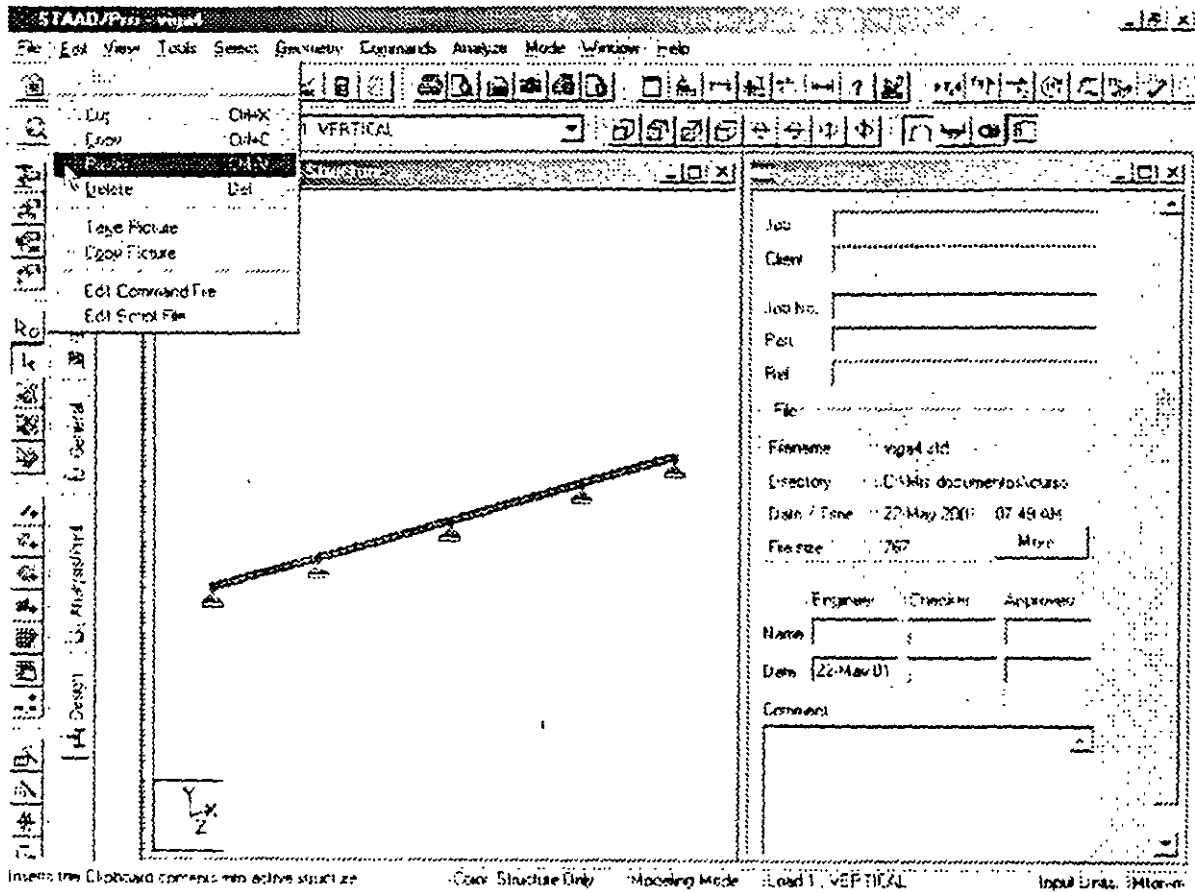


Figura 3.2 Menú **E**dit del módulo STAAD.

Las opciones del menú **E**dit permiten.

**Undo** Deshacer la acción anterior (última).

**Cut** Suprimir(borrar) los elementos seleccionados de la estructura (p.ej barras que aparecen en color en el área de dibujo) y los coloca en la memoria temporal.

- Copy** Copia a la memoria temporal los elementos seleccionados de la estructura (para poder insertarlos posteriormente), esta opción no borra a los elementos
- Paste** Insertar los elementos almacenados en la memoria temporal.
- Del** Borra los elementos seleccionados de la estructura.
- Edit command file** Ejecuta el editor modo texto mostrando el contenido del archivo de datos al que pueden realizársele cambios (añadir comandos o datos, suprimir o modificar parte de la información etc.).

## GENERACIÓN DE LA ESTRUCTURA

## CAPITULO 4

### 4.1 Generación de la geometría

Una vez iniciado STAAD/Pro aparece la ventana que se muestra en la figura 4.1, en donde se muestra la mayoría del contenido (iconos, menús) en gris, la mayoría de los cuales se activará una vez seleccionado el archivo de trabajo que está en el menú **FILE**, se elige New mostrándose la ventana de la figura 4.2, en donde es necesario especificar el tipo de estructura que se va a analizar teniéndose las siguientes opciones:

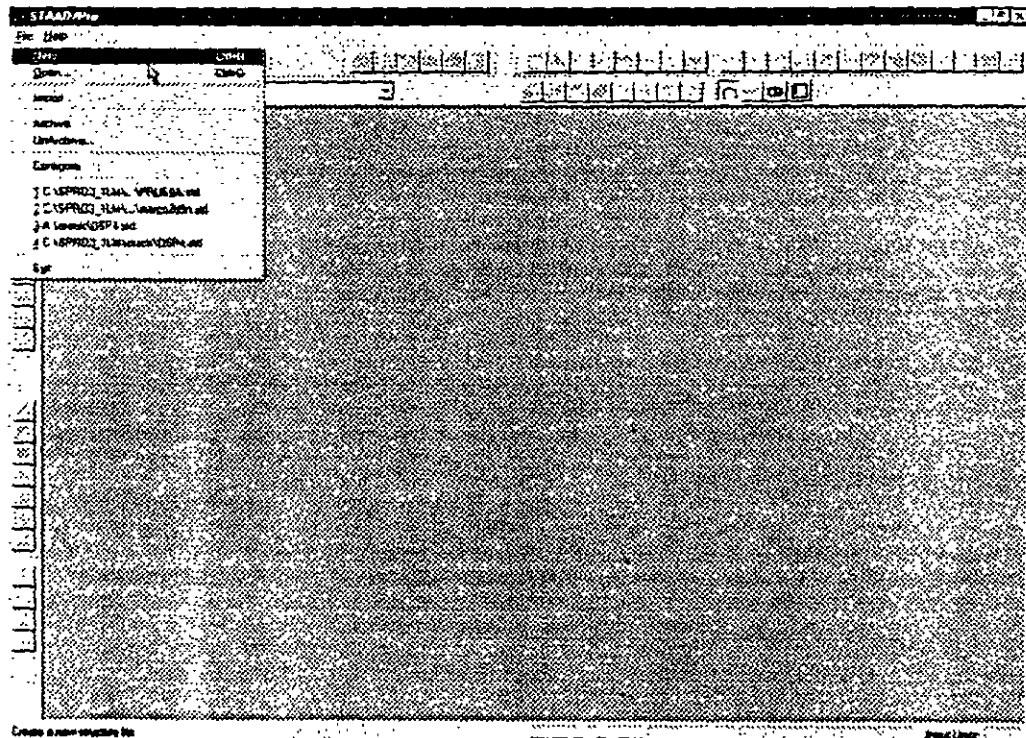


Figura 4.1 Ventana al iniciar STAAD/Pro

**SPACE-** Para estructuras tridimensionales considerando efectos de axial, torsión, flexión, en 2 direcciones y cortante también en 2 direcciones.

**PLANE-** Para cuando la estructura se puede considerar contenida en un plano (X, Y), entonces se le consideran efectos de flexión, cortante y axial.

# GENERACIÓN DE LA ESTRUCTURA

# CAPÍTULO 4

## 4.1 Generación de la geometría

Una vez iniciado STAAD/Pro aparece la ventana que se muestra en la figura 4.1, en donde se muestra la mayoría del contenido (iconos, menús) en gris, la mayoría de los cuales se activará una vez seleccionado el archivo de trabajo que está en el menú **FILE**, se elige New mostrándose la ventana de la figura 4.2, en donde es necesario especificar el tipo de estructura que se va a analizar teniéndose las siguientes opciones:

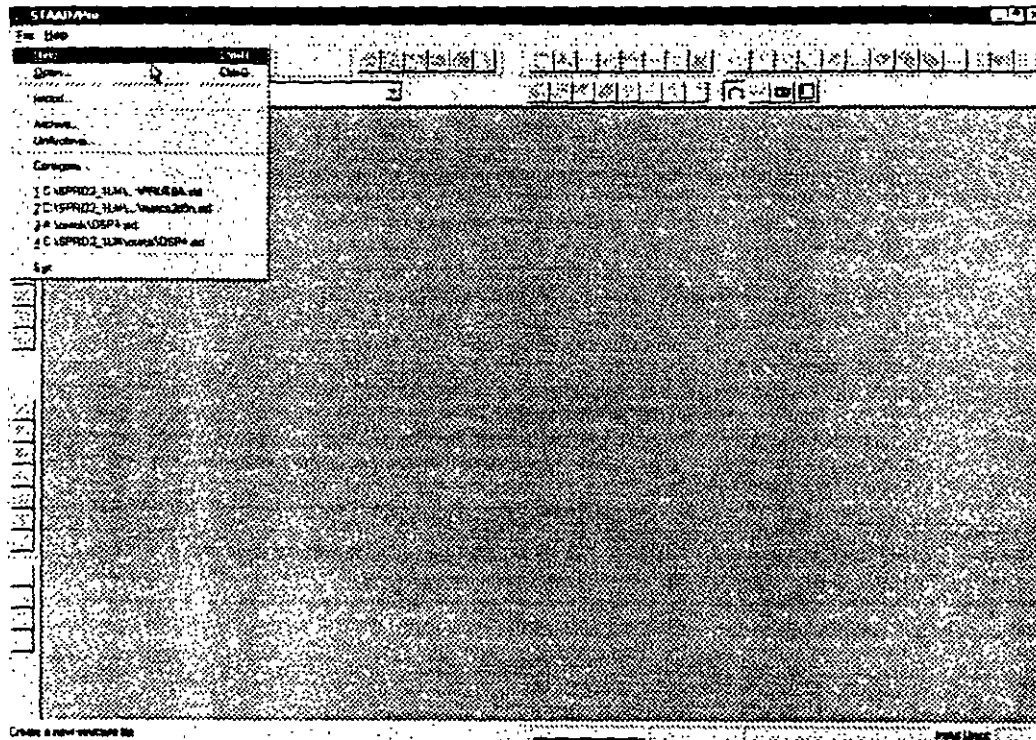


Figura 4.1 Ventana al iniciar STAAD/Pro

**SPACE-** Para estructuras tridimensionales considerando efectos de axial, torsión, flexión, en 2 direcciones y cortante también en 2 direcciones.

**PLANE-** Para cuando la estructura se puede considerar contenida en un plano (X, Y), entonces se le consideran efectos de flexión, cortante y axial.

FLOR- Es útil para analizar estructuras de piso (en el plano XZ, por ejemplo entrepisos y retículas de cimentación), considerándose sólo torsión, flexión y cortante.

TRUSS- Para cuando sólo se requiere considerar el efecto axial de las barras, la estructura, por ejemplo, armaduras planas o tridimensionales

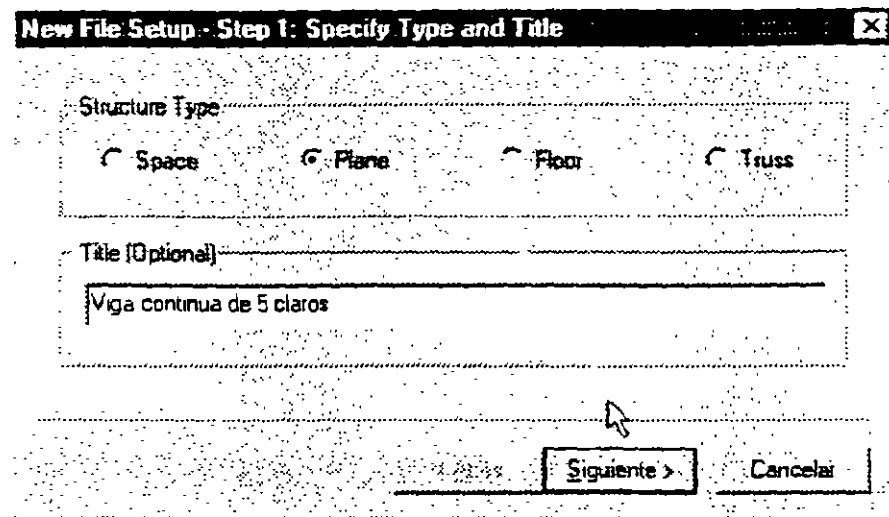


Figura 4.2 Definición del tipo de estructura

En el renglón en blanco puede introducirse de manera opcional, un título generalmente alusivo al nombre alusivo al nombre de la estructura por analizar (se aceptan espacios en blanco).

Enseguida se muestra el cuadro que se muestra en la figura 4.3 en donde es necesario seleccionar las unidades, tanto para las longitudes como para las fuerzas, en que se introducirán los datos de la estructura por analizar. después de hacer clic en siguiente STAAD/Pro nos muestra con la ventana de la figura 4.4 para finalmente haciendo clic en Finish se activen la mayoría de los menús y opciones disponibles y la estructura general del programa la cual se reproduce en la figura 4.5

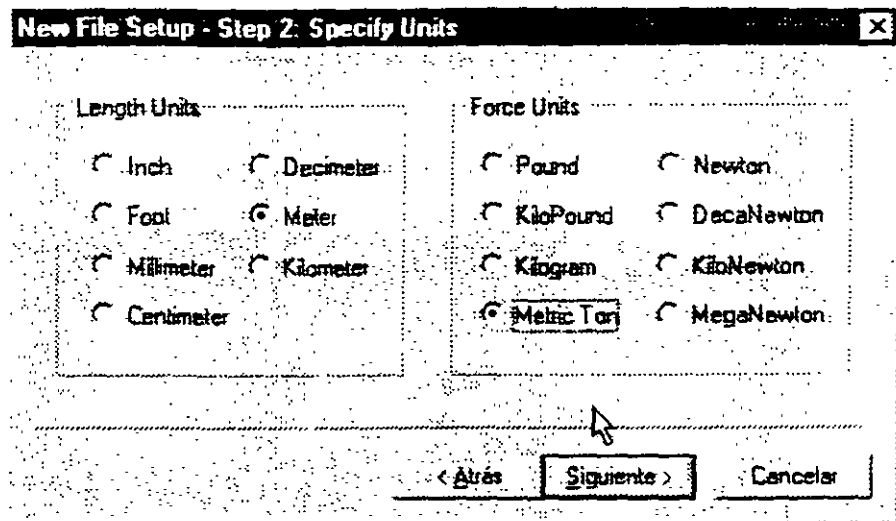


Figura 4.3 Definición de unidades

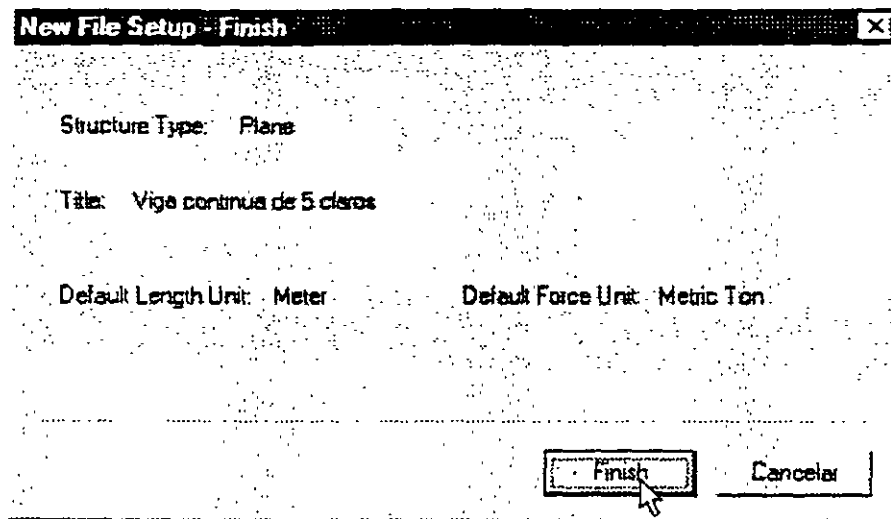
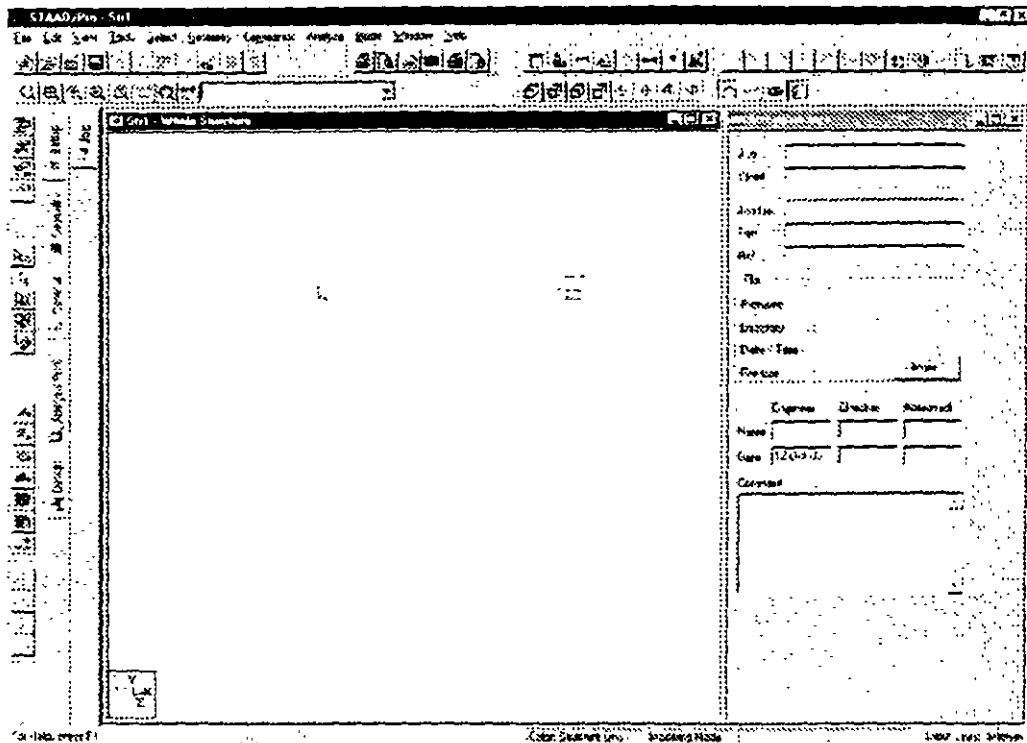


Figura 4.4 Datos de inicio definidos por el usuario



**Figura 4.5** Pantalla principal del programa STTAD/Pro

En esa figura, se reconocen tanto la barra de menús (file, edit, etc.) y debajo de ésta se muestran 2 renglones de iconos los cuales son accesos directos a una buena parte de las opciones contenidas en la mayoría de los menús, en la parte izquierda de la pantalla también se observa una columna de iconos y a la derecha de ésta, una columna de folders o carpetas los cuales agrupan a la información o datos de la estructura por categorías, por ejemplo, en la figura 4.5 se encuentra activado el folder Job y a la derecha del área en blanco se muestran los datos concernientes a esa carpeta de trabajo

Para iniciar la construcción de una malla se puede hacer clic en el folder geometría, mostrándose en el área en blanco una malla y a la derecha las características de esta, (véase figura 4.6). Comencemos con seleccionar el plano de trabajo xy, y además usaremos líneas de construcción espaciadas 2 m tanto en x como en y, esos datos se introducen en la ventana que tiene como título Snap node/ beam (véase figura 4.7).

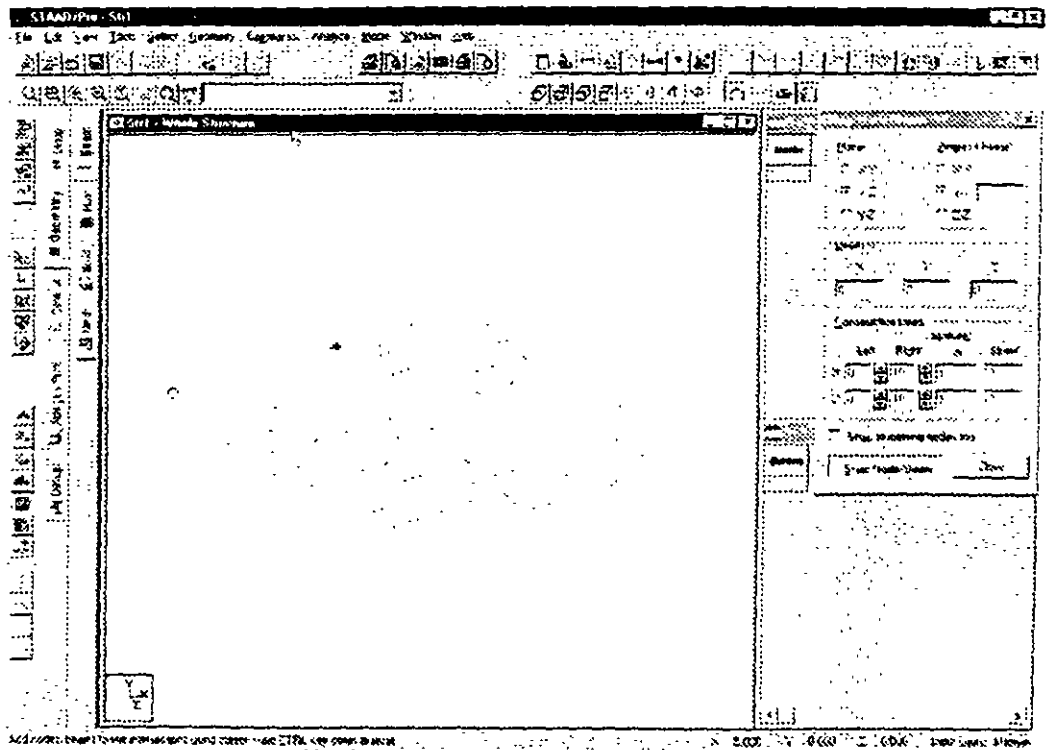


Figura 4.6 Carpeta geometria

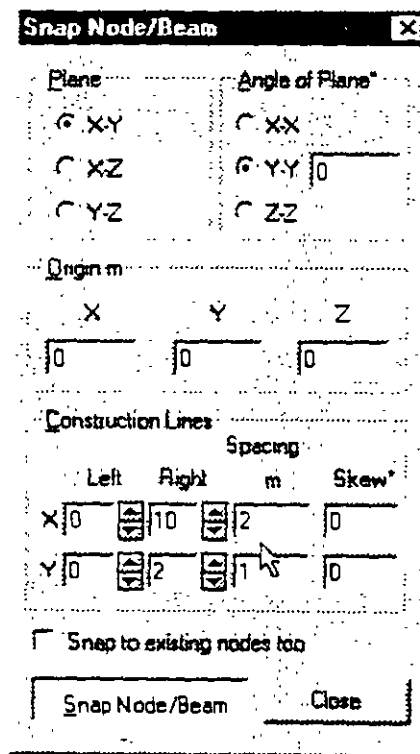


Figura 4.7 Especificación de las características de la malla



Una vez aceptadas las características de la malla de dibujo, se pueden empezar a dibujar las barras que componen al modelo, para ello se dirige el cursor a algún nudo de coordenadas específicas correspondiente a nuestro modelo para hacer clic, con lo que se muestra un círculo con un punto rojo indicando que es el nudo origen de un elemento barra, moviendo el ratón hacia la posición de la malla que definirá al otro extremo de la barra (nudo) y haciendo clic en ese punto que será el nudo final de una barra con lo que ésta se habrá definido, la creación de la geometría (adición de nuevas barras) se puede realizar de esa manera (véase figura 4.8).

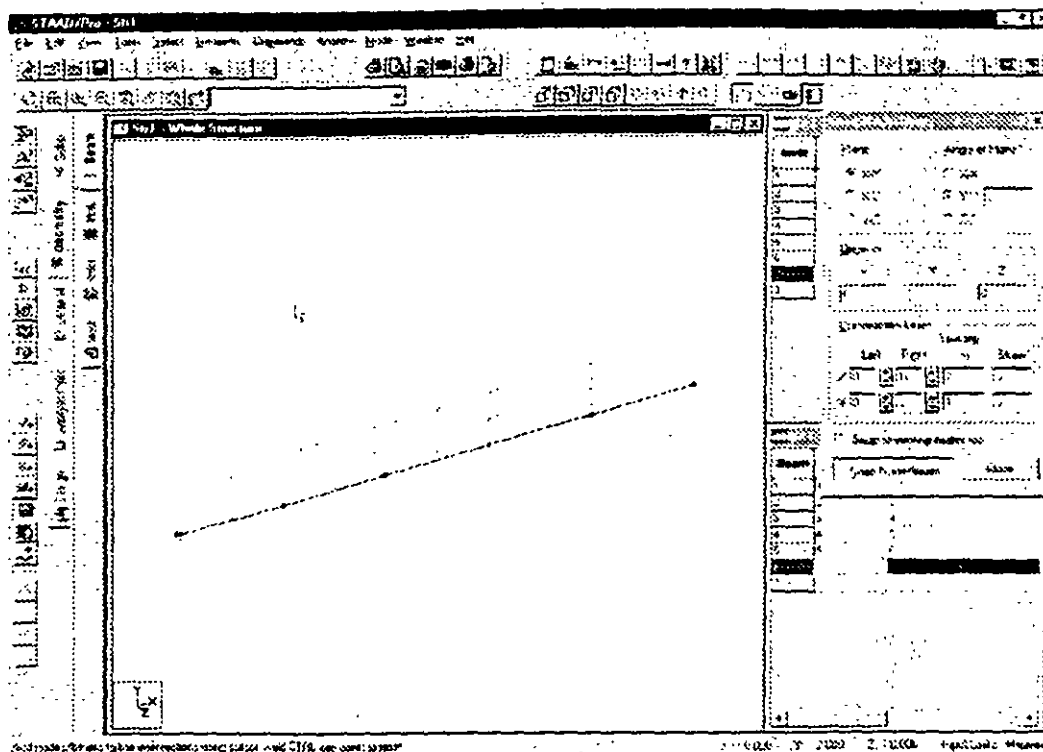
Para interrumpir el último punto referenciado antes de hacer clic en un nuevo nudo se mantiene oprimida la tecla Ctrl., con lo cual se podrán posicionar otras barras en otros nodos.

Al cerrar la ventana Snap node/beam desaparece del área en blanco la malla auxiliar de dibujo ya que no se podrán seguir introduciendo elementos barra y además se muestra a la derecha las características donde estos nudos (coordenadas) como de las barras (incidencias).

Para borrar elementos barra no deseados se puede proceder de la siguiente manera primero asegurarse que se está trabajando en el modo de selección (cursor de flecha), para ello se hace clic en el cursor icono flecha de la barra vertical de iconos, luego se dirige el cursor cerca del elemento que se va a seleccionar y después de hacer clic en dicho elemento (éste cambia a doble línea de color indicando que ha sido seleccionado) se oprime la tecla supr o bien se selecciona la opción delete del menú edit, enseguida aparecerá una ventana de mensaje indicando el número de elementos a eliminar, puede el usuario decidir (aceptar o cancelar) la acción.

Para seleccionar varios elementos se puede proceder a seleccionar uno por uno manteniendo oprimida la tecla Ctrl y/o marcar una ventana para ello se hace clic en un extremo del mismo y se arrastra el puntero del ratón hasta un extremo, de tal manera que los elementos a seleccionar estén casi completamente contenidos dentro de la ventana definida por el usuario).

Para adicionar nuevos elementos barras se puede hacer clic en el icono que tiene una malla y una barra inclinada, en la barra vertical de iconos o bien seleccionar del menú geometría la opción Snap/grid node.



**Figura 4.8 Geometría terminada**

La figura 4.8 muestra la geometría de una viga continua de 5 claros, cada uno con una longitud de 4 m.

#### **4.2 Asignación de propiedades geométricas**

STAAD/Pro puede trabajar de dos maneras: por comandos y de manera gráfica. de la primer manera primero es necesario seleccionar elementos (nudos, barras, etc.) y después elegir algún comando (asignar propiedades, apoyos, etc.), para la segunda será necesario, primero, definir alguna característica (sección, tipo de apoyo, etc.), luego asignarla a algún elemento (barra, nudo, etc.)

Por ejemplo, consideremos que la viga continua ya definida será de concreto con una sección transversal de 30 x 45 cm, para asignar estas características a las barras, primero las seleccionamos, enseguida en el menú comando / Member Property / Prismatic (ver figura 4.9)

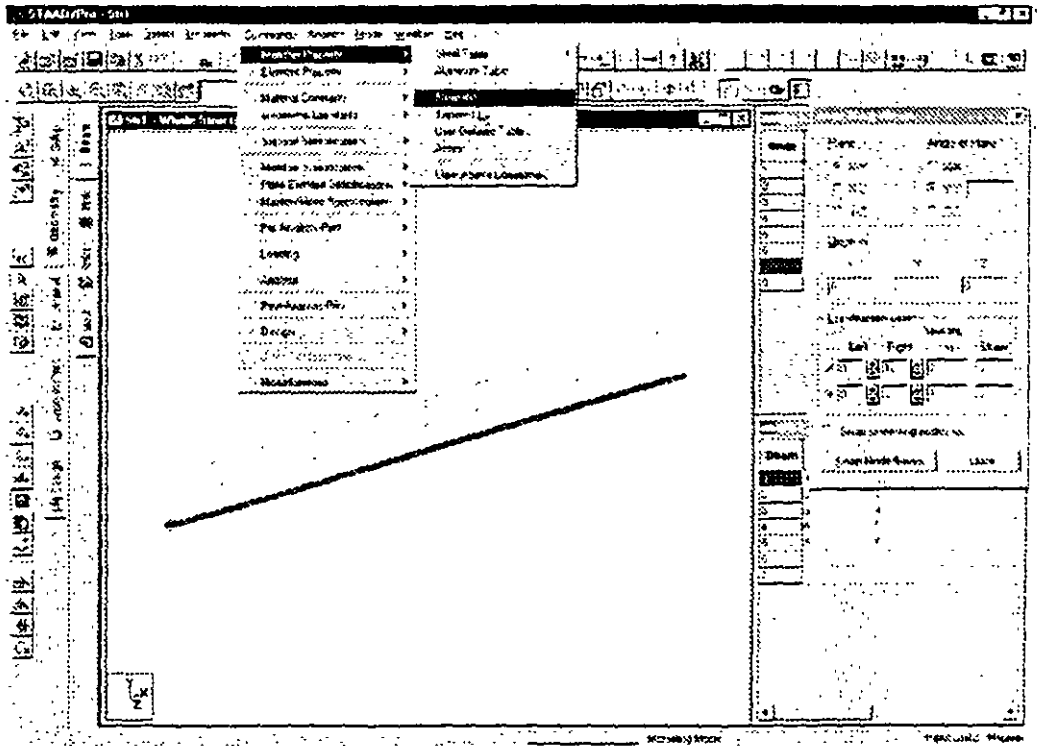


Figura 4.9 Definición de propiedades geométricas

Seleccionar la carpeta rectangle e introducir las dimensiones de la sección 0.45 y 0.30, como se muestra en la figura 4.10, enseguida hacer clic en el cuadro Assign con lo que las barras seleccionadas tendrán esa característica.

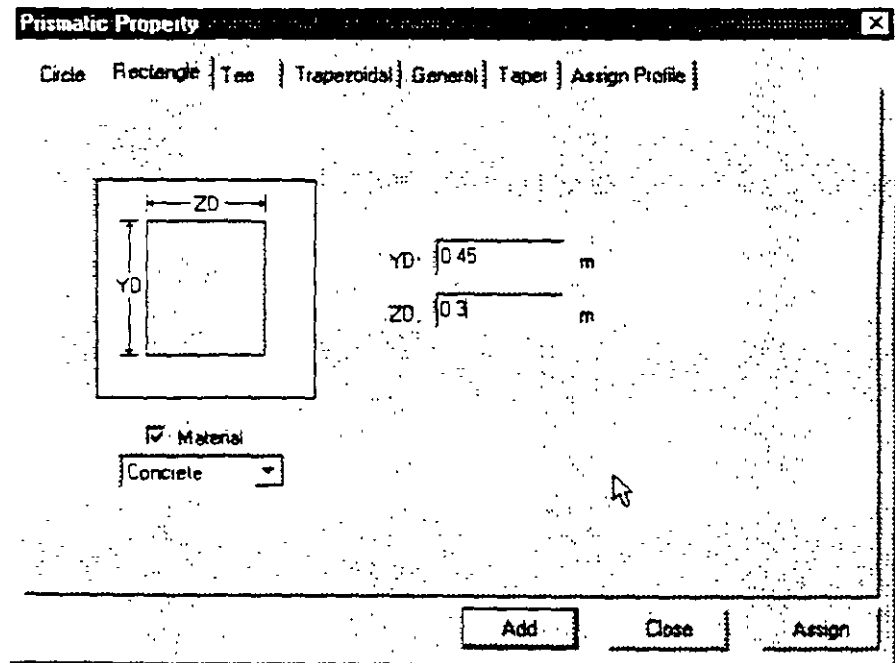


Figura 4.10 Características de la sección rectangular

#### 4.4 Tipos de apoyo

Ahora con el cursor de selección de nudos seleccionamos los nudos extremos (que tendrán apoyo empotrado) y del menú Commands / Support Specifications seleccionamos Fixed (ver figura 4.11).

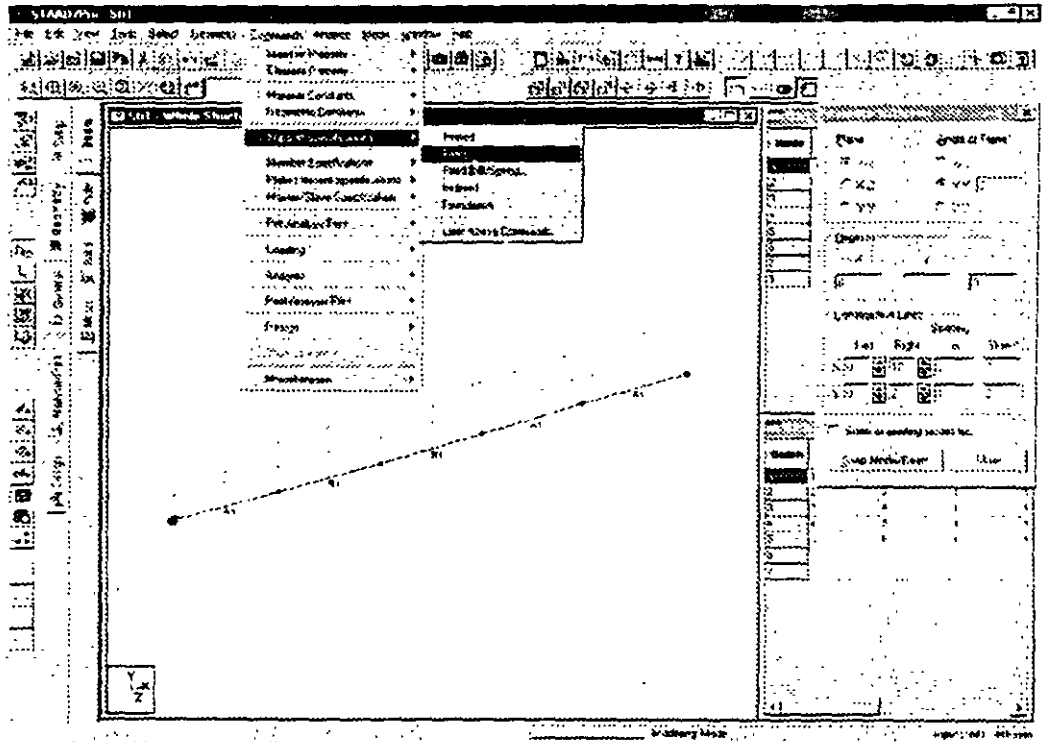


Figura 4.11 Especificación de apoyos

En la ventana que enseguida se muestra hacer clic en el cuadro Assign (Figura 4.12).

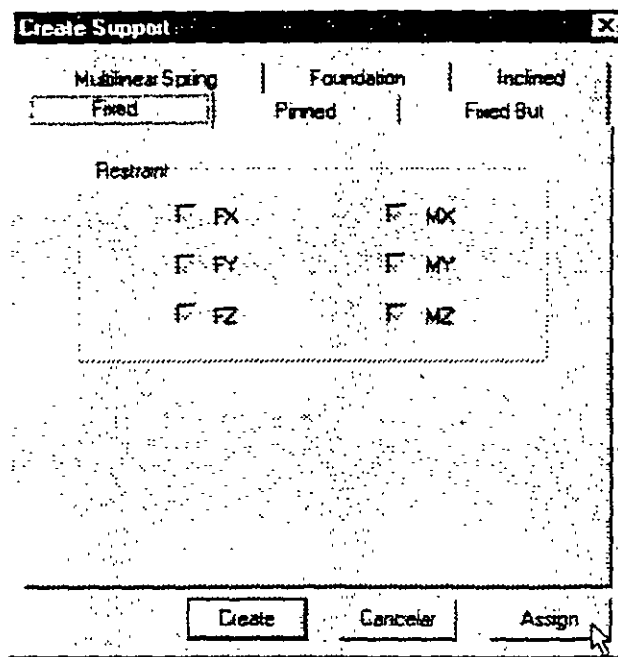


Figura 4.12 Tipos de apoyos

Repetimos el proceso, pero ahora con los nudos Intermedios asignándoles apoyos articulados (pinned)

#### 4.5 Asignación de fuerzas, condiciones y combinaciones de carga.

Para definir condiciones y combinaciones de carga se utiliza el menú Commands / Loading / Primary Loads (Figura 4.13)

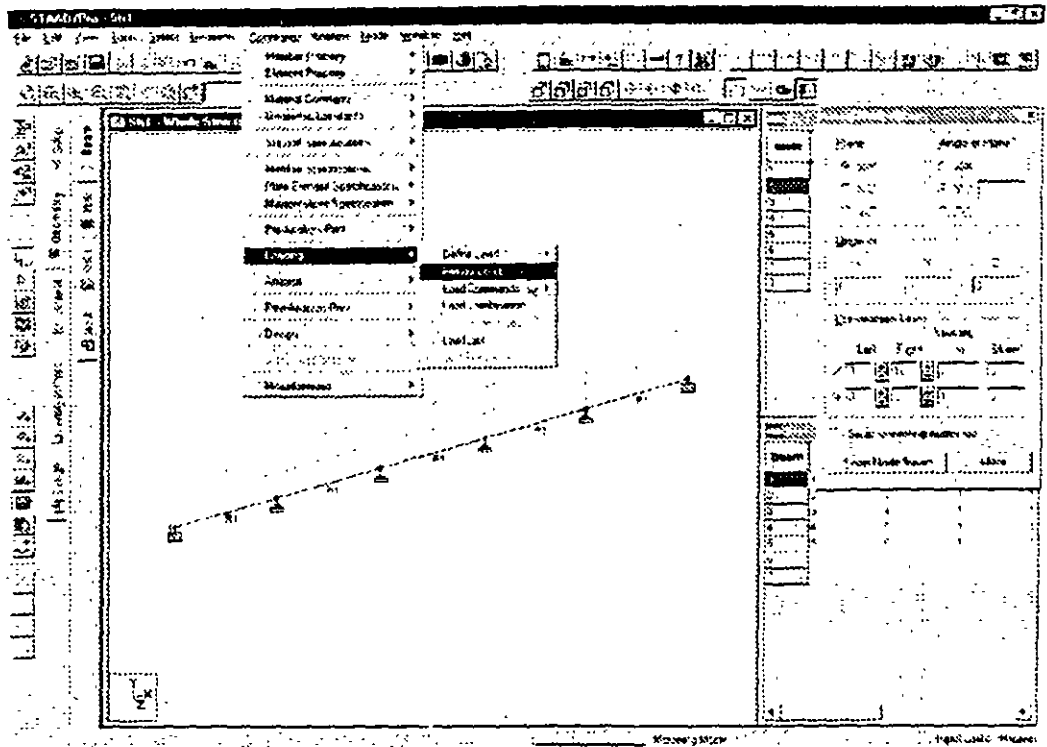


Figura 4.13 Definición de condiciones de carga

En la ventana mostrada (Figura 4.14), se puede seleccionar una existente o crear una nueva, en este último caso se puede introducir, en el cuadro en blanco, el título que se le dará a esa condición (Figura 4.14), por ejemplo, peso propio.

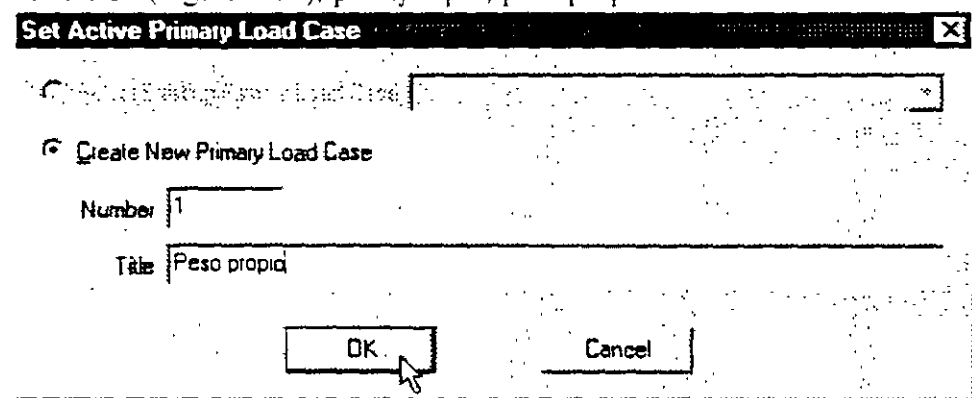


Figura 4.14 Creando una nueva condición de carga

Ahora para que el programa considere al peso propio como acción se hace clic en el cuadro Selfweight (Figura 4.15) enseguida se muestra el cuadro Selfweight Load, en el se hace clic en el cuadro Assign (Figura 4.15), con lo que en el cuadro en blanco (Load Specification) se muestra SELFWEIGHT Y - 1 con lo que el peso propio se incluirá en la condición de carga especificada

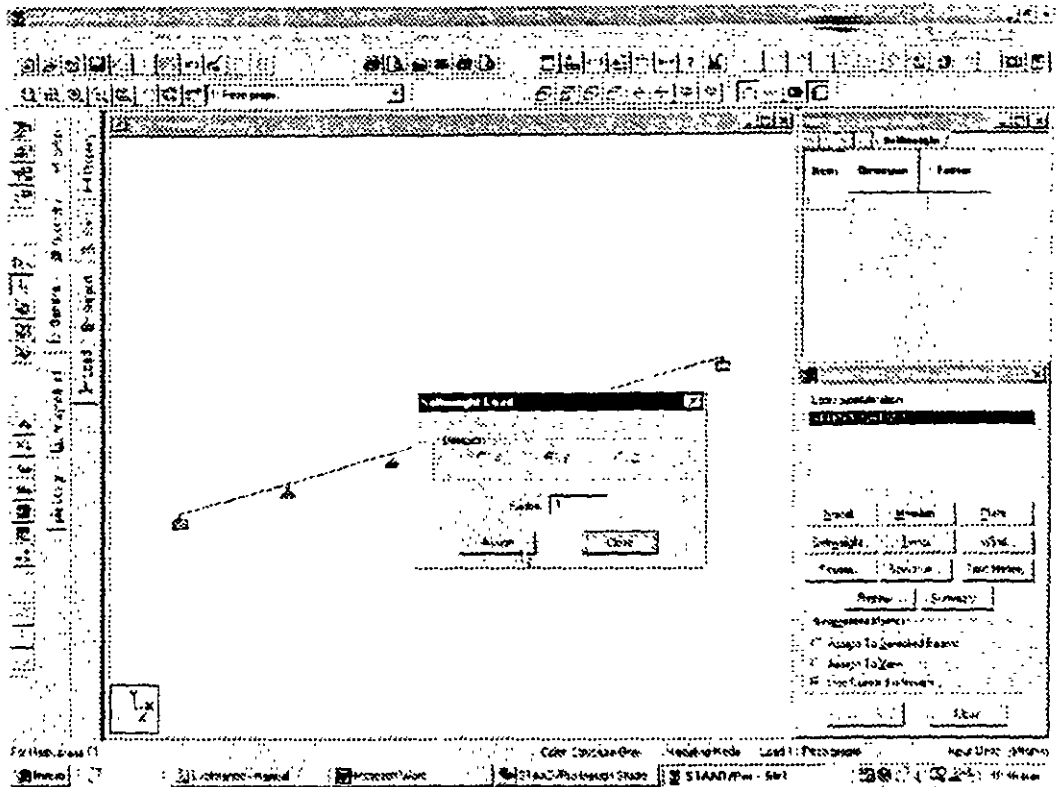


Figura 4.15 Consideración del peso propio en el análisis de la estructura

Ahora definiremos una nueva condición de carga seleccionando nuevamente **Commands / Loading / Primary Load** y en la ventana mostrada activar **Create New Primary Load Case** e introduciendo como título el mostrado en la figura 6.16 y después hacer clic en el cuadro **OK** introduzcamos una carga uniforme de  $-1.5 \text{ Ton/m}$  a las tres primeras barras, para ello, después de seleccionarlas y luego de hacer clic en el **Member** se introducen los valores que se muestran en la figura 4.17

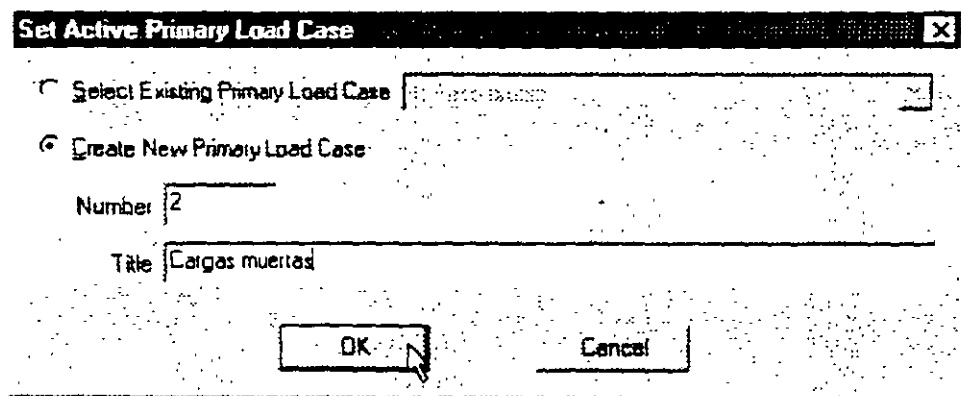


Figura 4.16 Definición de una nueva condición de carga

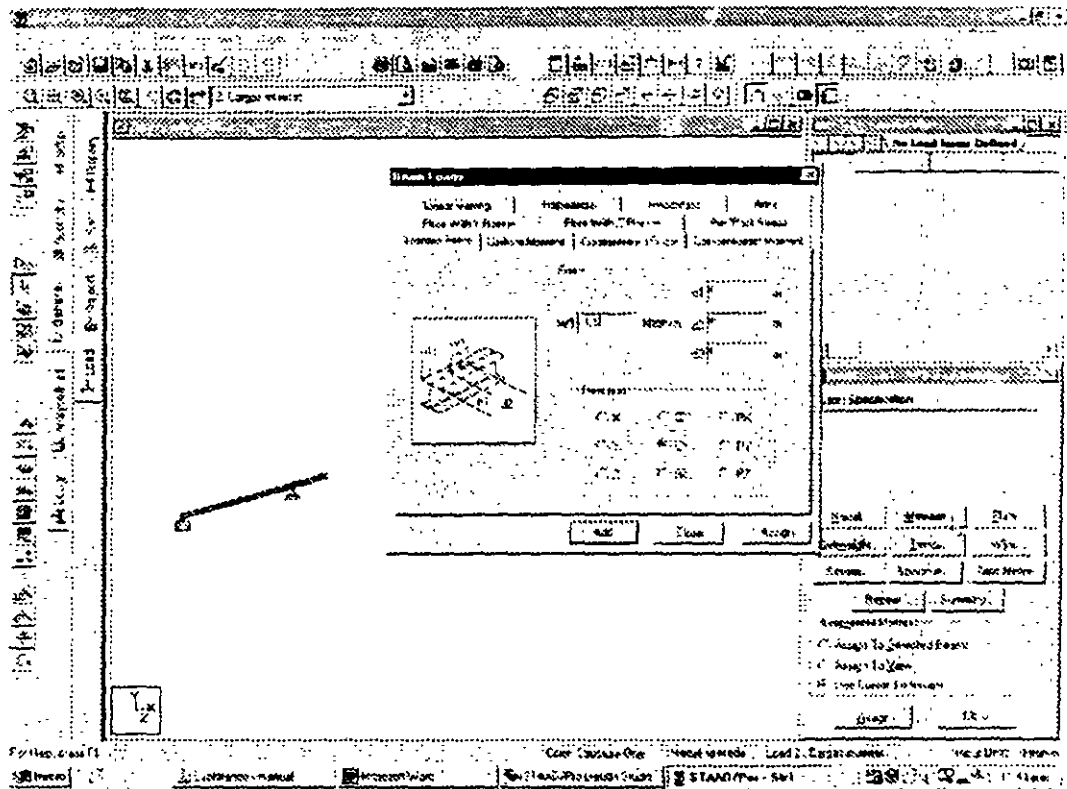


Figura 4.17 Especificación de parámetros para carga uniforme

Al hacer clic en el cuadro Assign se asignará la carga uniforme especificada a las barras seleccionadas. Enseguida de manera similar, se especificará una carga concentrada al centro de los dos tramos de la derecha. Primero se seleccionan las dos barras, luego se hace clic en el cuadro member y en la ventana que se muestra se hace clic en la carpeta Concentrated Force introduciendo los datos que se muestran en la figura 4.18 y después de hacer clic en el cuadro Assign la viga tiene las cargas mostradas en la figura 4.19



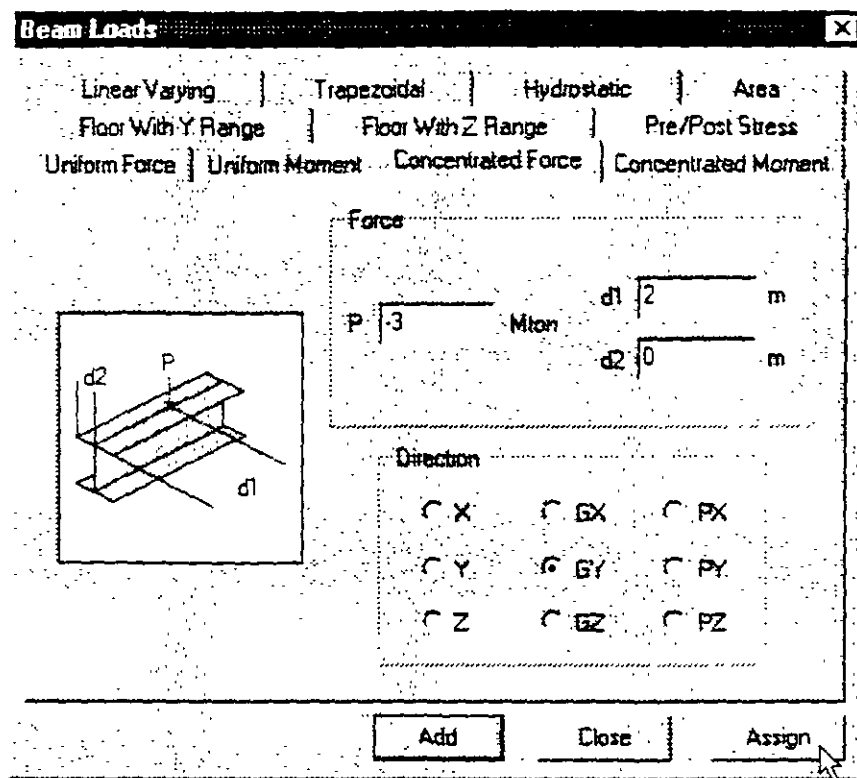


Figura 4.18 Especificación de cargas concentradas

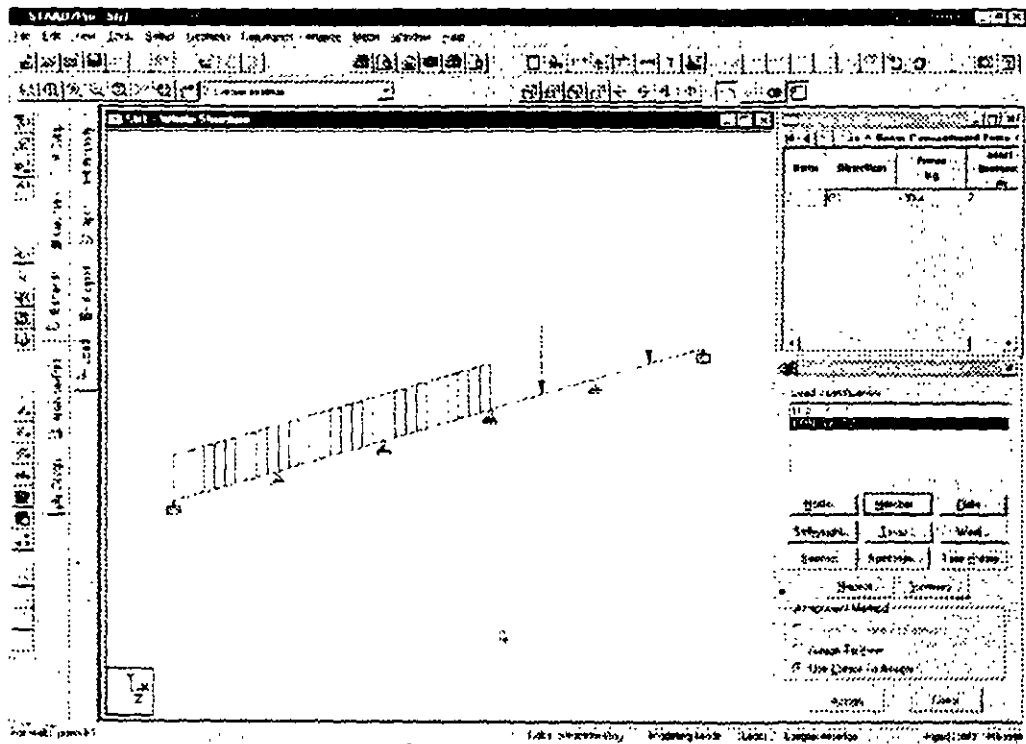


Figura 4.19 Cargas en la viga

Ahora especifiquemos una combinación de carga como la suma de las dos anteriores multiplicada por 1.4, para ello, nuevamente del menú Commands seleccionar Loading y de ahí Load Combination (Figura 4.20)

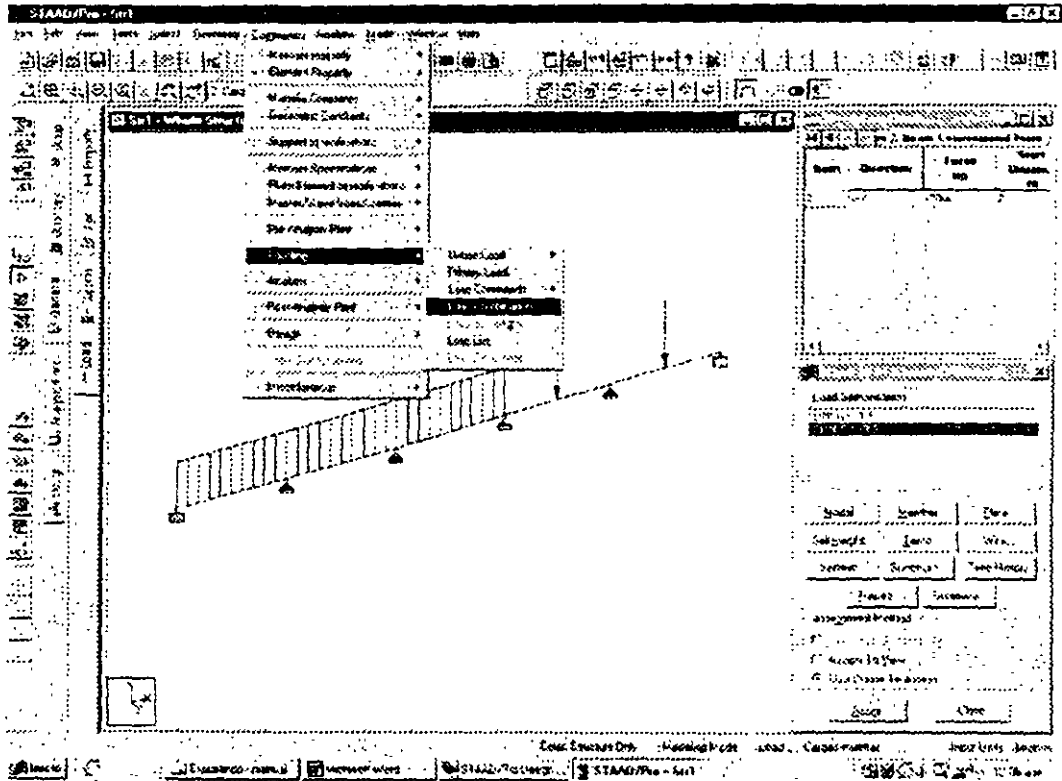


Figura 4.20 Definición de combinaciones de carga

En la ventana mostrada hacer clic en New e introducir el nombre de la combinación (Figura 4.21).

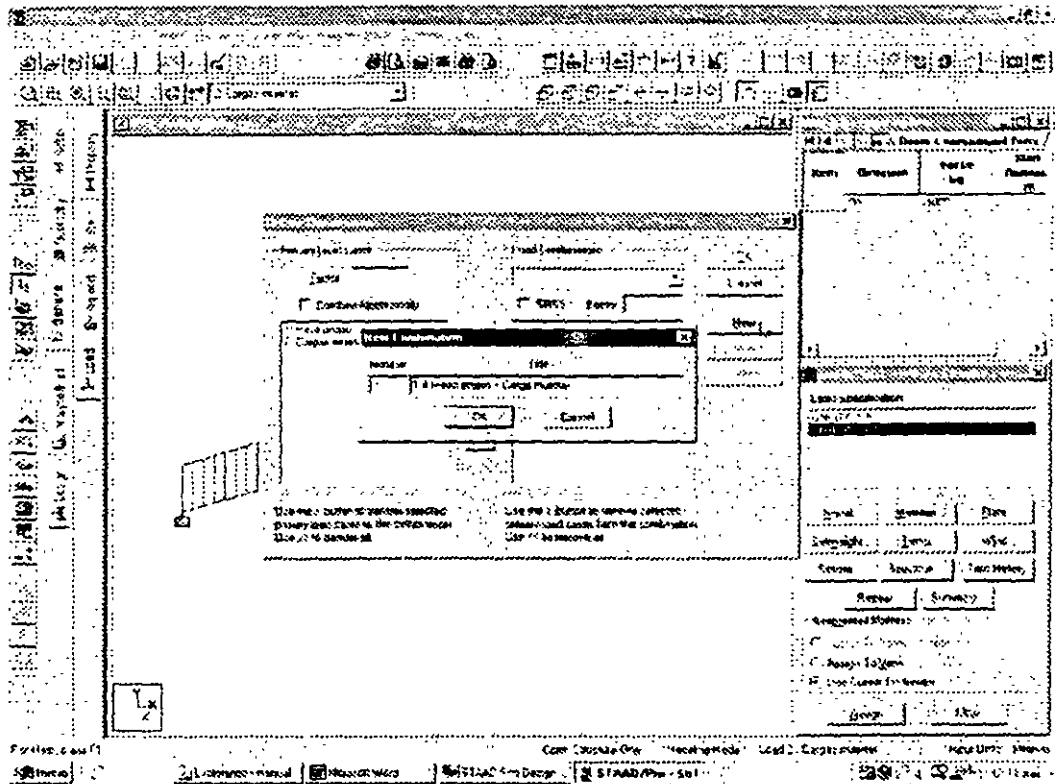
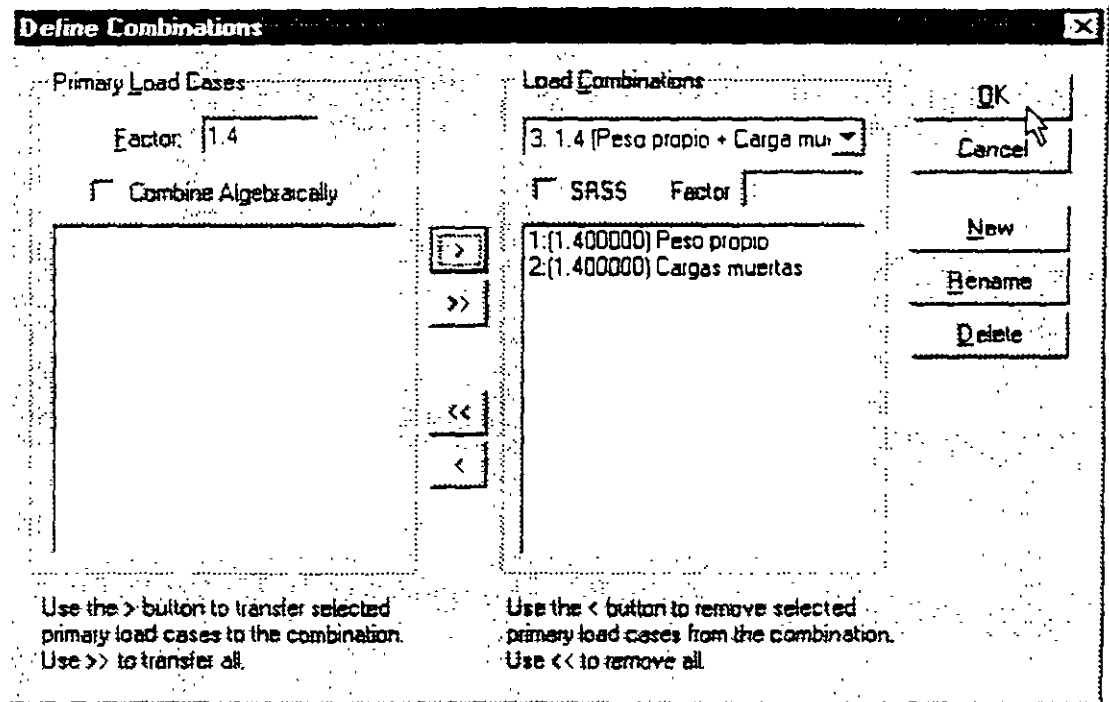


Figura 4.21 Nombre para la combinación de carga

Al hacer clic en el cuadro OK se muestra la ventana de la figura 4.22 en ella primero especifica el factor de carga (1.4 para este caso) y luego se selecciona, del cuadro a la izquierda, la condición que se quiere incluir y luego se hace clic en el botón > para pasarla al cuadro de la derecha con lo que se incluirá en la combinación de carga, para este caso se repite la operación anterior con la condición de carga 2 quedando como se muestra en la figura 4.22



**Figura 4.22** Datos para la combinación de carga

Se pueden especificar más combinaciones de carga haciendo clic en New en la ventana de la figura 4.21 o bien como se indicó al inicio de esta sección

Para terminar la especificación de las combinaciones de carga se hace clic en el cuadro OK de la figura 4.22.

#### 4.6 Opciones de análisis, selección de resultados

Para especificar la realización de un análisis elástico lineal se selecciona del menú Comands / Analysis la opción Perform Analysis (Figura 4.23) enseguida en la ventana que se muestra, se selecciona ALL (Figura 4.24) y después se hace clic OK con lo que el comando de análisis se adicionará al archivo de datos

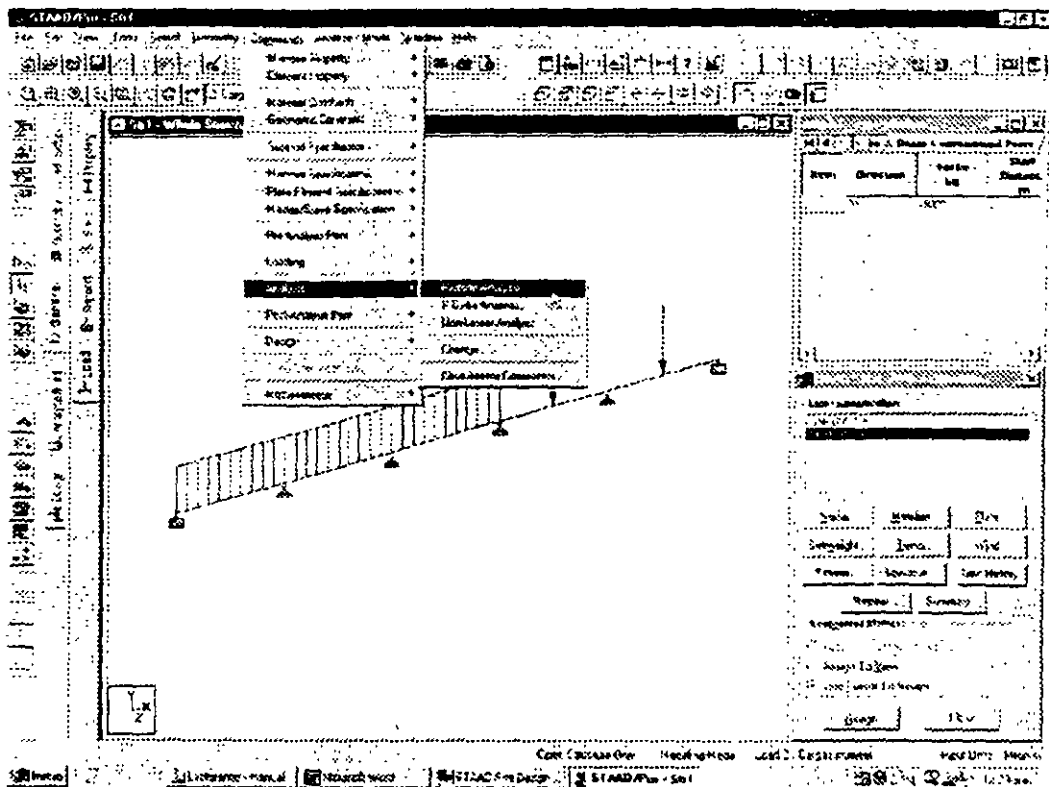


Figura 4.23 Opción de Análisis Elástico Lineal

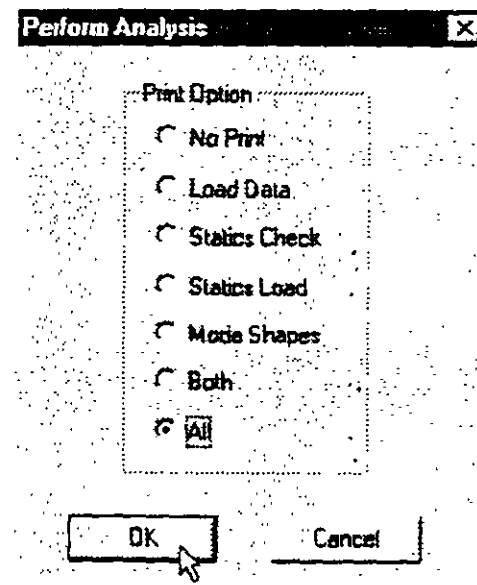


Figura 4.24 Parámetros de la opción Análisis

Para incluir resultados en el archivo de salida (\*.ani) se pueden seleccionar elementos (nodos, barras, etc.) y luego del Menú Commands seleccionar Post-Analysis Print y el tipo de resultados deseados (este comando se puede repetir varias veces), ver figura 4.25.

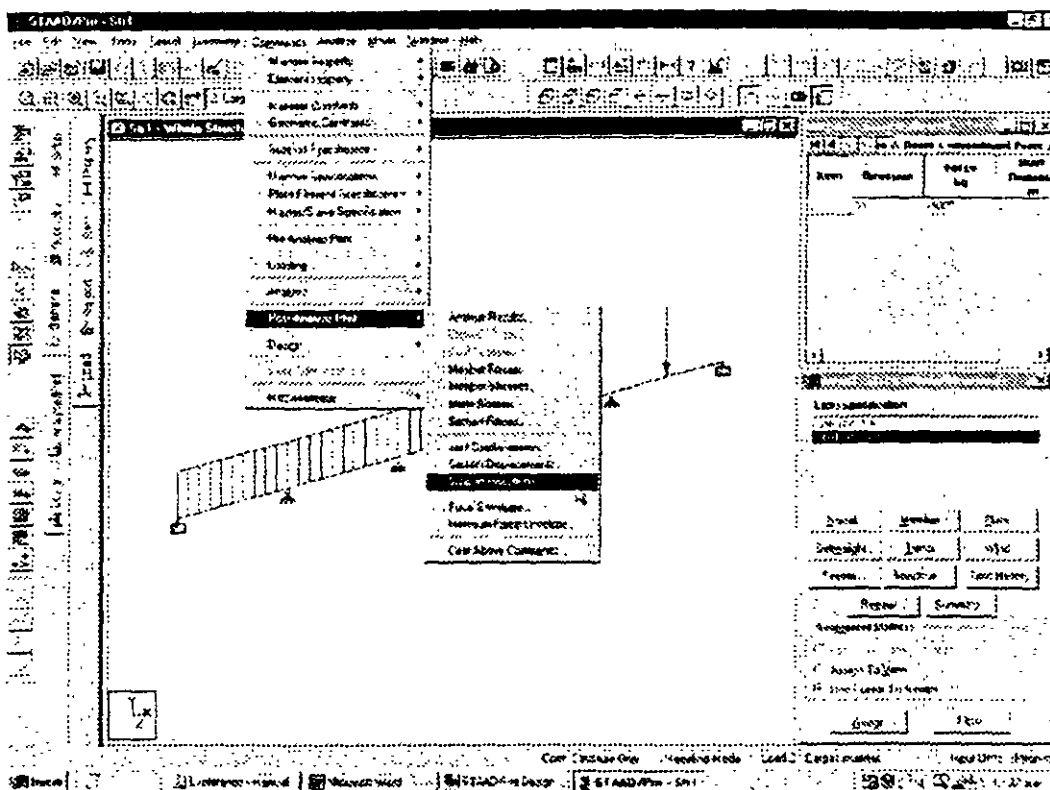


Figura 4.25 Selección de resultados

Si no se seleccionan elementos (nudos, barras, etc.) se incluirán (en el archivo de salida) los resultados de los elementos que estén presentes en la vista al momento de activar ese comando, el cual despliega una ventana, por ejemplo, como la mostrada en la figura 4.26, para la impresión de reacciones.

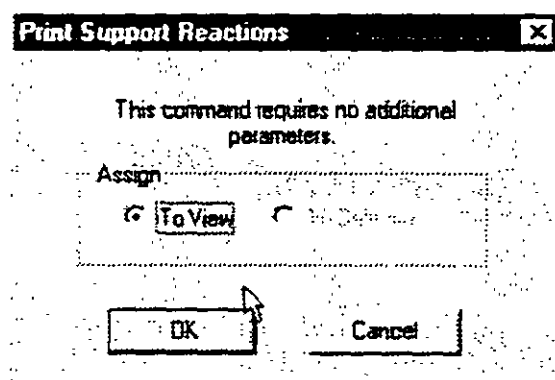


Figura 4.26 Selección de reacciones

# ANÁLISIS DE LA ESTRUCTURA

# CAPÍTULO 5

Antes de solicitar el análisis y una vez completados todos los datos de la estructura por analizar se recomienda guardar los datos en un archivo, para ello se selecciona la opción **Save** a **Save as** del menú **file** y en la ventana que aparece se especifica el nombre del archivo así como su ubicación (Figura 5.1).

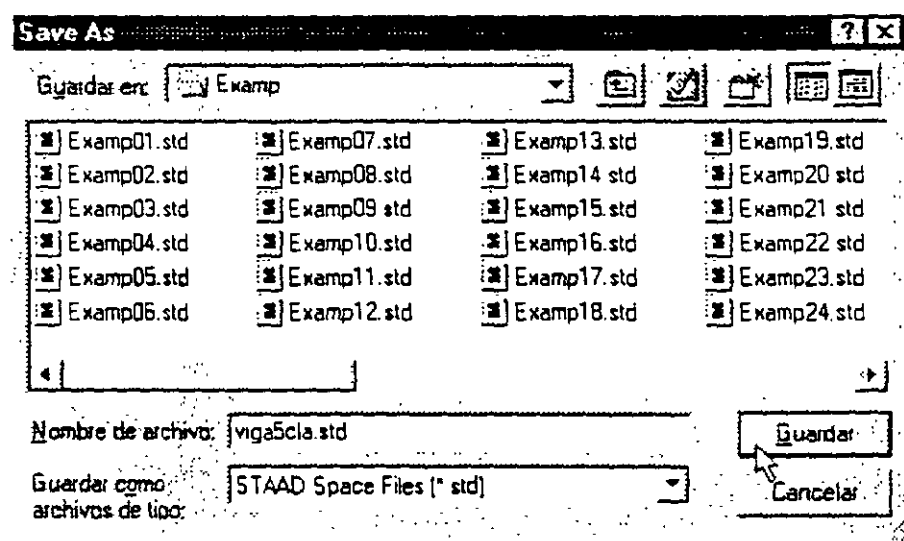


Figura 5.1 Almacenamiento de datos en un archivo

Ahora del menú **Analyze** hacer clic en **Run Analysis...** (Figura 5.2), con lo que se muestra la ventana de las figuras 5.3 en ella se selecciona la herramienta

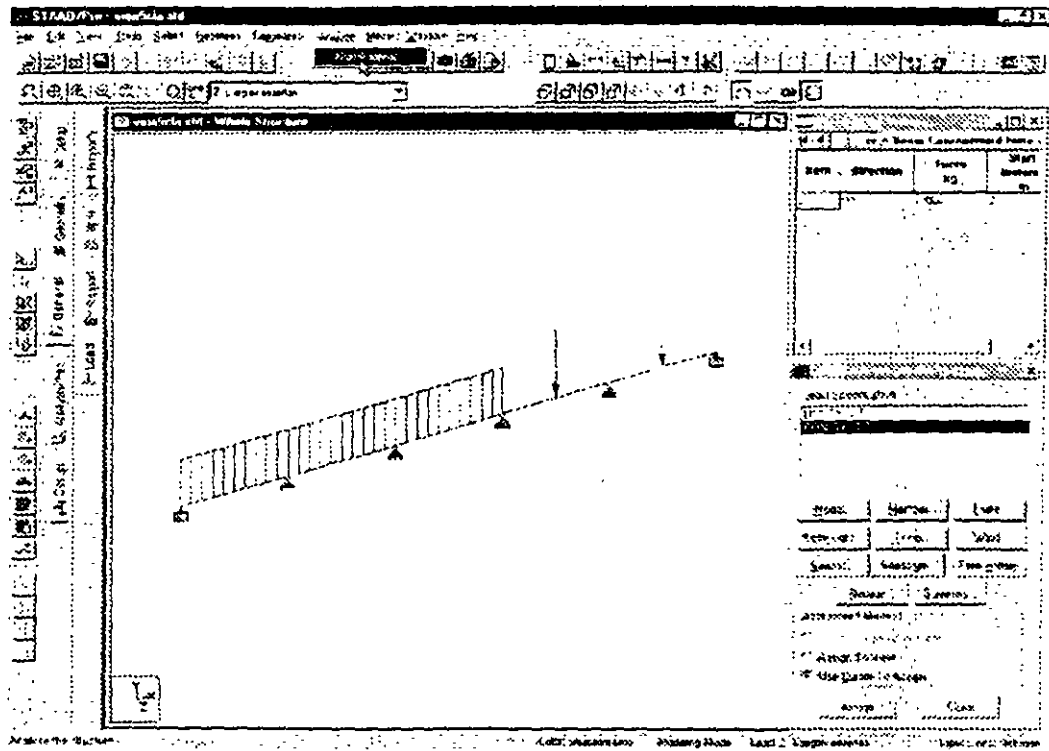


Figura 5.2 Análisis de la estructura

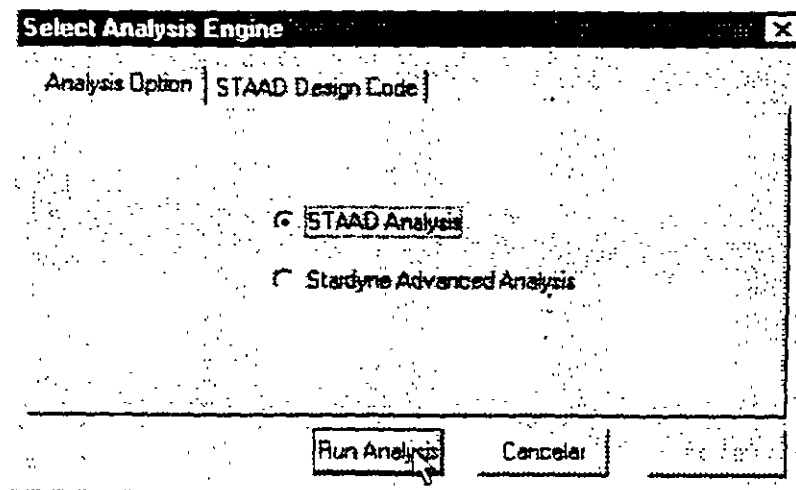


Figura 5.3 Selección de la herramienta de análisis

de análisis (STAAD Analysis o Stadyne...) y después se hace clic en Run Analysis para, enseguida, iniciarse el análisis.

STAAD/Pro procesa todas las instrucciones contenidas en el archivo de datos (\*.std) generando el archivo (\*.ani o \*.out, dependiendo de la herramienta o motor de análisis seleccionado, Staad o Stadyne respectivamente). Para la herramienta de análisis Staad el resultado de la fase de análisis se muestra en la figura 5.4



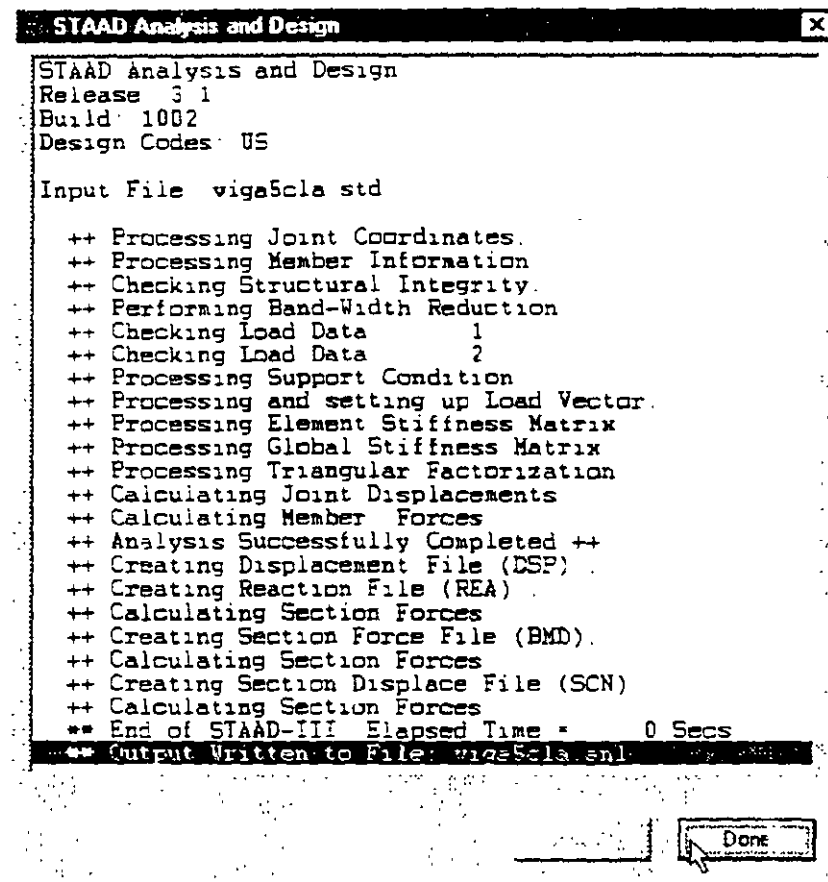


Figura 5.4 Resultados de la fase de análisis

# VER RESULTADOS

# CAPITULO 6

## 6.1 Introducción

Una vez realizado el análisis se recomienda revisar el contenido del archivo de resultados (\*.ani o \*.out) para ello se selecciona la opción STAAD Output de Output File en el submenú View del menú File (Figura 6.1).

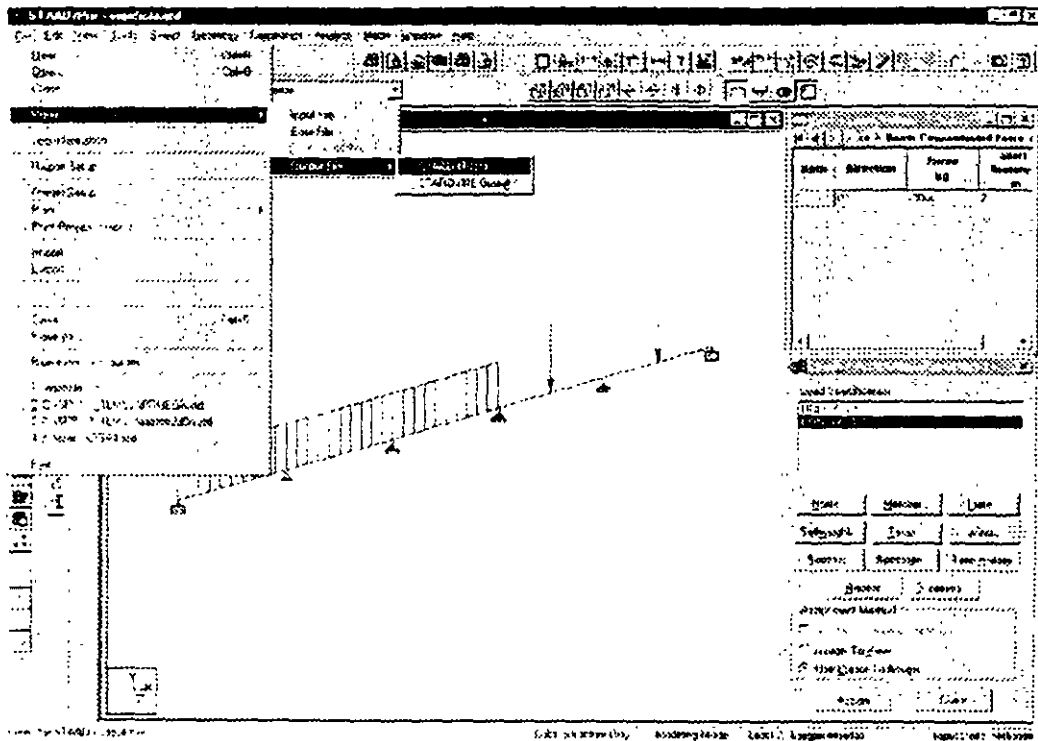


Figura 6.1 Ver el contenido del archivo de resultados

La secuencia anterior muestra el contenido del archivo de resultados en un "visor" (Figura 6.2), después de revisar su contenido (utilizando las teclas de flechas de desplazamiento o la barra para el mismo fin) haciendo clic en la opción exit del menú file del visor (Figura 6.2) regresamos al ambiente principal de STAAD/Pro.

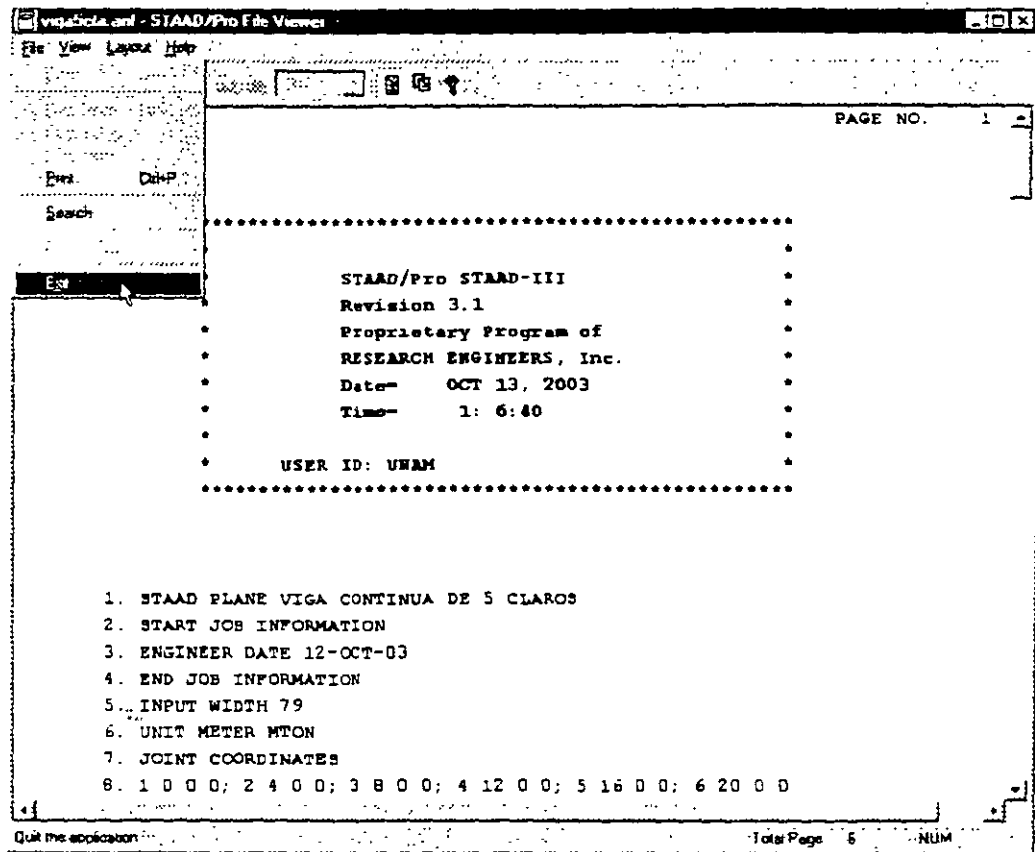


Figura 6.2 Visor de resultados

## 6.2 Ver estructura deformada

El modo postproceso (menú Mode, figura 6.3) permite, por ejemplo, ver de manera gráfica y numérica la mayoría de los resultados generados por la fase de análisis (aún algunos no contenidos en el archivo .ani).

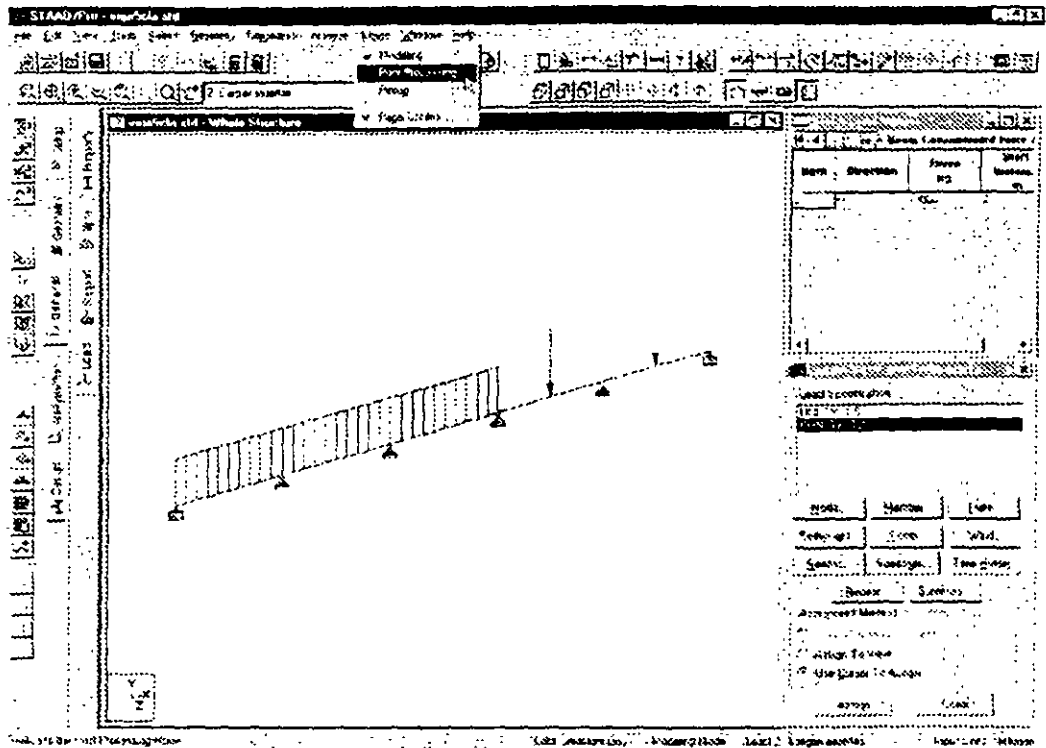


Figura 6.3 Modo Postproceso

El cambio al modo postproceso produce la ventana de la figura 6.4, en ella, por ejemplo, se pasan a la ventana de la derecha las condiciones de carga de las cuales se mostrarán los resultados.

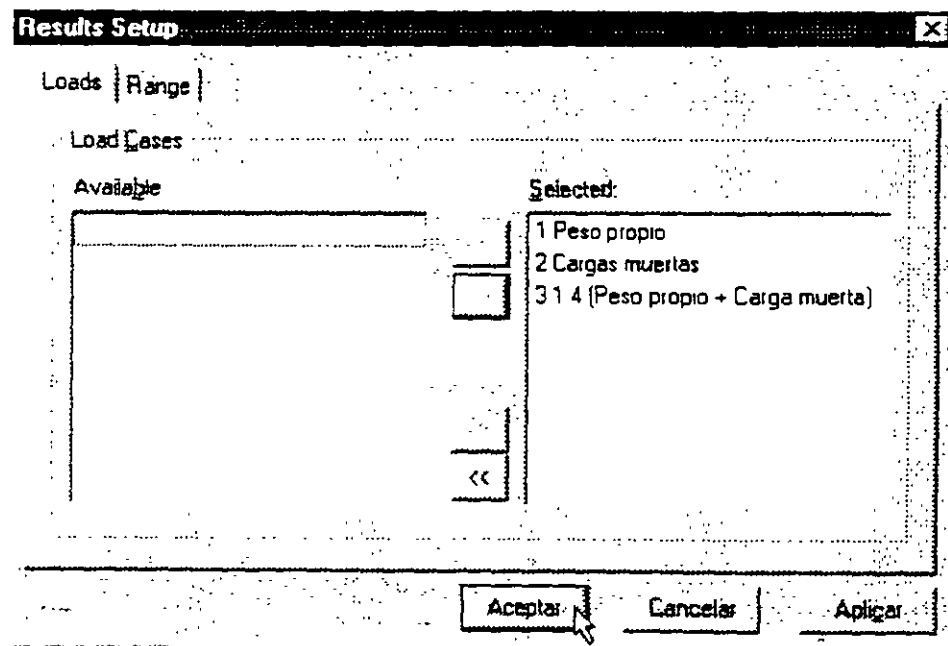


Figura 6.4 Selección de condiciones para mostrar resultados

Al hacer clic en aceptar STAAD/Pro muestra la pantalla y opciones del modo postproceso, en el área en blanco muestra la deformada de la estructura, la escala de esa deformada se puede modificar mediante la opción scale del menú Results (Figura 6.5).

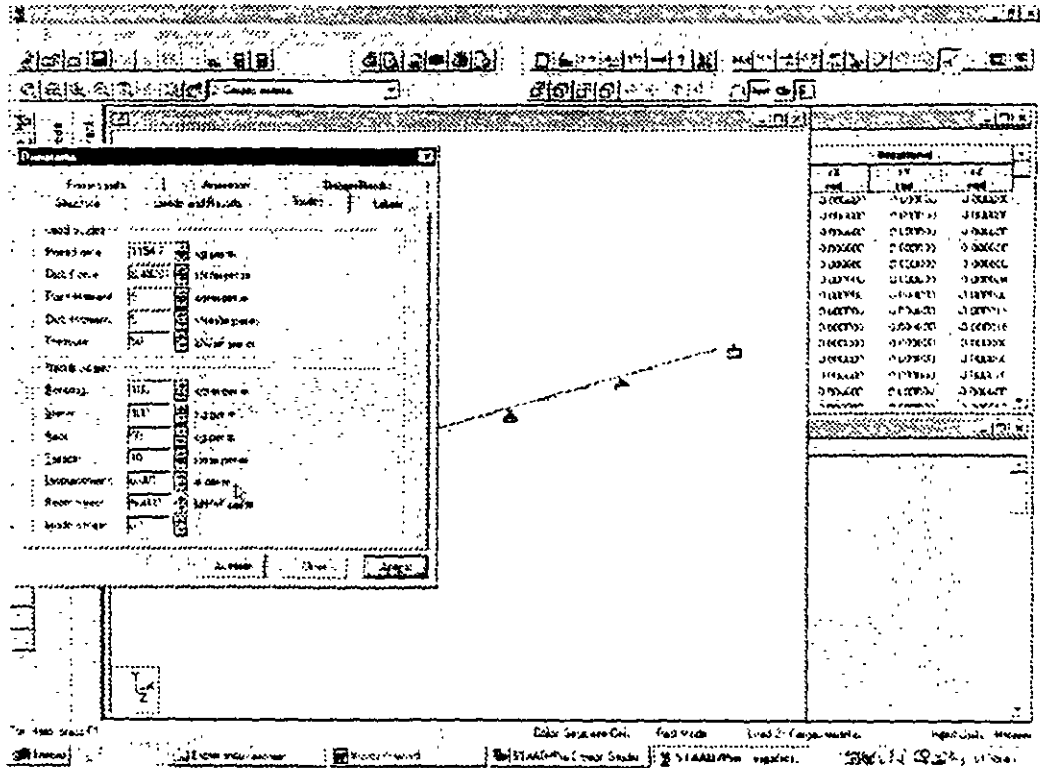


Figura 6.5 Cambio de escala

Para ver los resultados de otra condición o combinación de carga, por ejemplo, ésta se puede seleccionar del cuadro en blanco de la segunda barra horizontal de iconos (Figura 6.6)

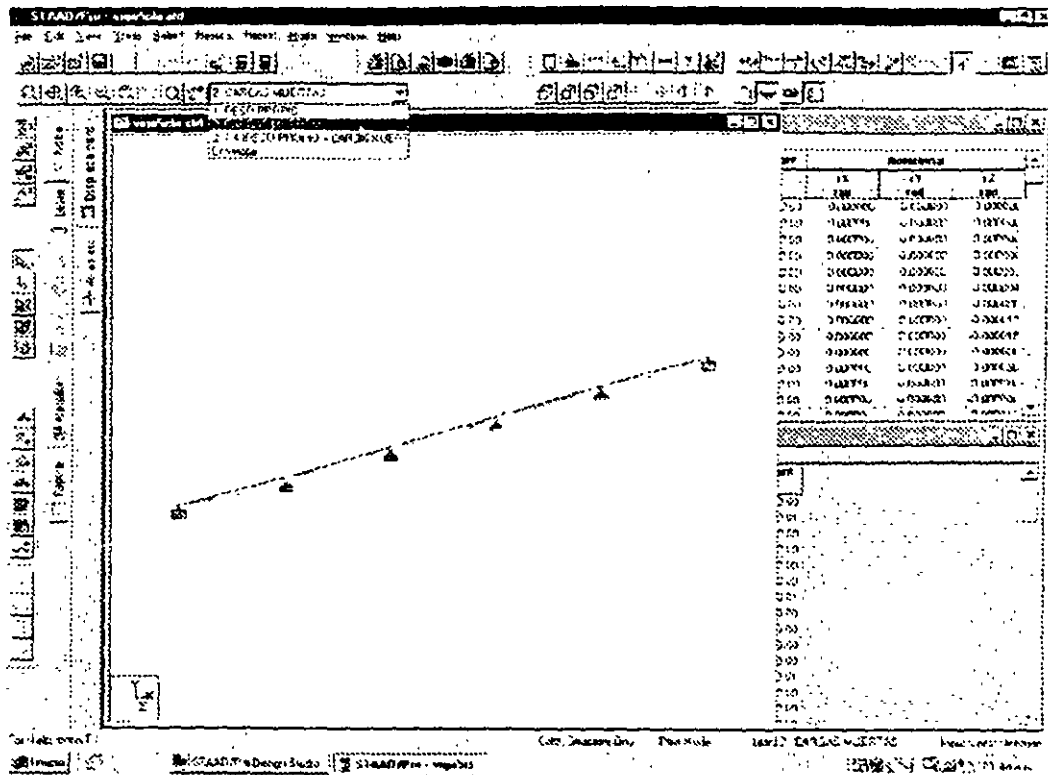


Figura 6.6 Seleccionando otra condición de carga

6.3 Ver diagramas de elementos mecánicos

Para que STAAD/Pro muestre los diagramas de elementos mecánicos, puede ser conveniente, primero quitar la deformada, para ello se deselecta la opción Section Displacement del menú Results (Figura 6.7)

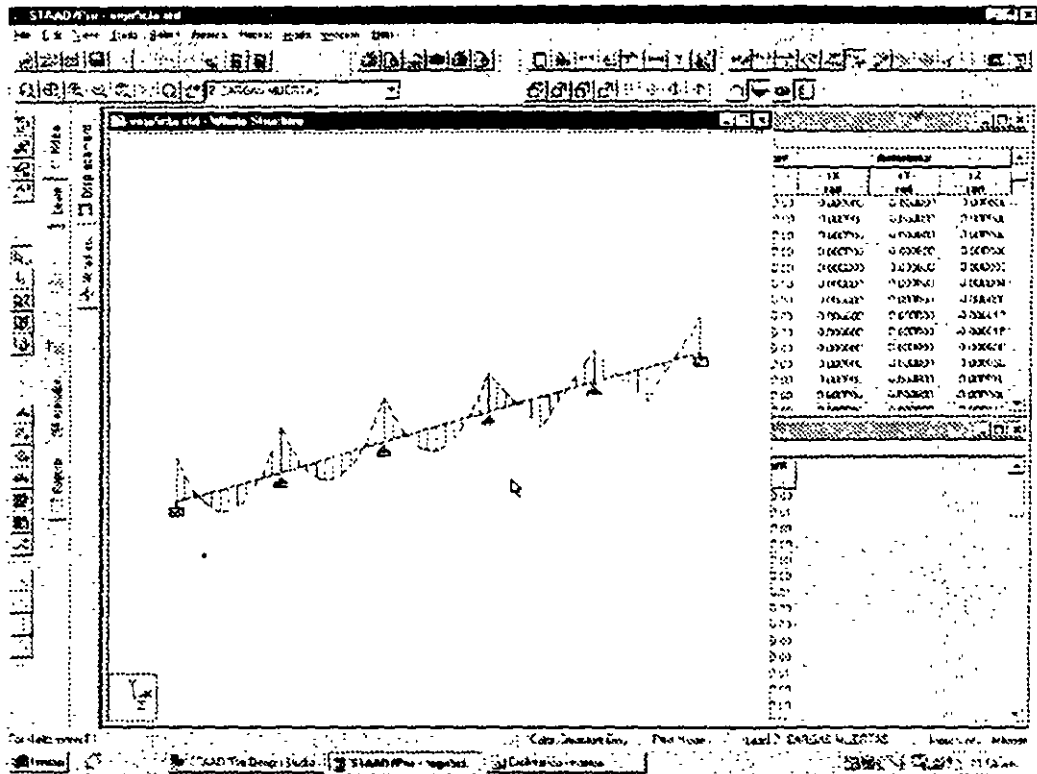


Figura 6.7 Ocultando la deformada

Enseguida, en el mismo menú (Results) seleccionar Bending Moment con lo que se mostrará el diagrama de momentos de la figura 6.8

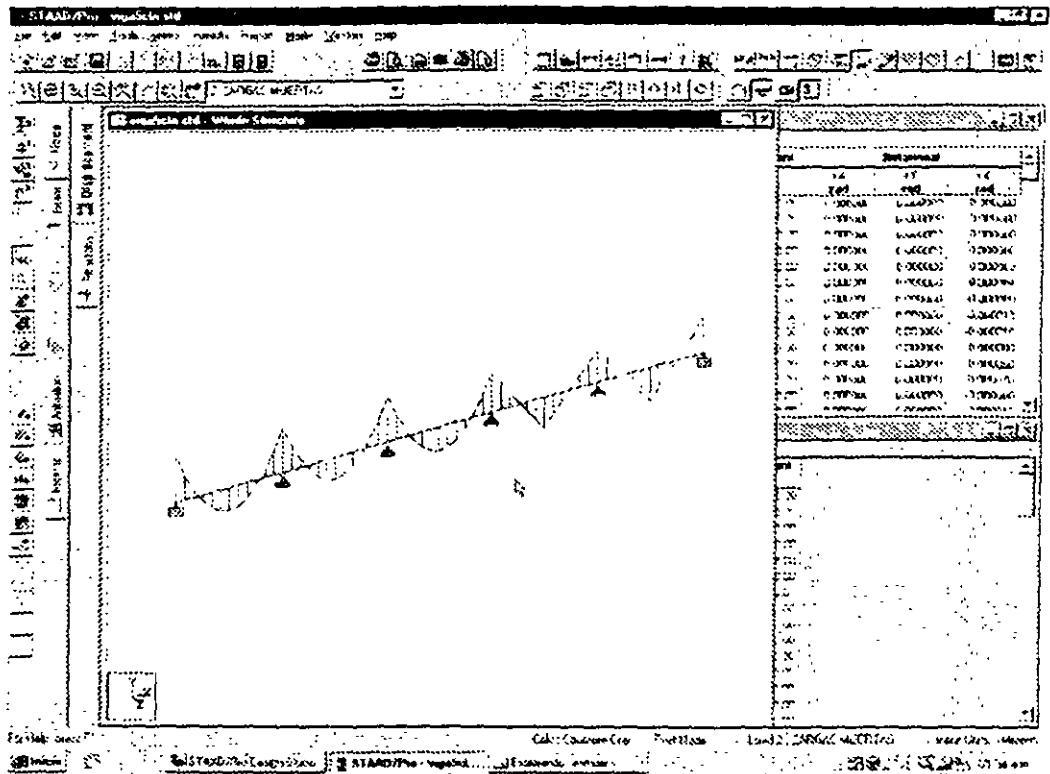


Figura 6.8 Diagrama de Momentos

Una manera de cambiar a otro tipo de elemento mecánico es, por ejemplo, hacer clic en el icono símbolos y etiquetas (Figura 6.8) con lo que se muestra la ventana de la figura 6.9, en ella hacer clic en la carpeta Loads and Results



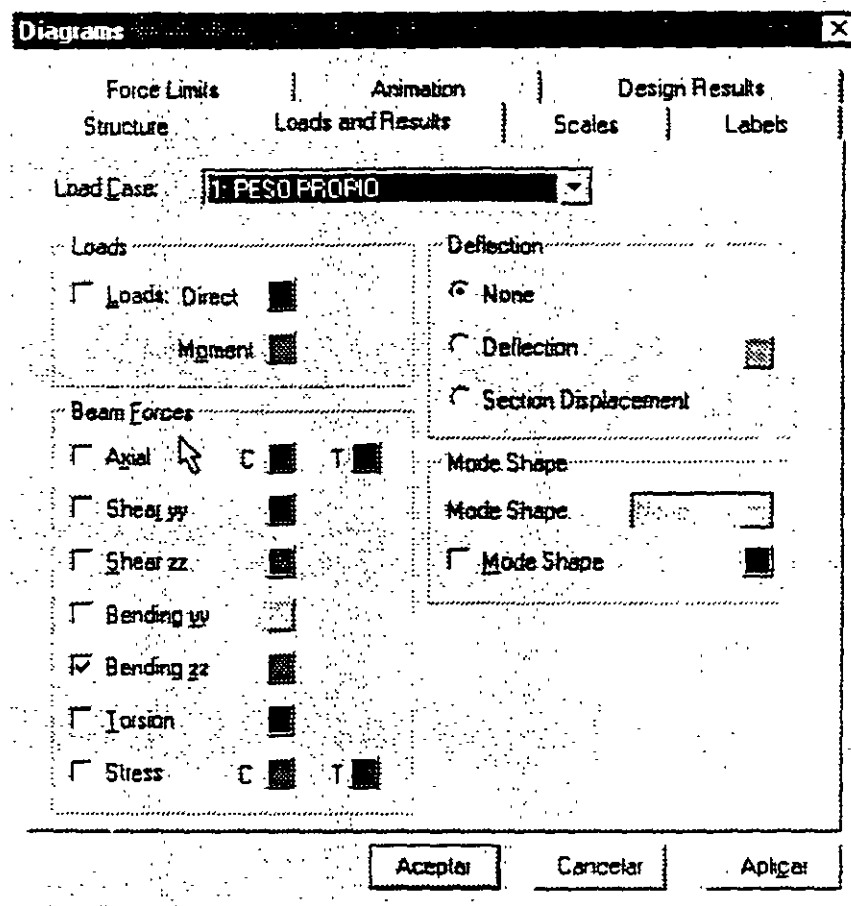


Figura 6.9 Selección de resultados a mostrar



**FACULTAD DE INGENIERÍA UNAM  
DIVISIÓN DE EDUCACIÓN CONTINUA**



**CA 354  
STAAD PRO PARA ANÁLISIS Y DISEÑO  
ESTRUCTURAL**

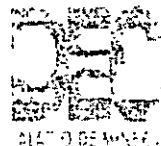
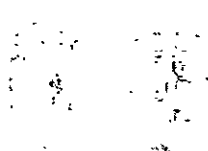
**TEMA:**

**INSTRUCTIVO PARA EL USO DEL PROGRAMA  
( VERSIÓN EN ESPAÑOL )**

**COORDINADOR: ING. JOSÉ LUIS ESQUIVEL ÁVILA**

**DEL 22 AL 29 DE NOVIEMBRE DE 2004**

**PALACIO DE MINERÍA**



SISTEMA DE  
ACCIÓN  
CONTINUA



**Programa 2004**

# CONTENIDO

## PRÓLOGO

## CAPÍTULO 1 EL PROGRAMA STAAD/Pro

- 1.1 Introducción al programa STAAD/Pro.
- 1.2 Introducción al programa STAAD

## CAPÍTULO 2 RECOMENDACIONES PARA 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 GENERACIÓN DE LA ESTRUCTURA

- 4.1 Introducción
- 4.2 Descripción general
- 4.3 Generación de la geometría

- 4.4 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**

## **CAPÍTULO 6 VER RESULTADOS**

- 6.1 Introducción
- 6.2 Ver estructura deformada
- 6.3 Ver diagramas de elementos mecánicos
- 6.4 Ver resultados de diseño
- 6.5 Otras características

## **CAPÍTULO 7 LOS MÓDULOS COMPLEMENTARIOS**

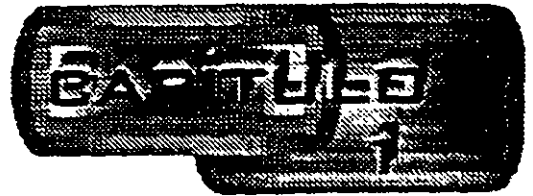
- 7.1 Introducción
- 7.2 Ver archivo de entrada
- 7.3 Ver archivo de salida

## **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 STAAD/Pro



### 1.1 Introducción al programa STAAD/Pro

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 más eficientemente y entre otras cosas facilitarnos la posibilidad de explorar varias alternativas de solución de problemas estructurales o bien considerar más variables en el modelo de las estructuras con el objeto de lograr un mejor entendimiento comportamiento de la estructura.

Tomando en cuenta lo anterior, **STAAD/Pro** 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 al programa que, junto con la relativa sencillez y facilidad de uso son algunas de sus principales características.

**STAAD/Pro** consta básicamente de una serie de módulos (véase figura 1), de ellos, en este instructivo se describirá sólo el módulo **STAAD**, en éste, el usuario puede seleccionar diversas opciones para poder 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.

**STAAD/Pro**, la siguiente generación del programa **STAAD-III**, es el principal software para Análisis y Diseño Estructural de Research Engineers. En **STAAD/Pro**, el enfoque principal está en la productividad. **STAAD/Pro** dirige el proceso completo de la Ingeniería Estructural, desde el desarrollo del modelo hasta el análisis, diseño, bosquejo y detallado de componentes estructurales, **STAAD/Pro** se diseñó para trabajar de manera similar a como se hace en un despacho de Proyecto Estructural.

**STAAD/Pro** es el ambiente de funcionamiento nativo con una ventana para la selección de los componentes que lo constituyen, permitiendo la construcción del modelo así como la visualización y comprobación de resultados. **STAAD/Pro** es el paquete principal con varios componentes optativos, que consisten en lo siguiente:

**STAAD** proporciona el análisis estructural y el diseño integrado de acero, concreto y madera.

**STARDYNE** proporciona características avanzadas de análisis. Construido alrededor de una biblioteca de elemento finito, **STARDYNE** proporciona poderosas opciones de análisis Dinámico, Sísmico, No-lineal, por temperatura, pandeo y otras capacidades avanzadas de análisis.

El ambiente **FEMkit** ofrece modelación de Elemento Finito orientada gráficamente, se complementa con tecnologías para generación de mallas 2D y 3D y herramientas poderosas para la comprobación del modelo.

**Visual DRAW** permite la generación de planos, elevaciones, secciones y dibujos de detalle. Totalmente integrado en el ambiente **STAAD/Pro**, **Visual DRAW** proporciona la generación de dibujos, con capacidades de edición y ploteo.

Los módulos siguientes también están disponibles como componentes de **STAAD/Pro**.

**STAAD.etc** es una colección de módulos de diseño de componentes estructurales, le permite al ingeniero completar el proyecto diseñando cimentaciones, muros de retención, mampostería, conexiones y otros componentes estructurales de utilidad.

**FabriCAD** es una herramienta integrada que realiza el detallado de acero, cálculos de fabricación y montaje, así como la generación de dibujos.

El componente **ADLPIPE** ofrece un sistema confiable para modelado y análisis. Este componente ofrece una solución completa para diseño de plantas industriales.

Poderoso y comprensivo, **STAAD/Pro** está basado en un diseño orientado a objetos que utiliza la tecnología MFC (Microsoft Foundation Class), aprovechando la computación de 32 bits. Una base de datos relacional, con enlaces OLE y DDE, permite intercambio de información entre múltiples aplicaciones integradas con todo el software basado en Windows.

## 1.2 Introducción al programa STAAD

El Sistema **STAAD/Pro** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis y Diseño de Estructuras bajo uno o mas sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura proporcionando, después del análisis, los desplazamientos de los nudos, elementos mecánicos, reacciones, formas modales y resultado del diseño.

STAAD fue desarrollado básicamente 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 estructuras utilizando elementos placa y sólido (elemento finito).

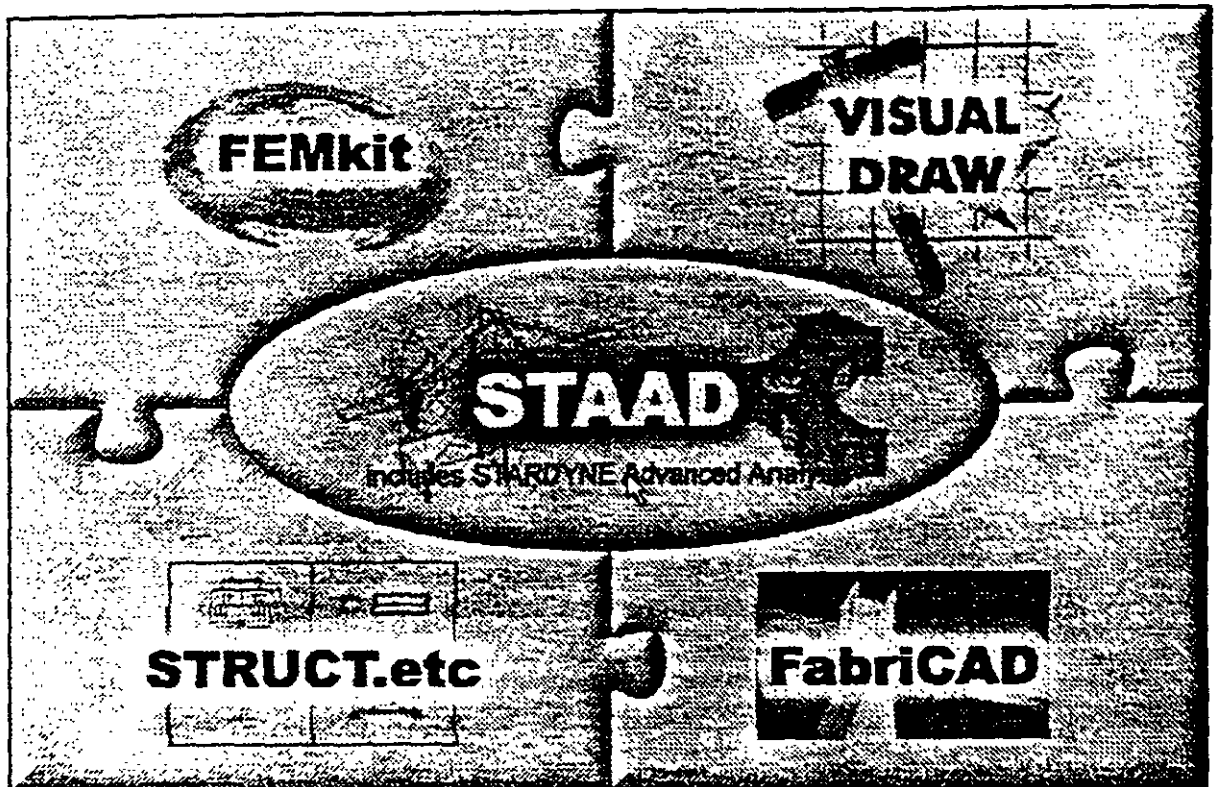


Figura 1.1 STAAD/Pro, programa principal y sistemas que lo integran.

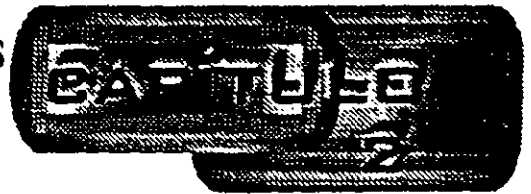
Una de las principales características del programa es la interacción que se puede establecer entre éste y el usuario, sin embargo, debido al número de opciones que el usuario puede activar, se requiere aprender su lenguaje específico para poder utilizarlo, ya que, el usuario puede seleccionar varias opciones y la ejecución de cada una de ellas genera otras más. STAAD es un programa orientado a eventos (seleccionar un elemento con el mouse, 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, 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 o generar la estructura, en el capítulo

5 se presentan las opciones de análisis, en el capítulo 6 se muestran las opciones para ver resultados del Análisis y Diseño, en el capítulo 7 se describen los módulos complementarios, el capítulo 8 contiene algunos ejemplos con la correspondiente interpretación de los resultados obtenidos por el programa STAAD, por último, en el capítulo 9 se incluyen algunos comentarios y sugerencias finales.



# RECOMENDACIONES PARA EL USO DEL PROGRAMA



## 2.1 Ejecución del programa

Una vez instalado, para iniciar la ejecución del programa STAAD, se puede hacer clic en inicio luego deslizar el puntero del ratón hasta programas, enseguida desplazarlo a la derecha y hacia abajo hasta la carpeta STAAD/Pro y por último a la derecha y hacia arriba (en la computadora donde se preparó este instructivo), para, finalmente hacer clic en STAAD (véase figura 2.1), con lo cual aparece la ventana de la figura 1.1, después de hacer clic en su zona central (STAAD) se muestra la ventana de la figura 2.2.

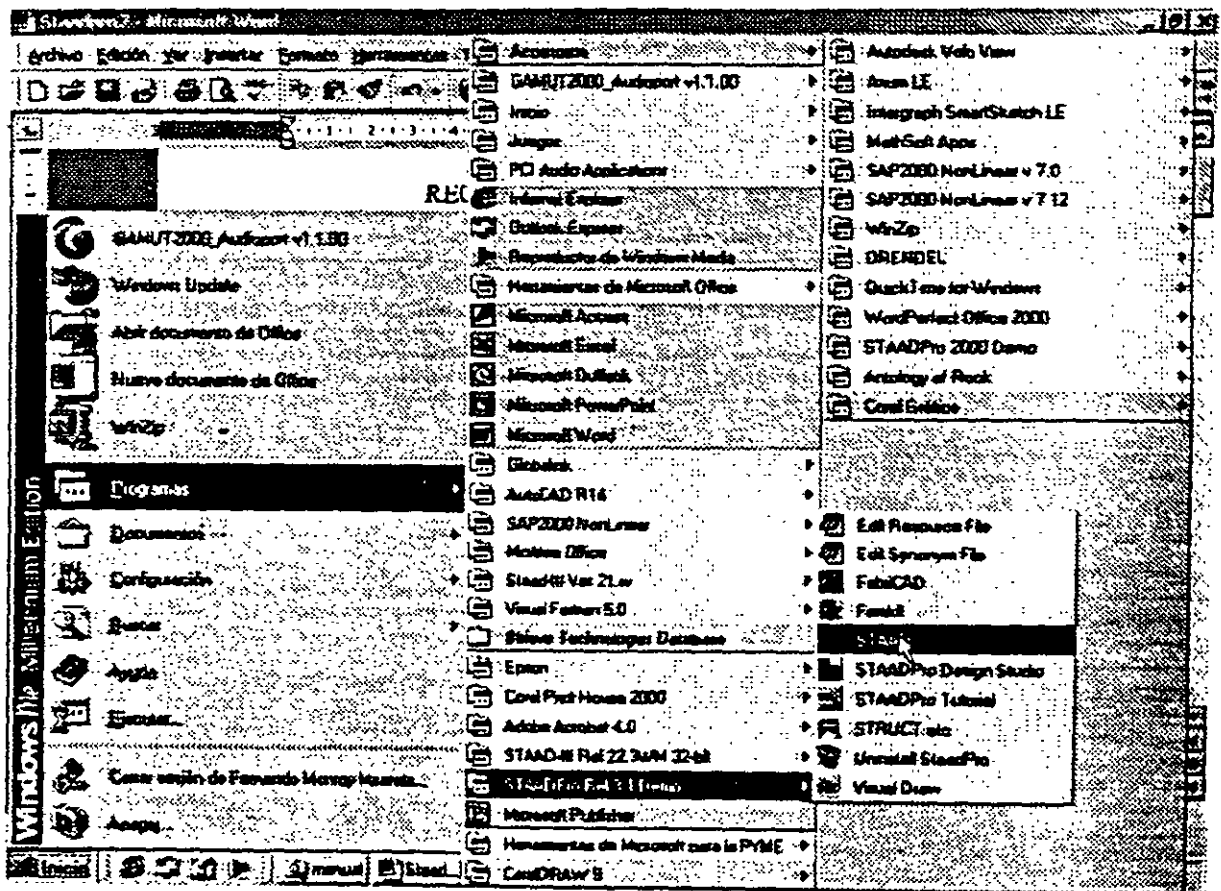


Figura 2.1 Ejecución del programa STAAD/Pro.

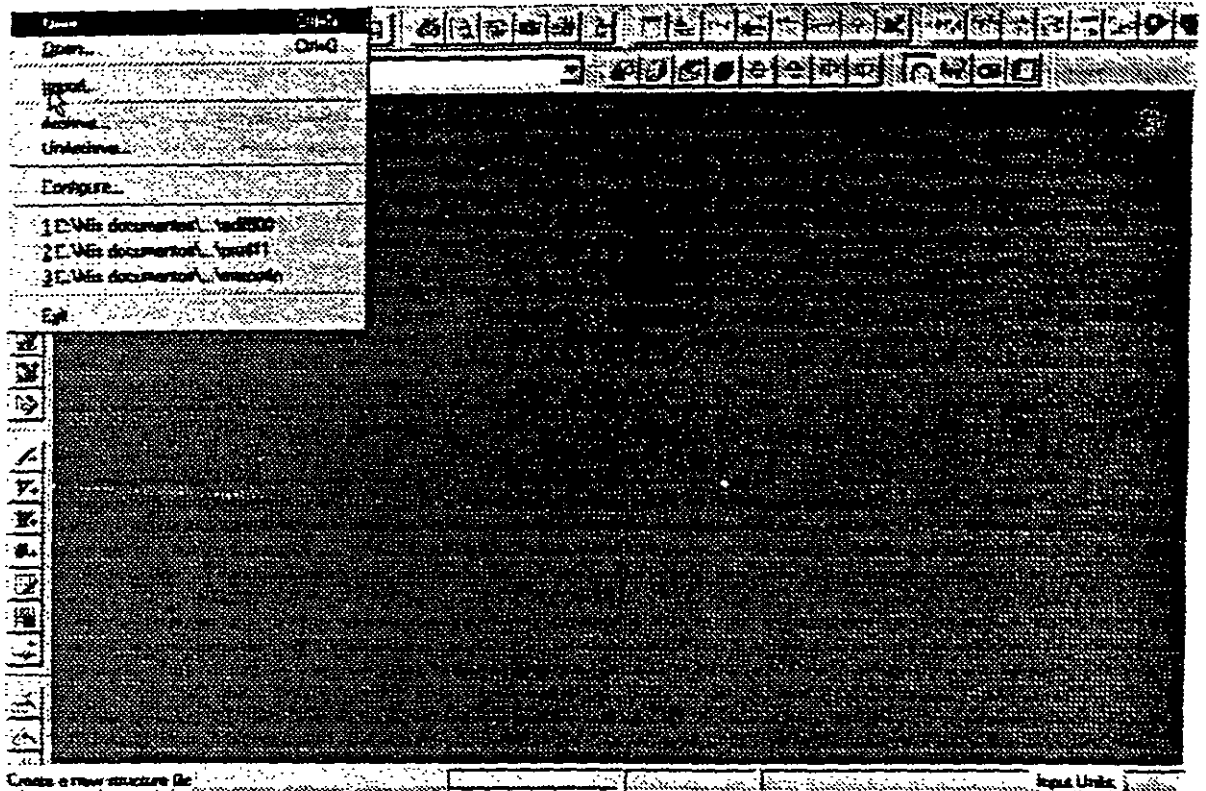


Figura 2.2 Inicio del programa STAAD.

## 2.2 Tipo de estructura y datos generales

Para iniciar la introducción de datos generales y el tipo de estructura por analizar se utiliza la opción **N**ew del menú **F**ile (véase figura 2.2) mostrándose la ventana de la figura 2.3

STAAD permite manejar la estructura a analizar como una de las siguientes:

- Truss
- Plane
- Floor
- Space

Para el caso de la estructura tipo **Truss** (armadura) esta puede ser plana o en 3 dimensiones (3D) en ambos casos en el análisis sólo se considerará el efecto axial.

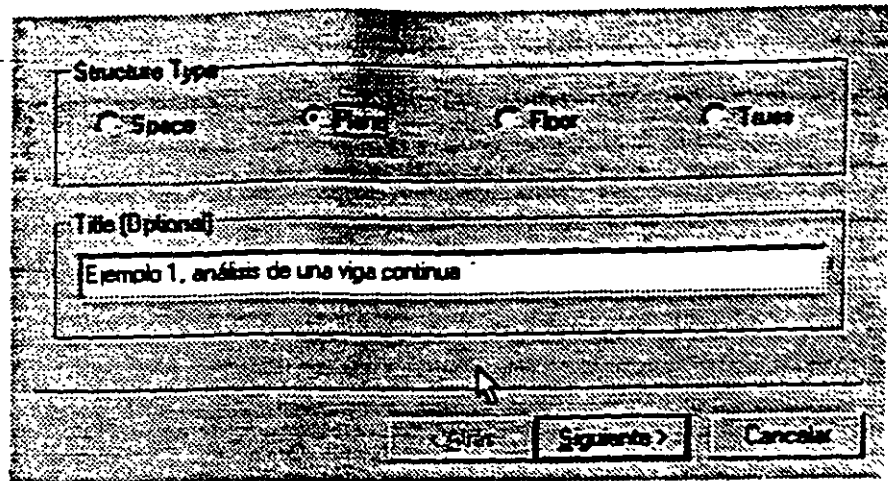


Figura 2.3 Datos generales al inicio del programa STAAD.

En la estructura tipo **Plane** se consideran cortante y axial en el plano de la estructura y flexión perpendicular a ese plano.

El tipo **Floor** permite analizar estructuras con acciones perpendiculares a su plano (retículas) considerando flexión en el plano, torsión, y cortante.

El caso general lo constituye el tipo **Space** en donde se consideran flexión y cortante en dos direcciones, torsión y axial, y 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 (por ejemplo diafragma rígido).

La opción que corresponda a la estructura por analizar, y la introducción de un título (opcional) como identificación que se incluirá dentro del archivo de datos, se realiza en la ventana de la figura 2.3, una vez introducidos los datos y seleccionado el tipo de estructura y después de hacer clic en el cuadro **Siguiente** se muestra la ventana de la figura 2.4, en donde han de seleccionarse las unidades para las fuerzas y longitudes de los datos de la estructura que se introducirán posteriormente (geometría, propiedades, cargas, etc.)

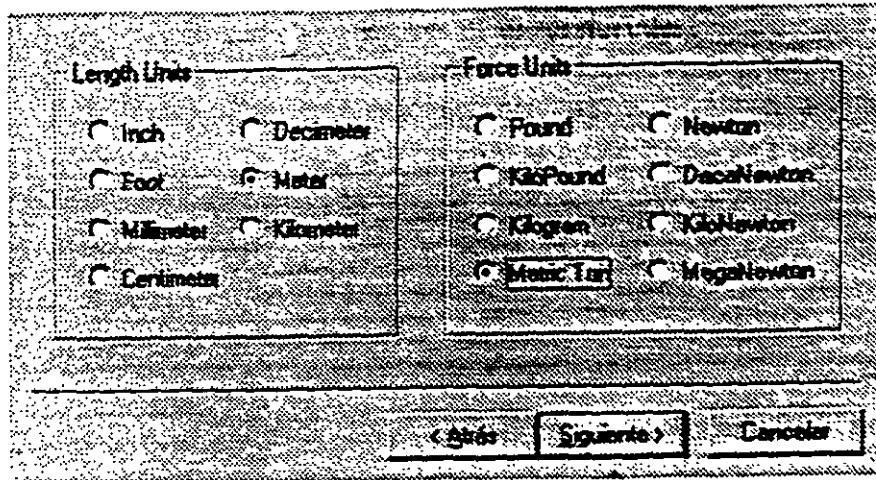


Figura 2.4 Datos de unidades al inicio del programa STAAD.

Una vez seleccionadas las unidades se hace clic en Siguiente para que se despliegue la ventana de la figura 2.5, finalmente, Finish conduce a la ventana de la figura 2.6 que es la ventana o módulo principal de STAAD.

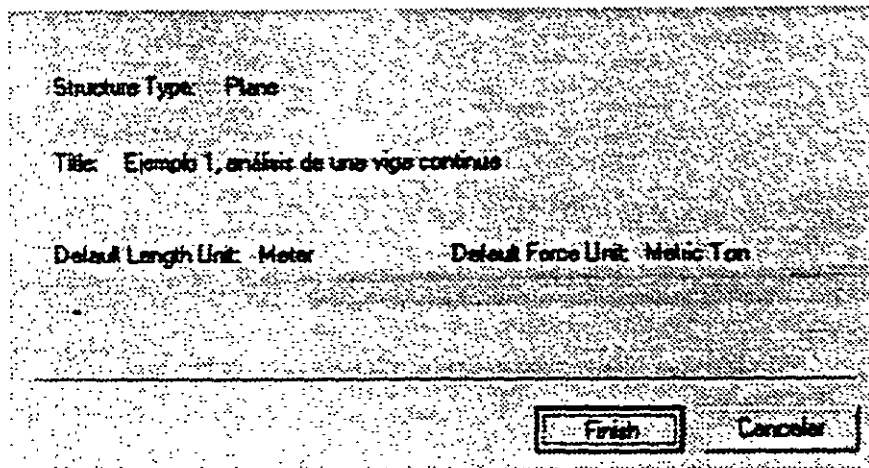


Figura 2.5 Datos seleccionados por el usuario al inicio del programa STAAD.

Obsérvese que en esta ventana (figura 2.6), en general, están contenidos algunos elementos típicos de varios programas desarrollados para ambiente o plataforma **Windows**, es decir, una barra de título (extremo superior de la ventana), una de menús desplegables (File, Edit, View, etc.), barras de iconos (algunos típicos de varios programas, y otros propios STAAD), una barra de estado en el extremo inferior de la ventana (for help press ....). En el extremo izquierdo se muestran algunos iconos y varias opciones agrupadas por categorías (Job, Setup, Geometry, etc.), seguidas por un área con fondo blanco que se utilizará para desplegar gráficamente la geometría y algunas características de la estructura

(apoyos, cargas, etc.), el área restante (a la derecha de la anterior) la utiliza el programa STAAD para mostrar, generalmente, información de los elementos de la estructura en forma numérica (coordenadas de los nudos, incidencias de las barras, fuerzas, etc.).

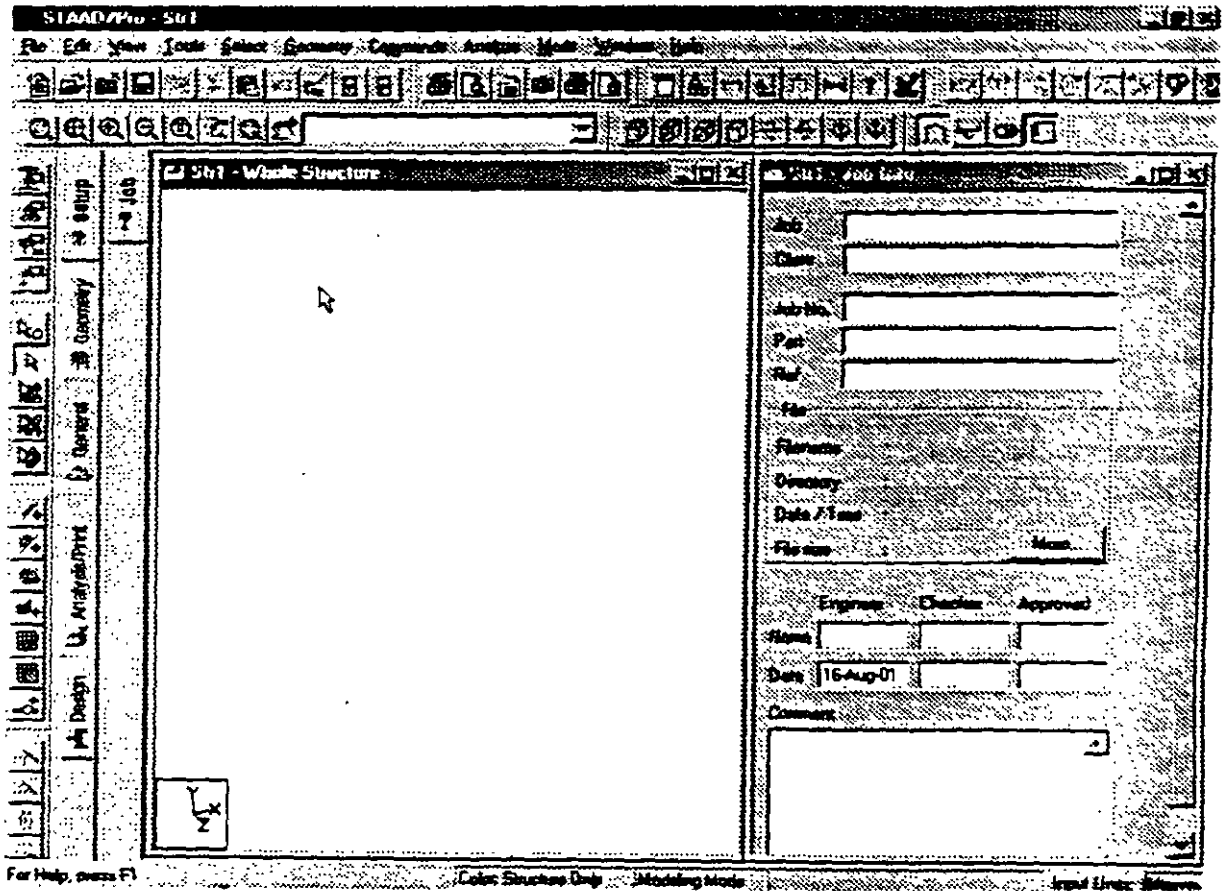


Figura 2.6 Ventana completa del programa STAAD.

### 2.3 Definición de la geometría

Antes de iniciar la ejecución del programa STAAD es conveniente tener completamente bien definida la geometría del modelo. La estructura por analizar se idealizará mediante una serie de elementos estructurales conectados entre sí, los cuales, de acuerdo a sus características o con fines de análisis se podrán modelar como elementos barra (trabes, columnas, diagonales, etc.), elementos finitos placa (losas, muros) o elementos finitos sólidos (elementos tridimensionales), 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 o ubicación de los elementos (barra, placa, sólido) se logra localizando sus nudos extremos, por ejemplo, en un sistema coordenado cartesiano. Proporcionando las coordenadas de esos nudos (o su longitud si es que el elemento barra es paralelo a alguno de los ejes de referencia) así como los nudos extremos (incidencias) de la barra queda definida su posición.

No es necesario numerar los nudos que forman parte de la estructura ya que el programa lo hace de manera automática. Es conveniente localizar nudos en donde se tenga cambio de propiedades geométricas o elásticas, recordando que el elemento barra requiere de dos nudos para posicionarlo, el elemento placa 3 ó 4 y el sólido desde 4 hasta 8 nudos (véase figura 2.7).

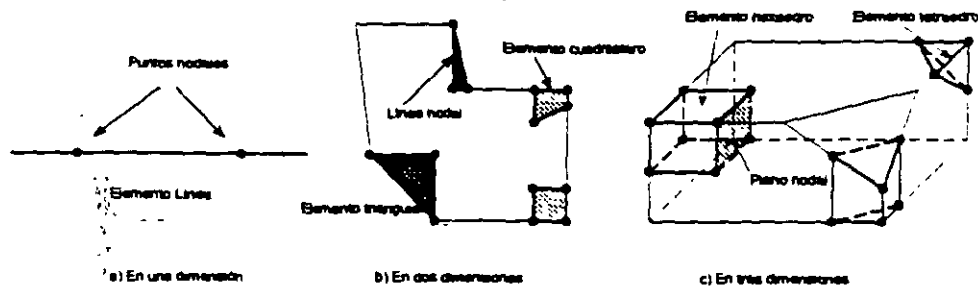


Figura 2.7 Tipos de nudos.

## 2.4 Definición de las propiedades geométricas de los elementos

Los siguientes son algunos de los tipos de elementos barra que permite manejar STAAD

- Prismáticos (rectangular, circular, etc.).
- Elementos estándar de acero.
- Elementos de acero definidos por el usuario.
- Sección I de peralte variable.
- Asignarles una forma específica.

Para elementos barra prismáticos de forma arbitraria se requiere proporcionar las siguientes propiedades referidas a ejes locales y centroidales de la barra.

$AX$  = Área de la sección transversal.

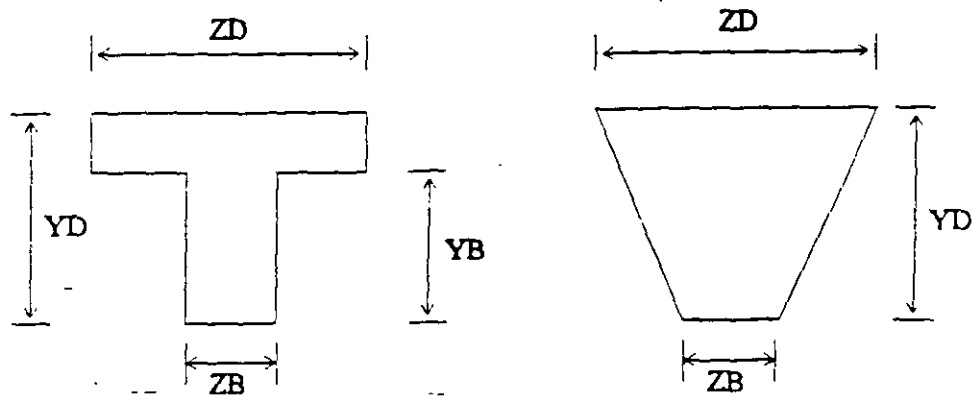
$IX$  = Constante de torsión.

$IY$  = Momento de inercia al rededor del eje y.

$IZ$  = Momento de inercia al rededor del eje z.

$AY$  = Área de cortante en dirección  $y$ .  
 $AZ$  = Área de cortante en dirección  $z$ .  
 $YD$  = Dimensión de la sección en dirección  $y$ .  
 $ZD$  = Dimensión de la sección en dirección  $z$ .

Para barras de sección trapezoidal o T el significado de  $YB$  y  $ZB$  se muestra en la figura 2.8.



**Figura 2.8** Características de secciones T y trapezoidal

Si al programa se le solicita el cálculo de esfuerzos o el diseño (revisión) en concreto o acero será necesario proporcionar los valores de  $YD$  y  $ZD$  en caso contrario se pueden omitir.

Si no se proporcionan las áreas de cortante el programa no considera ese efecto en el análisis, esto sólo es posible definiendo a las barras de tipo "General" e introduciendo los valores de sus propiedades.

Para secciones específicas (rectangular, circular, etc.) las propiedades son obtenidas por el programa sólo con proporcionar las dimensiones características según la forma de la sección transversal de la barra (p.ej  $B$  y  $D$  para la sección rectangular,  $D$  para la circular, etc.) en este caso serán considerados los efectos de deformación por cortante.

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 geométrica 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 para el análisis.

El programa STAAD permite asignar las propiedades geométricas de los elementos barra de acuerdo a una tabla de perfiles de acero estándar ( P.ej. tabla AISC) o tomarlas de una tabla definida por el usuario.

En el caso de secciones I de peralte variable los datos son los que se muestran en la figura 2.9

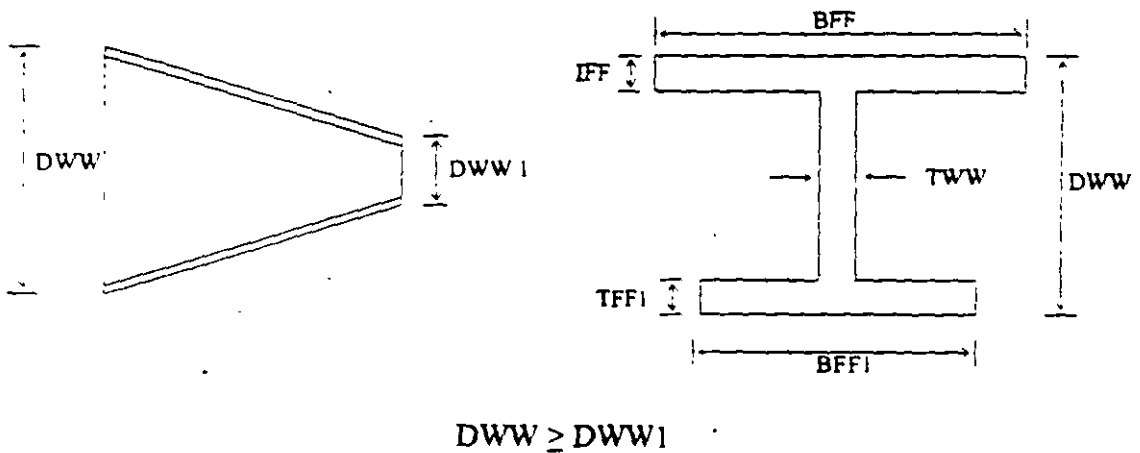


Figura 2.9 Características de la sección I de peralte variable.

Al programa se le pueden dar instrucciones para que, de manera automática, maneje a los elementos con secciones de formas específicas (sección T, o formada por uno o dos ángulos, etc )

Para el caso de los elementos placa será necesario proporcionar el espesor de la placa en cada esquina, para el sólido no es necesario proporcionar propiedades geométricas sólo constantes elásticas



## 2.7 Elección del tipo de análisis y los resultados

STAAD permite realizar un análisis elástico lineal de 1<sup>er</sup> orden y también de 2<sup>do</sup> orden, en el segundo caso se consideran efectos P- $\Delta$ , o un análisis no lineal por geometría en cuanto a considerar la geometría deformada de la estructura, por lo anterior habrá que decidir 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 resultados de diseño (revisión), y de que elementos se requieren; por ejemplo: algunos o todos los nudos, algunos o todos los elementos (barras, placas, etc.). Gráficas de la deformada, de algún marco o de toda la estructura, etc. Lo anterior se tendrá que especificar para una, algunas o todas las condiciones de carga y/o combinaciones. Si el usuario no selecciona o define los elementos (nudos, barras, etc.) y las condiciones y/o combinaciones, la impresión la realiza para todos los elementos y todos los sistemas de fuerza existentes.

## 2.8 Diseño de elementos

STAAD permite diseñar o revisar elementos de acero, concreto y madera por lo que será necesario especificar un código aplicable a utilizar (ACI, AISC, LRFD, ASSTHO, etc.), así como proporcionar los valores de los parámetros a utilizar ( $f_c$ ,  $f_y$ , etc.), e indicar los elementos que se diseñarán y el criterio a seguir para su diseño (viga, columna, etc.).

## 2.5 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 y sólido) como son E (Módulo elástico), y  $\mu$  (relación de Poisson) y, mediante la siguiente expresión se obtiene el modelo de rigidez a cortante.

$$G = \frac{E}{2(1 + \mu)}$$

Para incluir el peso propio es necesario proporcionar el peso volumétrico, si se consideran efectos de temperatura será necesario especificar el coeficiente lineal de dilatación térmica.

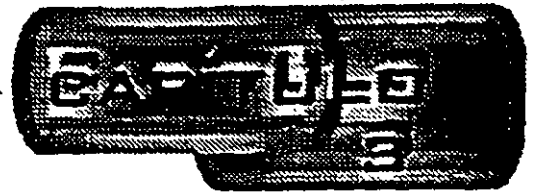
## 2.6 Tipos de fuerzas y combinaciones de carga

Es necesario tener completamente identificados 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, contar con las características de las fuerzas (tipo, magnitud, dirección, sentido y punto de aplicación) que componen cada sistema de fuerzas (condición de carga).

Por ejemplo, una condición de carga puede ser la carga muerta, que puede estar formada por fuerzas uniformes en algunas barras simulando el peso, por ejemplo, de los muros divisorios, o fuerzas concentradas que representan, por ejemplo, el peso de tanques, etc. Otra condición de carga, el sismo, puede ser representado por una serie de fuerzas estáticas (sismo estático) o dinámicas aplicadas a 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, pasillos, escaleras, etc.).

Los sistemas de carga independientes o primarios (como los llama el programa) pueden ser utilizados para formar sistemas de carga dependientes de los anteriores, 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 de referencia al Reglamento de Construcciones para el D.F. pensando en una estructura del grupo A, localizada en el D. F. una combinación será 1.5 de la carga muerta + 1.5 de la carga viva máxima, por lo que el factor de carga o participación de las condiciones anteriores 1 y 2 es 1.5, siendo 1 y 2 las condiciones de carga respectivas (1 la carga muerta y 2 la viva).

## MÓDULOS DEL PROGRAMA DESCRIPCION GENERAL



### 3.1 Introducción

Para poder introducir y/o hacer cambios a los datos o características de la estructura en el programa STAAD, además de contar con un editor en línea modo texto, principalmente tiene un editor gráfico integrado desde donde también se puede invocar al editor modo texto. Casi con cualquiera de los dos editores se puede:

- Manejar (Definir, mover, copiar, borrar, etc.) elementos estructurales (nudos, barras, placas sólidos).
- Especificar tipos de apoyo (fijo o con grados de libertad, resortes, apoyos inclinados, tipo "Foundation", etc.).
- Asignar propiedades geométricas de los elementos barra de acuerdo a: una tabla de perfiles estándar (AISC por ejemplo), una tabla previamente definida por el usuario, secciones prismáticas (circular rectangular, Te, trapezoidal, I de peralte constante o con variación lineal etc.), o introducir sus características particulares (propiedades geométricas, orientación de su sección transversal, etc.).
- Especificar espesores de los elementos placa.
- 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. 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, concreto, etc.) o se pueden introducir valores particulares.
- Especificar que elementos desempeñarán sólo una función estructural específica por ejemplo cable, barra en compresión, en tensión, armadura (tensión o compresión), con articulación o liberación a algún elemento mecánico en un extremo, ignorarlos y otras opciones. También se puede definir diafragmas rígidos.

- Definir cargas variables (móviles) pudiendo ser definidas por el usuario (tren de cargas concentradas), de acuerdo a AASHTO(HS20, HS15, H20, H15) o bien tomadas de un archivo externo.
- Especificar fuerzas definidas en el tiempo (fuerza-tiempo o aceleración-tiempo) tomando los valores de un archivo existente o introduciéndolos de acuerdo a una función (seno o coseno) proporcionando características dinámicas (amplitud y frecuencia), definiendo el lapso de tiempo de actuación de la fuerza así como también considerar el amortiguamiento.
- Definir características para generar cargas definidas por el UBC (Uniform Building Code).
- Definir cargas de viento especificando (hasta cinco) intensidades (presiones) actuando respectivamente en n alturas.
- Especificar fuerzas estáticas aplicadas a los nudos, desplazamientos prescritos de los apoyos, peso propio, etc. Para barras: fuerzas y/o momentos uniformes, fuerzas y/o momentos concentrados, fuerzas con variación lineal, presión hidrostática. Para los elementos placa: presión uniforme, lineal, hidrostática.
- Asignar carga uniforme por unidad de área en un nivel específico y en cierta área.
- Incluir en las barras, presfuerzo, incrementos de temperatura y ajustes en la longitud inicial de los elementos.
- Seleccionar el tipo de análisis como puede ser: elástico-lineal de primer orden, análisis no lineal P- $\Delta$ , análisis de segundo orden (especificando el número de iteraciones) y análisis dinámico.
- Y otras opciones más.

### 3.2 Descripción general

En la figura 3.1 se muestra la ventana deslizante correspondiente a la opción o menú File

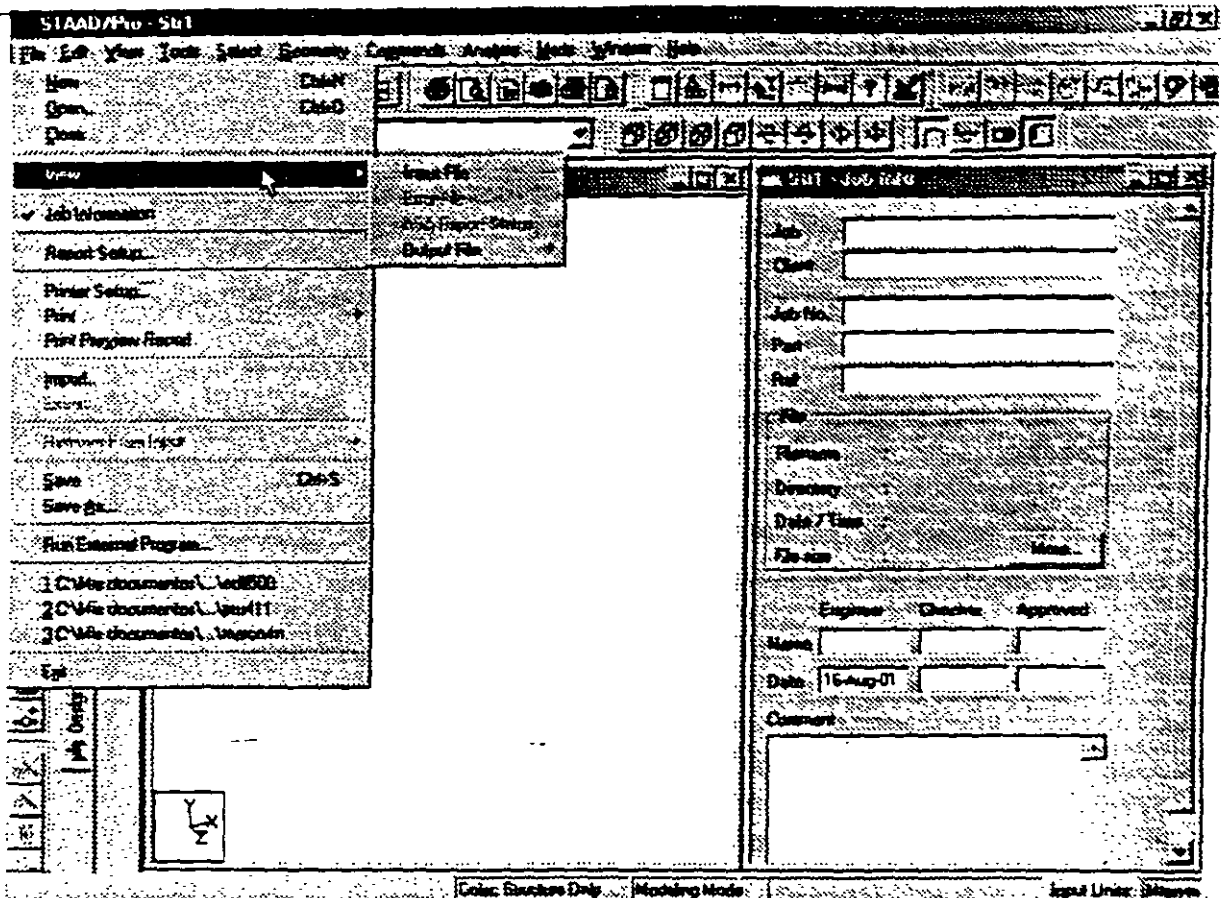


Figura 3.1 Menú File de STAAD

Algunas de las opciones del menú File permiten:

- |   |   |
|---|---|
| <b><u>N</u>ew</b>                               | Iniciar un problema nuevo   |
| <b><u>O</u>pen</b>                              | Abrir un archivo existente con datos de alguna estructura.  |
| <b><u>V</u>iew</b>                              | Ver el contenido del archivo de datos ( <b>I</b> nput <b>F</b> ile) o el archivo de resultados ( <b>O</b> utput <b>F</b> ile) |
| <b><u>P</u>rinter Setup</b>                     | Seleccionar una impresora o bien modificar sus propiedades  |
| <b><u>P</u>rint Input File</b>                  | Imprimir el contenido de un archivo de datos.   |
| <b><u>P</u>review <u>P</u>rint <u>I</u>nput</b> | Ver el contenido del archivo de datos antes de imprimir.  |
| <b><u>S</u>ave, <u>S</u>ave As</b>              | Permiten guardar el archivo de datos.   |

**Exit** Cerrar el programa

Existen, dentro del menú anterior, otras opciones que pueden ser de uso no muy frecuente.

Ahora en la figura 3.2 se presentan las opciones del menú Edit

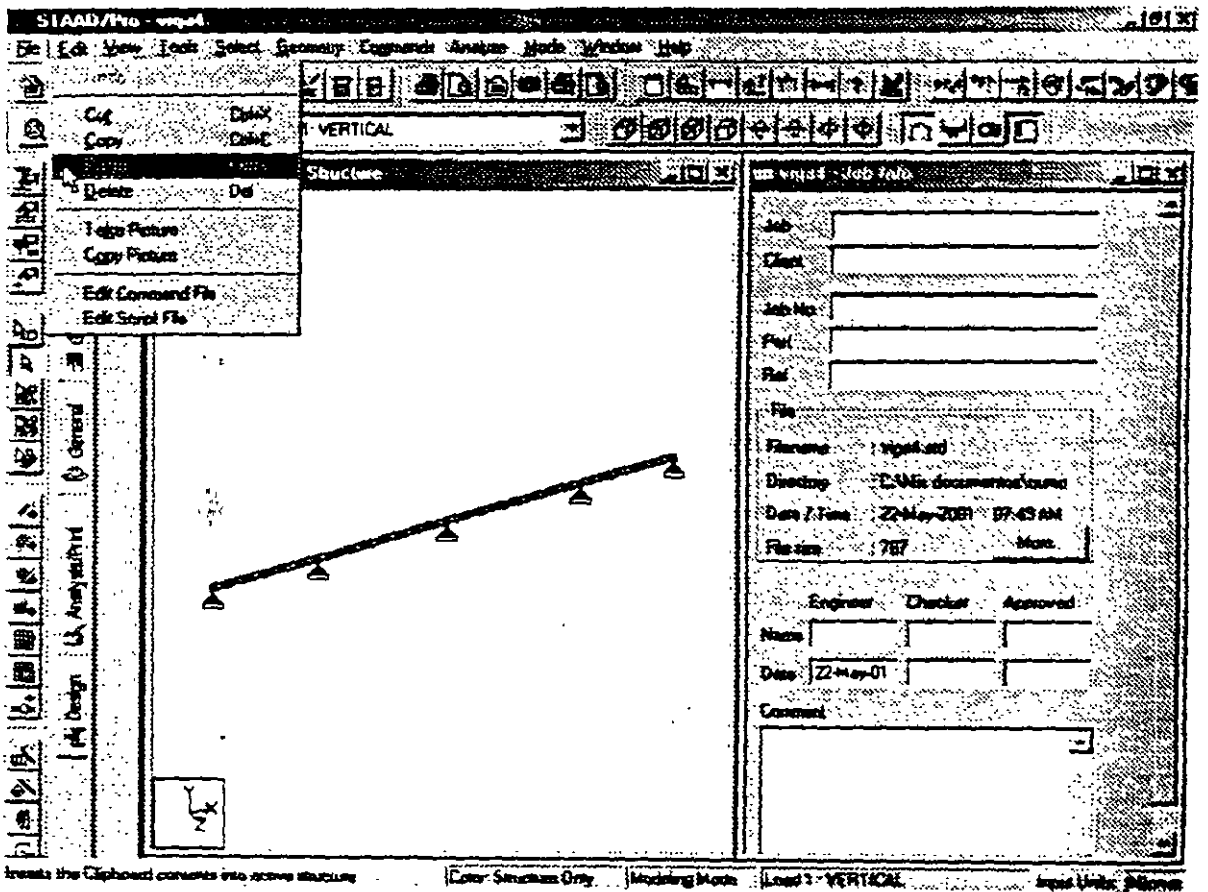


Figura 3.2 Menú Edit del módulo STAAD.

Las opciones del menú Edit permiten:

**Undo** Deshacer la acción anterior (última).

**Cut** Suprimir(borrar) los elementos seleccionados de la estructura (p.ej. barras que aparecen en color en el área de dibujo) y los coloca en la memoria temporal.

- Copy** Copia a la memoria temporal los elementos seleccionados de la estructura (para poder insertarlos posteriormente), esta opción no borra a los elementos
- Paste** Insertar los elementos almacenados en la memoria temporal.
- Del** Borra los elementos seleccionados de la estructura.
- Edit command file** Ejecuta el editor modo texto mostrando el contenido del archivo de datos al que pueden realizarse cambios (adicionar comandos o datos, suprimir o modificar parte de la información etc.).



Software licensed to Unknown User

Job No.	Sheet No. 1	Rev.
Part		
Ref		
By	Date 04-Oct-00	Chk
Client	File V3cslpro.std	Date/Time 23-Sep-2001 22:29

### Job Information

	Engineer	Checked	Approved
Name:			
Date:	04-Oct-00		

Structure Type | SPACE FRAME

Number of Nodes	4	Highest Node	4
Number of Elements	3	Highest Beam	3

Number of Basic Load Cases	1
Number of Combination Load Cases	0

Included in this printout are data for:

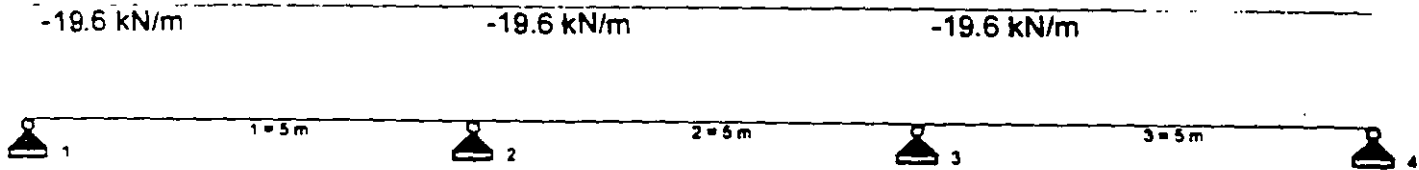
All	The Whole Structure
-----	---------------------





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	04-Oct-00	
Client	File	Date/Time
	V3c1apro.std	23-Sep-2001 22:26

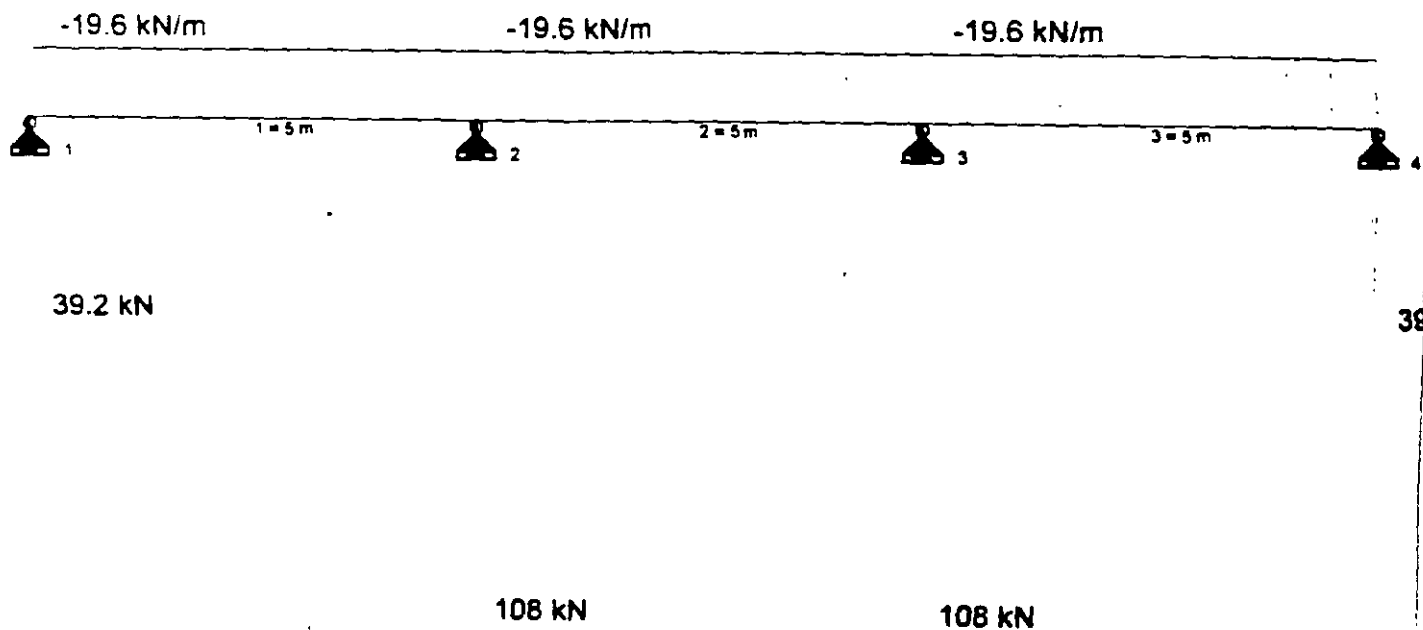


```
STAAD SPACE EJEMPLO
START JOB INFORMATION
ENGINEER DATE 04-Oct-00
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER MTON
JOINT COORDINATES
1 0 0 0; 2 5 0 0; 3 10 0 0; 4 15 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS YD 0.5 ZD 0.25
UNIT METER KN
CONSTANTS
E 2.5e+007 MEMB 1 TO 3
POISSON 0.17 MEMB 1 TO 3
DENSITY 24 MEMB 1 TO 3
ALPHA 1.1e-005 MEMB 1 TO 3
UNIT METER MTON
SUPPORTS
1 TO 4 PINNED
LOAD 1 VERTICAL
MEMBER LOAD
1 TO 3 UNI GY -2
PERFORM ANALYSIS PRINT ALL
PRINT ANALYSIS RESULTS
FINISH
```



Software licensed to Unknown User

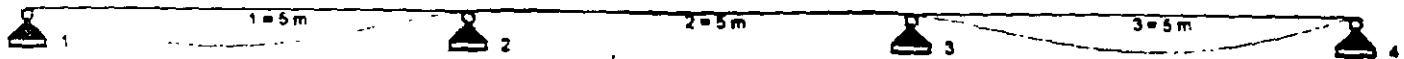
Job No	Sheet No <b>1</b>	Rev
Part	Ref	
By	Date <b>04-Oct-00</b>	Cr'd
Case#	File <b>V3clapro.std</b>	Date/Time <b>23-Sep-2001 22:4</b>





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	04-Oct-00	
Client	File	Date/Time
	V3ctapro.std	23-Sep-2001 22:41





Software licensed to Unknown User

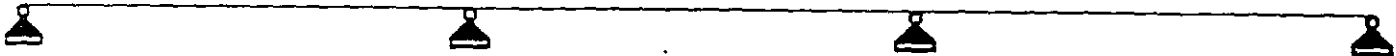
Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	04-Oct-00	
File	Date/Time	
V3clepro.std	23-Sep-2001 22:41	

Job Title

Client

Max: 48.9 kNm

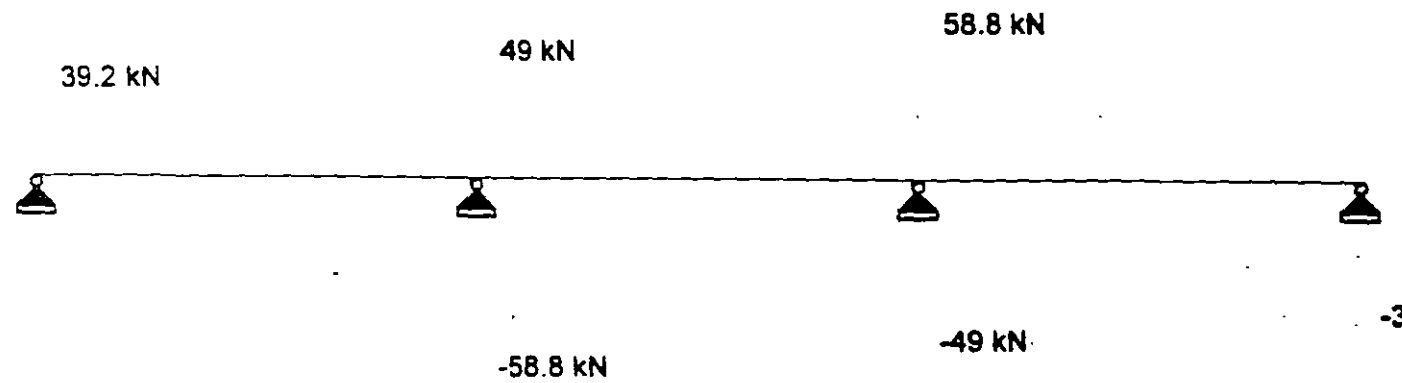
Max: 48.9 kNm





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	04-Oct-00	
Client	File	Date/Time
	V3c1epr0.std	23-Sep-2001 22:4



```

*****
*
*          STAAD/Pro STAAD-III
*          Revision 3.1
*          Proprietary Program of
*          RESEARCH ENGINEERS, Inc.
*          Date=   SEP 23, 2001
*          Time=   23:24:35
*
*          USER ID: Unknown User
*****

```

1. STAAD SPACE EJEMPLO
2. START JOB INFORMATION
3. ENGINEER DATE 04-CCT-00
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER MTON
7. JOINT COORDINATES
8. 1 0 0 0; 2 5 0 0; 3 10 0 0; 4 15 0 0
9. MEMBER INCIDENCES
10. 1 1 2; 2 2 3; 3 3 4
11. MEMBER PROPERTY AMERICAN
12. 1 TO 3 PRIS YD 0.5 ZD 0.25
13. UNIT METER KN
14. CONSTANTS
15. E 2.5E+007 MEMB 1 TO 3
16. POISSON 0.17 MEMB 1 TO 3
17. DENSITY 24 MEMB 1 TO 3
18. ALPHA 1.1E-005 MEMB 1 TO 3
19. UNIT METER MTON
20. SUPPORTS
21. 1 TO 4 PINNED
22. LOAD 1 VERTICAL
23. MEMBER LOAD
24. 1 TO 3 UNI GY -2
25. PERFORM ANALYSIS PRINT ALL

P R O B L E M   S T A T I S T I C S

---

```

NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =      4/      3/      4
ORIGINAL/FINAL BAND-WIDTH =      1/      1
TOTAL PRIMARY LOAD CASES =      1, TOTAL DEGREES OF FREEDOM =      12
SIZE OF STIFFNESS MATRIX =      72 DOUBLE PREC. WORDS
REQRD/AVAIL. DISK SPACE = 12.00/ 2047.7 MB, EXMEM = 1798.5 MB

```

EJEMPLO -- PAGE NO. 2

LOADING 1 VERTICAL

---

MEMBER LOAD - UNIT MTON METE

MEMBER	UDL	L1	L2	CON	L	LIN1	LIN2
--------	-----	----	----	-----	---	------	------

```

1   -2.000 GY   0.00   5.00
2   -2.000 GY   0.00   5.00
3   -2.000 GY   0.00   5.00

```

```

***TOTAL APPLIED LOAD ( MTON METE ) SUMMARY (LOADING 1 )
SUMMATION FORCE-X =      0.00
SUMMATION FORCE-Y =     -30.00
SUMMATION FORCE-Z =      0.00

```

```

SUMMATION OF MOMENTS AROUND THE ORIGIN-
MX=      0.00 MY=      0.00 MZ=     -225.00

```

```

-- Processing Element Stiffness Matrix.          23:24:35
-- Processing Global Stiffness Matrix.          23:24:35
-- Processing Triangular Factorization.         23:24:35

```

```

***WARNING - IMPROPER LOAD WILL CAUSE INSTABILITY AT JOINT 4
DIRECTION = MX PROBABLE CAUSE MODELING PROBLEM -0.728E-11
-- Calculating Joint Displacements.             23:24:35
-- Calculating Member Forces.                  23:24:35

```

```

***TOTAL REACTION ( MTON METE ) SUMMARY

```

```

LOADING 1

```

```

SUM-X=      0.00 SUM-Y=     30.00 SUM-Z=      0.00

```

```

SUMMATION OF MOMENTS AROUND ORIGIN-

```

```

MX=      0.00 MY=      0.00 MZ=     225.00

```

```

EXTERNAL AND INTERNAL JOINT LOAD SUMMARY-

```

JT	EXT FX/ INT FX	EXT FY/ INT FY	EXT FZ/ INT FZ	EXT MX/ INT MX	EXT MY/ INT MY	EXT MZ/ INT MZ
1	0.00	-5.00	0.00	0.00	0.00	-4.17
	0.00	1.00	0.00	0.00	0.00	4.17
2	0.00	-10.00	0.00	0.00	0.00	0.00
	0.00	-1.00	0.00	0.00	0.00	0.00
3	0.00	-10.00	0.00	0.00	0.00	0.00
	0.00	-1.00	0.00	0.00	0.00	0.00
4	0.00	-5.00	0.00	0.00	0.00	4.17
	0.00	1.00	0.00	0.00	0.00	-4.17

```

EJEMPLO

```

```

-- PAGE NO. 3

```

```

***** END OF DATA FROM INTERNAL STORAGE *****

```

```

26. PRINT ANALYSIS RESULTS

```

```

EJEMPLO

```

```

-- PAGE NO. 4

```



JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = SPACE

JOINT	LOAD	X-TRANS	Y-TRANS	Z-TRANS	X-ROTAN	Y-ROTAN	Z-ROTAN
1	1	0.0000	0.0000	0.0000	0.0000	0.0000	-0.0010
2	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0003
3	1	0.0000	0.0000	0.0000	0.0000	0.0000	-0.0003
4	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0010

EJEMPLO

-- PAGE NO. 5

SUPPORT REACTIONS -UNIT MTON METE STRUCTURE TYPE = SPACE

JOINT	LOAD	FORCE-X	FORCE-Y	FORCE-Z	MOM-X	MOM-Y	MOM Z
1	1	0.00	4.00	0.00	0.00	0.00	0.00
2	1	0.00	11.00	0.00	0.00	0.00	0.00
3	1	0.00	11.00	0.00	0.00	0.00	0.00
4	1	0.00	4.00	0.00	0.00	0.00	0.00

EJEMPLO

-- PAGE NO. 6

MEMBER END FORCES STRUCTURE TYPE = SPACE

ALL UNITS ARE -- MTON METE

MEMBER	LOAD	JT	AXIAL	SHEAR-Y	SHEAR-Z	TORSION	MOM-Y	MOM-Z
1	1	1	0.00	4.00	0.00	0.00	0.00	0.00
		2	0.00	6.00	0.00	0.00	0.00	-4.99
2	1	2	0.00	5.00	0.00	0.00	0.00	4.99
		3	0.00	5.00	0.00	0.00	0.00	-4.99
3	1	3	0.00	6.00	0.00	0.00	0.00	4.99
		4	0.00	4.00	0.00	0.00	0.00	0.00

\*\*\*\*\* END OF LATEST ANALYSIS RESULT \*\*\*\*\*

27. FINISH

\*\*\*\*\* END OF STAAD-III \*\*\*\*\*

\*\*\*\* DATE= SEP 23,2001 TIME= 23:24:35 \*\*\*\*

\*\*\*\*\*  
 \* FOR QUESTIONS REGARDING THIS VERSION OF PROGRAM \*  
 \* RESEARCH ENGINEERS, Inc at \*  
 \* West Coast: Ph- (714) 974-2500 Fax- (714) 921-2543 \*  
 \* East Coast: Ph- (978) 688-3636 Fax- (978) 685-7230 \*  
 \*\*\*\*\*



Software licensed to Unknown User

Job Title

Client

Job No

Sheet No

Rev

1

Part

Ref

By

Date 18-May-01

Chd

File marco1.stb

Date/Time 18-May-2001 02:11

### Job Information

	Engineer	Checked	Approved
Name:			
Date:	18-May-01		

Structure Type SPACE FRAME

Number of Nodes	5	Highest Node	11
Number of Elements	4	Highest Beam	13

Number of Basic Load Cases	1
Number of Combination Load Cases	0

Included in this printout are data for:

All The Whole Structure

Included in this printout are results for load cases:

Type	L/C	Name
Primary	1	VERTICAL

### Nodes

Node	X (m)	Y (m)	Z (m)
5	4.000	7.000	0.000
7	2.000	3.000	0.000
9	2.000	0.000	0.000
10	9.000	7.000	0.000
11	9.000	0.000	0.000



Software licensed to Unknown User

Job No	Sheet No <b>2</b>	Rev
Part		
Ref		
By	Date 18-May-01	Chd
Client	File marco1.std	Date/Time 18-May-2001 02:11

### Beams

Beam	Node A	Node B	Length (m)	Property	$\beta$ degrees
6	9	7	3.000	1	0
10	7	5	4.472	1	0
11	5	10	5.000	1	0
13	10	11	7.000	1	0

### Section Properties

Prop	Section	Area (m <sup>2</sup> )	$I_{yy}$ (m <sup>4</sup> )	$I_{zz}$ (m <sup>4</sup> )	J (m <sup>4</sup> )	Material
1	Rect 0.80X0.40	0.320	0.004	0.017	0.012	-

### Materials

Mat	Name	E (kN/mm <sup>2</sup> )	G (kN/mm <sup>2</sup> )	$\nu$	Density (kg/m <sup>3</sup> )	$\alpha$ (1/K)
1	Steel	205.000	82.000	0.250	77.000	12E -12
2	Concrete	25.000	10.684	0.170	24.000	12E -12
3	Aluminum	70.000	26.316	0.330	26.600	23E -12

### Supports

Node	X (kN/mm)	Y (kN/mm)	Z (kN/mm)	rX (kN/rad)	rY (kN/rad)	rZ (kN/rad)
9	Fixed	Fixed	Fixed	-	-	-
11	Fixed	Fixed	Fixed	-	-	-

### Releases

There is no data of this type.

### Basic Load Cases

Number	Name
1	VERTICAL

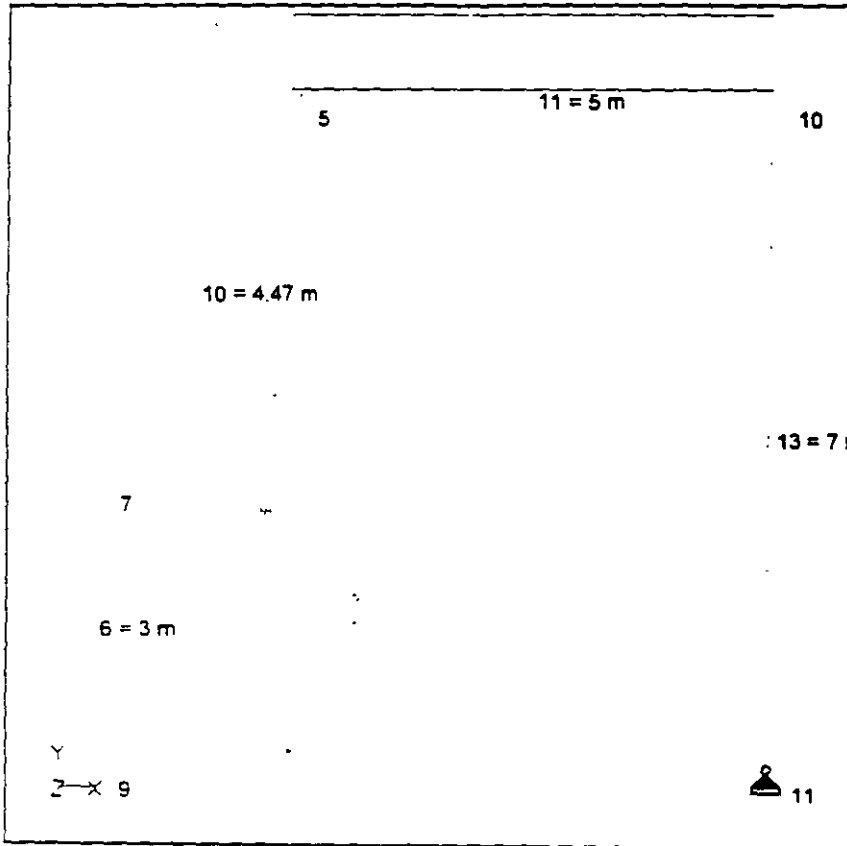


Software licensed to Unknown User

Job No	Sheet No	Rev
	3	
Part		
Ref		
By	Date 18-May-01	Chd
File marco1.std	Date/Time 18-May-2001 02:11	

## Combination Load Cases

There is no data of this type.

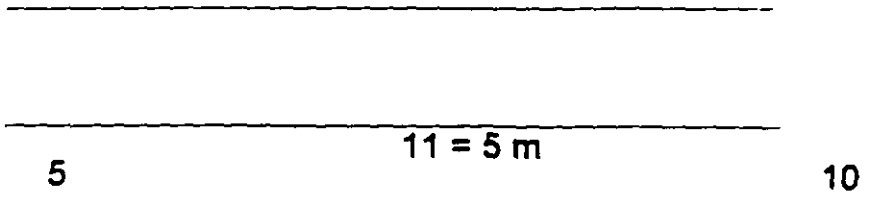


Whole Structure Loads 5kN:1m 1 VERTICAL



Software licensed to Unknown User

Job No	Sheet NO 4	Rev
Part		
Ref		
By	Date 18-May-01	Chg
Client	File marco1.std	Date/Time 18-May-2001 02:11



10 = 4.47 m



Whole Structure Loads 5kN:1m 1 VERTICAL

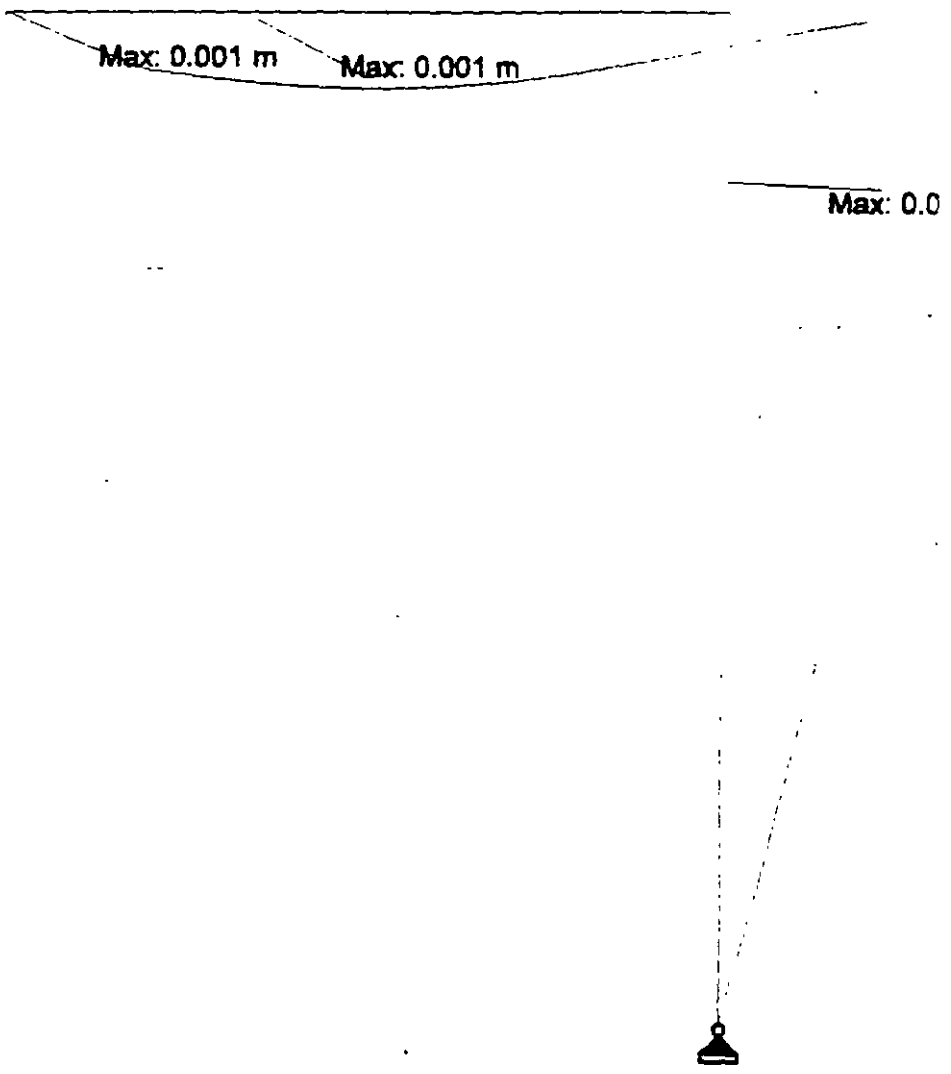


Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date 18-May-01	Cno
File marco1.std	Date/Time 18-May-2001 02:11	

Job Title

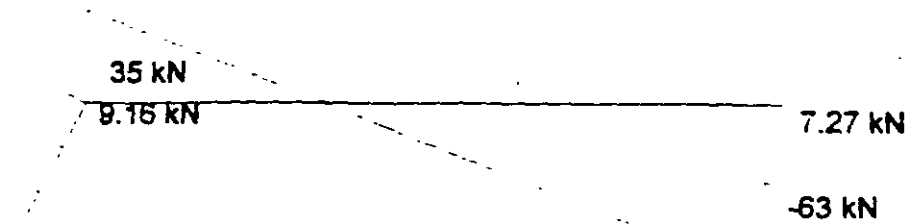
Client







Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Cr'd
	18-May-01	
Client	File	Date/Time
	marco1.std	18-May-2001 02



9.15 kN

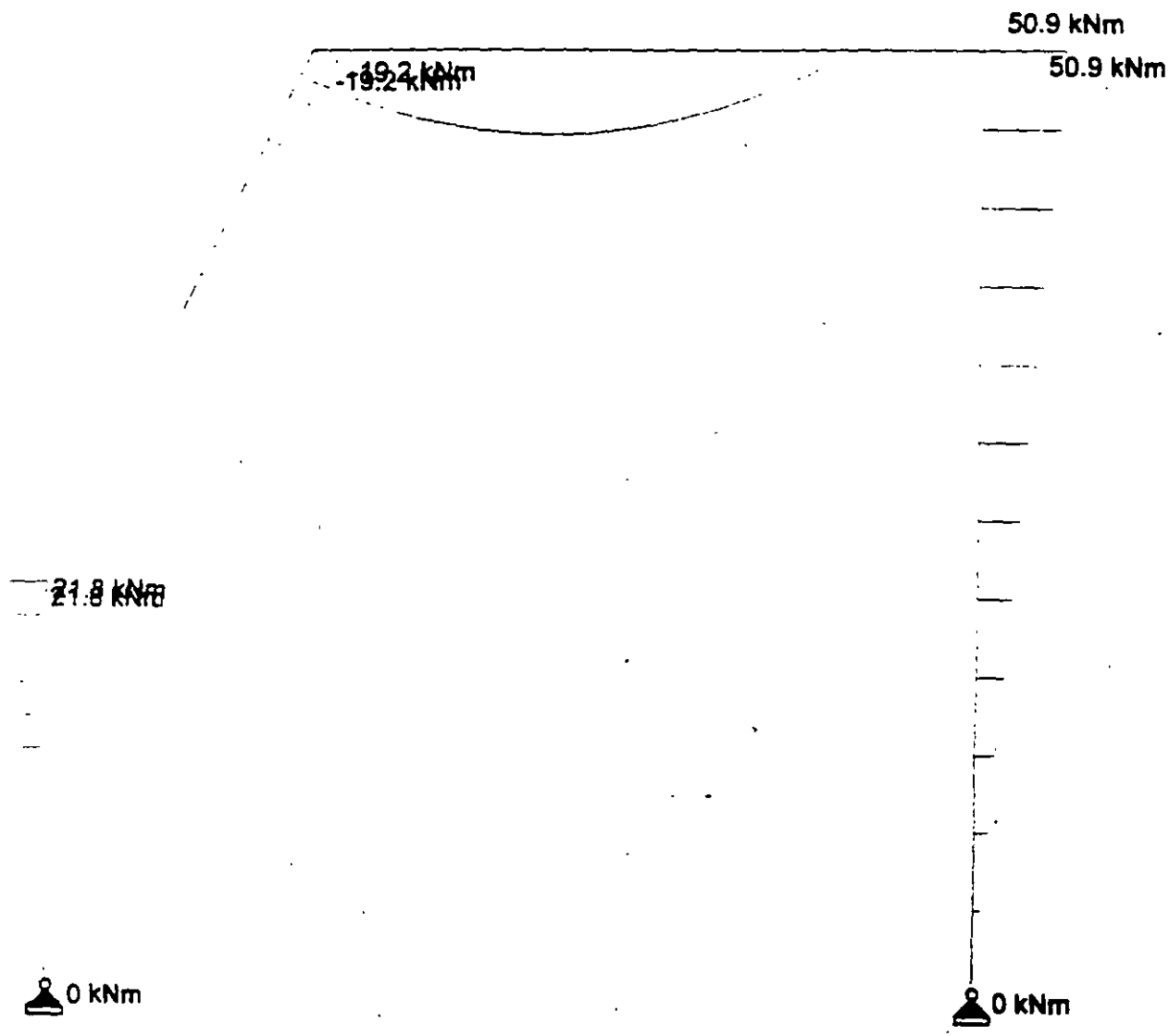
 -7.27 kN

 7.27 kN



Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	18-May-01	
Client	File	Date/Time
	marco1.std	18-May-2001 02:11







Software licensed to Unknown User

Job Title

Client

Job No

Sheet No

Rev

1

Part

Ref

By

Date 18-May-01

CHC

File

marco1.std

Date/Time

18-May-2001 02:11

-19.6 kN/m

N9

X = 7.269 kN  
Y = 35.024 kN  
Z = 0.000 kN  
MX = FREE  
MY = FREE  
MZ = FREE

N11

X = -7.269 kN  
Y = 63.043 kN  
Z = 0.000 kN  
MX = FREE  
MY = FREE  
MZ = FREE

```
STAAD PLANE VIGA EJEMPLO 2
START JOB INFORMATION
ENGINEER DATE 18-May-01
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER MTON
JOINT COORDINATES
5 4 7 0; 7 2 3 0; 9 2 0 0; 10 9 7 0; 11 9 0 0;
MEMBER INCIDENCES
6 9 7; 10 7 5; 11 5 10; 13 10 11;
MEMBER PROPERTY AMERICAN
6 10 11 13 PRIS YD 0.8 ZD 0.4
SUPPORTS
9 11 PINNED
UNIT METER KN
CONSTANTS
E 2.5e-007 MEMB 6 10 11 13
POISSON 0.17 MEMB 6 10 11 13
DENSITY 24 MEMB 6 10 11 13
ALPHA 1.1e-005 MEMB 6 10 11 13
UNIT METER MTON
LOAD 1 VERTICAL
MEMBER LOAD
11 UNI GY -2
PERFORM ANALYSIS PRINT ALL
PRINT SUPPORT REACTION ALL
PRINT JOINT DISPLACEMENTS ALL
PRINT MEMBER FORCES ALL
FINISH
```

```

*****
*
*          STAAD/Pro STAAD-III
*          Revision 3.1
*          Proprietary Program of
*          RESEARCH ENGINEERS, Inc.
*          Date=   SEP 24, 2001
*          Time=   0:35:15
*
*          USER ID: Unknown User
*****

```

1. STAAD PLANE VIGA EJEMPLO 2
2. START JOB INFORMATION
3. ENGINEER DATE 18-MAY-01
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER MTON
7. JOINT COORDINATES
8. 5 4 7 0; 7 2 3 0; 9 2 0 0; 10 9 7 0; 11 9 0 0
9. MEMBER INCIDENCES
10. 6 9 7; 10.7 5; 11 5 10; 13 10 11
11. MEMBER PROPERTY AMERICAN
12. 6 10 11 13 PRIS YD 0.8 ZD 0.4
13. SUPPORTS
14. 9 11 PINNED
15. UNIT METER KN
16. CONSTANTS
17. E 2.5E+007 MEMB 6 10 11 13
18. POISSON 0.17 MEMB 6 10 11 13
19. DENSITY 24 MEMB 6 10 11 13
20. ALPHA 1.1E-005 MEMB 6 10 11 13
21. UNIT METER MTON
22. LOAD 1 VERTICAL
23. MEMBER LOAD
24. 11 UNI GY -2
25. PERFORM ANALYSIS PRINT ALL

P R O B L E M   S T A T I S T I C S

```

-----
NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =      5/   4/   2
ORIGINAL/FINAL BAND-WIDTH =      3/   1
TOTAL PRIMARY LOAD CASES =      1, TOTAL DEGREES OF FREEDOM =      11
SIZE OF STIFFNESS MATRIX =      66 DOUBLE PREC. WORDS
REQRD/AVAIL. DISK SPACE = 12.01/ 2047.7 MB,  EXMEM = 1804.5 MB

```

VIGA EJEMPLO 2

-- PAGE NO. 2

LOADING 1 VERTICAL

MEMBER LOAD - UNIT MTON METE

MEMBER	UDL	L1	L2	CON	L	LIN1	LIN2
--------	-----	----	----	-----	---	------	------

11 -2.000 GY 0.00 5.00

\*\*\*TOTAL APPLIED LOAD ( MTON METE ) SUMMARY (LOADING 1 )

SUMMATION FORCE-X = 0.00  
SUMMATION FORCE-Y = -10.00  
SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-

MX= 0.00 MY= 0.00 MZ= -65.00

-- Processing Element Stiffness Matrix. 0:35:15  
-- Processing Global Stiffness Matrix. 0:35:15  
-- Processing Triangular Factorization. 0:35:15  
-- Calculating Joint Displacements. 0:35:15  
-- Calculating Member Forces. 0:35:15

\*\*\*TOTAL REACTION ( MTON METE ) SUMMARY

LOADING 1

SUM-X= 0.00 SUM-Y= 10.00 SUM-Z= 0.00

SUMMATION OF MOMENTS AROUND ORIGIN-

MX= 0.00 MY= 0.00 MZ= 65.00

EXTERNAL AND INTERNAL JOINT LOAD SUMMARY-

JT	EXT FX/	EXT FY/	EXT FZ/	EXT MX/	EXT MY/	EXT MZ/
	INT FX	INT FY	INT FZ	INT MX	INT MY	INT MZ
5	0.00	-5.00	0.00	0.00	0.00	-4.17
	0.00	5.00	0.00	0.00	0.00	4.17
7	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
9	0.00	0.00	0.00	0.00	0.00	0.00
	-0.74	-3.57	0.00	0.00	0.00	0.00
10	0.00	-5.00	0.00	0.00	0.00	4.17
	0.00	5.00	0.00	0.00	0.00	-4.17
11	0.00	0.00	0.00	0.00	0.00	0.00
	0.74	-6.43	0.00	0.00	0.00	0.00

\*\*\*\*\* END OF DATA FROM INTERNAL STORAGE \*\*\*\*\*

VIGA EJEMPLO 2

-- PAGE NO. 3

26. PRINT SUPPORT REACTION ALL

VIGA EJEMPLO 2

-- PAGE NO. 4

SUPPORT REACTIONS -UNIT MTON METE STRUCTURE TYPE = PLANE

JOINT	LOAD	FORCE-X	FORCE-Y	FORCE-Z	MOM-X	MOM-Y	MOM Z
9	1	0.74	3.57	0.00	0.00	0.00	0.00
11	1	-0.74	6.43	0.00	0.00	0.00	0.00

\*\*\*\*\* END OF LATEST ANALYSIS RESULT \*\*\*\*\*

27. PRINT JOINT DISPLACEMENTS ALL

VIGA EJEMPLO 2

-- PAGE NO. 5

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = PLANE

JOINT	LOAD	X-TRANS	Y-TRANS	Z-TRANS	X-ROTAN	Y-ROTAN	Z-ROTAN
5	1	0.0973	-0.0395	0.0000	0.0000	0.0000	-0.0002
7	1	0.0252	-0.0013	0.0000	0.0000	0.0000	-0.0001
9	1	0.0000	0.0000	0.0000	0.0000	0.0000	-0.0001
10	1	0.0968	-0.0055	0.0000	0.0000	0.0000	0.0001
11	1	0.0000	0.0000	0.0000	0.0000	0.0000	-0.0003

\*\*\*\*\* END OF LATEST ANALYSIS RESULT \*\*\*\*\*

28. PRINT MEMBER FORCES ALL

VIGA EJEMPLO 2

-- PAGE NO. 6

MEMBER END FORCES STRUCTURE TYPE = PLANE

ALL UNITS ARE -- MTON METE

MEMBER	LOAD	JT	AXIAL	SHEAR-Y	SHEAR-Z	TORSION	MOM-Y	MOM-Z
6	1	9	3.57	-0.74	0.00	0.00	0.00	0.00
		7	-3.57	0.74	0.00	0.00	0.00	-2.22
10	1	7	3.53	0.93	0.00	0.00	0.00	2.22
		5	-3.53	-0.93	0.00	0.00	0.00	1.95
11	1	5	0.74	3.57	0.00	0.00	0.00	-1.95
		10	-0.74	6.43	0.00	0.00	0.00	-5.19
13	1	10	6.43	0.74	0.00	0.00	0.00	5.19
		11	-6.43	-0.74	0.00	0.00	0.00	0.00

\*\*\*\*\* END OF LATEST ANALYSIS RESULT \*\*\*\*\*

29. FINISH

\*\*\*\*\* END OF STAAD-III \*\*\*\*\*

\*\*\*\* DATE= SEP 24,2001 TIME= 0:35:15 \*\*\*\*

\*\*\*\*\*  
\* FOR QUESTIONS REGARDING THIS VERSION OF PROGRAM \*  
\* RESEARCH ENGINEERS, Inc at \*  
\* West Coast: Ph- (714) 974-2500 Fax- (714) 921-2543 \*  
\* East Coast: Ph- (978) 688-3636 Fax- (978) 685-7230 \*  
\*\*\*\*\*

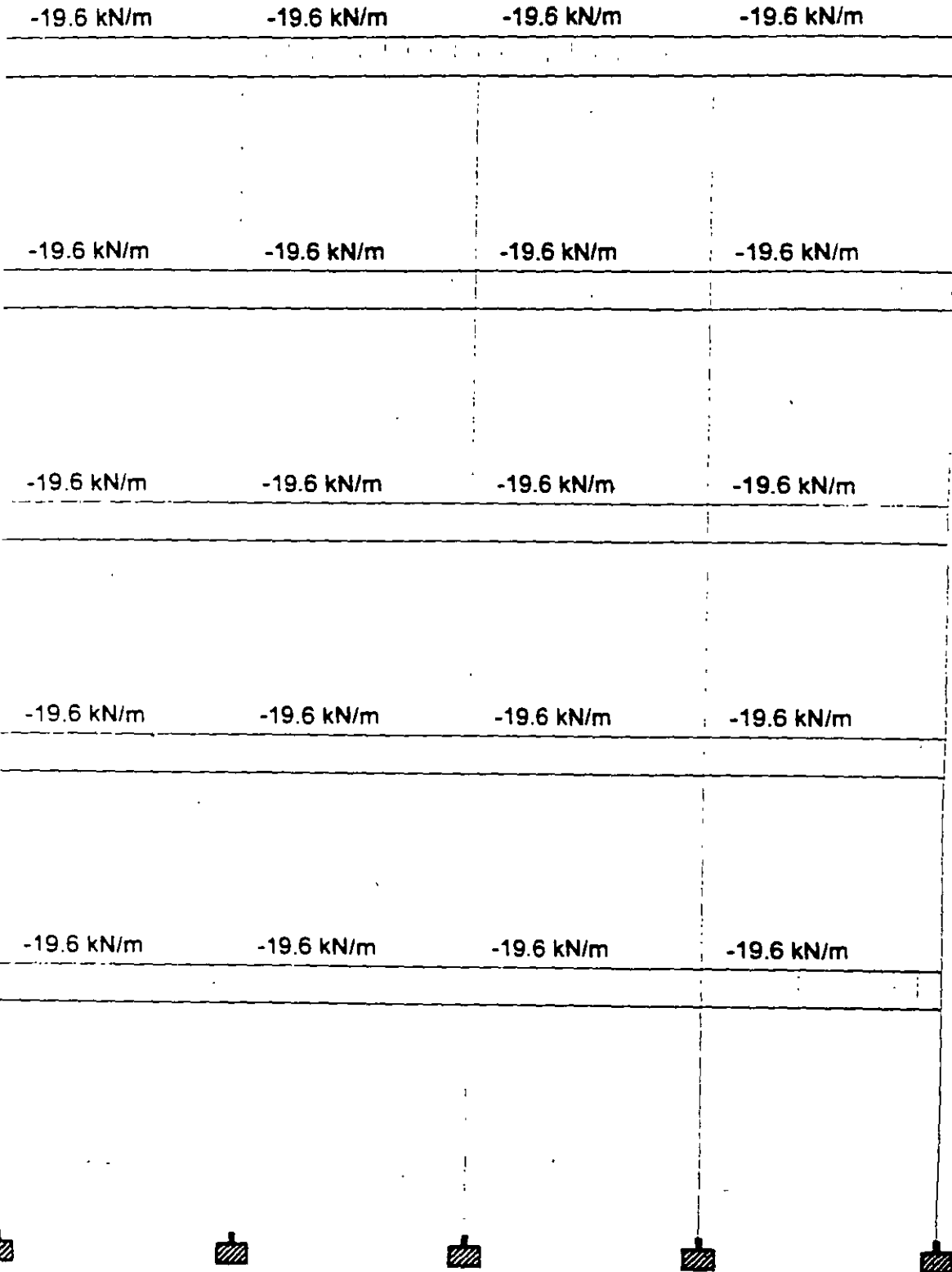


Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	31-May-01	
File	Date/Time	
marco4n.std	31-May-2001 22.39	

Job Title

Client





Software licensed to Unknown User

Job No	Sheet No 1	Rev
Part		
Rev		
By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 31-May-2001 22:39

98.1 kN

78.5 kN

58.8 kN

39.2 kN

19.6 kN





Software licensed to Unknown User

Job No	Sheet No 1	Rev
Part	Ref	
By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55

98.1 kN    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m

78.5 kN    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m

58.8 kN    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m

39.2 kN    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m

19.6 kN    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m    -19.6 kN/m







Software licensed to Unknown User

Job No

Sheet No

1

Rev

Part

Job Title

Ref

By

Desar 31-May-01

Chd

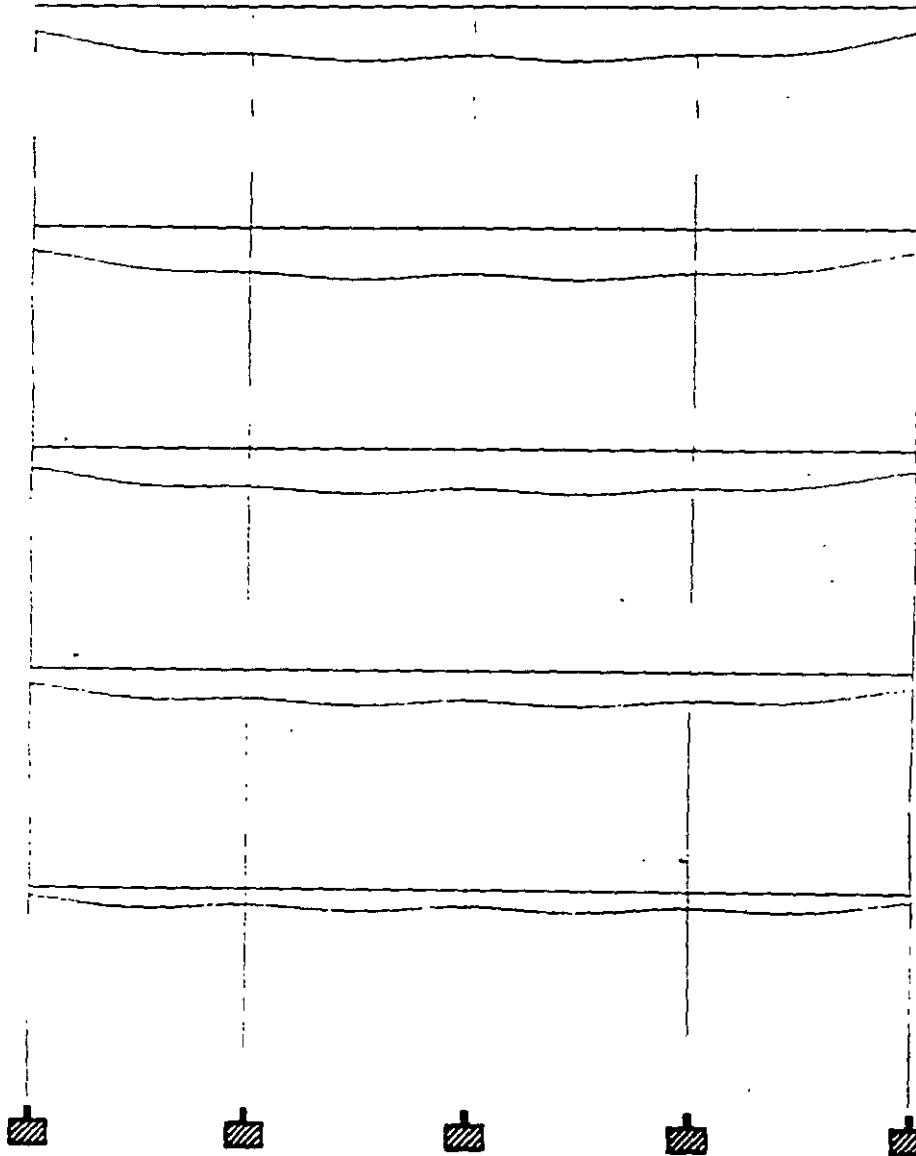
Client

File

marco4n.std

Date/Time

24-Sep-2001 00:55





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	

Part

Ref

By Date 31-May-01 Cnd

File marco4n.std Date/Time 24-Sep-2001 00:55

Job Title

Case:

Max: 0.021 m Max: 0.020 m Max: 0.020 m Max: 0.020 m Max: 0.020 m

Max: 0.018 m Max: 0.018 m Max: 0.018 m Max: 0.018 m Max: 0.018 m

Max: 0.015 m Max: 0.014 m Max: 0.014 m Max: 0.014 m Max: 0.014 m

Max: 0.010 m Max: 0.010 m Max: 0.010 m Max: 0.010 m Max: 0.010 m

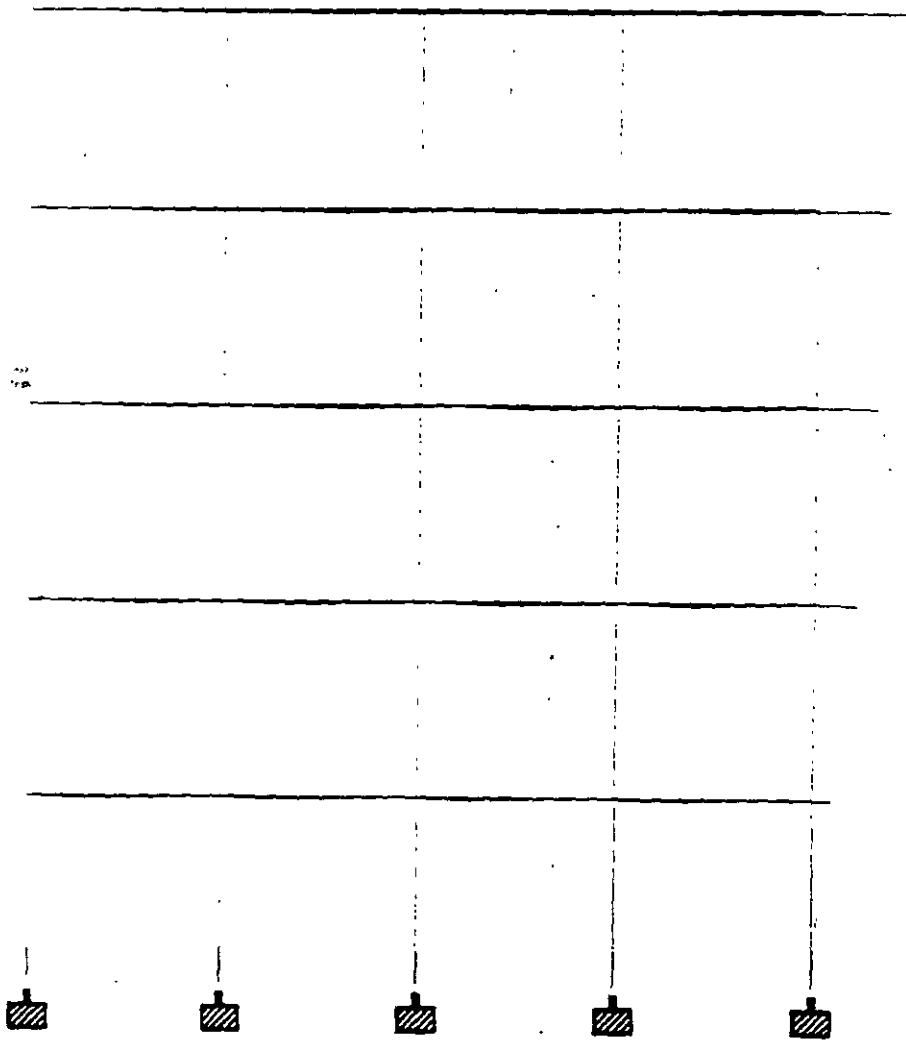
Max: 0.008 m Max: 0.004 m Max: 0.004 m Max: 0.004 m Max: 0.004 m





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	31-May-01	
Client	File	Date/Time
	marco4n.stl	24-Sep-2001 01:41





Software loaned to Unknown User

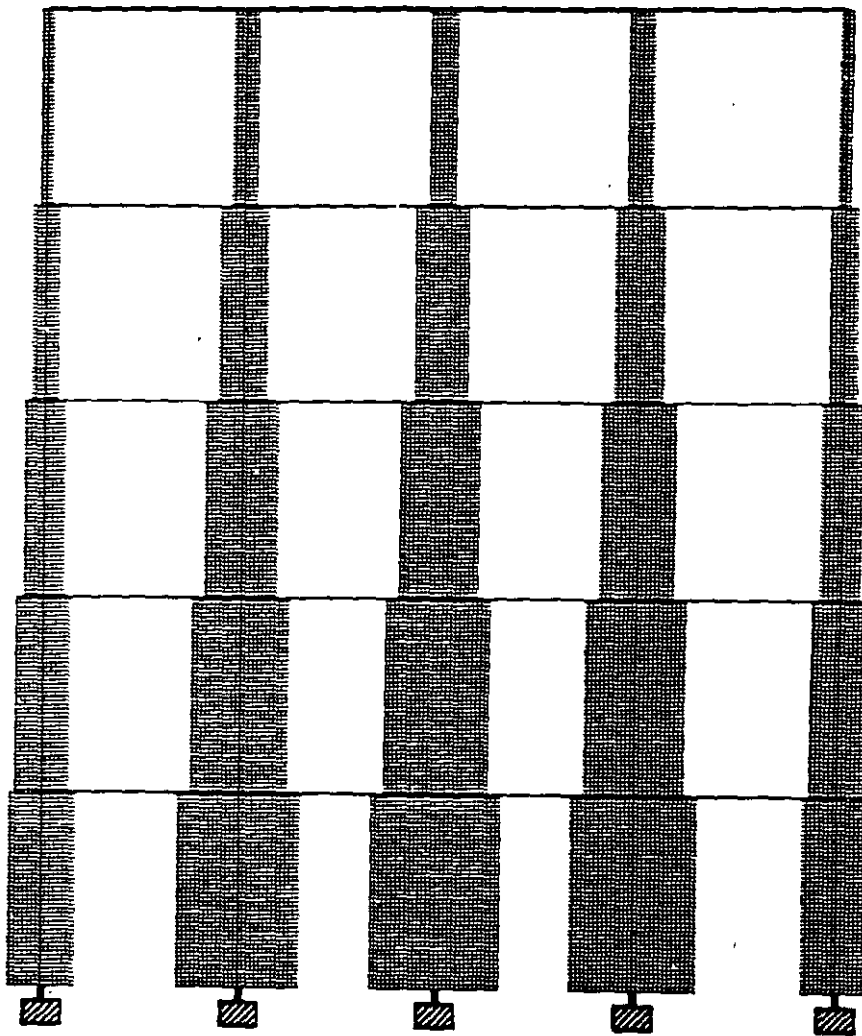
Job No	Sheet No <b>1</b>	Rev
--------	----------------------	-----

Part

Ref

By Date 31-May-01 Ctd

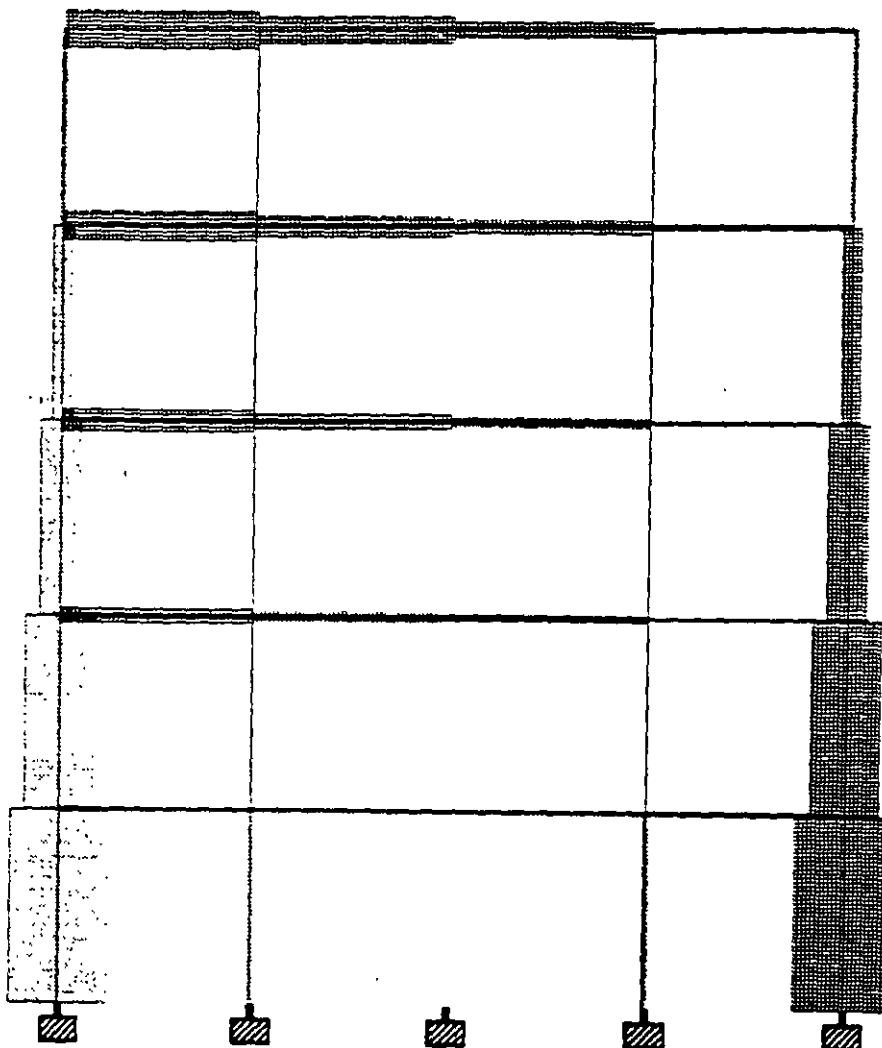
Client File marco4n.std Date/Time 24-Sep-2001 00:55





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Ord
	31-May-01	
Client	File	Date/Time
	marc04n.std	24-Sep-2001 00:55





Software licensed to Unknown User

Job No	Sheet No <b>1</b>	Rev
--------	----------------------	-----

Part:

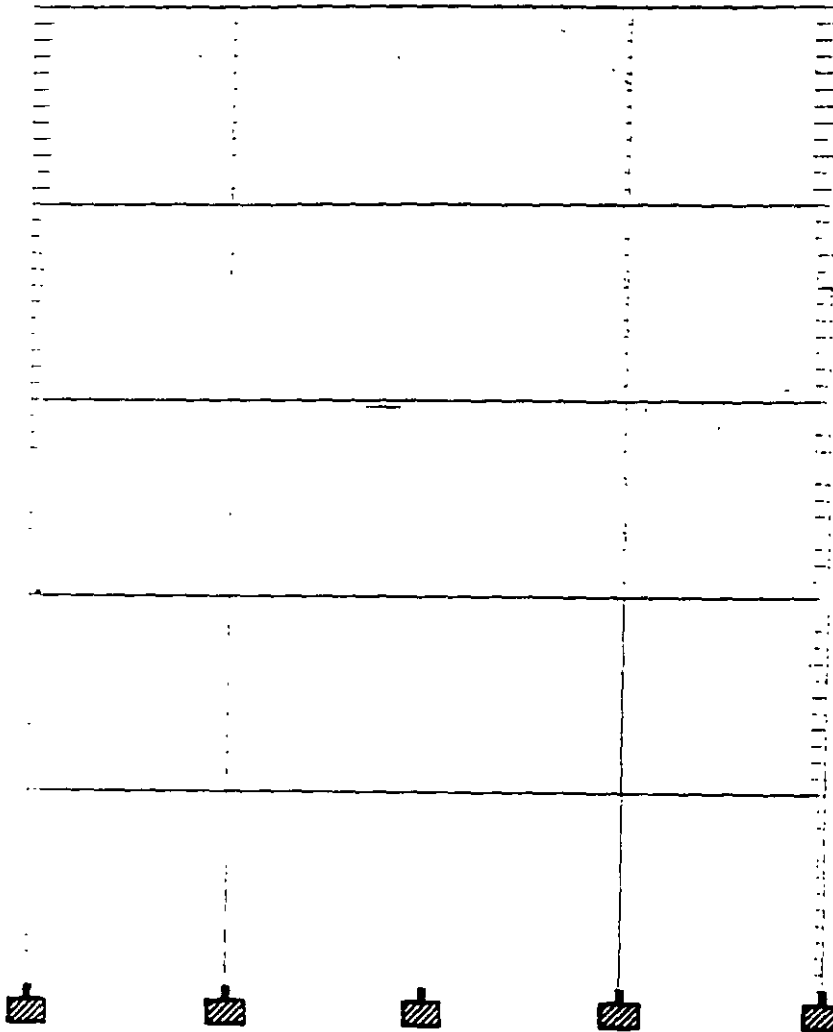
Job Title

Ref:

By: Date: 31-May-01 Cnd:

Client:

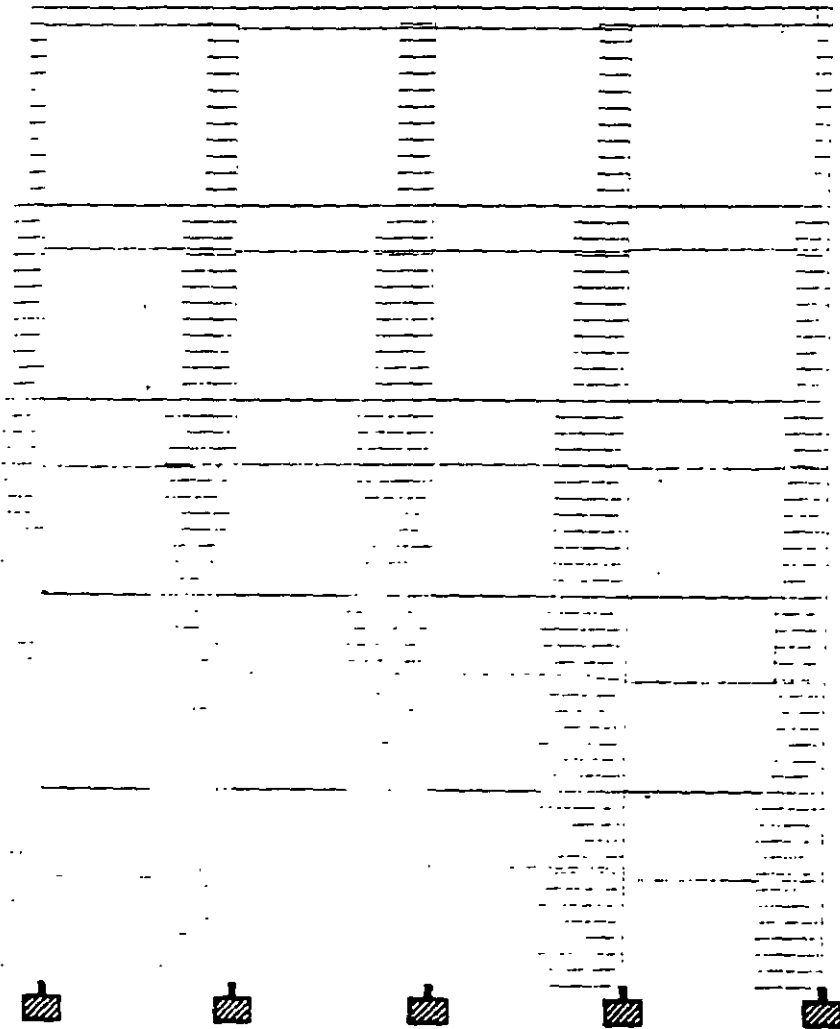
File: marcb4n.std Date/Time: 24-Sep-2001 00:55





Software licensed to Unknow User

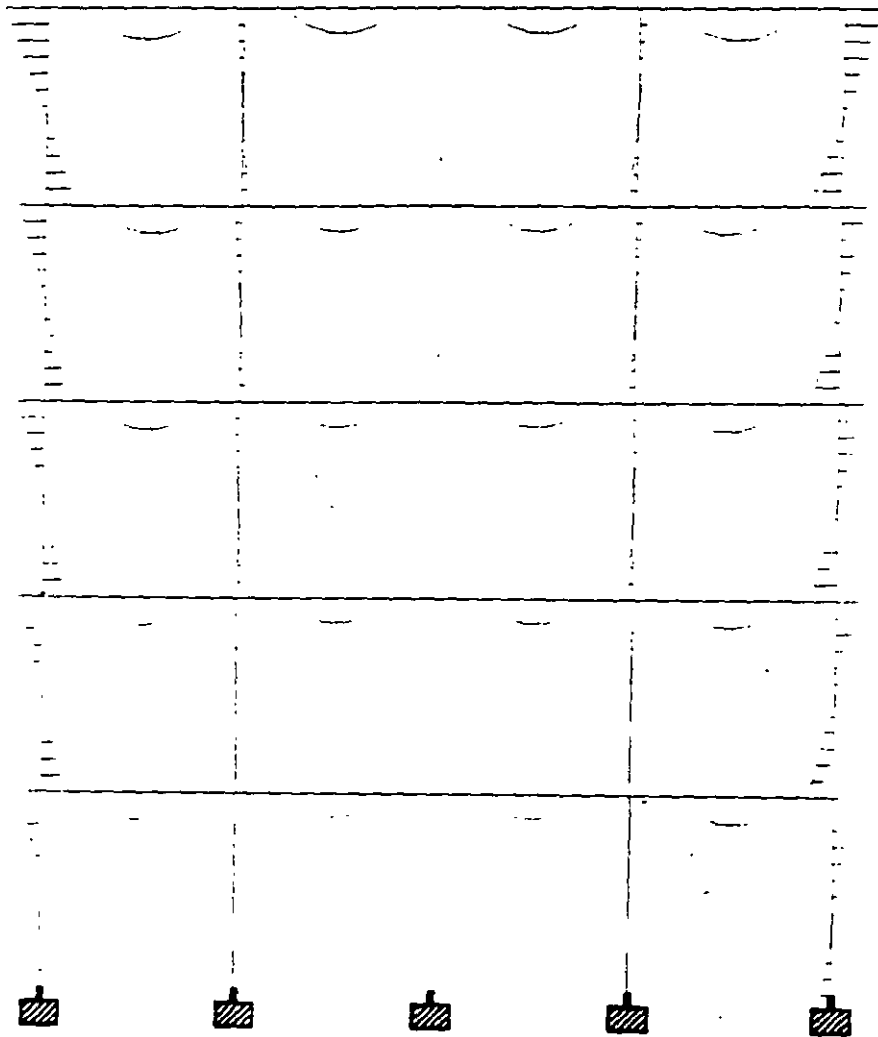
Job No	Sheet No 1	Rev
Part		
Ref		
By	Date 31-May-01	Chg
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55





Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	31-May-01	
Client	File	Date/Time
	marco4n.std	24-Sep-2001 00:55

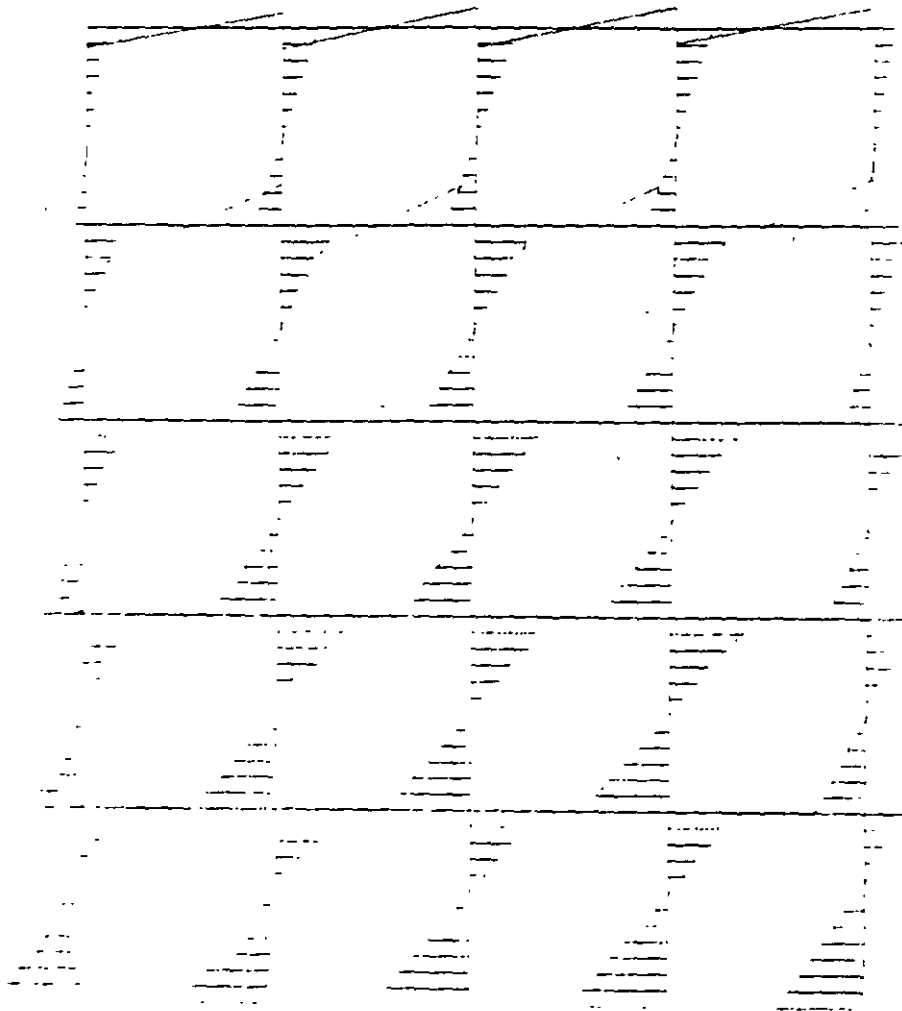






Software licensed to Unknown User

Job No	Sheet No 1	Rev
Part		
Ref		
By	Date 31-May-01	Crd
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55



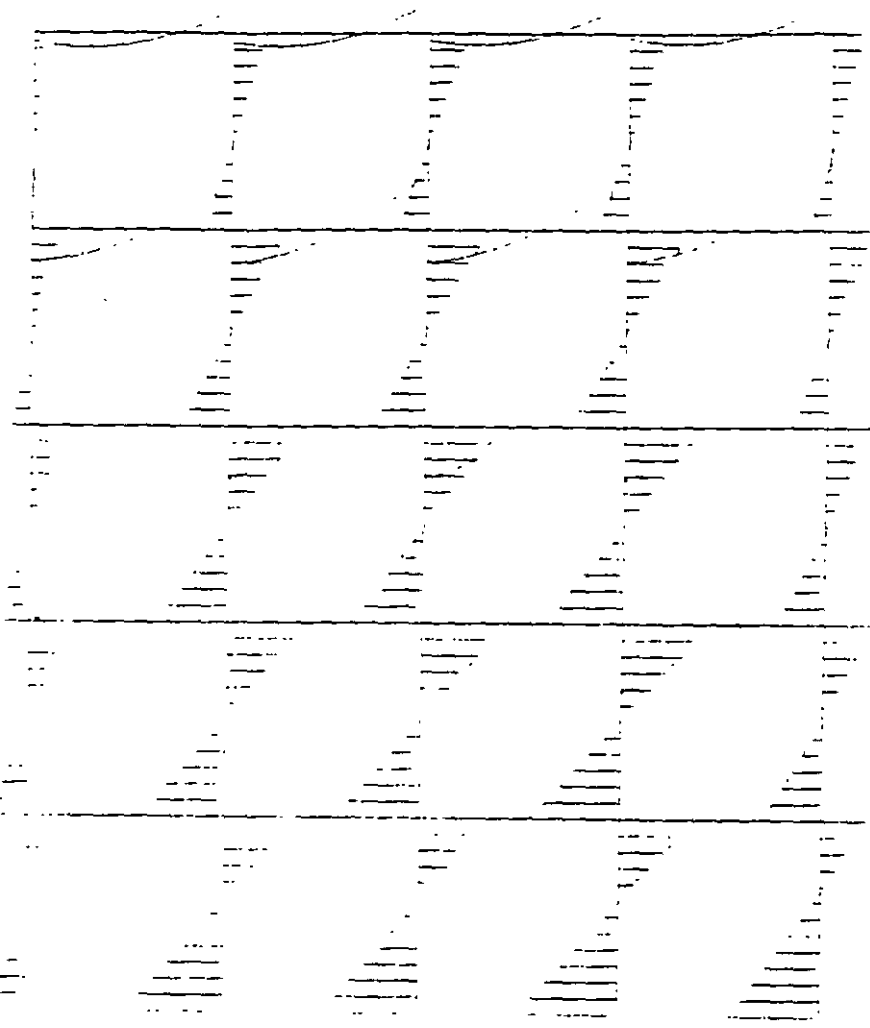


Software licensed to Unknown User

Job No	Sheet No 1	Rev
Part		
Ref		
By	Date 31-May-01	Chg
File marco4n.std	Date/Time 24-Sep-2001 00:55	

Job Title

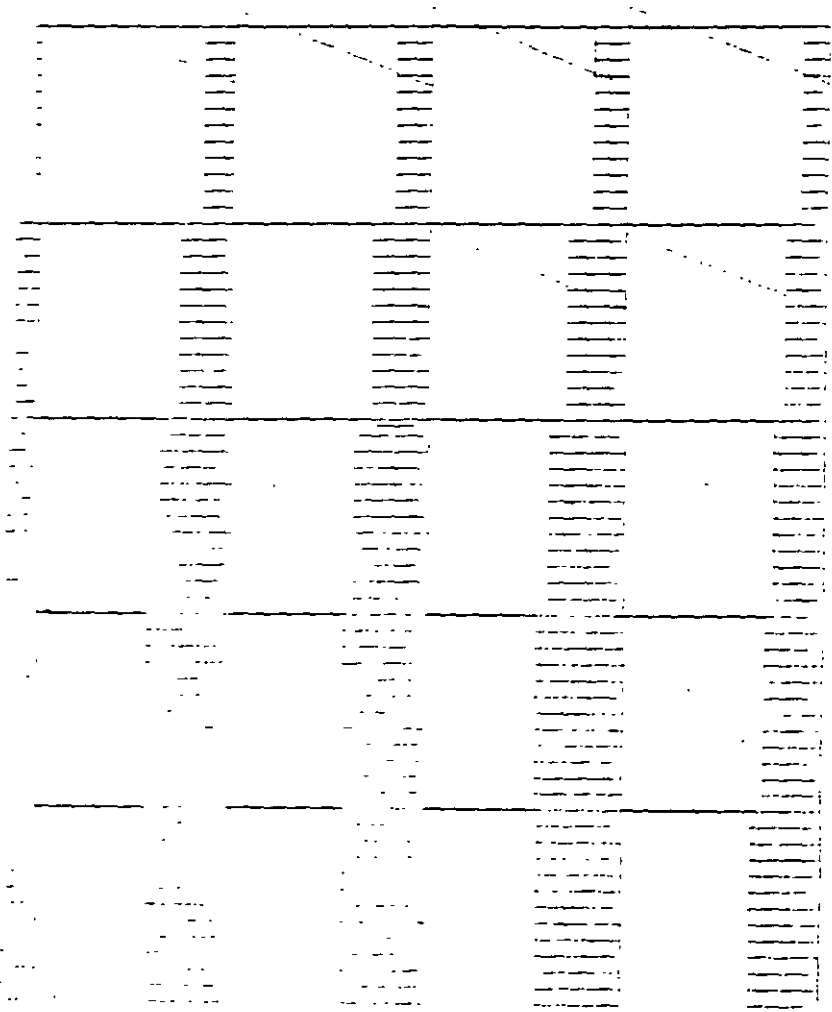
Client





Software licensed to Unknown User

Job No	Sheet No 1	Rev 58
Part		
Ref		
By	Date 31-May-01	Chg
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55



STAAD PLANE VIGA OCHO CLAROS

START JOB INFORMATION

ENGINEER DATE 31-May-01

END JOB INFORMATION

INPUT WIDTH 79

UNIT METER MTON

JOINT COORDINATES

1 0 3 0; 2 3 3 0; 3 6 3 0; 4 9 3 0; 5 12 3 0; 6 0 6 0; 7 3 6 0; 8 6 6 0;  
9 9 6 0; 10 12 6 0; 11 0 9 0; 12 3 9 0; 13 6 9 0; 14 9 9 0; 15 12 9 0;  
16 0 12 0; 17 3 12 0; 18 6 12 0; 19 9 12 0; 20 12 12 0; 21 0 15 0; 22 3 15 0;  
23 6 15 0; 24 9 15 0; 25 12 15 0; 26 0 0 0; 27 3 0 0; 28 6 0 0; 29 9 0 0;  
30 12 0 0;

MEMBER INCIDENCES

1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 6 7; 6 7 8; 7 8 9; 8 9 10; 9 11 12; 10 12 13;  
11 13 14; 12 14 15; 13 16 17; 14 17 18; 15 18 19; 16 19 20; 17 21 22; 18 22 23;  
19 23 24; 20 24 25; 21 26 1; 22 27 2; 23 28 3; 24 29 4; 25 30 5; 26 1 6;  
27 2 7; 28 3 8; 29 4 9; 30 5 10; 31 6 11; 32 7 12; 33 8 13; 34 9 14; 35 10 15;  
36 11 16; 37 12 17; 38 13 18; 39 14 19; 40 15 20; 41 16 21; 42 17 22; 43 18 23;  
44 19 24; 45 20 25;

MEMBER PROPERTY AMERICAN

1 TO 45 PRIS YD 0.4 ZD 0.4

SUPPORTS

26 TO 30 FIXED

UNIT METER KN

CONSTANTS

E 2.5e+007 MEMB 1 TO 45

POISSON 0.17 MEMB 1 TO 45

DENSITY 24 MEMB 1 TO 45

ALPHA 1.2e-011 MEMB 1 TO 45

UNIT METER MTON

LOAD 1 PESO PROPIO

MEMBER LOAD

1 TO 20 UNI GY -2

LOAD 2 Fuerza lateral

JOINT LOAD

1 FX 2

6 FX 4

11 FX 6

16 FX 8

21 FX 10

LOAD COMB 3 Combinación (suma de ambas)

1 1.0 2 1.0

PERFORM ANALYSIS PRINT ALL

PRINT SUPPORT REACTION ALL

FINISH

```

*****
*
*          STAAD/Pro STAAD-III
*          Revision 3.1
*          Proprietary Program of
*          RESEARCH ENGINEERS, Inc.
*          Date=   SEP 24, 2001
*          Time=   1:29:31
*
*          USER ID: Unknown User
*****

```

1. STAAD PLANE VIGA OCHO CLAROS
2. START JOB INFORMATION
3. ENGINEER DATE 31-MAY-01
4. END JOB INFORMATION
5. INPUT WIDTH 79
6. UNIT METER MTON
7. JOINT COORDINATES
8. 1 0 3 0; 2 3 3 0; 3 6 3 0; 4 9 3 0; 5 12 3 0; 6 0 6 0; 7 3 6 0; 8 6 6 0
9. 9 9 6 0; 10 12 6 0; 11 0 9 0; 12 3 9 0; 13 6 9 0; 14 9 9 0; 15 12 9 0
10. 16 0 12 0; 17 3 12 0; 18 6 12 0; 19 9 12 0; 20 12 12 0; 21 0 15 0; 22 3 15 0
11. 23 6 15 0; 24 9 15 0; 25 12 15 0; 26 0 0 0; 27 3 0 0; 28 6 0 0; 29 9 0 0
12. 30 12 0 0
13. MEMBER INCIDENCES
14. 1 1 2; 2 2 3; 3 3 4; 4 4 5; 5 6 7; 6 7 8; 7 8 9; 8 9 10; 9 11 12; 10 12 13
15. 11 13 14; 12 14 15; 13 16 17; 14 17 18; 15 18 19; 16 19 20; 17 21 22; 18 22 3
16. 19 23 24; 20 24 25; 21 26 1; 22 27 2; 23 28 3; 24 29 4; 25 30 5; 26 1 6
17. 27 2 7; 28 3 8; 29 4 9; 30 5 10; 31 6 11; 32 7 12; 33 8 13; 34 9 14; 35 10 15
18. 36 11 16; 37 12 17; 38 13 18; 39 14 19; 40 15 20; 41 16 21; 42 17 22; 43 18 3
19. 44 19 24; 45 20 25
20. MEMBER PROPERTY AMERICAN
21. 1 TO 45 PRIS YD 0.4 ZD 0.4
22. SUPPORTS
23. 26 TO 30 FIXED
24. UNIT METER KN
25. CONSTANTS
26. E 2.5E+007 MEMB 1 TO 45
27. POISSON 0.17 MEMB 1 TO 45
28. DENSITY 24 MEMB 1 TO 45
29. ALPHA 1.2E-011 MEMB 1 TO 45
30. UNIT METER MTON
31. LOAD 1 PESO PROPIO
32. MEMBER LOAD
33. 1 TO 20 UNI GY -2
34. LOAD 2 FUERZA LATERAL
35. JOINT LOAD
36. 1 FX 2
37. 6 FX 4
38. 11 FX 6
39. 16 FX 8
40. 21 FX 10
41. LOAD COMB 3 COMBINACIÓN (SUMA DE AMBAS)

42. 1 1.0 2 1.0

43. PERFORM ANALYSIS PRINT ALL

PROBLEM STATISTICS

-----  
NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS = 30/ 45/ 5  
ORIGINAL/FINAL BAND-WIDTH = 25/ 5  
TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 75  
SIZE OF STIFFNESS MATRIX = 1350 DOUBLE PREC. WORDS  
REQRD/AVAIL. DISK SPACE = 12.07/ 2047.7 MB, EXMEM = 1779.0 MB

VIGA OCHO CLAROS

-- PAGE NO. 3

LOADING 1 PESO PROPIO

-----  
MEMBER LOAD - UNIT MTON METE

MEMBER	UDL	L1	L2	CON	L	LIN1	LIN2
1	-2.000 GY	0.00	3.00				
2	-2.000 GY	0.00	3.00				
3	-2.000 GY	0.00	3.00				
4	-2.000 GY	0.00	3.00				
5	-2.000 GY	0.00	3.00				
6	-2.000 GY	0.00	3.00				
7	-2.000 GY	0.00	3.00				
8	-2.000 GY	0.00	3.00				
9	-2.000 GY	0.00	3.00				
10	-2.000 GY	0.00	3.00				
11	-2.000 GY	0.00	3.00				
12	-2.000 GY	0.00	3.00				
13	-2.000 GY	0.00	3.00				
14	-2.000 GY	0.00	3.00				
15	-2.000 GY	0.00	3.00				
16	-2.000 GY	0.00	3.00				
17	-2.000 GY	0.00	3.00				
18	-2.000 GY	0.00	3.00				
19	-2.000 GY	0.00	3.00				
20	-2.000 GY	0.00	3.00				

\*\*\*TOTAL APPLIED LOAD ( MTON METE ) SUMMARY (LOADING 1 )  
SUMMATION FORCE-X = 0.00  
SUMMATION FORCE-Y = -120.00  
SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-  
MX= 0.00 MY= 0.00 MZ= -720.00

LOADING 2 FUERZA LATERAL

-----  
JOINT LOAD - UNIT MTON METE

JOINT	FORCE-X	FORCE-Y	FORCE-Z	MOM-X	MOM-Y	MOM-Z
-------	---------	---------	---------	-------	-------	-------

1	2.00	0.00	0.00	0.00	0.00	0.00
6	4.00	0.00	0.00	0.00	0.00	0.00
11	6.00	0.00	0.00	0.00	0.00	0.00
16	8.00	0.00	0.00	0.00	0.00	0.00
21	10.00	0.00	0.00	0.00	0.00	0.00

VIGA OCHO CLAROS

-- PAGE NO. 4

\*\*\*TOTAL APPLIED LOAD ( MTON METE ) SUMMARY (LOADING 2 )

SUMMATION FORCE-X = 30.00  
 SUMMATION FORCE-Y = 0.00  
 SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-

MX= 0.00 MY= 0.00 MZ= -330.00

-- Processing Element Stiffness Matrix. 1:29:31  
 -- Processing Global Stiffness Matrix. 1:29:31  
 -- Processing Triangular Factorization. 1:29:31  
 -- Calculating Joint Displacements. 1:29:31  
 -- Calculating Member Forces. 1:29:31

\*\*\*TOTAL REACTION ( MTON METE ) SUMMARY

LOADING 1

SUM-X= 0.00 SUM-Y= 120.00 SUM-Z= 0.00

SUMMATION OF MOMENTS AROUND ORIGIN-

MX= 0.00 MY= 0.00 MZ= 720.00

EXTERNAL AND INTERNAL JOINT LOAD SUMMARY-

JT	EXT FX/ INT FX	EXT FY/ INT FY	EXT FZ/ INT FZ	EXT MX/ INT MX	EXT MY/ INT MY	EXT MZ/ INT MZ
1	0.00	-3.00	0.00	0.00	0.00	-1.50
	0.00	3.00	0.00	0.00	0.00	1.50
2	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
3	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
4	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
5	0.00	-3.00	0.00	0.00	0.00	1.50
	0.00	3.00	0.00	0.00	0.00	-1.50
6	0.00	-3.00	0.00	0.00	0.00	-1.50
	0.00	3.00	0.00	0.00	0.00	1.50
7	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
8	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
9	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
10	0.00	-3.00	0.00	0.00	0.00	1.50
	0.00	3.00	0.00	0.00	0.00	-1.50

11	0.00	-3.00	0.00	0.00	0.00	-1.50
	0.00	3.00	0.00	0.00	0.00	1.50
12	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00

VIGA OCHO CLAROS

13	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
14	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
15	0.00	-3.00	0.00	0.00	0.00	-1.50
	0.00	3.00	0.00	0.00	0.00	-1.50
16	0.00	-3.00	0.00	0.00	0.00	-1.50
	0.00	3.00	0.00	0.00	0.00	1.50
17	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
18	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
19	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
20	0.00	-3.00	0.00	0.00	0.00	1.50
	0.00	3.00	0.00	0.00	0.00	-1.50
21	0.00	-3.00	0.00	0.00	0.00	-1.50
	0.00	3.00	0.00	0.00	0.00	1.50
22	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
23	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
24	0.00	-6.00	0.00	0.00	0.00	0.00
	0.00	6.00	0.00	0.00	0.00	0.00
25	0.00	-3.00	0.00	0.00	0.00	1.50
	0.00	3.00	0.00	0.00	0.00	-1.50
26	0.00	0.00	0.00	0.00	0.00	0.00
	-0.27	-15.62	0.00	0.00	0.00	0.27
27	0.00	0.00	0.00	0.00	0.00	0.00
	-0.01	-29.30	0.00	0.00	0.00	0.02
28	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	-30.17	0.00	0.00	0.00	0.00
29	0.00	0.00	0.00	0.00	0.00	0.00
	0.01	-29.30	0.00	0.00	0.00	-0.02
30	0.00	0.00	0.00	0.00	0.00	0.00
	0.27	-15.62	0.00	0.00	0.00	-0.27

LOADING 2

SUM-X= -30.00 SUM-Y= 0.00 SUM-Z= 0.00

SUMMATION OF MOMENTS AROUND ORIGIN-

MX= 0.00 MY= 0.00 MZ= 330.00

EXTERNAL AND INTERNAL JOINT LOAD SUMMARY-

JT	EXT FX/ INT FX	EXT FY/ INT FY	EXT FZ/ INT FZ	EXT MX/ INT MX	EXT MY/ INT MY	EXT MZ/ INT MZ
----	-------------------	-------------------	-------------------	-------------------	-------------------	-------------------



1	2.00	0.00	0.00	0.00	0.00	0.00
	-2.00	0.00	0.00	0.00	0.00	0.00
2	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00

VIGA OCHO CLAROS

3	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
4	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
5	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
6	4.00	0.00	0.00	0.00	0.00	0.00
	-4.00	0.00	0.00	0.00	0.00	0.00
7	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
8	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
9	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
10	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
11	6.00	0.00	0.00	0.00	0.00	0.00
	-6.00	0.00	0.00	0.00	0.00	0.00
12	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
13	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
14	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
15	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
16	8.00	0.00	0.00	0.00	0.00	0.00
	-8.00	0.00	0.00	0.00	0.00	0.00
17	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
18	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
19	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
20	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
21	10.00	0.00	0.00	0.00	0.00	0.00
	-10.00	0.00	0.00	0.00	0.00	0.00
22	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
23	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
24	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
25	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
26	0.00	0.00	0.00	0.00	0.00	0.00
	5.13	23.71	0.00	0.00	0.00	-10.02
27	0.00	0.00	0.00	0.00	0.00	0.00
	6.64	-1.38	0.00	0.00	0.00	-11.49

28	0.00	0.00	0.00	0.00	0.00	0.00	0.00
	6.52	-0.04	0.00	0.00	0.00	0.00	-11.36
29	0.00	0.00	0.00	0.00	0.00	0.00	0.00
	6.62	1.29	0.00	0.00	0.00	0.00	-11.45
30	0.00	0.00	0.00	0.00	0.00	0.00	0.00
	5.09	-23.58	0.00	0.00	0.00	0.00	-9.95

VIGA OCHO CLAROS

-- PAGE NO. 7

LOAD COMBINATION NO. 3  
 COMBINACION (SUMA DE AMBAS)

LOADING- 1. 2.  
 FACTOR - 1.00 1.00

\*\*\*\*\* END OF DATA FROM INTERNAL STORAGE \*\*\*\*\*

44. PRINT SUPPORT REACTION ALL

VIGA OCHO CLAROS

-- PAGE NO. 8

SUPPORT REACTIONS -UNIT MTON METE STRUCTURE TYPE = PLANE

JOINT	LOAD	FORCE-X	FORCE-Y	FORCE-Z	MOM-X	MOM-Y	MOM Z
26	1	0.27	15.62	0.00	0.00	0.00	-0.27
	2	-5.13	-23.71	0.00	0.00	0.00	10.02
	3	-4.86	-8.09	0.00	0.00	0.00	9.75
27	1	0.01	29.30	0.00	0.00	0.00	-0.02
	2	-6.64	1.38	0.00	0.00	0.00	11.49
	3	-6.63	30.67	0.00	0.00	0.00	11.48
28	1	0.00	30.17	0.00	0.00	0.00	0.00
	2	-6.52	0.04	0.00	0.00	0.00	11.36
	3	-6.52	30.21	0.00	0.00	0.00	11.36
29	1	-0.01	29.30	0.00	0.00	0.00	0.02
	2	-6.62	-1.29	0.00	0.00	0.00	11.45
	3	-6.63	28.01	0.00	0.00	0.00	11.47
30	1	-0.27	15.62	0.00	0.00	0.00	0.27
	2	-5.09	23.58	0.00	0.00	0.00	9.95
	3	-5.36	39.19	0.00	0.00	0.00	10.22

\*\*\*\*\* END OF LATEST ANALYSIS RESULT \*\*\*\*\*

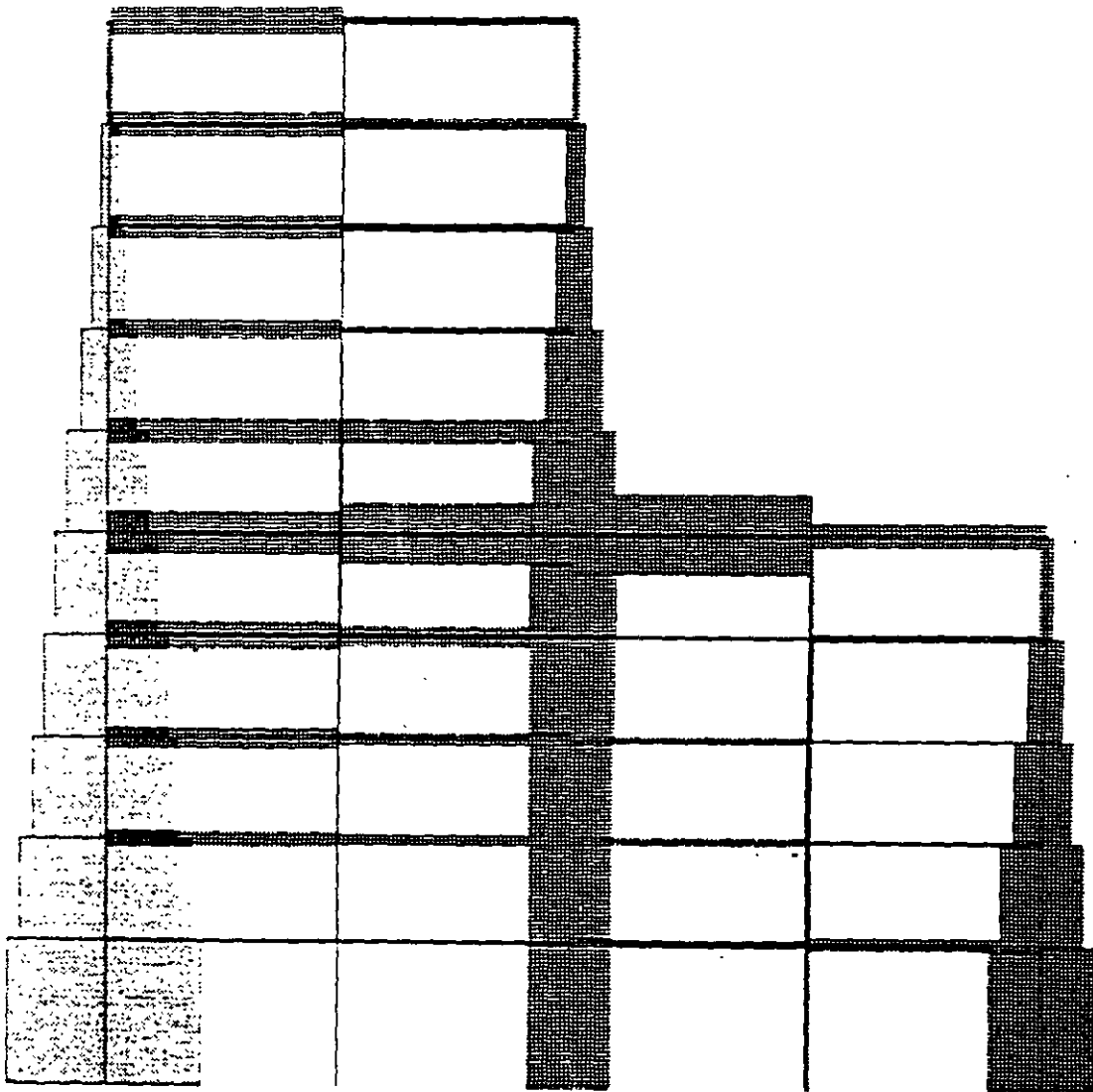
45. FINISH

\*\*\*\*\* END OF STAAD-III \*\*\*\*\*  
 \*\*\*\* DATE= SEP 24,2001 TIME= 1:29:31 \*\*\*\*  
 \*\*\*\*\*  
 \* FOR QUESTIONS REGARDING THIS VERSION OF PROGRAM \*  
 \* RESEARCH ENGINEERS, Inc at \*  
 \* West Coast: Ph- (714) 974-2500 Fax- (714) 921-2543 \*  
 \* East Coast: Ph- (978) 688-3636 Fax- (978) 685-7230 \*  
 \*\*\*\*\*



Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	24-Sep-01	
Client	File	Date/Time
	mar3-2d-10n.std	24-Sep-2001 18:07





Software licensed to Unknown User

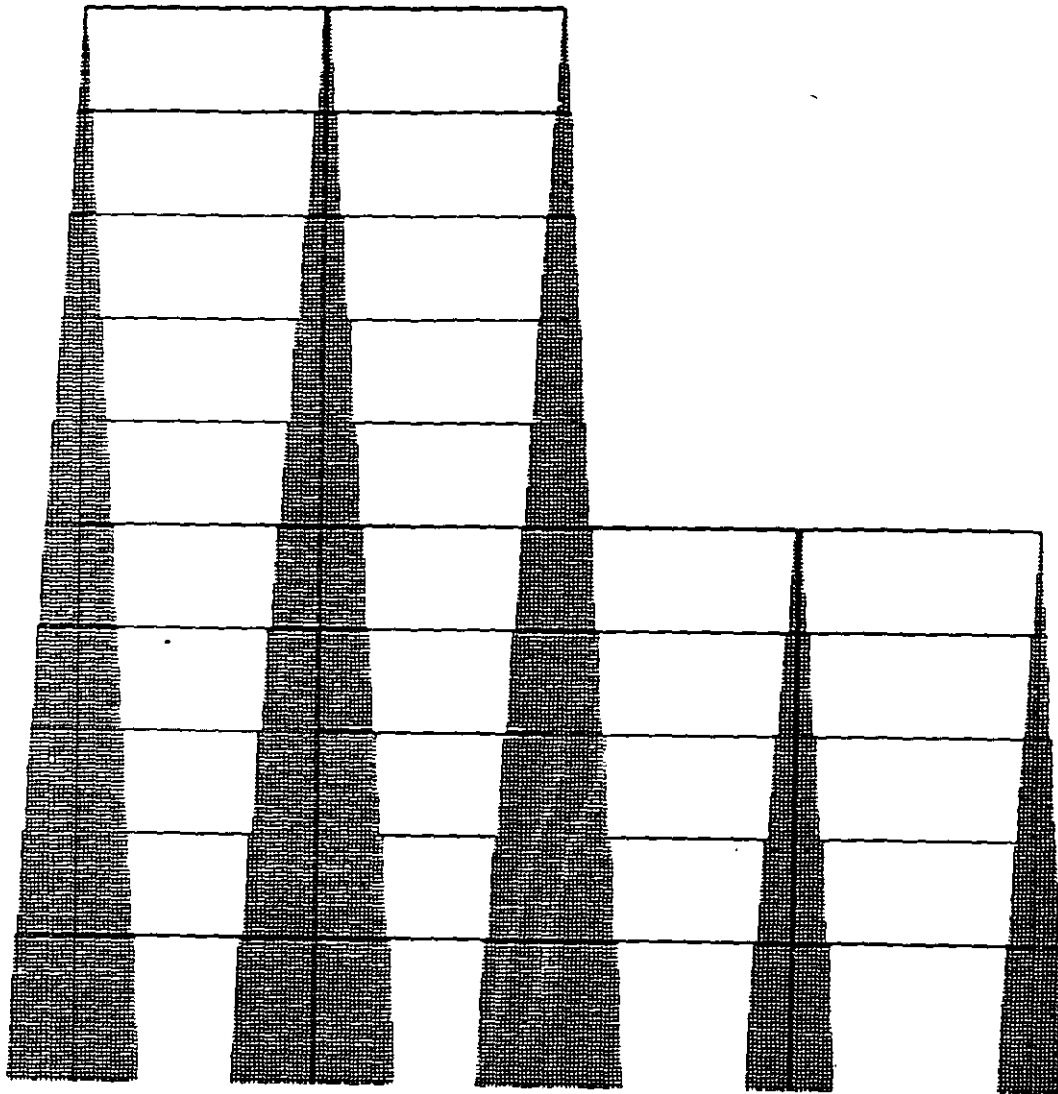
Job No	Sheet No	Rev
	1	

Part

Ref

By Date 24-Sep-01 Cnd

File mar3-2d-10n.std Date/Time 24-Sep-2001 16:07



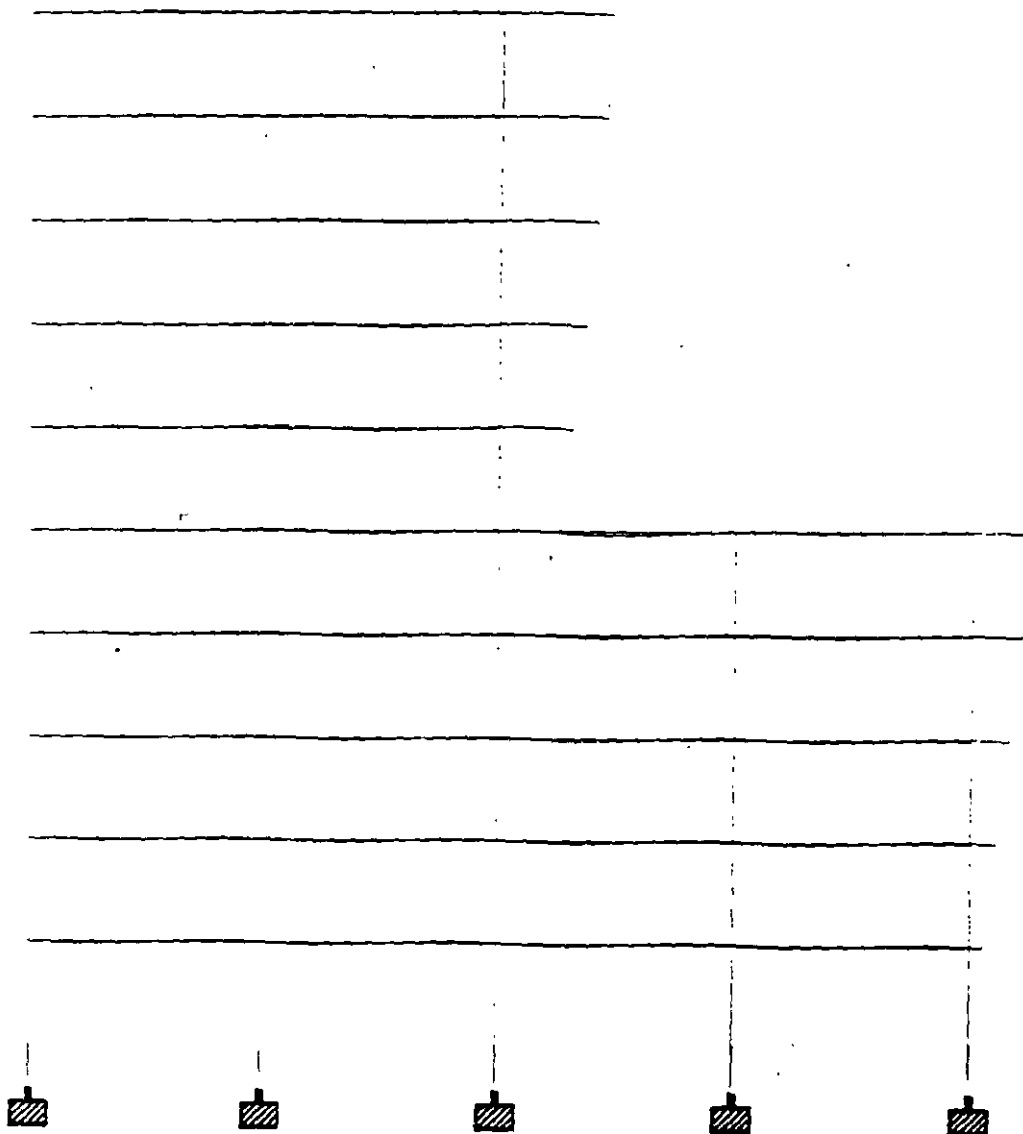


Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Chg
	24-Sep-01	
File	Date/Time	
mar3-2d-10n.std	24-Sep-2001 16:07	

Job Title

Client



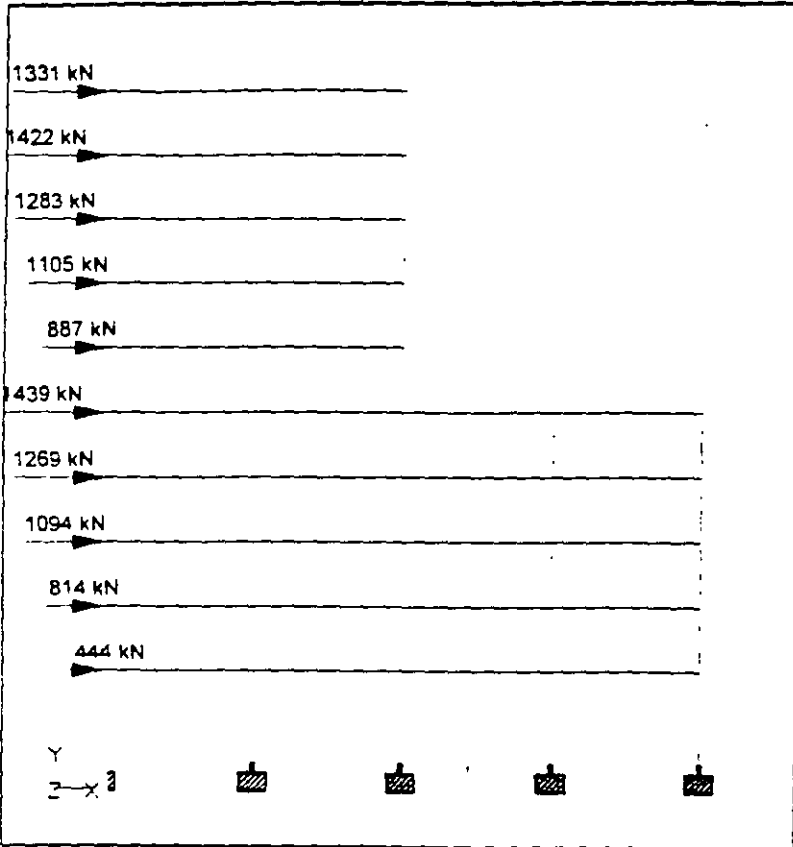


Software licensed to Unknown User

Job No	Sheet No <b>6</b>	Rev
Part		
Ref		
By	Date <b>24-Sep-01</b>	Chg
Client	File <b>mar3-2d-10n.std</b>	Date/Time <b>24-Sep-2001 16:07</b>

Job Title

Client



Whole Structure Loads 262.764kN:1m 2 FUERZAS LATERALES PARA RIGIDECES



Software licensed to Unknown User

Job No	Sheet No <b>5</b>	Rev
Part		
Ref		
By	Date 24-Sep-01	Chd
Client	File mar3-2d-10n.std	Date/Time 24-Sep-2001 18:07

### Section Properties

Prop	Section	Area (m <sup>2</sup> )	I <sub>yy</sub> (m <sup>4</sup> )	I <sub>zz</sub> (m <sup>4</sup> )	J (m <sup>4</sup> )	Material
1	Rect 0.95X0.95	0.902	0.068	0.068	0.115	-
2	Rect 0.95X0.35	0.332	0.003	0.025	0.010	-

### Materials

Mat	Name	E (kN/mm <sup>2</sup> )	G (kN/mm <sup>2</sup> )	v	Density (kg/m <sup>3</sup> )	α (1/°K)
1	Steel	205.000	82.000	0.250	77.000	12E -12
2	Concrete	25.000	10.684	0.170	24.000	12E -12
3	Aluminum	70.000	26.316	0.330	26.600	23E -12

### Supports

Node	X (kN/mm)	Y (kN/mm)	Z (kN/mm)	rX (kN/rad)	rY (kN/rad)	rZ (kN/rad)
1	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed
2	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed
3	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed
4	Fixed	-Fixed	Fixed	Fixed	Fixed	Fixed
5	Fixed	Fixed	Fixed	Fixed	Fixed	Fixed

### Releases

There is no data of this type.

### Basic Load Cases

Number	Name
1	PESO PROPIO
2	FUERZAS LATERALES PARA RIGIDECE:

### Combination Load Cases

There is no data of this type.



Software licensed to Unknown User

Job No	Sheet No <b>4</b>	Rev
Part		
Ref		
By	Date 24-Sep-01	Chd
Client	File mar3-2d-10n.std	Date/Time 24-Sep-2001 16:07

### Beams Cont...

Beam	Node A	Node B	Length (m)	Property	$\beta$ degrees
36	24	25	8.000	2	0
37	21	26	3.500	1	0
38	22	27	3.500	1	0
39	23	28	3.500	1	0
40	24	29	3.500	1	0
41	25	30	3.500	1	0
42	26	27	8.000	2	0
43	27	28	8.000	2	0
44	28	29	8.000	2	0
45	29	30	8.000	2	0
46	26	31	3.500	1	0
47	27	32	3.500	1	0
48	28	33	3.500	1	0
51	31	32	8.000	2	0
52	32	33	8.000	2	0
55	31	36	3.500	1	0
56	32	37	3.500	1	0
57	33	38	3.500	1	0
60	36	37	8.000	2	0
61	37	38	8.000	2	0
64	36	41	3.500	1	0
65	37	42	3.500	1	0
66	38	43	3.500	1	0
69	41	42	8.000	2	0
70	42	43	8.000	2	0
73	41	46	3.500	1	0
74	42	47	3.500	1	0
75	43	48	3.500	1	0
78	46	47	8.000	2	0
79	47	48	8.000	2	0
82	46	51	3.500	1	0
83	47	52	3.500	1	0
84	48	53	3.500	1	0
87	51	52	8.000	2	0
88	52	53	8.000	2	0





Software licensed to Unknown User

Job No	Sheet No <b>3</b>	Rev
Part		
Ref		
By	Date 24-Sep-01	Crd
Client	File mar3-2d-10n.std	Date/Time 24-Sep-2001 18:07

## Beams

Beam	Node A	Node B	Length (m)	Property	$\beta$ degrees
1	1	6	5.000	1	0
2	2	7	5.000	1	0
3	3	8	5.000	1	0
4	4	9	5.000	1	0
5	5	10	5.000	1	0
6	6	7	8.000	2	0
7	7	8	8.000	2	0
8	8	9	8.000	2	0
9	9	10	8.000	2	0
10	6	11	3.500	1	0
11	7	12	3.500	1	0
12	8	13	3.500	1	0
13	9	14	3.500	1	0
14	10	15	3.500	1	0
15	11	12	8.000	2	0
16	12	13	8.000	2	0
17	13	14	8.000	2	0
18	14	15	8.000	2	0
19	11	16	3.500	1	0
20	12	17	3.500	1	0
21	13	18	3.500	1	0
22	14	19	3.500	1	0
23	15	20	3.500	1	0
24	16	17	8.000	2	0
25	17	18	8.000	2	0
26	18	19	8.000	2	0
27	19	20	8.000	2	0
28	16	21	3.500	1	0
29	17	22	3.500	1	0
30	18	23	3.500	1	0
31	19	24	3.500	1	0
32	20	25	3.500	1	0
33	21	22	8.000	2	0
34	22	23	8.000	2	0
35	23	24	8.000	2	0



Software licensed to Unknown User

Job No	Sheet No <b>2</b>	Rev
Part		
Ref		
By	Date 24-Sep-01	Chd
Client	File mas3-2d-10n.std	Date/Time 24-Sep-2001 18:07

### Nodes Cont...

Node	X (m)	Y (m)	Z (m)
13	16.000	8.500	0.000
14	24.000	8.500	0.000
15	32.000	8.500	0.000
16	0.000	12.000	0.000
17	8.000	12.000	0.000
18	16.000	12.000	0.000
19	24.000	12.000	0.000
20	32.000	12.000	0.000
21	0.000	15.500	0.000
22	8.000	15.500	0.000
23	16.000	15.500	0.000
24	24.000	15.500	0.000
25	32.000	15.500	0.000
26	0.000	19.000	0.000
27	8.000	19.000	0.000
28	16.000	19.000	0.000
29	24.000	19.000	0.000
30	32.000	19.000	0.000
31	0.000	22.500	0.000
32	8.000	22.500	0.000
33	16.000	22.500	0.000
36	0.000	26.000	0.000
37	8.000	26.000	0.000
38	16.000	26.000	0.000
41	0.000	29.500	0.000
42	8.000	29.500	0.000
43	16.000	29.500	0.000
46	0.000	33.000	0.000
47	8.000	33.000	0.000
48	16.000	33.000	0.000
51	0.000	36.500	0.000
52	8.000	36.500	0.000
53	16.000	36.500	0.000



Software licensed to Unknown User

Job No	Sheet No 1	Rev
Part		
Ref		
By	Date 24-Sep-01	Chd
Client	File mar3-2d-10n.std	Date/Time 24-Sep-2001 16:07

### Job Information

	Engineer	Checked	Approved
Name:			
Date:	24-Sep-01		

Structure Type	SPACE FRAME
----------------	-------------

Number of Nodes	45	Highest Node	53
Number of Elements	70	Highest Beam	88

Number of Basic Load Cases	2
Number of Combination Load Cases	0

Included in this printout are data for:

All	The Whole Structure
-----	---------------------

Included in this printout are results for load cases:

Type	L/C	Name
Primary	1	PESO PROPIO
Primary	2	FUERZAS LATERALES PARA RIGIDECE:

### Nodes

Node	X (m)	Y (m)	Z (m)
1	0.000	0.000	0.000
2	8.000	0.000	0.000
3	16.000	0.000	0.000
4	24.000	0.000	0.000
5	32.000	0.000	0.000
6	0.000	5.000	0.000
7	8.000	5.000	0.000
8	16.000	5.000	0.000
9	24.000	5.000	0.000
10	32.000	5.000	0.000
11	0.000	8.500	0.000
12	8.000	8.500	0.000

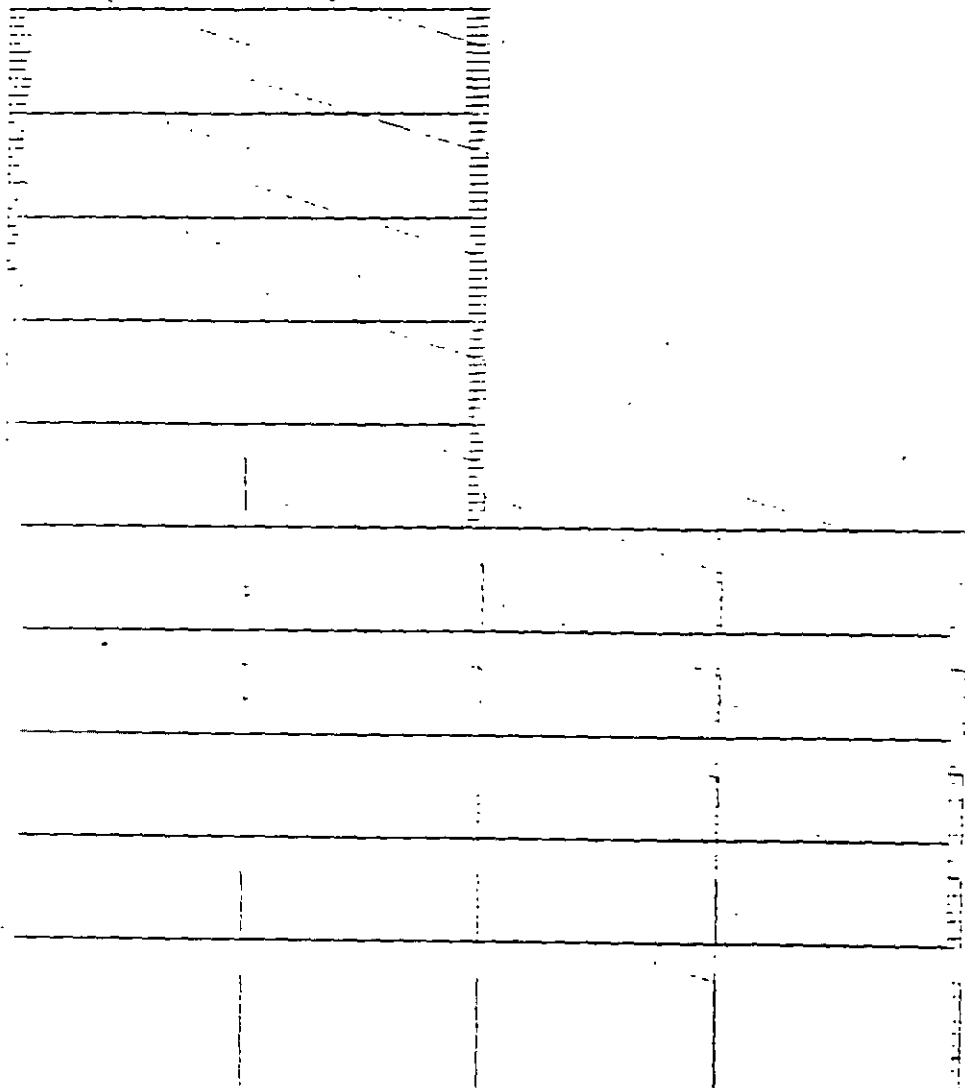


Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Ord
	24-Sep-01	
File	Date/Time	
mar3-2d-10n.std	24-Sep-2001 18:07	

Job Title

Client



JOINT	LOAD	X-TRANS	Y-TRANS	Z-TRANS	X-ROTAN	Y-ROTAN	Z-ROTAN
1	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
3	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
4	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
5	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
	2	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
6	1	-0.0024	-0.0269	0.0000	0.0000	0.0000	0.0000
	2	4.3083	0.2117	0.0000	0.0000	0.0000	-0.0113
7	1	-0.0017	-0.0337	0.0000	0.0000	0.0000	0.0000
	2	4.3180	0.0006	0.0000	0.0000	0.0000	-0.0100
8	1	-0.0010	-0.0303	0.0000	0.0000	0.0000	0.0000
	2	4.3054	-0.0897	0.0000	0.0000	0.0000	-0.0099
9	1	-0.0004	-0.0174	0.0000	0.0000	0.0000	0.0000
	2	4.2746	-0.0048	0.0000	0.0000	0.0000	-0.0098
10	1	0.0002	-0.0128	0.0000	0.0000	0.0000	0.0000
	2	4.2179	-0.1178	0.0000	0.0000	0.0000	-0.0111
11	1	-0.0035	-0.0436	0.0000	0.0000	0.0000	0.0000
	2	8.7115	0.3415	0.0000	0.0000	0.0000	-0.0116
12	1	-0.0033	-0.0545	0.0000	0.0000	0.0000	0.0000
	2	8.6402	0.0019	0.0000	0.0000	0.0000	-0.0106
13	1	-0.0030	-0.0488	0.0000	0.0000	0.0000	0.0000
	2	8.5935	-0.1528	0.0000	0.0000	0.0000	-0.0105
14	1	-0.0028	-0.0268	0.0000	0.0000	0.0000	0.0000
	2	8.5620	-0.0087	0.0000	0.0000	0.0000	-0.0105
15	1	-0.0027	-0.0197	0.0000	0.0000	0.0000	0.0000
	2	8.5491	-0.1820	0.0000	0.0000	0.0000	-0.0115
16	1	-0.0064	-0.0583	0.0000	0.0000	0.0000	0.0000
	2	13.0045	0.4525	0.0000	0.0000	0.0000	-0.0109
17	1	-0.0063	-0.0729	0.0000	0.0000	0.0000	0.0000
	2	12.9115	0.0039	0.0000	0.0000	0.0000	-0.0100
18	1	-0.0060	-0.0650	0.0000	0.0000	0.0000	0.0000
	2	12.8564	-0.2164	0.0000	0.0000	0.0000	-0.0100
19	1	-0.0059	-0.0337	0.0000	0.0000	0.0000	0.0000
	2	12.8268	-0.0127	0.0000	0.0000	0.0000	-0.0100
20	1	-0.0058	-0.0246	0.0000	0.0000	0.0000	0.0000
	2	12.8226	-0.2273	0.0000	0.0000	0.0000	-0.0109
21	1	-0.0103	-0.0711	0.0000	0.0000	0.0000	0.0000
	2	17.0099	0.5460	0.0000	0.0000	0.0000	-0.0104
22	1	-0.0102	-0.0890	0.0000	0.0000	0.0000	0.0000
	2	16.8814	0.0061	0.0000	0.0000	0.0000	-0.0094
23	1	-0.0102	-0.0787	0.0000	0.0000	0.0000	0.0000
	2	16.7878	-0.2809	0.0000	0.0000	0.0000	-0.0090
24	1	-0.0099	-0.0381	0.0000	0.0000	0.0000	0.0000
	2	16.7773	-0.0167	0.0000	0.0000	0.0000	-0.0091

MARCOS TIPO EN X (PARA TORSIN, FMM)

-- PAGE NO. 9

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = PLANE

JOINT	LOAD	X-TRANS	Y-TRANS	Z-TRANS	X-ROTAN	Y-ROTAN	Z-ROTAN
25	1	-0.0097	-0.0277	0.0000	0.0000	0.0000	0.0000
	2	16.7474	-0.2545	0.0000	0.0000	0.0000	-0.0095
26	1	-0.0131	-0.0820	0.0000	0.0000	0.0000	0.0000
	2	20.9477	0.6232	0.0000	0.0000	0.0000	-0.0107
27	1	-0.0134	-0.1026	0.0000	0.0000	0.0000	0.0000
	2	20.7485	0.0091	0.0000	0.0000	0.0000	-0.0099
28	1	-0.0140	-0.0900	0.0000	0.0000	0.0000	0.0000
	2	20.4624	-0.3464	0.0000	0.0000	0.0000	-0.0097
29	1	-0.0160	-0.0400	0.0000	0.0000	0.0000	0.0000
	2	20.0870	-0.0200	0.0000	0.0000	0.0000	-0.0060
30	1	-0.0175	-0.0289	0.0000	0.0000	0.0000	0.0000

16	111.60	0.00	0.00	0.00	0.00	0.00
	-111.60	0.00	0.00	0.00	0.00	0.00
17	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
18	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
19	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
20	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
21	129.40	0.00	0.00	0.00	0.00	0.00
	-129.40	0.00	0.00	0.00	0.00	0.00
22	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
23	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
24	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00

MARCOS TIPO EN X (PARA TORSIN, FMM)

-- PAGE NO. 7

25	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
26	146.70	0.00	0.00	0.00	0.00	0.00
	-146.70	0.00	0.00	0.00	0.00	0.00
27	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
28	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
29	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
30	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
31	90.45	0.00	0.00	0.00	0.00	0.00
	-90.45	0.00	0.00	0.00	0.00	0.00
32	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
33	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
34	112.70	0.00	0.00	0.00	0.00	0.00
	-112.70	0.00	0.00	0.00	0.00	0.00
35	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
36	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
37	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
38	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
39	130.85	0.00	0.00	0.00	0.00	0.00
	-130.85	0.00	0.00	0.00	0.00	0.00
40	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
41	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
42	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
43	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
44	145.00	0.00	0.00	0.00	0.00	0.00
	-145.00	0.00	0.00	0.00	0.00	0.00
45	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
46	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
47	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
48	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
49	135.70	0.00	0.00	0.00	0.00	0.00
	-135.70	0.00	0.00	0.00	0.00	0.00
50	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
51	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00

\*\*\*\*\* END OF DATA FROM INTERNAL STORAGE \*\*\*\*\*

56. PRINT JOINT DISPLACEMENTS ALL

MARCOS TIPO EN X (PARA TORSIN, FMM)

-- PAGE NO. 8

JOINT DISPLACEMENT (CM RADIANS) STRUCTURE TYPE = PLANE

	0.00	10.77	0.00	0.00	0.00	-4.26
36	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
37	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00
38	0.00	-10.77	0.00	0.00	0.00	4.26
	0.00	10.77	0.00	0.00	0.00	-4.26
41	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
42	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00
43	0.00	-10.77	0.00	0.00	0.00	4.26
	0.00	10.77	0.00	0.00	0.00	-4.26
46	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
47	0.00	-13.97	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
48	0.00	-10.77	0.00	0.00	0.00	4.26
	0.00	10.77	0.00	0.00	0.00	-4.26
51	0.00	-6.98	0.00	0.00	0.00	-4.26
	0.00	6.98	0.00	0.00	0.00	4.26
52	0.00	-10.17	0.00	0.00	0.00	0.00
	0.00	10.17	0.00	0.00	0.00	0.00
53	0.00	-6.98	0.00	0.00	0.00	4.26
	0.00	6.98	0.00	0.00	0.00	-4.26

LOADING 2

SUM-X= -1130.65 SUM-Y= 0.00 SUM-Z= 0.00

SUMMATION OF MOMENTS AROUND ORIGIN-

MX= 0.00 MY= 0.00 MZ= 25627.40

MARCOS TIPO EN X (PARA TORSIN, FMM)

-- PAGE NO. 6

EXTERNAL AND INTERNAL JOINT LOAD SUMMARY-

JO	EXT FX/ INT FX	EXT FY/ INT FY	EXT FZ/ INT FZ	EXT MX/ INT MX	EXT MY/ INT MY	EXT MZ/ INT MZ
1	0.00	0.00	0.00	0.00	0.00	0.00
	199.75	845.84	0.00	0.00	0.00	-838.75
2	0.00	0.00	0.00	0.00	0.00	0.00
	245.65	2.33	0.00	0.00	0.00	-913.66
3	0.00	0.00	0.00	0.00	0.00	0.00
	245.67	-358.40	0.00	0.00	0.00	-912.17
4	0.00	0.00	0.00	0.00	0.00	0.00
	244.05	-19.08	0.00	0.00	0.00	-905.87
5	0.00	0.00	0.00	0.00	0.00	0.00
	195.54	-470.69	0.00	0.00	0.00	-821.13
6	45.25	0.00	0.00	0.00	0.00	0.00
	-45.25	0.00	0.00	0.00	0.00	0.00
7	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
8	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
9	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
10	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
11	83.00	0.00	0.00	0.00	0.00	0.00
	-83.00	0.00	0.00	0.00	0.00	0.00

JT	EXT FX/ INT FX	EXT FY/ INT FY	EXT FZ/ INT FZ	EXT MX/ INT MX	EXT MY/ INT MY	EXT MZ/ INT MZ
1	0.00	-5.41	0.00	0.00	0.00	0.00
	-0.64	-107.49	0.00	0.00	0.00	1.32
2	0.00	-5.41	0.00	0.00	0.00	0.00
	-0.10	-134.53	0.00	0.00	0.00	0.36
3	0.00	-5.41	0.00	0.00	0.00	0.00
	0.05	-121.18	0.00	0.00	0.00	0.04
4	0.00	-5.41	0.00	0.00	0.00	0.00
	0.11	-69.44	0.00	0.00	0.00	-0.12
5	0.00	-5.41	0.00	0.00	0.00	0.00
	0.59	-51.29	0.00	0.00	0.00	-0.97
6	0.00	-12.40	0.00	0.00	0.00	-4.26
	0.00	12.40	0.00	0.00	0.00	4.26
7	0.00	-15.59	0.00	0.00	0.00	0.00
	0.00	15.59	0.00	0.00	0.00	0.00
8	0.00	-15.59	0.00	0.00	0.00	0.00
	0.00	15.59	0.00	0.00	0.00	0.00
9	0.00	-15.59	0.00	0.00	0.00	0.00
	0.00	15.59	0.00	0.00	0.00	0.00
10	0.00	-12.40	0.00	0.00	0.00	4.26
	0.00	12.40	0.00	0.00	0.00	-4.26
11	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
12	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00
13	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
14	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
15	0.00	-10.77	0.00	0.00	0.00	4.26
	0.00	10.77	0.00	0.00	0.00	-4.26
16	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
17	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00
18	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00
19	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00
20	0.00	-10.77	0.00	0.00	0.00	4.26
	0.00	10.77	0.00	0.00	0.00	-4.26
21	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
22	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.96	0.00	0.00	0.00	0.00

MARCOS TIPO EN X (PARA TORSIN, FMM)

-- PAGE NO. 5

23	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
24	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
25	0.00	-10.77	0.00	0.00	0.00	4.26
	0.00	10.77	0.00	0.00	0.00	-4.26
26	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
27	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
28	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
29	0.00	-10.17	0.00	0.00	0.00	0.00
	0.00	10.17	0.00	0.00	0.00	0.00
30	0.00	-6.98	0.00	0.00	0.00	4.26
	0.00	6.98	0.00	0.00	0.00	-4.26
31	0.00	-10.77	0.00	0.00	0.00	-4.26
	0.00	10.77	0.00	0.00	0.00	4.26
32	0.00	-13.96	0.00	0.00	0.00	0.00
	0.00	13.97	0.00	0.00	0.00	0.00
33	0.00	-10.77	0.00	0.00	0.00	4.26



PROBLEM STATISTICS

NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS = 45/ 70/ 5  
 ORIGINAL/FINAL BAND-WIDTH = 5/ 5  
 TOTAL PRIMARY LOAD CASES = 2, TOTAL DEGREES OF FREEDOM = 120  
 SIZE OF STIFFNESS MATRIX = 2160 DOUBLE PREC. WORDS  
 REQRD/AVAIL. DISK SPACE = 12.11/ 2047.7 MB, EXMEM = 1817.8 MB

MARCOS TIPO EN X (PARA TORSIN, FMM) -- PAGE NO. 3

LOADING 1 PESO PROPIO

SELFWEIGHT Y -1.000

ACTUAL WEIGHT OF THE STRUCTURE = 511.005 MTON

\*\*\*TOTAL APPLIED LOAD ( MTON METE ) SUMMARY (LOADING 1 )

SUMMATION FORCE-X = 0.00  
 SUMMATION FORCE-Y = -511.00  
 SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-

MX= 0.00 MY= 0.00 MZ= -6755.64

LOADING 2 FUERZAS LATERALES PARA RIGIDEZES

JOINT LOAD - UNIT MTON METE

JOINT	FORCE-X	FORCE-Y	FORCE-Z	MOM-X	MOM-Y	MOM-Z
51	135.70	0.00	0.00	0.00	0.00	0.00
46	145.00	0.00	0.00	0.00	0.00	0.00
41	130.85	0.00	0.00	0.00	0.00	0.00
36	112.70	0.00	0.00	0.00	0.00	0.00
31	90.45	0.00	0.00	0.00	0.00	0.00
26	146.70	0.00	0.00	0.00	0.00	0.00
21	129.40	0.00	0.00	0.00	0.00	0.00
16	111.60	0.00	0.00	0.00	0.00	0.00
11	83.00	0.00	0.00	0.00	0.00	0.00
6	45.25	0.00	0.00	0.00	0.00	0.00

\*\*\*TOTAL APPLIED LOAD ( MTON METE ) SUMMARY (LOADING 2 )

SUMMATION FORCE-X = 1130.65  
 SUMMATION FORCE-Y = 0.00  
 SUMMATION FORCE-Z = 0.00

SUMMATION OF MOMENTS AROUND THE ORIGIN-

MX= 0.00 MY= 0.00 MZ= -25627.40

- Processing Element Stiffness Matrix. 16: 7:45
- Processing Global Stiffness Matrix. 16: 7:45
- Processing Triangular Factorization. 16: 7:45
- Calculating Joint Displacements. 16: 7:45
- Calculating Member Forces. 16: 7:45

\*\*\*TOTAL REACTION ( MTON METE ) SUMMARY

LOADING 1

MARCOS TIPO EN X (PARA TORSIN, FMM) -- PAGE NO. 4

SUM-X= 0.00 SUM-Y= 511.00 SUM-Z= 0.00

SUMMATION OF MOMENTS AROUND ORIGIN-

MX= 0.00 MY= 0.00 MZ= 6755.64

EXTERNAL AND INTERNAL JOINT LOAD SUMMARY-

```

*****
*
*          STAAD/Pro STAAD-III
*          Revision 3.1
*          Proprietary Program of
*          RESEARCH ENGINEERS, Inc.
*          Date=   SEP 24, 2001
*          Time=   16: 7:45
*
*          USER ID: Unknown User
*****

```

1. STAAD PLANE MARCOS TIPO EN X (PARA TORSIN, FMM)
2. START JOB INFORMATION
3. ENGINEER DATE 24-SEP-01
4. END JOB INFORMATION
5. INPUT WIDTH 72
6. UNIT METER MTON
7. JOINT COORDINATES
8. 1 0 0 0; 2 8 0 0; 3 16 0 0; 4 24 0 0; 5 32 0 0; 6 0 5 0; 7 8 5 0
9. 8 16 5 0; 9 24 5 0; 10 32 5 0; 11 0 8.5 0; 12 8 8.5 0; 13 16 8.5 0
10. 14 24 8.5 0; 15 32 8.5 0; 16 0 12 0; 17 8 12 0; 18 16 12 0; 19 24 12 0
11. 20 32 12 0; 21 0 15.5 0; 22 8 15.5 0; 23 16 15.5 0; 24 24 15.5 0
12. 25 32 15.5 0; 26 0 19 0; 27 8 19 0; 28 16 19 0; 29 24 19 0; 30 32 19 0
13. 31 0 22.5 0; 32 8 22.5 0; 33 16 22.5 0; 36 0 26 0; 37 8 26 0
14. 38 16 26 0; 41 0 29.5 0; 42 8 29.5 0; 43 16 29.5 0; 46 0 33 0
15. 47 8 33 0; 48 16 33 0; 51 0 36.5 0; 52 8 36.5 0; 53 16 36.5 0
16. MEMBER INCIDENCES
17. 1 1 6; 2 2 7; 3 3 8; 4 4 9; 5 5 10; 6 6 7; 7 7 8; 8 8 9; 9 9 10
18. 10 6 11; 11 7 12; 12 8 13; 13 9 14; 14 10 15; 15 11 12; 16 12 13
19. 17 13 14; 18 14 15; 19 11 16; 20 12 17; 21 13 18; 22 14 19; 23 15 20
20. 24 16 17; 25 17 18; 26 18 19; 27 19 20; 28 16 21; 29 17 22; 30 18 23
21. 31 19 24; 32 20 25; 33 21 22; 34 22 23; 35 23 24; 36 24 25; 37 21 26
22. 38 22 27; 39 23 28; 40 24 29; 41 25 30; 42 26 27; 43 27 28; 44 28 29
23. 45 29 30; 46 26 31; 47 27 32; 48 28 33; 51 31 32; 52 32 33; 55 31 36
24. 56 32 37; 57 33 38; 60 36 37; 61 37 38; 64 36 41; 65 37 42; 66 38 43
25. 69 41 42; 70 42 43; 73 41 46; 74 42 47; 75 43 48; 78 46 47; 79 47 48
26. 82 46 51; 83 47 52; 84 48 53; 87 51 52; 88 52 53
27. START GROUP DEFINITION
28. \_COLU 84 83 82 75 74 73 66 65 64 57 56 55 48 47 46 41 40 39 38 37 32 -
29. 31 30 29 28 23 22 21 20 19 14 13 12 11 10 5 4 3 2 1
30. \_VIGS 88 87 79 78 70 69 61 60 52 51 45 44 43 42 36 35 34 33 27 26 25 -
31. 24 18 17 16 15 9 8 7 6
32. END
33. MEMBER PROPERTY AMERICAN
34. \_COLU PRIS YD 0.95 ZD 0.95
35. \_VIGS PRIS YD 0.95 ZD 0.35
36. CONSTANTS
37. E 2.2136E+006 ALL
38. DENSITY 2.4 ALL
39. SUPPORTS
40. 1 TO 5 FIXED
41. LOAD 1 PESO PROPIO

MARCOS TIPO EN X (PARA TORSIN, FMM)

42. SELFWEIGHT Y -1
43. LOAD 2 FUERZAS LATERALES PARA RIGIDEZES
44. JOINT LOAD
45. 51 FX 135.7
46. 46 FX 145
47. 41 FX 130.85
48. 36 FX 112.7
49. 31 FX 90.45
50. 26 FX 146.7
51. 21 FX 129.4
52. 16 FX 111.6
53. 11 FX 83
54. 6 FX 45.25
55. PERFORM ANALYSIS PRINT ALL

STAAD PLANE MARCOS TIPO EN X (PARA TORSIÓN, FMM)

START JOB INFORMATION

ENGINEER DATE 24-Sep-01

END JOB INFORMATION

INPUT WIDTH 72

UNIT METER MTON

JOINT COORDINATES

1 0 0 0; 2 8 0 0; 3 16 0 0; 4 24 0 0; 5 32 0 0; 6 0 5 0; 7 8 5 0;  
8 16 5 0; 9 24 5 0; 10 32 5 0; 11 0 8.5 0; 12 8 8.5 0; 13 16 8.5 0;  
14 24 8.5 0; 15 32 8.5 0; 16 0 12 0; 17 8 12 0; 18 16 12 0; 19 24 12 0;  
20 32 12 0; 21 0 15.5 0; 22 8 15.5 0; 23 16 15.5 0; 24 24 15.5 0;  
25 32 15.5 0; 26 0 19 0; 27 8 19 0; 28 16 19 0; 29 24 19 0; 30 32 19 0;  
31 0 22.5 0; 32 8 22.5 0; 33 16 22.5 0; 36 0 26 0; 37 8 26 0;  
38 16 26 0; 41 0 29.5 0; 42 8 29.5 0; 43 16 29.5 0; 46 0 33 0;  
47 8 33 0; 48 16 33 0; 51 0 36.5 0; 52 8 36.5 0; 53 16 36.5 0;

MEMBER INCIDENCES

1 1 6; 2 2 7; 3 3 8; 4 4 9; 5 5 10; 6 6 7; 7 7 8; 8 8 9; 9 9 10;  
10 6 11; 11 7 12; 12 8 13; 13 9 14; 14 10 15; 15 11 12; 16 12 13;  
17 13 14; 18 14 15; 19 11 16; 20 12 17; 21 13 18; 22 14 19; 23 15 20;  
24 16 17; 25 17 18; 26 18 19; 27 19 20; 28 16 21; 29 17 22; 30 18 23;  
31 19 24; 32 20 25; 33 21 22; 34 22 23; 35 23 24; 36 24 25; 37 21 26;  
38 22 27; 39 23 28; 40 24 29; 41 25 30; 42 26 27; 43 27 28; 44 28 29;  
45 29 30; 46 26 31; 47 27 32; 48 28 33; 51 31 32; 52 32 33; 55 31 36;  
56 32 37; 57 33 38; 60 36 37; 61 37 38; 64 36 41; 65 37 42; 66 38 43;  
69 41 42; 70 42 43; 73 41 46; 74 42 47; 75 43 48; 78 46 47; 79 47 48;  
82 46 51; 83 47 52; 84 48 53; 87 51 52; 88 52 53;

START GROUP DEFINITION

\_COLU 84 83 82 75 74 73 66 65 64 57 56 55 48 47 46 41 40 39 38 37 32 -  
31 30 29 28 23 22 21 20 19 14 13 12 11 10 5 4 3 2 1  
\_VIGS 88 87 79 78 70 69 61 60 52 51 45 44 43 42 36 35 34 33 27 26 25 -  
24 18 17 16 15 9 8 7 6

END

MEMBER PROPERTY AMERICAN

\_COLU PRIS YD 0.95 ZD 0.95

\_VIGS PRIS YD 0.95 ZD 0.35

CONSTANTS

E 2.2136e+006 ALL

DENSITY 2.4 ALL

SUPPORTS

1 TO 5 FIXED

LOAD 1 PESO PROPIO

SELFWEIGHT Y -1

LOAD 2 FUERZAS LATERALES PARA RIGIDECE

JOINT LOAD

51 FX 135.7

46 FX 145

41 FX 130.85

36 FX 112.7

31 FX 90.45

26 FX 146.7

21 FX 129.4

16 FX 111.6

11 FX 83

6 FX 45.25

PERFORM ANALYSIS PRINT ALL

PRINT JOINT DISPLACEMENTS ALL

PRINT SUPPORT REACTION

LOAD LIST 2

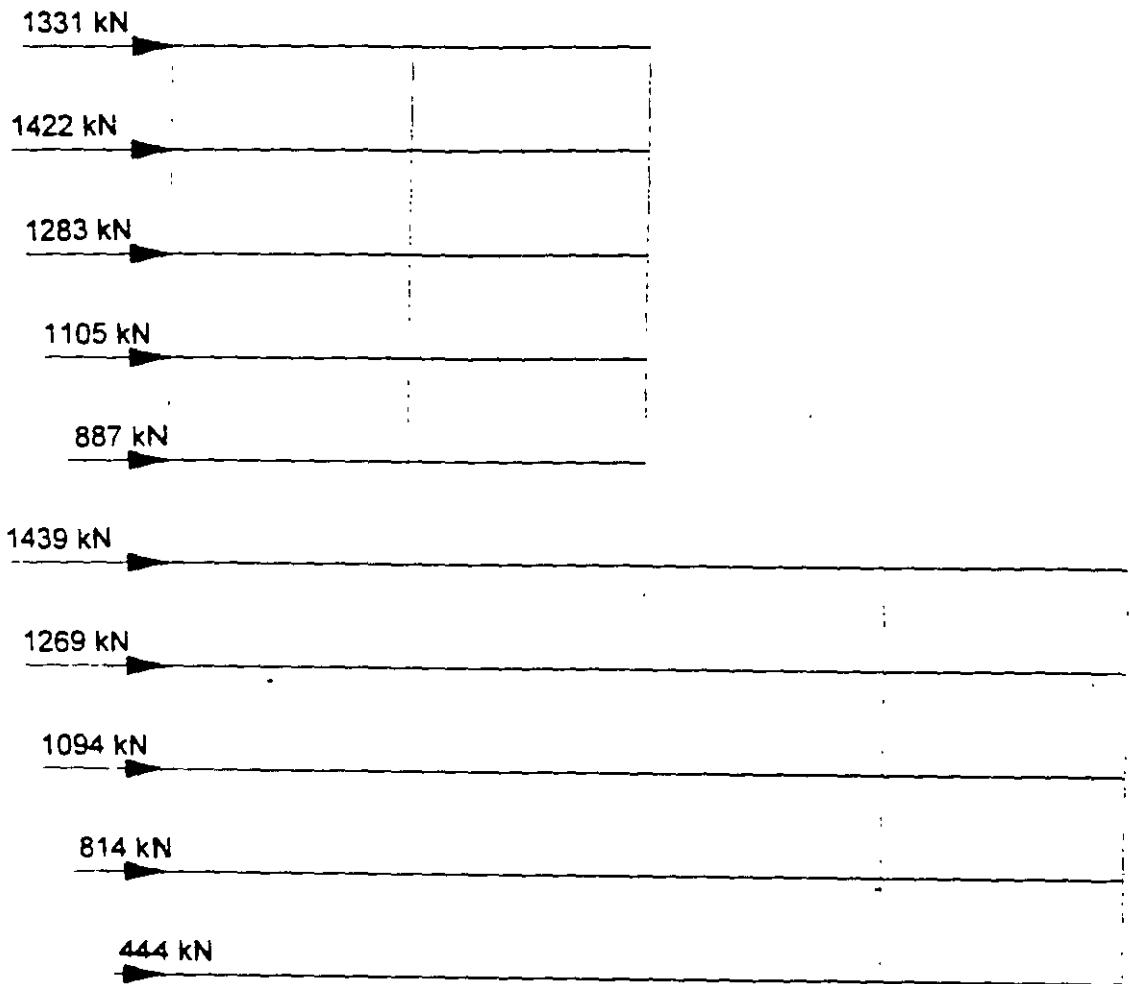
PRINT JOINT DISPLACEMENTS LIST 53 52 51 48 47 46 43 42 41 38 37 36 33 -  
32 31 30 29 28 27 26 25 24 23 22 21 20 19 18 17 16 15 14 13 12 11 10 9 -  
8 7 6

FINISH



Software licensed to Unknown User

Job No	Sheet No 1	Rev
Part		
Ref		
By	Date 24-Sep-01	Chg
Client	File mar3-2d-10n.std	Date/Time 24-Sep-2001 16:07



N1	N2	N3	N4	N5
X = -1958.840 kN	X = -2409.022 kN	X = -2409.167 kN	X = -2383.291 kN	X = -1917.567 kN
Y = -8294.874 kN	Y = -22.842 kN	Y = 3514.889 kN	Y = 187.081 kN	Y = 4815.932 kN
MZ = 8225.302 kNm	MZ = 8959.947 kNm	MZ = 8945.357 kNm	MZ = 8883.491 kNm	MZ = 8052.479 kNm



Software licensed to Unknown User

Job No

Sheet No

1

Rev

Part

Ref

By

Date 24-Sep-01

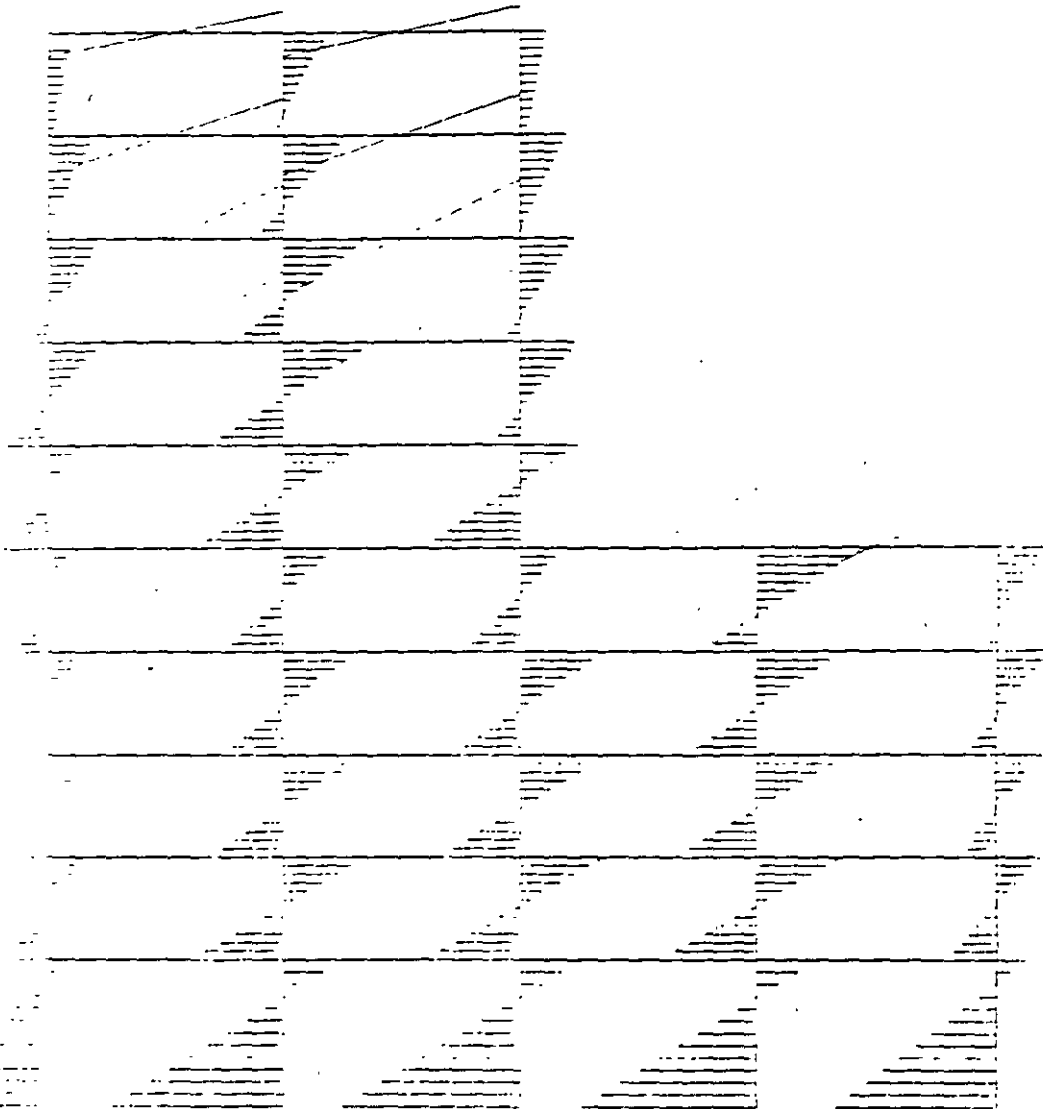
Chd

Client

File mar3-2d-10n.std

Date/Time

24-Sep-2001 16:07





Software licensed to Unknown User

Job No

Sheet No

1

Rev

Part

Job Title

Ref

By

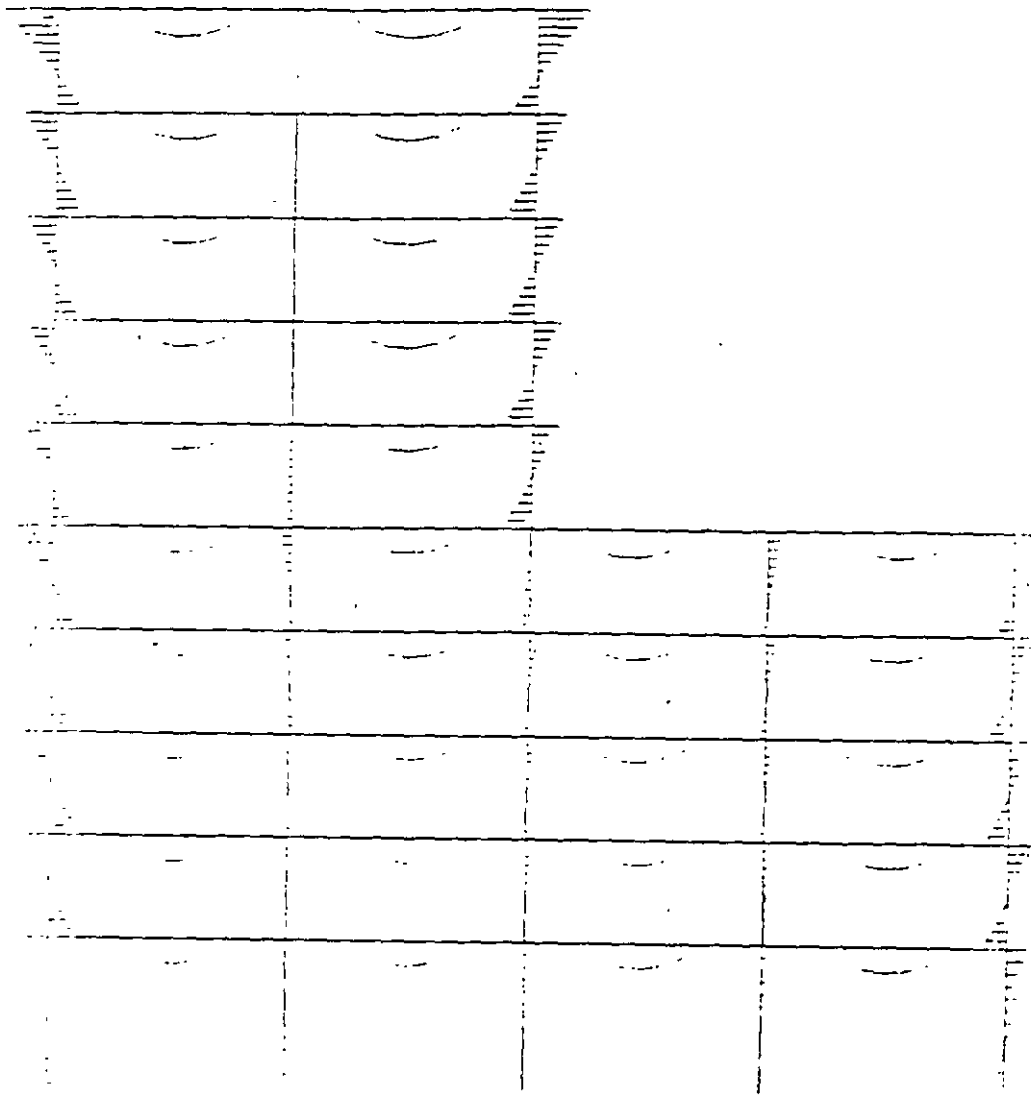
Date 24-Sep-01

Chd

Client

File mar3-2d-10n.std

Date/Time 24-Sep-2001 16:07



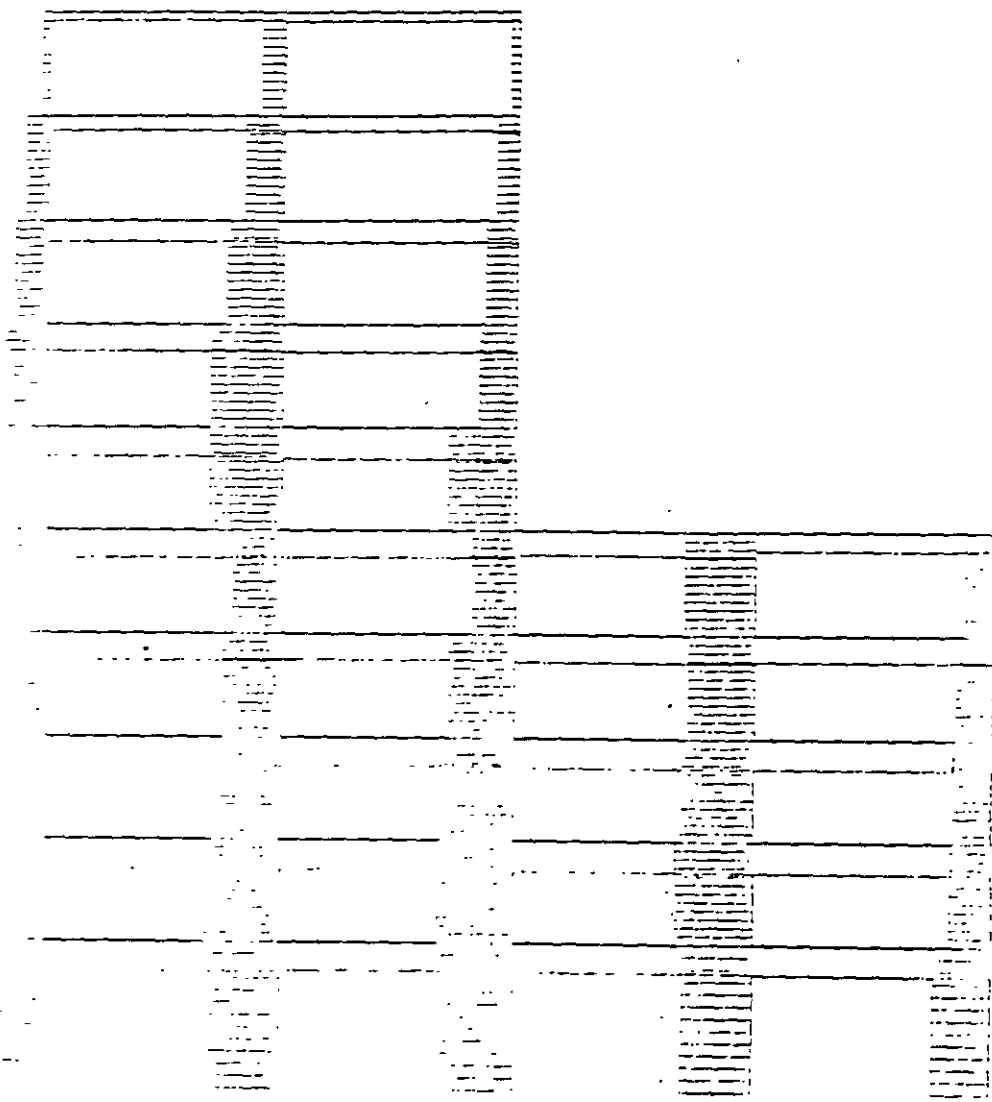


Software licensed to Unknown User

Job No	Sheet No	Rev
	1	
Part		
Ref		
By	Date	Ord
	24-Sep-01	
File	Date/Time	
mar3-2d-10n.std	24-Sep-2001 16:07	

Job Title

Client





**FACULTAD DE INGENIERÍA UNAM  
DIVISIÓN DE EDUCACIÓN CONTINUA**



**CA 354  
STAAD PRO PARA ANÁLISIS Y DISEÑO  
ESTRUCTURAL**

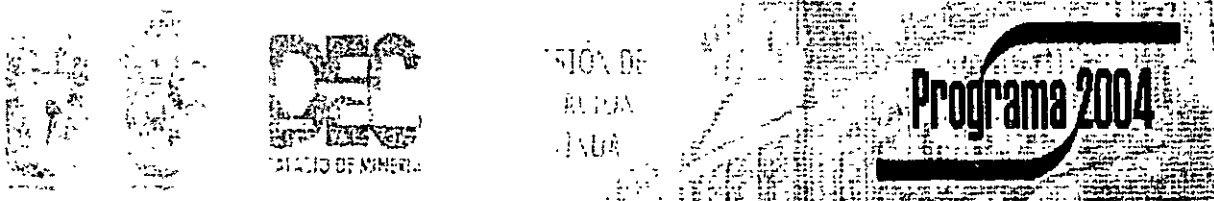
**TEMA:**

**EJEMPLOS DEL PROBLEMA No. 2**

**COORDINADOR: ING. JOSÉ LUIS ESQUIVEL ÁVILA**

**DEL 22 AL 29 DE NOVIEMBRE DE 2004**

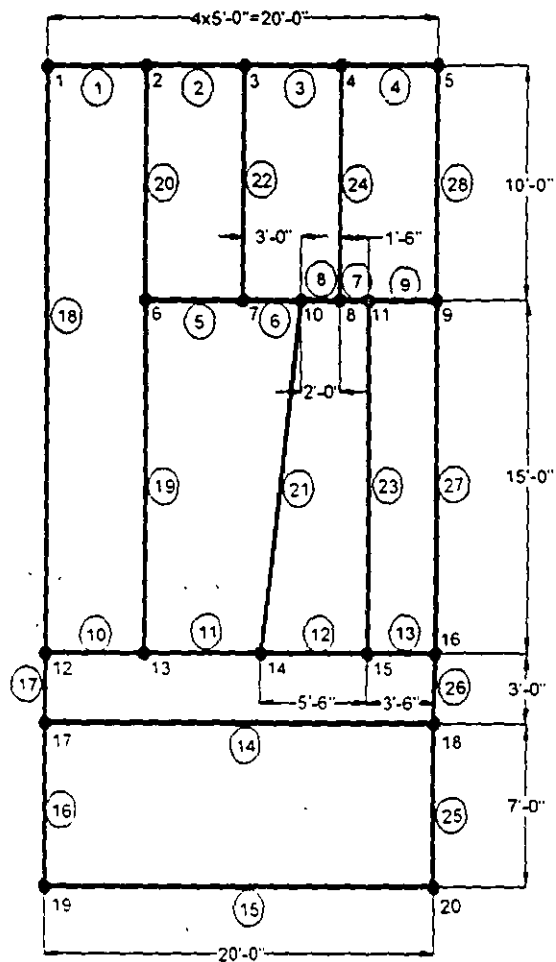
**PALACIO DE MINERÍA**





## Example Problem No. 2

A floor structure (bound by global X-Z axis) made up of steel beams is subjected to area load (i.e. load/area of floor). Load generation based on one-way distribution is illustrated in this example.



1. Select the *STAAD.Pro* icon from the STAAD.Pro 2001 program group.

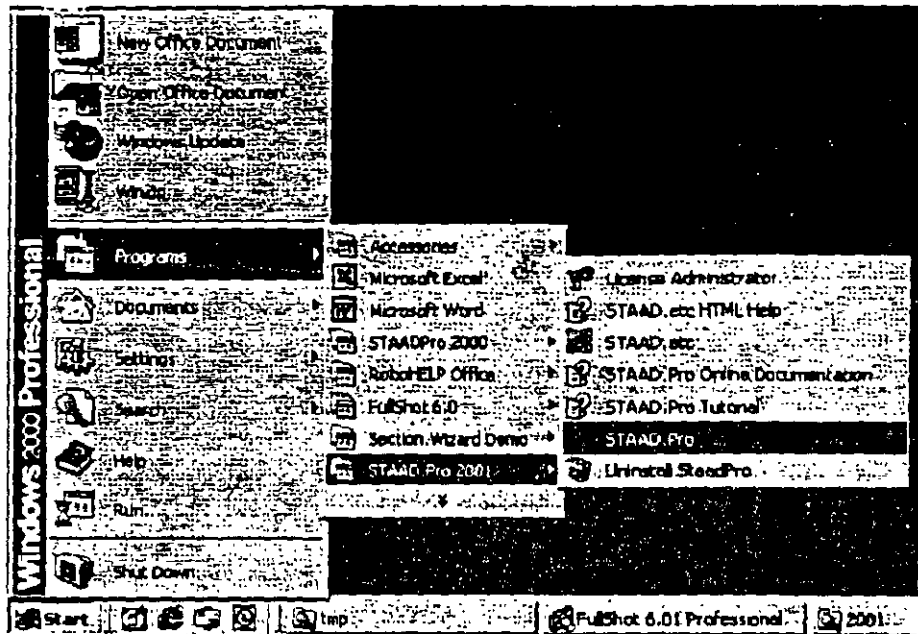


Figure 2. 1

The STAAD.Pro Graphical Environment will be invoked.

2. The units in which we wish to create this model are the *English* units. (feet, kip, etc.) The default unit system setting is whatever we chose during the installation of the program. If you had chosen *Metric* at the time of installation, you may want to change it to *English*. To do so, click on the *File / Configure* menu option (see Figure 2.2) and choose the appropriate one (*English* for our case). Then, click on the *Accept* button.

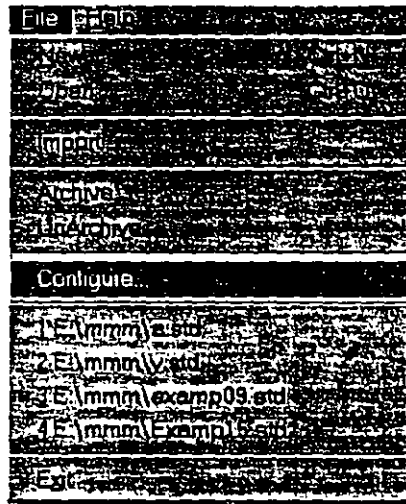


Figure 2. 2

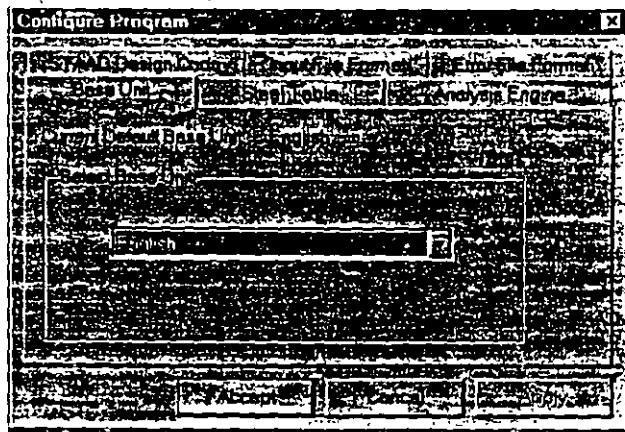


Figure 2. 3

3. To create a new structure, click on the *File / New* option in the STAAD.Pro screen that opens (as shown in Figure 2.4).

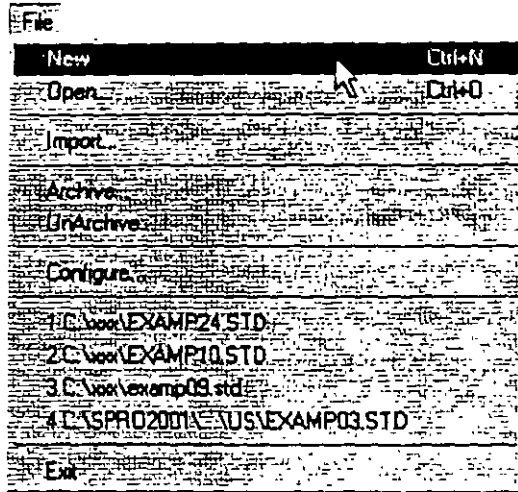


Figure 2.4

4. In the *New File Setup* dialog box, choose *Floor* as the *Structure Type* and specify an optional *Title* (A FLOOR FRAME DESIGN WITH AREA LOAD). Then click on the *Next* button as shown in Figure 2.5.

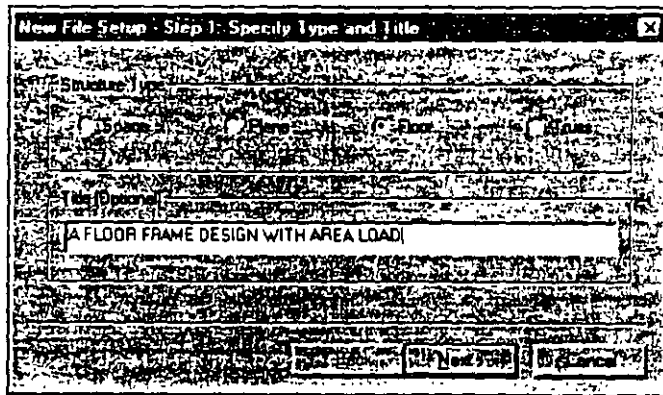


Figure 2.5

5. The next dialog box that comes up prompts us to select the length and force units in which we wish to start working in. So, specify the *Length Units* as *Foot*, the *Force Units* as *KiloPound* and click on the *Next* button as shown in Figure 2.6. Please note that the input units may be changed subsequently at any stage of building of the model.

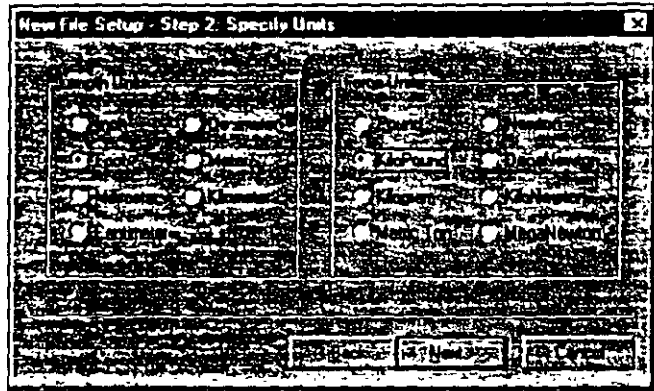


Figure 2. 6

6. This dialog box confirms the information of our previous selections. Press the *Finish* button. (see Figure 2.7)

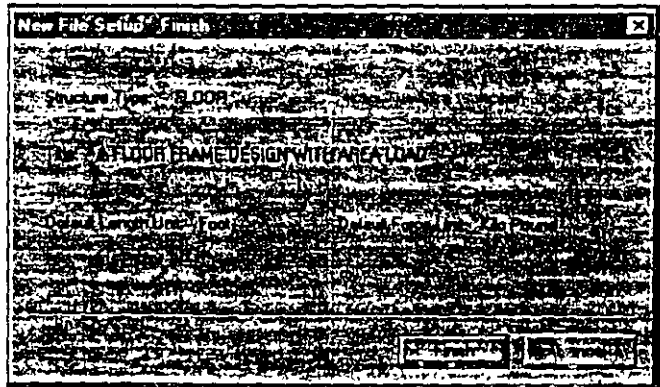


Figure 2. 7

Once we press the *Finish* button, the STAAD.Pro main window appears on the screen.

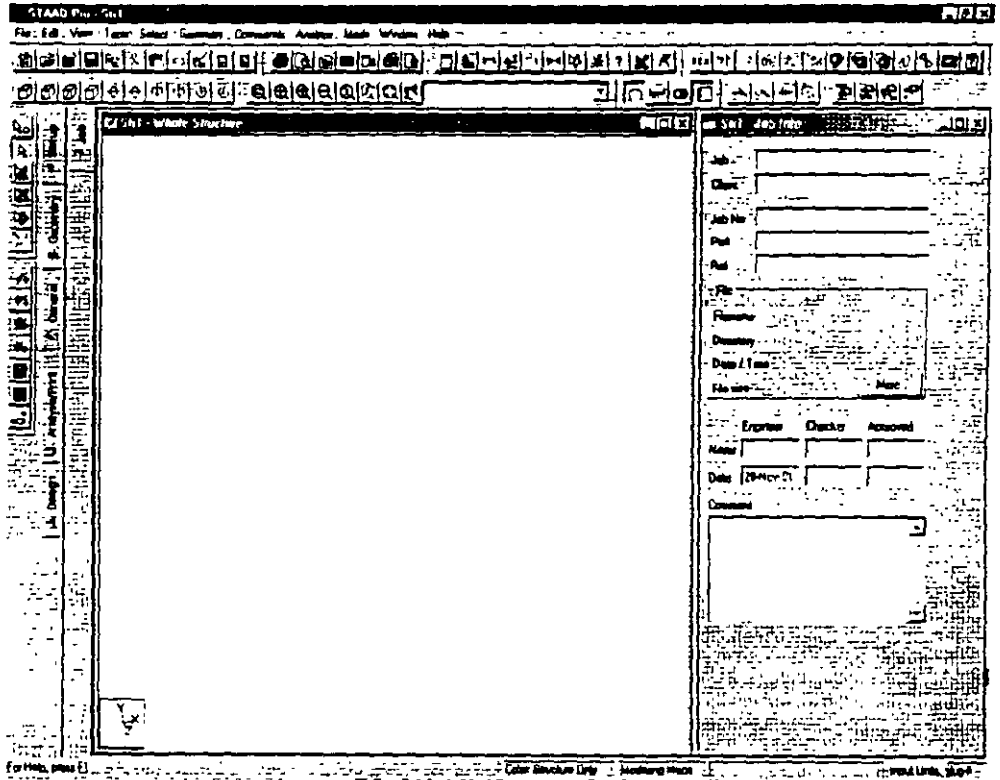


Figure 2. 8

## Creating Nodes 1 to 5 And Beams 1 to 4

7. Select *Geometry / Beam* Page from the left side of the screen. In the *Snap Node/Beam* dialog box that appears in the Data Area (on the right side of the screen), choose *X-Z* as the *Plane* and in the *Construction Lines* group, set *X* and *Z* to 20 with a spacing of 1ft. (see figure below) This 20X20 grid too is only a starting grid setting to enable us to start drawing the structure, it does not restrict our model to those limits as we will see later.

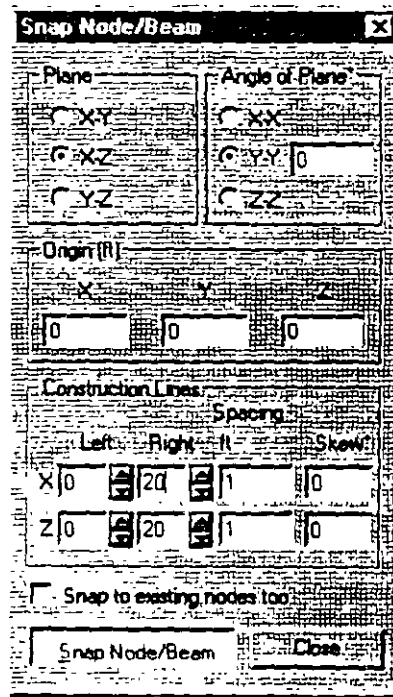


Figure 2.9

8. With the help of the mouse, click at the origin (0, 0) to create the first node. In a similar fashion, click on the following points to create nodes and automatically join successive nodes by beam members.

(5.0), (10.0), (15.0) and (20.0)

The exact location of the mouse arrow can be monitored on the status bar located at the bottom of the window where the X, Y, and Z coordinates of the current cursor position are continuously updated

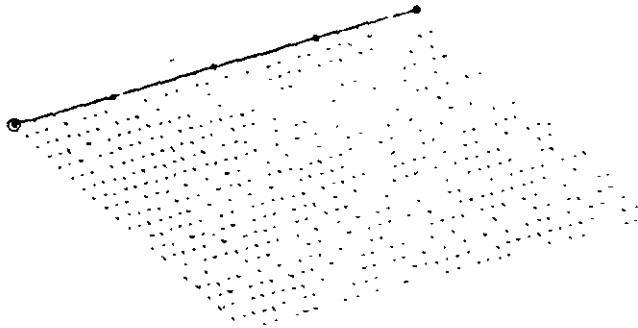


Figure 2. 10



9. After having created these four beams and five nodes, let us close the *Snap Node/Beam* dialog box.

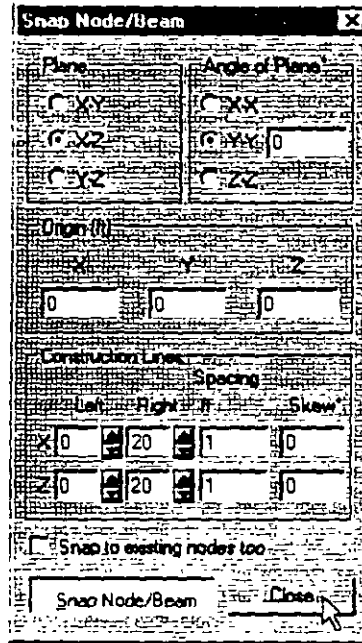


Figure 2. 11

## Switching On Node And Beam Labels

10. In order to display the *node* and *beam* numbers, right click anywhere within the drawing area. In the pop-up menu that appears, choose *Labels* as shown below.

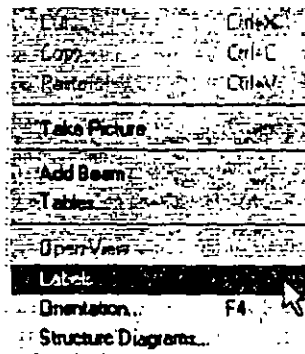


Figure 2. 12

Alternatively, one may access this option by selecting the *View* menu from the top menu bar followed by *Structure Diagrams*, and the *Labels* tab of the dialog box that opens.

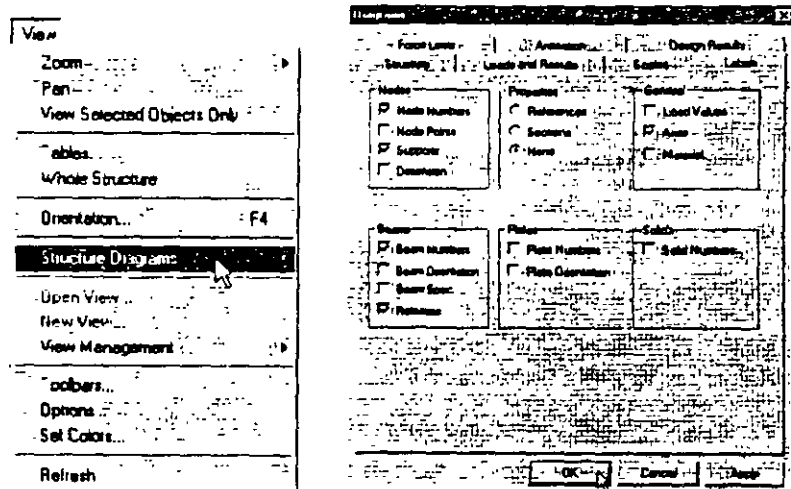


Figure 2. 13

11. In the *Diagrams* dialog box that appears, turn the *Node Numbers* and *Beam Numbers* on and then click on *OK*.

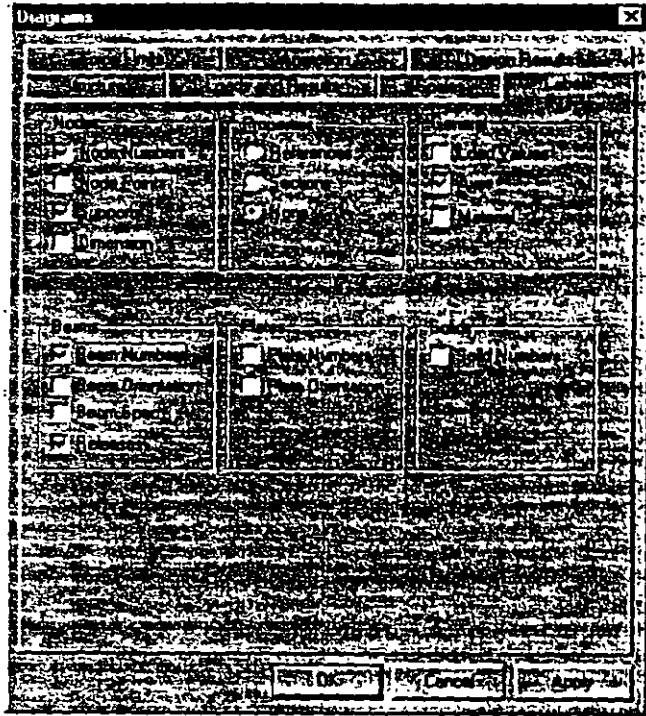


Figure 2. 14

The nodes and beams are now labeled on the drawing.

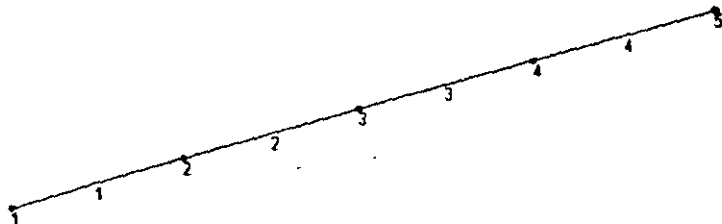



Figure 2. 15

## Creating Members 5 to 9

12. As shown on the title page of this example, beams 5 to 9 are located at the grid line  $Z = 10$ ft. We could create them in a manner similar to what we did for creating beams 1 to 5 by clicking at the relevant grid points. Alternatively, we may use STAAD's *Translational Repeat* facility to do the same.

Let us choose the latter method. First, select members 2, 3, and 4 with the help of the *Beam Cursor* . The *Beam Cursor* can also be selected by choosing the *Beam Cursor* option from the *Select* menu. To select multiple beams, hold down the 'Ctrl' key while clicking on the members.

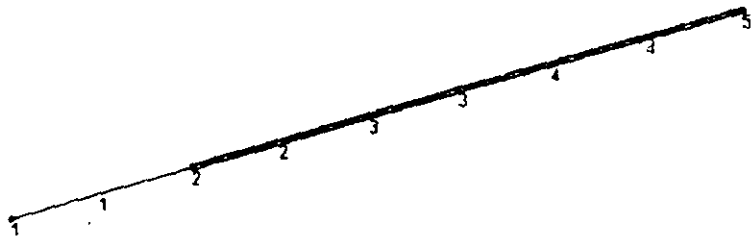


Figure 2. 16

Next, go to the *Geometry / Translational Repeat* menu option as shown below.

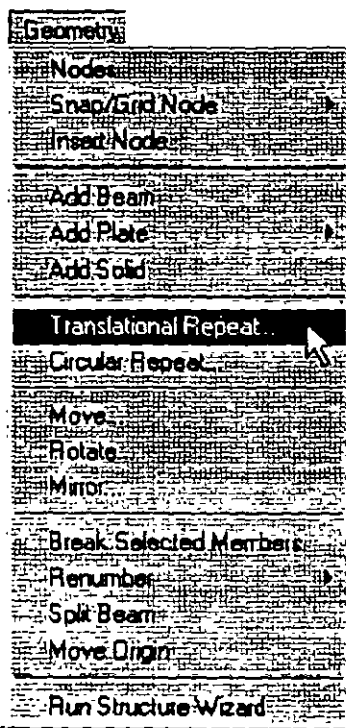


Figure 2. 17

13. In the *3D Repeat* dialog box that opens, specify the *Global Direction* as *Z*, *No of Steps* as *1* and the *Default Step Spacing* as *10ft*. Leave the *Link Steps* box unchecked. Then, click on *OK*.

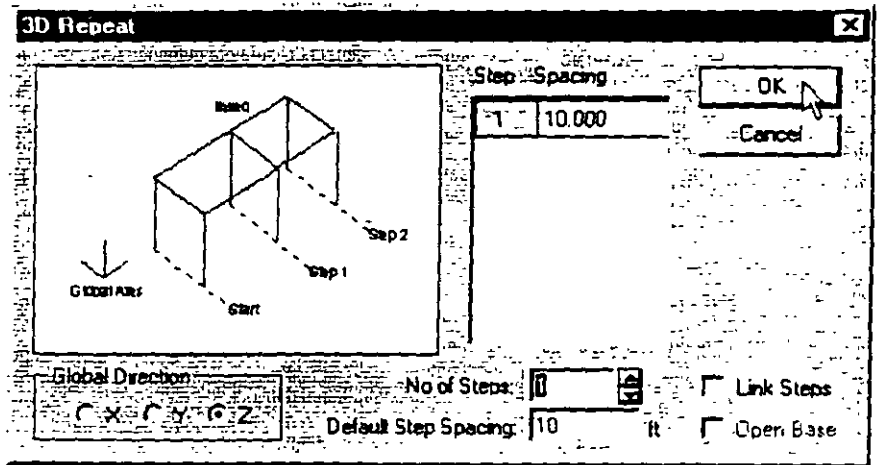


Figure 2. 18

After completing the translational repeat process, the structure should look as follows:

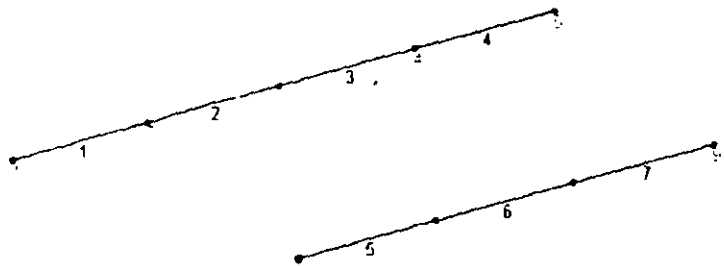


Figure 2. 19

14. Next, let us split member 6 into two parts of length 3ft and 2ft respectively. First, select the member by clicking on it and then click the right mouse button. In the pop-up menu that appears, choose the *Insert Node* option as shown below.

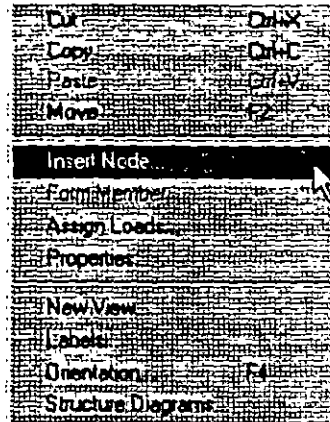


Figure 2. 20

Alternatively, one may access this option by going to the *Geometry* menu and choosing *Insert Node*.

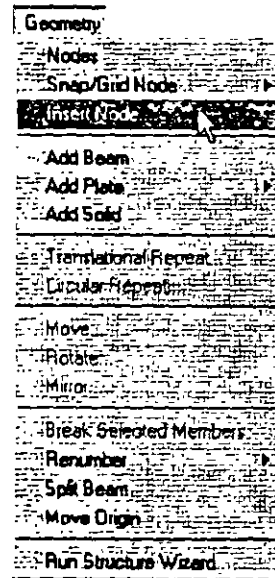


Figure 2. 21

15. In the *Insert Nodes* dialog box that opens, specify the *Distance* as 3ft, click on the *Add New Point* button, and click on *OK*.

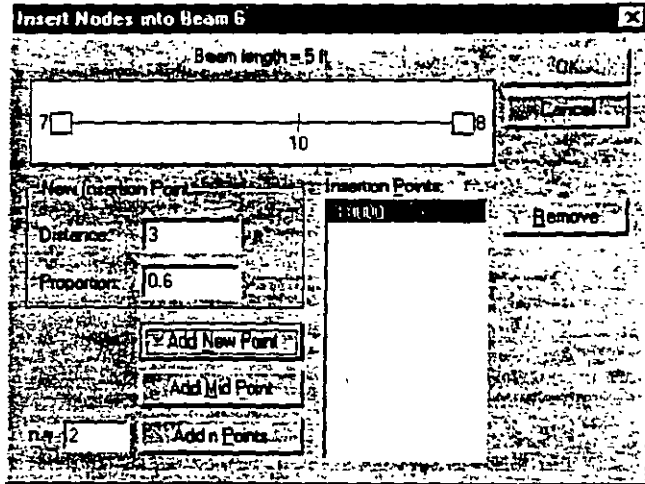


Figure 2. 22



After the insertion is done, the structure will look as shown below. Notice that a new node (number 10) has been added.

Before insertion

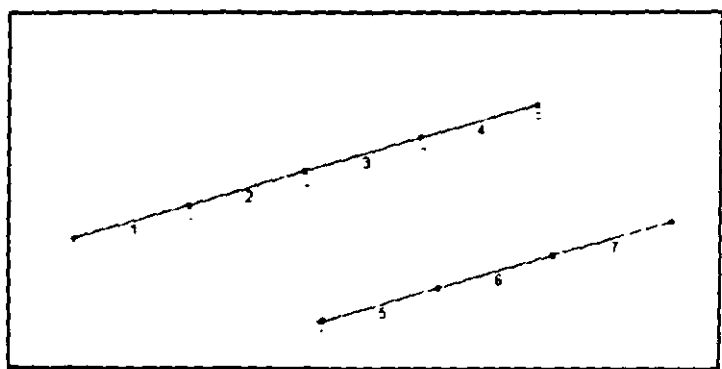


Figure 2. 23

After insertion

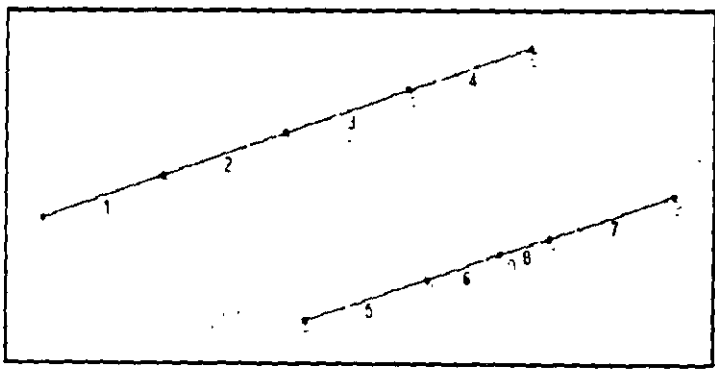


Figure 2. 24

16. Repeat this procedure to split member 7 to create node 11. In the *Insert Nodes* dialog box, specify the *Distance* as 1.5ft, click on the *Add New Point* button, and click on *OK*.

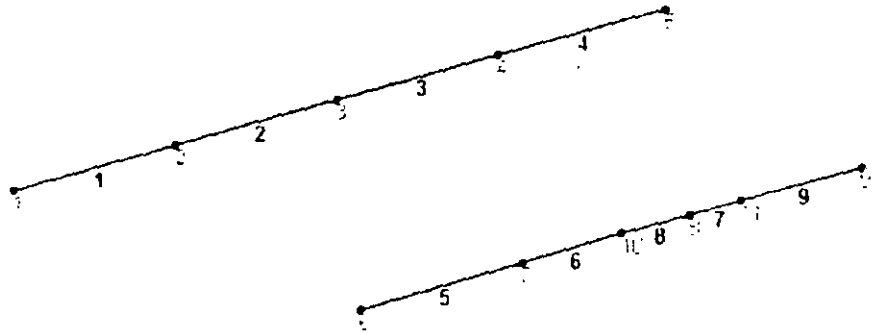


Figure 2. 25

## Creating Members 10 to 13

17. To create the X direction beams at  $Z = 25$ ft, just as we did before, we can use the *Translational Repeat* operation by using the X direction beams at  $Z = 0$  as the basis. Yet another method is the *Copy-Paste* facility from the *Edit* menu. To apply this method, we first select members 1, 2, 3, and 4. Click the right mouse button and choose *Copy* from the pop-up menu (or click on the *Edit* menu and choose *Copy*). Once again, click the right mouse button and select *Paste Beams* (or choose *Paste Beams* from the *Edit* menu) as shown below.

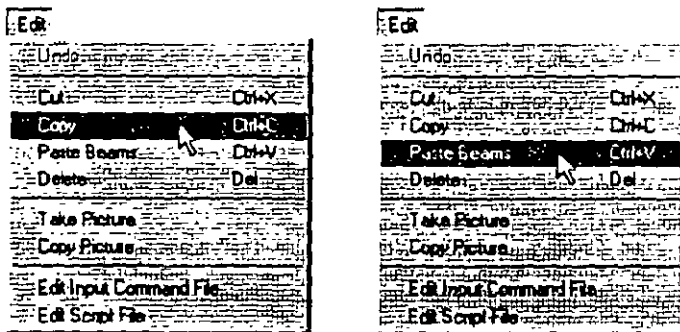


Figure 2. 26

18. Provide 0, 0, and 25 for X, Y and Z respectively and click on the *OK* button.

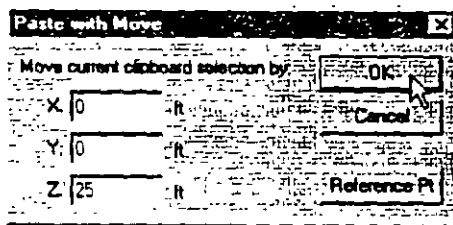


Figure 2. 27

19. As we click on the *OK* button, the following message box appears. This is only a reminder that we need to subsequently assign the required properties to these entities as well. Let us click on the *OK* button.

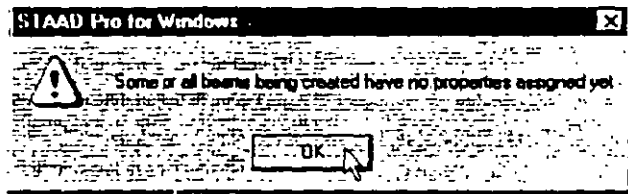


Figure 2. 28

Members 10 to 13 will appear on the model as shown below.

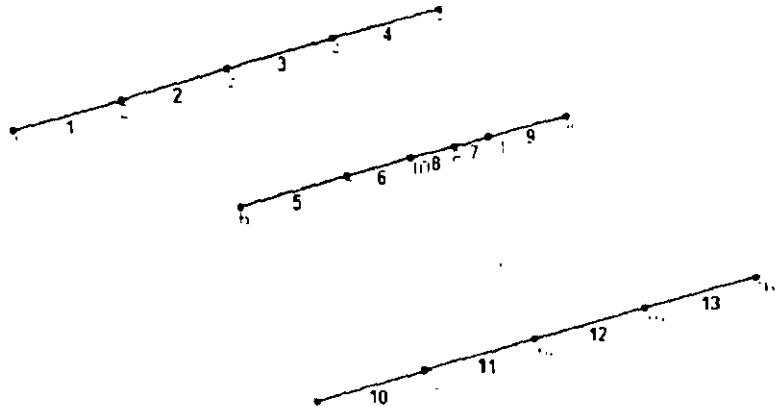

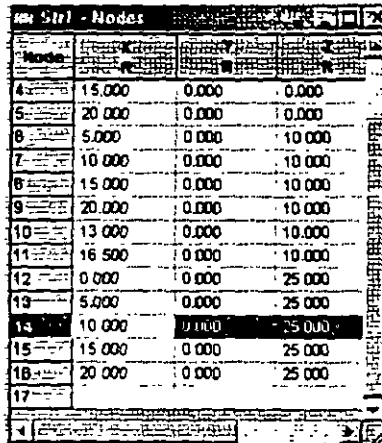


Figure 2. 29

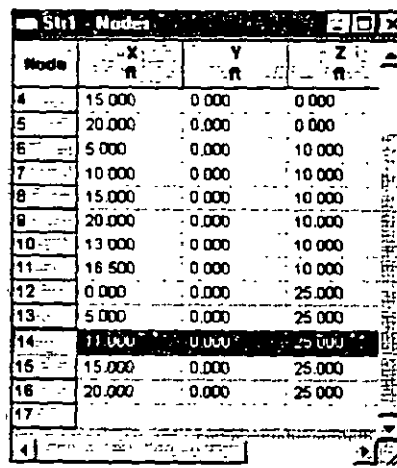
20. If we look at the figure on the title page, we will observe that beam 11 is 6ft long and not 5ft. So, the X co-ordinates of its end is at 11, not 10. To make this change, let us change the co-ordinates of Node 14. To do that, select that node using the *Nodes Cursor* . The data relating to Node 14 will be highlighted in the *Nodes* table located in the Data Area.



Node	X	Y	Z
4	15.000	0.000	0.000
5	20.000	0.000	0.000
6	5.000	0.000	10.000
7	10.000	0.000	10.000
8	15.000	0.000	10.000
9	20.000	0.000	10.000
10	13.000	0.000	10.000
11	16.500	0.000	10.000
12	0.000	0.000	25.000
13	5.000	0.000	25.000
14	10.000	0.000	25.000
15	15.000	0.000	25.000
16	20.000	0.000	25.000
17			

Figure 2. 30

21. In this table, change the value of the *X co-ordinate* from 10 to 11.



Node	X ft	Y ft	Z ft
4	15.000	0.000	0.000
5	20.000	0.000	0.000
6	5.000	0.000	10.000
7	10.000	0.000	10.000
8	15.000	0.000	10.000
9	20.000	0.000	10.000
10	13.000	0.000	10.000
11	16.500	0.000	10.000
12	0.000	0.000	25.000
13	5.000	0.000	25.000
14	11.000	0.000	25.000
15	15.000	0.000	25.000
16	20.000	0.000	25.000
17			

Figure 2. 31

22. We also need to change the co-ordinates of Node 15 from (15, 25) to (16.5, 25). However, instead of using the method described in the previous step, let us try a different approach. Let us first select Node 15 by clicking on it. Then, from the *Geometry* menu, select the *Move* option, specify the *X* direction distance as *1.5ft* and click on *OK*.

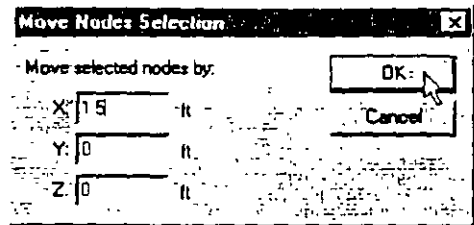
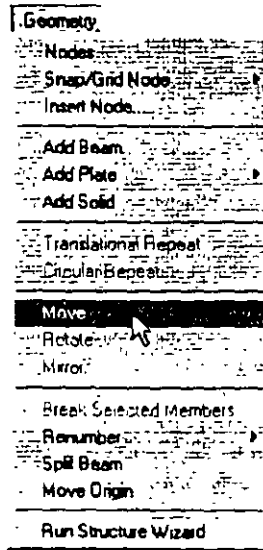


Figure 2. 32

## Creating Member 14

23. We now have to create the X direction beam at  $Z = 28$ ft. To do this, we shall adopt yet another method. Using the *Copy-Paste* facility of the *Edit* menu, let us create the joints at  $(0, 0, 28)$  and  $(20, 0, 28)$  using the joints at  $(0, 0, 0)$  and  $(20, 0, 0)$  as the basis. Then, we shall add a beam between the two new joints. The steps are as follows:

First, let us ensure that we have the *Nodes Cursor* selected. From the *Select* menu, verify that the check mark is against the *Nodes Cursor* option.

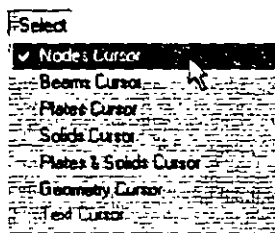


Figure 2. 33

Then, select Node 1 which has the co-ordinates  $(0, 0, 0)$  by clicking on it. It should be highlighted. To copy that Node, type  $\text{Ctrl}+\text{C}$  or select *Copy* from the *Edit* menu. To paste, type  $\text{Ctrl}+\text{V}$  or select *Paste Nodes* from the *Edit* menu.

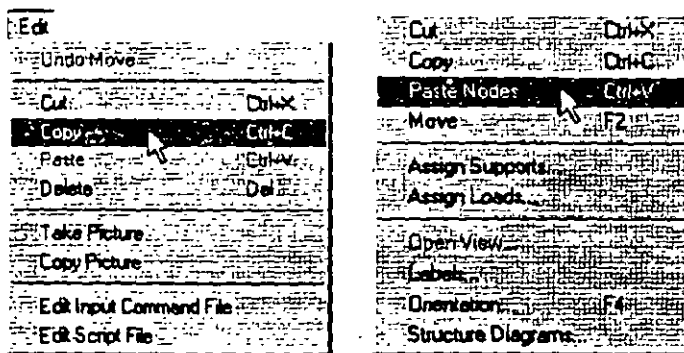


Figure 2. 34

When we select *Paste Nodes*, the following dialog box appears. Specify the *X* and *Y* values as *zero*, and *Z* as *28ft*. Then, click on the *OK* button.

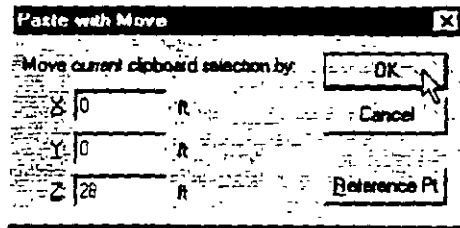


Figure 2. 35

Notice that a new node (no. 17) appears on the screen.

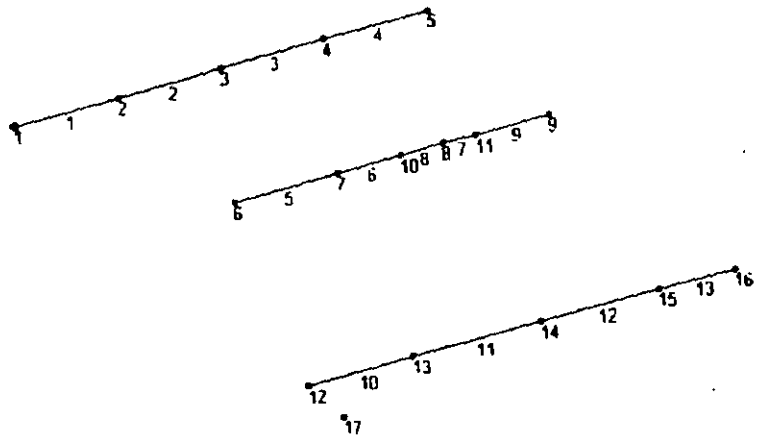


Figure 2. 36



24. In a similar fashion, copy node 5 (at 20, 0, 0) and paste it to create the node at (20, 0, 28).

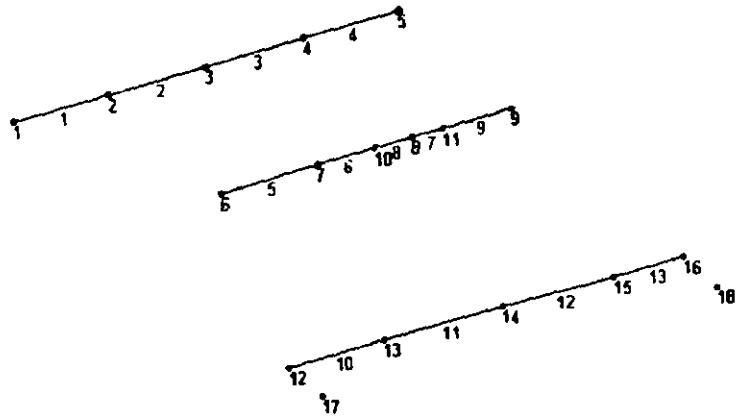


Figure 2.37

25. To add a beam between the two newly created nodes (17 and 18), select the *Add Beam* option from the *Geometry* menu. Then, click on the two nodes in succession and notice that the beam (no. 14) has been created. At this point, switch off the *Add Beam* option.

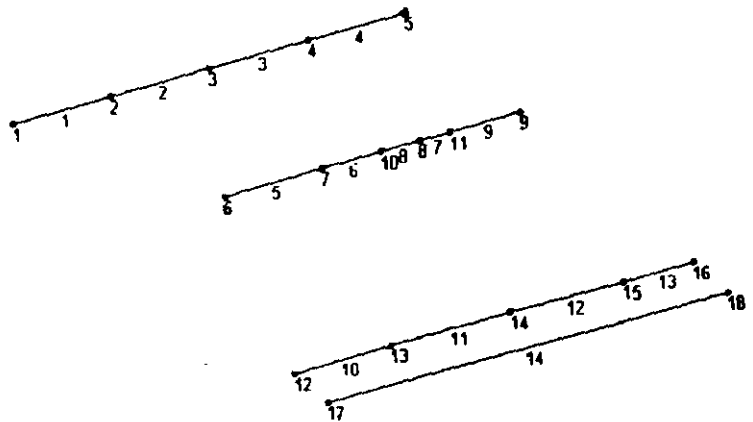


Figure 2.38

## Creating Member 15

26. To create the beam at  $Z = 35$ ft, we shall use the *Copy-Paste* technique of the *Edit* menu, and use Beam no. 14 as the basis. Select beam no. 14. *Copy* and *Paste* it at  $Z = 7$ ft. The value 7 is derived from the fact that  $Z = 35$  is 7ft away from  $Z = 28$ .

As we paste the beam, the following message box will appear. This is only a reminder that we need to subsequently assign the properties to this beam as well. Let us click on the *OK* button.

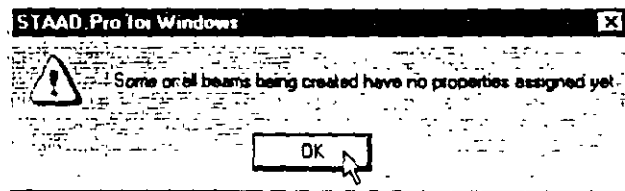


Figure 2. 39

The following figure shows the model with the newly created member 15.

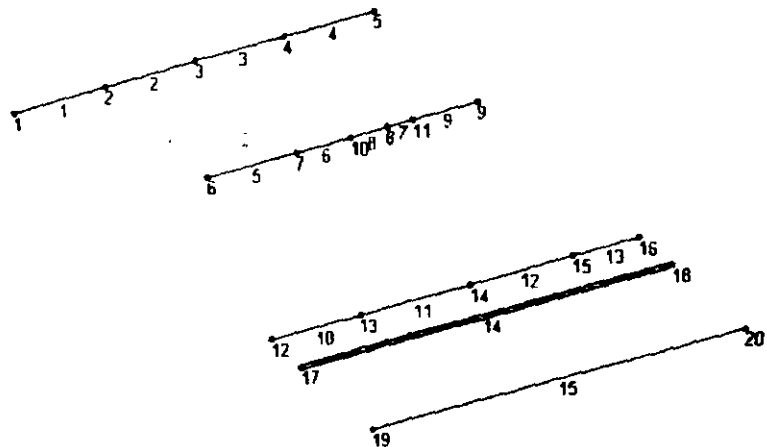



Figure 2. 40

## Creating Members 16 to 28

27. The remainder of the members in the model can be created by adding beams between existing nodes since all the nodes of the structure have already been created. So, let us select the *Add Beams* icon  (If you are unable to locate the icon, choose *Geometry / Add Beam* menu option.) The cursor will change as shown below.



You may choose to turn the *Beam Numbers* off to ease locating the node numbers. *Beam Numbers* can be switched off from *View / Structure Diagrams / Labels* tab and unchecking *Beam Numbers*.

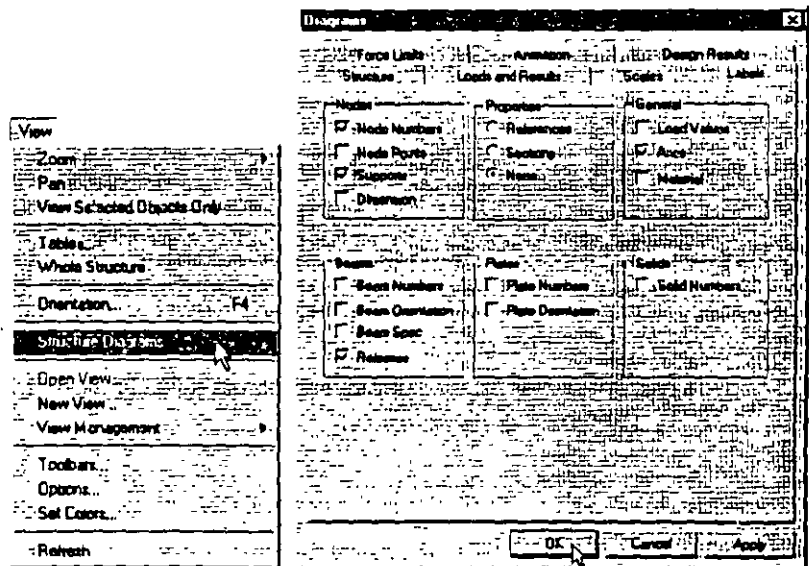


Figure 2. 41

28. Using the cursor, add new beams by clicking between the following pairs of nodes.

To create member #	Add beam between these nodes
16	17 and 19
17	12 and 17
18	1 and 12
19	6 and 13
20	2 and 6
21	10 and 14
22	3 and 7
23	11 and 15
24	4 and 8
25	18 and 20
26	16 and 18
27	9 and 16
28	5 and 9

29. After adding the beams, switch off the *Add Beam* icon to stop adding any more beams.

The structure will now look as shown below:

Isometric View

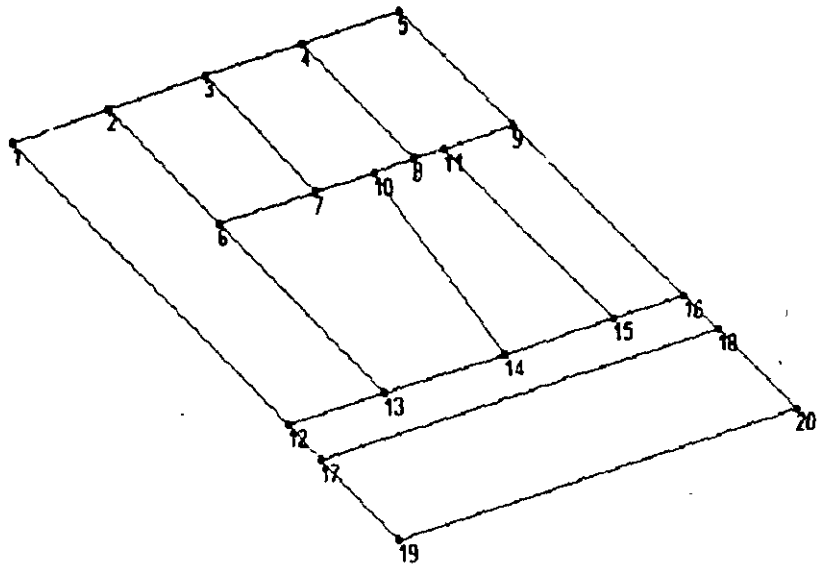


Figure 2. 42

Plan View

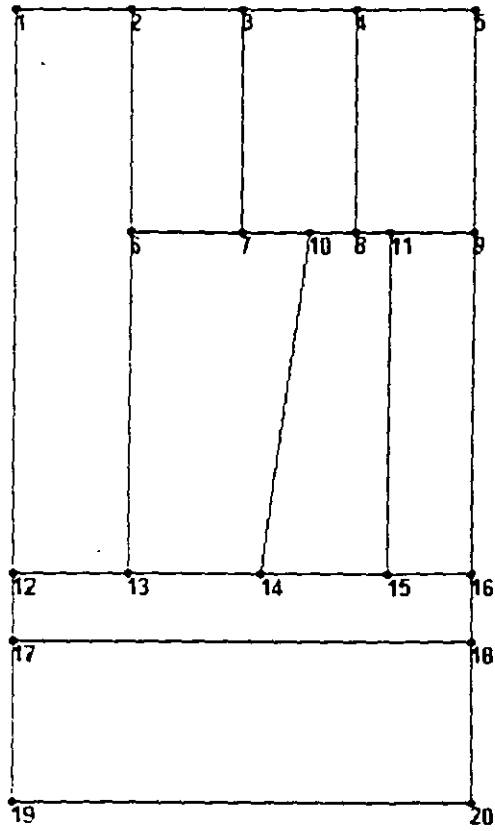


Figure 2. 43

## Assigning Member Properties

30. The next step is to define properties for the members. To do this, select *General / Property* Page from the left side of the screen. Then, click on the *Database* button in the *Properties* dialog box as shown below.

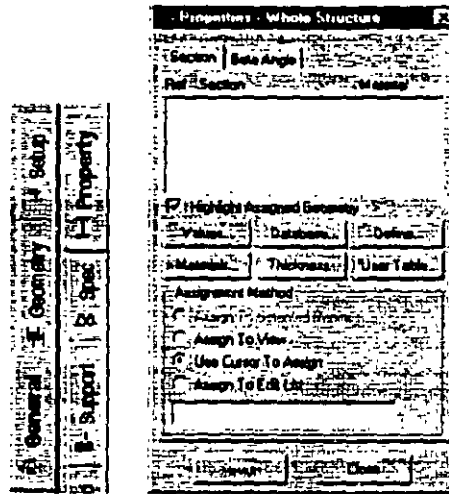


Figure 2. 44

31. In the *Select Country* dialog box that appears, choose the country name whose steel table you want to use, in our case, *American*. Then, click on *OK*.

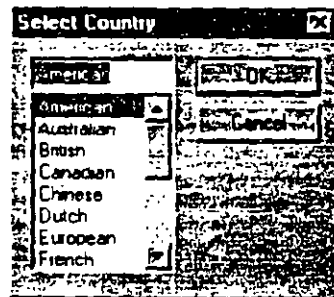


Figure 2. 45

32. In the *American Steel Table* dialog box, select the *W Shape* tab. Notice that the field called *Material* is presently on the "checked" mode. If we keep it that way, the material properties of steel (E, Poisson, Density, Alpha, etc.) will be assigned along with the cross-section name. The material property values so assigned will be the program defaults. We do not want default values, instead we will assign our own values later on. Consequently, let us uncheck the *Material* box. Choose *W12X26* as the beam size, *ST* as the section type and click on the *Add* button as shown in Figure 2.39. Detailed explanation of the terms such as ST, T, CM, TC, BC, etc. is available in Section 5 of the STAAD Technical Reference Manual.

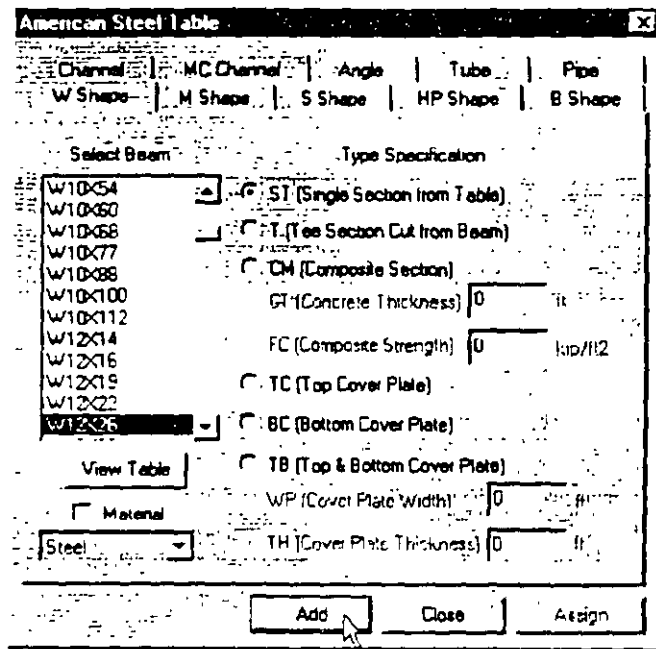


Figure 2. 46



33. Since the selected cross section has to be assigned to all the members in the structure, the simplest method to do that will be to set the assignment method as *Assign to View*. So, click on the *Assign to View* button in the *Properties* dialog box followed by the *Assign* button.

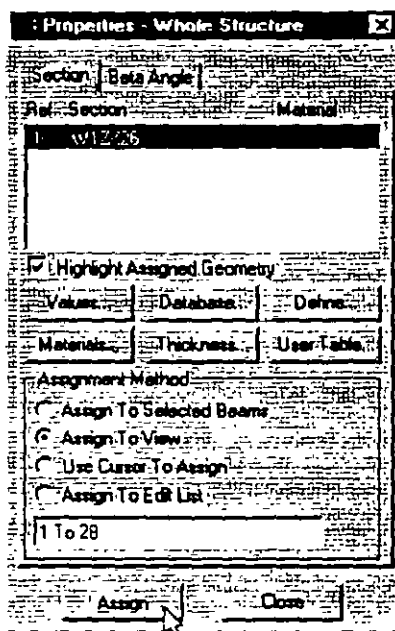


Figure 2. 47

A message box (shown below) asks us to re-confirm that we do indeed want to assign this property to all the members in the model. Let us click on *OK*.

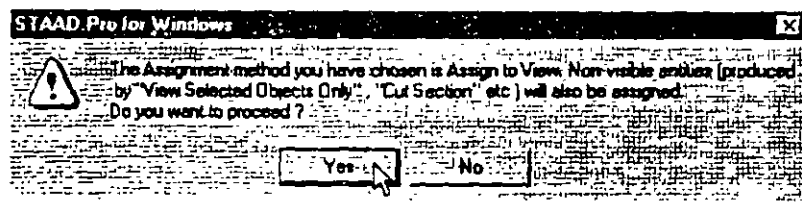


Figure 2. 48

After assigning the property, let us once again switch on the Beam Numbers (go to *View / Structure Diagrams - Labels - Beam Numbers*). The structure will now look as shown below.

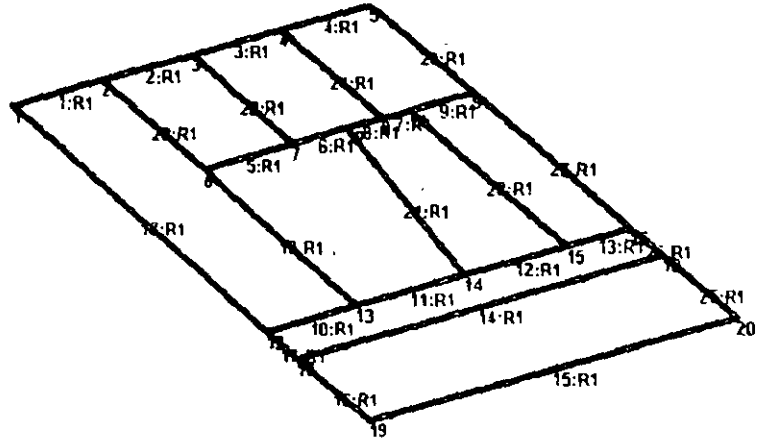


Figure 2. 49

Let us *Close* the *Properties* dialog box as shown below. Also, click anywhere in the drawing area to unhighlight the members.

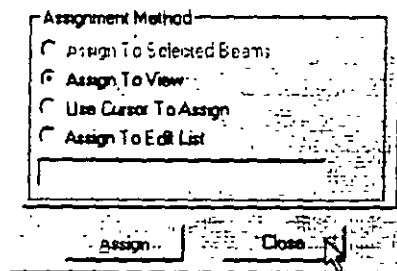


Figure 2. 50

## Assigning Member Releases

34. To assign member releases, first, go to *General / Spec* Page from the left side of the screen. Then, click on the *Beam* button in the *Specifications* table located in the Data Area.

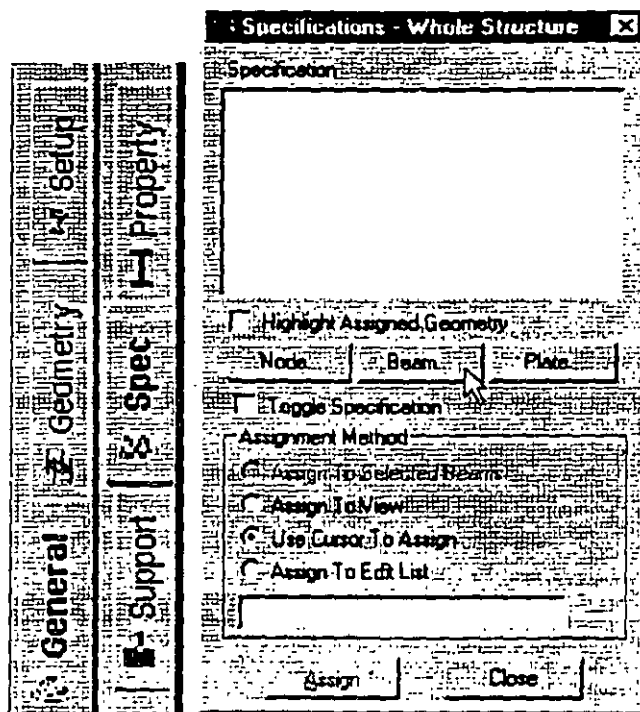


Figure 2. 51

35. In the *Beam Specs* dialog box that opens, select the *Release* tab which also happens to be the default. We want to apply the release at the start node, and hence it is convenient that 'Start' is the default. Check *MZ* under the *Release* option and click on the *Add* button.

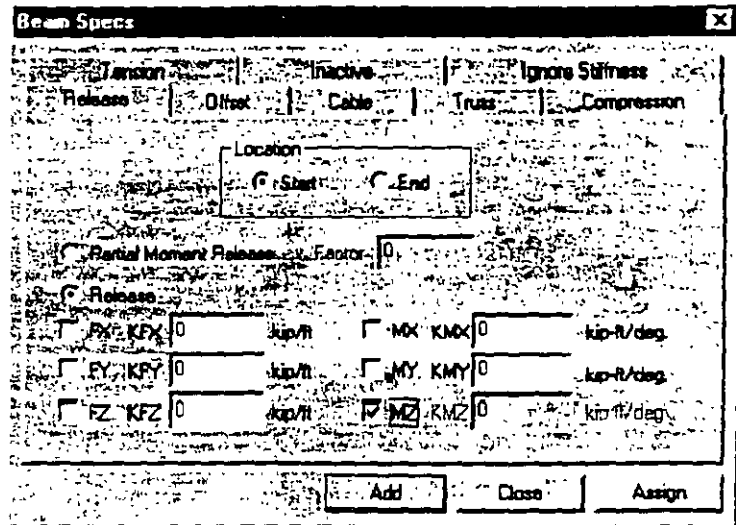


Figure 2. 52

36. Now select the members listed below that are to be released.

1, 5, 10, 14, 15, 18, 17, 28, 26, 20 to 24

One way to select these members is to go to *Select / By List / Beams* menu option. In the *Select Beams* dialog box, type the beam numbers in the *Enter list* box, and click on *OK* as shown below

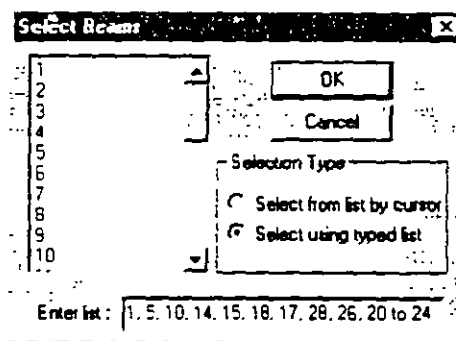


Figure 2. 53

Notice that as we select the members, the *Assignment Method* automatically sets to *Assign to Selected Beams*.

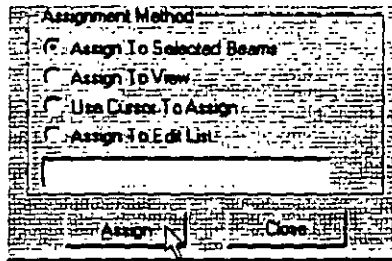


Figure 2. 54

Then, click on the *Assign* button in the *Specifications* dialog box. A message box (shown below) asks us to re-confirm that we do indeed want to assign this attribute to the selected members in the model. Let us click on *OK*.

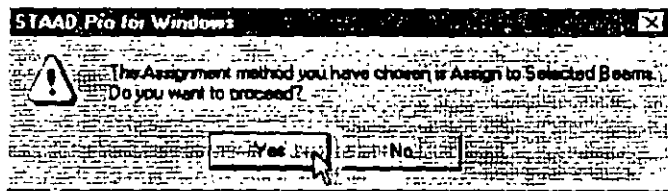


Figure 2. 55

After the releases have been assigned at the start, let us click anywhere in the drawing area to unhighlight all the members. The structure will look as follows:

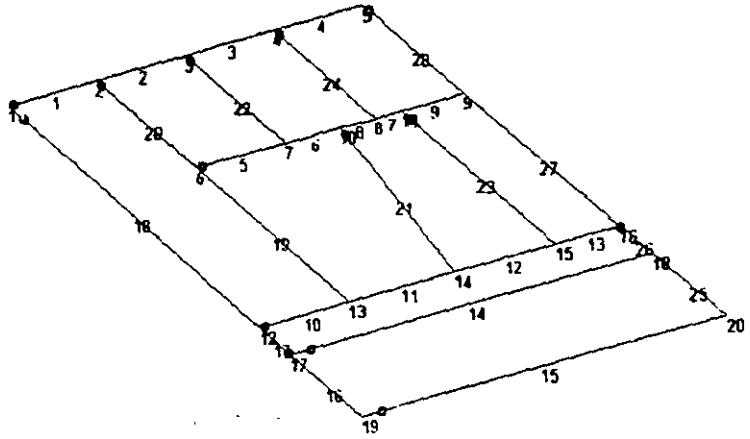


Figure 2. 56

37. To apply the releases at the beam ends, repeat the above procedure by clicking on the *Beam* button in the *Specifications* dialog box. Then, click on the *End* button, check *MZ* under the *Release* option and click on the *Add* button.

Assign this attribute to the following members.

4, 9, 13, 14, 15, 18, 16, 27, 25, 19, 21 to 24

After the releases have been assigned at the end, once again unhighlight the members by clicking anywhere in the drawing area. The structure will now look as shown below:

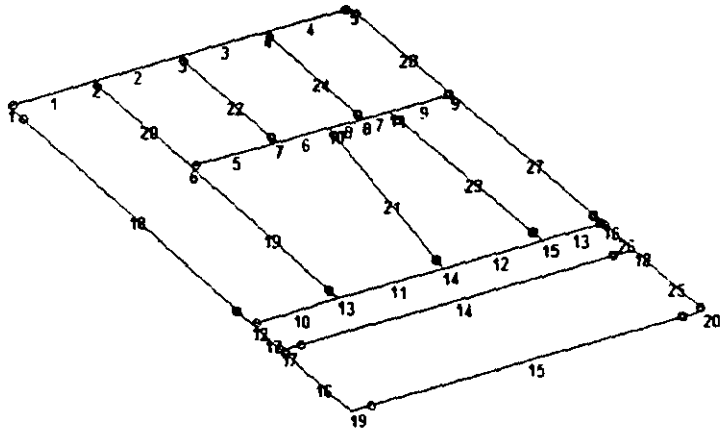


Figure 2. 57

## Specifying Material Constants

38. The Commands we wish to generate are:

```

CONSTANTS
E 4176E3 ALL
POISSON STEEL ALL
  
```

To do this, go to *Commands / Material Constants / Elasticity* option from the top menu bar as shown below.

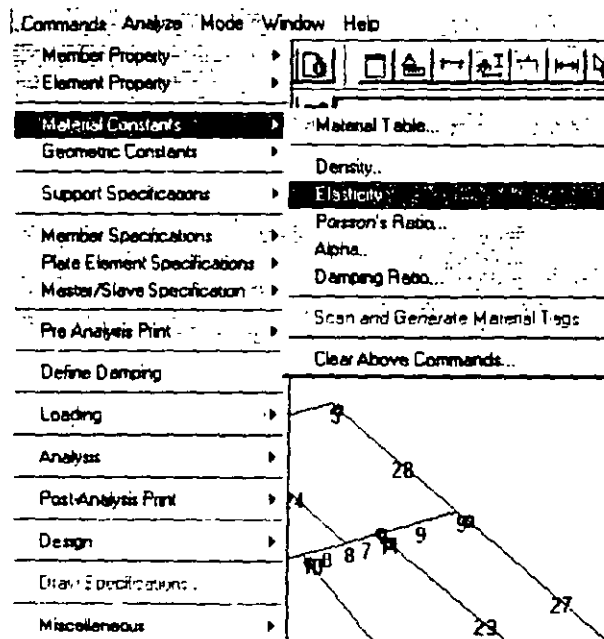


Figure 2. 58



39. In the *Material Constant* dialog box that appears, enter *4176E3* in the *Enter Value* box. Since the value has to be assigned to all the members of the structure, setting the assignment method *To View* allows us to achieve this easily. Then, click on *OK*.

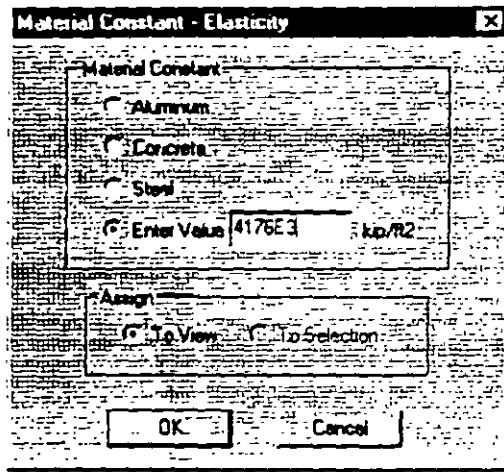


Figure 2. 59

40. In a similar fashion, set the *Poisson's Ratio* to the *Material Constant* for *Steel* and assign to all members in the view.

## Assigning Supports

41. The commands we wish to generate are:

1 5 12 16 19 20 FIXED

To do this, select the *General / Support* Page from the left side of the screen. In the *Supports* dialog box, click on the *Add* button.

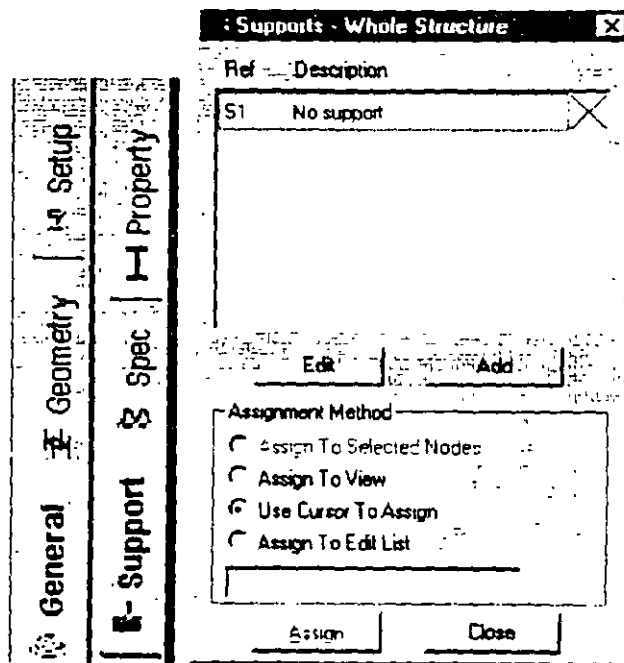


Figure 2. 60

42. In the *Create Support* dialog box that opens, select the *Fixed* tab (which also happens to be the default tab that comes up), and then click on the *Create* button. This creates a FIXED type of support where all 6 degrees of freedom are restrained.

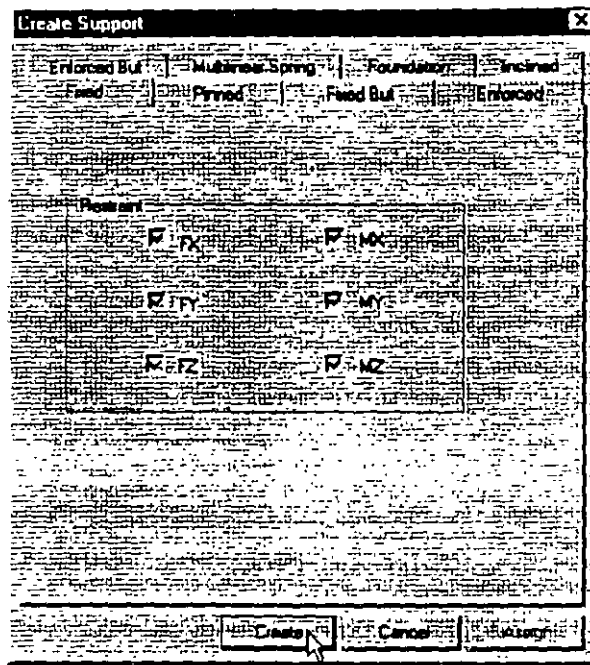


Figure 2. 61

43. To assign the support, first select the Support 2 specification in the *Supports* dialog box.

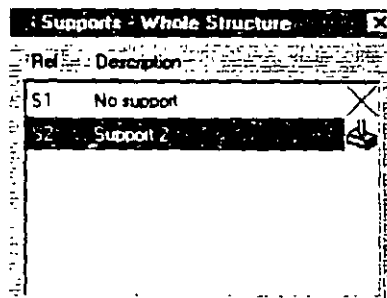


Figure 2. 62

Then, select the following nodes:

1, 5, 12, 16, 19, 20

To select these nodes, go to *Select / By List / Nodes* menu option. In the *Select Nodes* dialog box, type the node numbers in the *Enter list* box.

Notice that as we select the nodes, the *Assignment Method* automatically sets to *Assign to Selected Nodes*. Then, click on the *Assign* button in the *Specifications* dialog box.

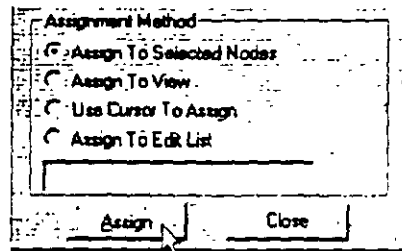


Figure 2. 63

A message box (shown below) asks us to re-confirm that we do indeed want to assign this support to the selected nodes. Let us click on *OK*.

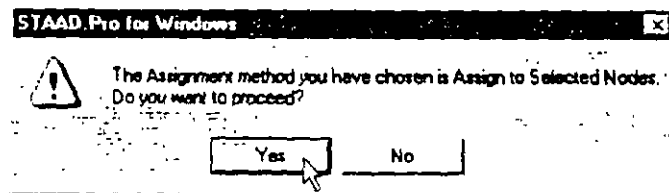


Figure 2. 64

After the supports are assigned, the structure will look as shown below:

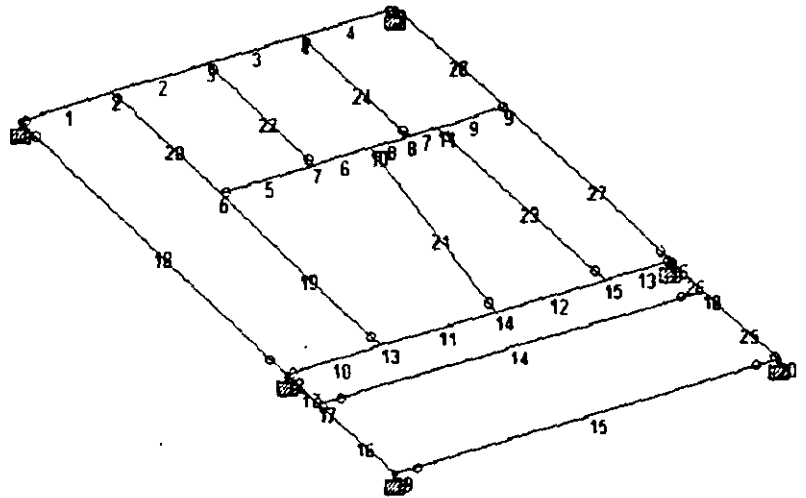


Figure 2. 65

## Specifying Loads

44. Load assignments are done from the *General / Load* Page as shown below.

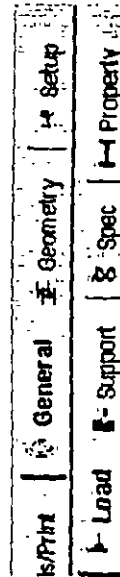


Figure 2. 66

45. For load case 1, we wish to generate the following load data:

**LOADING 1 300 POUNDS PER SFT DL + LL  
1 to 28 ALOAD -0.30**

In the *Set Active Primary Load Case* dialog box that opens, enter 300 POUNDS PER SFT DL - LL as the title for *Load Case 1* and click on *OK*.

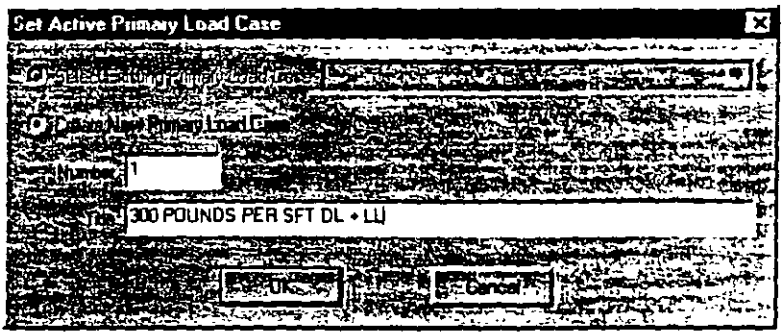


Figure 2. 67

46. In the *Loads* dialog box that appears, click on the *Member* button.

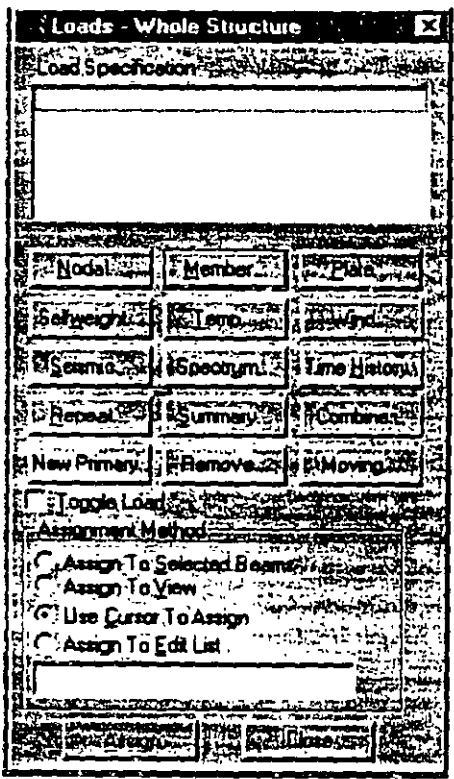


Figure 2. 68

47. In the *Beam Loads* dialog box that opens, select the *Area* tab. Enter -0.30 as the *Force* and click on the *Add* button.

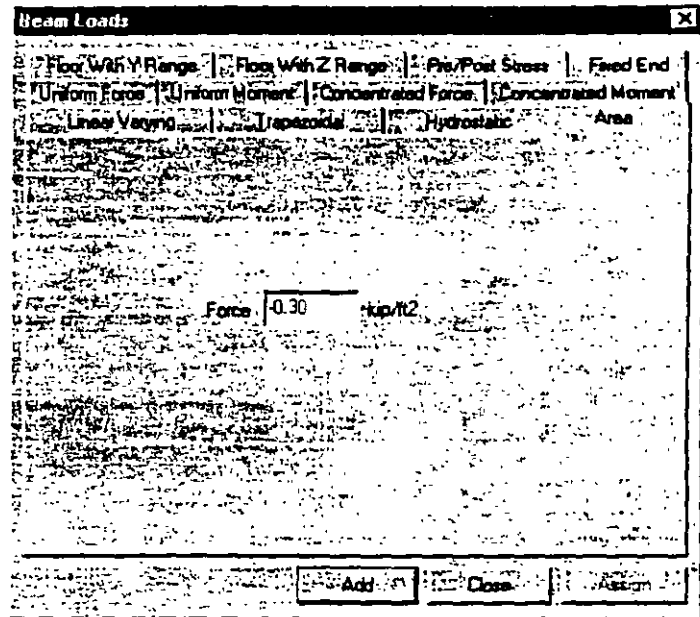


Figure 2. 69

As we click on the *Add* button, the following message box appears. In the case of loads such as joint and member loads, the magnitude and direction of the load at the applicable joints and members is directly known from the input. However, the Area load is a different sort of load where a load intensity on the given area has to be converted to joint and member loads. The calculations required to perform this conversion are done only during the analysis. Consequently, the loads generated from the Area load command can be viewed only after the analysis is completed.

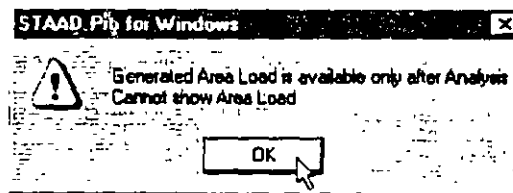


Figure 2. 70



48. Since this load is to be applied on all the beams of the model, set the *Assignment Method* to *Assign to View* in the *Loads* dialog box. Then, click on the *Assign* button followed by *Close*.

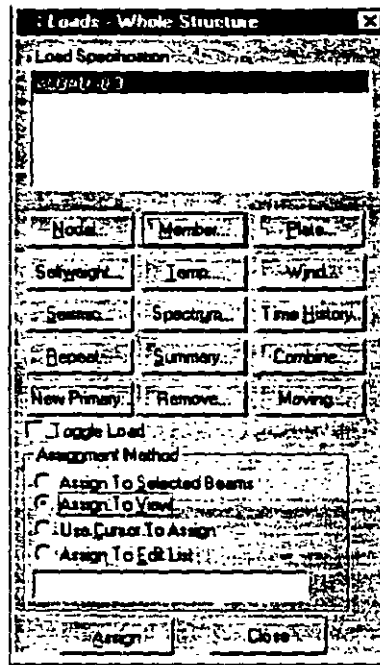


Figure 2. 71

A message box (shown below) asks us to re-confirm that we do indeed want to assign this load to all the members in the model. Let us click on *OK*.

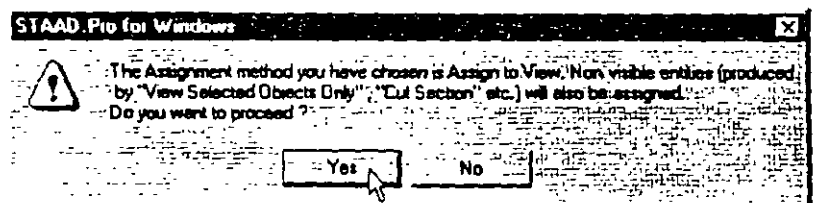


Figure 2. 72

## Specifying The Analysis Command

49. The next step is to assign the commands to perform the analysis  
We wish to generate the command:

### PERFORM ANALYSIS PRINT LOAD DATA

To do this, go to *Analysis/Print* Page from the left side of the screen. Then, click on the *Analysis* sub-page from the second row of pages as shown below.

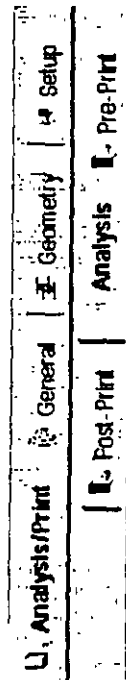


Figure 2. 73

50. Click on the *Define Commands* button in the Data Area on the right hand side of the screen.

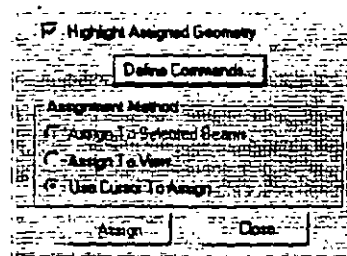


Figure 2. 74

51. In the *Analysis/Print Commands* dialog box that appears, select the *Perform Analysis* tab. Click on the *Load Data* option followed by the *Add* button and the *Close* button.

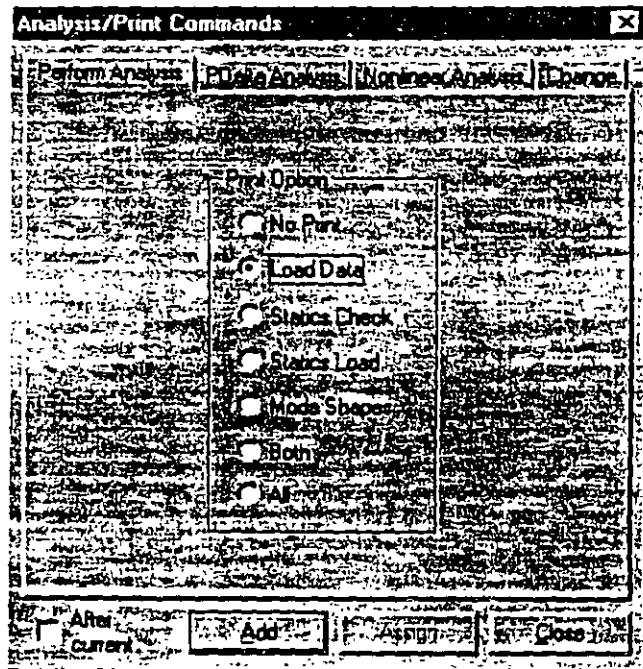


Figure 2. 75

## Specifying Steel Design Parameters

52. The next step is to specify steel design parameters. To do this, click on the *Design / Steel* Page from the left side of the screen. Make sure that under the *Current Code* selections on the top right hand side, *AISC* is selected. Then, click on the *Define Parameters* button in the *Steel Design* dialog box.

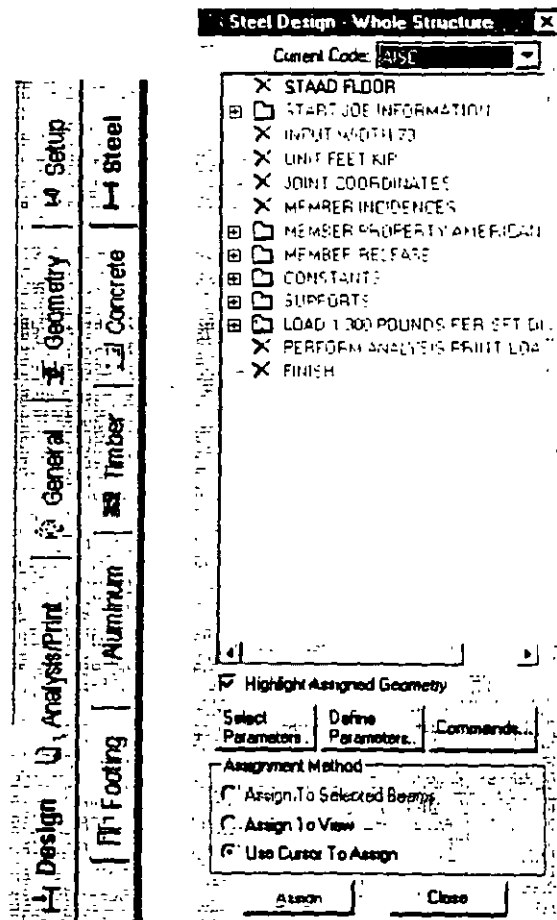


Figure 2. 76

The commands we wish to generate are:

BEAM 1 ALL  
D MAX 2.0 ALL  
D MIN 1.0 ALL  
UNT 1.0 ALL  
UNB 1.0 ALL

53. In the *Design Parameters* dialog box that opens, select the *Beam* tab. Then, define the *Beam Parameter* as 1 and click on the *Add* button.

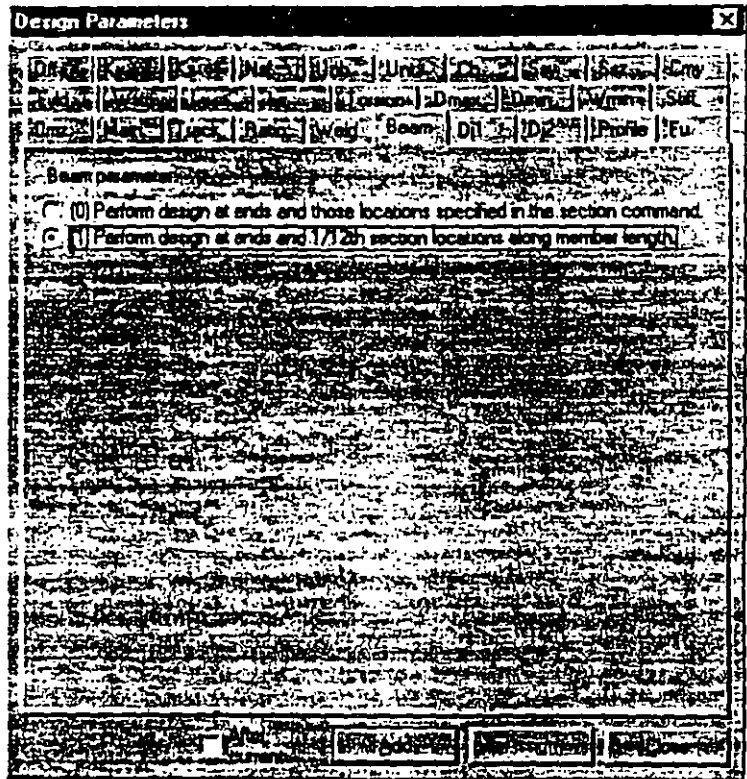


Figure 2. 77

54. In a similar fashion select the tabs *Dmax*, *Dmin*, *Unt*, and *Unb*. Then, enter the following values and click on the *Add* button.

Parameter Name	Value
DMAX	2.0
DMIN	1.0
UNT	1.0
UNB	1.0

55. After all the values have been added, click on the *Close* button in the *Design Parameters* dialog box.
56. Since each of these parameters has to be assigned to ALL the members in the view, do the following. Select each parameter, click on the *Assign to View* button, followed by the *Assign* button in the *Design Parameters* dialog box.

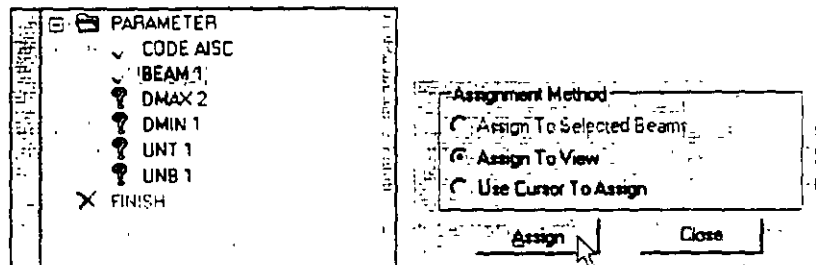


Figure 2. 78

As we click on the *Assign* button, a message box (shown below) asks us to re-confirm that we do indeed want to assign this parameter to all the members in the model. Let us click on *OK*.

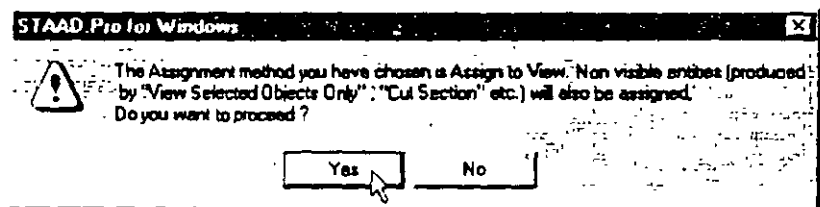
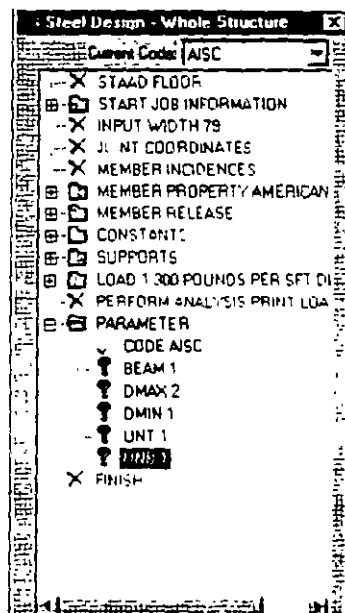


Figure 2. 79

Notice that before assigning the parameters, each of them will be preceded by  $\uparrow$ , whereas after assigning the parameters, they will be preceded by  $\downarrow$ .

Before



After

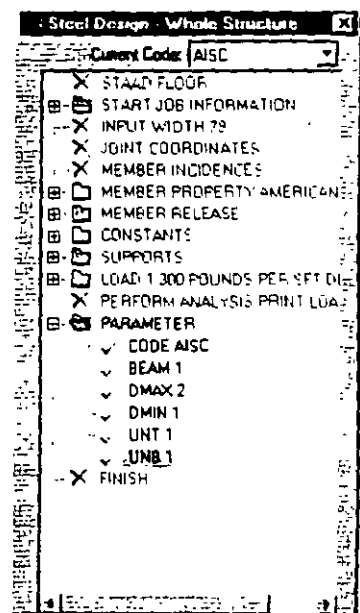


Figure 2. 80

## Specifying The SELECT MEMBER Command

57. Click on the *Commands* button in the *Steel Design* dialog box.

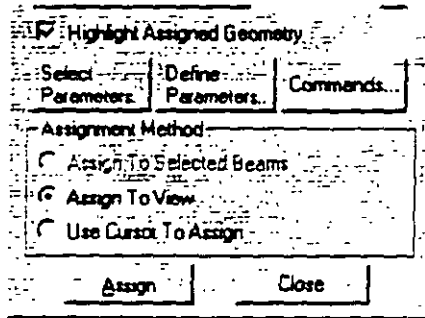


Figure 2. 81

58. In the *Design Commands* dialog box that opens, click on the *Select* tab followed by the *Add* and the *Close* buttons.

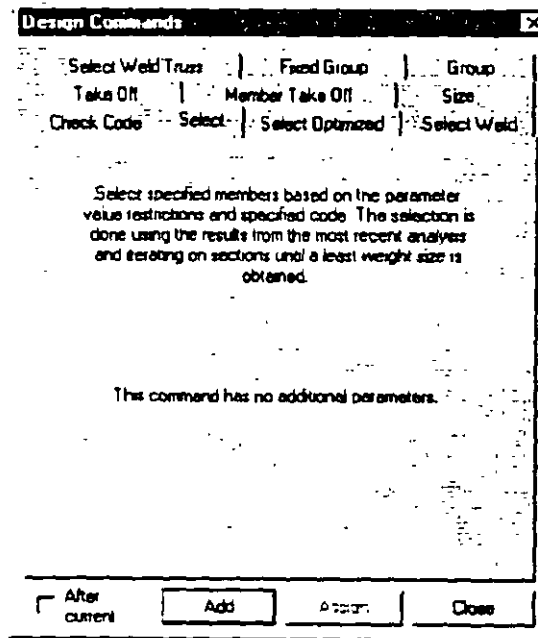


Figure 2. 82



59. Next, make sure that the **SELECT** parameter is selected in the *Steel Design* dialog box.

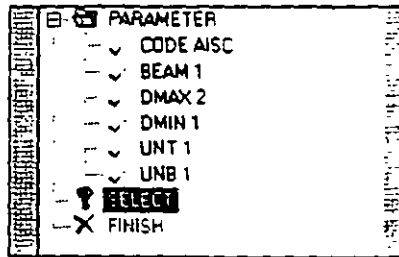


Figure 2. 83

Then, select the following members.

2, 6, 1, 14, 15, 16, 18, 19, 21, 23, 24, 27

By now, you should be familiar with the process of selecting members. In case you have forgotten, you may:

- Choose the *Beams Cursor* from the *Select* menu, and then click on those members in the drawing while keeping the 'Ctrl' key pressed.

or

- Choose *By List / Beams* from the *Select* menu, and type the member numbers in the *Enter list* box, followed by *OK*.

Notice that as we select the members, the *Assignment Method* automatically sets to *Assign To Selected Beams*

60. After the members are selected, click on the *Assign* button located in the *Steel Design* dialog box.

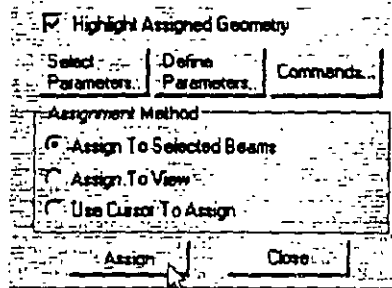


Figure 2. 84

A message box (shown below) asks us to re-confirm that we do indeed want to assign this command to the selected members in the model. Let us click on *OK*.

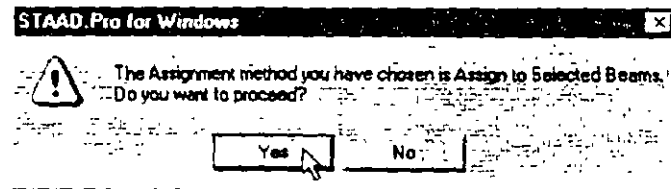


Figure 2. 85

The structure will now look as shown below.

Isometric View

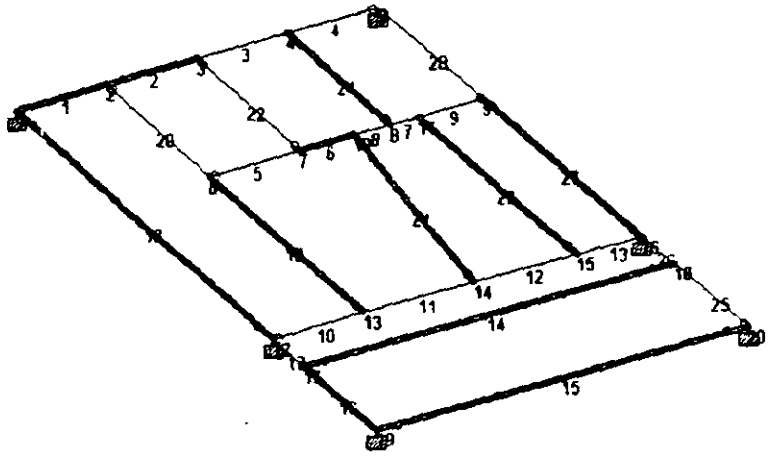


Figure 2. 86

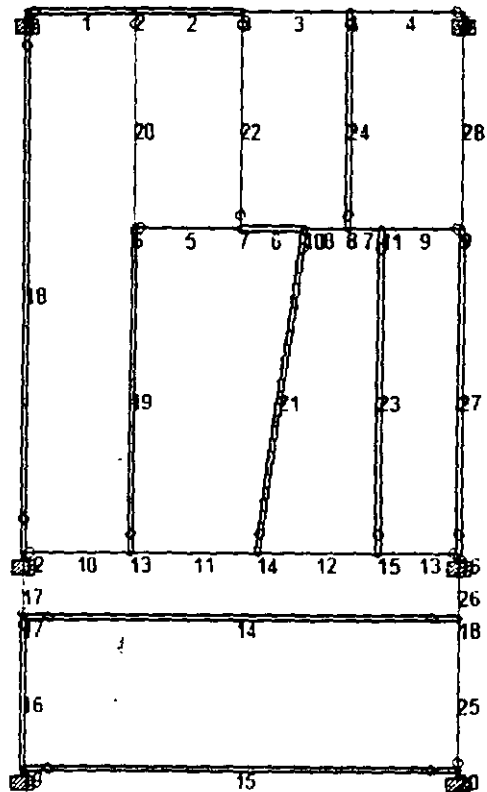
Plan View

Figure 2. 87

This concludes the assigning of all the data to the structure. From the *File* menu, select *Save*, and provide a file name, if you haven't already done so.



**FACULTAD DE INGENIERÍA UNAM  
DIVISIÓN DE EDUCACIÓN CONTINUA**



**CA 354  
STAAD PRO PARA ANÁLISIS Y DISEÑO  
ESTRUCTURAL**

**TEMA:**

**EJEMPLOS DEL PROBLEMA No. 5**

**COORDINADOR: ING. JOSÉ LUIS ESQUIVEL ÁVILA**

**DEL 22 AL 29 DE NOVIEMBRE DE 2004**

**PALACIO DE MINERÍA**



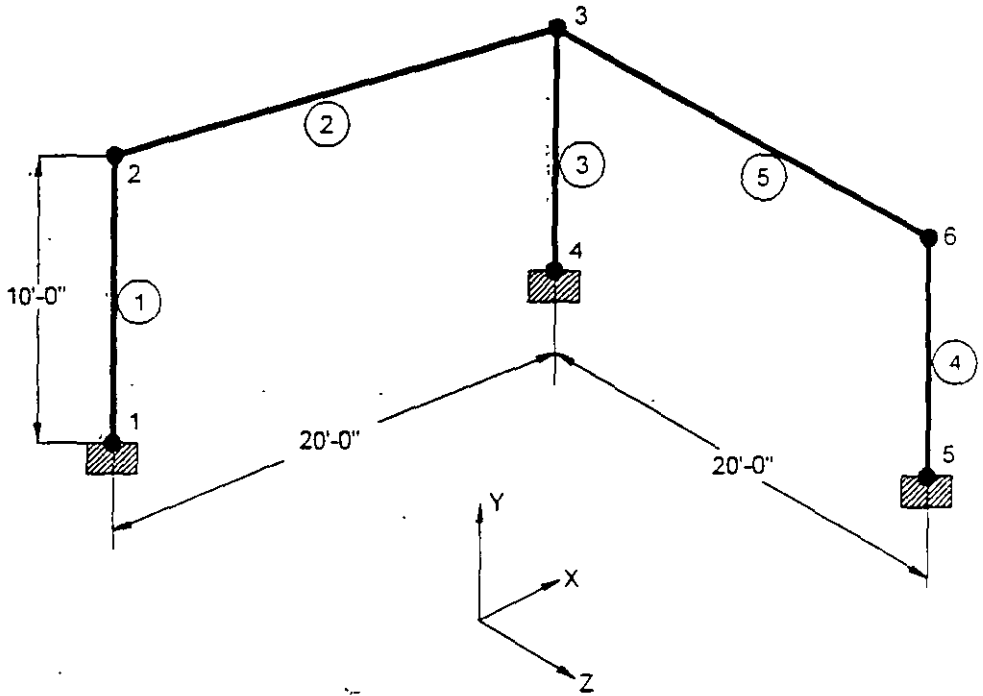
DIVISIÓN DE  
EDUCACIÓN  
CONTINUA



Programa 2004

## Example Problem No. 5

This example demonstrates the application of support displacement load (commonly known as sinking support) on a space frame structure.



1. Select the *STAAD.Pro* icon from the STAAD.Pro 2001 program group.

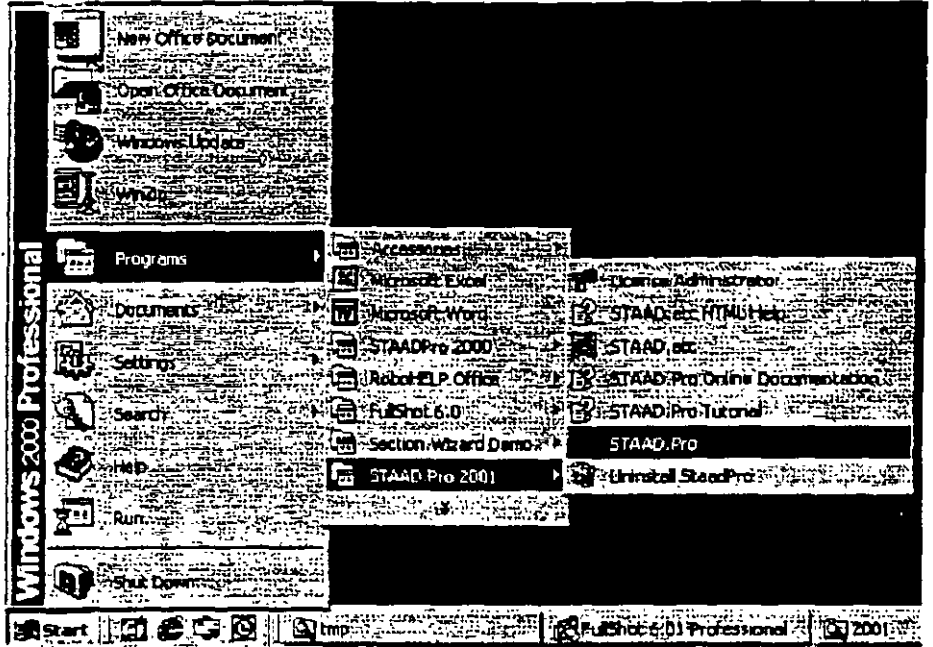


Figure 5. 1

The STAAD.Pro Graphical Environment will be invoked.

2. The units in which we wish to create this model are the *English* units. (feet, kip, etc.) The default unit system setting is whatever we chose during the installation of the program. If you had chosen *Metric* at the time of installation, you may want to change it to *English*. To do so, click on the *File / Configure* menu option (see Figure 5.2) and choose the appropriate one (*English* for our case). Then, click on the *Accept* button.

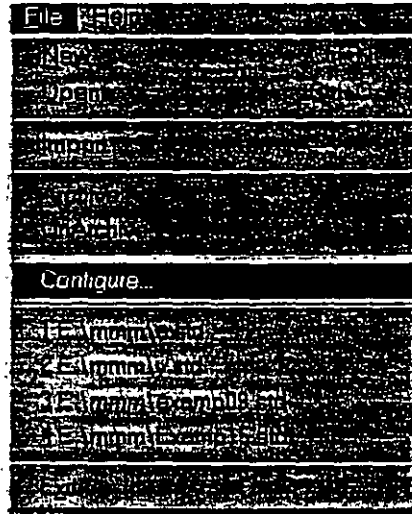


Figure 5.2

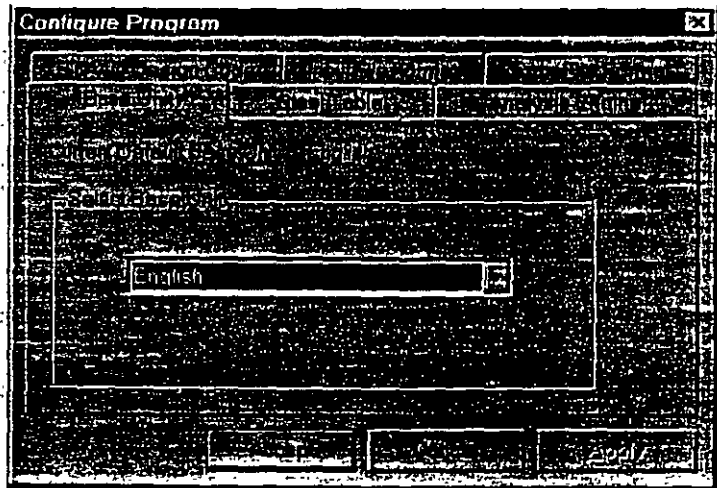


Figure 5.3



- To create a new structure, click on the *File / New* option in the STAAD.Pro screen that opens (as shown in Figure 5.4).

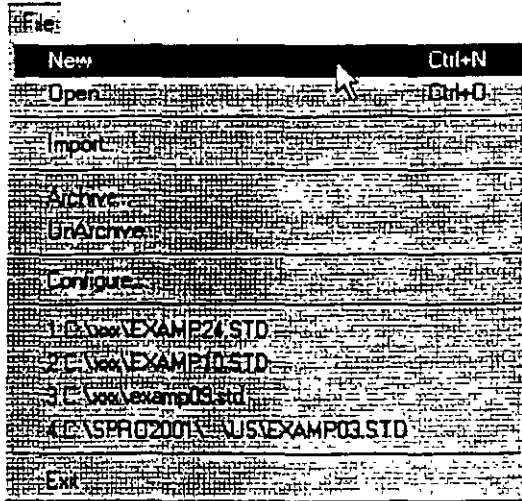


Figure 5. 4

- In the *New File Setup* dialog box, choose *Space* as the *Structure Type* and specify an optional *Title* (EXAMPLE PROBLEM NO. 5). Then click on the *Next* button as shown in Figure 5.5.

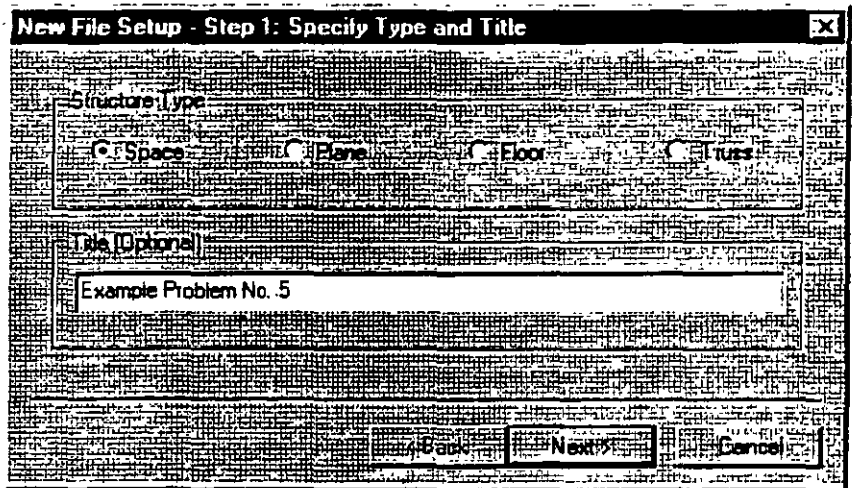


Figure 5. 5

5. The next dialog box that comes up prompts us to select the length and force units in which we wish to start working in. So, specify the *Length Units* as *Foot*, the *Force Units* as *KiloPound* and click on the *Next* button as shown in Figure 5.6. Please note that the input units may be changed subsequently at any stage of building of the model.

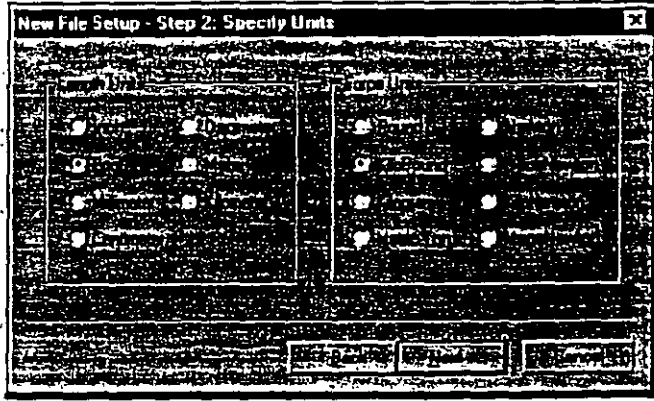


Figure 5. 6

6. This dialog box confirms the information of our previous selections. Press the *Finish* button. (see Figure 5.7)

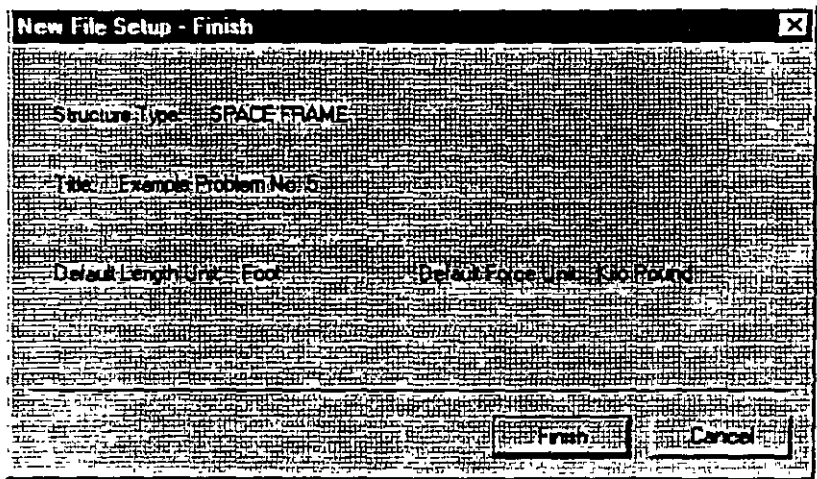


Figure 5. 7

Once we press the *Finish* button, the STAAD.Pro main window appears on the screen.

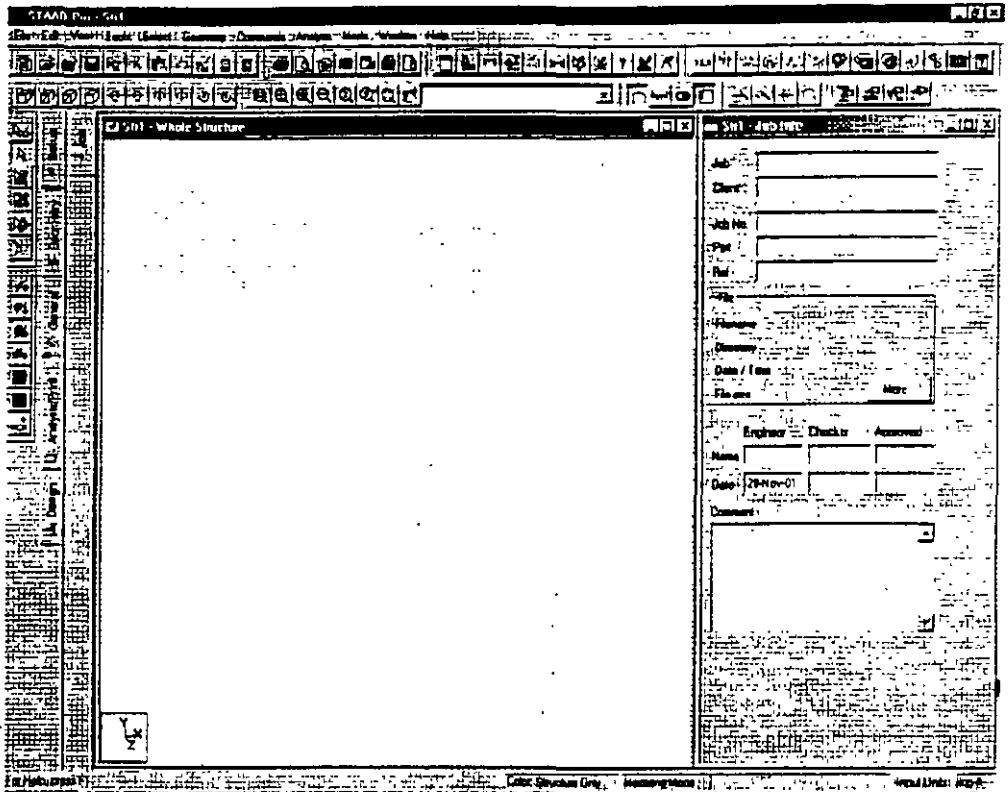


Figure 5. 8

## Creating Nodes 1 to 4 And Beams 1 to 3

7. Select *Geometry / Beam Page* from the left side of the screen. In the *Snap Node/Beam* dialog box that appears in the Data Area (on the right side of the screen), choose *X-Y* as the *Plane* and in the *Construction Lines* group, set *X* to *20* and *Y* to *10* with a spacing of *1ft.* (see Figure 5.9) This 20X10 grid too is only a starting grid setting to enable us to start drawing the structure, it does not restrict our model to those limits as we will see later.

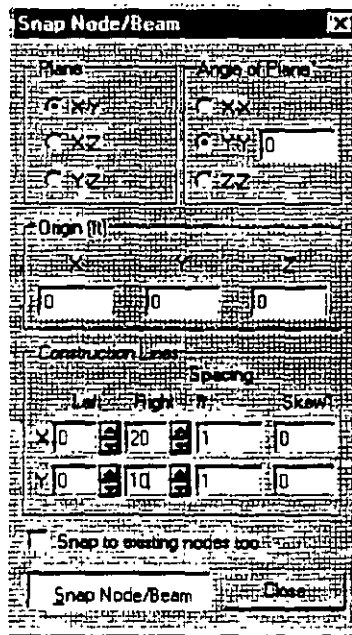
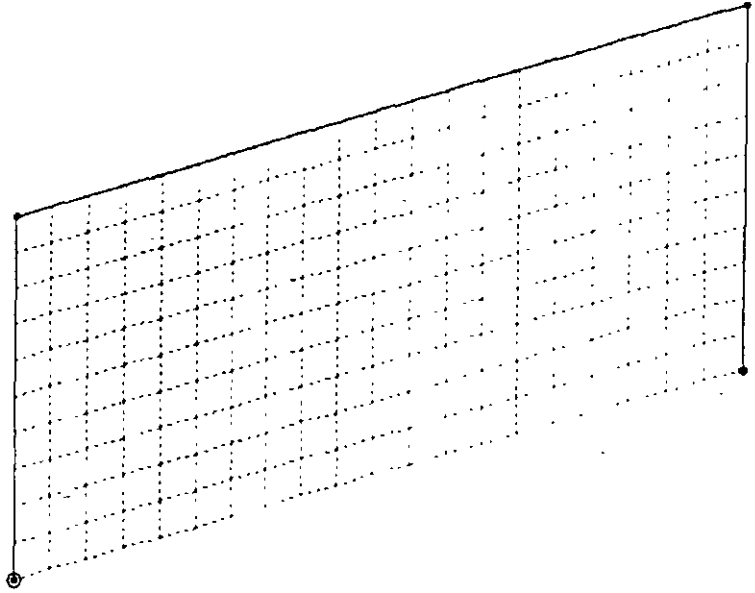


Figure 5. 9

8. With the help of the mouse, click at the origin (0, 0) to create the first node. In a similar fashion, click on the following points to create nodes and automatically join successive nodes by beam members. (see Figure 5.10)

(0, 10), (20, 10), (20, 0)

The exact location of the mouse arrow can be monitored on the status bar located at the bottom of the window where the X, Y, and Z coordinates of the current cursor position are continuously updated.



**Figure 5. 10**

9. After having created these three beams and four nodes, let us close the *Snap Node/Beam* dialog box.

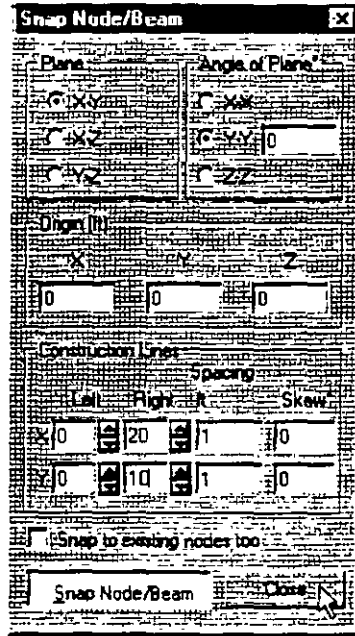


Figure 5. 11

## Switching On Node And Beam Labels

10. In order to display the *node* and *beam* numbers, right click anywhere within the drawing area. In the pop-up menu that appears, choose *Labels* as shown in Figure 5.12.

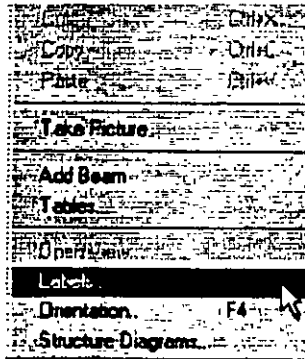


Figure 5.12

Alternatively, one may access this option by selecting the *View* menu from the top menu bar followed by *Structure Diagrams*, and the *Labels* tab of the dialog box that opens. (see Figure 5.13)

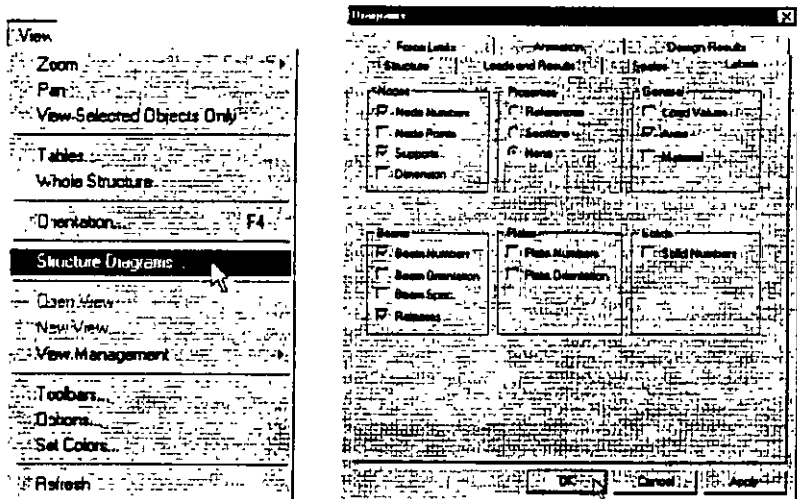


Figure 5.13

11. In the *Diagrams* dialog box that appears, turn the *Node Numbers* and *Beam Numbers* on and then click on *OK*.

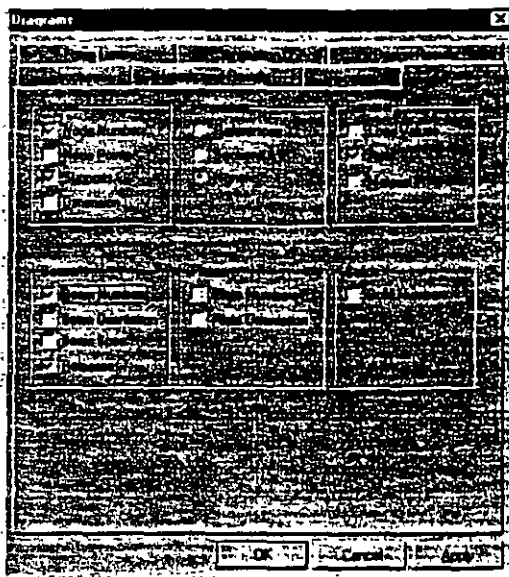


Figure 5. 14

The nodes and beams are now labeled on the drawing as shown below.

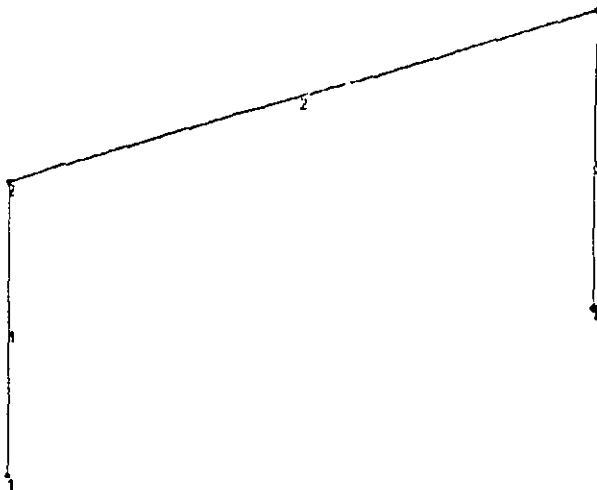



Figure 5. 15



## Creating Members 4 and 5

12. Looking at the diagram of our structure shown in the title page of this example, it can be seen that members 4 and 5 can be easily generated if we could first create a copy of members 1 and 2 and then rotate those copied units about a vertical line passing through the point (20, 0, 0) by 90 degrees. Fortunately, such a facility does exist which can be executed in a single step. It is called "*Circular Repeat*" and is available under the *Geometry* menu.
13. First, select Members 1 and 2 using the *Beams Cursor* . Then, go to the *Geometry / Circular Repeat* menu option as shown below.

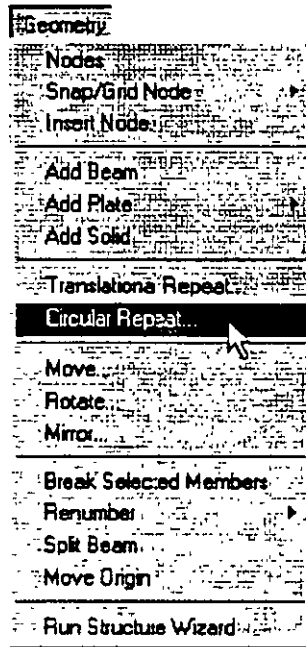


Figure 5. 16

14. In the *3D Circular* dialog box that comes up, specify the *Axis of Rotation* as *Y*, *Total Angle* as *90 degrees*, *No. of Steps* as *1* and passing through *Node 3*. Instead of specifying as passing through *Node 3*, one may also specify the *X and Z co-ordinates* as *20* and *0* respectively. Leave the *Link Steps* box unchecked and click on the *OK* button. (see figure below)

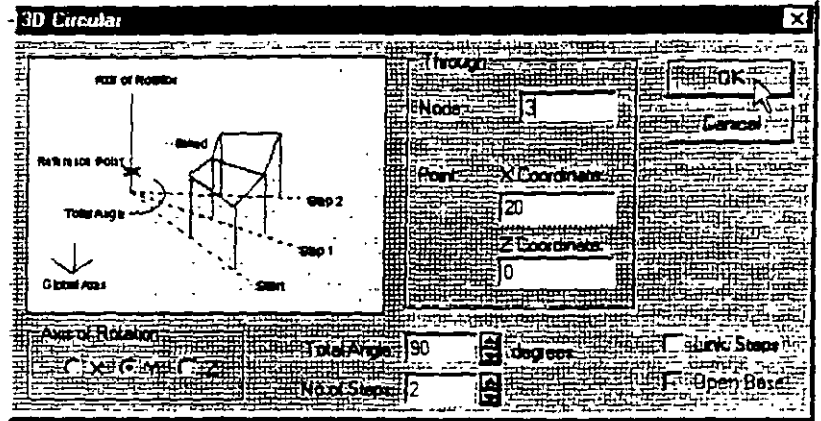


Figure 5.17

After completing the circular repeat procedure, the model will look as shown below.

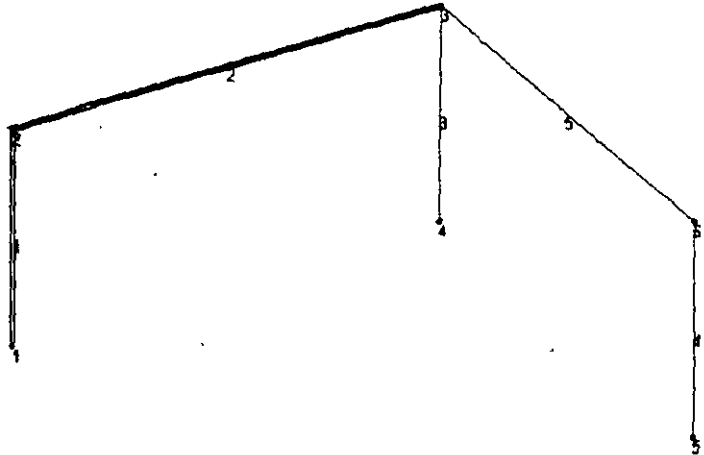



Figure 5.18

## Changing The Input Units Of Length

15. For specifying member property values, as a matter of convenience, it is simpler if our *length units* are *inches* instead of *feet*. To change the length units from feet to inch, either click on the *Input Units* icon  or select the *Tools / Set Current Unit* menu option as shown below.

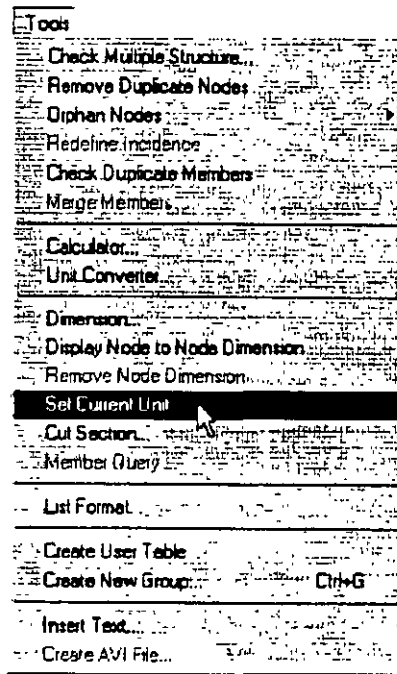


Figure 5. 19

16. In either case, the following dialog box comes up. Set the *Length Units* to *Inch* and click on the *OK* button.

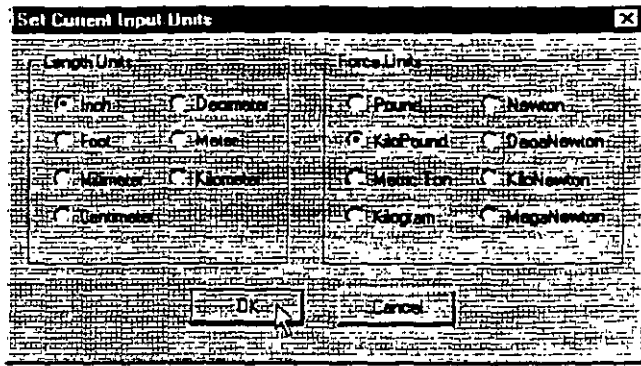


Figure 5.20

## Assigning Member Properties

17. The next step is to define properties for the members. The commands we wish to generate are:

**MEMB PROP**  
**1 TO 5 PRIS AX 10. IZ 300. IY 300. IX 10.**

To do this, select the *General / Property* Page from the left side of the screen. The property type we wish to assign is called **PRISMATIC**, and is available under the *Define* button in the *Properties* dialog box as shown below.

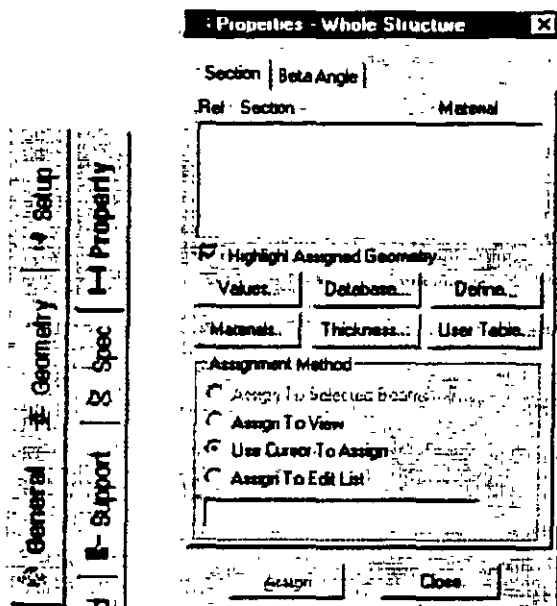


Figure 5. 21

18. In the *Prismatic Property* dialog box that comes up, select the *General* tab. Notice that the field called *Material* is presently on the checked mode. If we keep it that way, the material properties of steel (E, Poisson, Density, Alpha, etc.) will be assigned along with the cross-section name. The material property values so assigned will be the program defaults. We do not want default values, instead we will assign our own values later on. Consequently, let us uncheck the *Material* box. Then, enter the following values:

Parameter	Value
AX	10
IX	10
IY	300
IZ	300

Finally, click on the *Add* button as shown below.

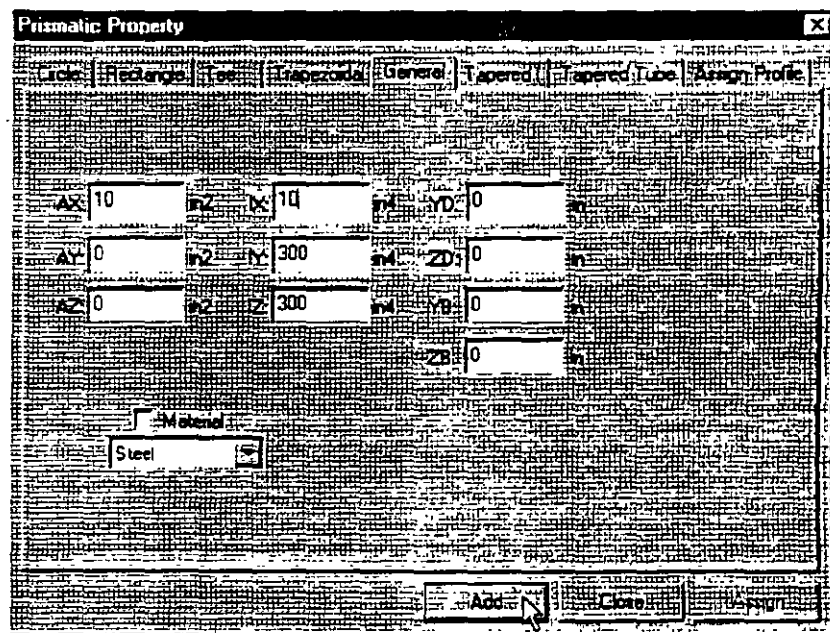


Figure 5. 22

19. Since the selected cross section has to be assigned to all the members in the structure, the simplest method to do that will be to set the *Assignment Method* as *Assign to View*. Click on the *Assign to View* button in the *Properties* dialog box followed by the *Assign* button. As we click on the *Assign* button, the following message box appears asking us to re-confirm that we do indeed want to assign this property to all the members in the model.

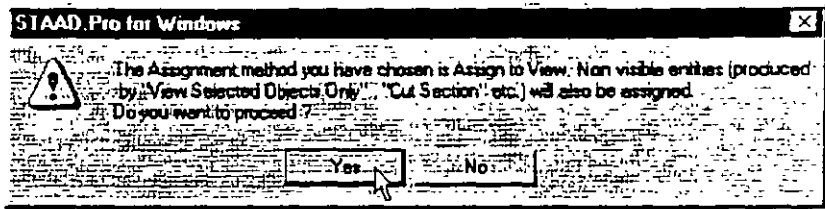


Figure 5. 23

Let us click on the *Yes* button and *Close* the *Properties* dialog box. (see Figure 5.24)

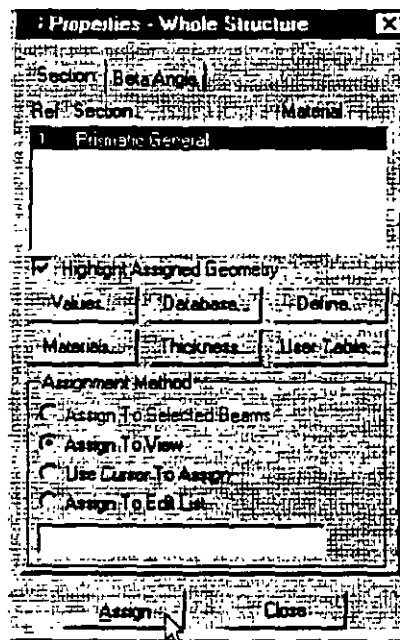


Figure 5. 24

After assigning the property, the structure will look as shown below.

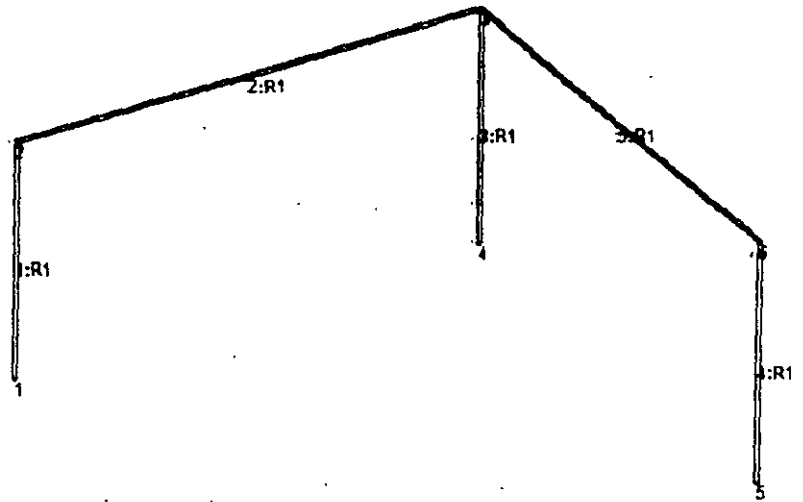


Figure 5. 25



## Specifying Material Constants

20. The Commands we wish to generate are:

```

CONSTANT
E 29000. ALL
POISSON STEEL ALL
  
```

To do this, go to *Commands / Material Constants / Elasticity* option from the top menu bar as shown below.

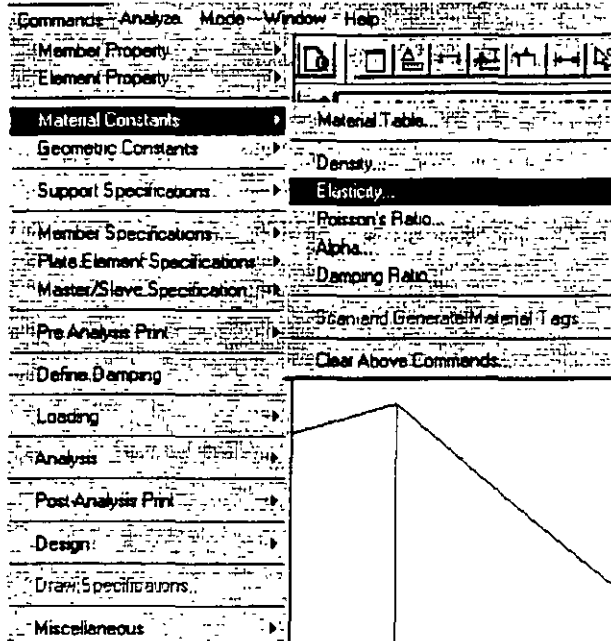


Figure 5. 26

21. In the *Material Constant* dialog box that appears, enter 29000. in the *Enter Value* box. Since the value has to be assigned to all the members of the structure, setting the assignment method *To View* allows us to achieve this easily. Then, click on *OK*. (see figure below)

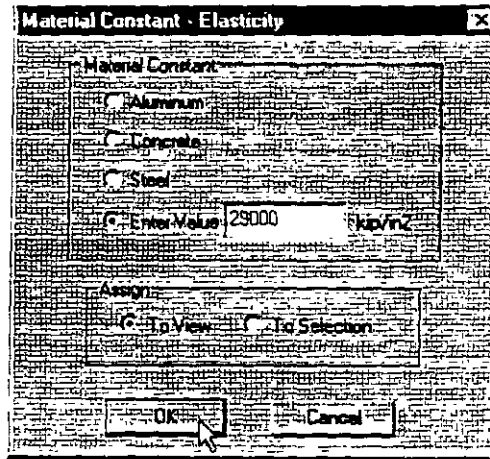


Figure 5. 27

22. In a similar fashion, set the *Poisson's Ratio* to the *Material Constant* for *Steel* and assign to all members in the view.

## Assigning Supports

23. The commands we wish to generate are:  $\gamma$

### 1 4 5-FIXED

To do this, select the *General / Support* Page from the left side of the screen. In the *Supports* dialog box, click on the *Add* button. (see figure below)

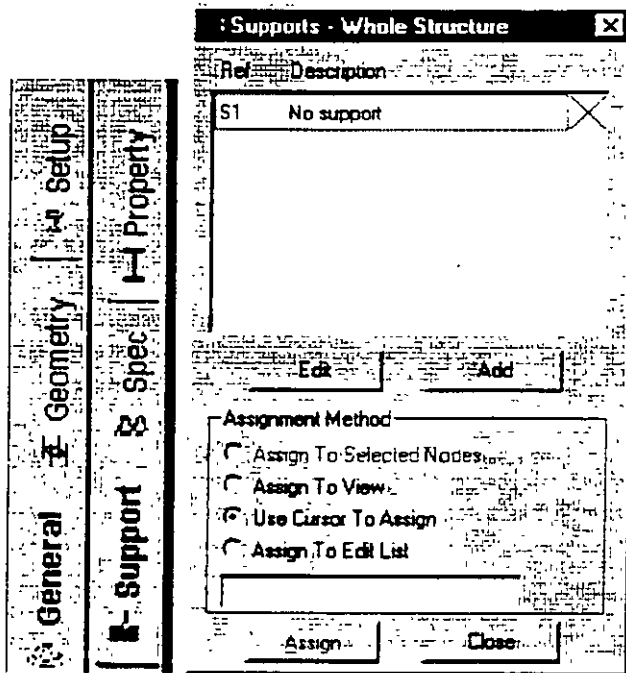


Figure 5. 28

24. In the *Create Support* dialog box that opens, select the *Fixed* tab and then click on the *Create* button as shown below. This creates a *FIXED* type of support where all 6 degrees of freedom are restrained.

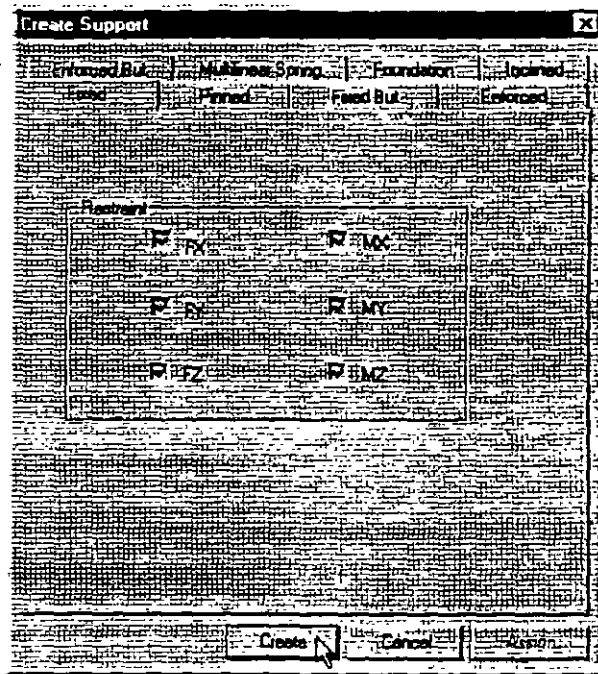


Figure 5. 29

25. To assign the support, first select nodes 1, 4 and 5. To select these nodes, go to *Select / By List / Nodes* menu option. In the *Select Nodes* dialog box, type the node numbers in the *Enter list* box.

26. Then, make sure that the Support 2 (Fixed) parameter is selected in the *Supports* dialog box.

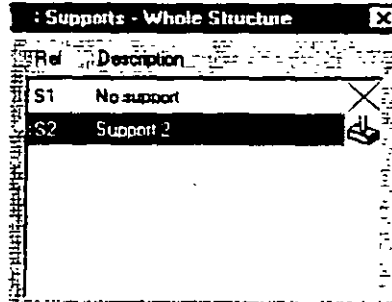


Figure 5.30

27. Notice that the *Assignment Method* becomes automatically set to *Assign to Selected Nodes*. Click on the *Assign* button in the *Supports* dialog box. (see figure below)

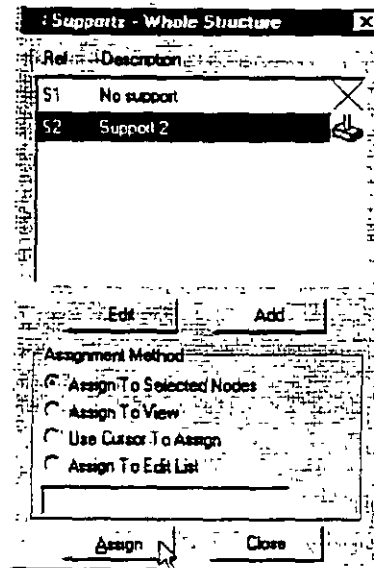


Figure 5.31

28. As we click on the *Assign* button, the following message box appears (shown below) asking us to re-confirm that we do indeed want to assign this support to the selected nodes in the model. Let us click on *OK*.

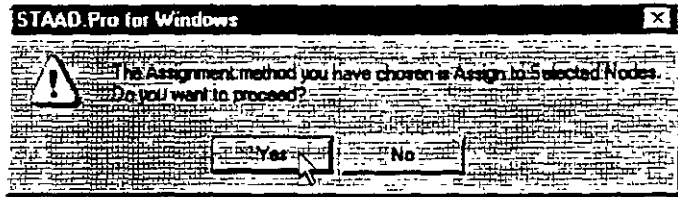


Figure 5. 32

After the supports are assigned, the structure will look as follows:

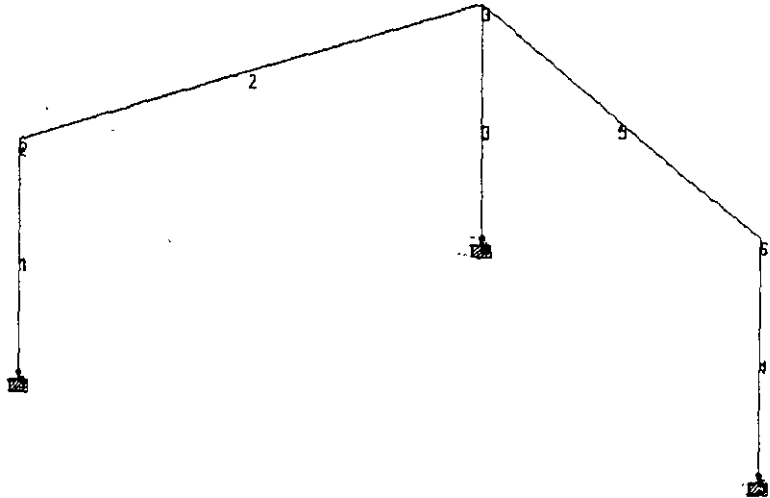


Figure 5. 33

## Specifying Loads

29. Load assignments are done from the *General / Load* Page as shown below.

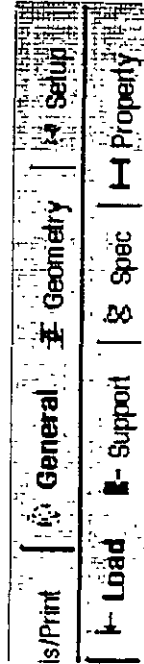


Figure 5. 34

For load case 1, we wish to apply a  $\frac{1}{2}$  inch displacement downwards in the global Y direction at the support at node 4.

The commands we wish to generate are:

```
LOADING 1 SINKING SUPPORT  
SUPPORT DISPLACEMENT LOAD  
4 FY -0.50
```

In the *Set Active Primary Load Case* dialog box that comes up, enter SINKING SUPPORT as the title for *Load Case 1* and click on *OK*. (see figure below)

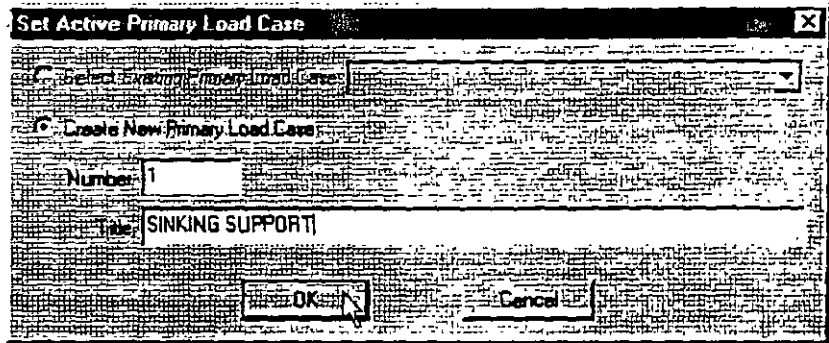


Figure 5. 35



- 30. The *Loads* dialog box will now appear in the Data Area on the right hand side of the screen. *Support Displacement* loads are assigned through the dialog boxes available under the *Nodal* type of loads. Hence, click on the *Nodal* button.(see figure below)

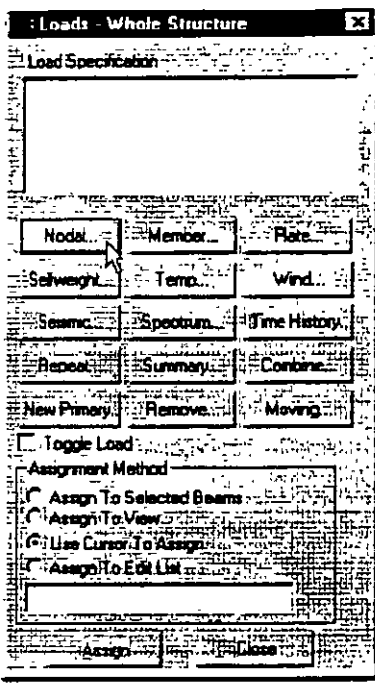


Figure 5. 36

31. In the *Node Loads* dialog box, select the *Support Displacement* tab. Enter  $-0.50$  as the *Displacement* value, set the *Direction* to  $F_y$  and click on the *Add* button.

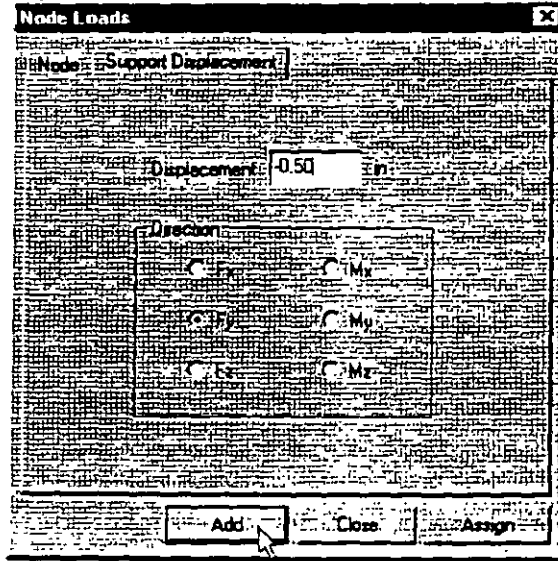


Figure 5. 37

32. This load is to be applied on node 4 of the model. We shall use a method of assignment called *Use Cursor to Assign*. To do so, first select the load from the *Loads* dialog box (it will become highlighted).

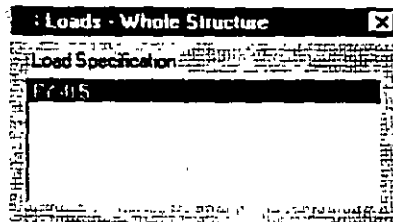


Figure 5. 38

Make sure that the *Nodes Cursor*  is selected so that we can select the nodes.

Click on the *Use Cursor To Assign* radio button. Then, click on the *Assign* button. The button will appear depressed and the label will change to *Assigning*. Using the cursor, click on node 4 to which the load is to be assigned. Click on the *Assign* button again to finish. (see figure below)

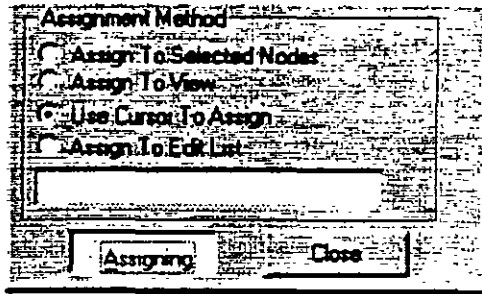


Figure 5. 39

## Specifying The Analysis Command

33. The next step is to assign the commands to perform the analysis and report the analysis results. We wish to generate the following commands:

**PERFORM ANALYSIS**  
**PRINT ANALYSIS RESULTS**

To do this, first go to *Analysis/Print* Page from the left side of the screen. Then, click on the *Analysis* sub-page from the second row of pages as shown below.



Figure 5. 40

34. Click on the *Define Commands* button in the Data Area on the right hand side of the screen.

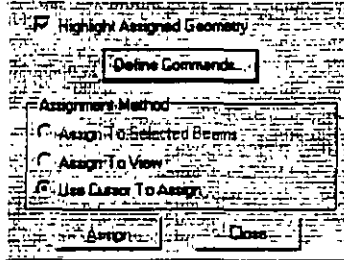


Figure 5. 41

35. In the *Analysis/Print Commands* dialog box that appears, select the *No Print* option. Then, click on the *Add* button followed by the *Close* button.

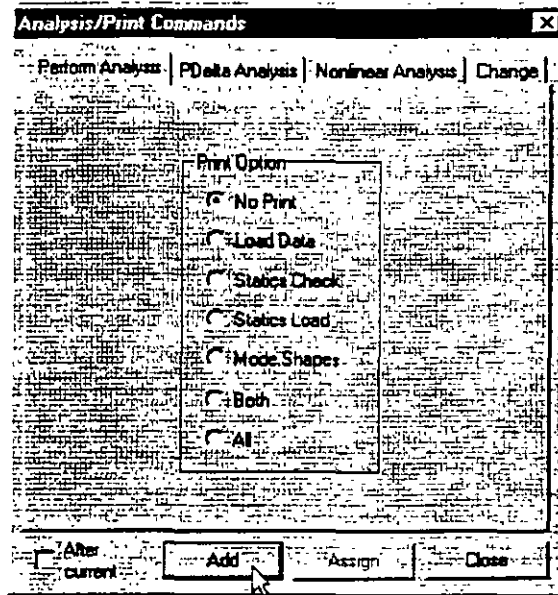


Figure 5. 42

- 36 The dialog box for specifying the "PRINT ANALYSIS RESULTS" command is nestled in the *Post-Print* sub-page of the *Analysis* Page.

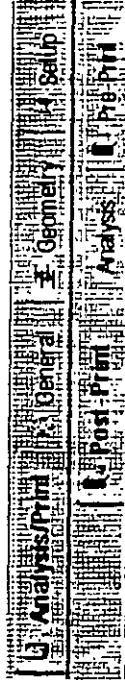


Figure 5. 43

37. Click on the *Define Commands* button in the Data Area on the right hand side of the screen.

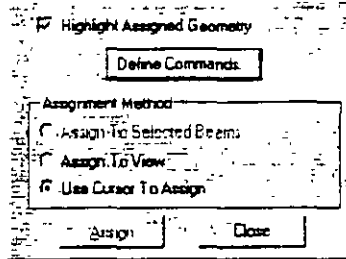


Figure 5. 44

38. In the *Analysis/Print Commands* dialog box that appears, select the *Analysis Results* tab. Then, click on the *Add* button followed by the *Close* button.

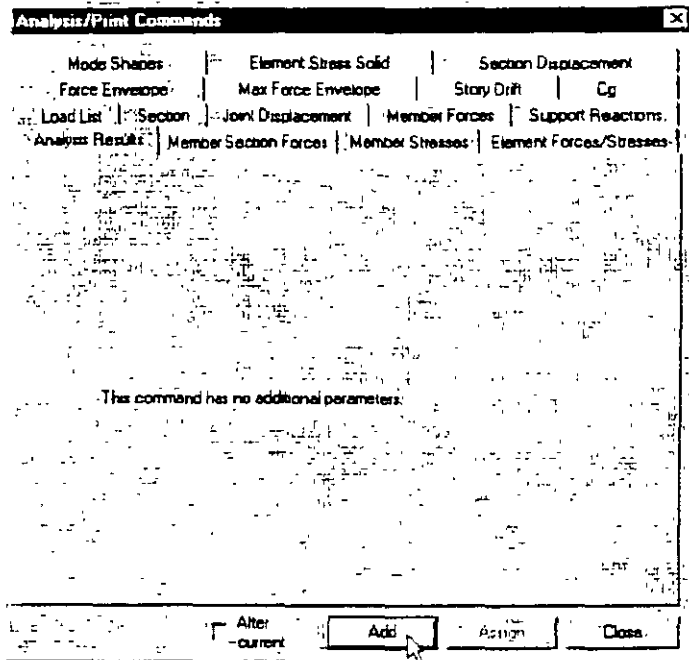


Figure 5. 45

This concludes the assigning of all the data to the structure. From the *File* menu, select *Save*, and provide a file name, if you haven't already done so

Citemos para terminar tres proyectos gigantescos que podrían algún día realizarse en la U.R.S.S.:

- Central de 5.000 MW en el golfo de Mezensk, con una amplitud media de marea de 5,80 m, área embalsada de 860 km<sup>2</sup> y longitud del dique de 50 km.
- Central de 10.000 MW en el golfo de Tugursk, con una amplitud media de marea de 5,80 m, área embalsada de 1.140 km<sup>2</sup> y longitud de dique de 17,4 km.
- Central de 3.500 MW en el golfo de Penzhinsk, con una amplitud media de marea de 6,8 m, área embalsada de 6.720 km<sup>2</sup> y longitud de dique de 31,6 km.

Las reservas mundiales de energía mareomotriz superan a las de todos los ríos del mundo y superan los 10<sup>9</sup> MW, pero su explotación es excesivamente costosa.

### 23.3. LA ENERGIA DE LAS OLAS

Es otra fuente de energía del mar aprovechable. Existen en la actualidad multitud de ideas, prototipos y patentes pero ninguna aplicación práctica importante. En la actualidad se estiman los costes por kW instalado de este tipo de energía ocho veces mayores que en la energía térmica convencional o nuclear. Un esquema reciente que según el inventor reduciría notablemente los costes, consiste en unidades que constan de un globo semiflotante alargado con una quilla de hormigón en su interior. Al pasar las olas comprimen y expansionan alternativamente el aire en los compartimentos internos, originando unos movimientos peristálticos, que bombean aire a través de válvulas y conductos instalados en la quilla. El aire finalmente acciona la turbina de aire acoplada al generador (5). Otro proyecto prometedor consiste en una instalación combinada de desalinización del agua del mar por el principio de la ósmosis invertida y de extracción de energía de las olas, cuya economía se presenta muy atractiva.

### PROBLEMAS

23-1. En este problema se despreciarán las pérdidas. Una turbina de aire tiene un diámetro de 3 m y es accionada por un viento de 50 km/h. La densidad del aire es 1,2 kg/m<sup>3</sup>.

Calcular la potencia máxima del aire.

23-2. La misma turbina del problema anterior atravesada ahora por un caudal de aire de la misma densidad de 160 m<sup>3</sup>/s. Las presiones medias antes y después de la turbina son 2, 5 y 2 mbar, respectivamente.

Calcular:

- a) la velocidad del viento;
- b) la fuerza axial que se ejerce sobre la turbina;
- c) la potencia del aire.

(5) Cf. *Ups and downs of wave energy*, en «Water Power, 31, 1 (1979)», 54-55. Véase también *Symposium über Meeresenergienutzung* (Simposio sobre la utilización de la energía del mar), en «BWK, 31, 2 (1979)», celebrado en septiembre de 1979.



## 24. *Turbomáquinas hidráulicas: Transmisiones hidrodinámicas*

### 24.1. INTRODUCCION

Para transmitir potencia a corta o larga distancia existen entre otras, además de la solución eléctrica, dos soluciones: la mecánica y la hidráulica. Las dos soluciones han sido empleadas en los barcos.

Un barco movido con turbina de vapor presenta el problema de que la turbina debe girar a gran velocidad; mientras que la hélice a poca velocidad.

- la *transmisión eléctrica* consistiría en hacer que la turbina accionara un generador, cuya potencia por cable se transmitiría al motor que movería la hélice. Este tipo de transmisión es frecuente en las locomotoras diesel.
- la *transmisión mecánica* en este caso consiste en utilizar engranajes helicoidales reductores. Esta solución es la más empleada en los barcos por su mayor sencillez y rendimiento (en otras aplicaciones las transmisiones mecánicas pueden incluir palancas, cadenas, correas, levas, etc.).
- la *transmisión hidráulica* es la que vamos a estudiar en este capítulo.

La transmisión hidráulica consta de:

- *bomba* que comunica la potencia del eje de entrada al fluido;
- *conducto* más o menos largo y complicado por donde circula el fluido transportador de potencia (en las transmisiones hidrodinámicas estudiadas en este capítulo este conducto prácticamente no existe);
- *motor hidráulico* accionado por el fluido, que comunica potencia al eje de salida.

Las transmisiones hidráulicas se dividen en

- *Transmisiones hidrostáticas*: la bomba y el motor son de *desplazamiento positivo*. Estas transmisiones se estudian en el Cap. 28 de este libro.
- *Transmisiones hidrodinámicas*: la bomba y el motor (turbina) son turbomáquinas, y el conjunto de la transmisión es una turbomáquina compuesta. Estas transmisiones se estudian en el presente capítulo. Existen dos tipos distintos de transmisiones hidrodinámicas: *acoplamientos hidrodinámicos* (Sec. 24.2) y *convertidores de par hidrodinámicos* (Sec. 24.3).

Las transmisiones hidrodinámicas fueron ideadas por el alemán Föttinger.



**FACULTAD DE INGENIERÍA UNAM  
DIVISIÓN DE EDUCACIÓN CONTINUA**



**CA 354  
STAAD PRO PARA ANÁLISIS Y DISEÑO  
ESTRUCTURAL**

**TEMA:**

**INSTRUCTIVO PARA EL USO DEL PROGRAMA**

**COORDINADOR: ING. JOSÉ LUIS ESQUIVEL ÁVILA**

**DEL 22 AL 29 DE NOVIEMBRE DE 2004**

**PALACIO DE MINERÍA**

**Programa 2004**



# STAAD/Pro Core

---

## TECHNICAL REFERENCE MANUAL

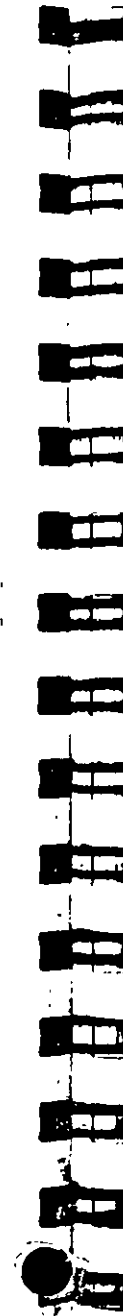


22700 Savi Ranch, Yorba Linda, CA 92887, U.S.A.  
Tel: (714) 974-2500 Fax: (714) 921-2543

STAAD/Pro is a suite of proprietary computer programs of Research Engineers, Inc. (REI), California, U S A. Although every effort has been made to ensure the accuracy of these programs, REI will not accept responsibility for any mistake, error or misrepresentation in or as a result of the usage of these programs

RELEASE 3.0

Copyright  
Research Engineers, Inc  
Published February, 1999



## About STAAD/Pro

STAAD/Pro, the next level of STAAD-III, is the leading Structural Analysis and Design software from Research Engineers. In STAAD/Pro, the focus is on productivity. STAAD/Pro addresses the entire process of Structural Engineering. From model development to analysis to design to drafting to detailing - and even component design - STAAD/Pro is designed to work the way the Structural Design Office works.

STAAD/Pro consists of a Core package and several optional Extension components. You may procure one or more Extension components along with the STAAD/Pro Core package. The Core and the Extension components of STAAD/Pro are described in the following paragraphs.

The STAAD/Pro Core package consists of the following:

The STAAD/Pro Design Studio, is a powerful Window's native working environment for model building, visualization and result verification.

The STAAD engine provides general-purpose structural analysis and integrated STEEL/CONCRETE/TIMBER design

The STARDYNE engine provides advanced analysis facilities. Built around a comprehensive finite element library, the STARDYNE engine provides powerful Dynamic, Seismic, Non-linear, Thermal, Buckling and other advanced analysis capabilities.

The FEMkit environment offers graphically oriented Finite Element modeling and verification facilities -complete with 2D/3D meshing technologies and powerful model verification tools.

The Visual DRAW CAD engine allows generation of Plans, Elevations, Sections and detail drawings. Fully integrated in the STAAD/Pro environment, Visual DRAW provides drawing generation, editing and plotting capabilities.

The following modules are available as STAAD/Pro Extension components:

STRUCT.etc - a suite of structural component design modules, allows the engineer to complete the project by designing Foundation, Retaining Walls, Masonry, Connections and other utility structures.

The **FabriCAD** detailing engine is an integrated tool that performs production Steel Detailing - from fabrication/erection calculations to detailing to generation of shop drawings.

The **ADLPIPE** component offers reliable Piping system modeling and analysis. Fully integrated with the STAAD component, this component offers a complete plant engineering solution.

Powerful and comprehensive, **STAAD/Pro** is based on an object-oriented design that utilizes MFC (Microsoft Foundation Class) technology, taking full advantage of 32 bit computing. A live relational database at the core, with OLE and DDE links, allows smooth workflow integration and seamless interdisciplinary information exchange with all Windows based software.

We welcome you to the world of **STAAD/Pro**



## About STAAD/Pro Core Documentation

The STAAD/Pro Core documentation consists of a set of manuals as described below. We strongly recommend that you go through the *Getting Started* manual to get an overview of STAAD/Pro.

### Getting Started and Examples

This manual is divided into three parts. Part-I explains the contents of the STAAD/Pro CD ROM, installation process, the system requirements, copy protection issues, and description on how to start different components of STAAD/Pro.

Part-II of this book provides a step-by-step tutorial for the STAAD/Pro environment as well as the STAAD Command File.

Part-III of this book offers *Application Examples*. The example set covers a wide range of Structural Analysis and Design facilities offered by STAAD/Pro. The examples represent various Structural problems commonly encountered by the Structural engineers.

### Graphical Environment

This manual contains detailed description of GUI (Graphical User Interface) of STAAD/Pro. The topics covered include Model generation, Structural Analysis/Design, Result verification, Report generation, and Printing. In addition, a Quick Reference Guide section provides a step-by-step description for the frequently asked modeling and post-processing tasks.

*Note* Please note that the STAAD Command File is explained in the STAAD/Pro Technical Reference manual.

### Technical Reference

This manual deals with the theory behind the STAAD and STARDYNE analysis engines offered by STAAD/Pro. It also includes explanation of all commands which you may use in the STAAD Command File.

### International Design Codes

STAAD/Pro supports all major international design codes, which are explained in this manual. You shall receive this manual if you have purchased one of these international design codes

### STAAD/Pro Verification

STAAD/Pro is developed and maintained using an extensive verification procedure. The STAAD/Pro Verification manual contains the problems used in the verification procedure. **This is distributed under the Quality Assurance program only.** Note that the set of verification problems is representative of various Analysis/Design facilities offered by STAAD/Pro. These problems have been derived from well-known references and textbooks.

### FEMkit Getting Started and Training

This manual contains the description of the FEMkit Finite element modeling environment along with a step-by-step Training section. The second part of this book also contains the *Appendices* which are referred to from the FEMkit Users manual

### FEMkit Users Manual

This manual contains the description of the FEMkit menu options and commands.

### Visual DRAW Users Manual

This manual describes different features and facilities of the Visual DRAW CAD software.

*Note: Please note that the above set of manuals comprise the STAAD/Pro Core documentation only. In addition, you may receive the documentation for the STAAD/Pro Extension components when you purchase these components*



## Table of Contents

STAAD & STARDYNE Technical Reference Manual

### Section 1 General Description

1.1	Introduction	1
1.2	Input Generation	1
1.3	Types of Structures	2
1.4	Unit Systems	3
1.5	Structure Geometry and Coordinate Systems	3
1.5.1	Global Coordinate System	4
1.5.2	Local Coordinate System	6
1.5.3	Relationship Between Global & Local Coordinates	8
1.6	Finite Element Information	9
1.6.1	Plate/Shell Element	9
1.6.2	Solid Element	18
1.7	Member Properties	21
1.7.1	Prismatic Properties	21
1.7.2	Built-In Steel Section Library	23
1.7.3	User Provided Steel Table	23
1.7.4	Tapered Sections	24
1.7.5	Assign Command	24
1.8	Member/Element Release	24
1.9	Truss/Tension - Only Members	25
1.10	Cable Members	25
1.11	Member Offsets	28
1.12	Material Constants	29
1.13	Supports	30
1.14	Master/Slave Joints	30
1.15	Loads	31
1.16.1	Joint Load	31
1.16.2	Member Load	31
1.16.3	Area Load	32
1.16.4	Fixed End Member Load	33
1.16.5	Prestress and Poststress Member Load	34
1.16.6	Temperature/Strain Load	36
1.16.7	Support Displacement Load	36
1.16.8	Loading on Elements	37
1.17	Load Generator	38

1.17.1 Moving Load Generator	38
1.17.2 UBC Seismic Load Generator	38
1.17.3 Wind Load Generator	39
1.18 Analysis Facilities	39
1.18.1 Stiffness Analysis	40
1.18.2 Second Order Analysis	44
1.18.2.1 P Delta Analysis	44
1.18.2.2 Non Linear Analysis	46
1.18.3 Dynamic Analysis	47
1.19 Member End Forces	50
1.19.1 Secondary Analysis	51
1.19.2 Member Forces at Intermediate Sections	51
1.19.3 Member Displacements at Intermediate Sections	52
1.19.4 Member Stresses at Specified Sections	52
1.19.5 Force Envelopes	52
1.20 Multiple Analyses	53
1.21 Steel/Concrete/Timber Design	54
1.22 Footing Design	54
1.23 Printing Facilities	54
1.24 Plotting Facilities	55
1.25 Miscellaneous Facilities	55
1.26 Post Processing Facilities	56

## Section 2 American Steel Design

2.1 Design Operations	59
2.2 Member Properties	60
2.2.1 Built-in Steel Section Library	60
2.3 Allowables per AISC Code	64
2.3.1 Tension Stress	64
2.3.2 Shear Stress	64
2.3.3 Stress Due To Compression	65
2.3.4 Bending Stress	65
2.3.5 Combined Compression and Bending	66
2.3.6 Singly Symmetric Sections	67
2.3.7 Torsion per Publication T114	67
2.3.8 Design of Web Tapered Sections	69
2.4 Design Parameters	69
2.5 Code Checking	70
2.6 Member Selection	70
2.6.1 Member Selection by Optimization	71
2.6.2 Deflection Check With Steel Design	71
2.7 Truss Members	72
2.8 Unsymmetric Sections	72
2.9 Plate Girders	76
2.10 Punching Shear Check	77
2.11 Tabulated Results of Steel Design	78

2.12 Weld Design	80
2.13 Steel Design per AASHTO Specifications	83
2.13.1 General Comments	83
2.13.2 Allowable Stresses per AASHTO Code	84
2.13.3 Stability Requirements per AASHTO Code	87
2.13.4 Minimum Metal Thickness Requirement	87
2.14 Steel Design per AISC/LRFD Specification	87
2.14.1 General Comments	87
2.14.2 LRFD Fundamentals	88
2.14.3 Analysis Requirements	90
2.14.4 Section Classification	90
2.14.5 Axial Tension	90
2.14.6 Axial Compression	91
2.14.7 Flexural Design Strength	92
2.14.8 Combined Axial Force And Bending	92
2.14.9 Design for Shear	93
2.14.10 Design Parameters	93
2.14.11 Code Checking and Member Selection	93
2.14.12 Tabulated Results of Steel Design	94

## Section 3 American Concrete Design

3.1 Design Operations	97
3.2 Section Types for Concrete Design	97
3.3 Member Dimensions	98
3.4 Design Parameters	99
3.5 Slenderness Effects and Analysis Consideration	99
3.6 Beam Design	103
3.7 Column Design	107
3.8 Slab/Wall Design	110

## Section 4 Timber Design

4.1 Timber Design	113
4.2 Design Operations	114
4.3 Input Specification	116
4.4 Code Checking	118
4.5 Orientation of Lamination	118
4.6 Member Selection	119

## Section 5 Commands and Input Instructions

5.1 Command Language Conventions	126
5.1.1 Elements of The Commands	127
5.1.2 Command Formats	129
5.1.3 Listing of Members by Specification of Global Ranges	132
5.2 Problem Initiation And Title	134

5.3	Unit Specification	136
5.4	Input/Output Width Specification	138
5.5	Set Command Specification	139
5.6	Separator Command	142
5.7	Page New Command	143
5.8	Page Length/Eject Command	144
5.9	Ignore Specifications	145
5.10	No Design Specification	146
5.11	Joint Coordinates Specification	147
5.12	Member Incidences Specification	152
5.13	Element Incidence Specification	156
5.14	Element Mesh Generation	160
5.15	Redefinition of Joint and Member Numbers	166
5.16	Listing of Members by Specification of GROUPS	168
5.17	Rotation of Structure Geometry	171
5.18	Inactive/Delete Specification	172
5.19	User Steel Table Specification	174
5.20	Member Property Specification	182
5.20.1	Specifying Properties from Steel Table	184
5.20.2	Prismatic Property Specification	187
5.20.3	Tapered Member Specification	188
5.20.4	Property Specification from User Provided Table	189
5.20.5	Assign Profile Specification	190
5.20.6	Examples of Member Property Specification	191
5.21	Element Property Specification	193
5.22	Member/Element Releases	194
5.22.1	Member Release Specification	195
5.22.2	Element Release Specification	198
5.23	Member Truss/Cable/Tension/Compression Specification	200
5.23.1	Member Truss Specification	201
5.23.2	Member Cable Specification	202
5.23.3	Member Tension/Compression Specification	203
5.24	Element Plane Stress and Ignore Inplane Rotation Specification	208
5.25	Member Offset Specification	210
5.26	Constant Specification	212
5.27	Support Specifications	215
5.27.1	Global Support Specification	216
5.27.2	Inclined Support Specification	218
5.27.3	Automatic Spring Support Generator for Foundations	220
5.27.4	Multi-linear Spring Support Specification	222
5.28	Master/Slave Specification	224
5.29	Draw Specifications	226
5.30	Cut-Off Frequency or Mode Shapes	230
5.31	Definition of Load Systems	231
5.31.1	Definition of Moving Load System	232
5.31.2	Definition of LBC Load	236
5.31.3	Colombian Seismic Load	239



5.31.4	Japanese Seismic Load	242
5.31.5	Definition of Wind Load	245
5.31.6	Definition of Time History Load	247
5.32	Loading Specifications	253
5.32.1	Joint Load Specification	254
5.32.2	Member Load Specification	255
5.32.3	Element Load Specification	258
5.32.4	Area Load/Floor Load Specification	261
5.32.5	Prestress Load Specification	266
5.32.6	Temperature Load Specification for Members and Elements	273
5.32.7	Fixed-End Load Specification	275
5.32.8	Support Displacement Load Specification	276
5.32.9	Selfweight Load Specification	278
5.32.10	Dynamic Loading Specification	279
5.32.10.1	Response Spectrum Specification	280
5.32.10.2	Application of Time Varying Load for Response History Analysis	283
5.32.11	Repeat Load Specification	285
5.32.12	Generation of Loads	287
5.33	Natural Frequency Specification	295
5.34	Modal Calculation Command	297
5.35	Load Combination Specification	298
5.36	Calculation of Problem Statistics	302
5.37	Analysis Specification	303
5.38	Change Specification	307
5.39	Load List Specification	309
5.40	Section Specification	311
5.41	Print Specifications	313
5.42	Print Section Displacement	319
5.43	Print Force Envelope Specification	321
5.44	Post Analysis Printer Plot Specifications	323
5.45	Post Analysis Graphics Display	324
5.45.1	Display Deflected Shapes	325
5.45.2	Display Section Displacements	326
5.45.3	Display Bending Moment/Shear Force Diagrams	327
5.45.4	Display Mode Shapes	328
5.45.5	Display Stress Contours	329
5.46	Size Specification	330
5.47	Steel Design Specifications	332
5.47.1	Parameter Specifications	334
5.47.2	Code Checking Specification	335
5.47.3	Member Selection Specification	336
5.47.4	Member Selection by Optimization	337
5.47.5	Weld Selection Specification	338
5.48	Group Specification	339
5.49	Steel Take Off Specification	341
5.50	Timber Design Specifications	342

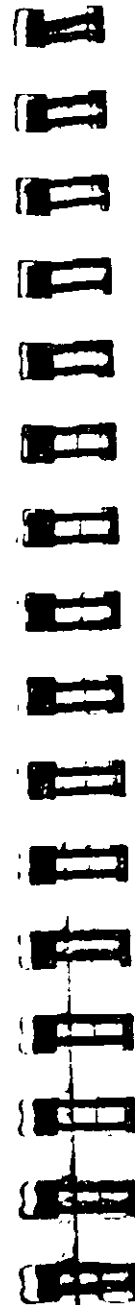


5 50 1	Timber Design Parameter Specifications	343
5 50 2	Code Checking Specification	344
5 50 3	Member Selection Specification	345
5 51	Concrete Design Specifications	346
5 51 1	Design Initiation	347
5 51 2	Concrete Design-Parameter Specification	348
5 51 3	Concrete Design Command	349
5 51 4	Concrete Take Off Command	350
5 51 5	Concrete Design Terminator	351
5 52	Footing Design Specifications	352
5 52 1	Design Initiation	355
5 52 2	Footing Design Parameter Specification	356
5 52 3	Footing Design Command	357
5 52 4	Footing Design Terminator	359
5 53	Miscellaneous Special Commands	360
5 53 1	Precision Command	361
5 53 2	Input Memory Command	362
5 54	Save Specification	363
5 55	Restore Specification	364
5 56	End Run Specification	365

## Section 6 STARDYNE for STAAD/Pro

6 1	Introduction	367
6 2	Input Generation - STARDYNE	368
6 3	Types of Structures - STARDYNE	368
6 4	Unit Systems - STARDYNE	368
6 5	Structure Geometry and Coordinate Systems-STARDYNE	368
6 5 1	Global Coordinate System	369
6 5 2	Local Element Coordinate System	369
6 6	Finite Element Information - STARDYNE	370
6 6 1	Plate/Shell Element	370
6 6 2	Solid Element	374
6 6 3	Matrix (General Spring) Element	376
6 6 4	Beam Element	376
6 6 4 1	Beam Element Prismatic Properties	378
6 6 4 2	Tapered Beam Elements	378
6 6 4 3	Pipe and Elbow Beam Elements	378
6 6 4 4	Beam Element End Releases	378
6 6 4 5	Tension - Only Elements	379
6 6 4 6	Cable Elements	379
6 6 4 7	Beam Element Offsets	379
6 7	Rigid System of Nodes - STARDYNE	379
6 8	Material Constants - STARDYNE	380
6 9	Supports - STARDYNE	381
6 10	Loads (STARDYNE)	381
6 10 1	Node Load	381

6 10 2	Element Loads	382
6 10 3	Node Temperature Load	383
6 10 4	Support Displacement Specification	384
6 11	STARDYNE Analysis Capabilities	384
6 11 1	STARDYNE Static Analysis	385
6 11 2	STARDYNE Second Order Static Analysis	389
6 11 3	STARDYNE Dynamic Response Analysis	393
6 11 3 1	STARDYNE Response Spectrum Analysis (Dynre4)	398
6 11 3 2	STARDYNE Linear Time History Response Analysis (Dynre1)	399
6 11 3 3	STARDYNE Steady State Harmonic Frequency Response Analysis (Dynre2)	402
6 11 3 4	STARDYNE Missing Mass Method	403
6 12	References (STARDYNE)	406



---

## General Description

---

## Section 1

### 1.1 Introduction

STAAD/Pro offers two analysis engines - the STAAD Analysis/Design engine and the STARDYNE Advanced Analysis engine. The STAAD analysis engine is described in this section. The STARDYNE analysis engine is discussed in section 6.

This section of the manual contains a general description of the analysis and design facilities available in STAAD. Specific information on Steel, Concrete, and Timber design are available in Sections 2, 3, and 4 of this manual, respectively. Detailed command formats and other specific user information is presented in Section 5.

The objective of this section is to familiarize the user with the basic principles involved in the implementation of the various analysis/design facilities offered by the STAAD engine. As a general rule, the sequence in which the facilities are discussed follows the recommended sequence of their usage in the input file.

### 1.2 Input Generation

The user communicates with STAAD through an input file. The input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design. The

*See section 5*

elements and conventions of the STAAD command language are described in Section 5 of this manual

The STAAD input file can be created through a text editor or the Modeling facility. In general, any text editor may be utilized to create the input file. The Modeling facility creates the input file through an interactive menu-driven graphics oriented procedure

### 1.3 Types of Structures

A STRUCTURE can be defined as an assemblage of elements. STAAD is capable of analyzing and designing structures consisting of both frame and plate/shell elements. Almost any type of structure can be analyzed by STAAD. Most general is the SPACE structure, which is a three dimensional framed structure with loads applied in any plane. A PLANE structure is bound by a global X-Y coordinate system with loads in the same plane. A TRUSS structure consists of truss members which can have only axial member forces and no bending in the members. A FLOOR structure is a two or three dimensional structure having no horizontal (global X or Z) applied loads or any load which may cause any horizontal movement of the structure. The floor framing (in global X-Z plane) of a building is an ideal example of a FLOOR structure. Columns can also be modeled with the floor in a FLOOR structure as long as the structure has no horizontal loading. If there is any horizontal load, it must be analyzed as a SPACE structure. Specification of the correct structure type reduces the number of equations to be solved during the analysis. This results in a faster and more economic solution for the user. The degrees of freedom associated with frame elements of different types of structures is illustrated in Figure 1.1.

For input, see section 5.2

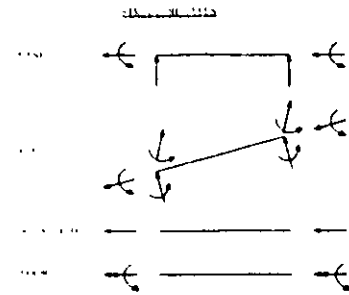


Figure 1.1

### 1.4 Unit Systems

The user is allowed to input data and request output in almost all commonly used engineering unit systems including MKS, SI and FPS. In the input file, the user may change units as many times as required. Mix and match between length and force units from different unit systems is also allowed. The input-unit for angles (or rotations) is degrees. However, in JOINT DISPLACEMENT output, the rotations are provided in radians. For all output, the units are clearly specified by the program.

For input, see section 5.3

### 1.5 Structure Geometry and Coordinate Systems

A structure is an assembly of individual components such as beams, columns, slabs, plates etc.. In STAAD, frame elements and plate elements may be used to model the structural components. Typically, modeling of the structure geometry consists of two steps:

- A. Identification and description of joints or nodes.
- B. Modeling of members or elements through specification of connectivity (incidences) between joints.

For input, see sections 5.11 to 5.17

In general, the term MEMBER will be used to refer to frame elements and the term ELEMENT will be used to refer to plate/shell elements. Connectivity for MEMBERS may be provided through the MEMBER INCIDENCE command while connectivity for ELEMENTS may be provided through the ELEMENT INCIDENCE command.

STAAD uses two types of coordinate systems to define the structure geometry and loading patterns. The GLOBAL coordinate system is an arbitrary coordinate system in space which is utilized to specify the overall geometry & loading pattern of the structure. A LOCAL coordinate system is associated with each member (or element) and is utilized in MEMBER END FORCE output or local load specification.

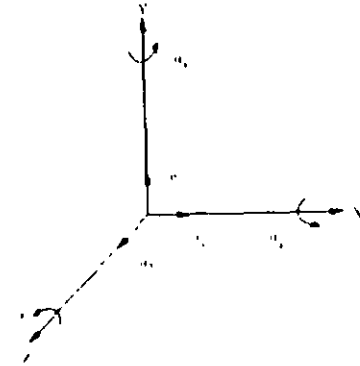
### 1.5.1 Global Coordinate System

The following coordinate systems are available for specification of the structure geometry

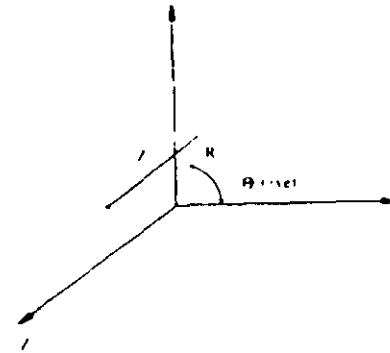
- A. **Conventional Cartesian Coordinate System.** This coordinate system (Fig. 1.2) is a rectangular coordinate system (X, Y, Z) which follows the orthogonal right hand rule. This coordinate system may be used to define the joint locations and loading directions. The translational degrees of freedom are denoted by  $u_1, u_2, u_3$  and the rotational degrees of freedom are denoted by  $u_4, u_5$  &  $u_6$ .
- B. **Cylindrical Coordinate System** In this coordinate system, (Fig. 1.3) the X and Y coordinates of the conventional cartesian system are replaced by R (radius) and  $\theta$  (angle in degrees). The Z coordinate is identical to the Z coordinate of the cartesian system and its positive direction is determined by the right hand rule.
- C. **Reverse Cylindrical Coordinate System:** This is a cylindrical type coordinate system (Fig. 1.4) where the R-  $\theta$  plane corresponds to the X-Z plane of the cartesian system. The right



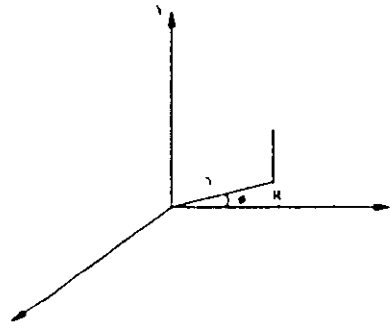
hand rule is followed to determine the positive direction of the Y axis.



Cartesian (Rectangular) Coordinate System  
Figure 1.2



Cylindrical Coordinate System  
Figure 1.3



Reverse Cylindrical Coordinate System  
Figure 1.4

### 1.5.2 Local Coordinate System

A local coordinate system is associated with each member. Each axis of the local orthogonal coordinate system is also based on the right hand rule. Fig. 1.5 shows a beam member with start joint 'i' and end joint 'j'. The positive direction of the local x axis is determined by joining 'i' to 'j' and projecting it in the same direction. The right hand rule may be applied to obtain the positive directions of the local y and z axes. The local y and z-axes coincide with the axes of the two principal moments of inertia. Note that the local coordinate system is always rectangular.

A wide range of cross-sectional shapes may be specified for analysis. These include rolled steel shapes, user specified prismatic shapes etc.. Fig. 1.6 shows local axis system(s) for these shapes.

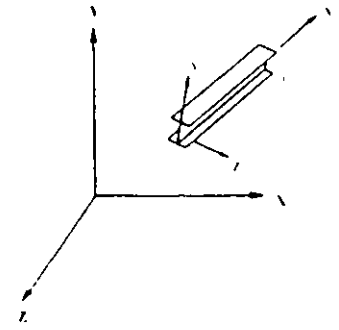
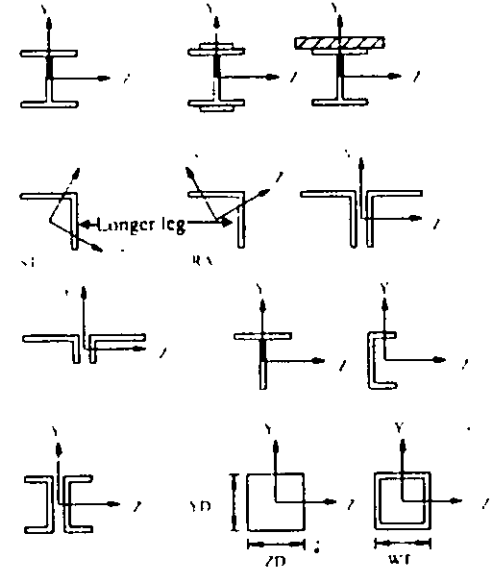


Figure 1.5



Local axis for different cross sections

Figure 1.6

NOTE: The local x-axis of the above sections are going into the paper

### 1.5.3 Relationship Between Global & Local Coordinates

Since the input for member loads can be provided in the local and global coordinate system and the output for member end-forces is printed in the local coordinate system, it is important to know the relationship between the local and global coordinate systems. This relationship is defined by an angle measured in the following specified way. This angle will be defined as the beta ( $\beta$ ) angle.

#### Beta Angle

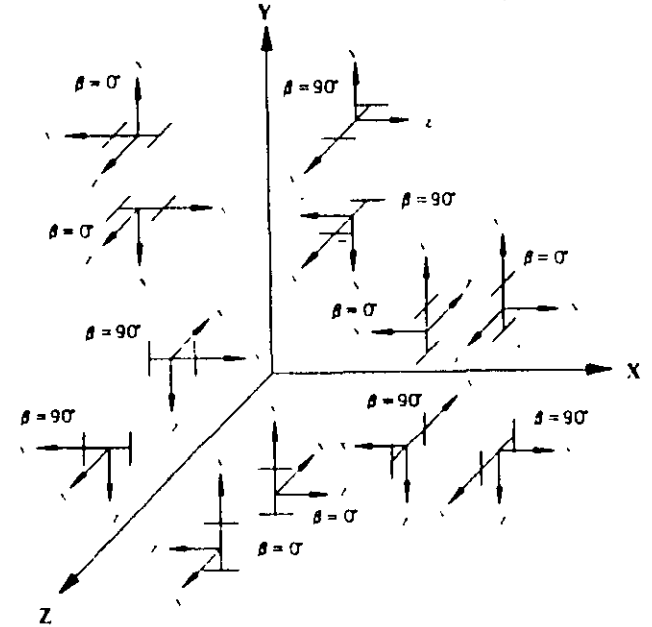
When the local x-axis is parallel to the global Y-axis, as in the case of a column in a structure, the beta angle is the angle through which the local z-axis has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis.

For input, see section 5.26

When the local x-axis is not parallel to the global Y-axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the local z-axis parallel to the global X-Z plane and the local y axis in the same positive direction as the global Y-axis. Figure 1.7 details the positions for beta equals 0 degrees or 90 degrees. When providing member loads in the local member axis, it is helpful to refer to this figure for a quick determination of the local axis system.

#### Reference Point

An alternative to providing the member orientation is to input the coordinates of an arbitrary reference point located in the member x-y plane but not on the axis of the member. From the location of the reference point, the program automatically calculates the orientation of the member x-y plane.



Relationship between Global and Local axes  
Figure 1.7

### 1.6 Finite Element Information

For input, see sections 5.11 - 5.13, 5.14 - 5.21, 5.24, and 5.32.3

STAAD is equipped with a state-of-the-art plate/shell and solid finite element. The features of each is explained below.

#### 1.6.1 Plate/Shell Element

The Plate/Shell finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (quadrilateral). If all the four nodes of a quadrilateral element do

not lie on one plane, it is advisable to model them as triangular elements. The thickness of the element may be different from one node to another.

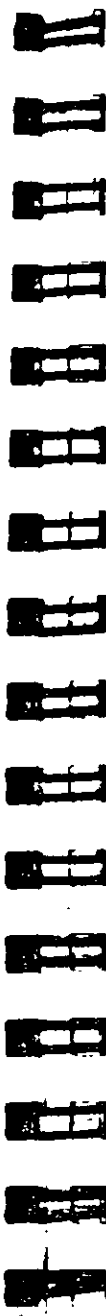
"Surface structures" such as walls, slabs, plates and shells may be modeled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, a MESH GENERATION facility is available. The facility is described in detail in Section 5.14.

The user may also use the element for PLANE STRESS action only. The ELEMENT PLANE STRESS command should be used for this purpose.

### Geometry Modeling Considerations

The following geometry related modeling rules should be remembered while using the plate/shell element

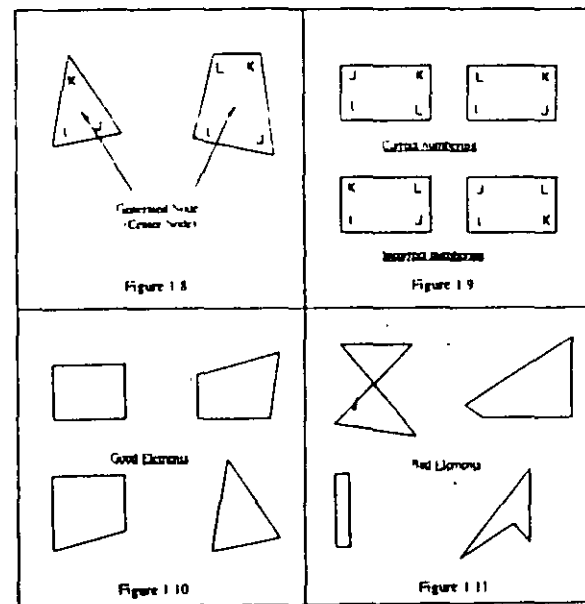
- 1) The program automatically generates a fifth node "O" (center node - see Fig. 1.8) at the element center
- 2) While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise (Fig. 1.9). For better efficiency, similar elements should be numbered sequentially
- 3) Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1
- 4) Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90 and never larger than 180.



### Element Load Specification

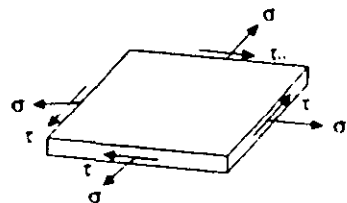
Following load specifications are available:

- 1) Joint loads at element nodes in global directions.
- 2) Concentrated loads at any user specified point within the element in global or local directions.
- 3) Uniform pressure on element surface in global or local directions
- 4) Partial uniform pressure on user specified portion of element surface in global or local directions
- 5) Linearly varying pressure on element surface in local directions.
- 6) Temperature load due to uniform increase or decrease of temperature.
- 7) Temperature load due to difference in temperature between top and bottom surfaces of the element



### Theoretical Basis

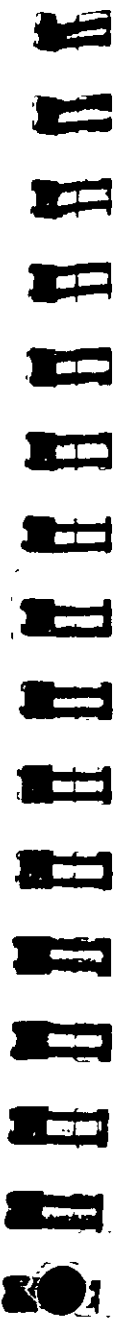
The STAAD plate finite element is based on hybrid finite element formulations. A complete quadratic stress distribution is assumed. For plane stress action, the assumed stress distribution is as follows.



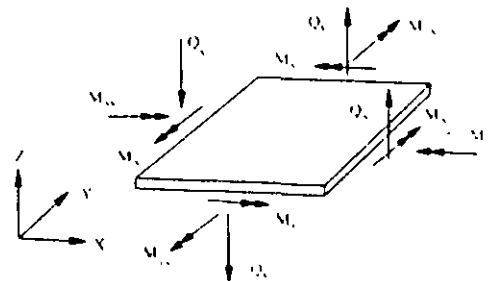
Complete quadratic assumed stress distribution

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & x^2 & 2xy & y^2 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & x^2 & 2xy \\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & 0 & 0 & -x^2 & 2xy \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \\ a_7 \\ a_8 \\ a_9 \\ a_{10} \\ a_{11} \\ a_{12} \\ a_{13} \end{pmatrix}$$

$a_1$  through  $a_{13}$  = constants of stress polynomials.



The following quadratic stress distribution is assumed for plate bending action:



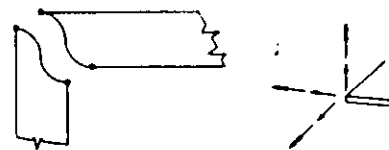
Complete quadratic assumed stress distribution:

$$\begin{pmatrix} M_x \\ M_y \\ M_{xy} \\ Q_x \\ Q_y \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & x^2 & xy & y^2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & x^2 & xy & y^2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & 0 & -xy & x^2 & y^2 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & -x & 0 & 2y \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & 0 & 0 & x & y & 2x \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \\ a_5 \\ a_6 \\ a_7 \\ a_8 \\ a_9 \\ a_{10} \\ a_{11} \\ a_{12} \\ a_{13} \\ a_{14} \\ a_{15} \\ a_{16} \\ a_{17} \end{pmatrix}$$

$a_1$  through  $a_{17}$  = constants of stress polynomials.

The distinguishing features of this finite element are:

- 1) Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see Fig. below) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.



- 2) The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not



treated as a dummy as is usually done in most commonly available commercial software.

- 3) Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely
- 4) These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.
- 5) These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.
- 6) These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.
- 7) Out of plane shear strain energy is incorporated in the formulation of the plate bending component. As a result, the elements respond to Poisson boundary conditions which are considered to be more accurate than the customary Kirchoff boundary conditions
- 8) The plate bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.
- 9) The plane stress triangle behaves almost on par with the well known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle which has very slow rates of convergence. Thus the triangular shell element is very useful in problems with double curvature where the quadrilateral element may not be suitable.
- 10) Stress retrieval at nodes and at any point within the element.

### Element Local Coordinate System

The precise orientation of local coordinates is determined as follows:

- 1) Designate the midpoints of the four or three element edges JJ, JK, KL, LI by M, N, O, P respectively.



- 2) The vector pointing from P to N is defined to be the local x-axis. (In a triangle, this is always parallel to IJ).
- 3) The cross-product of vectors PN and MO (for a triangle, ON and MK) defines the local z-axis, i.e.,  $z = PN \times MO$ .
- 4) The cross-product of vectors z and x defines the local y-axis, i.e.,  $y = z \times x$

The sign convention of output force and moment resultants is illustrated in Fig. 1.13

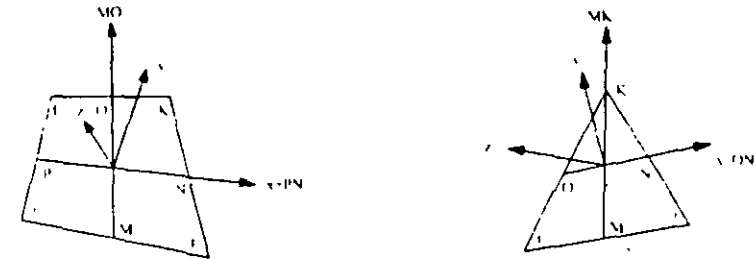


Figure 1.12

### Output of Element Forces

ELEMENT FORCE outputs are available at the following locations:

- A. Center node of the element.
- B. All corner nodes of the element.
- C. At any user specified point within the element.

Following are the items included in the ELEMENT FORCE output

QX, QY	Shear stresses (Force/ unit len./unit thk )
FX, FY, FXY	Membrane stresses (Force/unit len /unit thk)
MX, MY, MXY	Bending moments per unit width (Moment/unit len.)
SMAX, SMIN	Principal stresses (Force/unit area)
TMAX	Maxim. shear stress (Force/unit area)

**ANGLE** Orientation of the principal plane (Degrees)

Notes:

1. All element force output is in the local coordinate system. The direction and sense of the element forces are explained in Fig. I 13.
2. To obtain element forces at a specified point within the element, the user must provide the coordinate system for the element. Note that the origin of the local coordinate system coincides with the center node of the element.
3. Principal stresses (SMAX & SMIN), the maximum shear stress (TMAX) and the orientation of the principal plane (ANGLE) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.

**Sign Convention of Element Forces**

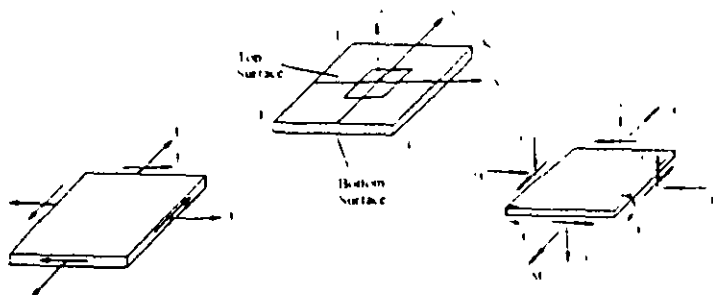


Figure I 13

Please note the following few restrictions in using the finite element portion of STAAD:

- 1) Both frame members and finite elements can be used together in a STAAD analysis. The ELEMENT INCIDENCES command must directly follow the MEMBER INCIDENCES input.
- 2) The selfweight of the finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.



- 3) Element forces are printed at the centroid and not along any edge.
- 4) In addition to the stresses shown in Fig I 13, the Von Mises stresses at the top and bottom surface of the element are also printed.

**Element Numbering**

During the generation of element stiffness matrix, the program verifies whether the element is same as the previous one or not. If it is same, repetitive calculations are not performed. The sequence in which the element stiffness matrix is generated is the same as the sequence in which elements are input in element incidences.

Therefore, to save some computing time, similar elements should be numbered sequentially. Fig. I 14 shows examples of efficient and non-efficient element numbering.

However the user has to decide between adopting a numbering system which reduces the computation time versus a numbering system which increases the ease of defining the structure geometry.

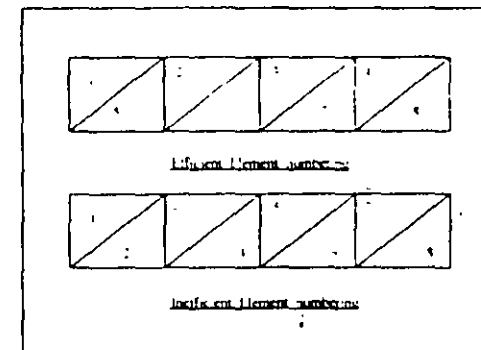


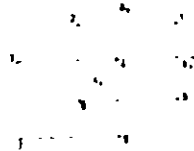
Figure I 14

### 1.6.2 Solid Element

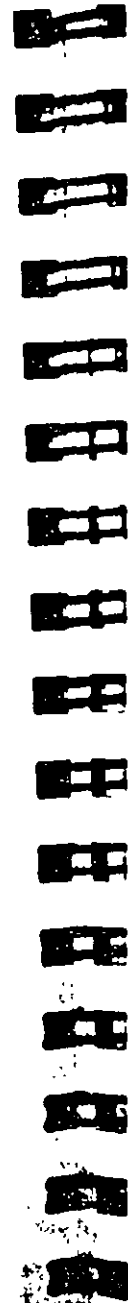
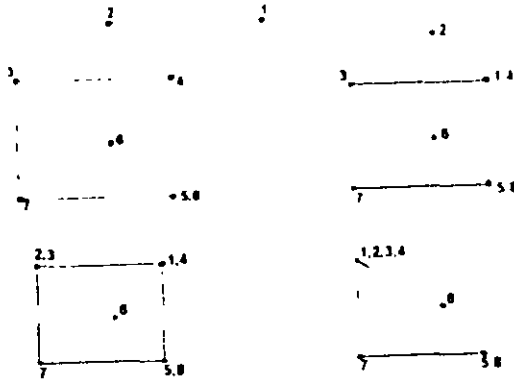
Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams, soil and rock strata where finite element analysis using solid elements provides a powerful tool

#### Theoretical Basis

The solid element used in STAAD is of eight noded isoparametric type. These elements have three translational degrees-of-freedom per node.



By collapsing various nodes together, an eight noded solid element can be degenerated to the following forms with four to seven nodes.



The stiffness matrix of the solid element is evaluated by numerical integration with eight Gauss-Legendre points. To facilitate the numerical integration, the geometry of the element is expressed by interpolating functions using natural coordinate system, (r,s,t) of the element with its origin at the center of gravity. The interpolating functions are shown below.

$$x = \sum_{i=1}^8 h_i x_i, \quad y = \sum_{i=1}^8 h_i y_i, \quad z = \sum_{i=1}^8 h_i z_i$$

where x, y and z are the coordinates of any point in the element and x<sub>i</sub>, y<sub>i</sub>, z<sub>i</sub>, i=1, 8 are the coordinates of nodes defined in the global coordinate system. The interpolation functions, h<sub>i</sub> are defined in the natural coordinate system, (r,s,t). Each of r,s and t varies between -1 and +1. The fundamental property of the unknown interpolation functions h<sub>i</sub> is that their values in natural coordinate system is unity at node, i, and zero at all other nodes of the element. The element displacements are also interpreted the same way as the geometry. For completeness, the functions are given below:

$$u = \sum_{i=1}^8 h_i u_i, \quad v = \sum_{i=1}^8 h_i v_i, \quad w = \sum_{i=1}^8 h_i w_i$$

where u, v and w are displacements at any point in the element and u<sub>i</sub>, v<sub>i</sub>, w<sub>i</sub>, i=1,8 are corresponding nodal displacements in the coordinate system used to describe the geometry.

## Local Coordinate System

The local coordinate system used in solid element is the same as the global system as shown below:

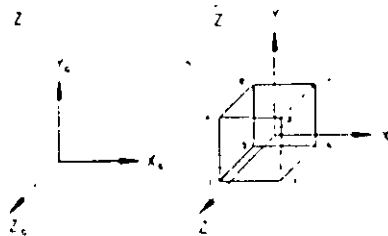


Figure 1.15

## Properties and Constants

Unlike members and shell (plate) elements, no properties are required for solid elements. However, the constants such as modulus of elasticity and Poisson's ratio are to be specified.

Also, Density needs to be provided if selfweight is included in any load case.

## Output of Element Stresses

Element stresses may be obtained at the center and at the joints of the solid element. The items that are printed are:

Normal Stresses	: SXX, SYY and SZZ
Shear Stresses	: SXY, SYZ and SZX
Principal stresses	: S1, S2 and S3.
Von Mises stresses	: SE
Direction cosines	: 6 direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions.

## 1.7 Member Properties

The following types of member property specifications are available in STAAD:

- PRISMATIC property specifications
- Standard Steel shapes from built-in section library
- User created steel tables
- TAPERED sections
- Through ASSIGN command

See section  
5.20

### 1.7.1 Prismatic Properties

The following prismatic properties are required for analysis:

- AX = Cross sectional area
- IX = Torsional constant
- IY = Moment of inertia about y-axis.
- IZ = Moment of inertia about z-axis

See section  
5.20.2

In addition, the user may choose to specify the following properties:

- AY = Effective shear area for shear force parallel to local y-axis
- AZ = Effective shear area for shear force parallel to local z-axis.
- YD = Depth of section parallel to local y-axis
- ZD = Depth of section parallel to local z-axis.

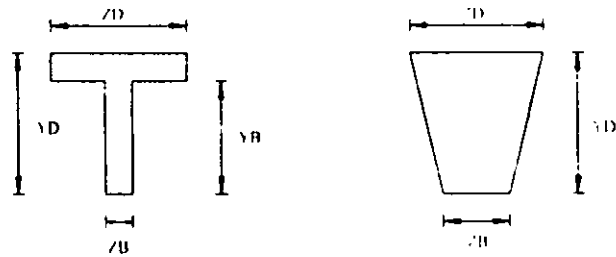
To specify T-beam or Trapezoidal beam, the following additional properties must be provided.

- YB = Depth of Web of T-section [See figure below]
- ZB = Width of web of T-section or bottom width of Trapezoidal section.

To specify T-beam, the user must specify YD, ZD, YB & ZB. Similarly for Trapezoidal sections, YD, ZD and YB must be provided.



If the shear areas are input, the program will automatically consider shear deformations in the analysis, and if they are left out, shear deformation will be ignored. In a frame structure, the ratio of shear deflection to bending deflection is so small that, in most cases, it can be ignored. The depths in the two major directions (YD and ZD) are used in the program to calculate the section moduli. These are needed only to calculate member stresses or to perform concrete design. The user can omit the YD & ZD values if stresses or design of these members are of no interest. The default value is 254 mm (10 inches) for YD and ZD. All the prismatic properties are input in the local member coordinates.



To define a concrete member, the user must not provide AX, but instead, provide YD and ZD for a rectangular section and just YD for a circular section. If no moment of inertia or shear areas are provided, the program will automatically calculate these from YD and ZD.

Table 1.1 is offered to assist the user in specifying the necessary section values. It lists, by structural type, the required section properties for any analysis. For the PLANE or FLOOR type analyses, the choice of the required moment of inertia depends upon the beta angle. If BETA equals zero, the required property is IZ.

Table 1.1 Required properties

Structural Type	Required Properties
TRUSS structure	AX
PLANE structure	AX, IZ or IY
FLOOR structure	IX, IZ or IY
SPACE structure	AX, IX, IY, IZ

## 1.7.2 Built-In Steel Section Library

This feature of the program allows the user to specify section names of standard steel shapes manufactured in different countries. Information pertaining to the American steel shapes is available in section 2.

See section  
2.2.1 and  
5.20.1

Contact Research Engineers for information on steel shapes for other countries.

Since the shear areas of the sections are built into the tables, shear deformation is always considered for these sections.

## 1.7.3 User Provided Steel Table

See sections  
5.19 and  
5.20.4

The user can provide a customized steel table with designated names and proper corresponding properties. The program can then find member properties from those tables. Member selection may also be performed with the program selecting members from the provided tables only.

These tables can be provided as a part of a STAAD input or as separately created files from which the program can read the properties. The user who does not use standard rolled shapes or who uses a limited number of specific shapes may create permanent member property files. Analysis and design can be limited to the sections in these files.



### 1.7.4 Tapered Sections

*See section 5.20.3* Properties of tapered I-sections may be provided through MEMBER PROPERTY specifications. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis. Specification of TAPERED sections is described in Section 5 of this manual.

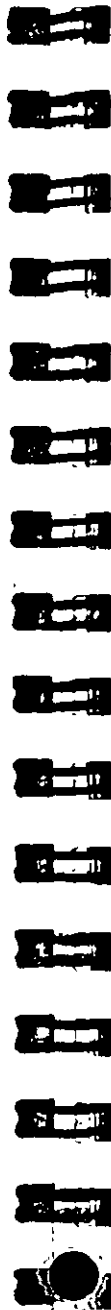
### 1.7.5 Assign Command

*See section 5.20.5* Through this command, the user may instruct the program to automatically select a steel section from the table for analysis and subsequent design. The section types that may be ASSIGNED include BEAM, COLUMN, CHANNEL, ANGLE and DOUBLE ANGLE. When a BEAM or COLUMN is specified, the program will assign an I-beam section (WF for AISC) and subsequent member selection or optimization will be performed with a similar type section.

### 1.8 Member/Element Release

STAAD allows releases for both members and elements

*See section 5.22* One or both ends of a member or element can be released. Members/Elements are assumed to be rigidly framed into joints in accordance with the structural type specified. When this full rigidity is not applicable, individual force components at either end of the member can be set to zero with member release statements. By specifying release components, individual degrees of freedom are removed from the analysis. Release components are given in the local coordinate system for each member. Note that PARTIAL moment release is also allowed.



### 1.9 Truss/Tension - Only Members

*See section 5.23* For analyses which involve members that carry axial loads only, i.e. truss members, there are two methods for specifying this condition. When all the members in the structure are truss members, the type of structure is declared as TRUSS whereas, when only some of the members are truss members (e.g. bracings of a building), the MEMBER TRUSS command can be used where those members will be identified separately. A considerable amount of analysis and design time is saved by declaring an axial member as a TRUSS member, specification of such members as frame members with both ends pinned should be avoided wherever possible.

In STAAD, a member may be declared as a Tension-only member. The analysis will be performed accordingly. Refer to Section 5.23.3 for details on this facility.

### 1.10 Cable Members

*See section 5.23* Cable members may be specified by using the MEMBER CABLE command. While specifying cable members, the initial tension in the cable must be provided. The following paragraph explains how cable stiffness is calculated.

The increase in length of a loaded cable is a combination of two effects. The first component is the elastic stretch, and is governed by the familiar spring relationship:

$$F = Kx \text{ where } K_{\text{elastic}} = \frac{EA}{L}$$

The second component of the lengthening is due to a change in geometry (as a cable is pulled taut, sag is reduced). This relationship can be described by

$$F = Kx \text{ but here } K_{\text{sag}} = \frac{12T^3}{w^2L^3}$$

$J = \text{Comba, deformación.}$   
(Catenaria)

where  $w = \text{weight per unit length of cable}$   
 $T = \text{tension in cable}$

Therefore, the "stiffness" of a cable depends on the initial installed tension (or sag). These two effects may be combined as follows

$$K_{\text{comb}} = \frac{1}{1/K_{\text{sag}} + 1/K_{\text{elastic}}}$$

$$K_{\text{comb}} = EA^{-1} \left[ 1 - w^2 L^2 EA^{-1} 12T^3 \right]^{-1}$$

Note. When  $T = 0$ ,  $K_{\text{comb}} = EA/L$

It may be noticed that as the tension increases (sag decreases) the combined stiffness approaches that of the pure elastic situation.

The following points need to be considered when using the cable member in STAAD

- 1) The cable member is only a truss member whose properties accommodate the sag factor and initial tension. The behavior of the cable member is identical to that of the truss member. It can carry axial loads only. As a result, the fundamental rules involved in modeling truss members have to be followed when modeling cable members. For example, when two cable members meet at a common joint, if there isn't a support or a 3rd member connected to that joint, it is a point of potential instability.
- 2) Due to the reasons specified in 1) above, applying a transverse load on a cable member is not advisable. The load will be converted to two concentrated loads at the 2 ends of the cable and the true deflection pattern of the cable will never be realized.

- 3) A cable member offers no resistance to a compressive force applied at its ends. When the end joints of the member are subjected to a compressive force, they "give in" thereby (Per me) causing the cable to sag. Under these circumstances, the cable member has zero stiffness and this situation has to be accounted for in the stiffness matrix and the displacements have to be recalculated. But in STAAD, merely declaring the member to be a cable member does not guarantee that this behavior will be accounted for. It is also important that the user declare the member to be a tension only member by using the MEMBER TENSION command. This will ensure that the program will test the nature of the force in the member after the analysis and if it is compressive, the member is switched off and the stiffness matrix re-calculated.

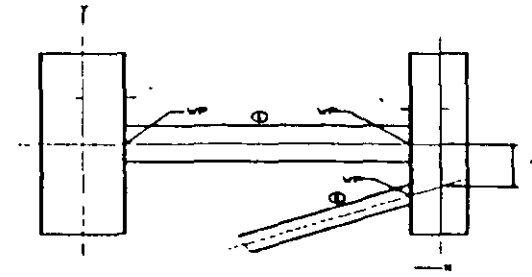
- 4) Due to potential instability problems explained in item 1 above, users should also avoid modeling a catenary by breaking it down into a number of straight line segments. The cable member in STAAD cannot be used to simulate the behavior of a catenary. By catenary, we are referring to those structural components which have a curved profile and develop axial forces due their self weight. This behavior is in reality a non-linear behavior where the axial force is caused because of either a change in the profile of the member or induced by large displacements, neither of which are valid assumptions in an elastic analysis. A typical example of a catenary is the main U shaped cable used in suspension bridges.

- 5) The increase of stiffness of the cable as the tension in it increases under applied loading is not accounted for during the analysis.

## 1.11 Member Offsets

See section  
3.25

Some members of a structure may not be concurrent with the incident joints thereby creating offsets. This offset distance is specified in terms of global coordinate system (i.e. global X, Y and Z distance from the incident joint). Secondary forces induced, due to this offset connection, are taken into account in analyzing the structure and also to calculate the individual member forces. The new offset centroid of the member can be at the start or end incidences and the new working point will also be the new start or end of the member. Therefore, any reference from the start or end of that member will always be from the new offset points.



**MEMBER OFFSET**  
 1 START 7  
 1 END -6  
 2 END -6 -9

Figure 1.15

## 1.12 Material Constants

See  
section 3.26

The material constants are: modulus of elasticity (E); weight density (DEN); Poisson's ratio (POISS), coefficient of thermal expansion (ALPHA), and beta angle (BETA) or coordinates for any reference (REF) point.

E value for members must be provided or the analysis will not be performed. Weight density (DEN) is used only when selfweight of the structure is to be taken into account. Poisson's ratio (POISS) is used to calculate the shear modulus (commonly known as G) by the formula.

$$G = 0.5 \times E / (1 + \text{POISS})$$

If Poisson's ratio is not provided G will be 1/2 E. Coefficient of thermal expansion (ALPHA) is used to calculate the expansion of the members if temperature loads are applied. The temperature unit for temperature load and ALPHA has to be the same.



BETA angle and REFERENCE point are discussed in Sec 1.5.3 and are input as part of the member constants.

## 1.13 Supports

STAAD allows specifications of supports that are parallel as well as inclined to the global axes.

Supports are specified as PINNED, FIXED, or FIXED with different releases. A pinned support has restraints against all translational movement and none against rotational movement. In other words, a pinned support will have reactions for all forces but will resist no moments. A fixed support has restraints against all directions of movement.

The restraints of a fixed support can also be released in any desired direction as specified in section 5.

Translational and rotational springs can also be specified. The springs are represented in terms of their spring constants. A translational spring constant is defined as the force to displace a support joint one length unit in the specified global direction. Similarly, a rotational spring constant is defined as the force to rotate the support joint one degree around the specified global direction.

## 1.14 Master/Slave Joints

The master/slave option is provided to enable the user to model rigid links in the structural system. This facility can be used to model special structural elements like a rigid floor diaphragm. Several slave joints may be provided which will be assigned same displacements as the master joint. The user is also allowed the flexibility to choose the specific degrees of freedom for which the displacement constraints will be imposed on the slaved joints. If all degrees of freedom ( $F_x$ ,  $F_y$ ,  $F_z$ ,  $M_x$ ,  $M_y$  and  $M_z$ ) are provided as constraints, the joints will be assumed to be rigidly connected.

## 1.15 Loads

Loads in a structure can be specified as joint load, member load, temperature load and fixed-end member load. STAAD can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self-weight can also be applied in any desired direction.

### 1.16.1 Joint Load

Joint loads, both forces and moments, may be applied to any free joint of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single joint, in which case the loads will be additive on that joint.

See section  
5.32.1

### 1.16.2 Member Load

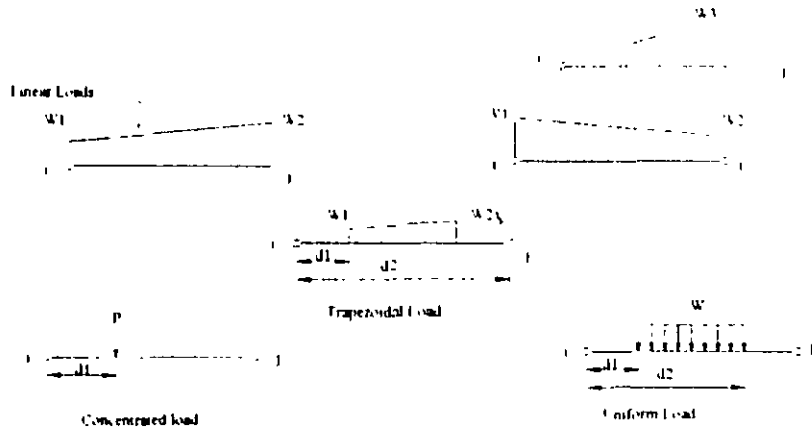
Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a member. Trapezoidal linearly varying loads act over the full or partial length of a member. Trapezoidal loads are converted into a uniform load and several concentrated loads.

See section  
5.32.2

Any number of loads may be specified to act upon a member in any independent loading condition. Member loads can be specified in the member coordinate system or the global coordinate system. Uniformly distributed member loads provided in the global coordinate system may be specified to act along the full or projected member length. Refer to Fig. 1.3 to find the relation of the member to the global coordinate systems for specifying member loads. Positive forces act in the positive coordinate directions, local or global, as the case may be.



See  
section 5.28



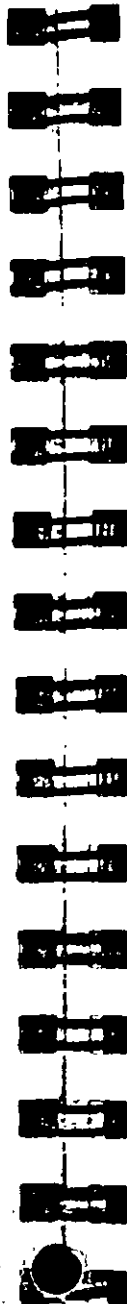
Member Load Configurations Figure 1.16

### 1.16.3 Area Load

Many times a floor (bound by X-Z plane) is subjected to a uniformly distributed load. It could require a lot of work to calculate the member load for individual members in that floor. However, with the AREA LOAD command, the user can specify the area loads (unit load per unit square area) for members. The program will calculate the tributary area for these members and provide the proper member loads. The following assumptions are made while transferring the area load to member load:

See section 5.32.4

- The member load is assumed to be a linearly varying load for which the start and the end values may be of different magnitude.
- Tributary area of a member with an area load is calculated based on half the spacing to the nearest approximately parallel members on both sides. If the spacing is more than or equal to the length of the member, the area load will be ignored.



c) Area load should not be specified on members declared as MEMBER CABLE, MEMBER TRUSS or MEMBER TENSION.

Figure 1.17 shows a floor structure with area load specification of 0.1.

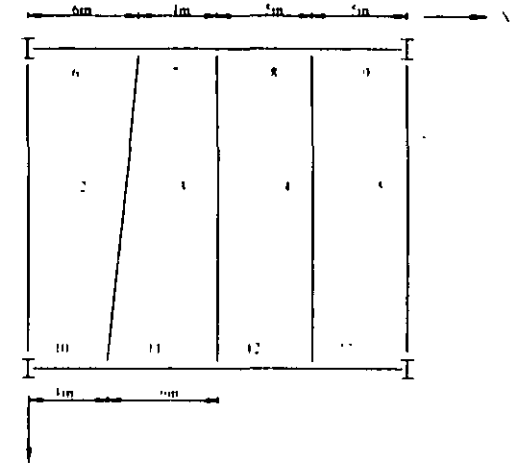


Figure 1.17

Member 1 will have a linear load of 0.3 at one end and 0.2 at the other end. Members 2 and 4 will have a uniform load of 0.5 over the full length. Member 3 will have a linear load of 0.45 and 0.55 at respective ends. Member 5 will have a uniform load of 0.25. The rest of the members, 6 through 13, will have no contributory area load since the nearest parallel members are more than each of the member lengths apart. However, the reactions from the members to the girder will be considered.

### 1.16.4 Fixed End Member Load

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load.

See section 5.32.7

on the member. Each end of a member can have six forces: axial; shear  $y$ , shear  $z$ , torsion; moment  $y$ , and moment  $z$ .

### 1.16.5 Prestress and Poststress Member Load

See section  
5.32.5

Members in a structure may be subjected to prestress load for which the load distribution in the structure may be investigated. The prestressing load in a member may be applied axially or eccentrically. The eccentricities can be provided at the start joint, at the middle, and at the end joint. These eccentricities are only in the local  $y$ -axis. A positive eccentricity will be in the positive local  $y$ -direction. Since eccentricities are only provided in the local  $y$ -axis, care should be taken when providing prismatic properties or in specifying the correct BETA angle when rotating the member coordinates, if necessary. Two types of prestress load specification are available; PRESTRESS, where, due to the load, reactions are generated during the application of prestress load and POSTSTRESS, assumed to be applied after the prestress load is in place, which does not generate reactions.

- 1) The cable is assumed to have a generalized parabolic profile. The equation of the parabola is assumed to be

$$y = ax^2 + bx + c$$

$$\text{where } a = \frac{1}{L^2}(2es - 4em + 2ee)$$

$$b = \frac{1}{L}(4em - ee - 3es)$$

$$c = es$$

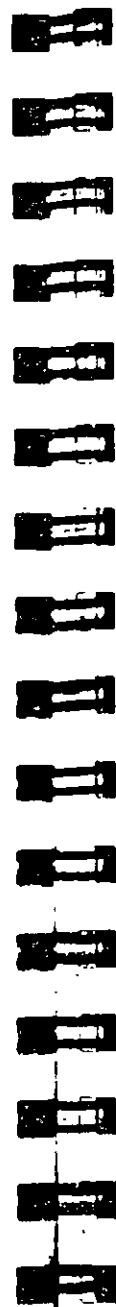
where  $es$  = eccentricity of cable at start of member (in local  $y$ -axis)

$em$  = eccentricity of cable at middle of member (in local  $y$ -axis)

$ee$  = eccentricity of cable at end of member (in local  $y$ -axis)

$L$  = Length of member

- 2) The angle of inclination of the cable with respect to the local  $x$ -axis (a straight line joining the start and end joints of the



member) at the start and end points is small which gives rise to the assumption that

$$\sin \theta = \theta = dy / dx$$

Hence, if the axial force in the cable is  $P$ , the vertical component of the force at the ends is  $P(dy/dx)$  and the horizontal component of the cable force is,

$$P \sqrt{1 - \left(\frac{dy}{dx}\right)^2}$$

Users are advised to ensure that their cable profile meets this requirement. An angle under 5 degrees is recommended.

- 3) The member is analyzed for the prestressing/poststressing effects using the equivalent load method. This method is well documented in most reputed books on Analysis and Design of Prestressed concrete. The magnitude of the uniformly distributed load is calculated as

$$udl = \frac{8Pe}{L^2}$$

where  $P$  = axial force in the cable

$$e = \frac{(es + ee)}{2} - em$$

$L$  = length of the member

- 4) The force in the cable is assumed to be same throughout the member length. No reduction is made in the cable forces to account for friction or other losses
- 5) The term MEMBER PRESTRESS as used in STAAD signifies the following condition. The structure is constructed first. Then, the prestressing force is applied on the relevant members. As a result, the members deform and depending on their end conditions, forces are transmitted to other members in the structure. In other words, "PRE" refers to the time of placement of the member in the structure relative to the time of stressing.

- 6) The term MEMBER POSTSTRESS as used in STAAD signifies the following condition. The members on which such load is applied are first cast in the factory. Following this, the prestressing force is applied on them. Meanwhile, the rest of the structure is constructed at the construction site. Then, the prestressed members are brought and placed in position on the partially built structure. Due to this sequence, the effects of prestressing are "experienced" by only the prestressed members and not transmitted to the rest of the structure. In other words, "POST" refers to the time of placement of the member in the structure relative to the time of stressing.
- 7) As may be evident from Item (6) above, it is not possible to compute the displacements of the ends of the POSTSTRESSED members for the effects of POSTSTRESSING, and hence are assumed to be zero. As a result, displacements of intermediate sections (See SECTION DISPLACEMENT command) are measured relative to the straight line joining the start and end joints of the members as defined by their initial JOINT COORDINATES.

### 1.16.6 Temperature/Strain Load

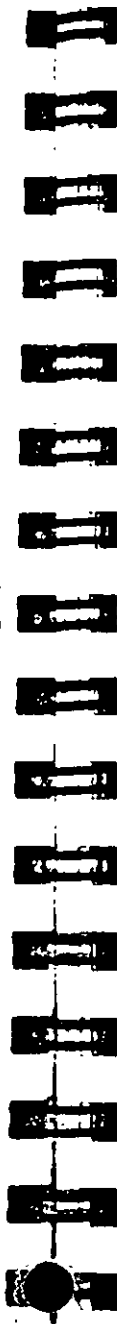
See section  
5.32.6

Temperature difference through the length of a member as well as differences of both faces of members and elements may also be specified. The program calculates the axial strain (elongation and shrinkage) due to the temperature difference. From this it calculates the induced forces in the member and the analysis is done accordingly. The strain intervals of elongation and shrinkage can be input directly.

### 1.16.7 Support Displacement Load

See section  
5.32.8

Loads can be applied to the structure in terms of the displacement of the supports. Displacement can be translational or rotational. Translational displacements are provided in the specified length while the rotational displacements are always in degrees. Note that displacements can be specified only in directions in which the support is restrained and not in directions in which it is released.



### 1.16.8 Loading on Elements

On Plate/Shell elements, the types of loading that are permissible are:

- 1) Pressure loading which consists of loads which act perpendicular to the surface of the element. The pressure loads can be of uniform intensity or trapezoidally varying intensity over a small portion or over the entire surface of the element.
- 2) Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.
- 3) Temperature loads which may be constant across the depth of the element (causing only in-plane elongation / shortening) or may vary across the depth of the element causing bending on the element. The coefficient of thermal expansion for the material of the element must be provided in order to facilitate computation of these effects.
- 4) The self-weight of the elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.

On Solid elements, the only two loading types available are

- 1) The self-weight of the solid elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided in order to facilitate computation of the self-weight.
- 2) Joint loads which are forces or moments that are applied at the joints in the direction of the global axes.

## 1.17 Load Generator

STAAD is equipped with built-in algorithms to generate moving loads and lateral seismic loads (per the Uniform Building Code and the IS 1893 code) on a structure. Use of the load generation facility consists of two parts:

- 1) Definition of the load system(s).
- 2) Generation of primary load cases using previously defined load system(s).

The following sections describe the salient features of the moving load generator, the seismic load generator and the wind load generator available.

### 1.17.1 Moving Load Generator

This feature enables the user to generate moving loads on a structure. Moving load system(s) consisting of concentrated loads at fixed specified distances in both directions on a plane can be defined by the user. A user specified number of primary load cases will be subsequently generated by the program and taken into consideration in analysis. American Association of State Highway and Transportation Officials (AASHTO, 1983) loadings are available within the program and can be specified using standard AASHTO designations.

See sections  
5.31.1 and  
5.32.12

### 1.17.2 UBC Seismic Load Generator

The STAAD seismic load generator follows the UBC procedure of equivalent lateral load analysis. It is assumed that the lateral loads will be exerted in X and Z directions and Y will be the direction of the gravity loads. Thus, for a building model, Y axis will be perpendicular to the floors and point upward (all Y joint coordinates positive). The user is required to set up his model accordingly. Total lateral seismic force or base shear is

See sections  
.31.2 and  
.32.12



automatically calculated by STAAD using the appropriate UBC equation. Note both 1994 and 1985 specifications may be used.

For load generation per the 1994 code, the user is required to provide seismic zone coefficient, importance factor, co-efficient  $R_w$  and site soil coefficient. For UBC 1985 load generation, seismic zone coefficient, importance factor & horizontal force factor  $k$  must be provided. Specification of site character period  $T_s$  is optional.

Instead of using approximate UBC formulas to estimate the building period in a certain direction, the program calculates the period using Raleigh quotient technique. This period is then utilized to calculate seismic coefficient  $C$ .

After the base shear is calculated from the appropriate equation, it is distributed among the various levels and roof per UBC specifications. The distributed base shears are subsequently applied as lateral loads on the structure. These loads may then be utilized as normal load cases for analysis and design.

### 1.17.3 Wind Load Generator

The STAAD Wind Load generator is capable of calculating wind loads on the structure from user specified wind intensities and exposure factors. Different wind intensities may be specified for different height zones of the structure. Openings in the structure may be modeled using exposure factors. An exposure factor is associated with each joint of the structure and is defined as the fraction of the influence area on which the wind load acts. Built-in algorithms automatically calculate the wind load on a SPACE structure and distribute the loads as lateral joint loads.

See sections  
5.31.3 and  
5.32.12

## 1.18 Analysis Facilities

Following analysis facilities are available in STAAD

- 1) Stiffness Analysis
- 2) Second Order Analysis
  - P-Delta Analysis
  - Non-Linear Analysis
- 3) Dynamic Analysis

Salient features of each type of analysis are discussed in the following sections. Detailed theoretical treatments of these features are available in standard structural engineering textbooks.

### 1.18.1 Stiffness Analysis

See section  
5.37

The stiffness analysis implemented in STAAD is based on the matrix displacement method. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural components (frame members or finite elements). Each component has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibility at the joints.

Structural systems such as slabs, plates, spread footings, etc., which transmit loads in 2 directions have to be discretized into a number of 3 or 4 noded finite elements connected to each other at their nodes. Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the joints. The plane stress effects as well as the plate bending effects are taken into consideration in the analysis.

#### Assumptions of the Analysis

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

- 1) The structure is idealized into an assembly of beam and plate type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be both forces and moments which may act in any specified direction.

- 2) A beam member is a longitudinal structural member having a constant, doubly symmetric or near-doubly symmetric cross section along its length. Beam members always carry axial forces. They may also be subjected to shear and bending in two arbitrary perpendicular planes, and they may also be subjected to torsion. From this point these beam members are referred to as 'members' in the manual.
- 3) A plate element is a three or four noded element having constant thickness. These plate elements are referred to as 'elements' in the manual.
- 4) Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any member, six degrees of freedom are considered at each node (i.e. three translational and three rotational) in the generation of relevant matrices. If the member is defined as truss member (i.e. carrying only axial forces) then only the three degrees (translational) of freedom are considered at each node.
- 5) Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems.

Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. Global coordinate axes are a common datum established for all idealized elements so that element forces and displacements may be related to a common frame of reference.

#### Basic Equation

The complete stiffness matrix of the structure is obtained by systematically summing the contributions of the various member and element stiffness. The external loads on the structure are represented as discrete concentrated loads acting only at the nodal points of the structure.

The stiffness matrix relates these loads to the displacements of the nodes by the equation:

$$A_j = a_j + S_j \times D_j$$

This formulation includes all the joints of the structure, whether they are free to displace or are restrained by supports. Those components of joint displacements that are free to move are called degrees of freedom. The total number of degrees of freedom represent the number of unknowns in the analysis

### Method to Solve for Displacements

There are many methods to solve the unknowns from a series of simultaneous equations. An approach which is particularly suited for structural analysis is called the method of decomposition. This method has been selected for use in STAAD. Since the stiffness matrices of all linearly elastic structures are always symmetric, an especially efficient form of the decomposition called Modified Cholesky's method may be applied to these problems. This method is very accurate and cost effective and well suited for the Gaussian elimination process in solving the simultaneous equations.

### Consideration of Bandwidth

The method of decomposition is particularly efficient when applied to a symmetrically banded matrix. For this type of matrix fewer calculations are required due to the fact that elements outside the band are all equal to zero.

STAAD takes full advantage of this bandwidth during solution, as it is important to have the least bandwidth to obtain the most efficient solution. For this purpose, STAAD offers features by which the program can internally rearrange the joint numbers to provide a better bandwidth.

### Structural Integrity

The integrity of the structure is a very important requirement that must be satisfied by all models. Users must make sure that the



model developed represents one single structure only, not two or more separate structures.

An "integral" structure or "one" structure may be defined as a system in which proper "stiffness connections" exist between the members/elements. The entire model functions as a single integrated load resisting system. Two or more independent structures within one model results in erroneous mathematical formulation and therefore, generates numerical problems. STAAD checks structural integrity using a sophisticated algorithm and reports detection of multiple structures within the model.

### Modeling and Numerical Instability Problems

Instability problems can occur due to two primary reasons.

#### 1) Modeling problem

There are a variety of modeling problems which can give rise to instability conditions. They can be classified into two groups.

- a) Local instability - A local instability is a condition where the fixity conditions at the end(s) of a member are such as to cause an instability in the member about one or more degrees of freedom. Examples of local instability are:
  - (i) Member Release: Members released at both ends for any of the following degrees of freedom (FX, FY, FZ, and MX) will be subjected to this problem.
  - (ii) A framed structure with columns and beams where the columns are defined as "TRUSS" members. Such a column has no capacity to transfer shears or moments from the superstructure to the supports.
- b) Global Instability - These are caused when the supports of a structure are such that they cannot offer any resistance to sliding or overturning of the structure in one or more directions. For example, a 2D structure (frame in the XY

plane) which is defined as a SPACE FRAME with pinned supports and subjected to a force in the Z direction will topple over about the X-axis. Another example is that of a space frame with all the supports released for FX, FY or FZ.

## 2) Math precision

A math precision error is caused when numerical instabilities occur in the matrix inversion process. One of the terms of the equilibrium equation takes the form  $1/(1-A)$ , where  $A = k1/(k1+k2)$ ;  $k1$  and  $k2$  being the stiffness coefficients of two adjacent members. When a very "stiff" member is adjacent to a very "flexible" member, viz., when  $k1 \gg k2$ , or  $k1+k2 \cong k1$ ,  $A=1$  and hence,  $1/(1-A) = 1/0$ . Thus, huge variations in stiffnesses of adjacent members are not permitted.

Math precision errors are also caused when the units of length and force are not defined correctly for member lengths, member properties, constants etc.

Users also have to ensure that the model defined represents one single structure only, not two or more separate structures. For example, in an effort to model an expansion joint, the user may end up defining separate structures within the same input file. Multiple structures defined in one input file can lead to grossly erroneous results.

## 1.18.2 Second Order Analysis

STAAD offers the capability to perform second order stability analyses. Two methods are available - a simplified method called P-Delta Analysis and an elaborate method called Non Linear Analysis. Both methods are explained below.

See  
section 5.37

### 1.18.2.1 P-Delta Analysis

Structures subjected to lateral loads often experience secondary forces due to the movement of the point of application of vertical loads. This secondary effect, commonly known as the P-Delta

See

section 5.37



effect, plays an important role in the analysis of the structure. In STAAD, a unique procedure has been adopted to incorporate the P-Delta effect into the analysis. The procedure consists of the following steps:

- 1) First, the primary deflections are calculated based on the provided external loading.
- 2) Primary deflections are then combined with the originally applied loading to create the secondary loadings. The load vector is then revised to include the secondary effects.

Note that the lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. The REPEAT LOAD facility (see Section 5.32.11) has been created with this requirement in mind. This facility allows the user to combine previously defined primary load cases to create a new primary load case.

- 3) A new stiffness analysis is carried out based on the revised load vector to generate new deflections.
- 4) Element/Member forces and support reactions are calculated based on the new deflections.

It may be noted that this procedure yields very accurate results with all small displacement problems. STAAD allows the user to go through multiple iterations of the P-Delta procedure if necessary. The user is allowed to specify the number of iterations based on the requirement.

The P-Delta analysis is recommended by the ACI code (in lieu of moment magnification methods) and the AISC LRFD code for calculation of more realistic forces and moments.

P-Delta effects are calculated for frame members only. They are not calculated for finite elements or solid elements.



### 1.18.2.2 Non Linear Analysis

STAAD also offers the capability to perform non-linear analysis based on geometric non-linearity. The non-linear analysis algorithm incorporates both geometric stiffness corrections and secondary loadings.

See  
section 5.37

Non linear analysis methodology is generally adopted for structures subject to large displacements. As large displacements generally result in significant movement of the point of application of loads, consideration of secondary loadings becomes an important criteria. In addition, geometric stiffness corrections are applied to take into consideration the modified geometry. Since the geometric stiffness corrections are based on generated displacements, they are different for different load cases. This makes the non-linear analysis option load dependent. The STAAD non-linear analysis algorithm consists of the following steps:

- 1) First, primary displacements are calculated for the applied loading.
- 2) Stiffness corrections are applied on the member/element stiffness matrices based on observed displacements. New global stiffness matrix is assembled based on revised member/element stiffness matrices.
- 3) Load vectors are revised to include the secondary effects due to primary displacements.
- 4) The new set of equations are solved to generate new displacements.
- 5) Element/Member forces and support reactions are calculated from these new displacements.
- 6) The STAAD non-linear analysis algorithm allows the user to go through multiple iterations of the above procedure. The number of iterations may be specified by the user based on the requirement. It may be noted, however, that multiple iterations



may increase the computer resource requirements and execution time substantially.

Note: The following points may be noted with respect to the non-linear analysis facility:

- 1) Since the procedure is load dependent, the user is required to use the SET NL and CHANGE commands properly. The SET NL command must be provided to specify the total number of primary load cases. The CHANGE command should be used to reset the stiffness matrices.
- 2) As the geometric corrections are based on displacements, all loads that are capable of producing significant displacements must be part of the load case(s) identified for non-linear analysis.

### 1.18.3 Dynamic Analysis

Currently available dynamic analysis facilities include solution of the free vibration problem (eigenproblem), response spectrum analysis and forced vibration analysis.

#### Solution of the Eigenproblem

See sections  
5.30  
5.32.10, 5.34

The eigenproblem is solved for structure frequencies and mode shapes considering a lumped mass matrix, with masses at all active d.o.f. included. Two solution methods are available: the determinant search method, and the subspace iteration method, with solution selection based on problem size.

#### Mass Modeling

The natural frequencies and mode shapes of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forcing function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. Results of the frequency and mode shape calculations may vary significantly

depending upon the mass modeling. This variation, in turn, affects the response spectrum and forced vibration analysis results. Thus, extreme caution should be exercised in mass modeling in a dynamic analysis problem. Active masses should be modeled as loads. All masses that are capable of moving should be modeled as loads applied in all possible directions of movement. In response spectrum analysis, as a bare minimum, all masses that are capable of moving in the direction of the spectrum, must be provided as loads acting in that direction

**Response Spectrum Analysis**

This capability allows the user to analyze the structure for seismic loading. For any supplied response spectrum (either acceleration vs. period or displacement vs. period), joint displacements, member forces, and support reactions may be calculated. Modal responses may be combined using either the square root of the sum of squares (SRSS) or the complete quadratic combination (CQC) method to obtain the resultant responses. Results of the response spectrum analysis may be combined with the results of the static analysis to perform subsequent design. To account for reversibility of seismic activity, load combinations can be created to include either the positive or negative contribution of seismic results

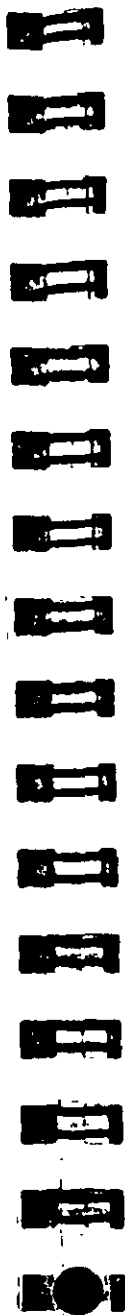
See section 5.32.10

**Response Time History Analysis**

STAAD is equipped with a facility to perform a response history analysis on a structure subjected to time varying forcing function loads at the joints and/or a ground motion at its base. This analysis is performed using the modal superposition method. Hence, all the active masses should be modeled as loads in order to facilitate determination of the mode shapes and frequencies. Please refer to the section above on "mass modeling" for additional information on this topic. In the mode superposition analysis, it is assumed that the structural response can be obtained from the "p" lowest modes. The equilibrium equations are written as

See Sections 5.31.4 and 32.10.2

$$[m]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = \{P\} \dots \dots (1)$$



Using the transformation

$$\{x\} = \sum_{i=1}^p \phi_i q_i \dots \dots (2)$$

Equation 1 reduces to "p" separate uncoupled equations of the form

$$q_i + 2\zeta_i \omega_i \dot{q}_i + \omega_i^2 q_i = R_i(t)$$

where  $\zeta_i$  is the modal damping ratio and  $\omega_i$  the natural frequency for the  $i^{th}$  mode.

These are solved by the Wilson-  $\theta$  method which is an unconditionally stable step by step scheme. The time step for the response is chosen as 0.1 T where T is the period of the highest mode that is to be included in the response. The  $q_i$ s are substituted in equation 2 to obtain the displacements  $\{x\}$  at each time step

**Time History Analysis for a Structure Subjected to a Harmonic Loading**

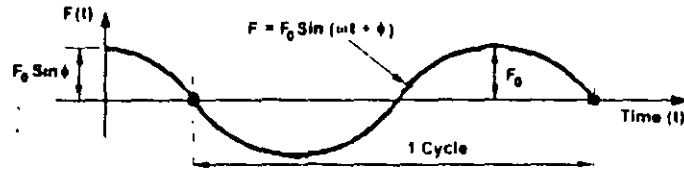
A Harmonic loading is one in which can be described using the following equation

$$F(t) = F_0 \sin(\omega t + \phi)$$

In the above equation,

- F(t) = Value of the force at any instant of time "t"
- F<sub>0</sub> = Peak value of the force
- ω = Frequency of the forcing function
- φ = Phase Angle

A plot of the above equation is shown in the figure below



**Definition of Input in STAAD for the Above Forcing Function**

As can be seen from its definition, a forcing function is a continuous function. However, in STAAD, a set of discrete time-force pairs is generated from the forcing function and an analysis is performed using these discrete time-forcing pairs. What that means is that based on the number of cycles that the user specifies for the loading, STAAD will generate a table consisting of the magnitude of the force at various points of time. The time values are chosen from this time '0' to  $n \cdot t_c$  in steps of "STEP" where  $n$  is the number of cycles and  $t_c$  is the duration of one cycle. STEP is a value that the user may provide or may choose the default value that is built into the program. Users may refer to section 5.31.4 of this manual for a list of input parameters that need to be specified for a Time History Analysis on a structure subjected to a Harmonic loading.

The relationship between variables that appear in the STAAD input and the corresponding terms in the equation shown above is explained below.

- $F_0$  = AMPLITUDE
- $\omega$  = FREQUENCY
- $\phi$  = PHASE

**1.19 Member End Forces**

Member end forces and moments in the member result from loads applied to the structure. These forces are in the local member

See section 5.41

coordinate system. Figure 1.18 shows the member end actions with their directions.

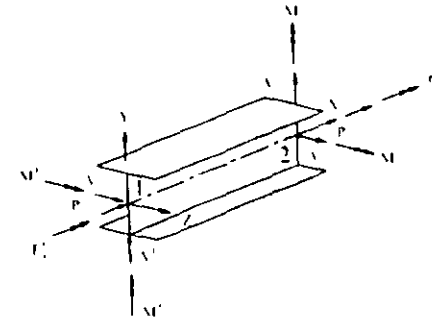


Figure 1.18

**1.19.1 Secondary Analysis**

See sections 5.40, 5.41, 5.42 and 5.43

Solution of the stiffness equations yield displacements and forces at the joints or end points of the member. STAAD is equipped with the following secondary analysis capabilities to obtain results at intermediate points within a member.

- 1) Member forces at intermediate sections
- 2) Member displacements at intermediate sections.
- 3) Member stresses at specified sections
- 4) Force envelopes.

The following sections describe the secondary analysis capabilities in detail.

**1.19.2 Member Forces at Intermediate Sections**

See sections 5.40 and 5.41

With the SECTION command, the user may choose any intermediate section of a member where forces and moments need to be calculated. These forces and moments may also be used in design of the members. The maximum number of sections specified may not exceed five, including one at the start and one at the end of a member. If no intermediate sections are requested, the

program will consider the start and end member forces for design. However, of the sections provided, they are the only ones to be considered design.

### 1.19.3 Member Displacements at Intermediate Sections

See sections  
5.42 and  
5.45.2

Like forces, displacements of intermediate sections of members can be printed or plotted. This command may not be used for truss or cable members.

### 1.19.4 Member Stresses at Specified Sections

Member stresses can be printed at specified intermediate sections as well as at the start and end joints. These stresses include

See  
sections  
5.40 and  
5.41

- Axial stress, which is calculated by dividing the axial force by the cross sectional area.
- Bending-y stress, which is calculated by dividing the moment in local-y direction by the section modulus in the same direction.
- Bending-z stress, which is the same as above except in local-z direction.
- Shear stresses (in y and z directions), and
- Combined stress, which is the sum of axial, bending-y and bending-z stresses.

All the stresses are calculated as the absolute value.

### 1.19.5 Force Envelopes

See section  
43

Force envelopes of the member forces FX (axial force), FY (Shear-y), and MZ (moment around local z-axis, i.e. strong axis) can be printed for any number of intermediate sections. The force values include maximum and minimum numbers representing maximum positive and maximum negative values. The following is the sign convention for the maximum and minimum values:



FX A positive value is compression, and negative tension

FY A positive value is shear in the positive y-direction, and negative in the negative y-direction.

FZ Same as above, except in local z-direction.

MZ A positive moment will mean a moment causing tension at the top of the member. Conversely, a negative moment will cause tension at the bottom of the member. The top of a member is defined as the side towards positive local y-axis

MY Same as above, except about local z axis.

## 1.20 Multiple Analyses

Structural analysis/design may require multiple analyses in the same run. STAAD allows the user to change input such as member properties, support conditions etc. in an input file to facilitate multiple analyses in the same run. Results from different analyses may be combined for design purposes.

For structures with bracing, it may be necessary to make certain members inactive for a particular load case and subsequently activate them for another. STAAD provides an INACTIVE facility for this type of analysis. The INACTIVE option is discussed in detail in the following paragraph.

### Inactive Members

With the INACTIVE command, members can be made inactive. These inactive members will not be considered in the stiffness analysis or in any printout. The members made inactive by the INACTIVE command are made active again with the CHANGE command. This can be useful in an analysis where tension-only bracing is desired, so a set of members should be inactive for certain load cases. This can be accomplished by:

See section  
5.18

- making the desired members inactive;

- b) providing the relevant load cases for which the members are inactive;
- c) performing the analysis;
- d) using the CHANGE command to make all the inactive members active;
- e) and making the other set of members inactive and providing the proper load cases for which the members are meant to be inactive, performing the analysis and repeating the procedure as necessary

## 1.21 Steel/Concrete/Timber Design

*See sections 2, 3 and 4*  
Extensive design capabilities are available in STAAD for steel, concrete and timber sections. Detailed information on steel, concrete and timber design is presented in Sections 2, 3 and 4 respectively.

## 1.22 Footing Design

*See section 5.52*

A footing design facility capable of designing individual footings for user specified support(s) is available. All active load cases are checked and design is performed for the support reaction(s) which requires the maximum footing size. Parameters are available to control the design. Output includes footing dimensions and reinforcement details. Dowel bars and development lengths are also calculated and included in the design output. Detailed description and command specification(s) for footing design is available in section 5.52 of this manual.

## 1.23 Printing Facilities

All input data and output may be printed using PRINT commands available in STAAD. The input is normally echoed back in the output. This is important from a documentation point of view. However, if required, the echo can be switched off.



Extensive listing facilities are provided in almost all PRINT commands to allow the user to select joints or members (elements) for which values are required.

## 1.24 Plotting Facilities

*See sections 5.29, and 5.45*

Two types of plotting facilities are available in STAAD. The first type allows the user to view structure geometry, deflected shape, bending moment/shear force diagrams, stress contours etc. using the Post Processing module. For certain capabilities (like deflected shape, bending moment diagrams etc.) a PLOT file containing the relevant info must first be created through STAAD.

In addition to the graphics capabilities of the Post Processing module, a wide range of PRINTER PLOT options are available in STAAD. These capabilities may be utilized to generate PRINTER PLOTS of structure geometry, deflected shapes, bending moment diagrams etc. as part of the output.

## 1.25 Miscellaneous Facilities

STAAD offers the following miscellaneous facilities for problem solution.

### Perform Rotation

*See section 5.17*

After the geometry has been specified, this command can be used to rotate the structure shape through any desired angle about any global axis. The rotated configuration can be used for further analysis and design.

### Substitute

*See section 5.15*

Joint and member numbers may be redefined in STAAD through the use of the SUBSTITUTE command. After a new set of numbers are assigned, input and output values will be in accordance with the new numbering scheme. This facility allows the user to specify

numbering schemes that will result in simple input specification as well as easy interpretation of data.

### Calculation of Center of Gravity

See section  
5.41

STAAD is capable of calculating the center of gravity of the structure. The PRINT CG command may be utilized for this purpose.

### Print Problem Statistics

See  
section 5.36

The user may take advantage of this facility to review analysis related characteristics (size of the stiffness matrix, disk storage requirement etc.) of the problem before actually running the job. This option is especially useful for the estimation of storage requirements before running a large problem which may require large amounts of storage.

### Input Memory

See section  
5.53.2

This option may be used for problems running on the PC requiring a large amount of memory. It should be remembered that the use of this option may result in slow program execution.

## 1.26 Post Processing Facilities

All output from the STAAD run may be utilized for further processing either by other modules or external programs. Files containing relevant information must be created through STAAD for this purpose. The following options are available.

### Save/Restore

See sections  
5.54 and  
5.55

The save/restore features enable the user to save all the data and results associated with a problem and reactivate (restore) the problem and resume processing at a later time.



## American Steel Design

## Section 2

### 2.1 Design Operations

STAAD contains a broad set of facilities for designing structural members as individual components of an analyzed structure. The member design facilities provide the user with the ability to carry out a number of different design operations. These facilities may be used selectively in accordance with the requirements of the design problem. The operations to perform a design are:

Specify the members and the load cases to be considered in the design.

Specify whether to perform code checking or member selection.

Specify design parameter values, if different from the default values.

These operations may be repeated by the user any number of times depending upon the design requirements.

Steel Design may be performed based on the following codes: AISC-ASD, AISC-LRFD and AASHTO. A brief description of each is presented in the following pages.

Currently STAAD supports steel design of wide flange, S, M, HP shapes, angle, double angle, channel, double channel, beams with

cover plate, composite beams and code checking of prismatic properties.

## 2.2 Member Properties

For specification of member properties of standard American steel sections, the steel section library available in STAAD may be used. The syntax for specifying the names of built-in steel shapes is described in the next section.

### 2.2.1 Built-in Steel Section Library

The following sections describe specification of steel sections from the AISC (9th Edition, 1989) Steel Tables.

#### AISC Steel Table

Almost all AISC steel shapes are available for input. Following are the descriptions of all the types of sections available:

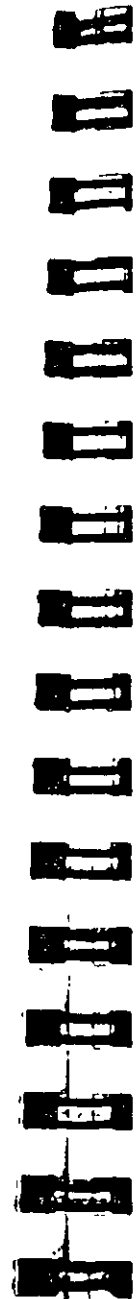
#### Wide Flanges (W shapes)

All wide flange sections as listed in AISC/LRFD-89 are available the way they are written, e.g. W10X49, W21X50, etc.

**20 TO 30 TA ST W10X49**  
**33 36 TA ST W18X86**

#### C, MC, S, M, HP Shapes

The above shapes are available as listed in AISC (9th Edition) without decimal points. For example, C8X11.5 will be input as C8X11 and S15X42.9 will be input as S15X42, omitting the decimal weights. (Exception: MC6X151 for MC6X15.1 and MC6X153 for MC6X15.3.)



**10 TO 20 BY 2 TA ST C15X40**  
**1 2 TA ST MC8X20**

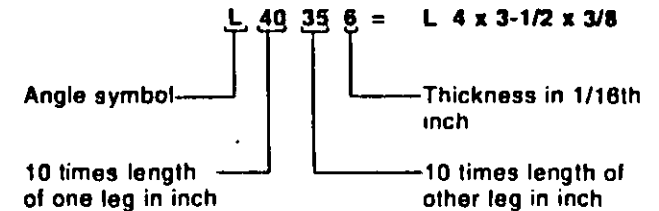
#### Double Channels

Back to back double channels, with or without spacing between them, are available. The letter D in front of the section name will specify a double channel.

**21 22 24 TA D MC9X25**  
**55 TO 60 TA D C8X18**

#### Angles

Angle specifications in STAAD are different from those in the AISC manual. The following example illustrates angle specifications.



Similarly, L505010 = L 5 x 5 x 5/8 and L904016 = L 9 x 4 x 1

At present, there are two ways to define the local y and z-axes for an angle section. To make the transition from the AISC Manual to the program data easy, the standard section for an angle is specified:

**51 52 53 TA ST L40356**

This specification has the local z-axis (i.e., the minor axis) corresponding to the Z-Z axis specified in the steel tables. Many

engineers are familiar with a convention used by some other programs in which the local y-axis is the minor axis. STAAD provides for this convention by accepting the command:

**54 55 56 TA RA L40356** (RA denotes reverse angle)

**Double Angles**

Short leg back to back or long leg back to back double angles can be specified by inputting the word SD or LD, respectively, in front of the angle size. In case of an equal angle either LD or SD will serve the purpose.

**14 TO 20 TA LD L35304 SP 0.5** Long leg back to back  
L3-1/2x3x1/4 with 0.5 space

**23 27 TA SD L904012** Short leg back to back  
L9x4x3/4

**Tees**

Tees are not input by their actual names, as they are listed in the AISC manual, but instead by designating the beam shapes (W and S) from which they are cut. For example,

**125 8 TAT WBX24** tee cut from WBX24 which is WT4X12

**Pipes**

Two types of specifications can be used for pipe sections. In general pipes may be input by their outer and inner diameters. For example,

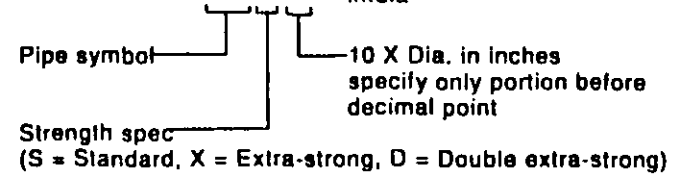
**1 TO 9 TA ST PIPE OD 2.0 ID 1.875** will mean a pipe with O.D. of 2.0 and I.D. of 1.875 in current input units.



Pipe sections listed in the AISC manual can be specified as follows.

**5 TO 10 TA ST PIPX20**

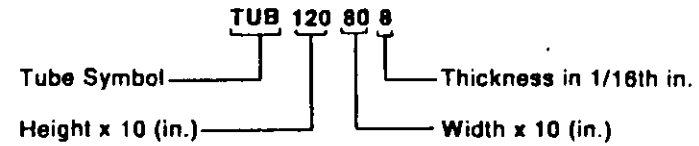
**PIP X 20** denotes extra strong pipe of 2 in. dia



**Tubes**

Tubes from the AISC tables can be specified as follows

**5 TO 10 TA ST TUB120808**



Tubes, like pipes, can be input by their dimensions (Height, Width and Thickness) as follows.

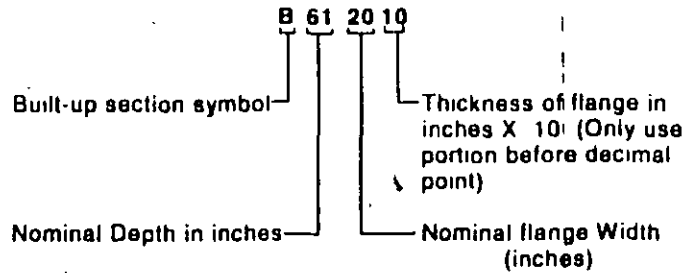
**6 TA ST TUBE DT 8.0 WT 6.0 TH 0.5** is a tube that has a height of 8, a width of 6, and a wall thickness of 0.5.

Member Selection cannot be performed on tubes specified in the latter way. Only code checking can be performed on these sections.

**Welded Plate Girders**

Welded plate girders from the AISC manual may be specified as follows.





Example

I TO 10 TA ST B612017  
 15 16 TA ST B682210

## 2.3 Allowables per AISC Code

For steel design, STAAD compares the actual stresses with the allowable stresses as defined by the American Institute of Steel Construction (AISC) Code. The ninth edition of the AISC Code, as published in 1989, is used as the basis of this design (except for tension stress). Because of the size and complexity of the AISC codes, it would not be practical to describe every aspect of the steel design in this manual. Instead, a brief description of some of the major allowable stresses are described herein.

### 2.3.1 Tension Stress

Allowable tensile stress on the net section is calculated as  $0.60 F_y$ .

### 2.3.2 Shear Stress

Allowable shear stress on the gross section,

$$F_v = 0.4F_y$$

For shear on the web, the gross section is taken as the product of the total depth and the web thickness. For shear on the flanges, the gross section is taken as 2/3 times the total flange areas.



### 2.3.3 Stress Due To Compression

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the formula E-1 in the AISC Code, when the largest effective slenderness ratio ( $Kl/r$ ) is less than  $C_c$ . If  $Kl/r$  exceeds  $C_c$ , allowable compressive stress is decreased as per formula E2-2 of the Code.

$$C_c = \sqrt{2\pi^2 E / F_y}$$

### 2.3.4 Bending Stress

Allowable bending stress for tension and compression for a symmetrical member loaded in the plane of its minor axis, as given in Section 1.5.1.4 is:

$$F_b = 0.66F_y$$

If meeting the requirements of this section of:

- $b_f / 2t_f$  is less than or equal to  $65/\sqrt{F_y}$
- $b_f / t_f$  is less than or equal to  $190/\sqrt{F_y}$
- $d/t$  is less than or equal to  $640(1 - 3.74(F_y / F_y)) / \sqrt{F_y}$  when  $(F_y / F_y) < 0.16$ , or than  $257/\sqrt{F_y}$  if  $(F_y / F_y) > 0.16$
- The laterally unsupported length shall not exceed  $76.0b_f / F_y$  (except for pipes or tubes), nor  $20,000/(dF_y / A_f)$
- The diameter-thickness ratio of pipes shall not exceed  $3300/F_y$

If for these symmetrical members,  $b_f / 2t_f$  exceeds  $65/\sqrt{F_y}$ , but is less than  $95/\sqrt{F_y}$ ,  $F_b = F_y(0.79 - 0.002(b_f / 2t_f) \sqrt{F_y})$

For other symmetrical members which do not meet the above,  $F_b$  is calculated as the larger value computed as per AISC formulas F1-6 or F1-7 and F1-8 as applicable, but not more than  $0.60F_y$ . An unstiffened member subject to axial compression or compression due to bending is considered fully effective when the width-thickness ratio is not greater than the following:

- 76.0/ $\sqrt{F}$ , for single angles or double angles with separators.
- 95.0/ $\sqrt{F}$ , for double angles in contact.
- 127/ $\sqrt{F}$ , for stems of tees.

When the actual width-thickness ratio exceeds these values, the allowable stress is governed by B5 of the AISC code

Tension and compression for the doubly symmetric (I & H) sections with  $b_f/2t_f$  less than  $65/\sqrt{F}$  and bent about their minor axis,  $F_b = 0.75F_y$ . If  $b_f/2t_f$  exceeds  $65/\sqrt{F}$ , but is less than  $95/\sqrt{F}$ ,  $F_b = F_y(1 - 0.005(b_f/2t_f)\sqrt{F})$

For tubes, meeting the subparagraphs b and c of this Section, bent about the minor axis,  $F_b = 0.66F_y$ , failing the subparagraphs b and c but a width-thickness ratio less than  $238/\sqrt{F}$ ,  $F_b = 0.6F_y$

### 2.3.5 Combined Compression and Bending

Members subjected to both axial compression and bending stresses are proportioned to satisfy AISC formula H1-1 and H1-2 when  $f_a/F_a$  is greater than 0.15, otherwise formula H1-3 is used. It should be noted that during code checking or member selection, if  $f_a/F_a$  exceeds unity, the program does not compute the second and third part of the formula H1-1, because this would result in a misleadingly liberal ratio. The value of the coefficient  $C_m$  is taken as 0.85 for sidesway and  $0.6 - 0.4(M1/M2)$ , but not less than 0.4 for no sidesway.

### 2.3.6 Singly Symmetric Sections

For double angles and Tees which have only one axis of symmetry, the  $KL/r$  ratio about the local Y-Y axis is determined using the clauses specified on page 3-53 of the AISC Manual.

### 2.3.7 Torsion per Publication T114

The AISC 89 code of specifications for steel design currently does not have any provisions specifically meant for design of sections for Torsion. However, AISC has published a separate document called "Torsional Analysis of Steel Members" which provides guidelines on transforming torsional moments into normal stresses and shear stresses which can then be incorporated into the interaction equations explained in Chapter H of the AISC 89 code. The guidelines of the publication have now been incorporated into the AISC-89 steel design modules of STAAD.

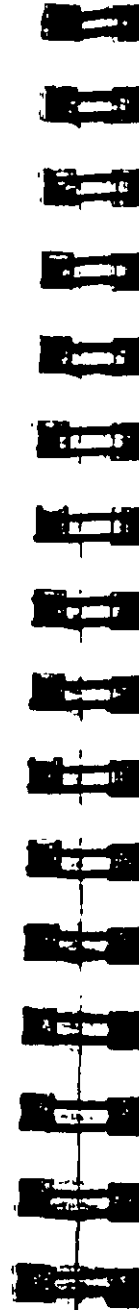
To consider stresses due to torsion in the code checking or member selection procedure, specify the parameter TORSION with a value of 1.0. See Table 2.1 for more details

#### Methodology

If the user were to request design for torsion, the torsional properties required for calculating the warping normal stresses, warping shear stresses and pure shear stresses are first determined. These depend of the "boundary" conditions that prevail at the ends of the member. These boundary conditions are defined as "Free", "Pinned" or "Fixed". They are explained below:

**Free** : "Free" represents the boundary condition such as that which exists at the free end of a cantilever beam. It means that there is no other member connected to the beam at that point.

**Pinned** : "Pinned" represents the condition that corresponds to either a pinned support defined at the joint through the Support



demand or a release of any of the moments at the joint through a Member Release specification.

**Fixed:** "Fixed" represents the condition where a fixed support exists at the joint. In the absence of a support at that joint, it represents a condition where a rigid frame connection exists between the given member and at least one other member connected to that joint. Also, no member releases should be present at that joint on the given member.

After the boundary conditions are determined, the normal and shear stresses are determined. The guidelines specified in the publication T114 for concentrated torsional moments acting at the ends of the member are used to determine these stresses.

The normal stresses are added to the axial stresses caused by axial load. These are then substituted into the interaction equations in Chapter H of the AISC 89 code for determining the ratio. The plane shear and warping shear stresses are added to the shear stresses caused by actual shear forces and compared against the allowable shear stresses on the cross section.

### Restrictions

This facility is currently available for Wide Flange shapes (W.M & S), Channels, Tee shapes, Pipes and Tubes. It is not available for Single Angles, Double Angles, members with the PRISMATIC property specification, Composite sections (Wide Flanges with concrete slabs or plates on top), or Double Channels. Also, the stresses are calculated based on the rules for concentrated torsional moments acting at the ends of the member.



## 2.3.8 Design of Web Tapered Sections

Appendix F of AISC-89 provides specifications for design of Web-Tapered members. These specifications have been incorporated into STAAD to perform code checking on web tapered wide flange shapes. Please note that member selection cannot be performed on web-tapered members.

## 2.4 Design Parameters

The program contains a large number of parameter names which are needed to perform designing and code checking. These parameter names, with their default values, are listed in Table 2.1. These parameters communicate design decisions from the engineer to the program.

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements of an analysis, some or all of these parameter values may have to be changed to exactly model the physical structure. For example, by default the KZ (k value in local z-axis) value of a member is set to 1.0, while in the real structure it may be 1.5. In that case, the KZ value in the program can be changed to 1.5, as shown in the input instructions (Section 6). Similarly, the TRACK value of a member is set to 0.0, which means no allowable stresses of the member will be printed. If the allowable stresses are to be printed, the TRACK value must be set to 1.0.

Note that parameter names PROFILE, DMAX and DMIN are only used for member selection.

See Table  
2.1 and  
Section  
5.47.1

## 2.5 Code Checking

See Section  
6.47.2 and  
Example 1

The purpose of code checking is to check whether the provided section properties of the members are adequate. The adequacy is checked as per AISC-89. Code checking is done using the forces and moments at specified sections of the members. If no sections are specified, the program uses the start and end forces for code checking.

When code checking is selected, the program calculates and prints whether the members have passed the code or have failed, the critical condition of the AISC code (like any of the AISC specifications or compression, tension, shear, etc.); the value of the ratio of the critical condition (overstressed for a value more than 1.0 or any other specified RATIO value); the governing load case, and the location (distance from the start of the member) of forces in the member where the critical condition occurs.

Code checking can be done with any type of steel section listed in Section 2.2 of this manual.

## 2.6 Member Selection

STAAD is capable of performing design operations on specified members. Once an analysis has been performed, the program can select the most economical section, i.e. the lightest section, which fulfills the code requirements for the specified member. The section selected will be of the same type section as originally designated for the member being designed. A wide flange will be selected to replace a wide flange, etc. Several parameters are available to guide this selection. If the PROFILE parameter is provided, the search for the lightest section is restricted to that profile. Up to three (3) profiles may be provided for any member with a section being selected from each one. Member selection can also be constrained by the parameters DMAX and DMIN which limit the maximum and minimum depth of the members. Note that

See Section  
17.3



if the PROFILE parameter is provided for specified members, DMAX or DMIN parameters will be ignored by the program in selecting these members.

Member selection can be performed with all the types of steel sections listed in Section 2.2 of this manual. Note that for beams with cover plates, the sizes of the cover plate are kept constant while the beam section is iterated.

Selection of members, whose properties are originally input from a user created table, will be limited to sections in the user table.

Member selection can not be performed on members whose section properties are input as prismatic.

### 2.6.1 Member Selection by Optimization

See Section  
5.47.4

Steel table properties of an entire structure can be optimized by STAAD. This optimization method involves a state-of-the-art technique which requires automatic multiple analyses. The user can start with no properties for the members but provide the type of profile-spec they have (e.g. BEAM, COLUMN, CHANNEL, ANGLE etc. Refer to Section 2.7.5). The sizes of the members are optimized by the stiffness they are contributing and the amount of loads they are receiving. Based on this, a balanced size is selected for each member. This method requires extensive computer time and hence should be used with caution.

### 2.6.2 Deflection Check With Steel Design

This facility allows the user to consider deflection as a criteria in the CODE CHECK and MEMBER SELECTION processes. The deflection check may be controlled using three parameters which are described in Table 2.1. Note that deflection is used in addition to other strength and stability related criteria. The local deflection calculation is based on the latest analysis results.

## 2.7 Truss Members

As mentioned earlier, a truss member is capable of carrying only axial forces. So in design no time is wasted calculating the allowable bending or shear stresses, thus reducing design time considerably. Therefore, if there is any truss member in an analysis (like bracing or strut, etc.), it is wise to declare it as a truss member rather than as a regular frame member with both ends pinned.

## 2.8 Unsymmetric Sections

For unsymmetric sections (like angle, double angle and tee), STAAD considers the smaller section modulus for checking against bending. For some cases, this approach may produce slightly conservative results.

Appendix C of the AISC code has been implemented for stress reduction of unstiffened compression elements. In addition, the AISC specification for SINGLE ANGLE design has been fully implemented including lateral torsional buckling effects.

Table 2.1 - AISC Parameters

Parameter Name	Default Value	Description
<u>KY</u>	1.0	K value in local y-axis. Usually, this is minor axis.
<u>K</u>	1.0	K value in local z-axis. Usually, this is major axis.
<u>LY</u>	Member Length	Length in local y-axis to calculate slenderness ratio.
<u>LZ</u>	Member Length	Same as above except in z-axis (major).

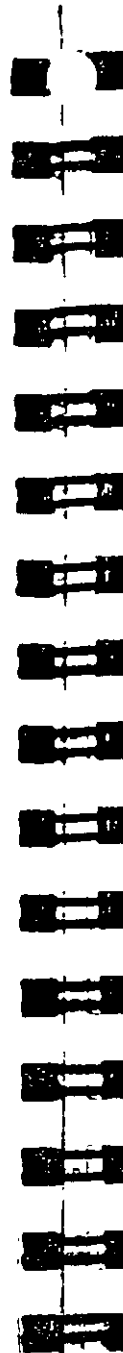


Table 2.1 - AISC Parameters Cont

Parameter Name	Default Value	Description
<u>FYLD</u>	36 KSI	Yield strength of steel in current units.
<u>NSF</u>	1.0	Net section factor for tension members.
<u>UNT</u>	Member Length	Unsupported length of the top flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the top flange.
<u>UNB</u>	Member Length	Unsupported length of the bottom flange for calculating allowable bending compressive stress. Will be used only if flexural compression is on the bottom flange.
<u>CB</u>	1.0	Cb value as used in section 1.5 of AISC. 0.0 = Cb value to be calculated. Any other value will mean the value to be used in design.
<u>SSY</u>	0.0	0.0 = Sidesway in local y-axis 1.0 = No sidesway
<u>SSZ</u>	0.0	Same as above except in local z-axis.
<u>CMY</u>	0.85 for sidesway* and calculated for no sidesway	Cm value in local y & z axes
<u>CMZ</u>		
<u>MAIN</u>	0.0	0.0 = check for slenderness 1.0 = suppress slenderness check
<u>STIFF</u>	Member length	Spacing of stiffeners for plate girder design
<u>PUNCH</u>	See sect. 2.10	Parameter for punching shear
<u>TRACK</u>	0.0	0.0 = Suppress critical member stresses 1.0 = Print all critical member stresses 2.0 = Print expanded output (see Figure 2.1)
<u>DMAX</u>	45 in.	Maximum allowable depth.
<u>DMIN</u>	0.0 in.	Minimum allowable depth.
<u>RATIO</u>	1.0	Permissible ratio of the actual to allowable stresses.

Table 2.1 - AISC Parameters Cont

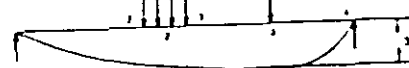
Parameter Name	Default Value	Description
WELD	1 for closed sections 2 for open sections	Weld type, as explained in section 2.11. A value of 1 will mean welding is on one side only except for wide-flange or tee sections, where the web is always assumed to be welded on both sides. A value of 2 will mean welding on both sides. For closed sections like pipe or tube, the welding will be on one side only.
BEAM	0 0	0 0 = design only for end moments or those at locations specified by the SECTION command 1 0 = calculate moments at twelfth points along the beam, and use the maximum, Mz location for design.
WMIN	1/16 inch	Minimum welding thickness.
WSTR	0.4 x FYLD	Allowable welding stress.
DFE	None (Mandatory for deflection check)	"Deflection Length" / Maxm allowable local deflection
DJ1	Start Joint of member	Joint No. denoting starting point for calculation of "Deflection Length" (See Note 1)
DJ2	End Joint of member	Joint No. denoting end point for calculation of "Deflection Length" (See Note 1)
TORSION	0 0	0 0 = No torsion check performed. 1 0 = Perform torsion check based on rules of AISC T114.
APER	1.0	0.0 = Design tapered I-section based on rules of Chapter F and Appendix B only. Do not use the rules of Appendix F. 1.0 = Design tapered I-sections based on rules of Appendix F of AISC-89.

\* In the case of no sideways, which means the member is restrained at both ends, the m value is computed based on the formula in Section 1.6.1 of the AISC code.



NOTE:

1) "Deflection Length" is defined as the length that is used for calculation of local deflections within a member. It may be noted that for most cases the "Deflection Length" will be equal to the length of the member. However, in some situations, the "Deflection Length" may be different. For example, refer to the figure below where a beam has been modeled using four joints and three members. Note that the "Deflection Length" for all three members will be equal to the total length of the beam in this case. The parameters DJ1 and DJ2 should be used to model this situation. Also the straight line joining DJ1 and DJ2 is used as the reference line from which local deflections are measured. Thus, for all three members here, DJ1 should be "1" and DJ2 should be "4".



D = Maximum local deflection for members 1, 2 and 3

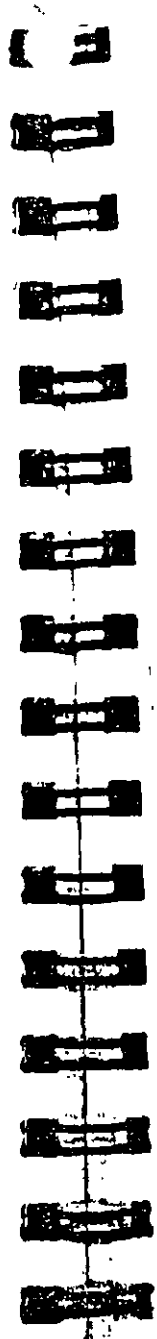
EXAMPLE : PARAMETERS  
DFE 300. ALL  
DJ1 1 ALL  
DJ2 4 ALL

- If DJ1 and DJ2 are not used, "Deflection Length" will default to the member length and local deflections will be measured from original member line.
- It is important to note that unless a DFE value is specified, STAAD will not perform a deflection check. This is in accordance with the fact that there is no default value for DFE (see Table 2.1).
- A critical difference exists between the parameter UNL and the parameters LY & LZ. UNL represents the laterally unsupported length of the compression flange. It is defined in Chapter F, page 5-47 of the specifications in the AISC 1989 ASD manual as the distance between cross sections braced against twist or lateral displacement of the compression flange. UNL is used to calculate the allowable compressive stress (FCZ and FCY) for behavior as a beam. LY and LZ on the other hand are the unbraced lengths for behavior as a column and are used to calculate the KL/r ratios and the allowable axial compressive stress FA.

SSY and CMY are 2 parameters which are based upon 2 values defined in page 5-55, Chapter H of the AISC 9th ed. manual. SSY is a variable which allows the user to define whether or not the member is subject to sidesway in the local Y direction. CMY is a variable used for defining the expression called  $C_m$  in the AISC manual. When SSY is set to 0 (which is the default value), it means that the member is subject to sidesway in the local Y direction. When SSY is set to 1.0, it means that the member is not subject to sidesway in the local Y direction. The only effect that SSY has is that it causes the program to calculate the appropriate value of CMY. If SSY is set to 0 and CMY is not provided, STAAD will calculate CMY as 0.85. If SSY is set to 1 and CMY is not provided, STAAD will calculate CMY from the equation on page 5-55. However, if the user provides CMY, the program will use that value and not calculate CMY at all, regardless of what the user defines SSY to be.

### 2.9 Plate Girders

Plate girders may be designed according to Chapter G of the AISC specifications. The generalized ISECTION specification capability available in the User Table facility may be used to specify the plate girder sections. The AISC welded plate girder shapes (pages 2-230 and 2-231 - AISC 9th Edition) are available in the Steel Section library of the program. Both CODE CHECKING & MEMBER SELECTION may be performed when sections are specified from a standard AISC plate girder section table or through the generalized ISECTION facility. Parameter STIFF (see Table 2.1) may be used to specify stiffener spacing so that the right allowable shear stress may be calculated. The program automatically calculates the flange stresses according to the requirements of section G2.



### 2.10 Punching Shear Check

For tubular members, punching shear may be checked according to the American Petroleum Institute (API) specifications. Parameter PUNCH may be used to identify members for which punching shear check is required. This parameter should also be used to specify joint type and geometry. Refer to the following table for the value of parameter to Table 2.1.

Type of Joint and Geometry	Req. Value of Parameter PUNCH
K (overlap)	1.0
K (gap)	2.0
T & Y	4.0
CROSS (w/diaphragms)	4.0
CROSS (w/o diaphragms)	5.0

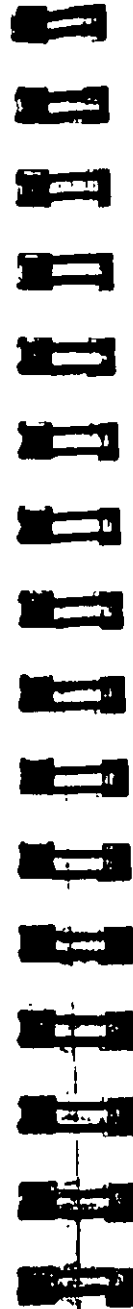
**Note:**

- 1) A value representing joint type & geometry must be provided for parameter PUNCH
- 2) For detailed description of joint type & geometry, refer to API code (Section 2.5.5)

## 2.11 Tabulated Results of Steel Design

For code checking or member selection, the program produces the results in a tabulated fashion. The items in the output table are explained as follows:

- a) MEMBER refers to the member number for which the design is performed.
- b) TABLE refers to the AISC steel section name which has been checked against the steel code or has been selected
- c) RESULT prints whether the member has PASSED or FAILED. If the RESULT is FAIL, there will be an asterisk (\*) mark in front of the member number
- d) CRITICAL COND refers to the section of the AISC code which governed the design.
- e) RATIO prints the ratio of the actual stresses to allowable stresses for the critical condition. Normally a value of 1.0 or less will mean the member has passed.
- f) LOADING provides the load case number which governed the design.
- g) FX, MY and MZ provide the axial force, moment in local y-axis and moment in local z-axis respectively. Although STAAD does consider all the member forces and moments to perform design, only FX MY and MZ are printed since they are the ones which are of interest, in most cases



- h) LOCATION specifies the actual distance from the start of the member to the section where design forces govern.
- i) If the parameter TRACK is set to 1.0, the program will block out part of the table and will print the allowable bending stresses in compression (FCY & FCZ) and tension (FTY & FTZ), allowable axial stress in compression (FA), and allowable shear stress (FV), all in kips per square inch. In addition, member length, area, section moduli, governing KL/r ratio and CB are also printed.
- j) In the output for TRACK 2.0, the items Fey and Fez are as follows:

$$F_{ey} = \frac{12\pi^2 E}{23(K_y L_y / r_y)^2}$$

$$F_{ez} = \frac{12\pi^2 E}{23(K_z L_z / r_z)^2}$$



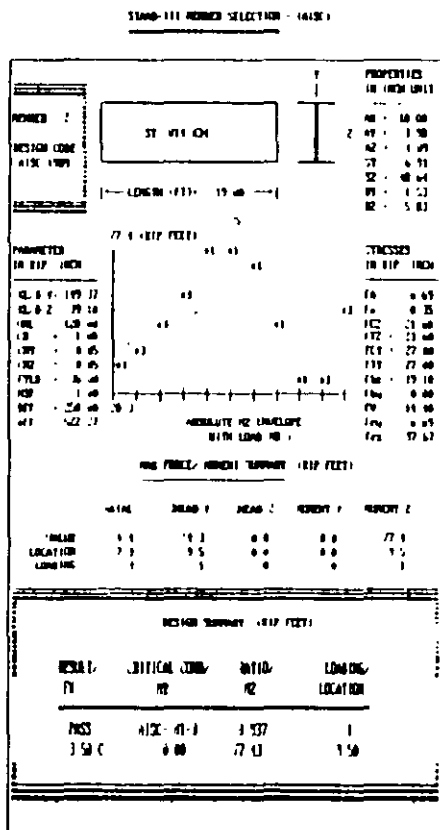


Figure 2.1

## 2.12 Weld Design

Section 7.5

STAAD is able to select weld thickness for connections and tabulate the various stresses. The weld design is limited to the members having properties from wide flange, tee, single angle, single channel, pipe and tube section tables only. The parameters WELD, WMIN and WSTR (as explained in Table 2.1) govern the weld design



Since the thickness of a weld is very small in comparison to its length, the properties of the weld can be calculated as line member. Therefore, the cross-sectional area (AZ) of the weld will actually be the length of the weld. Similarly, the units for the section moduli (SY and SZ) will be length-squared and for the polar moments of inertia (JW) will be length-cubed. The following table shows the different available weld lines, their type and their coordinate axes.

WELD TYPE	ANGLE	WIDE FLANGE	TEE	CHANNEL	PIPE	TUBE
1						
2					-	-

Actual stresses, calculated from the member forces, can be specified by three names, based on their directions.

**Horizontal Stress** - as produced by the local z-shear force and torsional moment.

**Vertical Stress** - as produced by the axial y-shear force and torsional moment.

**Direct Stress** - as produced by the axial force and bending moments in the local y and z directions.

The Combined Stress is calculated by the square root of the summation of the squares of the above three principal stresses.

Following are the equations:

Forces

MX = Torsional moment

MY = Bending in local y-axis

$MZ$  = Bending in local z-axis

$F_X$  = Axial force

$V_Y$  = Shear in local y-axis

$V_Z$  = Shear in local z-axis

### Properties of Weld

$A_X$  = Area of the weld as the line member

$S_Y$  = Section modulus around local y-axis

$S_Z$  = Section modulus around local z-axis

$J_W$  = Polar moment of inertia

$CH$  = Distance of the extreme fiber for horizontal (local z) forces

$CV$  = Distance of the extreme fiber for vertical (local y) forces

Stress Equations:

$$\text{Horizontal stress, } F_h = \frac{V_Z}{A_X} + \frac{CH \times M_X}{J_W}$$

$$\text{Vertical stress, } F_v = \frac{V_Y}{A_X} + \frac{CV \times M_X}{J_W}$$

$$\text{Direct stress, } F_d = \frac{F_X}{A_X} + \frac{M_Z}{S_Z} + \frac{M_Y}{S_Y}$$

- Note that the moments  $M_Y$  and  $M_Z$  are taken as the absolute values, which may result in some conservative results for asymmetrical sections like angle, tee and channel

$$\text{Combined force } F_{\text{comb}} = \sqrt{F_h^2 + F_v^2 + F_d^2}$$

$$\text{Weld thickness} = \frac{F_{\text{comb}}}{F_w}$$

where  $F_w$  = Allowable weld stress, default value is 0.4 FYLD (Table 2.1).



The thickness  $t$  is rounded up to the nearest 1/16th of an inch and all the stresses are recalculated. The tabulated output prints the latter stresses. If the parameter TRACK is set to 1.0, the output will include the weld properties. Note that the program does not calculate the minimum weld thickness as needed by some codes, but checks only against the minimum thickness as provided by the user (or 1/16th inch if not provided).

When the TRUSS qualifier is used with SELECT WELD command, the program will design the welds required for truss angle and double angle members that are attached to gusset plates. The program reports the number of welds (two for single angles, four for double angles), and the length required for each weld. The thickness of the weld is taken as 1/4 inch (6 mm) for members up to 1/4 inch (6 mm) thick, and 1/16 inch (1.5 mm) less than the angle thickness for members greater than 1/4 inch (6 mm) thick. Minimum weld length is taken as four times weld thickness.

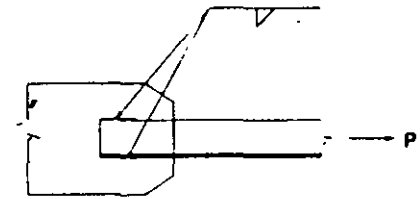


Figure 2.2 Weld design for SELECT WELD TRUSS

## 2.13 Steel Design per AASHTO Specifications

### 2.13.1 General Comments

This section presents some general statements regarding the implementations of American Association of State Highway and Transportation Officials (AASHTO) specifications for structural steel design in STAAD. The design philosophy and procedural logistics for member selection and code checking is based upon the principles of allowable stress design. Two major failure modes are recognized: failure by overstressing and failure by stability

considerations. The following sections describe the salient features of the allowable stresses being calculated and the stability criteria being used. Members are proportioned to resist the design loads without exceeding the allowable stresses and the most economical section is selected on the basis of the least weight criteria. The code checking part of the program also checks the slenderness requirements, the minimum metal thickness requirements and the width-thickness requirement. It is generally assumed that the user will take care of the detailing requirements like provision of stiffeners and check the local effects like flange buckling, web crippling etc.

### 2.13.2 Allowable Stresses per AASHTO Code

As mentioned before, the member design and code checking in STAAD is based upon the allowable stress design method. It is a method for proportioning structural members using design loads and forces, allowable stresses, and design limitations for the appropriate material under service conditions. It is beyond the scope of this manual to describe every aspect of structural steel design per AASHTO specifications because of practical reasons. This section will discuss the salient features of the allowable stresses specified by the AASHTO code. Table 10.32.1A of the AASHTO code specifies the allowable stresses.

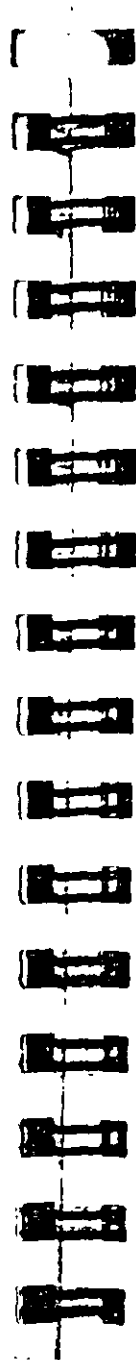
#### Axial Stress

Allowable tension stress, as calculated in AASHTO is based on the net section. This tends to produce a slightly conservative result. Allowable tension stress on the net section is given by,

$$F_t = 0.55F_y$$

Allowable compressive stress on the gross section of axially loaded compression members is calculated based on the following formula:

$$F_a = \frac{F_y}{F.S.} \frac{(1 - (Kl/r)^2 F_y)}{4\pi^2 E} \quad \text{when } (Kl/r) < C_c$$



$$F_a = \frac{\pi^2 E}{F.S. (Kl/r)^2} \quad \text{when } (Kl/r) > C_c$$

$$\text{with } C_c = \left( \frac{2\pi^2 E}{F_y} \right)^{1/2} \quad \text{and F.S.} = 2.12$$

It can be mentioned here that AASHTO does not have a provision for increase in allowable stresses for a secondary member and when  $l/r$  exceeds a certain value.

#### Bending Stress

Allowable stress in bending compression for rolled shape girders and built-up sections whose compression flanges are supported laterally through its full length by embedment in concrete is given by

$$F_b = 0.55F_y$$

For similar members with unsupported or partially supported flange lengths, the allowable bending compressive stress is given by

$$F_b = 0.55F_y \left( 1 - \frac{(l/r)^2 F_y}{4\pi^2 E} \right) \quad \text{when } (Kl/r) < C_c$$

$$\text{with } (r^2) = b^2 / 12$$

The AASHTO code does not have a specification for maximum allowable tensile stress in members subject to bending. The corresponding AISC specification is  $0.66F_y$  for compact hot rolled or built-up sections symmetrical about, and loaded in, the plane of their minor axes and meeting other requirements of Section 1.5.1.4.1 of the AISC specifications. A common practice among bridge designers is to use  $0.55F_y$  as the allowable bending tensile stress for members meeting other requirements of Section 1.5.1.4.1 of the AISC specifications. This practice is slightly conservative and is being used here in the STAAD implementation of the AASHTO steel code.

### Shear Stress

Allowable shear stress on the gross section is given by

$$F_v = 33F_y$$

For shear on the web, the gross section is defined as the product of the total depth and the web thickness. The AASHTO code does not specify any allowable stress for shear on flanges. The program assumes the same allowable for shear stress (0.33F<sub>y</sub>) for both shear on the web and shear on the flanges. For shear on the flanges, the gross section is taken as 2/3 times the total flange area.

### Bending-Axial Stress Interaction

Members subjected to both axial and bending stresses are proportioned according to section 10.36 of the AASHTO steel code. All members subject to bending and axial compression are required to satisfy the following formula:

$$\frac{f_a}{F_a} + \frac{C_{mx} f_{bx}}{(1 - f_a / F_{ax}) F_{bx}} + \frac{C_{my} f_{by}}{(1 - f_a / F_{ay}) F_{by}} < 1.0$$

at intermediate points, and

$$\frac{f_a}{0.72F_y} + \frac{f_{bx}}{F_{bx}} + \frac{f_{by}}{F_{by}} < 1.0$$

at support points.

The AASHTO steel code does not provide any specification for combined axial tension and bending interaction. The AISC procedure of checking bending-axial tension interaction is used in the AASHTO implementation of STAAD. All members subject to combined axial tension and bending are required to satisfy provisions of AISC 1.6.2.

## 2.13.3 Stability Requirements per AASHTO Code

According to AASHTO code, for compression members, the slenderness ratio, KL/r, shall be limited to 120 for main members and to 140 for secondary members. The AASHTO code defines secondary members as those whose primary purpose is to brace the structure against lateral or longitudinal force, or to reduce the unbraced length of other members, main or secondary. For tension members, KL/r is 200 for main members and 240 for secondary members.

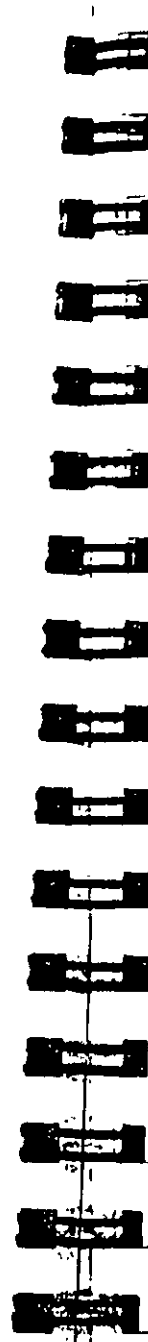
## 2.13.4 Minimum Metal Thickness Requirement

The AASHTO code has a minimum thickness requirement for all structural steel. According to this requirement, all structural steel except for webs of certain rolled shapes, closed ribs in orthotropic decks, fillers and railings, shall not be less than 0.3125 inches. The web thickness of rolled beams or channels shall be not less than 0.23 inches. These requirements have been incorporated in the AASHTO implementation of STAAD.

## 2.14 Steel Design per AISC/LRFD Specification

### 2.14.1 General Comments

The design philosophy embodied in the Load and Resistance Factor Design (LRFD) Specification is built around the concept of limit state design, the current state-of-the-art in structural engineering. Structures are designed and proportioned taking into consideration the limit states at which they would become unfit for their intended use. Two major categories of limit-state are recognized--ultimate and serviceability. The primary considerations in ultimate limit state design are strength and stability, while that in serviceability is deflection. Appropriate load and resistance factors are used so that a uniform reliability is achieved for all steel structures under various loading conditions



and at the same time the chances of limits being surpassed are acceptably remote.

In the STAAD implementation of LRFD, members are proportioned to resist the design loads without exceeding the limit states of strength, stability and serviceability. Accordingly, the most economic section is selected on the basis of the least weight criteria as augmented by the designer in specification of allowable member depths, desired section type, or other such parameters. The code checking portion of the program checks that code requirements for each selected section are met and identifies the governing criteria.

The following sections describe the salient features of the LRFD specifications as implemented in STAAD steel design. A detailed description of the design process along with its underlying concepts and assumptions is available in the LRFD manual. However, since the design philosophy is drastically different from the conventional Allowable Stress Design (ASD), a brief description of the fundamental concepts is presented here to initiate the user into the design process.

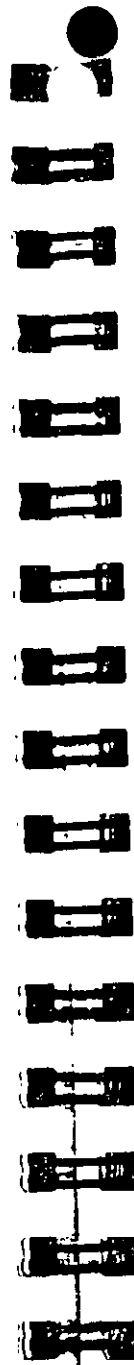
## 2.14.2 LRFD Fundamentals

The primary objective of the LRFD Specification is to provide a uniform reliability for all steel structures under various loading conditions. This uniformity can not be obtained with the allowable stress design (ASD) format.

The ASD method can be represented by the inequality

$$\sum Q_i < R_n / F.S.$$

The left side is the required strength, which is the summation of the load effects,  $Q_i$  (forces and moments). The right side, the design strength, is the nominal strength or resistance,  $R_n$ , divided by a factor of safety. When divided by the appropriate section property (area or section modulus), the two sides of the inequality become the actual stress and allowable stress respectively. ASD,



then, is characterized by the use of unfactored "working" loads in conjunction with a single factor of safety applied to the resistance. Because of the greater variability and, hence, unpredictability of the live load and other loads in comparison with the dead load, a uniform reliability is not possible.

LRFD, as its name implies, uses separate factors for each load and resistance. Because the different factors reflect the degree of uncertainty of different loads and combinations of loads and of the accuracy of predicted strength, a more uniform reliability is possible. The LRFD method may be summarized by the inequality

$$\sum \gamma_i Q_i < R_n \phi$$

On the left side of the inequality, the required strength is the summation of the various load effects,  $Q_i$ , multiplied by their respective load factors,  $\gamma_i$ . The design strength, on the right side, is the nominal strength or resistance,  $R_n$ , multiplied by a resistance factor,  $\phi$ .

In the STAAD implementation of LRFD, it is assumed that the user will use appropriate load factors and create the load combinations necessary for analysis. The design portion of the program will take into consideration the load effects (forces and moments) obtained from analysis. In calculation of resistances of various elements (beams, columns etc.), resistance (nominal strength) and applicable resistance factor will be automatically considered.

### 2.14.3 Analysis Requirements

The types of construction recognized by AISC specification have not changed, except that both "simple framing" (formerly Type 2) and "semi-rigid framing" (formerly Type 3) have been combined into the same category. Type PR (partially restrained), "Rigid Framing" (formerly Type 1) is now Type FR (fully restrained). Type FR construction is permitted unconditionally. Type PR construction may necessitate some inelastic, but self-limiting, deformation of a structural steel element. Thus, when specifying Type PR construction, the designer should take into consideration the effects of partial restraint on the stability of the structure, lateral deflections and second order bending moments. As stated in Sect. C1 of the LRFD specification, an analysis of second order effects is required. Thus, when using LRFD code for steel design, the user must use the P-Delta analysis feature of STAAD.

### 2.14.4 Section Classification

The LRFD specification allows inelastic deformation of section elements. Thus local buckling becomes an important criterion. Steel sections are classified as compact, noncompact or slender element sections depending upon their local buckling characteristics. This classification is a function of the geometric properties of the section. The design procedures are different depending on the section class. STAAD is capable of determining the section classification for the standard shapes and user specified shapes and design accordingly.

### 2.14.5 Axial Tension

The criteria governing the capacity of tension members is based on two limit states. The limit state of yielding in the gross section is intended to prevent excessive elongation of the member. The second limit state involves fracture at the section with the minimum effective net area. The net section area may be specified by the user through the use of the parameter NSF (see Table 2.2). STAAD calculates the tension capacity of a given member based

on these two limit states and proceeds with member selection or code check accordingly.

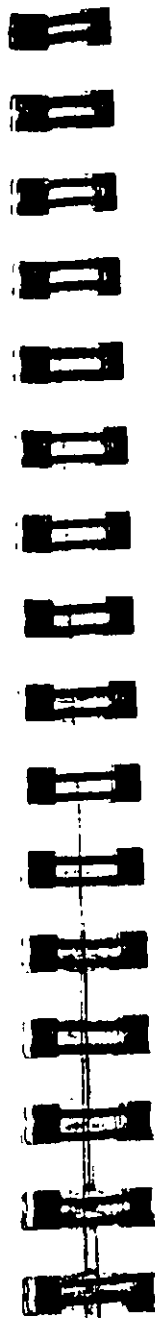
### 2.14.6 Axial Compression

The column strength equations have been revised in LRFD to take into account inelastic deformation and other recent research in column behavior. Two equations governing column strength are available, one for inelastic buckling and the other for elastic or Euler buckling. Both equations include the effects of residual stresses and initial out-of-straightness. Compression strength for a particular member is calculated by STAAD according to the procedure outlined in Chapter E of the LRFD specifications. For slender elements, the procedure described in Appendix B5.3 is used.

Singly symmetric and unsymmetric compression members are designed on the basis of the limit states of flexural-torsional and torsional buckling. The procedure of Appendix E3 is implemented for the determination of design strength for these limit states.

Effective length for calculation of compression resistance may be provided through the use of the parameters KY, KZ and/or LY, LZ. If not provided, the entire member length will be taken into consideration.

In addition to the compression resistance criterion, compression members are required to satisfy slenderness limitations which are a function of the nature of use of the member (main load resisting component, bracing member, etc.). In both the member selection and code checking process, STAAD immediately does a slenderness check on appropriate members before continuing with other procedures for determining the adequacy of a given member.



### 2.14.7 Flexural Design Strength

In LRFD, the flexural design strength of a member is determined by the limit state of lateral torsional buckling. Inelastic bending is allowed and the basic measure of flexural capacity is the plastic moment capacity of the section. The flexural resistance is a function of plastic moment capacity, actual laterally unbraced length, limiting laterally unbraced length, buckling moment and the bending coefficient. The limiting laterally unbraced length  $L_r$  and buckling moment  $M_r$  are functions of the section geometry and are calculated as per the procedure of Chapter F. The purpose of bending coefficient  $C_b$  is to account for the influence of the moment gradient on lateral-torsional buckling. This coefficient can be specified by the user through the use of parameter CB (see Table 2.2) or may be calculated by the program (if CB is specified as 0.0). In the absence of the parameter CB, a default value of 1.0 will be used. The procedure for calculation of design strength for flexure also accounts for the presence of residual stresses of rolling. To specify laterally unsupported length, either of the parameters UNL and UNF (see Table 2.2) can be used.

### 2.14.8 Combined Axial Force And Bending

The interaction of flexure and axial forces in singly and doubly symmetric shapes is governed by formulas H1-1a and H1-1b. These interaction formulas cover the general case of biaxial bending combined with axial force. They are also valid for uniaxial bending and axial force.



### 2.14.9 Design for Shear

The procedure of Sect. F2 of the LRFD Specification is used in STAAD to design for shear forces in members. Shear strength as calculated in LRFD is governed by the following limit states: Eq. F2-1a by yielding of the web; Eq. F2-2a by inelastic buckling of the web; Eq. F2-3a by elastic buckling of the web. Shear in wide flanges and channel sections is resisted by the area of the web, which is taken as the overall depth times the web thickness.

### 2.14.10 Design Parameters

Design per LRFD specifications is requested by using the CODE parameter (see Section 6.47). Other applicable parameters are summarized in Table 2.2. These parameters communicate design decisions from the engineer to the program and thus allow the engineer to control the design process to suit an application's specific needs.

See Table  
2.2 and  
Section  
6.47.1

The default parameter values have been selected such that they are frequently used numbers for conventional design. Depending on the particular design requirements, some or all of these parameter values may be changed to exactly model the physical structure.

Note that parameters PROFILE, DMAX and DMIN may only be used for member selection.

### 2.14.11 Code Checking and Member Selection

Both code checking and member selection options are available in STAAD LRFD implementation. For general information on these options, refer to Sections 2.5 and 2.6. For information on specification of these commands, refer to Section 5.47.1.

## 2.14.12 Tabulated Results of Steel Design

Results of code checking and member selection are presented in a tabular format. A detailed discussion of the format is provided in Section 2.11. Following exceptions may be noted: CRITICAL COND refers to the section of the LRFD specifications which governed the design.

If the TRACK is set to 1.0, member design strengths will be printed out.

Table 2.2 - LRFD Parameters

Parameter Name	Default Value	Description
KY	1.0	K value for bending about Y-axis. Usually this is minor axis.
KZ	1.0	K value for bending about Z-axis. Usually this is major axis.
LY	Member Length	Length to calculate slenderness ratio for bending about Y-axis
LZ	Member Length	Length to calculate slenderness ratio for bending about Z-axis
EYLD	36.0 ksi	Yield strength of steel.
NSF	1.0	Net section factor for tension members



Table 2.2 - LRFD Parameters Cont.

Parameter Name	Default Value	Description
UNL	Member Length	Unsupported length ( $L_b$ ) of the top flange for calculating flexural strength. Will be used only if flexural compression is on the top flange.
UNB	Member Length	Unsupported length ( $L_b$ ) of the bottom flange for calculating flexural strength. Will be used only if flexural compression is on the bottom flange.
CB	1.0	Co-efficient $C_b$ per Chapter F. If $C_b$ is set to 0.0, it will be calculated by the program. Any other value will be directly used in design.
TRACK	0.0	0.0 = Suppress all design strengths. 1.0 = Print all design strengths. 2.0 = Print expanded design output.
DMAX	45.0 in.	Maximum allowable depth.
DMIN	0.0 in.	Minimum allowable depth
RATIO	1.0	Permissible ratio of actual load effect and design strength.
BEAM	0.0	0.0 = design only for end moments and those at locations specified by SECTION command. *1.0 = calculate moment at twelve points along the beam, and use maximum $M_z$ for design.

Note: For deflection check, parameters DFF, DJ1 and DJ2 from Table 2.1 may be used. All requirements remain the same.





# American Concrete Design

## Section 3

### 3.1 Design Operations

STAAD has the capabilities for performing concrete design. It will calculate the reinforcement needed for the specified concrete section. All the concrete design calculations are based on the current ACI 318.

### 3.2 Section Types for Concrete Design

The following types of cross sections can be defined for concrete design.

For Beams

Prismatic (Rectangular & Square),

Trapezoidal and T-shapes

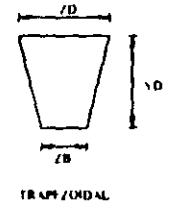
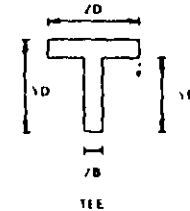
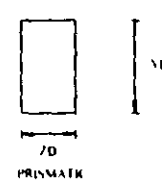
For Columns

Prismatic (Rectangular, Square and Circular)

For Slabs

Finite element with a specified thickness.

Walls/Plates



### 3.3 Member Dimensions

Concrete members which will be designed by the program must have certain section properties input under the MEMBER PROPERTY command. The following example shows the required input.

```

UNIT INCH
MEMBER PROPERTY
1 3 TO 7 9 PRISM YD 18. ZD 12. IZ 2916 IY 1296
11 13 PR YD 12.
14 TO 16 PRIS YD 24. ZD 48. YB 18. ZB 12.
17 TO 19 PR YD 24. ZD 18. ZB 12.

```

In the above input, the first set of members are rectangular (18 inch depth and 12 inch width) and the second set of members, with only depth and no width provided, will be assumed to be circular with 12 inch diameter. Note that no area (AX) is provided for these members. For concrete design, this property must not be provided. If shear areas and moments of inertias are not provided, the program calculates these values from YD and ZD. Notice that in the above example the IZ and IY values provided are actually 50% of the values calculated using YD and ZD. This is a conventional practice which takes into consideration revised section parameters due to cracking of section.

Note that the third and the fourth set of members in the above example represent a T-shape and a TRAPEZOIDAL shape respectively. Depending on the properties (YD, ZD, YB, ZB, etc.) provided, the program will determine whether the section is rectangular, trapezoidal or T-shaped and the BEAM design will be done accordingly.

### 3.4 Design Parameters

The program contains a number of parameters which are needed to perform design by the ACI code. Default parameter values have been selected such that they are frequently used numbers for conventional design requirements. These values may be changed to suit the particular design being performed. Table 3.1 is a complete list of the available parameters and their default values.

See Section  
5.51

Section 5.51.2 of this manual describes the commands required to provide these parameters in the input file. For example, the values of SFACE and EFACE (parameters that are used in shear design), the distances of the face of supports from the end nodes of a beam, are assigned values of zero by default but may be changed depending on the actual situation. Similarly, beams and columns are designed for moments directly obtained from the analyses without any magnification. The factor MMAG may be used for magnification of column moments. For beams, the user may generate load cases which contain loads magnified by the appropriate load factors.

### 3.5 Slenderness Effects and Analysis Consideration

Slenderness effects are extremely important in designing compression members. The ACI-318 code specifies two options by which the slenderness effect can be accommodated (Section 10.10 & 10.11 ACI-318). One option is to perform an exact analysis which will take into account the influence of axial loads and variable moment of inertia on member stiffness and fixed-end moments, the effect of deflections on moments and forces, and the effect of the duration of loads. Another option is to approximately magnify design moments.

STAAD has been written to allow the use of both the options. To perform the first type of analysis, use the command PDELTA ANALYSIS instead of PERFORM ANALYSIS. This analysis

Method will accommodate the requirements as specified in Section 10 of the ACI-318 Code, except for the effects of the duration of the loads. It is felt that this effect may be safely ignored because experts believe that the effects of the duration of loads are negligible in a normal structural configuration. If it is desired, STAAD can also accommodate any arbitrary moment magnification factor (second option) as an input, in order to provide some safety due to the effects of the duration of loads.

Table 3.1

Parameter Name	Default Value	Description
FYMAIN	* 60,000 psi	Yield Stress for main reinforcing steel.
FYSEC	* 60,000 psi	Yield Stress for secondary steel.
FC	* 4,000 psi	Compressive Strength of Concrete.
CLT	* 1.5 inch	Clear cover for top reinforcement.
CLB	* 1.5 inch	Clear cover for bottom reinforcement.
CLS	* 1.5 inch	Clear cover for side reinforcement.
MINMAIN **	Number 4 bar	Minimum main reinforcement bar size (Number 4 - 18)
MINSEC **	Number 4 bar	Minimum secondary reinforcement bar size
MAXMAIN **	Number 18 bar	Maximum main reinforcement bar size
SFACE	* 0.0	Face of support location at start of beam. If specified, the shear force at start is computed at a distance of SFACE+d from the start joint of the member.



EFACE

\* 0.0

Face of support location at end of beam (Note: Both SFACE & EFACE are input as positive numbers) If specified, the shear force at end is computed at a distance of EFACE+d from the end joint of the member..

REINF

0.0

Tied Column. A value of 1.0 will mean spiral.

Table 3.1 Cont.

Parameter Name	Default Value	Description
MMAG	1 0 (for columns only)	A factor by which the column design moments will be magnified
WIDTH	*ZD	Width of concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES
DEPTH	*YD	Depth of concrete member. This value defaults to YD as provided under MEMBER PROPERTIES.
NSECTION	12	Number of equally-spaced sections to be considered in finding critical moments for beam design.
TRACK	0 0	<p><b>BEAM DESIGN:</b></p> <p>With TRACK set to 0 0, Critical Moment will not be printed out with beam design report. A value of 1 0 will mean a print out. A value of 2 0 will print out required steel areas for all intermediate sections specified by NSECTION.</p> <p><b>COLUMN DESIGN:</b></p> <p>TRACK 0 0 prints out detailed design results.            TRACK 1 0 prints out column interaction analysis results in addition to TRACK 0 0 output.            TRACK 2 0 prints out a schematic interaction diagram and intermediate interaction values in addition to all of above.</p>

\* These values must be provided in the current unit system being used.

\*\* When using metric units for ACI design, provide values for these parameters in actual 'mm' units instead of the bar number. The following metric bar sizes are available: 6 mm, 8 mm, 10 mm, 12 mm, 16 mm, 20 mm, 25 mm, 32 mm, 40 mm, 50 mm and 60 mm.



Although ignoring load duration effects is somewhat of an approximation, it must be realized that the approximate evaluation of slenderness effects is also an approximate method. In this method, moment-magnification is based on empirical formula and assumptions on sideways.

Considering all this information, it is our belief, that a PDELTA ANALYSIS, as performed by STAAD, is most appropriate for the design of concrete members. However, the user must note, to take advantage of this analysis, all combinations of loadings must be provided as primary load cases and not as load combinations. This is due to the fact that load combinations are just algebraic combinations of forces and moments, whereas a primary load case is revised during the pdelta analysis based on the deflections. Also note that the proper factored loads (such as 1.4 for DL etc.) should be provided by the user. STAAD does not factor the loads automatically.

### 3.6 Beam Design

Beams are designed for flexure, shear and torsion. For all these forces, all active beam loadings are prescanned to locate the possible critical sections. The total number of sections considered is 12 (twelve) unless this number is redefined with an NSECTION parameter. All of these equally spaced sections are scanned to determine moment and shear envelopes.

#### Design for Flexure

Reinforcement for positive and negative moments are calculated on the basis of the section properties provided by the user. If the section dimensions are inadequate to carry the applied load, that is if the required reinforcement is greater than the maximum allowable for the cross section, the program reports that beam fails in maximum reinforcement. Effective depth is chosen as Total depth - (Clear cover + diameter of stirrup + half the dia. of main reinforcement), and a trial value is obtained by adopting proper bar sizes for the stirrups and main reinforcements. The relevant clauses

Equations 10.2 to 10.6 of ACI 318 are utilized to obtain the actual amount of steel required as well as the maximum allowable and minimum required steel. These values are reported as ROW, ROWMX and ROWMN in the output and can be printed using the parameter TRACK 10 (see Table 3.1). In addition, the maximum, minimum and actual bar spacing are also printed.

It is important to note that beams are designed for flexural moment MZ only. The moment MY is not considered in the flexural design.

### Design for Shear

Shear reinforcement is calculated to resist both shear forces and torsional moments. Shear forces are calculated at a distance (d+SFACE) and (d+EFACE) away from the end nodes of the beam. SFACE and EFACE have default values of zero unless provided under parameters (see Table 3.1). Note that the value of the effective depth "d" used for this purpose is the update value and accounts for the actual e.g. of the main reinforcement calculated under flexural design. Clauses 11.1 through 11.6 of ACI 318 are used to calculate the reinforcement for shear forces and torsional moments. Based on the total stirrup reinforcement required, the size of bars, the spacing, the number of bars and the distance over which they are provided are calculated. Stirrups are always assumed to be 2-legged.

### Design for Anchorage

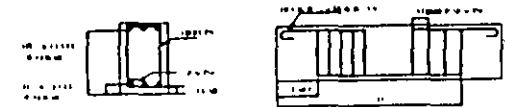
In the output for flexural design, the anchorage details are also provided. At any particular level, the START and END coordinates of the layout of the main reinforcement is described along with the information whether anchorage in the form of a hook or continuation is required or not at these START and END points. Note that the coordinates of these START and END points are obtained after taking into account the anchorage requirements. Anchorage length is calculated on the basis of the Clauses described in Chapter 12 of ACI 318.



### Description of Output for Beam Design

Table 3.2 shows a sample output of an actual reinforcement pattern developed by the program. The following annotations apply to the Table 3.2:

1)	LEVEL	Serial number of bar level which may contain one or more bar group
2)	HEIGHT	Height of bar level from the bottom of beam.
3)	BAR INFO	Reinforcement bar information specifying number of bars and bar size
4)	FROM	Distance from the start of the beam to the start of the reinforcement bar
5)	TO	Distance from the start of the beam to the end of the reinforcement bar.
6)	ANCHOR (STA/END)	States whether anchorage, either a hook or continuation, is needed at start (STA) or at the end.
7)	ROW	Actually required flexural reinforcement ( $A_s/bd$ ) where $b$ = width of cross section (ZD for rectangular and square section) and $d$ = effective depth of cross section (YD - distance from extreme tension fiber to the e.g. of main reinforcement).
8)	ROWMN	Minimum required flexural reinforcement ( $A_{min}/bd$ )
9)	ROWMX	Maximum allowable flexural reinforcement ( $A_{max}/bd$ )
10)	SPACING	Distance between centers of adjacent bars of main reinforcement
11)	Vu	Factored shear force at section.
12)	Vc	Nominal shear strength provided by concrete.
13)	Vs	Nominal shear strength provided by shear reinforcement.
14)	Tu	Factored torsional moment at section
15)	Tc	Nominal torsional moment strength provided by concrete.
16)	Ts	Nominal torsional moment strength provided by torsion reinforcement



**Table 3.2**  
**(Actual Output from Design)**

.....

BEAM NO. 14 DESIGN RESULTS - MOMENTURE

LEN = 20.00 FT    FY = 50000    FC = 4000    SIZE = 15.00 X 21.00 INCHES

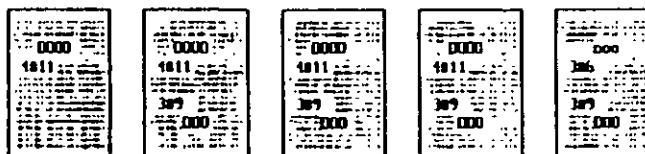
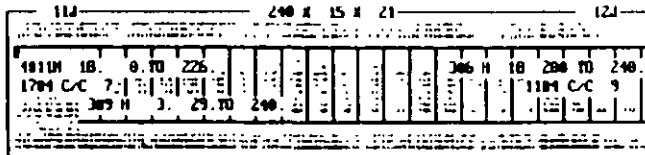
LEVEL	HEIGHT FT	BAR INFO IN.	FROM FT	TO FT	ANCHOR STA	END
1	0 - 2-5/8	3-NUM. 9	2 - 4-5/8	20 - 3-0/0	NO	YES
CRITICAL POS. MOMENT = 89.77 KIP-FT AT 12.00 FT. LOAD REQD STEEL = 2.48 IN <sup>2</sup> ROR = 0.0090    RORMAX = 0.0214    RORMIN = 0.0033 MAX. MIN/ACTUAL BAR SPACING = 1.07 / 2.26 / 4.94 INCH BASIC REQD. DEVELOPMENT LENGTH = 17.95 / 31.42 INCH						
2	1 - 6-1/8	4-NUM. 11	0 - 3-0/0	18 - 3-0/0	YES	NO
CRITICAL NEG. MOMENT = 369.86 KIP-FT AT 0.00 FT. LOAD REQD STEEL = 5.47 IN <sup>2</sup> ROR = 0.0199    RORMAX = 0.0214    RORMIN = 0.0033 MAX. MIN/ACTUAL BAR SPACING = 7.00 / 2.82 / 3.20 INCH BASIC REQD. DEVELOPMENT LENGTH = 19.20 / 53.91 INCH						
3	1 - 6-1/8	3-NUM. 9	16 - 3-1/4	20 - 3-0/0	NO	YES
CRITICAL NEG. MOMENT = 105.87 KIP-FT AT 20.00 FT. LOAD REQD STEEL = 1.31 IN <sup>2</sup> ROR = 0.0047    RORMAX = 0.0214    RORMIN = 0.0033 MAX. MIN/ACTUAL BAR SPACING = 7.50 / 1.75 / 5.13 INCH BASIC REQD. DEVELOPMENT LENGTH = 16.10 / 27.75 INCH						

.....

BEAM NO. 14 DESIGN RESULTS - SHEAR

AT START SUPPORT - Vu = 83.90 KIP    Vc = 35.10 KIP    Vd = 53.82 KIP  
 PROVIDE NUM. 4 BARS AT 7.0 IN. C/C FOR 1.08 IN

AT END SUPPORT - Vu = 57.52 KIP    Vc = 35.10 KIP    Vd = 32.46 KIP  
 PROVIDE NUM. 4 BARS AT 9.3 IN. C/C FOR .94 IN



### 3.7 Column Design

Columns design in STAAD per the ACI code is performed for axial force and uniaxial as well as biaxial moments. All active loadings are checked to compute reinforcement. The loading which produces the largest amount of reinforcement is called the critical load. Column design is done for square, rectangular and circular sections. For rectangular and circular sections, reinforcement is always assumed to be equally distributed on all faces. This means that the total number of bars for these sections will always be a multiple of four (4). If the MMAG parameter is specified, the column moments are multiplied by the MMAG value to arrive at the ultimate moments on the column. Since the ACI code no longer requires any minimum eccentricity conditions to be satisfied, such checks are not made.

**Method used:** Bresler Load Contour Method

**Known Values:** Pu, Mu, Muz, B, D, Clear cover, Fc, Fy  
 Ultimate Strain for concrete : 0.003

**Steps involved :**

- 1) Assume some reinforcement. Minimum reinforcement (1%) is a good amount to start with.
- 2) Find an approximate arrangement of bars for the assumed reinforcement.
- 3) Calculate PNMAX = 0.85 Po, where Po is the maximum axial load capacity of the section. Ensure that the actual nominal load on the column does not exceed PNMAX. If PNMAX is less than Pu/PHI, (PHI is the strength reduction factor) increase the reinforcement and repeat steps 2 and 3. If the reinforcement exceeds 8%, the column cannot be designed with its current dimensions.
- 4) For the assumed reinforcement, bar arrangement and axial load, find the uniaxial moment capacities of the column for the Y and the Z axes, independently. These values are referred to as MYCAP and MZCAP respectively.



5) Solve the Interaction equation

$$\left(\frac{M_{ny}}{M_{ycap}}\right)^\alpha + \left(\frac{M_{nz}}{M_{zcap}}\right)^\alpha \leq 1.0$$

where  $\alpha = 1.24$

If the column is subjected to a uniaxial moment,  $\alpha$  is chosen as 1.0

- 6) If the Interaction equation is satisfied, find an arrangement with available bar sizes, find the uniaxial capacities and solve the interaction equation again. If the equation is satisfied now, the reinforcement details are written to the output file.
- 7) If the interaction equation is not satisfied, the assumed reinforcement is increased (ensuring that it is under 8%) and steps 2 to 6 are repeated.

### Column Interaction

The column interaction values may be obtained by using the design parameter TRACK 1.0 or TRACK 2.0 for the column member. If a value of 2.0 is used for the TRACK parameter, 12 different Pn-Mn pairs, each representing a different point on the Pn-Mn curve are printed. Each of these points represents one of the several Pn-Mn combinations that this column is capable of carrying about the given axis, for the actual reinforcement that the column has been designed for. In the case of circular columns, the values are for any of the radial axes. The values printed for the TRACK 1.0 output are:

- P0 = Maximum purely axial load carrying capacity of the column (zero moment).
- Pnmax = Maximum allowable axial load on the column (Section 10.3.5 of ACI 318).
- P-bal = Axial load capacity at balanced strain condition.
- M-bal = Uniaxial moment capacity at balanced strain condition.
- c-bal = M-bal / P-bal = Eccentricity at balanced strain condition.
- M0 = Moment capacity at zero axial load.
- P-tens = Maximum permissible tensile load on the column.



Des. Pn = Pu/PHI where PHI is the Strength Reduction Factor and Pu is the axial load for the critical load case.  
 Des Mn = Mu\*MMAG/PHI where PHI is the Strength Reduction Factor and Mu is the bending moment for the appropriate axis for the critical load case. For circular columns,

$$M_u = \sqrt{M_{uy}^2 + M_{uz}^2}$$

c/h = (Mn/Pn)/h where h is the length of the column.

### Column Design Output

The following table illustrates different levels of the column design output

Table 3.3

The following output is generated without any TRACK specification

```

.....
C O L U M N   N O   5   D E S I G N   R E S U L T S
FY = 60000  FC = 4000  PSI   SQRE  SIZE =  2 00  X  2 00  INCHES,  TIED
AREA OF STEEL REQUIRED =  1.988  SQ  IN
BAR CONFIGURATION   REINF  PCT   LOAD   LOCATION   PHI
.....
3  NUMBER 9          5.556   2     STA     0.700
(PROVIDE EQUAL NUMBER OF BARS AT EACH FACE)
    
```

TRACK 1.0 generates the following additional output.

COLUMN INTERACTION MOMENT ABOUT Z-AXIS (KIP-FT)					
PO	Pn max	P-bal	M-bal	e-bal (inch)	
897.12	717.70	189.56	158.50	10.33	
NO	P-tens	Des Pn	Des Mn	e	
137.46	-432.00	323.12	9.88	0.303	

COLUMN INTERACTION MOMENT ABOUT Y-AXIS (KIP-FT)					
PO	Pn max	P-bal	M-bal	e-bal (inch)	
897.12	717.70	189.56	158.50	10.33	
NO	P-tens	Des Pn	Des Mn	e	
137.46	-432.00	323.12	108.08	0.333	

TRACK 2.0 generates the following output in addition to all of the above.

	Pn	Mn	Pn	Mn	(k)
PO	210.44	166.60	149.93	169.69	
NO	163.81	171.74	127.39	165.12	
NOMINAL AXIAL COMPRESSION	210.44	166.60	149.93	169.69	
P-tens	163.81	171.74	127.39	165.12	

### 3.8 Slab/Wall Design

Slab and walls are designed per ACI specifications. To design a slab or wall, it must be modeled using finite elements.

Element design will be performed only for the moments MX and MY at the center of the element. Design will not be performed for FX, FY, FXY, QX, QY or MXY. Also, design is not performed at any other point on the surface of the element.

A typical example of element design output is shown in Table 3.4. The reinforcement required to resist  $M_x$  moment is denoted as longitudinal reinforcement and the reinforcement required to resist  $M_y$  moment is denoted as transverse reinforcement (Figure 3.1). The parameters  $F_{YMAIN}$ ,  $F_C$ , and  $CLEAR$  listed in Table 3.1 are relevant to slab design. Other parameters mentioned in Table 3.1 are not applicable to slab design.

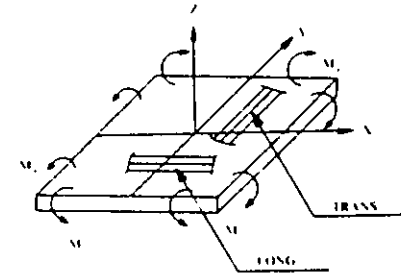


Table 3.4 (Actual Output from Design)

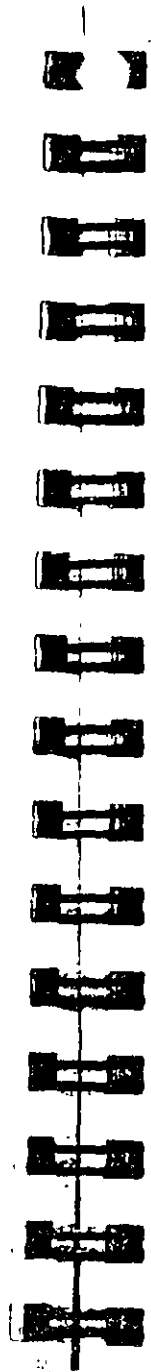
ELEMENT FORCES		FORCE LENGTH UNITS = KIP FEET		FORCE LENGTH/WIDTH		MOMENT		FORCE LENGTH/WIDTH	
ELEMENT	LOAD	QX	QY	FX	FY	FX	FY	FX	FY
13		0.30	0.34	0.14	0.06	0.00	0.00	0.00	0.00
TOP	SHAX=	9.35	SMIN=	2.09	THAX=	3.63	ANGLE=	0.0	
BOTT	SHAX=	2.74	SMIN=	0.56	THAX=	1.65	ANGLE=	0.0	
		0.00	0.03	0.10	0.04	0.00	0.00	0.00	0.00
TOP	SHAY=	5.44	SMIN=	0.74	THAY=	2.15	ANGLE=	18.7	
BOTT	SHAY=	1.37	SMIN=	1.81	THAY=	1.59	ANGLE=	35.5	

ELEMENT DESIGN SUMMARY				
ELEMENT	LONG. REINF (SQ. IN/FT)	NON-X /LOAD (K-FT/FT)	TRANS. REINF (SQ. IN/FT)	NON-Y /LOAD (K-FT/FT)
13 TOP	0.130	0.14	0.10	0.06 / 1
BOTT	0.000	0.00 / 0	0.300	0.30 / 0







## Timber Design

### Section 4

#### 4.1 Timber Design

STAAD Timber design module offers design of Glulam timber sections as per AITC Codes (Timber Construction Manual, 3rd. Edition, 1985). It also conforms to the National Design Specification for Wood Construction and Supplement (NDS) and building codes like Uniform Building Code (UBC), Basic/National Building Code and Standard Building Code. Some of the main features of the program are:

1. This feature is for Glulam Timber only.
2. Code check and design of members as per TCM - AITC.
3. Design values for Structural Glued Laminated Timber tables are in-built into the program. The program accepts Table no., Combination and Species specifications as inputs (e.g., 1:16F-V3-SP/SP) and reads design values from in-built tables.
4. Incorporates all the following Allowable stress modifiers:
  - i) Duration of Load Factor
  - ii) Size Factor
  - iii) Form Factor
  - iv) Lateral stability of Beams and Columns
  - v) Moisture Content Factor
  - vi) Temperature and Curvature factors.

The allowable stresses for bending, tension, compression, shear and Moduli of elasticities are modified accordingly.
5. Determines slenderness for beams and columns (Short, intermediate and long) and checks for min. eccentricity, lateral

stability, buckling, bending and compression, bending and tension and horizontal shear against both axes

- The output results show sections provided or chosen, actual and allowable stresses, governing condition and ratios of interaction formulae and the relevant AITC clause nos. etc for each individual member.

## 4.2 Design Operations

### Explanation of Terms and Symbols Used in This Section

Symbols	Description
$f_a$	Actual compression or tension stress (in PSI). For tension, the axial load is divided by net sectional area (i.e. NSF x X-area).
FA	Allowable design value for compression or tension (in PSI) modified with applicable modifiers or calculated based on slenderness in case of compression.
$f_{bz}$ , $f_{by}$	Actual bending stresses about local Z and Y axis (in PSI)
FBZ, FBY	Allowable design values for bending stresses about local Z and Y axis (in PSI) modified by the applicable modifiers.
JZ, JY	Modifier for P-DELTA effect about the Z and Y axis respectively as explained in formula 5-18 of TCM.
$f_{vz}$ , $f_{vy}$	Actual horizontal shear stresses.
FVZ, FVY	Allowable horizontal shear stresses.
VZ, VY	Shear in local Z and local Y direction.
ZD, YD	Depth of section in local Z and Y axis.
EZ, EY	Minimum eccentricity along Z and Y axis.
CFZ, CFY	CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively.
CLZ, CLY	CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y axis respectively.
RATIO	Permissible ratio of the stresses as provided by the user. The default value is 1.



## Combined Bending and Axial Stresses

### Bending and Axial Tension:

The following interaction formulae are checked:

- $f_a/FA + f_{bz}/(FBZ \times CFZ) + f_{by}/(FBY \times CFY) \leq \text{RATIO}$
- Lateral stability check with Net compressive stress.  
 $- f_a/FA + f_{bz}/(FBZ \times CLZ) + f_{by}/(FBY \times CLY) \leq \text{RATIO}$

### Bending and Axial Compression:

- $f_a/FA + f_{bz}/(FBZ \cdot JZ \times f_a) + f_{by}/(FBY \cdot JY \times f_a) \leq \text{RATIO}$

Applicability of the size factor.

- When  $CF < 1.00$ ,  
 if  $f_a > FBZ \times (1 - CFZ)$ , FBZ is not modified with CFZ. if  $f_a > FBY \times (1 - CFY)$ , FBY is not modified with CFY.

if  $f_a < FBZ \times (1 - CFZ)$  FBZ is taken as  $FBZ \times CFZ + f_a$  but shall not exceed  $FBZ \times CLZ$

if  $f_a < FBY \times (1 - CFY)$  FBY is taken as  $FBY \times CFY + f_a$  but shall not exceed  $FBY \times CLY$

- When  $CF \geq 1.00$ , the effect of CF and CL are cumulative  
 FBZ is taken as  $FBZ \times CFZ \times CLZ$  FBY is taken as  $FBY \times CFY \times CLY$

### Min. Eccentricity:

The program checks against min. eccentricity in following cases

- The member is a FRAME member and not a truss member and under compression.
- The value of actual axial compressive stress does not exceed 30% of the allowable compressive stress.
- The actual moments about both axes are less than moments that would be caused due to min. eccentricity. In this

approach, the moment due to min. eccentricity is taken as the compressive load times an eccentricity of 1 in. or 0.1 x depth whichever is larger.

In case of min. eccentricity,

$f_{bx}$  is taken as  $f_x \times (6+1.5 \times JZ)/(EZ/ZD)$   $f_{by}$  is taken as  $f_y \times (6+1.5 \times JY)/(EY/YD)$

the following conditions are checked

$f_x/FA + f_{bx}/(FBZ \cdot JZ \times f_x) \leq \text{RATIO}$  and  $f_y/FA + f_{by}/(FBY \cdot JY \times f_y) \leq \text{RATIO}$

Shear Stresses:

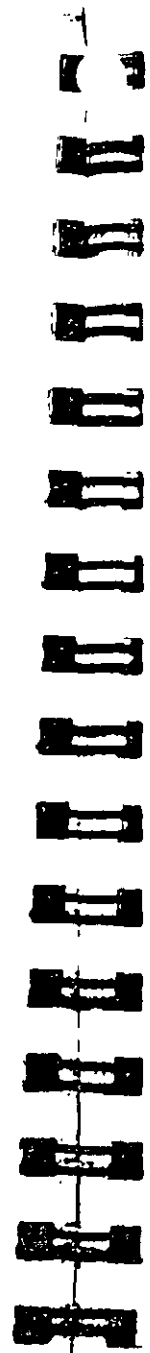
Horizontal stresses are calculated and checked against allowable values:

$f_{vx} = 3 \times VY / (2 \times \text{Area} \times \text{NSF}) \leq FVZ$   $f_{vy} = 3 \times VZ / (2 \times \text{Area} \times \text{NSF}) \leq FVY$

**4.3 Input Specification**

A typical set of input commands for STAAD TIMBER DESIGN is listed below:

UNIT KIP INCH  
 PARAMETER  
 CODE TIMBER  
 GLULAM 1:16F-V3-DF/DF MEMB 1 TO 14  
 GLULAM 1:22F-E5-SP/SP MEMB 15 TO 31  
 GLULAM 2:3-DF MEMB 32 TO 41  
 LAMIN 1.375 LY 168.0 MEMB 5 9 15 TO 31  
 LZ 176.0 MEMB 1 TO 4 6 7 8 10 TO 14  
 LUZ 322.6 ALL  
 LUY 322.6 ALL  
 WET 1.0 ALL  
 CDT 1.33  
 NSF 0.85  
 BEAM 1.0 ALL



**CHECK CODE 1 TO 14  
 SELECT MEMB 15 TO 31**

**Input Commands and Parameters Explained**

Specify PARAMETER and then CODE TIMBER to start TIMBER DESIGN before specifying the input parameters. The user must provide the timber grade (GLULAM GRADE) for each member he intends to design. The parameters can be specified for all or specified list of members. If a parameter is not specified, the default value is assigned to it. See following INPUT PARAMETERS LIST TABLE for description and default values of the parameters.

**Glulam Grade & Allowable Stresses from Table**

The allowable stresses for GLULAM members are read in from Table-1 and Table-2 of AITC for design values for Structural Glued Laminated Timber. The structural members are to be specified in the following manner:

Table - 1 Members :

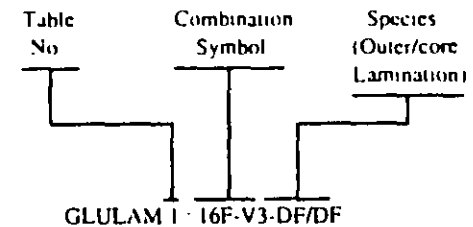
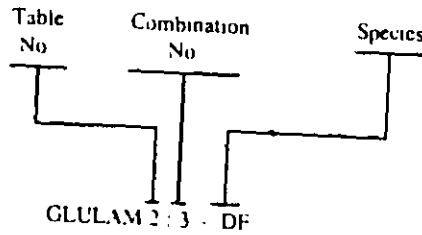


Table - 2 Members :



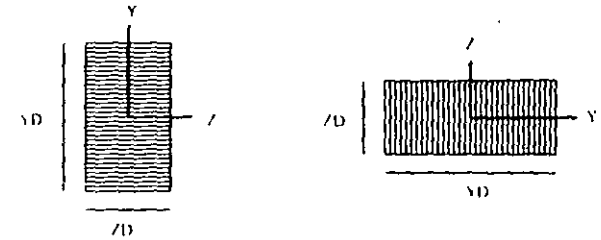
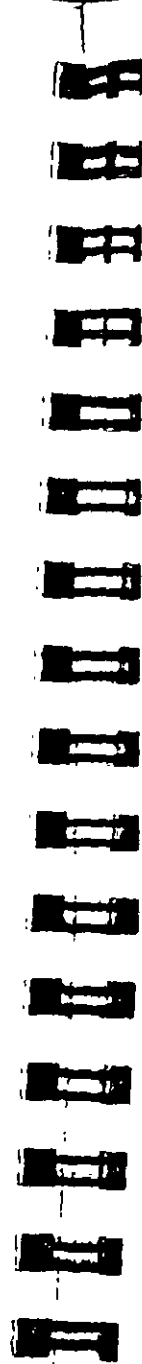
For TABLE-2 members, the applicable stress values are selected based on the depth and the number of laminations. Please note here that lamination thickness (in inch) can be provided by the user and in case it is not provided the default is taken as 1.5 inch. Usually, it is either 1.5 inch or 1.375 inch.

### 4.4 Code Checking

The CHECK CODE command enables the user to check the adequacy of the size ( YD X ZD ) provided in the MEMBER PROPERTIES for the most critical forces and moments. The program prints whether the member has PASSED or FAILED, the critical conditions and the value of the ratio.

### 4.5 Orientation of Lamination

Laminations are always assumed to lie along the local Z-plane of the member. The user may please note that in MEMBER PROPERTIES section, YD always represents the depth of the section across the grain and ZD represents the width along the grain.



### 4.6 Member Selection

The SELECT MEMBER command starts with the min. permissible depth (or min. depth provided thru DMIN parameter) and checks the code. If the member fails with this depth, the thickness is increased by one lamination thickness and the code requirements are checked again. The process is continued till the section passes all the code requirements. This ensures the least weight section for the member. If the depth of the section reaches max. allowable or available depth and the member still fails, the user can have the following options for redesign:

- i) Change the width or increase the max. allowable depth (DMAX)
- ii) Change the timber grade
- iii) Change the design parameters.

Table 4.1 - Timber Design Parameters

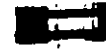
Parameter Name	Default Value	Description
LZ	Length of the Member(L)	Effective length of the column in z-axis.
LY	-DO-	Same as above in y-axis
LUZ	1.92*L	Unsupported effective length for beam in z.
LUY	1.92*L	Unsupported effective length for beam in y.

Table 4.1 - Timber Design Parameters Cont.

Parameter Name	Default Value	Description
WET	0 0	0 0 - dry condition 1 0 - wet condition wet use factors are in-built
NSF	1 0	Net section factor for tension members. (both shear and tension stresses are based on sectional area x nsf )
CDT	1 0	Duration of load factor
CSF	1 0	Form factor
CTM	1.0	Temp factor
CCR	1 0	Curvature factor.
RATIO	1 0	Permissible ratio of actual to allowable stresses.
LAMINATION	1.50 inch	Thickness of lamination in inch (1 50 or 1 375)
BEAM	0 0	0.0 = design for end forces or at locations specified by section command 1 0 = calculate moments at tenth along the beam and use the max for design.

Note:

- i. In case the column buckling is restrained in Y and/or Z direction provide LY and/or LZ as zero(s). Similarly, lateral beam buckling in Y and/or Z direction could be restrained by providing LUY and/or LUZ as zeros.
- ii. Size Factor, lateral stability and moisture content factors and few others are either calculated or read from tables within the program.



SAMPLE OUTPUT RESULTS

STAAD CODE CHECKING - (AUTO)  
\*\*\*\*\*

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
2	8	000X15 000	FAIL	TCN CL 5-18	1 205 2
		2 24 C	0 00	15.38	3 3000

MEMB- 2 GLULAM GRADE=16F-V1-DF/WH LAM.=1.500 UNITS ROUND-INCH  
 LZ=240 00 LY=240 00 LUZ=240 00 LUY=240 00 JZ =0.370 JY =1.000 CDT=1.000  
 CSF=1.000 WET=0 0 CCR=1.000 CTM=1 00 CFZ=0.98 CFY=1.00 CLZ=1 000 CLY=1 000  
 ACTUAL STRESSES : fa= 18 67,fbz=1815 08,fbx= 0 00,fcz= 49 17,fcy= 0 00  
 ALLOW STRESSES : FA= 366.67,FBZ=1579 19,FBY= 950.00,FVZ=140 00,FVY=130 00

3	8	300X15 000	FAIL	TCN CL 5-18	1 239 2
		10 64 C	0 00	43.94	3 3000

MEMB- 3 GLULAM GRADE=16F-V1-DF/WH LAM.=1.500 UNITS ROUND-INCH  
 LZ=180 00 LY=180 00 LUZ=180 00 LUY=180 00 JZ =0.374 JY =0.997 CDT=1 000  
 CSF=1 00 WET=0 0 CCR=1.00 CTM=1 00 CFZ=0 98 CFY=1.00 CLZ=1 000 CLY=1 000  
 ACTUAL STRESSES : fa= 88 68,fbz=1757 49,fbx= 0 00,fcz= 36 01,fcy= 0 00  
 ALLOW STRESSES : FA= 552.22,FBZ=1600 00,FBY= 950 00,FVZ=140 00,FVY=130 00

STAAD MEMBER SELECTION - (AUTO)  
\*\*\*\*\*

ALL UNITS ARE - KIP FEET (UNLESS OTHERWISE NOTED)

MEMBER	TABLE	RESULT/ FX	CRITICAL COND/ MY	RATIO/ MZ	LOADING/ LOCATION
2	8	000X18 000	PASS	TCN CL 5-18	0 860 2
		2 24 C	0 00	45.38	3 3000

MEMB- 2 GLULAM GRADE=16F-V1-DF/WH LAM.=1.500 UNITS ROUND-INCH  
 LZ=240 00 LY=240 00 LUZ=240 00 LUY=240 00 JZ =0 173 JY =1.000 CDT=1 000  
 CSF=1 00 WET=0 0 CCR=1.00 CTM=1.00 CFZ=0.96 CFY=1.00 CLZ=1 000 CLY=1 000  
 ACTUAL STRESSES : fa= 15.56,fbz=1260 46,fbx= 0 00,fcz= 41 24,fcy= 0 00  
 ALLOW STRESSES : FA= 366.67,FBZ=1545 08,FBY= 950 00,FVZ=140 00,FVY=130 00

3	8	300X18 000	PASS	TCN CL 5-18	0 876 2
		10 64 C	0 00	43.94	0 0000

MEMB- 3 GLULAM GRADE=16F-V1-DF/WH LAM.=1.500 UNITS ROUND-INCH  
 LZ=180 00 LY=180 00 LUZ=180 00 LUY=180 00 JZ =0 000 JY =0.997 CDT=1 000  
 CSF=1 00 WET=0 0 CCR=1.00 CTM=1.00 CFZ=0.96 CFY=1.00 CLZ=1 000 CLY=1 000  
 ACTUAL STRESSES : fa= 71 90,fbz=1220 48,fbx= 0 00,fcz= 30 51,fcy= 0 00  
 ALLOW STRESSES : FA= 652 22,FBZ=1600 00,FBY= 950 00,FVZ=140 00,FVY=130 00

## Output Results and Parameters Explained

For CODE CHECKING and/or MEMBER SELECTION the output results are printed as shown in the previous section. The items are explained as follows.

- a) MEMBER refers to the member number for which the design is performed.
- b) TABLE refers to the size of the PRISMATIC section (B X D or ZD X YD).
- c) RESULT prints whether the member has PASSEd or FAILed.
- d) CRITICAL COND refers to the CLAUSE or FORMULA NO from the TIMBER CONSTRUCTION MANUAL (3rd. Edition, AITC-1985) which governed the design. See following table:

Critical Condition	Governing Criteria
CLAUSE 5-19	Axial Compression and Bending with MINIMUM ECCENTRICITY.
CLAUSE 5-18	Axial Compression and Bending
CLAUSE 5-42	Axial Tension and Bending
CLAUSE 5-24	Horizontal Shear
CLAUSE 5-40	Lateral stability for net compressive stress in case of Tension and Bending

- e) RATIO prints the ratio of the actual stresses to allowable stresses for the critical condition. This ratio is usually the cumulative ratio of stresses in the interaction formula. In case of shear governing the design, it means the ratio of the actual shear stress to allowable shear stress. If this value exceeds the allowable ratio (default 1.0) the member is FAILed.
- f) LOADING provides the load case number that governed.
- g) FX, MY and MZ provide the design axial force, moment in local Y axes and moment in local Z axes respectively. FX value is followed by a letter C or T to denote COMPRESSION or TENSION.



- h) LOCATION specifies the actual distance from the start of the member to the section where design forces govern in case BEAM command or SECTION command is specified.

OUTPUT parameters that appear within the box are explained as follows.

- a) MEMB refers to the same member number for which the design is performed.
- b) GLULAM GRADE refers to the grade of the timber.
- c) LAM refers to lamination thickness provided in the input or assumed by the program. See INPUT PARAMETERS section.
- d) LZ, LY, LUZ and LUY are the effective lengths as provided or calculated. See INPUT PARAMETERS section.
- e) JZ and JY are the modifiers for the P-DELTA effect about Z-axis and Y-axis respectively. These are calculated by the program.
- f) CDT, CSF, WET, CCR, CTM are the allowable stress modifiers explained in the INPUT PARAMETERS section.
- g) CFZ and CFY are values of the size factors in the Z-axis and Y-axis respectively. CLZ and CLY represent the factors of lateral stability for beams about Z-axis and Y-axis respectively. These values are printed to help the user see the intermediate design values and re-check the design calculations.
- h)  $f_x$ ,  $f_{bx}$ ,  $f_{by}$ ,  $f_{vx}$  and  $f_{vy}$  are the actual axial stress, bending stresses about Z and Y axes and horizontal shear stresses about Z and Y axes respectively. If the bending moments about both axes are less than the eccentric moments based on min. eccentricity then bending stresses are calculated based on the min. eccentricity. Refer DESIGN OPERATIONS section for details.
- i) FA, FBZ, FBY, FVZ and FVY are the final allowable axial, bending (Z and Y axes) and horizontal shear (Z and Y axes) stresses. Refer DESIGN OPERATIONS section for details.



## STAAD Commands and Input Instructions

### Section 5

This section of the manual describes in detail various commands and related instructions for STAAD. The user utilizes a command language format to communicate instructions to the program. Each of these commands either supplies some data to the program or instructs it to perform some calculations using the data already specified. The command language format and conventions are described in Section 5.1. This is followed by a description of the available commands.

Although STAAD input can be created through the Modeling mode. It is very important to understand the command language. With the knowledge of this language, it is easy to understand the problem and add or comment data as necessary. The general sequence in which the commands should appear in an input file should ideally follow the same sequence in which they are presented in this section. However, the commands can be provided in any order with the following exceptions.

- i) All design related data can be provided only after the analysis command.
- ii) All load cases and load combinations must be provided together, except in a case where the CHANGE and RESTORE commands are used. Additional load cases can be provided in the latter part of input.

All input data provided is stored by the program. Data can be added, deleted or modified within an existing data file.

## Input Instructions

### 5.1 Command Language Conventions

This section describes the command language used in STAAD. First, the various elements of the language are discussed and then the command format is described in detail.

#### 5.1.1 Elements of The Commands

- a) **Integer Numbers:** Integer numbers are whole numbers written without a decimal point. These numbers are designated as  $i_1, i_2, \dots$ , etc., and should not contain any decimal point. Signs (+ or -) are permitted in front of these numbers. If the sign is omitted, it is assumed to be positive (+).
- b) **Floating Point Numbers:** These are real numbers which may contain a decimal portion. These numbers are designated as  $f_1, f_2, \dots$ , etc.. Values may have a decimal point and/or exponent. When specifying numbers with magnitude less than 1/100, it is advisable to use the E format to avoid precision related errors.

##### Example

```
5055.32 0.73 -8.9 732
5E3 -3.4E-6
etc.
```

When the sign is omitted, it is assumed to be positive (+). Also note that the decimal point may be omitted, if the decimal portion of the number is zero.

- c) **Alphanumeric:** These are characters which are used to construct the names for data, titles or commands. Alphabetic characters may be input in upper or lower case letters. No quotation marks are needed to enclose them.

##### Example

```
MEMBER PROPERTIES
1 TO 8 TABLE ST W8X35
```

- d) **Repetitive Data:** Repetitive numerical data may be provided by using the following format:



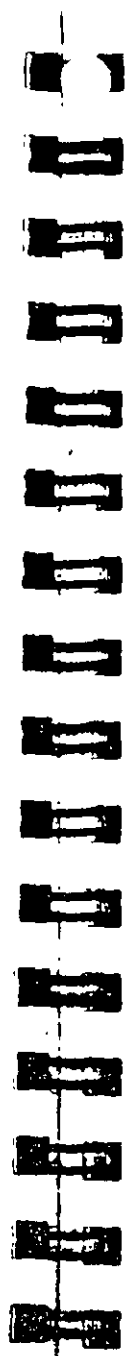
n\*f  
 where n = number of times data has to be repeated  
 f = numeric data, integer and floating point

**Example**

**JOINT COORDINATES**  
**1 3 \* 0.**

This joint coordinate specification is same as:

**1 0. 0. 0.**



**5.1.2 Command Formats**

- a) **Free-Format Input:** All input to STAAD is in free-format style. Input data items should be separated by blank spaces or commas from the other input data items. Quotation marks are never needed to separate any alphabetic words such as data, commands or titles.
- b) **Commenting Input:** For documentation of a STAAD data file, the facility to provide comments is available. Comments can be included by providing an asterisk (\*) mark as the first non-blank character in any line. The line with the comment is "echoed" in the output file but not processed by the program.

**Example**

**JOINT LOAD**  
**\* THE FOLLOWING IS AN EQUIPMENT LOAD**  
**2 3 7 FY 35.0**  
 etc.

- c) **Meaning of Underlining in the Manual:** Exact command formats are described in the latter part of this section. Many words in the commands and data may be abbreviated. The full word intended is given in the command description with the portion actually required (the abbreviation) underlined.

For example, if the word MEMBER is used in a command, only the portion MEMB need be input. It is clearer for others reading the output if the entire word is used, but an experienced user may desire to use the abbreviations.

- d) **Meaning of Braces and Parenthesis:** In some command formats, braces enclose a number of choices, which are arranged vertically. One and only one of the choices can be selected. However, several of the listed choices may be selected if an asterisk (\*) mark is located outside the braces.

**Example**

$$\left\{ \begin{array}{l} XY \\ YZ \\ XZ \end{array} \right\}$$

In the above example, the user must make a choice of XY or YZ or XZ.

**Example**

$$\left\{ \begin{array}{l} FX \\ FY \\ FZ \end{array} \right\}$$

Here the user can choose one or all of the listing (FX, FY and FZ) in any order. Parentheses, ( ), enclosing a portion of a command indicate that the enclosed portion is optional. The presence or absence of this portion affects the meaning of the command, as is explained in the description of the particular command.

**Example**

**PRINT (MEMBER) FORCES**  
**PERFORM ANALYSIS (PRINT LOAD DATA)**

In the first line, the word MEMBER may be omitted with no change of the meaning of the command. In the second line,

**PRINT LOAD DATA**

command may also be omitted, in which case the load data will not be printed.

- e) **Multiple Data Separator:** Multiple data can be provided on a single line, if they are separated by a semicolon (;) character

One restriction is that consecutive commands can not be separated by a semicolon. They must appear on separate lines.

**Example**

**MEMBER INCIDENCES**

1 1 2; 2 2 3; 3 3 4

etc.

**Possible Error:**

**PRINT FORCES; PRINT STRESSES**

In the above case, only the PRINT FORCES command is processed and the PRINT STRESSES command is ignored.

- f) **Listing Data:** In some STAAD command descriptions, the word "list" is used to identify a list of joints, members/elements or loading cases. The format of a list can be defined as follows:

$$\text{list} = \left\{ \begin{array}{l} i_1, i_2, i_3, \dots \\ i_1 \text{ TO } i_2 \text{ (BY } i_3) \\ \underline{X} \text{ or } \underline{Y} \text{ or } \underline{Z} \end{array} \right\}$$

TO means all integers from the first ( $i_1$ ) to the second ( $i_2$ ) inclusive. BY means that the numbers are incremented by an amount equal to the third data item ( $i_3$ ). If BY  $i_3$  is omitted, the increment will be set to one. Sometimes the list may be too long to fit on one line, in which case the list may be continued to the next line by providing a hyphen preceded by a blank. Also note that only a list may be continued and not any other type of data.

Instead of a numerical list, the specification X ( or Y or Z) may be used. This specification will include all MEMBERS parallel to the global direction specified. Note that this is not applicable to JOINTS or ELEMENTS.

**Example**

2 4 7 TO 13 BY 2 19 TO 22 -  
 28 31 TO 33 FX 10.0  
 This list of items is the same as:  
 2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0  
 Possible Error:  
 3 5 TO 9 11 15 -  
 FX 10.0

In this case, the continuation mark for list items is used when list items are not continued. This will result in an error message or possibly unpredictable results.

### 1.3 Listing of Members by Specification of Global Ranges

This command allows the user to specify lists of members/elements by providing global ranges. The general format of the specification is as follows.

General format:

$\left\{ \begin{array}{l} \text{X} \text{RANGE} \\ \text{Y} \text{RANGE} \\ \text{Z} \text{RANGE} \end{array} \right\} f_1, f_2$

where,  
 XRANGE, YRANGE, ZRANGE = direction of range (parallel to global X, Y, Z directions respectively)

f1, f2 = values (in current unit system) that defines the specified range.



#### Notes

- 1) Only one range direction (XRANGE, YRANGE etc.) is allowed per list.
- 2) The values defining the range (f1, f2) must be in the current unit system.

#### Example

**MEMBER TRUSS**  
**XRANGE 20. 70.**  
**CONSTANTS**  
**E STEEL YRANGE 10. 55.**

In the above example, a XRANGE is specified with values of 20. and 70. This range will include all members lying entirely within a range parallel to the global X-axis and limited by X=20 and X=70.

# STAAD Commands

## 2 Problem Initiation And Title

### Purpose

This command initiates the STAAD run, allows the user to specify the type of the structure and an optional title.

### General format:

STAAD  $\left\{ \begin{array}{l} \text{PLANE} \\ \text{SPACE} \\ \text{TRUSS} \\ \text{FLOOR} \end{array} \right\}$  (any title a<sub>1</sub>)

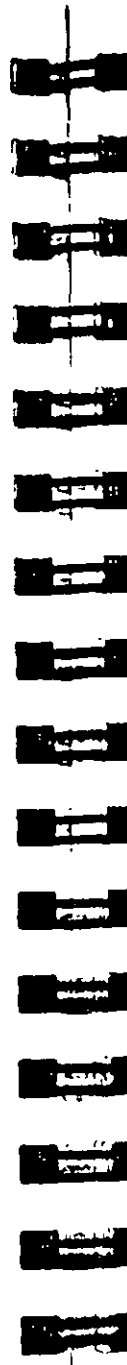
### Description

Any STAAD input has to start with the word STAAD. Following type specifications are available:

- PLANE = Plane frame structure
- SPACE = Space frame structure
- TRUSS = Plane or space truss structure
- FLOOR = Floor structure

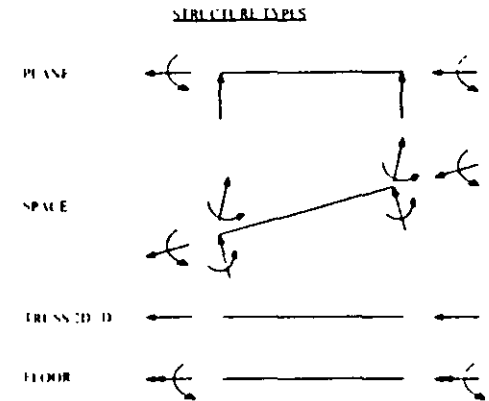
a<sub>1</sub> = Any title for the problem. This title will appear on the top of every output page. To include additional information in the page header, use a comment line containing the pertinent information as the second line of input.

See Section 1.3



### Notes

- 1) The user should be careful about choosing the type of the structure. The choice is dependent on the various degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedoms considered in the various type specifications. Detailed discussions are available in Section 1.3.



- 2) The optional title provided by the user is printed on top of every page of the output. The user can use this facility to customize his output

### 5.3 Unit specification

**Purpose**

This command allows the user to specify or change length and force units for input and output.

**General format:**

UNIT     { length-unit  
                  force-unit }

length-unit = { INCHES  
                  FEET or FT  
                  CM  
                  METER  
                  MMS  
                  DME  
                  KM }

force-unit = { KIP  
                  POUND  
                  KG  
                  MTON  
                  NEWTON  
                  KNS  
                  MNS  
                  DNS }

**Note:**

DME denotes Decimeters. MNS denotes mega Newtons (1000 Newtons) and DNS denotes DecaNewtons (10 newtons). MTON denotes Metric Ton (1000 kilograms). All other units are self explanatory.

**Description**

The UNIT command can be specified any number of times during an analysis. All data is assumed to be in the most recent unit specification preceding that data. Also note that the input-unit for

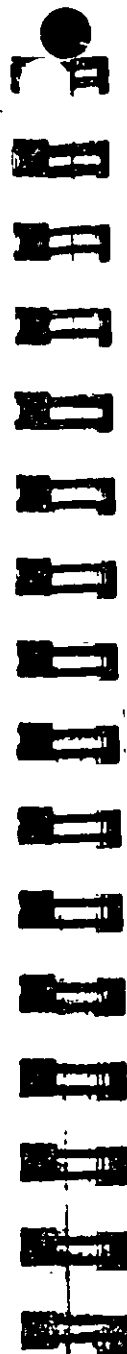
angles is always degrees. However, the output unit for joint rotations (in joint displacement) is radians. For all output, the units are clearly specified by the program.

**Example**

UNIT KIP FT  
UNIT INCH  
UNIT CM MTON

**Notes**

This command may be used as frequently as needed to specify data or generate output in the desired length and/or force units. Note that mix and match between different unit systems (Imperial, Metric, SI etc ) are allowed.



## 5.4 Input/Output Width Specification

### Purpose

These commands may be used to specify the width(s) of the lines of the input and/or output file(s).

### General format:

```

{ INPUT
  OUTPUT } WIDTH i1
    
```

For INPUT WIDTH.

$i_1$  = Number of characters per line in the input file to be processed.

For OUTPUT WIDTH.

$i_1$  = 72 or 118 depending on narrow or wide output.

### Description

The user may specify the required input/output width, as required, using this command. For INPUT width, any number up to 79 may be provided. The default input width is 72. If the input width is more than 72, the output must be printed on wider paper, on a printer capable of handling up to 118 characters. It should be remembered that a narrower input width results in faster line scanning and therefore increases the speed of execution. The program can create output using two different output widths - 72 (default) and 118. The 72-character width may be used for display on most CRTs and for printing on 8-1/2" wide paper. The 118-character width may be used for printing on 11" wide paper.

### Notes

This is a customization facility that may be used to improve the presentation quality of the run documents.

## 5.5 Set Command Specification

### Purpose

This command allows the user to set various general specifications for the analysis/design run.

### General format:

```

SET { NL
     CONNECTIVITY
     DISPLACEMENT } i1
    { ECHO { ON
           OFF } } i2
    Z UP i3
    
```

where.

- $i_1$  = Maximum number of primary load cases (NL)
- $i_2$  = Maximum number of members (or elements) connected to a joint.
- $i_3$  = Maximum allowable displacement for any joint in the structure.

### Description

The SET NL command is used in a multiple analysis run if the user wants to add more primary load cases after one analysis has been performed. Specifically, for those examples, which use the CHANGE or RESTORE command, if the user wants to add more primary load cases, the NL value should be set to the maximum number with the SET NL command. The program will then be able to set aside additional core space for information to be added later. Note that this command should be provided before any joint, member or load specifications. The value for  $i_1$  should not be greater than the maximum number of primary load cases.

The SET CONNECTIVITY command may also be used to specify the maximum number of members (or elements) that may be connected to a joint. The default value is set to 16 in the program.

See Sections  
5.18 and 5.3

If it is necessary to connect more than 16 members (or elements) to a joint, the SET CONNECTIVITY option may be used. This command should be positioned before the JOINT COORDINATE specifications.

The SET DISPLACEMENT command is used to specify the limiting value of displacement for use in convergence checks of displacements wherever applicable such as a NON-LINEAR ANALYSIS. This command should be placed before the JOINT COORDINATE specification.

The SET ECHO ON command will activate and the SET ECHO OFF command will deactivate the echoing of input file commands in the output file. In the absence of the SET ECHO command, input file commands will be echoed back to the output file.

By default, the Y-axis is the vertical axis. However, the SET Z UP command may be used to model situations where Z-axis represents the vertical axis (direction of gravity load) of the structure. This situation may arise if the input geometry is created through some CAD software. Note that this command will affect the default BETA angle specification. However, BETA can be set to a certain value for all members parallel to a particular global axis by using the MEMBER X (or Y or Z) type of listing. For additional information, see the CONSTANTS specification (Section 5.26).

**Notes**

The SET Z UP Command directly influences the values of the following input:

- 1) JOINT COORDINATE
- 2) Input for the PERFORM ROTATION Command
- 3) BETA ANGLE



The following features of STAAD cannot be used with the SET Z UP command:

- 1) Wind Load Generation
- 2) Floor Load Generation
- 3) Automatic Generation of Spring Supports for Mat Foundations

## 6 Separator Command

### Purpose

This command may be used to specify the desired separator character that can be used to separate multiple lines of data on a single line of input.

### General format:

**SEPARATOR a<sub>1</sub>**

### Description

The semicolon (;) is the default character which functions as the separator for multiple line data on one line. However, this separator character can be changed by the SEPARATOR command to any character a<sub>1</sub>, other than the comma or asterisk.

### Notes

Comma (,) or asterisk (\*) may not be used as a separator character



## 5.7 Page New Command

### Purpose

This command may be used to instruct the program to start a new page of output.

### General format:

**PAGE NEW**

### Description

With this command, a new page of output can be started. This command provides the flexibility, the user needs, to design the output format.

### Notes

The presentation quality of the output document may be improved by using this command properly.



## 5.8 Page Length/Eject Command

### Purpose

These commands may be used to specify the page length of the output and the desired page eject character.

### General format:

$$\text{PAGE} \begin{cases} \text{LENGTH} & i \\ \text{EJECT} & a_1 \end{cases}$$

The page length in STAAD output is based on a default value of 60 lines. However, the user may change the page length to any number  $i$  (number of lines per page) desired.

### Description

Standard page eject character (CTRL L for PCs and I for Mini/Mirm) is embedded in the STAAD program. The PAGE EJECT command with the input of the character  $a_1$  will alter the default page eject character in the program. A blank character will suppress page ejection.

## 5.9 Ignore Specifications

### Purpose

This command allows the user to provide member lists in a convenient way without triggering error messages pertaining to non-existent member numbers.

### General format:

IGNORE LIST

### Description

IGNORE LIST may be used if the user wants the program to ignore any nonexistent member that may be included in a member list specification. For example, for the sake of simplicity, a list of members may be specified as MEMB 3 TO 40 where members 10 and 11 do not exist. An error message can be avoided in this situation by providing the IGNORE LIST command anywhere in the beginning of input. A warning message, however, will appear for each nonexistent member.

## 10 No Design Specification

### Purpose

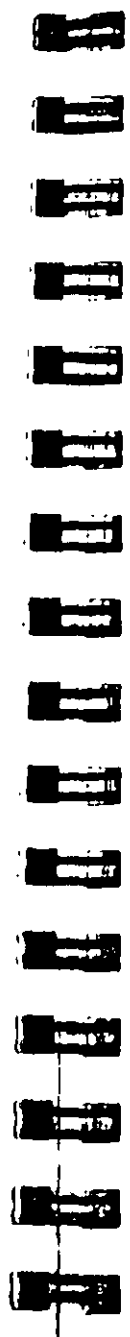
This command allows the user to declare that no design operations will be performed during the run. The memory reserved for design will be released to accommodate larger analysis jobs.

### General format:

**INPUT NODESIGN**

### Description

STAAD always assumes that at some point in the input, the user may wish to perform design for steel or concrete members. These design processes require more computer memory. If memory availability is a problem, the above command may be used to eliminate extra memory requirements.



## 5.11 Joint Coordinates Specification

### Purpose

These commands allow the user to specify and generate the coordinates of the JOINTS of the structure. The JOINT COORDINATES command initiates the specification of the coordinates. The REPEAT and REPEAT ALL commands allow easy generation of coordinates using repetitive patterns.

### General format:

**JOINT COORDINATES (CYLINDRICAL (REVERSE)) (NOCHECK) band-spec**

$i_1, x_1, y_1, z_1, (i_2, x_2, y_2, z_2, i_3)$

**REPEAT** n,  $x_1, y_1, z_1, (x_2, y_2, z_2, \dots, x_n, y_n, z_n)$

**REPEAT ALL** n,  $x_1, y_1, z_1, (x_2, y_2, z_2, \dots, x_n, y_n, z_n)$

**band-spec = (NOREDUCE BAND)**

**NOCHECK=** Do not perform check for multiple structures or orphan joints.

### Description

The command JOINT COORDINATES specifies a Cartesian Coordinate System (see Figure 2.2). Joints are defined using the global X, Y and Z coordinates. The command JOINT COORDINATES CYLINDRICAL specifies a Cylindrical Coordinate System (see Figure 2.3). Joints are defined using the r,  $\theta$  and z coordinates. JOINT COORDINATES CYLINDRICAL REVERSE specifies a Reverse Cylindrical Coordinate system (see Figure 2.4). Joints are defined using the r, y and  $\theta$  coordinates. NOREDUCE BAND causes the program to execute without performing a bandwidth reduction.

See Section 1.5.1

### Example

**JOINT COORDINATES NOREDUCE BAND**

The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified coordinate increments. The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command, or all joint data if no previous REPEAT ALL command has been given. (When using the REPEAT and REPEAT ALL commands, joint numbering must be consecutive and should begin with 1.)

$i_1 =$  The joint number for which the coordinates are provided. Any integer number (five digit max.) is permitted.

$x_1, y_1$  and  $z_1 =$  X, Y & Z (R,  $\theta$  & Z for cylindrical or R, Y &  $\theta$  for cylindrical reverse) coordinates of the joint.

For PLANE analyses  $z_1$  is an optional data item when defining input for individual joints.  $z_1$  is always required for joint generation. The following are used only if joints are to be generated.

$i_2 =$  The second joint number to which the joint coordinates are generated.

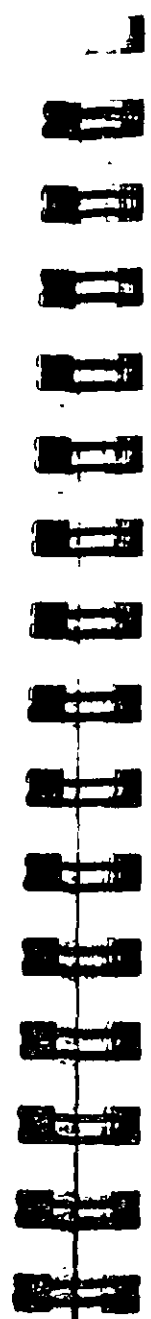
$x_2, y_2,$  and  $z_2 =$  X, Y & Z (R,  $\theta$  & Z for cylindrical or R, Y &  $\theta$  for cylindrical reverse) coordinates of the joint  $i_2$ .

$i_3 =$  Joint number increment by which the generated joints will be incremented. Defaults to 1 if left out.

$n =$  Number of times repeat is to be carried out. Note that "n" cannot exceed 98 in any one single REPEAT command.

$x1_k, y1_k$  &  $z1_k =$  X, Y & Z (R,  $\theta$  & Z (R, Y &  $\theta$ ) coordinate increments for k th repeat.

The X, Y and Z (R,  $\theta$  & Z (R, Y &  $\theta$ ) coordinates will be equally spaced between  $i_1$  and  $i_2$



### Example 1

#### JOINT COORDINATES

```
1 10.5 2.0 8.5
2 0.0 0.0 0.0
3 5.25 0.0 8.5 6 50.25 0.0 8.5
```

In this example, X Y Z coordinates of joints 1 to 6 are provided. Note that the joints between 3 & 6 will be generated with joints equally spaced from 3 to 6. Hence, joint 4 will have coordinates of 20.25 0.0 8.5 and joint 5 will have coordinates of 35.25 0.0 8.5.

### Example 2

#### JOINT COORDINATES

```
1 0.0 0.0 0.0 4 45 0.0 0.0
REPEAT 4 0.0 0.0 15.0
REPEAT ALL 10 0.0 10.0 0.0
```

Here, the 220 joint coordinates of a ten story 3 X 4-bay structure are generated. The REPEAT command repeats the first input line 4 times, incrementing each Z coordinate by 15. Thus, the first 2 lines are sufficient to create a "floor" of twenty joints.

```
1 0.0 0.0 ; 2 15.0 0.0 ; 3 30.0 0.0 ; 4 45.0 0.0
5 0.0 0.15 ; 6 15.0 0.15 ; 7 30.0 0.15 ; 8 45.0 0.15
17 0.0 0.60 ; 18 15.0 0.60 ; 19 30.0 0.60 ; 20 45.0 0.60
```

The REPEAT ALL command repeats all previous data (i.e. the 20 joint "floor") ten times, incrementing the Y coordinate by 10 each time. This creates the 200 remaining joints of the structure:

**Example 3**

```

21 0.0 10.0 0.0 ; 22 15.0 10.0 0.0 ; ... ;
40 45.0 10.0 60.0 ; 41 0.0 20.0 0.0 ; ... ;
200 45.0 90.0 60.0 ; 201 0.0 100.0 0.0 ; ... ;
219 30.0 100.0 60.0 ; 220 45.0 100.0 60.0

```

The following examples illustrate various uses of the REPEAT command.

```
REPEAT 10 5. 10. 5.
```

The above REPEAT command will repeat the last input line 10 times using the same set of increments (i.e.  $x = 5$ ,  $y = 10$ ,  $z = 5$ )

```
REPEAT 3 2. 10. 5. 3. 15. 3. 5. 20. 3.
```

The above REPEAT command will repeat the last input line three times. Each repeat operation will use a different increment set.

```
REPEAT 10 0. 12. 0. 15*0 0. 10. 0. 9*0
```

The above REPEAT command will repeat the last input line 10 times; six times using  $x$ ,  $y$  and  $z$  increments of 0, 12, and 0, and four times using increments of 0, 10, and 0. Each  $x$ ,  $y$  and  $z$  value of 0 represents no change from the previous increment. To create the 2nd through 6th repeats, five sets of 0, 0, and 0. (15\*0) are supplied. The seventh repeat is done with increments of 0, 10 and 0. The 8th through 10th repeats are done with the same increments as 7, and is represented as 9\*0.

**Notes**

The PRINT JOINT COORDINATE command may be used to verify the joint coordinates provided or generated by REPEAT and REPEAT ALL commands.

Also, use the Post Processing facility to verify geometry graphically.

## 5.12 Member Incidences Specification

### Purpose

This set of commands is used to specify MEMBERS by defining connectivity between JOINTS. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The member/clement incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD is capable of detecting multiple structures automatically.

### General format:

#### MEMBER INCIDENCES

$i_1, i_2, i_3, (i_4, i_5, i_6)$

REPEAT  $n, m_1, j_1$

REPEAT ALL  $n, m_1, j_1$

### Description

The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified member and joint increments. The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command or to the beginning of the specification if no previous REPEAT ALL command has been issued. (When using REPEAT and REPEAT ALL commands, member numbering must be consecutive).

$i_1$  = Member number for which incidences are provided. Any integer number (maximum six digits) is permitted.

$i_2$  = Start joint number.

$i_3$  = End joint number.



The following data are used for member generation only:

- $i_4$  = Second member number to which members will be generated.
- $i_5$  = Member number increment for generation.
- $i_6$  = Joint number increment which will be added to the incident joints. ( $i_5$  and  $i_6$  will default to 1 if left out.)
- $n$  = Number of times repeat is to be carried out.
- $m_1$  = Member number increment
- $j_1$  = Joint number increment

### Example

#### MEMBER INCIDENCES

```
1 1 2
2 5 7 5
7 11 13 13 2 3
```

In this example, member 1 goes from joint 1 to 2. Member 2 is connected between joints 5 and 7. Member numbers from 3 to 5 will be generated with a member number increment of 1 and a joint number increment 1 (by default). That is, member 3 goes from 6 to 8, member 4 from 7 to 9, member 5 from 8 to 10. Similarly, in the next line, member 9 will be from 14 to 16, 11 from 17 to 19 and 13 from 20 to 22.

### Additional example

#### MEMBER INCIDENCES

```
1 1 21 20
21 21 22 23
REPEAT 4 3 4
38 21 25 39
REPEAT 3 4 4
REPEAT ALL 9 51 20
```

This example creates the 510 members of a ten story 3 X 4-bay structure (this is a continuation of the example started in Section 5.12). The first input line creates the twenty columns of the first floor:

```
1 1 21 ; 2 2 22 ; 3 3 23 ; ... ; 19 19 39 ; 20 20 40
```

The two commands (21 21 22 23 and REPEAT 4 3 4) create 15 members which are the second floor "floor" beams running, for example, in the east-west direction.

```
21 21 22; 22 22 23; 23 23 24
24 25 26; 25 26 27; 26 27 28
... ..
33 37 38; 34 38 39; 35 39 40
```

The next two commands (36 21 25 39 and REPEAT 3 4 4) function similar to the previous two commands, but here create the 16 second floor "floor" beams running in the north-south direction.

```
36 21 25; 37 22 26; 38 23 27; 39 24 28
40 25 29; 41 26 30; 42 27 31; 43 28 32
... ..
48 33 37; 49 34 38; 50 35 39; 51 36 40
```

The preceding commands have created a single floor unit of both beams and columns, a total of 51 members. The REPEAT ALL now repeats this unit nine times, generating 459 new members and finishing the ten story structure. The member number is incremented by 51 (the number of members in a repeating unit) and the joint number is incremented by 20, (the number of joints on one floor).



**Notes**

The PRINT MEMBER INFO command may be used to verify the member incidences provided or generated by REPEAT and REPEAT ALL commands.

Also, use the Post Processing facility to verify geometry graphically

## i.13 Element Incidence Specification

### Purpose

This set of commands is used to specify ELEMENTs by defining the connectivity between JOINTs. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD is capable of detecting multiple structures automatically.

### Plate / Shell Element Incidences

#### General format

**ELEMENT INCIDENCES (SHELL)**

$i_1, i_2, i_3, i_4, (i_5), (IQ i_6, i_7, i_8)$

**REPEAT**  $n, e, j$

**REPEAT ALL**  $n, e, j$

#### Description

ELEMENT INCIDENCES SHELL must be provided immediately after MEMBER INCIDENCES (if any) are specified. The REPEAT command causes the previous line of input to be repeated 'n' number of times with specified element and joint increments. The REPEAT ALL command functions similar to the REPEAT command, except that it repeats all previously specified input back to the most recent REPEAT ALL command; or to the beginning of the specification if no previous REPEAT ALL command had been issued.

5 Section  
16

- $i_1$  = Element number (any number up to six digits). If MEMBER INCIDENCE is provided, this number must not coincide with any MEMBER number.
- $i_2 \dots i_5$  = Clockwise or counterclockwise joint numbers which represent the element connectivity. Note that  $i_5$  is not needed for triangular (3 noded) elements.

The following data is needed if elements are to be generated:

- $i_6$  = Last element number to which elements are generated.
- $i_7$  = Element number increment by which elements are generated. Defaults to 1 if omitted.
- $i_8$  = Joint number increment which will be added to incident joints. Defaults to 1 if omitted.

The following data is needed if REPEAT or REPEAT ALL commands are used to generate elements:

- $n$  = Number of times repeat is to be carried out.
- $e$  = Element number increment.
- $j$  = Joint number increment.

#### Example

##### ELEMENT INCIDENCE

```
1 1 2 7 6
2 3 4 8
3 8 9 11 10 TO 8
9 1 3 7 TO 14
```

#### Notes

The PRINT ELEMENT INFO command may be used to verify the element incidences provided or generated by REPEAT and REPEAT ALL commands.

Also, use the Post Processing facility to verify geometry graphically.

### Solid Element Incidences

#### General format

The element incidences for solid elements are to be identified using the expression SOLID to distinguish them from PLATE/SHELL elements.

#### ELEMENT INCIDENCES SOLID

$i_1, i_2, i_3, i_4, i_5, i_6, i_7, i_8, i_9, (TO i_{10}, i_{11}, i_{12})$

REPEAT  $n, e_i, j_i$

REPEAT ALL  $n, e_i, j_i$

#### Description

ELEMENT INCIDENCES SOLID must be provided immediately after MEMBER INCIDENCES (if any) are specified as well as after the ELEMENT INCIDENCES SHELL (if any)

- $i_1$  = Element number
- $i_2, i_3, i_4$  = Joint numbers of the solid element
- $i_{10}$  = Last element number to be generated
- $i_{11}$  = Element number increment
- $i_{12}$  = Joint number increment
- $n$  = Number of times REPEAT is to be carried out
- $e_i$  = Element number increment
- $j_i$  = Joint number increment

Specify the four nodes of any of the faces of the solid element in a counter-clockwise direction as viewed from the outside of the element and then go to the opposite face and specify the four nodes of that face in the same direction used while specifying the nodes of the first face.



#### Example

```

ELEMENT INCIDENCE SOLID
1 1 5 6 2 21 25 26 22 TO 3
4 21 25 26 22 41 45 46 42 TO 6
    
```



## 5.14 Element Mesh Generation

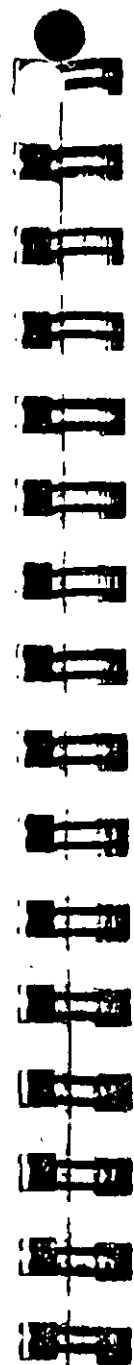
### Purpose

This set of commands is used to generate finite element meshes. The procedure involves the definition of super-elements which are subsequently divided into smaller elements.

### Description

This is the second method for the generation of element incidences. If the user needs to divide a big element into a number of small elements, he may use this facility which generates the joint numbers and joint coordinates, the element numbers and the element incidences automatically. Use of this feature consists of two parts:

1. Definition of the super element boundary points: A super-element may be defined by either 4 boundary points or 8 boundary points ( see figure on next page). A boundary point is denoted by a unique alphabet (A-Z in upper case or a-z in lower case) and its corresponding coordinates. Hence, any 4 or 8 of the 52 characters may be used to define the super-element boundary. If 4 points are used to define the super element, each side of the super-element will be assumed to have a straight edge connecting the 2 points defining that side. If 8 points are used, each side will be a smooth curve connecting the 3 points defining that side.
2. Generation of sub-elements: define the super-element using boundary points (4 or 8 as explained above) and specify the total number of sub-elements required.



General Format:

```

DEFINE MESH ( { CYL } (x0,y0,z0)
A1 x1 y1 z1
...
A1 x1 y1 z1
GENERATE ELEMENT { (QUADRILATERAL) }
                  { TRIANGULAR }

MESH A1 A1 ..... n1 (n2)
MESH Am An ..... n3 (n4)
.....
    
```

where

A<sub>1</sub>, A<sub>j</sub> = Alphabets A - Z or alphabets a - z. That is max 52.

x<sub>1</sub>, y<sub>1</sub>, z<sub>1</sub> = Coordinates for boundary point A<sub>1</sub>.

If CYL or RCYL is defined, above coordinates will be in cylindrical or reverse cylindrical coordinates system. Optional coordinates x<sub>0</sub>, y<sub>0</sub> and z<sub>0</sub> will be the cartesian coordinates for the origin of the cylindrical coordinates. Defaults to 0, 0, 0 if not provided.

A<sub>1</sub>, A<sub>j</sub>, A<sub>k</sub> = A rectangular super-element defined by four or eight boundary points

n<sub>1</sub> = Number of elements along the side A<sub>1</sub> A<sub>j</sub> of the super-element. (Should not exceed 20)

n<sub>2</sub> = Number of elements along the side A<sub>j</sub> A<sub>k</sub> of the super-element. (Should not exceed 20)

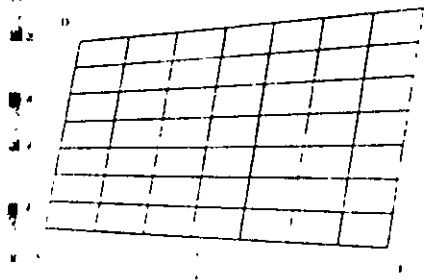
If n<sub>2</sub> is omitted, that is, only n<sub>1</sub> is provided, then n<sub>1</sub> will indicate the total number of elements within the super-element. In this case, n<sub>1</sub> must be the square of an integer.

### Notes

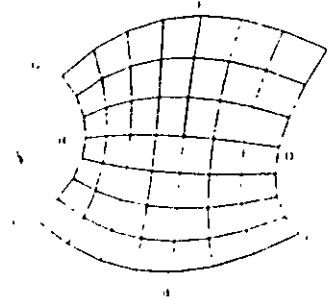
All coordinates are in current unit system. While using this facility the user has to keep the following points in mind:

- 1) All super-elements must be 4-noded or 8-noded. Generated elements for 4-noded super-elements will retain the straight-line edges of the super-elements, while joints of elements

generated from 8-noded super-elements will lie on a curved trajectory

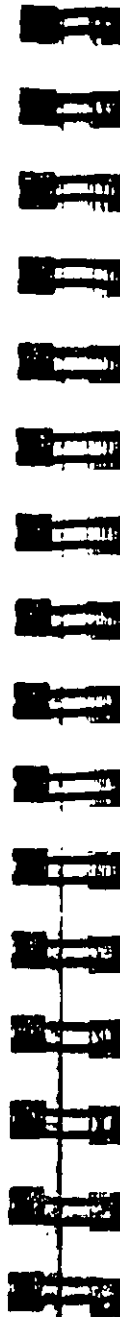


Mesh generated for a 4-noded super element

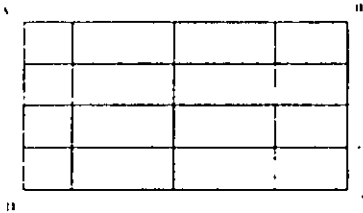


Mesh generated for a 4-noded super element

- 2) 2 super-elements which have a common boundary must have the same number of elements along their common boundary.
- 3) Sequence of super-elements - MESH commands define the super-elements. The sequence of this MESH command should be such that once one is defined, the next super-elements should be the ones connected to this. Therefore, for convenience, the first super-element should be the one which is connected by the largest number of super-elements. In the example shown here for the tank, the bottom super-element is specified first.
- 4) This command must be used after the MEMBER INCIDENCE & ELEMENT INCIDENCE section and before the MEMBER PROPERTIES & ELEMENT PROPERTIES section. The elements that are created internally are numbered sequentially with an increment of one starting from the last member/element number plus one. Similarly the additional joints created internally are numbered sequentially with an increment of one starting from the last joint number plus one. It is advisable that users keep the joint numbers and member/element numbers in a sequence with an increment of one starting from one.



- 5) If there are members embracing a super-element which is being meshed, the users will have to take care of the required additions/modifications in the MEMBER INCIDENCE section themselves since a few more new joints might appear on the existing common boundary as a result of meshing the super-element. See the following figure:



Note: If a member exists between points A and B, the user must breakup this member into 4 parts. Members will not be meshed automatically.

- 6) The sub-elements will have the same direction (Clockwise or Anti-clockwise) as the super-elements. For a super-element bounded by four points A, B, C and D, if ABCD, BCDA etc. are in clockwise direction, CBAD or DCBA etc. are in anti-clockwise direction. If the particular super-element is denoted as ABCD, all the sub-elements in it will have a clockwise element incidence in this example.
- 7) Element incidences of the generated sub-elements may be obtained by providing the command PRINT ELEMENT INFORMATION after the 'MESH, .' command in the input file.
- 8) If the STAAD input file contains commands for JOINT COORDINATES, MEMBER INCIDENCES, ELEMENT INCIDENCES and MESH GENERATION, they should be specified in the following order:  
 STAAD SPACE  
 UNIT KIP FEET  
 JOINT COORDINATES  
 ...  
 MEMBER INCIDENCES

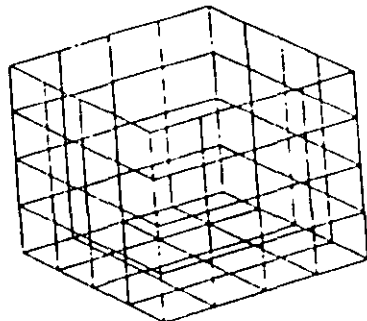
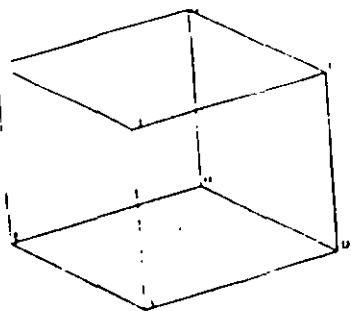
ELEMENT INCIDENCES

DEFINE MESH

GENERATE ELEMENT

**Example**

The following section of input illustrates the use of MESH GENERATION facility. The user may compare this with the geometry inputs for Example Prob. No. 10 in the STAAD example manual.



**STAAD SPACE TANK STRUCTURE WITH**

**\* MESH GENERATION**

**UNIT FEET KIPS**

**DEFINE MESH**

**A 0 0 0 ; B 0 20 0 ; C 20 20 0**

**D 20 0 0 ; E 0 0 -20 ; F 0 20 -20**

**G 20 20 -20 ; H 20. 0. -20**

**GENERATE ELEMENT**

**MESH AEHD 16**

**MESH EABF 16**

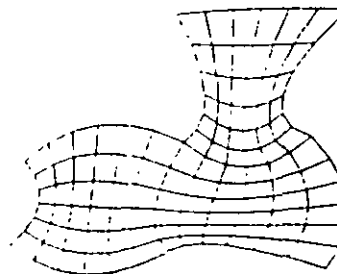
**MESH ADCB 16**

**MESH HEFG 16**

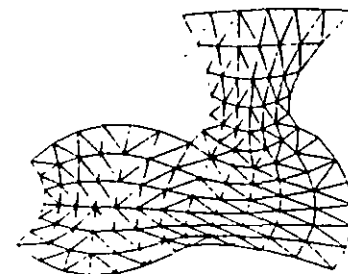
**MESH DHGC 16**



**Typical generated Quad and Triangular elements:**



*Typical generated Quad elements*



*Typical generated Triangular elements*

## 5.15 Redefinition of Joint and Member Numbers

### Purpose

This command may be used to redefine JOINT and MEMBER numbers. Original JOINT and MEMBER numbers are substituted by new numbers.

### General Format:

$$\text{SUBST} \left\{ \begin{array}{l} \text{JOINT} \\ \text{MEMBER} \\ \text{COLUMN} \end{array} \right\} \left\{ \begin{array}{l} \text{X RANGE} \\ \text{Y RANGE} \\ \text{Z RANGE} \end{array} \right\} f_1, f_2 \text{ START } i$$

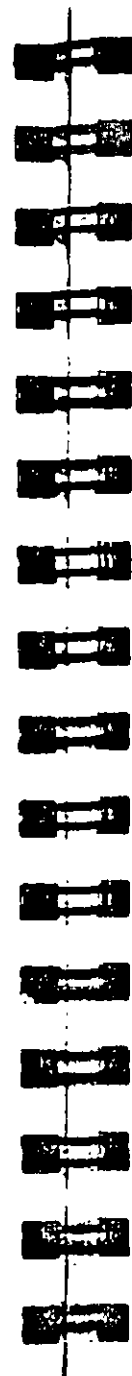
where,  $f_1$  and  $f_2$  are two range values of x, y, or z and  $i$  is the new starting number.

### Description

Joint and member numbers can be redefined in STAAD through the use of the SUBSTITUTE command. After a new set of numbers are assigned, input and output values will be in accordance with the new numbering scheme. The user can design numbering schemes that will result in simple input specification as well as easy interpretation of results. For example, all joints in first floor of a building may be renumbered as 101, 102 ....., all second floor joints may be renumbered as 201, 202 ....., etc.

### Example

```
UNIT FT
SUBST JOINT YR 9.99 10.0 START 101
SUBST COLUMN START 901
```



Joints with Y coordinates ranging from 9.99 to 10 ft will have a new number starting from 101. Columns will be renumbered starting with the new number 901.

### Note

Meaningful respecification of JOINT and MEMBER numbers may significantly improve ease of interpretation of results.

## 5.16 Listing of Members by Specification of GROUPS

This command allows the user to specify a group of members/joints and save the information using a 'group-name'. The 'group-name' may be subsequently used in the input file instead of a member/joint list to specify other attributes. This extremely useful feature allows avoiding of multiple specifications of the same member/joint list. Following is the general format required for the GROUP command.

### General format:

#### START GROUP DEFINITION

#### (GEOMETRY)

\_(group-name) joint-list

.....

OR

#### JOINT

\_(group-name) joint-list

.....

#### MEMBER

\_(group-name) member-list

.....

#### ELEMENT

\_(group-name) element-list

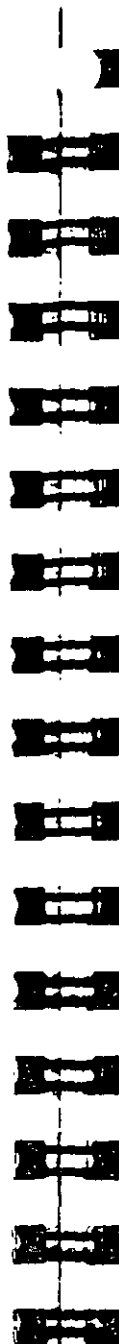
.....

#### SOLID

\_(group-name) solid element-list

.....

#### END GROUP DEFINITION



where,

group-name = an alpha-numeric name specified by the user to identify the group. The group-name must start with the '\_' (underscore) character and is limited to eight characters.

member-list/ = the list of members/joints belonging to the group.  
joint-list

### Notes

- 1) The GROUP definition must start with the START GROUP DEFINITION command and end with the END command.
- 2) More than one GROUP name may be specified within the same definition specification.
- 3) The words JOINT, MEMBER, ELEMENT and SOLID may be provided if the user wishes to identify the group name and lists with those specific items. However, if the group name and list is merely a means of grouping together more than one type of structural component under a single heading, the word GEOMETRY may be provided. In the absence of any of those five words (GEOMETRY, JOINT, MEMBER, ELEMENT or SOLID), the list is assumed to be that for GEOMETRY.

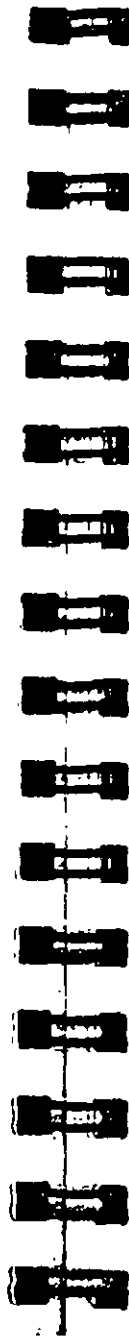
Example

```
START GROUP DEFINITION
_TRUSS 1 TO 20 25 35
_BEAM 40 TO 50
END
MEMBER PROPERTIES
_TRUSS TA LD L40304
_BEAM TA ST W12X26
```

Example

```
START GROUP DEFINITION
JOINT
_TAGA 1 TO 10
MEMBER
_TAGB 40 TO 50
GEOMETRY
_TAGC 101 TO 135
END

MEMBER PROPERTIES
_TAGB TA LD L40304
_TAGC TA ST W12X26
```



## 5.17 Rotation of Structure Geometry

### Purpose

This command may be used to rotate an existing structure geometry about the global axes.

### General format:

$$\text{PERFORM ROTATION } \begin{Bmatrix} X & d_1 \\ Y & d_2 \\ Z & d_3 \end{Bmatrix}$$

where,  $d_1, d_2, d_3$  are the rotations (in degrees) about the X, Y and Z global axes respectively. After the Joint Coordinates and Member Incidences are specified, this command can be used to rotate the structure geometry by any desired angle about any global axis. The rotated configuration is used for analysis and design. While specifying this command, note that the sense of the rotation should conform to the right hand rule.

### Description

This command can be used to rotate the geometric shape through any desired angle about any global axis. The rotated configuration can be used for analysis and design.

### Example

```
PERFORM ROTATION X 20 Z -15
```

### Note

This command should be provided immediately following member incidences if there are only members and after element incidences if there are members and elements.

## 5.18 Inactive/Delete Specification

### Purpose

This set of commands may be used to temporarily INACTIVATE or permanently DELETE specified JOINTs or MEMBERs

### General format:

```

INACTIVE  MEMBERS  member-list
DELETE   { MEMBERS  member-list
             JOINTS   joint-list }
  
```

### Description

These commands can be used to specify that certain joints or members be deactivated or completely deleted from a structure. The INACTIVE command makes the members temporarily inactive, the user must re-activate them during the later part of the input for further processing. The DELETE command will completely delete the members from the structure; the user cannot re-activate them. These commands must be provided immediately after all member/element incidences are provided.

### Notes

- The DELETE MEMBER command will automatically delete all joints associated with deleted members, provided the joints are not connected by any other active members or elements
- This command will also delete all the joints which were not connected to the structure in the first place. For example, such joints may have been generated for ease of input of joint coordinates and were intended to be deleted. Hence, if a DELETE MEMBER command is used, a DELETE JOINT command should not be used.
- The DELETE MEMBER command is applicable for deletion of members as well as elements. If the list of members to be deleted extends beyond one line, it should be continued on to

the next line by providing a blank space followed by a hyphen (-) at the end of the current line

### Example

```

INACTIVE  MEMBERS  5 7 TO 10
DELETE   MEMBERS  29 TO 34 43
  
```

- The INACTIVE MEMBER command cannot be used in situations where inactivating a member results in joints becoming unconnected in space.
- The INACTIVE MEMBER command should not be used if the MEMBER TENSION command is used.
- The INACTIVated members may be restored for further processes (such as an analysis or design for a 2<sup>nd</sup> set of load cases) by using the CHANGE command. See Section 5.37 and Example 4 for more information.
- The DELETE MEMBER command should be used to delete elements too. Specify the command as DELETE MEMBER j where j is the element number of the element you wish to delete. In the example shown above, 29 to 34 and 43 are element numbers.
- Loads that have been defined on members declared as INACTIVE members will not be considered in the analysis. This applies to SELFWEIGHT, MEMBER LOADS, PRESTRESS and POSTSTRESS LOADS, TEMPERATURE LOADS, etc.

## 5.19 User Steel Table Specification

### Purpose

STAAD allows the user to create and use customized Steel Section Table (s) for Property specification, Code checking and Member Selection. This set of commands may be used to create the table(s) and provide necessary data.

### General format:

```
START USER TABLE
TABLE i1 (In)
section-type
section-name
property-spec
END
```

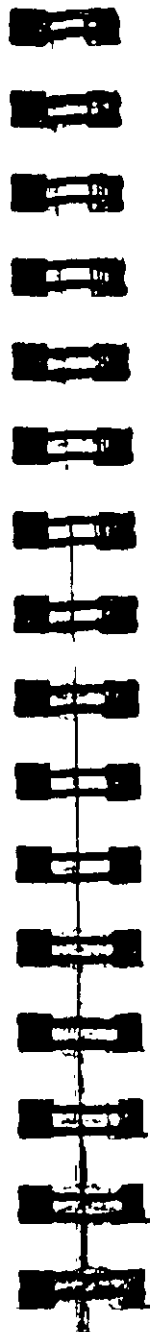
where,

$i_1$  = table number (1 to 20). During the analysis process, the data in each user provided table is stored in a corresponding file with an extension .U0?. For example, the data of the 5th table is stored in U05. The first part of the input file name is the same as that of the STAAD input file. These files are located in the same working directory as the input file. Hence, they may later be used as external user provided tables for other input files.

$I_n$  = external file name containing the section name and corresponding properties.

section-type = a steel section name including: WIDE FLANGE, CHANNEL, ANGLE, DOUBLE ANGLE, TEE, PIPE, TUBE, GENERAL, ISECTION & PRISMATIC.

section-name = Any user designated section name, within 12 characters. First three characters of Pipes and Tubes must be PIP and TUB respectively. Only alphanumeric characters and digits are allowed for defining section names. (Blank spaces, asterisks,



question marks, colon, semi-colon etc. are not permitted.)  
 property-spec = Properties for the section. The requirements are different for each section type as follows. Note that shear areas AY and AZ must be provided to ensure proper shear stress or shear strength calculations during design.  
 The default length units for properties are inch (for American version) and cm (for other versions). However, the user may specify the desired length unit by using the UNIT command as the first command in the table (see example following this description).

### Description

Following section types are available under this option

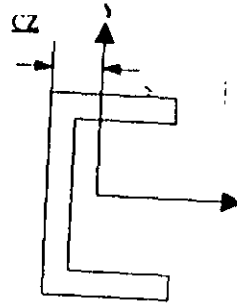
#### Wide Flange

- 1) AX = Cross section area
- 2) D = Depth of the section
- 3) TW = Thickness of web
- 4) WF = Width of the flange
- 5) TF = Thickness of flange
- 6) IZ = Moment of inertia about local z-axis (usually strong axis)
- 7) IY = Moment of inertia about local y-axis
- 8) IX = Torsional constant
- 9) AY = Shear area in local y-axis. If zero, shear deformation is ignored in the analysis.
- 10) AZ = Same as above except in local z-axis.



**Channel**

- 1) AX, 2) D, 3) TW, 4) WF, 5) TF, 6) IZ, 7) IY, 8) IX, 9) CZ,
- 10) AY, 11) AZ



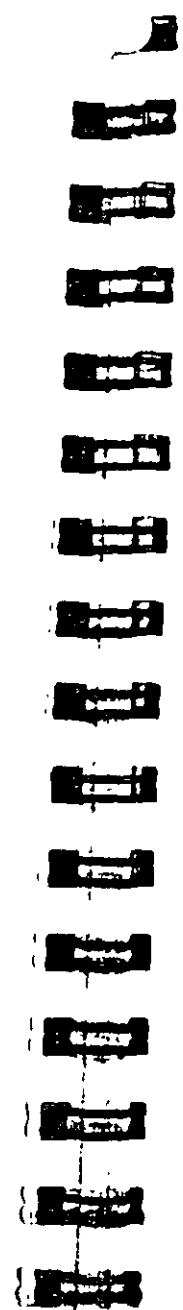
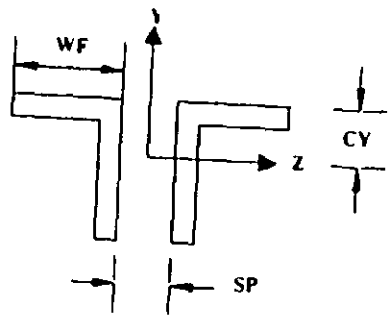
**Angle**

- 1) D, 2) WF, 3) TF, 4) R, 5) AY, 6) AZ

R = radius of gyration about principal axis, shown as r(Z-Z) in the AISC manual

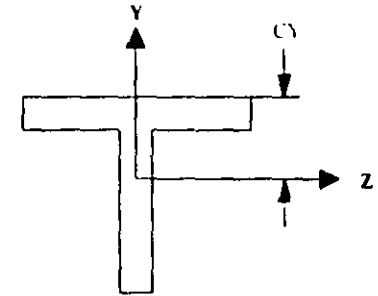
**Double Angle**

- 1) D, 2) WF, 3) TF, 4) SP, 5) IZ, 6) IY, 7) IX, 8) CY, 9) AY,
- 10) AZ



**Tee**

- 1) AX, 2) D, 3) WF, 4) TF, 5) TW, 6) IZ, 7) IY, 8) IX, 9) CY,
- 10) AY, 11) AZ



**Pipe**

- 1) OD = Outer diameter
- 2) ID = Inner diameter
- 3) AY, 4) AZ

**Tube**

- 1) AX, 2) D, 3) WF, 4) TF, 5) IZ, 6) IY, 7) IX, 8) AY, 9) AZ

**General**

The following cross-sectional properties should be used for this section-type. Note that this facility allows the user to specify a built-up or unconventional Steel Section.

- 1) AX = Cross section area.
- 2) D = Depth of the section.
- 3) TD = Thickness associated with section element parallel to depth (usually web). To be used to check depth/thickness ratio.
- 4) B = Width of the section.
- 5) TB = Thickness associated with section element parallel to flange. To be used to check width/thickness ratio.

- 6) IZ = Moment of inertia about local z-axis
- 7) IY = Moment of inertia about local y-axis.
- 8) IX = Torsional Constant.
- 9) SZ = Section modulus about local z-axis
- 10) SY = Section modulus about local y-axis
- 11) AY = Shear area for shear parallel to local y-axis
- 12) AZ = Shear area for shear parallel to local z-axis
- 13) PZ = Plastic modulus about local z-axis.
- 14) PY = Plastic modulus about local y-axis
- 15) HSS = Warping constant for lateral torsional buckling calculations
- 16) DEE = Depth of web For rolled sections, distance between fillets should be provided

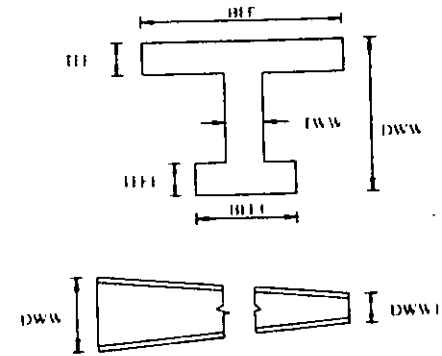
**Note:**

Properties PZ, PY, HSS and DEE must be provided for code checking/member selection per plastic and limit state based codes (AISC LRFD, British, French, German and Scandinavian codes). For codes based on allowable stress design (AISC-ASD, AASHTO, Indian codes), zero values may be provided for these properties.

**Isection**

This section type may be used to specify a generalized I-shaped section. The cross-sectional properties required are listed below. Note that this facility can be utilized to specify tapered I-shapes.

- 1) DWW = Depth of section at start node
- 2) TWW = Thickness of web.
- 3) DWW1 = Depth of section at end node.
- 4) BFF = Width of top flange.
- 5) TFF = Thickness of top flange.
- 6) BFF1 = Width of bottom flange.
- 7) TFF1 = Thickness of bottom flange.
- 8) AYF = Shear area for shear parallel to Y-axis.
- 9) AZF = Shear area for shear parallel to Z-axis.
- 10) XIF = Torsional constant.



**NOTE:**

- 1) DWW should never be less than DWW1. The user should provide the member incidences accordingly.
- 2) The user is allowed the following options for the values AYF, AZF and XIF.
  - a) If positive values are provided, they are used directly by the program.
  - b) If zero is provided, the program calculates the properties using the following formula
 
$$AYF = D \times TWW \text{ (where } D = \text{Depth at section under consideration)}$$

$$AZF = 0.66 ((BFF \times TFF) + (BFF1 \times TFF1))$$

$$XIF = 1/3 ((BFF \times TFF^3) + (DEE \times TWW^3) + (BFF1 \times TFF1^3))$$
 (where DEE = Depth of web of section)
  - c) If negative values are provided, they are applied as factors on the corresponding value(s) calculated by the program using the above formula. The factor applied is always the absolute of the value provided, i.e. if the user provides the value of XIF as -1.3, then the program will multiply the value of XIF, calculated by the above formula, by a factor of 1.3.

**Prismatic**

The property-spec for the PRISMATIC section-type is as follows -

- 1) AX = Cross-section area
- 2) IZ = Moment of inertia about the local z-axis
- 3) IY = Moment of inertia about the local y-axis
- 4) IX = Torsional constant
- 5) AY = Shear area for shear parallel to local y-axis.
- 6) AZ = Shear area for shear parallel to local z-axis.
- 7) YD = Depth of the section in the direction of the local y-axis.
- 8) ZD = Depth of the section in the direction of the local z-axis.

**Example**

**START USER TABLE**

**TABLE 1**

**UNIT INCHES**

**WIDE FLANGE**

**W14X30**

8.85 13.84 .27 6.73 .385 291. 19.6 .38 0 0

**W21X50**

14.7 20.83 .38 6.53 .535 984 24.9 1.14 7.92 0

**W14X109**

32. 14.32 .525 14.605 .86 1240 447 7.12 7.52 0

**TABLE 2**

**UNIT INCHES**

**ANGLES**

**L25255**

2.5 2.5 0.3125 .489 0 0

**L40404**

4. 4. .25 .795 0 0

**END**

- Note that these section-names must be provided in ascending order by weight, since the member-selection process uses these tables and the iteration starts from the top. The above example can also be input as follows:



**START USER TABLE**

**TABLE 1 TFILE1**

**TABLE 2 TFILE2**

**END**

Where TFILE1 and TFILE2 are names of files which must be created prior to running STAAD, and where the file TFILE1 will contain the following:

**UNIT INCHES**

**WIDE FLANGE**

**W14X30**

8.85 13.84 .27 6.73 .385 291. 19.6 .38 0 0

**W21X50**

14.7 20.83 .38 6.53 .535 984 24.9 1.14 7.92 0

**W14X109**

32. 14.32 0.525 14.605 .86 1240 447 7.12 7.52 0

and the file TFILE2 will contain:

**UNIT INCHES**

**ANGLES**

**L25255**

2.5 2.5 .3125 .489 0 0

**L40404**

4. 4. .25 .795 0 0

**Notes**

The User-Provided Steel Table(s) may be created and maintained as separate file(s). The same files may be used for all models using sections from these tables. These files should reside in the same directory where the input file is located.

## 5.20 Member Property Specification

### Purpose

This set of commands may be used for specification of section properties for frame members

### General format:

```

MEMBER PROPERTIES {
    AUSTRALIAN
    CANADIAN
    EUROPEAN
    FRENCH
    INDIAN
    AMERICAN
    BRITISH
    GERMAN
    JAPANESE
}

member-list {
    TABLE type-spec table-name
              (additional-spec)
    PRISMATIC property-spec
    TAPERED argument-list
    UPTABLE i1 section-name
    ASSIGN profile-spec
}
    
```

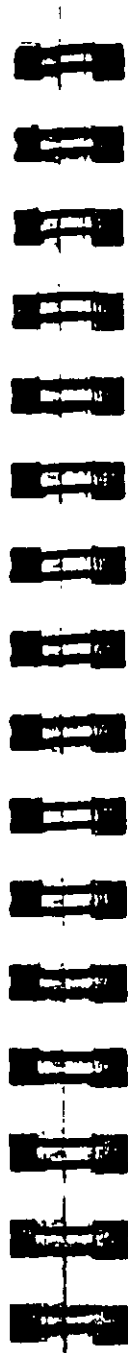
AMERICAN, BRITISH, EUROPEAN (etc ) option will instruct the program to pick up properties from the appropriate steel table. The default depends on the country of distribution.

### Description

This command initiates the specification of MEMBER PROPERTY. Following are the various options available:

- a) Specification from built-in steel tables. (Section 5.19.1)
- b) Specification of prismatic properties. (Section 5.19.2)
- c) Specification of tapered members. (Section 5.19.3)
- d) Specification from user provided table. (Section 5.19.4)
- e) Specification by ASSIGNing a profile. (Section 5.19.5)

Each specification is described in detail in the following sections. Examples are available in Section 5.20.6.



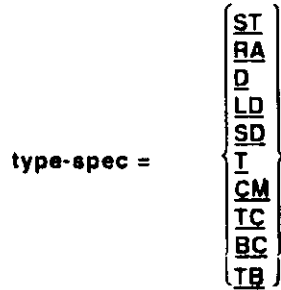
### 5.20.1 Specifying Properties from Steel Table

**Purpose**

The following commands are used for specifying section properties from built-in steel table(s).

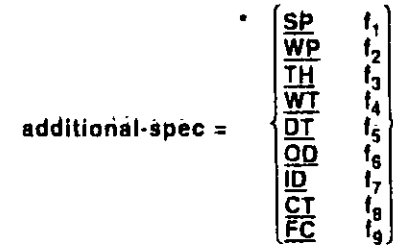
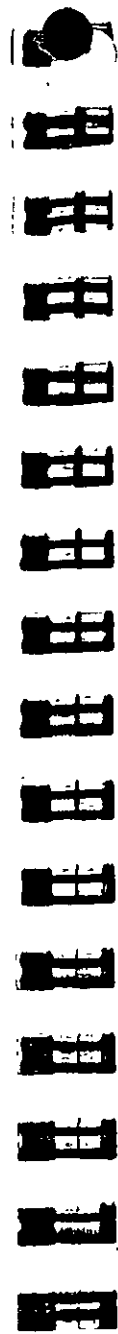
**General format:**

**type-spec . table-name additional-spec.**



- ST specifies single section from the standard built-in tables.
- RA specifies single angle with reverse Y-Z axes (see Section 1.5.2).
- D specifies double channel.
- LD specifies long leg, back to back, double angle
- SD specifies short leg, back to back, double angle.
- T specifies tee section cut from I shaped beams.
- CM specifies composite section, available with I shaped beams.
- TC specifies beams with top cover plate.
- BC specifies beams with bottom cover plate.
- TB specifies beams with top and bottom cover plates.

**table-name** = Table section name like W8X18, C15X33 etc. The documentation on steel design per individual country codes contains information regarding their steel section specification also. For details on specifying sections from the American steel tables, see Section 2.2.1 of this manual.



See Section 1.7.2

- SP f<sub>1</sub>= This set describes the spacing (f<sub>1</sub>) between angles or channels if double angles or double channels are used. f<sub>1</sub> defaults to 0.0 if not given.
- WP f<sub>2</sub>= Width (f<sub>2</sub>) of the cover plate if a cover plate is used with I shaped sections.
- TH f<sub>3</sub>= Thickness (f<sub>3</sub>) of plates or tubes.
- WT f<sub>4</sub>= Width (f<sub>4</sub>) of tubes, where TUBE is the table-name.
- DT f<sub>5</sub>= Depth (f<sub>5</sub>) of tubes.
- OD f<sub>6</sub>= Outside diameter (f<sub>6</sub>) of pipes, where PIPE is the table-name.
- ID f<sub>7</sub>= Inside diameter (f<sub>7</sub>) of pipes.
- CT f<sub>8</sub>= Concrete thickness (f<sub>8</sub>) for composite sections.
- FC f<sub>9</sub>= Compressive strength (f<sub>9</sub>) of the concrete for composite sections.

**Example**  
See Section 5.20.6

**Notes**

All values  $f_{1..9}$  must be supplied in current units.

Some important points to note in the case of the composite section are:

- 1) The width of the concrete slab is assumed to be the width of the top flange of the steel section + 16 times the thickness of the slab
- 2) In order to calculate the section properties of the cross-section, the modular ratio is calculated assuming that:  
 $E_s =$  Modulus of elasticity of steel = 29000 Ksi.  
 $E_c =$  Modulus of elasticity of concrete =  $1802.5\sqrt{FC}$  Ksi  
 where FC (in Ksi) is defined earlier



See Section 1.7.1

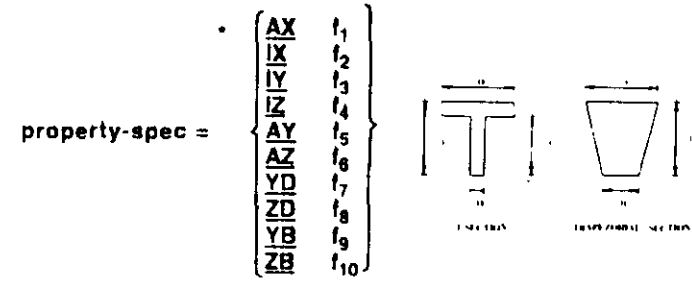
**5.20.2 Prismatic Property Specification**

**Purpose**

The following commands are used to specify section properties for prismatic cross-sections.

**General format:**

For the PRISMATIC specification, properties are provided directly as follows:



- AX  $f_1$  = Cross sectional area of the member. If omitted, the area is calculated from the YD and ZD dimensions.
- IX  $f_2$  = Torsional constant.
- IY  $f_3$  = Moment of inertia about local y-axis.
- IZ  $f_4$  = Moment of inertia about local z-axis (usually major).
- AY  $f_5$  = Effective shear area in local y-axis.
- AZ  $f_6$  = Effective shear area in local z-axis.
- YD  $f_7$  = Depth of the member in local y direction. (Diameter of section for circular members)
- ZD  $f_8$  = Depth of the member in local z direction. If ZD is not provided and YD is provided, the section will be assumed to be circular.
- YB  $f_9$  = Depth of stem for T-section.
- ZB  $f_{10}$  = Width of stem for T-section or bottom width for TRAPEZOIDAL section.

### 5.20.3 Tapered Member Specification

**Purpose**

The following commands are used to specify section properties for tapered I-shapes.

**General format:**

$$\text{argument-list} = f_1 f_2 f_3 f_4 f_5 (f_6 f_7)$$

where,

$f_1$  = Depth of section at start node

$f_2$  = Thickness of web.

$f_3$  = Depth of section at end node.

$f_4$  = Width of top flange.

$f_5$  = Thickness of top flange

$f_6$  = Width of bottom flange Defaults to  $f_4$  if left out.

$f_7$  = Thickness of bottom flange. Defaults to  $f_5$  left out

**Example**

**MEMBER PROPERTY**

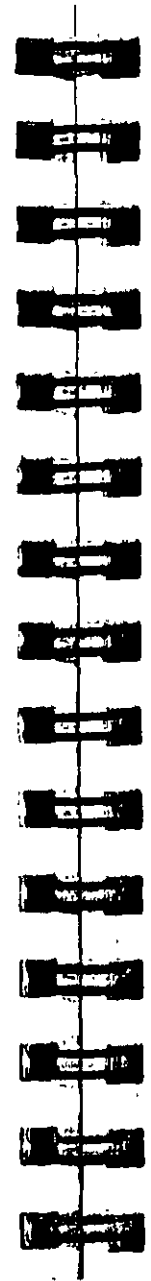
**1 TO 5 TAPERED 13.98 0.285 13.98 6.745 0.455 6.745 0.455**

**Notes**

1. All dimensions ( $f_1, f_2, \dots, f_7$ ) should be in current units.
2.  $f_1$  (Depth of section at start node) should always be greater than  $f_3$  (Depth of section at end node). The user should provide the member incidences accordingly.

Section

1 / 4



### 5.20.4 Property Specification from User Provided Table

**Purpose**

The following commands are used to specify section properties from a previously created USER-PROVIDED STEEL TABLE.

**General format:**

$$\text{member-list UPTABLE } i_1 \text{ section-name}$$

UPTABLE stands for user-provided table

$i_1$  = table number as specified previously (1 to 4)

section-name = section name as specified in the table. (Refer to Section 5.19)

**Example**

See Section 5.20.6

See Section  
1 / 3

### 5.20.5 Assign Profile Specification

**Purpose**

The ASSIGN command may be used to instruct the program to assign a suitable steel section to a frame member based on the profile-spec shown below.

**General format:**

profile-spec = 

BEAM
COLUMN
CHANNEL
ANGLE (DOUBLE)

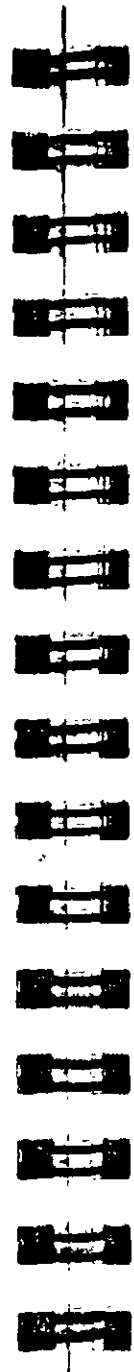
See Section 5

**Example**

See Section 5.20.6

**Notes**

Sections are always chosen from the relevant built-in steel table. To find out the details of the sections that are chosen, the command PRINT MEMBER PROPERTIES should be provided after specification of all member properties.



### 5.20.6 Examples of Member Property Specification

This section illustrates the various options available for MEMBER PROPERTY specification

**Example**

```

UNIT INCHES
MEMBER PROPERTIES
1 TO 5 TABLE ST W8X31
9 10 TABLE LD L40304 SP 0.25
12 TO 15 PRISMATIC AX 10.0 IZ 1520.0
17 18 TA ST PIPE OD 2.5 ID 1.75
20 TO 25 TA ST TUBE DT 12. WT 8. TH 0.5
27 29 32 TO 40 -
42 PR AX 5. IZ 400. IY 33. IX 0.2 YD 9. ZD 3.
43 TO 47 UPT 1 W10X49
50 51 UPT 2 L40404
52 TO 55 ASSIGN COLUMN
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CM W14X34 CT 5.0 FC 3.0
    
```

This example shows each type of member property input. Members 1 to 5 are wide flanges selected from the AISC tables; 9 and 10 are double angles selected from the AISC tables; 12 to 15 are prismatic members with no shear deformation; 17 and 18 are pipe sections; 20 to 25 are tube sections; 27, 29, 32 to 40, and 42 are prismatic members with shear deformation; 43 to 47 are wide flanges selected from the user input table number 1; 50 and 51 are single angles from the user input table number 2; 52 through 55 are designated as COLUMN members using the ASSIGN specification. The program will assign a suitable I-section from the steel table for each member.

Member 56 is a wide flange W12X26 with a 4.0 in. wide cover plate of thickness 0.3 inches at the top. Member 57 is a composite



section with a concrete slab thickness of 5.0 inches at the top of a wide flange W14X34. The compressive strength of the concrete in the slab is 3 0 ksi.



## 5.21 Element Property Specification

### Purpose

This set of commands may be used to specify properties of plate finite elements.

Unlike members and plate/shell elements, no properties are required for solid elements. However, constants such as modulus of elasticity and Poisson's ratio are to be specified.

### General Format:

#### ELEMENT PROPERTY

element-list THICKNESS  $t_1$  ( $t_2, t_3, t_4$ )

$t_1$  = Thickness of the element.

$t_2, \dots, t_4$  = Thicknesses at other nodes of the element, if different from  $t_1$ .

### Description

See  
Section 1.6

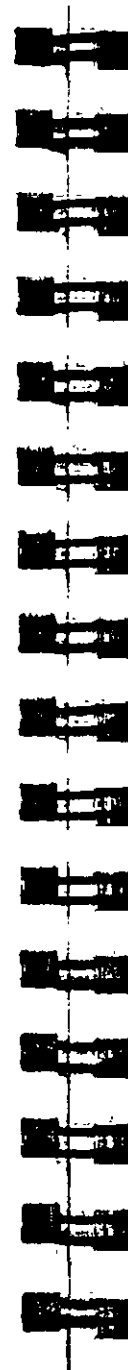
Elements of uniform thickness may be modeled using this command. Note that the value of the thickness must be provided in current units.

### Example

```
UNIT INCH
ELEMENT PROPERTY
1 TO 8 14 16 TH 0.25
```

## 5.22 Member/Element Releases

STAAD allows specification of releases of degrees of freedom for frame members and plate elements. Section 5.22.1 describes MEMBER release options and Section 5.22.2 describes ELEMENT release options.



### 5.22.1 Member Release Specification

#### Purpose

This set of commands may be used to fully release specific degrees of freedom at the ends of frame members. They may also be used to describe a mode of attachment where the member end is connected to the joint for specific degrees of freedom through the means of springs.

General format:

#### MEMBER RELEASES

member-list	<table border="0"> <tr> <td rowspan="2">}</td> <td>START</td> </tr> <tr> <td>END</td> </tr> </table>	}	START	END	<table border="0"> <tr> <td rowspan="6">}</td> <td>FX</td> </tr> <tr> <td>FY</td> </tr> <tr> <td>FZ</td> </tr> <tr> <td>MX</td> </tr> <tr> <td>MY</td> </tr> <tr> <td>MZ</td> </tr> </table>	}	FX	FY	FZ	MX	MY	MZ	<table border="0"> <tr> <td rowspan="6">}</td> <td>KFX 11</td> </tr> <tr> <td>KFY 12</td> </tr> <tr> <td>KFZ 13</td> </tr> <tr> <td>KMX 14</td> </tr> <tr> <td>KMY 15</td> </tr> <tr> <td>KMZ 16</td> </tr> </table>	}	KFX 11	KFY 12	KFZ 13	KMX 14	KMY 15	KMZ 16
			}	START																
END																				
}	FX																			
	FY																			
	FZ																			
	MX																			
	MY																			
	MZ																			
}	KFX 11																			
	KFY 12																			
	KFZ 13																			
	KMX 14																			
	KMY 15																			
	KMZ 16																			

where FX through MZ and KFX through KMZ represent force-x through moment-z degrees of freedom in the member local axes and 11 through 16 are spring constants for these degrees of freedom. If FX through MZ is used, it signifies a full release for that d.o.f. and if KFX through KMZ is used, it signifies a spring attachment

#### Example

#### MEMBER RELEASE

```
1 3 TO 9 11 12 START KFX 1000.0 MY MZ
1 10 11 13 TO 18 END MZ KMX 200.0
```

In the above example, for members 1, 3 to 9, 11 and 12, the moments about the local Y and Z axes are released at their start joints (as specified in MEMBER INCIDENCES). Further, these members are attached to their START joint along their local x axis through a spring whose stiffness is 1000.0 units of force/length.

For members 1, 10, 11 and 13 to 18, the moment about the local Z axis is released at their end joint. Also, the members are attached to their END joint about their local x axis through a moment-spring whose stiffness is 200.0 units of force-length/Degree. Note that members 1 and 11 are released at both start and end joints, though not necessarily in the same degrees of freedom

**Partial Moment Release**

Moments at the end of a member may be released partially. This facility may be used to model partial fixity of connections. The following format may be used to provide a partial moment release. Note that this facility is provided under the MEMBER RELEASE option and is in addition to the existing RELEASE capabilities.

**General Format:**

**MEMBER RELEASE**

member-list { START } MP  $f_1$   
                   { END }

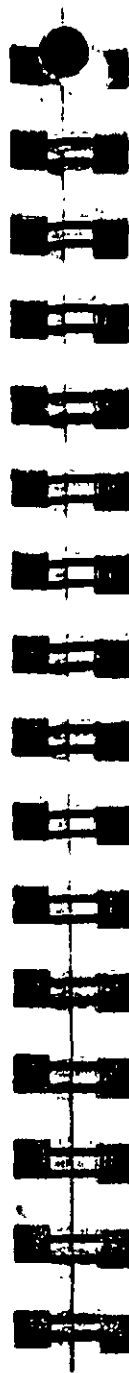
See Section where  $f_1$  = release factor.

The moment related stiffness co-efficient will be multiplied by a factor of  $(1-f_1)$  at the specified end

**Example**

**MEMBER RELEASE**  
**15 TO 25 START MP 0.25**

The above RELEASE command will apply a factor of 0.75 on the moment related stiffness co-efficients at START of members 15 to 25.



**Notes**

It is important to note that the factor  $f_1$  indicates a reduction in the stiffness corresponding to the rotational degrees of freedom MX, MY and MZ. In other words, the user should not expect the moment on the member to reduce by a factor of  $f_1$ . It may be necessary for the user to perform a few trials in order to arrive at the right value of  $f_1$  which results in the desired reduction in moment.

Also, note that START and END is based on the MEMBER INCIDENCE specification.

**At any end of the member, for any particular DOF, full, partial and spring release cannot be applied simultaneously. Only one out of the three is permitted.**

## F.22.2 Element Release Specification

### Purpose

This set of commands may be used to release specified degrees of freedoms at the end of plate finite elements.

### General Format:

#### ELEMENT RELEASE

$$\text{element-list} \begin{Bmatrix} J1 \\ J2 \\ J3 \\ J4 \end{Bmatrix} \begin{Bmatrix} FX \\ FY \\ FZ \\ MX \\ MY \\ MZ \end{Bmatrix}$$

See  
Section 1.8

where the keywords J1, J2, J3 and J4 signify joints in the order of the specification of the element incidence. For examples, if the incidences of the element were defined as 35 42 76 63, J1 represents 35, J2 represents 42, J3 represents 76, and J4 represents 63. Please note that element releases at multiple joints cannot be specified in a single line. Those must be specified separately as shown below.

FX through MZ represents forces/moments to be released per local axis system.

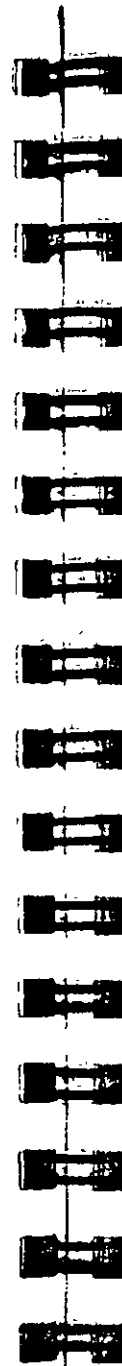
### Example

#### Correct Usage

```
ELEMENT RELEASE
10 TO 50 J1 MX MY
10 TO 50 J2 MX MY
10 TO 50 J3 MY
10 TO 50 J4 MY
```

#### Incorrect Usage

```
ELEMENT RELEASE
10 TO 50 J1 J2 MX MY
10 TO 50 J3 J4 MY
```

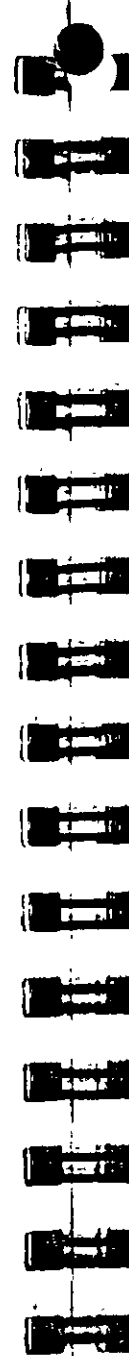


### Notes

All releases are in the local axis system.

## 5.23 Member Truss/Cable/Tension/Compression Specification

STAAD allows frame members to be designated as TRUSS members, CABLE members and TENSION/COMPRESSION-only members. Sections 5.23.1 through 5.23.3 describe these specifications.



### 5.23.1 Member Truss Specification

#### Purpose

This command may be used to model a specified set of members as TRUSS members.

#### Description

This specification may be used to specify TRUSS type members in a PLANE, SPACE or FLOOR structure. The TRUSS members are capable of carrying only axial forces. Typically, bracing members in a PLANE or SPACE frame will be of this nature

#### General format:

```
MEMBER TRUSS  
member-list
```

Note that this command is superfluous when a TRUSS type structure has already been specified.

See Sections  
19 and 110

#### Example

```
MEMB TRUSS  
1 TO 8 10 12 14 15
```

#### Notes

The TRUSS member has only one degree of freedom-the axial deformation. It is not equivalent to a frame member with moment releases at both ends.

### 5.23.2 Member Cable Specification

**Purpose**

This command may be used to model a specified set of members as CABLE members.

**Description**

The CABLE members, in addition to elastic axial deformation, are also capable of accommodating the effect of initial tension. Theoretical discussions of CABLE members are presented in Section 1 of this manual

**General format:**

**MEMBER CABLE**

**member-list TENSION  $f_1$**

where  $f_1$  = Initial Tension in cable member  
(in current units)

See Sections  
9, 1.10

**Example**

**MEMB CABLE  
20 TO 25 TENSION 15.5**

**Notes**

The TENSION specified in the CABLE member is applied on the structure as an external load as well as is used to modify the stiffness of the member. See Section 1.10 for details.



### 5.23.3 Member Tension/Compression Specification

**Purpose**

This command may be used to designate certain members as Tension-only or Compression-only members.

**General Format:**

**MEMBER TENSION  
member - list**

**MEMBER COMPRESSION  
member - list**

**Description**

Tension-only members are truss members that are capable of carrying tensile forces only. Thus, they are automatically inactivated for load cases that are capable of causing compression on them.

Compression-only members are truss members that are capable of carrying compressive forces only. Thus, they are automatically inactivated for load cases that are capable of causing tension on them.

See Section  
19

The procedure for analysis of Tension-only or Compression-only members requires iterations for every load case and therefore may be quite involved. The user may also consider using the INACTIVE specification if the solution time becomes unacceptably high.

It is very important to recognize that the input data must be provided in such a way that only one primary load case is provided for each PERFORM ANALYSIS command. Also, the SET NL and CHANGE commands must be used to convey to STAAD that multiple analyses and multiple structural conditions are involved.

**Example**

**MEMBER TENSION**  
**25 TO 30 35 36**

**Example**

**MEMBER COMPRESSION**  
**43 57 98 102 145**

**Example**

**MEMBER TENSION**  
**12 17 19 TO 37 65**  
**MEMBER COMPRESSION**  
**5 13 46 TO 53 87**

**Notes**

- 1) Loads that have been defined on members declared as MEMBER TENSION or MEMBER COMPRESSION will be active even when the member becomes INACTIVE during the process of analysis. This applies to SELFWEIGHT, MEMBER LOADS, PRESTRESS & POSTSTRESS LOADS, TEMPERATURE LOAD, etc.
- 2) A member declared as a TENSION only member or a COMPRESSION only member will carry axial forces only. It will not carry moments or shear forces. In other words, it is a truss member.
- 3) The MEMBER TENSION and MEMBER COMPRESSION commands should not be specified if the INACTIVE MEMBER command is specified.
- 4) The following is the general sequence of commands in the input file if the MEMBER TENSION or MEMBER COMPRESSION command is used. This example is for the



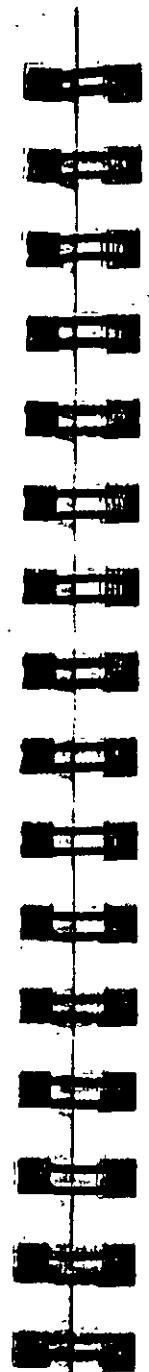
MEMBER TENSION command. Similar rules are applicable for the MEMBER COMPRESSION command. The dots indicate other input data items.

STAAD ...  
 SET NL ...  
 UNITS ...  
 JOINT COORDINATES  
 ...  
 MEMBER INCIDENCES  
 ...  
 ELEMENT INCIDENCES  
 ...  
 CONSTANTS  
 ...  
 MEMBER PROPERTY  
 ...  
 ELEMENT PROPERTY  
 ...  
 SUPPORTS  
 ...  
 MEMBER TENSION  
 ...  
 LOAD 1  
 ...  
 PERFORM ANALYSIS  
 CHANGE  
 MEMBER TENSION  
 ...  
 LOAD 2  
 ...  
 PERFORM ANALYSIS  
 CHANGE  
 MEMBER TENSION  
 ...  
 LOAD 3  
 ...

```

PERFORM ANALYSIS
CHANGE
MEMBER TENSION
...
LOAD 4
...
PERFORM ANALYSIS
CHANGE
MEMBER TENSION
...
LOAD 5
...
LOAD COMBINATION 6
...
LOAD COMBINATION 7
...
PERFORM ANALYSIS
CHANGE
LOAD LIST ALL
PRINT ...
PRINT ...
PARAMETER
...
CHECK CODE ...
SELECT MEMBER ...
FINISH
    
```

- a) See Section 5.5 for explanation of the SET NL command. The number that follows this command is the total number of primary load cases in the file.



- b) The principle used in the analysis is the following.
- The program reads the list of members declared as MEMBER TENSION.
  - The analysis is performed for the entire structure and the member forces are computed.
  - For the members declared as MEMBER TENSION, the program checks the axial force to determine whether it is tensile or compressive. If it is compressive, the member is "switched off" from the structure.
  - The analysis is performed again without the switched off members.
- c) In the example shown, only one LOAD case is specified per ANALYSIS. This is because, a member which is under tension for one load case may be in compression for another load case. The stiffness matrix for an analysis can account for the structural condition of only one of these two load cases
- d) Note that the MEMBER TENSION command and its accompanying list of members is provided after each (except the last) CHANGE command. This is because, each CHANGE command signifies that the previous MEMBER TENSION command is defunct thereby necessitating the specification of the MEMBER TENSION command again.
- e) The MEMBER TENSION command should not be used if the following load cases are present. Response Spectrum load case, Time History Load case, UBC Load case, Moving Load case.



## 5.24 Element Plane Stress and Ignore Inplane Rotation Specification

### Purpose

These commands allow the user to model the following conditions on plate elements

- a) PLANE STRESS condition
- b) IGNORing the in-plane rotation

### General Format:

```

ELEMENT { PLANE STRESS
         IGNORE ( INPLANE ROTATION ) }
element-list
    
```

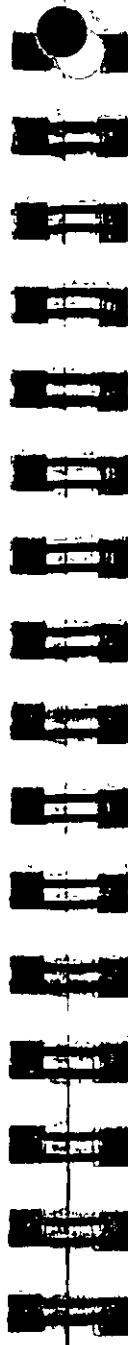
### Description

The PLANE STRESS specification allows the user to model selected elements for PLANE STRESS action only.

Similarly, the IGNORE INPLANE ROTATION command causes the program to ignore "in-plane rotation" actions. The STAAD plate element formulation includes this important action automatically.

However, it may be noted that some element formulations ignore this action by default. This user may utilize this option to compare STAAD results with solutions from these programs.

Section 16



### Example

```

ELEMENT PLANE STRESS
1 TO 10 15 20 25 35
ELEMENT IGNORE
30 50 TO 55
    
```

## 5.25 Member Offset Specification

### Purpose

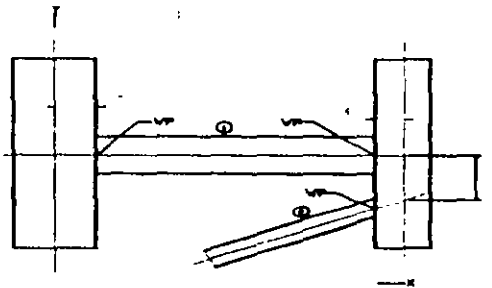
This command may be used to model the offset conditions existing at the ends of frame members.

### General format:

#### MEMBER OFFSETS

```
member-list { START } f1, f2, f3
              { END   }
```

### Description



$f_1$ ,  $f_2$ , and  $f_3$  correspond to the distance, measured in the global coordinate system, from the joint (START or END as specified) to the centroid of the starting or ending point of the members listed. MEMBER OFFSET command can be used for

any member whose starting or ending point is not concurrent with the given incident joint. This command enables the user to account for the secondary forces which are induced due to the eccentricity of the member. Member offsets can be specified in any direction, including the direction which may coincide with the local  $\bar{x}$ -axis of the member.

$wp$  in the diagram refers to the location of the centroid of the starting or ending point of the member.

S  
/ Section



### Example

```
MEMBER OFFSET
1 START 7.0
1 END -6.0 0.0
2 END -6.0 -9.0
```

### Notes

- 1) If a MEMBER LOAD (see MEMBER LOAD specification) is applied on a member for which MEMBER OFFSETS have been specified, the location of the load is measured not from the coordinates of the starting joint. Instead, it is measured from the offset location of the starting joint.
- 2) START and END is based on the user's specification of MEMBER INCIDENCE for the particular member.

## 5.26 Constant Specification

### Purpose

This command may be used to specify the material properties (Modulus of Elasticity, Poisson's ratio, Density and Co-efficient of linear expansion) of the members and elements. In addition, this command may also be used to specify the member orientation (BETA angle or REFERENCE point).

### General format:

#### CONSTANTS

$$\left. \begin{array}{l} E \\ \text{POISSON} \\ \text{DENSITY} \\ \text{BETA} \\ \text{ALPHA} \end{array} \right\} f_1 \quad \left\{ \begin{array}{l} \text{MEMBER memb/elem-list} \\ (\text{ALL}) \end{array} \right\}$$

REF  $f_2, f_3, f_4$       MEMBER memb/elem-list

- E specifies Young's Modulus. This value must be provided as the first item in the Constants list.
- POISSON specifies Poisson's Ratio. This value is used for calculating the Shear Modulus ( $G=0.5 \times E / (1 + \text{POISSON})$ ).
- DENSITY specifies weight density.
- ALPHA Co-efficient of thermal expansion.
- BETA specifies member rotation angle in degrees (see Section 2).

Note : Single angle sections are oriented according to their principal axes by default. If it is necessary to orient them such that their legs are parallel to the global axes, the BETA specification must be

See Section 5.3



used. STAAD offers the following additional specifications for this purpose :

BETA ANGLE  
BETA RANGLE

Both of the above options will result in an orientation with the legs parallel to the global axis. The 'ANGLE' option rotates the section through the angle "theta" (where "theta" = angle between the principal axis system and the global axis system). The 'RANGLE' option rotates the section through an angle equal to (180 - "theta"). Both options will work the same way for equal angles. For unequal angles, the right option must be used based on the required orientation.

$f_1$  Value of the corresponding constants. For E, POISSON and DENSITY, material names can be provided instead of  $f_1$ . Appropriate values will be automatically assigned. Current list of material names includes STEEL, CONCRETE & ALUMINUM.

$f_2, f_3, f_4$  Global X, Y, and Z coordinates for the reference point, from which the BETA angle can be calculated by the program.

### Example

```
CONSTANTS
E 29000.0 ALL
BETA 45.0 MEMB 5 7 TO 18
DENSITY STEEL MEMB 14 TO 29
BETA 90 MEMB X
```

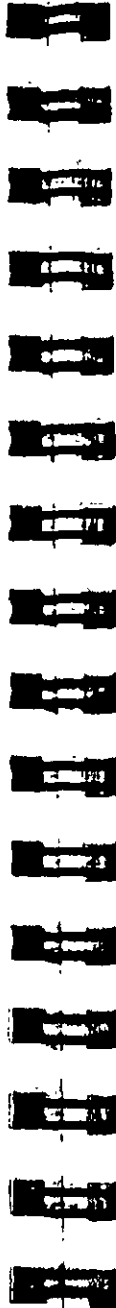
Note that the last command in the above example will set BETA as 90° for all members parallel to the X-axis.

**Notes**

- 1) The value for E must always be given first in the Constants list.
- 2) All numerical values must be provided in the current units.
- 3) It is not necessary nor possible to specify the units of temperature or ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for temperature (see Section 5.32.6).
- 4) If the POISSON RATIO is not specified, the program will use a default value of 0.0

**5.27 Support Specifications**

STAAD support specifications may be either parallel or inclined to the global axes. Specification of supports parallel to the global axes is described in Section 5.27.1. Specification of inclined supports is described in Section 5.27.2.



## 5.27.1 Global Support Specification

### Purpose

This set of commands may be used to specify the SUPPORT conditions for supports parallel to the global axes.

### General format:

#### SUPPORTS

joint-list { PINNED  
FIXED (BUT release-spec{spring-spec.}) }

release-spec =  $\begin{Bmatrix} FX \\ FY \\ FZ \\ MX \\ MY \\ MZ \end{Bmatrix}$

spring-spec =  $\begin{Bmatrix} KFX & f_1 \\ KFY & f_2 \\ KFZ & f_3 \\ KMX & f_4 \\ KMY & f_5 \\ KMZ & f_6 \end{Bmatrix}$

### Description

PINNED support is a support which has translational, but no rotational restraints. In other words, the support has no moment carrying capacity. A FIXED support has both translational and rotational restraints. A FIXED support can be released in the global directions as described in release-spec (FX for force-X through MZ for moment-Z). Also, a fixed support can have spring constants as described in spring-spec (translational spring in global X-axis as KFX through rotational spring in global Z-axis as KMZ). Corresponding spring constants are  $f_1$  through  $f_6$ . Note that the

rotational spring constants are always per degree of rotation. No more than five releases may be provided. If both release specifications and spring specifications are to be supplied for the same support joint, release specifications must come first.

### Example

#### SUPPORTS

1 TO 4 7 PINNED

5 6 FIXED BUT FX MZ

8 9 FIXED BUT MZ KFX 50.0 KFY 75.

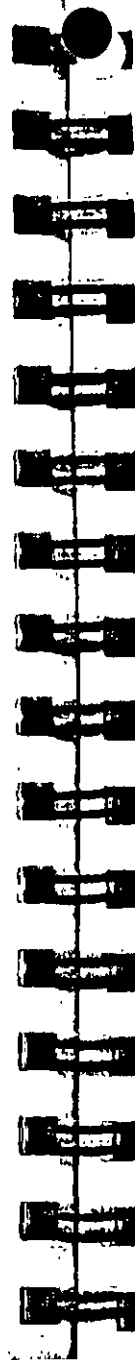
18 21 FIXED

27 FIXED BUT KFY 125.0

In this example, joints 1 to 4 and joint 7 are pinned. No moments are carried by those supports. Joints 5 and 6 are fixed for all DOF except in force-X and moment-Z. Joints 8 and 9 are fixed for all DOF except moment-Z and have springs in the global X and Y directions with corresponding spring constants of 50 and 75 units respectively. Joints 18 and 21 are fixed for all translational and rotational degrees of freedom. At joint 27, all the DOF are fixed except the FY DOF where it has a spring with 125 units spring constant

### Notes

- 1) Users are urged to refer to Section 5.38 for information on specification of SUPPORTS along with the CHANGE command specifications.
- 2) Spring constants must be provided in the current units.



### 5.27.2 Inclined Support Specification

**Purpose**

These commands may be used to specify supports that are inclined with respect to the global axes.

**General Format:**

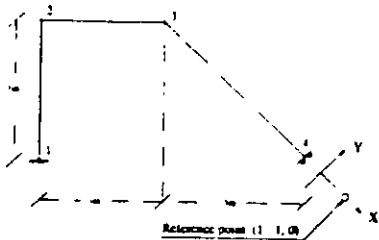
**SUPPORT**

joint-list **INCLINED**  $f_1$   $f_2$   $f_3$  { **PINNED**  
**FIXED (BUT release-spec (spring-spec.))** }

where  $f_1$ ,  $f_2$ ,  $f_3$  are coordinates of the "reference point" necessary for the "Inclined Support Axis System" (see below).

Section 1.13

Note the release-spec and spring-spec are the same as in the previous section (5.27.1). However, please note that FX through MZ and KFX through KMZ refer to forces/moments and spring constants in the "Inclined Support Axis System" (see below).



An inclined support should not be defined at a joint to which a finite element is connected.

### Inclined Support Axis System

The INCLINED SUPPORT specification is based on the "Inclined Support axis system". The local x-axis of this system is defined by assuming the inclined support joint as the origin and joining it with a "reference point" with co-ordinates of  $f_1$ ,  $f_2$  and  $f_3$  (see figure) measured from the inclined support joint in the global coordinate system.

The Y and Z axes of the inclined support axis system have the same orientation as the local Y and Z axes of an imaginary member whose BETA ANGLE is zero and whose incidences are defined from the inclined support joint to the reference point. Users may refer to section 1.5.3 of this manual for more information on these concepts.

**Example**

**SUPPORT**  
**4 INCLINED 1.0 -1.0 0.0 FIXED BUT FY MX MY MZ**



### 5.27.3 Automatic Spring Support Generator for Foundations

STAAD has a facility for automatic generation of spring supports to model footings and foundation mats. This command is specified under the SUPPORT command.

**General Format:**

**SUPPORT**

Joint-list { FOOTING f1 (f2) } DIRECTION  $\begin{pmatrix} X \\ Y \\ Z \end{pmatrix}$  SUBGRADE f3

where

- f1, f2 = Length and width of the footing. If f2 is not given, the footing is assumed to be a square with sides f1
- f3 = Soil sub-grade modulus in force/area/length units
- X, Y, Z = Global direction in which soil springs are to be generated

**The FOOTING option :** If you want to specify the influence area of a joint yourself and have STAAD simply multiply the area you specified by the sub-grade modulus, use the FOOTING option. Situations where this may be appropriate are such as when a spread footing is located beneath a joint where you want to specify a spring support. Please note that it is absolutely imperative that you provide f1 (and f2 if its a non-square footing) if you choose the FOOTING option.

**The ELASTIC MAT option :** If you want to have STAAD calculate the influence area for the joint (instead of you specifying an area yourself) and use that area along with the sub-grade modulus to determine the spring stiffness value, use the MAT option. Situations where this may be appropriate are such as when a slab is on soil and carries the weight of the structure above. You



may have modeled the entire slab as finite elements and wish to generate spring supports at the nodes of the elements. Note that the word ELASTIC is optional.

**The DIRECTION option :** The keyword DIRECTION is followed by one of the alphabets X, Y or Z which indicate the direction of resistance of the spring supports.

**The SUBGRADE option :** The keyword SUBGRADE is followed by the value of the subgrade reaction. Please note the value should be provided in the current unit system signified by the most recent UNIT statement prior to the SUPPORT command.

**Example**

**SUPPORTS  
1 TO 126 ELASTIC MAT DIREC Y SUBG 200.**

The above command instructs STAAD to internally generate supports for all nodes 1 through 126 with elastic springs. STAAD first calculates the influence area perpendicular to the global Y axis of each node and then multiplies the corresponding influence area by the soil subgrade modulus of 200 0 to calculate the spring constant to be applied to the node.

## 7.4 Multi-linear Spring Support Specification

When soil is modeled as spring supports, the varying resistance it offers to external loads can be modeled using this facility, such as when its behavior in tension differs from its behavior in compression.

General format:

### MULTILINEAR SPRINGS

joint-list SPRINGS  $d_1$   $s_1$   $d_2$   $s_2$  .....  $d_n$   $s_n$

Where (di si) pairs represent displacement and spring constant pairs.

Example

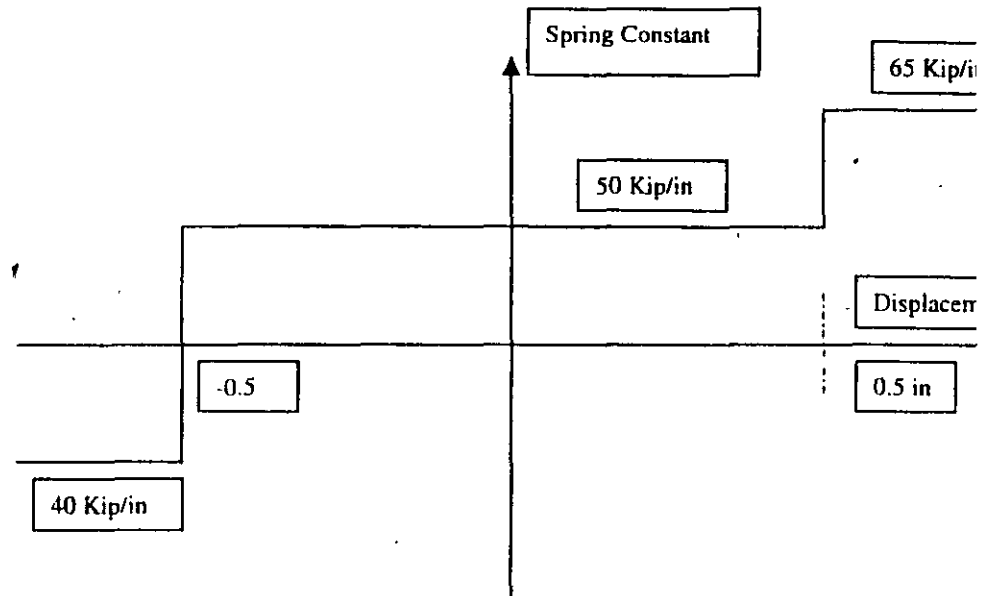
UNIT KIP INCH  
SUPPORT

1 PINNED ; 2 4 FIXED BUT KFY 40.0

MULTILINEAR SPRINGS

2 4 SPRINGS -0.5 40.0 0.0 50.0 0.5 65.0

Load-Displacement characteristics of soil can be represented by a multi-linear curve. Slope of this curve will represent the spring characteristic of the soil at different displacement values. A typical spring characteristic of soil may be represented as the step curve as show in below. In the above example the multi-linear spring command specifies soil spring at joints 2 and 4.



The multi-linear spring command will trigger a multiple analysis and convergence check cycle. The cycle will continue till the support displacements computed in the previous analysis cycle are close enough with the support displacements computed in the current analysis cycle.



## .28 Master/Slave Specification

### Purpose

This set of commands may be used to model rigid links through the specification of MASTER and SLAVE joints.

### General format:

```

SLAVE {
  RIGID
  FX
  FY
  FZ
  MX
  MY
  MZ
} MASTER | JOINT joint-spec
    
```

```

joint-spec = {
  joint-list
  X RANGE
  Y RANGE
  Z RANGE
} t1, t2
    
```

### Description

The master/slave option provided in STAAD allows the user to model rigid links in the system. The support specifications must be provided before this command is used. Notice that instead of providing a joint list for the slaved joints, a range of coordinate values (in global system) may be used. All joint coordinates within a specified range are assumed to be slaved joints. No finite elements may be connected to the slaved joints. The joint list or coordinate range specified for slaved joints may include the master joint. Fx, Fy etc. are the directions in which they are slaved to the master. If all directions are provided, the joints are rigidly connected. The following examples illustrate the use of this option.

See Section 1

### Example

```

SLAVE FX MZ MASTER 9 JOINT 1 TO 15 -
17 19 20
SLAVE FX FY MASTER 37 JOINT YR 19.9 20.1
    
```

The SLAVE RIGID specification may be used to model a rigid diaphragm directly. All degrees of freedom will be taken into consideration for slaving and the rigid body rotation will be automatically considered.

### Example

```

SLAVE RIGID MASTER 22 JOINTS 10 TO 45
SLAVE RIGID MASTER 70 JOINT YR 25.5 27.5
    
```

### Notes

Proper stiffness connection is necessary between the master and the slaved joints for this type of modeling.



### 3 Draw Specifications

#### Purpose

This set of commands may be used to generate printer plots of structure geometry and results as part of the output

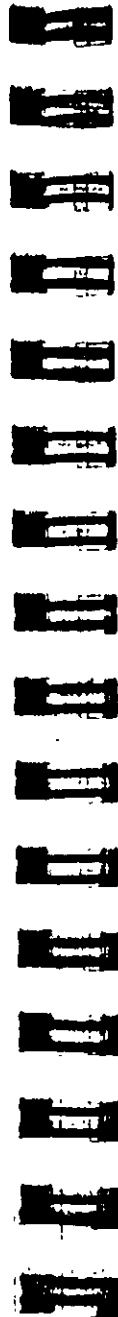
#### Description

Besides interactive graphics, STAAD has features to provide commands to plot structural geometry, analysis results etc. as part of the STAAD output file.

Note that these output files ( ANL files) should be printed only through the *Print Output* option of the main menu of STAAD. Plots can also be displayed by the *View Output* option of the main menu of STAAD.

Plots are of high-resolution and most of the 8/9/24 pin dot matrix and laser printers are supported.

The DRAW command is used to create the plots in the output. The following is the format of the DRAW command.



<u>DRAW</u>	ISOMETRIC	} (LIST list-spec)
	ROTATE rotate-spec	
	SECTION section-spec	
	ZOOM f <sub>1</sub>	
	SHIFT x y	
	JOINT	
	MEMBER	
	SUPPORT	
	PROPERTY	
	SHAPE	
	HIDDEN - LINE - REMOVED	
	SHRINK f <sub>2</sub>	
	LOAD ln	
	DFDRAW ln	
	MODRAW sn	
	SCDRAW ln	
	MSDRAW ln force-spec	
	BMDRAW ln force-spec	
ENVELOP force-spec		
SCALE f <sub>3</sub>		
VALUE		
STRESS CONTOUR ln		

$$\text{rotate-spec} = \begin{Bmatrix} X & x_a \\ Y & y_a \\ Z & z_a \end{Bmatrix} \quad \text{section-spec} = \begin{Bmatrix} XY \\ YZ \\ XZ \end{Bmatrix} \quad r_1 \quad r_2$$

$$\text{force-spec} = \begin{Bmatrix} FX \\ FY \\ MZ \\ FZ \\ MY \end{Bmatrix}$$

- f<sub>1</sub> = Zoom factor by which structure is to be reduced or enlarged. A value less than 1.0 is for reduction and greater than 1.0 is to enlarge.
- f<sub>2</sub> = Shrink factor by which member/elements be shrunk. Value varies from 0.1 to 0.9 ;
- f<sub>3</sub> = Scale factor by which deflected shapes to be multiplied. Normally all scales are automatically computed. However, the user may change this by this command.
- x,y = x and y shift values based on structure coordinates.
- ln = Load number to be considered

- ... = Mode-shapes number to be displayed.  
 xa,ya,za = X, Y and Z angles by which the structure is to be rotated.  
 r<sub>1</sub>,r<sub>2</sub> = Maximum and minimum values defining the range in the direction perpendicular to the specified section plane.

**Notes**

- 1) Following commands may be used anywhere in the input.

- ISOMETRIC = Draw isometric view.  
 ROTATE = Rotate as specified in rotate-spec.  
 SECTION = Draw section as specified in section-spec.  
 ZOOM = Draw with a zoom factor of value f<sub>1</sub>  
 SHIFT = Shift the structure to the specified x, y values.  
 JOINT = Display joint numbers.  
 MEMBER = Display member numbers.  
 SUPPORT = Display support icons.  
 PROPERTY = Display property names.  
 SHAPE = Display the shape of the member property with proper BETA angle orientation.  
 HIDE = Remove hidden lines when elements are present.  
 SHRINK = Shrink all member/elements by the factor of f<sub>2</sub>  
 LOAD = Display load icons. Obviously, this command can be used only after the loadings are provided.

- 2) Following commands are related to results and should be used only after the PERFORM ANALYSIS command.

- DFDRAW = Draw deflected shape.  
 MODRAW = Draw mode shape.  
 SCDRAW = Draw section displacement.  
 MSDRAW = Display force/moment diagram on the entire structure for specified In (load number).  
 BMDRAW = Display force/moment diagram for independent members as listed in LIST. No more than 2 member lists are allowed. Use multiple DRAW commands to display force/moment diagrams for independent members.



- ENVELOP = Same as MSDRAW except worst of all active load cases.  
 SCALE = Scale factor by which deflected shapes to be multiplied.  
 VALUE = Display values of Force/Moments, displacements.  
 STRESS-CONTOUR = Draw stress-contour for finite elements.  
 Only the contour for the Absolute maximum principal stress can be plotted.

**Example**

```

DRAW ISOMETRIC MEMBER SUPPORT PROPERTY
DRAW SHAPE SUPPORT
DRAW SECTION XY 14.9 15.1
DRAW ROTATE X -20 Y 30 Z 20 HIDE
DRAW ISOMET MSDRAW 2 MZ VALUE

```

### 5.30 Cut-Off Frequency or Mode Shapes

**Purpose**

These commands are used in conjunction with dynamic analysis. They may be used to specify the highest frequency or the number of mode shapes that need to be considered.

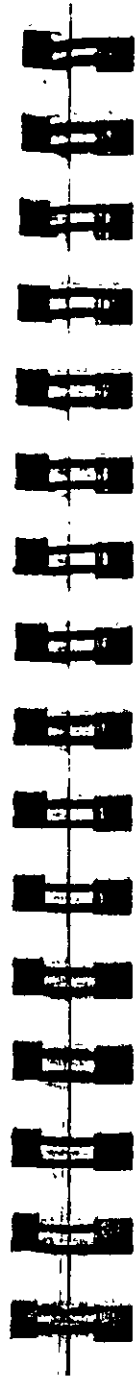
**General Format:**

```
CUT (OFF) {
    FREQUENCY f1
    MODE SHAPE i1
}
```

**Where,**

- f<sub>1</sub> = Highest frequency (cycle/sec) to be considered for dynamic analysis.
  - i<sub>1</sub> = Number of mode shapes to be considered for dynamic analysis.
- If the cut off frequency command is not provided the cut off frequency will default to 108 cps. If the cut off mode shape command is not provided, the first three modes will be calculated. These commands should be provided prior to the loading specifications

See Section 1.18.3



### 5.31 Definition of Load Systems

**Purpose**

This section describes the specifications necessary for defining various load systems for automatic generation of Moving loads, UBC Seismic loads and Wind loads. In addition, this section also describes the specification of Time History load for Time History analysis.

**Description:**

STAAD has built-in algorithms to generate moving loads, lateral seismic loads (per the Uniform Building Code), and wind loads on a structure. Use of the load generation facility consists of two parts:

- 1) Definition of the load system(s).
- 2) Generation of primary load cases using previously defined load system(s).

See Section 1.17

Definition of the load system(s) must be provided before any primary load case is specified. This section describes the specification of load system(s). Information on how to generate primary load cases using the defined load system(s) is available in Section 5.32.12.

### 5.31.1 Definition of Moving Load System

**Purpose**

This set of commands may be used to define the moving load system.

**General format:**

**DEFINE MOVING LOAD(FILE file-name)**

**TYPE j** { **LOAD**  $f_1, f_2, \dots, f_n$  **DISTANCE**  $d_1, d_2, \dots, d_{n-1}$  (**WIDTH**  $w$ ) }  
 { **load-name (f)** }

Note that the MOVING LOAD system may be defined in two possible ways - directly within the input file or using an external file.

See Section 5.12.12

The FILE option should be used only in the second case when the data is to be read from an external file. The filename should be limited to 16 characters.

Moving Loads can be generated for frame members only. They will not be generated for finite elements.

**Define Moving Load within input file**

Use the first TYPE specification.

**TYPE j LOAD**  $f_1, f_2, \dots, f_n$  **DISTANCE**  $d_1, d_2, d_{(n-1)}$  (**WIDTH**  $w$ )

Where,

- $j$  = moving load system type number. (integer)
- $f_i$  = value of conc.  $i^{th}$  load
- $d_i$  = distance between the  $(i+1)^{th}$  load and the  $i^{th}$  load in the direction of movement



$w$  = spacing between loads perpendicular to the direction of movement. If left out, one dimensional loading is assumed.

**Define Moving Load using an external file**

Use the second TYPE specification.

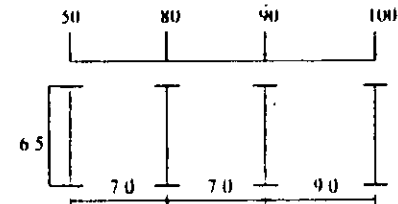
**TYPE j load-name (f)**

Where,

- load-name** Is the name of the moving load system
- and **f =** Optional multiplying factor to scale up or down the value of the loads. (default = 1.0)

Following is a typical file containing the data.

```
CS200      ----- name of load system (load-name)
50. 80. 90. 100. ----- loads
7. 7. 9.   ----- distance between loads
6.5       ----- width
```



Note that several load systems may be repeated within the same file.

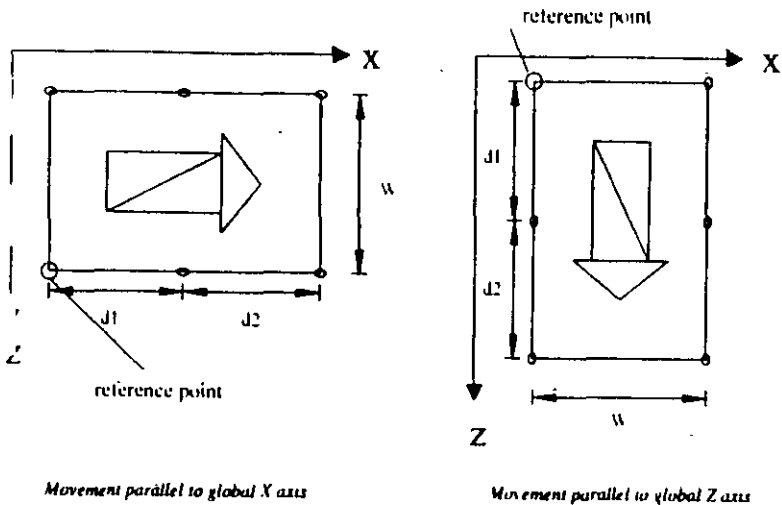
All loads and distances are in current unit system.

The STAAD moving load generator assumes:

- 1) All loads are acting in the negative global vertical (Y or Z) direction. The user is advised to set up the structure model accordingly.
- 2) Resultant direction of movement is determined from the X, Y and Z increments of movements as provided by the user.

**Reference Load**

The first specified concentrated load in the moving load system is designated as the reference load. While generating subsequent primary load cases, the initial position of the load system and the direction of movement are defined with respect to the reference load location. Also note that, when selecting the reference load location, the value of the width must be positive in the applicable global X or Z direction. The following figures illustrate the concept of reference load.



**Specifying standard AASHTO loadings**

General format:

TYPE i  $\left\{ \begin{matrix} \text{HS20} \\ \text{HS15} \\ \text{H20} \\ \text{H15} \end{matrix} \right\} (f) (vs)$

where,

- i = moving load system type no. (integer).
- f = optional multiplying factor (default = 1.0)

vs = variable spacing as defined by AASHTO, for HS series trucks (default = 14 ft.)

**Example**

```
UNIT KIP FEET
DEFINE MOVING LOAD
TYPE 1 LOAD 10.0 20.0 15.0 DISTANCE 5.0 7.5 WIDTH 8.0
TYPE 2 LOAD 20.0 20.0 DISTANCE 10.0 WIDTH 7.5
TYPE 3 HS20 0.80 22.0
```

**Notes**

All loads and distances must be provided in the current unit system.

**Example**

When data is provided through an external file called MOVLOAD

**Data in input file**

```
UNIT KIP FEET
DEFINE MOVING LOAD FILE MOVLOAD
TYPE 1 AXLTYP1
TYPE 2 AXLTYP2 1.25
```

**Data in external file MOVLOAD**

```
AXLTYP1
10 20 15
5.0 7.5
6.0
AXLTYP2
20 20
10
7.5
```



### 5.31.2 Definition of UBC Load

**Purpose**

This set of commands may be used to define the parameters for generation of UBC-type equivalent static lateral loads for seismic analysis.

**Description**

The STAAD seismic load generator assumes that the lateral loads will be exerted in X and Z directions and Y will be the direction of the gravity loads. Thus, for a building model, Y axis will be perpendicular to the floors and point upward (all Y coordinates positive). The user is advised to set up the model accordingly

Total lateral seismic force or base shear is automatically calculated by STAAD using the appropriate UBC equation(s).

See Sections  
17.2 and  
12.12

$$V = \frac{ZIC}{R_w} W \quad (\text{per UBC 1994}) \quad (1)$$

$$V = ZIKCSW \quad (\text{per UBC 1985}) \quad (2)$$

**Note:**

- 1) All symbols and notations are per UBC
- 2) Base shear V may be calculated by STAAD using either the 1994 procedure (equation 1) or the 1985 procedure (equation 2). The user should use the appropriate "ubc-spec" (see General Format below) to instruct the program accordingly.

STAAD/ISDS utilizes the following procedure to generate the lateral seismic loads.

1. User provides seismic zone co-efficient and desired "ubc-spec" (1985 of 1994) through the DEFINE UBC LOAD command.
2. Program calculates the structure period T.



3. Program calculates C from appropriate UBC equation(s) utilizing T.
4. Program calculates V from appropriate equation(s). W is obtained from SELFWEIGHT, JOINT WEIGHT(s) and MEMBER WEIGHT(S) provided by the user through the DEFINE UBC LOAD command.
5. The total lateral seismic load (base shear) is then distributed by the program among different levels of the structure per UBC procedures.

General format:

**DEFINE UBC (ACCIDENTAL) LOAD**

ZONE f<sub>1</sub> ubc-spec

SELFWEIGHT  
JOINT WEIGHT  
joint-list WEIGHT w  
MEMBER WEIGHT

mem-list  $\left\{ \begin{array}{l} \underline{UNI} \quad v_1 \quad v_2 \quad v_3 \\ \underline{CON} \quad v_4 \quad v_5 \end{array} \right\}$

ubc-spec =  $\left\{ \begin{array}{l} \underline{f} \\ \underline{RWX} \\ \underline{RWZ} \\ \underline{S} \\ \underline{(CT)} \\ \underline{(PX)} \\ \underline{(PZ)} \end{array} \right\} \begin{array}{l} f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \\ f_7 \\ f_8 \\ f_9 \\ f_{10} \\ f_{11} \end{array}$

ubc-spec =  $\left\{ \begin{array}{l} \underline{K} \\ \underline{(TS)} \end{array} \right\} \begin{array}{l} f_6 \\ f_7 \\ f_8 \end{array}$

for UBC 1994      for UBC 1985

where.

- f<sub>1</sub> = seismic zone coefficient (0.2, 0.3 etc.)
- f<sub>2</sub> = importance factor
- f<sub>3</sub> = numerical co-efficient R<sub>w</sub> for lateral load in X-direction
- f<sub>4</sub> = numerical co-efficient R<sub>w</sub> for lateral load in Z-directions
- f<sub>5</sub> = site co-efficient for soil characteristics
- f<sub>6</sub> = horizontal force factor
- f<sub>7</sub> = importance factor

- $f_8$  = site characteristic period (Referred to as  $T_s$  in the UBC code). Default value is 0.5.
- $f_9$  = optional CT value to calculate alternative period per method A of UBC.
- $f_{10}$  = Period of structure (in seconds) in the X- direction
- $f_{11}$  = Period of structure (in seconds) in the Z- direction.
- w = joint weight associated with list
  
- UNI** = specifies a uniformly distributed load with a value of  $v_1$  starting at a distance of  $v_2$  (from the start of the member) and ending at a distance of  $v_3$  (from the start of the member). If  $v_2$  and  $v_3$  are omitted, the load is assumed to cover the entire length of the member
  
- CON** = specifies a concentrated force with a value of  $v_4$  applied at a distance of  $v_5$  (from the start of the member). If  $v_5$  is omitted, the load is assumed to act at the center of the member.

**Notes**

- 1) If the option ACCIDENTAL is used, the accidental torsion will be calculated per UBC specifications. The value of the accidental torsion is based on the "center of mass" for each level. The "center of mass" is calculated from the SELFWEIGHT, JOINT WEIGHTs and MEMBER WEIGHTs specified by the user
- 2) In "ubc-spec" for 1985 code, specification of TS is optional. If TS is specified, resonance co-efficient S is determined from the building period T and user provided TS using UBC equations. If TS is not specified, the default value of 0.5 is assumed.
- 3) By providing either PX or PZ or both, you may override the period calculated by STAAD and the user defined value will then be used for the base shear calculation. If you do not define PX or PZ, the period will be calculated by the program.
- 4) Some of the items in the output for the UBC analysis are explained below.



**CALC / USED PERIOD**

The CALC PERIOD is the period calculated using the Rayleigh method (Method B as per UBC code). For UBC in the x-direction, the USED PERIOD is PX. For the UBC in the z-direction, the USED PERIOD is PZ. If PX and PZ are not provided, then the used period is the same as the calculated period for that direction. The used period is the one substituted into the critical equation of the UBC code to calculate the value of C.

**C, C-ALT**

According to the UBC code, C in Eq. 34-2 has to be calculated as per a period calculated by method B and a period calculated by Method A. The former is represented in the output as C. The latter is multiplied by 0.8 (80%) and the resulting value is represented as C-ALT.

- 5) In the analysis for UBC loads, all the supports of the structure have to be at the same level and have to be at the lowest elevation level of the structure.

**Example**

See Section 5.31.12 under Generation of UBC Seismic Load.

**5.31.3 Colombian Seismic Load**

**Purpose**

The purpose of this command is to define and generate static equivalent seismic loads as per Colombian specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in X or Z direction(s).



**Description**

The STAAD seismic load generator assumes that the lateral loads will be exerted in X and Z directions. Y will be the direction of gravity loads. Thus, for a building model, the Y axis will be perpendicular to the floors and point upward (all Y coordinates positive). The user is advised to set up the model accordingly

**Methodology :**

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE COLOMBIAN LOAD command.

Program calculates the natural period of building T utilizing clause 1628.2.2 of UBC 1994.

Design spectral coefficient ( Sa ) is calculated utilizing T as,

$$Sa = \begin{cases} Aa I (1.0 + 5.0 T) & \text{when, } 0 \leq T \leq 0.3 \text{ sec} \\ 2.5 Aa I & \text{when, } 0.3 < T \leq 0.48 S \text{ in sec} \\ 1.2 Aa S / T & \text{when, } 0.48 S < T \leq 2.4 S \text{ in sec} \\ Aa I / 2 & \text{when, } 2.4 S < T \end{cases}$$

where,

$$\begin{aligned} Aa &= \text{Seismic Risk factor (user input)} \\ S &= \text{Soil Site Coefficient (user input)} \\ I &= \text{Coefficient of Importance (user input)} \end{aligned}$$

Base Shear, Vs is calculated as

$$Vs = W * Sa$$

Where,

$$W = \text{Total weight on the structure}$$

Total lateral seismic load, Vs is distributed by the program among different levels as,

$$Fx = Cvx * Vs$$

Where,

$$Cvx = (Wx * hxK) / \sum_{ni=1} (Wx * hxK)$$



Where,

$$\begin{aligned} Wx &= \text{Weight at the particular level} \\ hx &= \text{Height of that particular level} \\ K &= \begin{cases} 1.0 & \text{when, } T \leq 0.5 \text{ sec} \\ 0.75 + 0.5 * T & \text{when, } 0.5 < T \leq 2.5 \text{ sec} \\ 2.0 & \text{when, } 2.5 < T \end{cases} \end{aligned}$$

**General Format**

```
DEFINE COLOMBIAN LOAD
ZONE I1 abc-spec
SELFWEIGHT
JOINT WEIGHT
Joint-list WEIGHT w
MEMBER WEIGHT
Mem-list (UNI ....)
```

$$abc-spec = (I I2, S I3)$$

Where, I1, I2 and I3 are Seismic Risk factor, Soil Site Coefficient and Coefficient of Importance.

General format to provide Colombian Seismic load in any load case:

```
LOAD i
COLMBIAN LOAD {X/Y/Z} (f)
```

where i and f are the load case number and factor to multiply horizontal seismic load respectively.

**Examples**

```
DEFINE COLOMBIAN LOAD
ZONE 0.38 I 1.0 S 1.5
JOINT WEIGHT
51 58 93 100 WEIGHT 1440
101 108 143 150 WEIGHT 1000
151 158 193 200 WEIGHT 720
```

**LOAD 1 ( SEISMIC LOAD IN X  
DIRECTION )  
COLOMBIAN LOAD X**

**.31.4 Japanese Seismic Load**

**Purpose**

The purpose of this command is to define and generate static equivalent seismic loads as per Japanese specifications using a static equivalent approach similar to those outlined by UBC. Depending on this definition, equivalent lateral loads will be generated in X or Z direction(s).

**Description**

The STAAD seismic load generator assumes that the lateral loads will be exerted in X and Z directions and Y will be the direction of gravity loads. Thus, for a building model, Y axis will be perpendicular to the floors and point upward ( all Y coordinates positive). The user is advised to set up the model accordingly.

**Methodology :**

Seismic zone coefficient and parameter values are supplied by the user through the DEFINE AIJ LOAD command.

Program calculates the natural period of building T utilizing the following equation

$$T = h ( 0.02 + 0.01 \alpha )$$

where,

- h = height of building
- $\alpha$  = ratio of steel part

Design spectral coefficient ( Rt ) is calculated utilizing T and Tc as follows

$$R_t = \begin{cases} 1.0 & \text{when } T < T_c \\ 1 - 0.2 ( T/T_c - 1 )^2 & \text{when } T_c \leq T \leq 2T_c \\ 1.6 T_c / T & \text{when } 2T_c \leq T \end{cases}$$

$\alpha_i$  is calculated from the weight provided by the user in Define AIJ Load command.

Seismic coefficient of floor Ci is calculated using appropriate equations

$$C_i = Z R_t A_i C_o$$

Where,

- Z = zone factor
- C<sub>o</sub> = normal coefficient of shear force
- A<sub>i</sub> =  $1 + ( 1 / \sqrt{\alpha_i} - \alpha_i ) 2T / ( 1 + 3T )$

The total lateral seismic load is distributed by the program among different levels.

**General Format**

```

DEFINE AIJ LOAD
ZONE f1 ubc-spec
SELFWEIGHT
JOINT WEIGHT
Joint-list WEIGHT w
MEMBER WEIGHT
Mem-list (UNI ....)
    
```

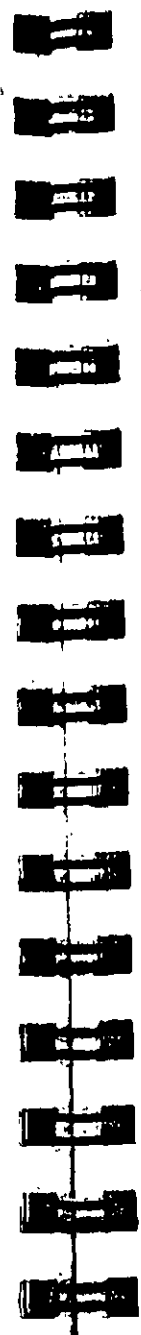
$$ubc-spec = ( f_2, C O f_3, T C f_4 )$$

Where, f1, f2, f3 and f4 are Zone factor, Ratio of Steel Part, Normal coefficient of shear force and Value needed for calculation of Rt.

General format to provide Japanese Seismic load in any load case:

```

LOAD i
AIJ LOAD {X/Y/Z} (f)
    
```



Where, i and f are load case number and factor to multiply horizontal seismic load respectively.

**Example**

```

DEFINE AIJ LOAD
ZONE 0.8 I 0.0 CO 0.2 TC 0.6
JOINT WEIGHT
51 56 93 100 WEIGHT 1440
101 106 143 150 WEIGHT 1000
151 156 193 200 WEIGHT 720
LOAD 1 ( SEISMIC LOAD IN X)
AIJ LOAD X
    
```



### 5.31.5 Definition of Wind Load

**Purpose**

This set of commands may be used to define the parameters for automatic generation of wind loads on the structure.

**General Format:**

```

DEFINE WIND LOAD
TYPE j
INTENSITY p1 p2 p3 ... pn HEIGHT h1 h2 h3 ... hn

EXPOSURE { e1 JOINT joint-list
            e1 YRANGE f1 f2 }

EXPOSURE e2 -do-

...

EXPOSURE en -do-
    
```

where,

- j = wind load system type number (integer)
- p<sub>1</sub>, p<sub>2</sub>, p<sub>3</sub> ... p<sub>n</sub> wind intensities (pressures) in force/area. Up to 5 different intensities can be defined in the input file.
- h<sub>1</sub>, h<sub>2</sub>, h<sub>3</sub> ... h<sub>n</sub> corresponding heights in global vertical direction up to which the above intensities occur.
- e<sub>1</sub>, e<sub>2</sub>, e<sub>3</sub> ... e<sub>n</sub> exposure factors. A value of 1.0 means that the wind force is applied on the full influence area associated with the joint(s).
- joint-list Joint list associated with exposure factor
- f<sub>1</sub> and f<sub>2</sub> global vertical coordinate values to specify vertical range for exposure factor.

If the command EXPOSURE is not specified, the exposure factor is chosen as 1.0.

**Description**

All loads and heights are in current unit system. In the list of intensities, the first value of intensity acts from the ground level up to the first height. The second intensity ( $p_2$ ) acts in the Global vertical direction between the first two heights ( $h_1$  and  $h_2$ ) and so on. The program assumes that the ground level has the lowest global vertical coordinate.

See Section 3

Exposure factor (e) is the fraction of the influence area associated with the joint(s) on which the load acts. Total load on a particular joint is calculated as follows.

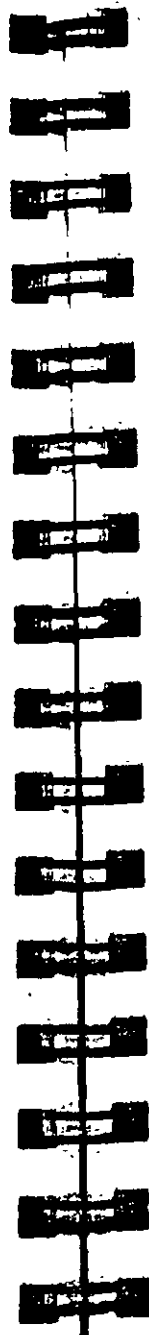
Joint load = (Exposure Factor) X (Influence Area) X (Wind Intensity)  
 The exposure factor may be specified by a joint-list or by giving a vertical range within which all joints will have the same exposure. If exposure factor is not specified, it defaults to 1.0 in which case the entire influence area associated with the joint(s) will be considered.

For PLANE FRAMES, influence area for each joint is calculated considering unit width (1 inch) perpendicular to the plane of the structure. Note that the user can accommodate the actual width by incorporating it in the Exposure Factor as follows

$$\text{Exposure Factor (User Specified)} = (\text{Fraction of influence area}) \times (\text{influence width for joint})$$

**Notes**

All intensities, heights and ranges must be provided in the current unit system.



**5.31.6 Definition of Time History Load**

**Purpose**

This set of commands may be used to define parameters for Time History loading on the structure.

**General format:**

```
DEFINE TIME HISTORY (DT x)
TYPE i { ACCELERATION } (SAVE)
      { FORCE }
```

See Sections 1.18.3 and 5.32.10.2

```
{ READ t_n
  t_1 p_1 t_2 p_2 ... t_n p_n
  function-spec }
```

```
ARRIVAL TIME
a_1 a_2 a_3 ..... a_n
(DAMPING d)
```

where

```
FUNCTION { SINE }
        { COSINE }
```

function-spec =

```
AMPLITUDE t_1 { FREQUENCY } t_2 (PHASE t_3) CYCLES t_4 (STEP t_5)
              { RPM }
```

= solution time step used in the step-by-step integration of the uncoupled equations. The default value of DT is determined as follows-

a) If the highest mode to be included in the response has a frequency larger than 60 cps.  
 DT = 0.0016sec.

b) If the highest mode (Nf th mode) has a frequency less than 60 cps,  $DT = 1/10$ th the period of the Nfth mode.

i = type number of time varying load (integer). Up to 6 types may be provided. ACCELERATION indicates that the time varying load type is a ground motion. FORCE indicates that it is a forcing function.

Save = The save option results in the creation of a file (input file name with a "Tim" extension) containing the history of the displacements of every node of the structure at every time step.  
Syntax: TYPE 1 FORCE SAVE

$t_1 p_1 t_2 p_2 \dots$  = values of time(sec.) and corresponding force (current force unit) or acceleration (current length unit/sec<sup>2</sup>) depending on whether the time varying load is a forcing function or a ground motion. If the data is specified through the input file, up to 299 pairs can be provided for each type in the ascending value of time. More than one line may be used if necessary. However, if the data is provided through an external file, an unlimited number of time-force pairs may be specified.

$a_1 a_2 a_3 \dots a_n$  = Values of the various possible arrival times (seconds) of the various dynamic load types. Arrival time is the time at which a load type begins to act at a joint (forcing function) or at the base of the structure (ground motion). The same load type may have different arrival times for different joints and hence all those values must be specified here. The arrival times and the time-force pairs for the load types are used to create the load vector needed for each time step of the analysis. Refer to Section 5.32.10.2 for information on input specification for application of the forcing function and/or ground motion loads. Up to 99 arrival time values may be specified.

d = Modal damping ratio. Default value is 0.05. Only one modal damping ratio can be used for the entire structure.

Note that the "function-spec" option may be used to specify harmonic loads. Both "sine" and "cosine" harmonic functions may be specified. The program will automatically calculate the harmonic load time history based on the following specifications -

- $f_1$  - Amplitude of motion in current units.
- $f_2$  - If FREQUENCY, then cyclic frequency (cycles / sec.)  
If RPM, then revolutions per minute.
- $f_3$  - Phase Angle in degrees, default = 0
- $f_4$  - No. of cycles of loading.
- $f_5$  - time step of loading, default = one tenth of the period corresponding to the frequency of the harmonic loading.

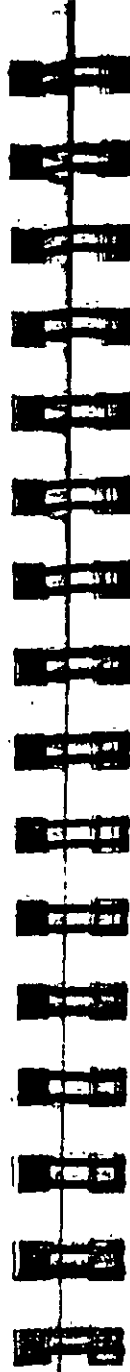
**Example**

```

UNIT FT
DEFINE TIME HISTORY
TYPE 1 FORCE
0.0 1.0 1.0 1.2 2.0 1.8 3.0 2.2
4.0 2.8 5.0 2.8
TYPE 2 ACCELERATION
0.0 2.5 0.5 2.7 1.0 3.2 1.5 3.8
2.0 4.2 2.5 4.5 3.0 4.5 3.5 2.8
ARRIVAL TIME
0.0 1.0 1.8 2.2 3.5 4.4
DAMPING 0.075
    
```

**Notes**

The 'READ fn' command is to be provided only if the history of the time varying load is to be read from an external file. fn is the file name. The data in the external file must be provided as one time-force pair per line as shown in the following example.



Data in Input file

```

UNIT KIP FEET
DEFINE TIME HISTORY
TYPE 1 FORCE
READ THFILE
ARRIVAL TIME
0.0
DAMPING 0.075
    
```

Data in the External file "THFILE"

```

0.0 1.0
1.0 1.2
2.0 1.8
3.0 2.2
4.0 2.8
5.0 2.8
    
```

Example for Harmonic Loading Generator

```

UNIT KIP
DEFINE TIME HISTORY
TYPE 1 FORCE
*Following lines for Harmonic Loading Generator
FUNCTION SINE
AMPLITUDE 6.2831 FREQUENCY 60 CYCLES 100 STEP 0.02
ARRIVAL TIME
0.0
DAMPING 0.075
    
```

To define more than one sinusoidal load, the input specification is as follows :

```

DEFINE TIME HISTORY
TYPE 1 FORCE
FUNCTION SINE
AMPLITUDE 1.925 RPM 10794.0 CYCLES 1000
TYPE 2 FORCE
FUNCTION SINE
AMPLITUDE 1.511 RPM 9794.0 CYCLES 1000
TYPE 3 FORCE
FUNCTION SINE
AMPLITUDE 1.488 RPM 1785.0 CYCLES 1000
ARRIVAL TIME
0.0 0.0013897 0.0084034
DAMPING 0.04
  
```

#### Note

The response (displacements, forces etc.) will contain the contribution of only those modes whose frequency is less than or equal to 60 cps. Contribution of modes with frequency greater than 60 cps is not considered.

## 5.32 Loading Specifications

### Purpose

This section describes the various loading options available in STAAD. The following command may be used to initiate a new load case.

### General format:

**LOADING  $i_1$  (any load title)**

$i_1$  = any unique integer number (upto five digits) to identify the load case. This number need not be sequential with the previous load number.

The **LOADING** command initiates a new load case. Under this heading, all different loads related to this loading number can be input. These different kinds of loads are described below.

### 5.32.1 Joint Load Specification

**Purpose**

This set of commands may be used to specify JOINT loads on the structure.

**General format:**

JOINT LOAD

$$\text{joint-list} \left\{ \begin{array}{l} \text{FX} \\ \text{FY} \\ \text{FZ} \\ \text{MX} \\ \text{MY} \\ \text{MZ} \end{array} \right. \left\{ \begin{array}{l} f_1 \\ f_2 \\ f_3 \\ f_4 \\ f_5 \\ f_6 \end{array} \right.$$

FX, FY and FZ specify a force in the corresponding global direction.

MX, MY and MZ specify a moment in the corresponding global direction.

$f_1, f_2 \dots f_6$  are the values of the loads.

**Example**

JOINT LOAD

3 TO 7 9 11 FY -17.2 MZ 180.0

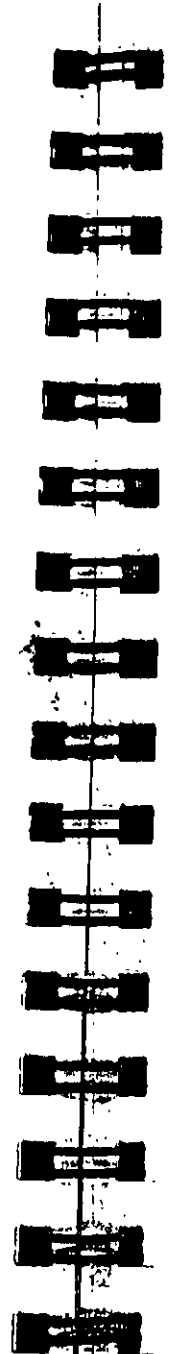
5 8 FX 15.1

12 MX 180.0 FZ 6.3

**Notes**

Joint numbers may be repeated where loads are meant to be additive in the joint.

Section 16.1



### 5.32.2 Member Load Specification

**Purpose**

This set of commands may be used to specify MEMBER loads on frame members.

**General format:**

MEMBER LOAD

$$\text{member-list} \left\{ \begin{array}{l} \text{UNI or UMOM} \\ \text{CON or CMOM} \\ \text{LIN} \\ \text{TRAP} \end{array} \right. \left\{ \begin{array}{l} \text{direction-spec} \\ \text{direction-spec} \\ \text{local-spec} \\ \text{direction-spec} \end{array} \right. \left\{ \begin{array}{l} f_1, f_2, f_3, f_4 \\ f_5, f_6, f_4 \\ f_7, f_8, f_9 \\ f_{10}, f_{11}, f_{12}, f_{13} \end{array} \right.$$

$$\text{direction-spec} = \left\{ \begin{array}{l} X \\ Y \\ Z \\ GX \\ GY \\ GZ \\ PX \\ PY \\ PZ \end{array} \right. \quad \text{local-spec} = \left\{ \begin{array}{l} X \\ Y \\ Z \end{array} \right.$$

UNI or UMOM specifies a uniformly distributed load or moment with a value of  $f_1$ , at a distance of  $f_2$  from the start of the member to the start of the load, and a distance of  $f_3$  from the start of the member to the end of the load. The load is assumed to cover the full member length if  $f_2$  and  $f_3$  are omitted.

CON or CMOM specifies a concentrated force or moment with a value of  $f_5$  applied at a distance of  $f_6$  from the start of the member.  $f_6$  will default to half the member length if omitted.

$f_4 =$  Perpendicular distance from the member shear center to the plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis.



... specifies a linearly decreasing or increasing, or a triangular load. If the load is linearly increasing or decreasing then  $f_7$  is the value at the start of the member and  $f_8$  is the value at the end. If the load is triangular, then  $f_7$  and  $f_8$  are input as zero and  $f_9$  is the value of the load in the middle of the member.

TRAP specifies a trapezoidal linearly varying load which may act over the full or partial length of a member and in a local, global or projected direction. The starting load value is given by  $f_{10}$  and the ending load value by  $f_{11}$ . The loading location is given by  $f_{12}$ , the loading starting point, and  $f_{13}$ , the stopping point. Both are measured from the start of the member. If  $f_{12}$  and  $f_{13}$  are not given, the load is assumed to cover the full member length.

X, Y, & Z in the direction-spec and local-spec specify the direction of the load in the local (member) x, y and z-axes.

GX, GY, & GZ in the direction-spec specify the direction of the load in the global X, Y, and Z-axes.

PX, PY and PZ may be used if the load is to be along the projected length of the member in the corresponding global direction. Load start and end distances are measured along the member length and not the projected length.

**Notes**

Specification of global axes is not permissible for the linear load (LIN option).

If the member being loaded has offset distances (see MEMBER OFFSET specification), the location of the load is measured not from the coordinates of the starting node but from the offset distance.

Trapezoidal loads are converted into uniform loads plus several concentrated loads.



**Example**

```
MEMBER LOAD
619 CON GY -2.35 5.827
68 TO 72 UNI GX -0.088 3.17 10.0
186 TRAP GY -0.24 -0.35 0.0 7.96
3212 LIN X -5.431 -3.335
41016 UNI PZ -0.075
```

### 5.32.3 Element Load Specification

**Purpose**

This command may be used to specify various types of ELEMENT LOADS on the plate elements.

**General format:**

**ELEMENT LOAD**

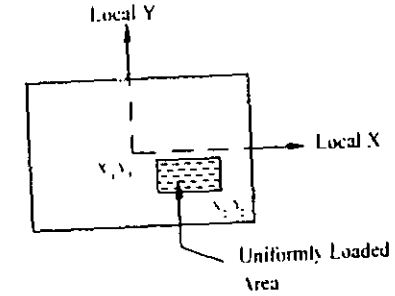
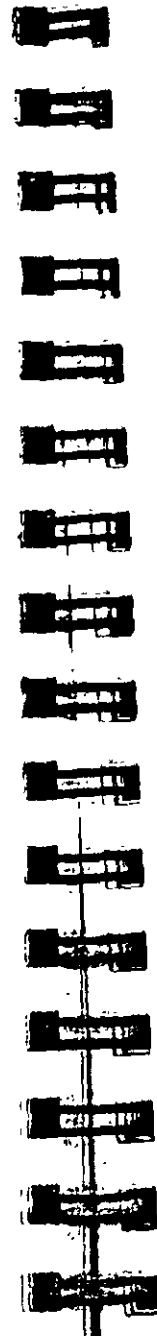
$$\text{element-list} \left\{ \begin{array}{l} \text{PRESSURE} \left\{ \begin{array}{l} \text{GX} \\ \text{GY} \\ \text{GZ} \end{array} \right\} f_1 (x_1 y_1 x_2 y_2) \\ \text{TRAP} \left\{ \begin{array}{l} \text{X} \\ \text{Y} \end{array} \right\} f_2 f_3 \end{array} \right\}$$

**Description**

The PRESSURE option should be used when a UNIFORM pressure needs to be specified. The uniform PRESSURE may be provided on the entire element or an user specified portion of the element (defined by  $x_1, y_1$  and  $x_2, y_2$  - see description below). If  $x_1, y_1$  and  $x_2, y_2$  are not provided, the pressure is applied on the entire element. If only  $x_1, y_1$  is provided, the load is assumed as a concentrated load applied at the specified point.

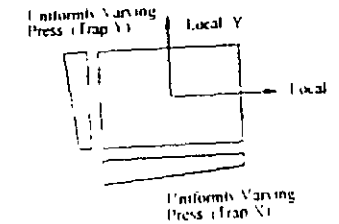
Note that the PRESSURE may be provided either in GLOBAL (GX, GY, GZ) directions or in local Z direction (normal to the element). If the GLOBAL direction is omitted, the applied loading is assumed to be in the local Z direction.

GX,GY,GZ Global direction specification for pressure denotes global X, Y, or Z direction respectively.



See Section 1.6

$f_1$   
 $x_1, y_1$  &  $x_2, y_2$   
 Element pressure (force/square of length) or concentrated load (force). Note that  $f_1$  is assumed as a concentrated load if  $x_2$  and  $y_2$  are omitted. Co-ordinate points in local co-ordinate system (center node is origin) defining the rectangular area on which the pressure is applied.



The TRAP option should be used when a linearly varying pressure needs to be specified. The variation must be provided over the entire element.

X or Y Direction of variation of element pressure. The TRAP X/Y option indicates that the variation of the Trapezoid is in the local X or in the local Y direction. The load always acts in the local Z axis.  
 $f_2$  Pressure intensity at start.  
 $f_3$  Pressure intensity at end

**Note:**

- 1 "Start" and "end" defined above is based on positive directions of the local X or Y axis.
- 2 While the X or Y indicates the direction of variation of the trapezoidal load, the load itself acts in the local Z direction.

Example

```
LOAD 4
ELEMENT LOAD
1 7 TO 10 PR 2.5
11 12 PR 2.5 1.5 2.5 5.5 4.5
15 TO 25 TRAP X 1.5 4.5
34 PR 5.0 2.5 2.5
35 TO 45 PR -2.5
```



### 5.32.4 Area Load/Floor Load Specification

#### Purpose

These commands may be used to specify AREA LOADs or FLOOR LOADs on a structure. The AREA LOAD may be used for modeling one-way distribution and the FLOOR LOAD may be used for modeling two-way distribution.

#### General format for AREA LOAD:

#### AREA LOAD

member-list ALOAD  $f_1$

See Section  
1.16.3

$f_1$  = The value of the area load (unit weight over square length unit). This load always acts along the positive local y-axis. For the members of a FLOOR analysis, this direction will coincide with global vertical axis in most cases. (For detailed description, refer to Section 1.)

#### Example

```
AREA LOAD
2 4 TO 8 ALOAD -.250
12 16 ALOAD -.500
```

#### Note

Area load should not be specified on members declared as MEMBER CABLE, MEMBER TRUSS or MEMBER TENSION.

**General Format for FLOOR LOAD:**

**FLOOR LOAD**

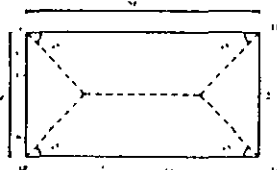
**YRANGE  $f_1$   $f_2$  FLOOR  $f_3$  (XRANGE  $f_4$   $f_5$  ZRANGE  $f_6$   $f_7$ )**

Where

- $f_1$   $f_2$  Global vertical coordinate values to specify vertical range. The floor load will be calculated for all members lying in the global horizontal plane within the specified global vertical range.
- $f_3$  The value of the floor load (unit weight over square length unit). This load always acts parallel to the global vertical axis. A positive value signifies that the load is acting in the positive global Y direction. A negative value indicates a load in the negative global Y direction.
- $f_4$  -  $f_7$  Global X or Z coordinate values to define the corner points of the area on which the specified floor load ( $f_3$ ) acts. If not specified, the floor load will be calculated for all members in all floors within the specified global vertical range.

**Notes**

- 1) The structure has to be modeled in such a way that the global vertical axis remains perpendicular to the floor plane(s)
- 2) For the FLOOR LOAD specification, a two-way distribution of the load is considered. For the AREA LOAD specification, a one-way action is considered.
- 3) FLOOR LOAD from a slab is distributed on the adjoining members as trapezoidal and triangular loads depending on the length of the sides as shown in the diagram. Internally, these loads are converted to multiple point loads.

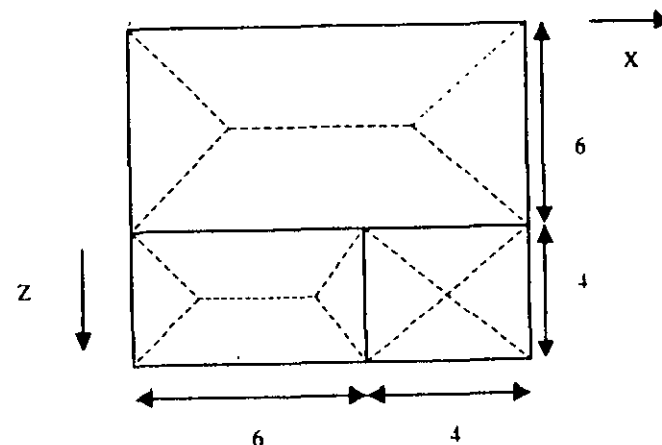


Members 1 and 2 get full trapezoidal and triangular loads respectively. Members 3 and 4 get partial trapezoidal loads and 5 and 6 get partial triangular load.



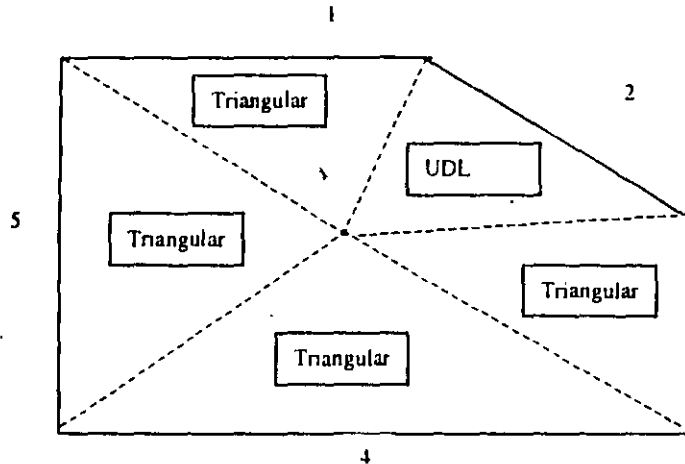
- 4) The load per unit area may not vary for a particular panel and it is assumed to be continuous and without holes.
- 5) The FLOOR LOAD facility is not available if the SET Z UP command is used (See Section 5.5.)

The load distribution pattern depends upon the shape of the panel. If the panel is Rectangular, the distribution will be Trapezoidal and triangular as explained in the following diagram.



For a panel which is not rectangular, the distribution is described in following diagram.

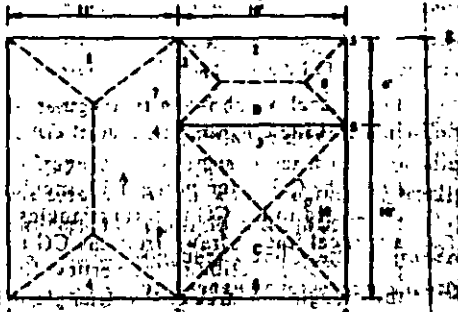
First, the CG of the polygon is calculated. Then, each corner is connected to the CG to form triangles as shown. For each triangle, a vertical line is drawn from the CG to the opposite side. If the point of intersection of the vertical line and the side falls outside the triangle, the area of that triangle will be calculated and an equivalent uniform distributed load will be applied on that side. Otherwise a triangular load will be applied on the side.



**Example**

The input for FLOOR LOAD is explained through an example.

Let us consider the following floor plan at  $y = 12'$ .



If the entire floor has a load of 0.25 (force/unit area), then the input will be as follows:

```
...
LOAD 2
FLOOR LOAD
YR 12.0 12.0 FLOAD -0.25
```

If in the above example, panel A has a load of 0.25 and panels B and C have a load of 0.5, then the input will be as follows:

Note the usage of X RANGE, Y RANGE and Z RANGE specifications.

```
...
LOAD 2
FLOOR LOAD
YR 11.9 12.1 FLOAD -0.25 XR 0.0 11.0 ZR 0.0 16.0
YR 11.9 12.1 FLOAD -0.5 XR 11.0 21.0 ZR 0.0 16.0
LOAD 3
```

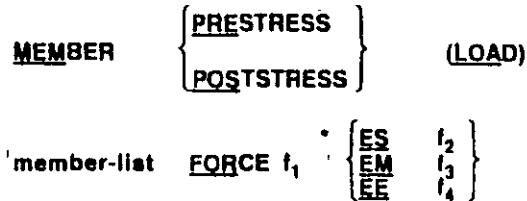
The program internally identifies the panels (shown as A, B and C in the figure). The floor loads are distributed as trapezoidal and triangular loads as shown by dotted lines in the figure. The negative sign for the load signifies that it is applied in the downward global Y direction.

### 5.32.5 Prestress Load Specification

**Purpose**

This command may be used to specify PRESTRESS loads on members of the structure.

**General Format:**



- $f_1$  = Prestressing force. This value is positive in the direction of the local x-axis.
- ES = specifies eccentricity of the prestress force at the start of the member at a distance  $f_2$  from the centroid.
- EM = specifies eccentricity of the prestress force at the mid-point of the member at a distance  $f_3$  from the centroid.
- EE = specifies eccentricity of the prestress force at the end of the member at a distance  $f_4$  from the centroid.

**Description**

The first option, (MEMBER PRESTRESS LOAD), considers the effect of the prestressing force during its application. Thus, transverse shear generated at the ends of the member(s) subject to the prestressing force is transferred to the adjacent members.

The second option, (MEMBER POSTSTRESS LOAD), considers the effect of the existing prestress load after the prestressing operation. Thus, transverse shear at the ends of the member(s) subject to the prestressing force is not transferred to the adjacent members.

See Section 1.1 5



**Example**

```
MEMBER PRESTRESS
2 TO 7 11 FORCE 50.0
MEMBER POSTSTRESS
8 FORCE 30.0 ES 3.0 EM -6.0 EE 3.0
```

In the first example, a prestressing force of 50 kips is applied through the centroid (i.e. no eccentricity) of members 2 to 7 and 11. In the second example, a poststressing force of 30 kips is applied with an eccentricity of 3 inches at the start, -6.0 inches at the middle, and 3.0 at the end of member 8.

One of the limitations in using this command is that under any one load case, on any given member, a prestress or poststress load may be applied only once. If the given member carries multiple stressed cables or has a PRESTRESS and POSTSTRESS load condition, such a situation will have to be specified through multiple load cases for that member. See example below.

**Incorrect Input**

```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
6 FORCE 150 ES 3 EM -6 EE 3
PERFORM ANALYSIS
```

**Correct Input**

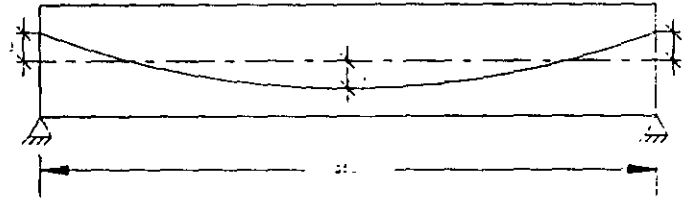
```
LOAD 1
MEMBER PRESTRESS
6 7 FORCE 100 ES 2 EM -3 EE 2
LOAD 2
MEMBER PRESTRESS
6 FORCE 150 ES 3 EM -6 EE 3
LOAD COMB 3
1 1.0 2 1.0
```

**PERFORM ANALYSIS**

**Examples for Modeling Techniques**

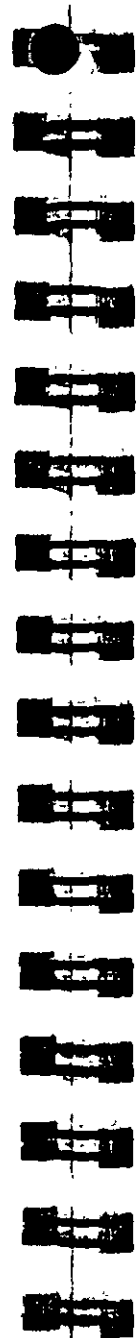
The following examples describe the partial input data for the members and cable profiles shown below.

**Example 1**

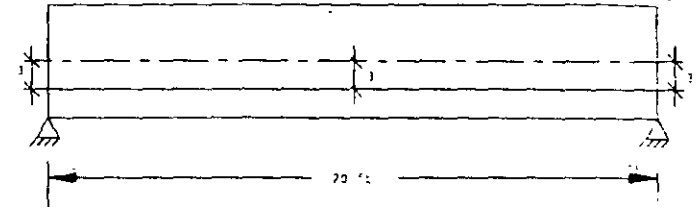


**JOINT COORD**  
 1 0 0 ; 2 10 0  
**MEMB INCI**  
 1 1 2

**UNIT INCH**  
**LOAD 1**  
**MEMBER POSTSTRESS**  
 1 FORCE 100 ES 3 EM -3 EE 3  
**PERFORM ANALYSIS**



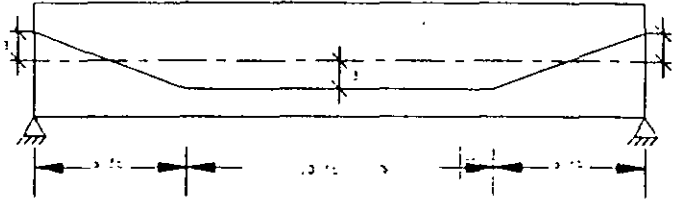
**Example 2**



**JOINT COORD**  
 1 0 0 ; 2 20 0  
**MEMB INCI**  
 1 1 2

**UNIT INCH**  
**LOAD 1**  
**PRESTRESS LOAD**  
 1 FORCE 100 ES -3 EM -3 EE -3  
**PERFORM ANALYSIS**

**Example 3**

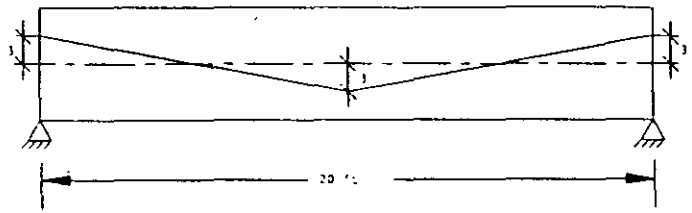


**JOINT COORD**  
 1 0 0 ; 2 5 0 ; 3 15 0 0 ; 4 20 0  
**MEMB INCI**  
 1 1 2 ; 2 2 3 ; 3 3 4

**UNIT INCH**  
**LOAD 1**  
**PRESTRESS LOAD**  
 1 FORCE 100 ES 3 EM 0 EE -3  
 2 FORCE 100 ES -3 EM -3 EE -3  
 3 FORCE 100 ES -3 EM 0 EE 3  
**PERFORM ANALYSIS**



**Example 4**



**JOINT COORD**  
 1 0 0 ; 2 10 0 ; 3 20 0 0  
**MEMB INCI**  
 1 1 2 ; 2 2 3

**UNIT INCH**  
**LOAD 1**  
**PRESTRESS LOAD**  
 1 FORCE 100 ES 3 EM 0 EE -3  
 2 FORCE 100 ES -3 EM 0 EE 3  
**PERFORM ANALYSIS**



**Example 5**



**JOINT COORD**  
 1 0 0 ; 2 10 0 ; 3 20 0 0  
**MEMB INCI**  
 1 1 2 ; 2 2 3

**UNIT INCH**  
**LOAD 1**  
**PRESTRESS LOAD**  
 1 FORCE 100 ES 3 EM -3 EE 3  
 2 FORCE 100 ES 3 EM -3 EE 3  
**PERFORM ANALYSIS**



### 5.32.6 Temperature Load Specification for Members and Elements

**Purpose**

This command may be used to specify TEMPERATURE loads or strain loads on members and elements.

**General format:**

**TEMPERATURE LOAD**

$$\text{memb/elem-list} \left\{ \begin{array}{l} \text{TEMP } t_1 \ t_2 \\ \text{STRAIN } t_3 \end{array} \right\}$$

$t_1$  = The change in temperature which will cause axial elongation in the members or uniform volume expansion in elements. The temperature unit is the same as the unit chosen for the coefficient of thermal expansion ALPHA under the CONSTANT command.

$t_2$  = The temperature differential from the top to the bottom of the member or element ( $T_{\text{top surface}} - T_{\text{bottom surface}}$ ). If  $t_2$  is omitted, no bending will be considered.

$t_3$  = Initial axial elongation (+)/ shrinkage (-) in member due to misfit, etc. in length unit.

See Section 1.16.6

**Example**

**TEMP LOAD**  
 1 TO 9 15 17 TEMP 70.0  
 18 TO 23 TEMP 80.0 66.0  
 8 TO 13 STRAIN 0.45E-4

**Note**

It is not necessary nor possible to specify the units for temperature or for ALPHA. The user must ensure that the value provided for ALPHA is consistent in terms of units with the value provided for the temperature load. (see Section 5.26)

**5.32.7 Fixed-End Load Specification****Purpose**

This command may be used to specify FIXED-END loads on members of the structure.

**General format:****FIXED ( END ) LOAD**

**member-list  $f_1, f_2, \dots, f_{12}$**

See Section  
1.16.4

$f_1 \dots f_6$  = Force-x, shear-y, shear-z, torsion, moment-y, moment-z  
(all in local coordinates) at the start of the member.

$f_7 \dots f_{12}$  = Same as above except at the end of the member

### 5.32.8 Support Displacement Load Specification

**Purpose**

This command may be used to specify SUPPORT DISPLACEMENT load on supports of the structure.

**General format:**

**SUPPORT DISPLACEMENT ( LOAD )**

support joint-list  $\left\{ \begin{array}{l} FX \\ FY \\ FZ \\ MX \\ MY \\ MZ \end{array} \right\} f_1$

With this command, the support displacement is modeled as a load. Note that displacement cannot be specified in a direction in which the support is released.

See Section 16.7

FX, FY, FZ specify translational displacements in global X, Y, and Z directions respectively. MX, MY, MZ specify rotational displacements in global X, Y, and Z directions.

$f_1$  = Value of the corresponding displacement. For translational displacements, the unit is in the currently specified length unit, while for rotational displacements the unit is always in degrees.



**Example**

**UNIT INCHES  
SUPPORT DISPL  
5 TO 11 13 FY -0.25  
19 21 TO 25 MX 15.0**

In this example, the joints of the first support list will be displaced by 0.25 inch in the negative global Y direction. The joints of the second support list will be rotated by 15 degrees about the global X-axis.

**Notes**

Support displacements can be applied in upto 4 load cases only

The support displacement load should not be applied on a structure which contains finite elements.

### 5.32.9 Selfweight Load Specification

**Purpose**

This command may be used to calculate and apply the SELFWEIGHT of the structure for analysis.

**General format:**

$$\text{SELFWEIGHT} \left\{ \begin{matrix} X \\ Y \\ Z \end{matrix} \right\} f_1$$

This command is used if the self-weight of the structure is to be considered. The self-weight of every active member is calculated and applied as a uniformly distributed member load.

X, Y, & Z represent the global direction in which the selfweight acts.

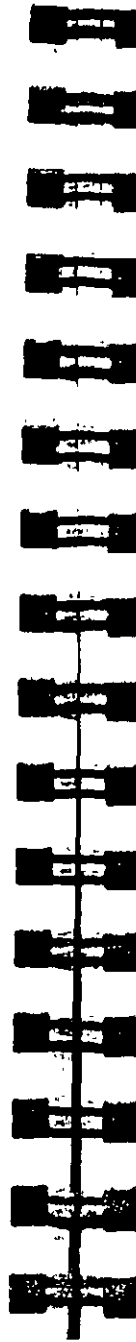
$f_1$  = The factor to be used to multiply the selfweight.

This command may also be used without any direction and factor specification. Thus, if specified as "SELFWEIGHT", loads will be applied in the negative global Y direction with a factor of unity

**Notes**

Density must be provided for calculation of the self weight.

The selfweight of finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.



### 5.32.10 Dynamic Loading Specification

**Purpose**

The command specification needed to perform response spectrum analysis and time-history analysis is explained in the following sections.

### 5.32.10.1 Response Spectrum Specification

#### Purpose

This command may be used to specify and apply the RESPONSE SPECTRUM loading for dynamic analysis.

#### General Format:

$$\text{SPECTRUM} \begin{Bmatrix} \text{SRSS} \\ \text{CQC} \end{Bmatrix} \begin{Bmatrix} \text{X} \\ \text{Y} \\ \text{Z} \end{Bmatrix} \begin{Bmatrix} f_1 \\ f_2 \\ f_3 \end{Bmatrix} \begin{Bmatrix} \text{ACC} \\ \text{DIS} \end{Bmatrix} (\text{SCALE } f_4) (\text{DAMP } f_5)$$

$$P_1, V_1; P_2, V_2; P_3, V_3; P_4, V_4 \dots P_n, V_n$$

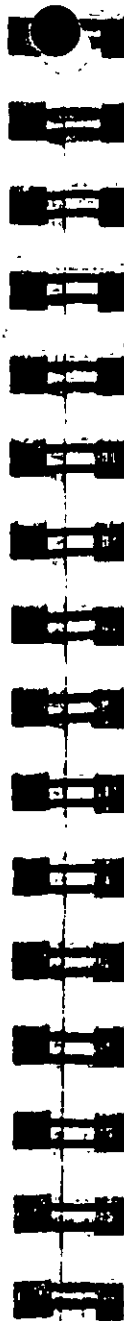
Where,  $f_1, f_2, f_3$  are the factors for input spectrum to be applied in X, Y & Z directions. Any one or all directions can be input. Directions not provided will default to zero.

Where,  $f_4$  = Scale factor by which the response spectrum is modified. Defaults to 1.0 if not specified.

Where,  $f_5$  = Damping Factor. Default value is 0.05 (5% damping). This value is necessary for the CQC Method only

Where, ACC or DIS stands for Acceleration or Displacement.

Where,  $P_1, V_1; P_2, V_2; \dots; P_n, V_n$  = Values of periods (sec) and corresponding acceleration (current length unit/sec<sup>2</sup>) or displacement (current length unit) as the case may be. More than one line may be used if necessary. Use of hyphens (-) at the end of the line is not permitted to continue data to the next line. Spectrum pairs should be provided in ascending value of period, with a maximum of 99 spectrum pairs.



#### Description

Note that if SPECTRUM SRSS is used, modal combinations are done according to SRSS (square root of summation of squares) method. Otherwise, the CQC (complete quadratic combination) method is used.

This command should appear as part of a loading specification. If it is the first occurrence, it should be accompanied by the load data to be used for frequency and mode shape calculations. Additional occurrences need no additional information. Maximum response spectrum load cases allowed in one run is 4.

See Sections  
1 18 3, 5 30,  
and 5 34

Results of frequency and mode shape calculations may vary significantly depending upon the mass modeling. All active masses should be modeled as loads. All masses that are capable of moving should be modeled as loads, applied in all possible directions of movement. In response spectrum analysis, all masses that are capable of moving in the direction of the spectrum must be provided as loads acting in that direction. An illustration of mass modeling is available, with explanatory comments, in Example Problem No. 11

#### Example

##### LOAD 2 SPECTRUM IN X-DIRECTION

SELFWEIGHT X 1.0

SELFWEIGHT Y 1.0

SELFWEIGHT Z 1.0

JOINT LOAD

10 FX 17.5

10 FY 17.5

10 FZ 17.5

SPECTRUM SRSS X 1.0 ACC SCALE 32.2

0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47

1.2 0.5 ; 1.4 0.65 ; 1.6 0.87 ; 1.8 0.55 ; 2.0 0.43

### Multiple Response Spectra

If there is more than one response spectrum defined in the input file, the load data should accompany the first set of spectrum data only. In the subsequent load cases, only the spectra should be defined. See example below.

#### LOAD 1 SPECTRUM IN X-DIRECTION

SELFWEIGHT X 1.0  
 SELFWEIGHT Y 1.0  
 SELFWEIGHT Z 1.0

#### JOINT LOAD

10 FX 17.5  
 10 FY 17.5  
 10 FZ 17.5

#### SPECTRUM SRSS X 1.0 ACC SCALE 32.2

0.20 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.0 0.47  
 1.2 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43

#### LOAD 2 SPECTRUM IN Y-DIRECTION

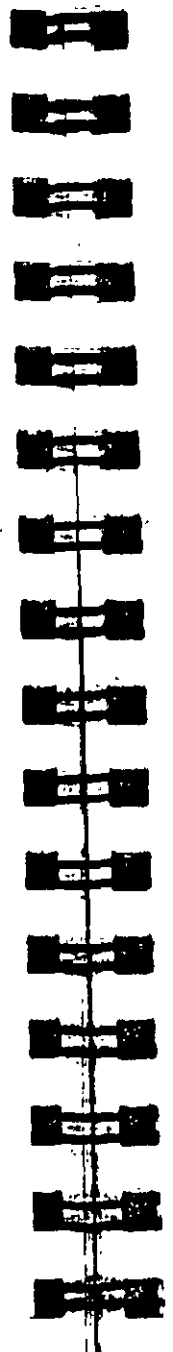
#### SPECTRUM SRSS Y 1.0 ACC SCALE 32.2

0.20, 0.1 ; 0.40 0.15 ; 0.60 0.33 ; 0.80 0.45 ; 1.00 0.48  
 1.20 0.51 ; 1.4 0.63 ; 1.6 0.67 ; 1.8 0.54 ; 2.0 0.42

#### LOAD 3 SPECTRUM IN Z-DIRECTION

#### SPECTRUM SRSS Z 1.0 ACC SCALE 32.2

0.20, 0.2 ; 0.40 0.25 ; 0.60 0.35 ; 0.80 0.43 ; 1.00 0.47  
 1.20 0.5 ; 1.4 0.65 ; 1.6 0.67 ; 1.8 0.55 ; 2.0 0.43



### 5.32.10.2 Application of Time Varying Load for Response History Analysis

#### Purpose

This set of commands may be used to model Time History loading on the structure for Response Time History analysis. Note that both nodal time histories and ground motion time histories may be provided.

#### General format:

#### TIME LOAD

joint list  $\left\{ \begin{matrix} FX \\ FY \\ FZ \\ MX \\ MY \\ MZ \end{matrix} \right\} \quad t_1 \quad t_2$

See Sections  
 1.18.3, and  
 5.31.4

GROUND MOTION  $\left\{ \begin{matrix} X \\ Y \\ Z \end{matrix} \right\} \quad t_1 \quad t_2$

Where  $t_1$  = type number of time varying load (see Section 5.31.4)  
 $t_2$  = arrival time number (see Section 5.31.4) (integer).  
 This is the sequential number of the arrival time in the list explained in section 5.31.4. Thus the arrival time number of  $a_3$  is 3 and of  $a_n$  is n.

Note that either TIME LOAD or GROUND MOTION or both may be specified under one load case. More than one load case for time history analysis is not permitted.

**Example**

```

LOAD 1
SELFWEIGHT X 1.0
SELFWEIGHT Y 1.0
SELFWEIGHT Z 1.0
MEMBER LOADS
5 CON GX 7.5 10.0
5 CON GY 7.5 10.0
5 CON GZ 7.5 10.0
TIME LOAD
2 3 FX 1 3
5 7 FX 1 6
GROUND MOTION X 2 1
    
```

In the above example, the permanent masses in the structure are provided in the form of "selfweight" and "member loads" for obtaining the mode shapes and frequencies. The rest of the data is the input for application of the time varying loads on the structure. Forcing function type 1 is applied at joints 2 and 3 starting at arrival time number 3. (Arrival time number 3 is 1.8 seconds in example shown in section 5.31.4) Similarly, forcing function type 1 is applied at joints 5 and 7 starting at arrival time number 6 (4.4 seconds). A ground motion (type 2) acts on the structure in the x-direction starting at arrival time number 1 (0.0 seconds).



### 5.32.11 Repeat Load Specification

**Purpose**

This command is used to create a primary load case using combinations of previously defined primary load cases.

**General format:**

REPEAT LOAD  
 $l_1, f_1, l_2, f_2 \dots l_n, f_n$

where,

$l_1, l_2 \dots l_n$  = primary load case numbers  
 $f_1, f_2 \dots f_n$  = corresponding factors

**Description**

This command may be used to create a primary load case using combinations of previously defined primary load case(s). The REPEAT load differs from the load COMBINATION command (Section 5.35) in two ways:

- 1) A REPEAT LOAD is treated as a new primary load. Therefore, a P-Delta analysis will reflect correct secondary effects. (LOAD COMBINATIONS, on the other hand, algebraically combine the effects of previously defined primary loadings evaluated independently).
- 2) In addition to previously defined primary loads, the user can also add new loading conditions.
- 3) The REPEAT LOAD option is available with load cases with JOINT LOADS and MEMBER LOADS containing UNI, UMOM and CON specifications only. It is not available for MEMBER LOADS with LIN and TRAP specifications. It can also be used on load cases with ELEMENT PRESSURE loads

(without the TRAP specification). SUPPORT DISPLACEMENTS LOAD, TEMPERATURE LOAD on members or elements, RESPONSE SPECTRUM LOAD and TIME HISTORY LOAD should not be used in REPEAT LOAD. It is also not available for loads generated using the program's load generation facilities such as UBC LOAD Generation, WIND LOAD Generation, MOVING LOAD Generation, etc.

#### Example

```
LOAD 1 DL + LL
SELFWEIGHT Y -1.4
MEMBER LOAD
1 TO 7 UNIFORM Y -3.5
LOAD 2 DL + LL + WL
REPEAT LOAD
1 1.10
```

- 5) The maximum number of load cases that can be combined using a REPEAT LOAD command is 14.



### 5.32.12 Generation of Loads

#### Purpose

This command is used to generate Moving Loads, UBC Seismic loads and Wind Loads using previously specified load definitions.

Primary load cases may be generated using previously defined load systems. The following sections describe generation of moving loads, UBC seismic loads and Wind Loads.

#### Generation of Moving Loads

Pre-defined moving load system types may be used to generate the desired number of primary load cases, each representing a particular position of the moving load system on the structure. This procedure will simulate the movement of a vehicle in a specified direction on a specified plane on the structure.

See  
Sections  
1.17 and  
5.31.1

#### General format:

**LOAD GENERATION n (ADD LOAD i)**

$$\text{TYPE } j \ x_1 \ y_1 \ z_1 \ \left\{ \begin{array}{l} \text{XINC} \ f_1 \\ \text{YINC} \ f_2 \\ \text{ZINC} \ f_3 \end{array} \right\} \left( \left\{ \begin{array}{l} \text{YRANGE} \\ \text{ZRANGE} \end{array} \right\} r \right)$$

where,

- n = total no. of primary load cases to be generated.
- i = load case no. for the previously defined load case to be added to the generated loads.
- j = type no. of previously defined load system.
- $x_1, y_1, z_1$  = x, y and z coordinates (global) of the initial position of the reference load.
- $f_1, f_2, f_3$  = x, y or z (global) increments of position of load system to be used for generation of subsequent load cases.
- r = (Optional) defines section of the structure along global vertical direction to carry moving load. This r value is



added to the reference vertical coordinate ( $y_1$  or  $z_1$ ) in the positive global vertical direction. The moving load will be externally distributed among all members within the vertical range thus generated.  $r$  always should be a positive number. In other words, the program always looks for members lying in the range  $Y_1$  and  $Y_1+ABS(r)$  or  $Z_1$  and  $Z_1+ABS(r)$ .

The ADD LOAD specification may be used to add a previously defined load case to all the load cases generated by the LOAD GENERATION command. In the example below, the SELFWEIGHT specified in load case 1 is added to all the generated load cases.

Sequential load case numbers will be assigned to the series of generated primary load cases. Numbering will begin at one plus the immediate previous load case number. Allow for these when specifying load cases after load case generation.

**Notes**

Primary load cases can be generated from Moving Load systems for frame members only. This feature does not work on finite elements.

**Example**

```
LOAD 1 DL ONLY
SELFWEIGHT
LOAD GENERATION 20 ADD LOAD 1
TYPE 1 0. 5. 10. XI 10.
TYPE 2 0. 10. 10. Z1 15.
LOAD 22 LIVE LOAD ON PAVEMENT
MEMB LOAD
10 TO 20 30 TO 40 UNI GY -5.0
LOAD COMBINATION 31
10 0.75 22 0.75
PERFORM ANALYSIS
```



**Generation of UBC Seismic Load**

See Sections  
1 17 2 and  
5 31 2

Built-in algorithms will automatically distribute the base shear among appropriate levels and the roof per UBC specifications. The following general format should be used to generate UBC load in a particular direction.

**General Format:**

$$\text{LOAD } i \quad \left\{ \begin{matrix} X \\ Y \\ Z \end{matrix} \right\} \quad (f)$$

where  $i$  = load case number  
 $f$  = factor to be used to multiply the UBC Load  
(default = 1.0)

**Example**

```
DEFINE UBC LOAD
ZONE 0.2 K 1.0 I 1.5 TS 0.5
SELFWEIGHT
JOINT WEIGHT
1 TO 100 WEIGHT 5.0
101 TO 200 WEIGHT 7.5
LOAD 1 UBC IN X-DIRECTION
UBC LOAD X
JOINT LOAD
5 25 30 FY -17.5
LOAD 2 UBC IN Z-DIRECTION
UBC LOAD Z
LOAD 3 DEAD LOAD
SELFWEIGHT
LOAD COMBINATION 4
1 0.75 2 0.75 3 1.0
```

In the above example, notice that the first two load cases are UBC load cases. They are specified before any other load cases

### Notes

- 1) The UBC load cases should be provided as the first set of load cases. Non-UBC primary load case specified before a UBC load case is not acceptable. Additional loads such as MEMBER LOADS and JOINT LOADS may be specified along with the UBC load under the same load case.

### Incorrect usage

```
LOAD 1
SELFWEIGHT Y -1
LOAD 2
JOINT LOAD
3 FX 45
LOAD 3
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
LOAD 4
UBC LOAD Z 1.2
MEMBER LOAD
3 UNI GY -4.5
PERFORM ANALYSIS
```

### Correct usage

```
LOAD 1
UBC LOAD X 1.2
JOINT LOAD
3 FY -4.5
LOAD 2
UBC LOAD Z 1.2
MEMBER LOAD
```



```
3 UNI GY -4.5
LOAD 3
SELFWEIGHT Y -1
LOAD 4
JOINT LOAD
3 FX 45
PERFORM ANALYSIS
```

- 2) All load cases involving UBC Load generation must be provided before the ANALYSIS specification. In other words, multiple analyses in which the UBC load generation is performed in the separate analyses is not permitted.

### Incorrect usage

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
LOAD 2
UBC LOAD Z 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
PDELTA ANALYSIS
```

### Correct usage

```
LOAD 1
UBC LOAD X 1.2
SELFWEIGHT Y -1
JOINT LOAD
3 FY -4.5
```

**LOAD 2**  
**UBC LOAD Z 1.2**  
**SELFWEIGHT Y -1**  
**JOINT LOAD**  
**3 FY -4.5**  
**PDELTA ANALYSIS**

- 3) REPEAT LOAD specification cannot be used for load cases involving UBC load generation. For example,

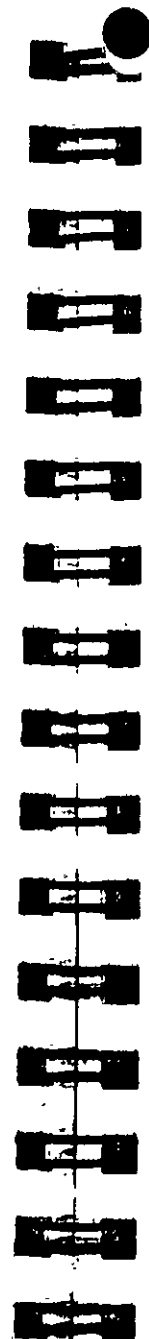
**Incorrect usage**

**LOAD 1**  
**UBC LOAD X 1.0**  
**LOAD 2**  
**SELFWEIGHT Y -1**  
**LOAD 3**  
**REPEAT LOAD**  
**1 1.4 2 1.2**  
**PDELTA ANALYSIS**

- 4) If UBC load generation is performed for the X and the Z directions, the command for the X direction must precede the command for the Z direction.

**Incorrect usage**

**LOAD 1**  
**UBC LOAD Z 1.2**  
**SELFWEIGHT Y -1**  
**LOAD 2**  
**UBC LOAD X 1.2**  
**SELFWEIGHT Y -1**  
**PDELTA ANALYSIS**



**Correct usage**

**LOAD 1**  
**UBC LOAD X 1.2**  
**SELFWEIGHT Y -1**  
**LOAD 2**  
**UBC LOAD Z 1.2**  
**SELFWEIGHT Y -1**  
**PDELTA ANALYSIS**

**Generation of Wind Load**

The built-in wind load generation facility can be used to calculate the wind loads based on the parameters defined in Section 5.31.3. The following general format should be used to perform the wind load generation.

**General Format:**

$$\begin{matrix} \text{LOAD } i \\ \text{WIND LOAD} \end{matrix} \left\{ \begin{matrix} X \\ Y \\ Z \end{matrix} \right\} (f) \text{ TYPE } j$$

See  
 Sections  
 1 17.3 and  
 5 31.3

**Where**

- i Load case number
- X or Z Direction of wind in global axis system.
- j Type number of previously defined systems
- f The factor to be used to multiply the wind load. Negative signs maybe used to indicate opposite direction of wind (default=1.0)

## Example

```

DEFINE WIND LOAD
TYPE 1
INTENSITY 0.1 0.12 HEIGHT 100 200
EXP 0.6 JOI 1 TO 25 BY 7 29 TO 37 BY 4 22 23
TYPE 2
INT 0.1 0.12 HEIGHT 100 900
EXP 0.3 YR 0 500
LOAD 1
SELF Y -1.0
LOAD 2
WIND LOAD Z 1.2 TYPE 2
LOAD 3
WIND LOAD X TYPE 1

```



## 5.33 Natural Frequency Specification

## Purpose

This command may be used to calculate the natural frequency of the structure for vibration corresponding to the general direction of deflection generated by the load case which precedes this command. Thus, this command typically follows a load case.

## General format:

**CALCULATE NATURAL (FREQUENCY)**

## Description

This command is specified after all other load specifications of any primary load case for which the natural frequency is calculated. This natural frequency calculation is based on the Rayleigh iteration method. If a full-scale eigensolution is required, the MODAL CALCULATION command (see next section) may be used. Note that an eigensolution is automatically performed if a RESPONSE SPECTRUM is specified in any load case.

See Section  
1.18.3

## Example

```

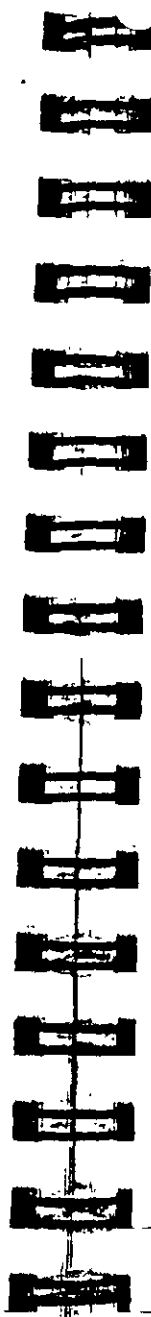
LOADING 1 DEAD AND LIVE LOAD
AREA LOAD
1 TO 23 ALLOAD -200.0
CALCULATE NATURAL FREQ
LOADING 2 WIND LOAD

```

In this example, the natural frequency for load case 1 will be calculated. The output will produce the value of the natural frequency in cycles per second (cps), the maximum deflection along with the global direction and the joint number where it occurs.

**Notes**

This command is based on Rayleigh method of iteration. The frequency calculated estimates frequency for the mode shape that corresponds to the static deflected shape generated by the loads in the load case.



## 5.34 Modal Calculation Command

**Purpose**

This command may be used to obtain a full scale eigensolution to calculate relevant frequencies and mode shapes.

**General Format:**

### MODAL (CALCULATION REQUESTED)

This command is typically used in a load case after all loads are specified. The loads will be treated as masses for eigensolutions. Static displacements and forces will be calculated for the load case.

This command must only be used for one load case, and can not be used in conjunction with a response spectrum analysis (see Verification Problem No. 2 for sample implementation).

See Sections  
5.31 and  
2.18.3

**Notes**

The eigensolution initiated by this command will treat the loads specified in the load case as masses. The user is advised to specify the loads keeping this in mind.

## 5.35 Load Combination Specification

### Purpose

This command may be used to combine the results of the analysis. The combination may be algebraic, SRSS and a combination of both.

### General format:

**LOAD COMBINATION (SRSS) i a<sub>1</sub>**

**i<sub>1</sub>, f<sub>1</sub>, i<sub>2</sub>, f<sub>2</sub> ... (f<sub>SRSS</sub>)**

- i** = Load combination number ( any integer smaller than 100000 that is not the same as any previously defined primary load case number.)
- a<sub>1</sub>** = Any title for the load combination.
- i<sub>1</sub>, i<sub>2</sub> ..** represents the load case numbers which are to be combined.
- f<sub>1</sub>, f<sub>2</sub> ..** represents corresponding factors to be applied to loadings.
- f<sub>SRSS</sub>** = optional factor to be applied as a multiplying factor on the combined result of the SRSS load combination (see examples below).

### Notes

- 1) In the LOAD COMBINATION SRSS option, if the minus sign precedes any load case no., then that load case will be combined algebraically with the SRSS combination of the rest.
- 2) The total number of combination load cases cannot exceed 150.

### Description

Results from analyses may be combined both algebraically and using the SRSS (Square Root of Summation of Squares) method. The combination scheme may be mixed if required. For example, in the same load combination case, results from load cases may be combined in the SRSS manner and then combined algebraically with other load cases. Refer to the following examples for illustration -

### Example

Several combination examples are provided to illustrate the possible combination schemes -

#### Simple Algebraic and SRSS Combinations

```
LOAD COMBINATION 7 DL+LL+WL
1 0.75 2 0.75 3 1.33
LOAD COMBINATION SRSS 8 DL+SEISMIC
1 1.0 2 0.4 3 0.4
```

The first item above (LOAD COMBINATION 7) illustrates a simple algebraic combination. The second item (LOAD COMBINATION 8) illustrates a pure SRSS load combination with a default SRSS factor of 1. The following combination scheme will be used -

$$v = 1.0 \sqrt{1 \times L1^2 + 0.4 \times L2^2 + 0.4 \times L3^2}$$

where v = the combined value and L1 - L3 = values from load cases 1, 2 and 3.

Note that since a SRSS factor is not provided, the default value of 1.0 is being used.

**Algebraic & SRSS Combination in the Same Load Combination Case**

**Example 1**

**LOAD COMBINATION SRSS 9**  
**-1 0.75 2 1.3 3 2.42 0.75**

The combination formula will be as follows -

$$v = 0.75 \times L1 + 0.75 \sqrt{1.3 \times L2^2 + 2.42 \times L3^2}$$

where v = combined value  
 L2 & L3 = values from load cases 2 & 3.

In the above specification, note that a minus sign precedes load case 1. Thus, Load 1 is combined algebraically with the result obtained from combining load cases 2 and 3 in the SRSS manner. Note that the SRSS factor of 0.75 is applied on the SRSS combination of 2 and 3.

**Example 2**

**LOAD COMBINATION SRSS 10**  
**-1 0.75 -2 0.572 3 1.2 4 1.7 0.63**

Here, both load cases 1 and 2 are combined algebraically with the SRSS combination of load cases 3 and 4. Note the SRSS factor of 0.75. The combination formula will be as follows.

$$v = 0.75 \times L1 + 0.572 \times L2 + 0.63 \sqrt{1.2 \times L3^2 + 1.7 \times L4^2}$$

**Notes**

- 1) This option combines the results of the analysis in the specified manner. It does not analyze the structure for the combined loading.

- 2) If the secondary effects of combined load cases is to be obtained through a PDELTA ANALYSIS, the LOAD COMBINATION command is inappropriate for the purpose. See the REPEAT LOAD command (section 5.32.11) for details.
- 3) In a load combination specification, a value of 0 (zero) as a load factor is not permitted. In other words, a specification such as

**LOAD COMB 7**  
**1 1.35 2 0.0 3 1.2 4 0.0 5 1.7**

is not permitted. What happens is that due to the way the program processes the data, as soon as it encounters a 0.0, it stops reading the data at the 0.0 and does not read any further. Thus, in the above loading case, the results will consist only of forces due to load 1 multiplied by a factor of 1.35. The contribution from 1.2 \* load 3 and 1.7 \* load 5 will not be there because the program simply did not read those data.

- 6) All combination load cases must be provided immediately after the last primary load case.
- 7) The maximum number of load cases that can be combined using a LOAD COMBINATION command is 30.



## 5.36 Calculation of Problem Statistics

### Purpose

This command may be used for obtaining run statistics like storage space required, band-width info etc.

### General format:

**PRINT PROBLEM STATISTICS**

### Description

This command provides an estimate of the hard disk space required to run a file before actually running it. It is particularly recommended for PC users running a large problem. This command should be used just after the loading specifications and in place of the PERFORM ANALYSIS command.

See Section 1.18

## 5.37 Analysis Specification

### Purpose

STAAD analysis options include linear static analysis, P-Delta (or second order analysis), Nonlinear analysis, and several types of Dynamic analysis

This command is used to specify the analysis request. In addition, this command may be used to request various analysis related data like load info, statics check info, mode shapes etc.

### General format:

**{ PERFORM  
NONLINEAR (n) } ANALYSIS (PRINT { LOAD DATA  
STATICS CHECK  
STATICS LOAD  
MODE SHAPES  
BOTH  
ALL })**

**PDELTA (n) ANALYSIS (CONVERGE (m)) (PRINT LOAD DATA)**

Where n = no. of iterations desired (default value of n = 1).

See Section 1.18

This command directs the program to perform the analysis which includes:

- Checking whether all information is provided for the analysis;
- Forming the joint stiffness matrix;
- Checking the stability of the structure;
- Solving simultaneous equations, and
- Computing the member forces and displacements.
- If P-Delta analysis is specified, forces and displacements are recalculated, taking into consideration the P-Delta effect.
- Non-linear analysis will take the geometric non-linearity as well as the P-Delta effects into account (see Section 1.18.2.2).



- h) If a RESPONSE SPECTRUM is specified within a load case or the MODAL CALCULATION command is used, dynamic analysis is performed.
- i) In each of the "n" iterations of the PDELTA analysis, the load vector will be modified to include the secondary effect generated by the displacements caused by the previous analysis.

There are two options to carry out P-Delta analysis.

- 1) When the CONVERGE command is not specified: The member end forces are evaluated by iterating "n" times. The default value of "n" is 1 (one).
- 2) When the CONVERGE command is included: The member end forces are evaluated by performing a convergence check on the joint displacements. In each step, the displacements are compared with those of the previous iteration in order to check whether convergence is attained. In case "m" is specified, the analysis will stop after that iteration even if convergence has not been achieved. If convergence is achieved in less than "m" iterations, the analysis is terminated.

### Example

Followings are some example on use of the command for P-Delta analysis.

```
PDELTA ANALYSIS
PDELTA 5 ANALYSIS
PDELTA ANALYSIS CONVERGE
PDELTA ANALYSIS CONVERGE 5
```

Without one of these analysis commands, no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

Note that a PDELTA ANALYSIS will correctly reflect the secondary effects of a combination of load cases only if they are defined using the REPEAT LOAD specification (Section 5.32.11). Secondary effects will not be evaluated correctly for LOAD COMBINATIONS.

If the PRINT LOAD DATA command is specified, the program will print an interpretation of all the load data.

PRINT STATICS CHECK will provide a summation of the applied loads and support reactions as well as a summation of moments of the loads and reactions taken around the origin.

PRINT STATICS LOAD prints everything that PRINT STATICS CHECK does, plus it prints a summation of all internal and external forces at each joint (generates voluminous output).

PRINT MODE SHAPES prints mode shape values at the joints for all calculated mode shapes.

PRINT BOTH is equivalent to PRINT LOAD DATA plus PRINT STATICS CHECK.

PRINT ALL is equivalent to PRINT LOAD DATA plus PRINT STATICS LOAD.

**Notes**

STAAD allows multiple analyses in the same run. Multiple analyses may be used for the following purposes:

- 1) Successive analysis and design cycles in the same run result in optimized design. STAAD's live relational database automatically updates changes in member cross-sectional sizes. Thus the entire process is automated.

Refer to Example 1 in the Getting Started & Examples manual for detailed illustration.

- 2) Multiple analyses may be used for load-dependent structures. For example, structures with bracing members are analyzed in several steps. The bracing members are assumed to take Tension load only. Thus, they need to be activated and inactivated based on the direction of lateral loading.

The entire process can be modeled in one STAAD run using multiple PERFORM ANALYSIS commands. The STAAD run database automatically keeps track of results for different runs and is capable of generating a design based on load combinations provided.

Refer to Example 4 in the Getting Started & Examples manual for detailed illustration

- 3) The user may also use Multiple Analyses to model change in other characteristics like SUPPORTS, RELEASES, SECTION PROPERTIES etc.
- 4) Multiple Analyses may require use of additional commands like the SET NL command and the CHANGE command.
- 5) PDELTA effects are computed for frame members only. They are not calculated for finite elements or solid elements.



### 5.38 Change Specification

**Purpose**

This command is used to reset the stiffness matrix. Typically, this command is used when multiple analyses are required in the same run.

**General format:**

**CHANGE**

This command indicates that input, which will change the stiffness matrix, will follow. This command should only be used when an analysis has already been performed. The CHANGE command does the following:

- a) sets the stiffness matrix to zero.
- b) makes members active if they had been made inactive by a previous INACTIVE command, and
- c) allows the respecification of the supports with another SUPPORT command which causes the old supports to be ignored. The SUPPORT specification must be such that the number of "releases" before the CHANGE must be greater than or equal to the number of "releases" after the CHANGE. Also, the supports must be specified in the same order before and after the CHANGE command.

**Example**

Before CHANGE

```

1 PINNED
2 FIXED BUT FX MY MZ
3 FIXED BUT FX MX MY MZ
    
```

After CHANGE

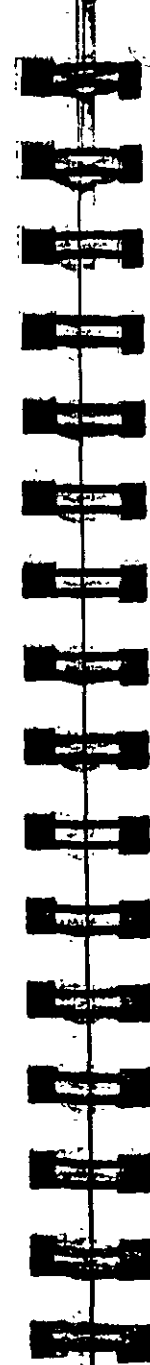
- 1 PINNED
- 2 FIXED
- 3 FIXED BUT FX MZ

The CHANGE command is not necessary when only member properties are revised to perform a new analysis. This is typically the case in which the user has asked for a member selection and then uses the PERFORM ANALYSIS command to reanalyze the structure based on the new member properties.

**Notes**

- 1) If new load cases are specified after the CHANGE command such as in a structure where the INACTIVE MEMBER command is used, the user needs to define the total number of primary load cases using the SET NL option (see Section 5.5 and Example 4).
- 2) Multiple Analyses using the CHANGE command should not be performed if the input file contains load cases involving UBC Analysis, Response Spectrum Analysis, Time History Analysis, Wind Load Generation or Moving Load Generation.

Section



### 5.39 Load List Specification

**Purpose**

This command allows specification of a set of active load cases. All load cases made active by this command remain active until a new load list is specified.

**General format:**

**LOAD LIST** { load-list }  
                   ALL

**Description**

This command is used to activate the load cases listed in this command and, in a sense, deactivate all other load cases not listed in this command. In other words, the loads listed are used for printing output and in design for performing the specified calculations. Note that, when PERFORM ANALYSIS command is used, the program internally uses all load cases, regardless of LOAD LIST command, except after a CHANGE or RESTORE command. In these two cases, the LOAD LIST command allows the program to perform analysis only on those loads in the list. If the LOAD LIST command is never used, the program will assume all load cases to be active.

**Example**

```
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 3
PRINT SUPPORT REACTIONS
CHECK CODE ALL
```

In this example, member forces will be printed for all load cases, whereas loading 1 and 3 will be used for printing support reactions and code-checking of all members.

**Notes**

The LOAD LIST command may be used for multiple analyses situations when an analysis needs to be performed with a selected set of load cases only. Please note that all load cases are automatically active before a CHANGE or RESTORE command is used.



**5.40 Section Specification**

**Purpose**

This command is used to specify sections along the length of frame member for which forces and moments are required.

**General format:**

**SECTION**  $f_1, f_2 \dots f_5$   $\left\{ \begin{array}{l} \text{MEMBER memb-list} \\ \text{( ALL )} \end{array} \right\}$

**Description**

This command specifies the sections, in terms of fractional member lengths, at which the forces and moments are considered for further processing.

$f_1, f_2 \dots f_5$  = Section (in terms of the fraction of the member length) provided for the members. Maximum number of sections is 5, including one at the start and one at the end. In other words, no more than three intermediate sections are permissible per SECTION command.

*See Sections 1.19.2 and 1.19.4*

**Example**

**SECTION 0.0 0.5 1.0 MEMB 1 2**  
**SECTION 0.25 0.75 MEMB 3 TO 7**  
**SECTION 0.6 MEMB 8**

In this example, first the members 1 and 2 are set to section values of 0.0, 0.5, and 1.0, i.e. at the start, mid point and end. The members 3 to 7 are specified by the next SECTION command where sections 0.25 and 0.75 are set.

In the next SECTION command, member 8 has its section specified at 0.6. The remainder of the members will have no sections

provided for them. If no section value is given for any member, it defaults to 0 0 and 1.0 (i.e. start and end). For example, the start and end forces of the members will be used for design, if no section is specified. As mentioned earlier, no more than three intermediate sections are allowed per SECTION command. However, if more than three intermediate sections are involved, they can be examined by repeating the SECTION command after completing the required calculations. The following example will clarify.

**Example**

```
SECTION 0.2 0.4 0.5 ALL
PRINT SECTION FORCES
SECTION 0.6 0.75 0.9 ALL
PRINT SECTION FORCES
```

In this example, first forces at 3 intermediate sections (namely 0.2, 0.4 and 0.5) are printed and then forces at an additional 3 sections (namely 0.6, 0.75 and 0.9) are printed. This gives the user the section forces at more than three intermediate sections.

**Notes**

- 1) The SECTION command just specifies the sections. Use the PRINT SECTION FORCES command after this command to print out the forces and moments at the specified sections.
- 2) This is a secondary analysis command. Note that the analysis must be performed before this command may be used.



## 5.41 Print Specifications

**Purpose**

This command is used to direct the program to print various modeling information and analysis results. STAAD offers a number of versatile print commands that can be used to customize the output.

General format for data related print commands:

```
PRINT {
    JOINT COORDINATES
    MEMBER INFORMATION
    ELEMENT INFORMATION SOLID
    MEMBER PROPERTIES
    MATERIAL PROPERTIES
    SUPPORT INFORMATION
    or
    ALL
} { (ALL)
    LIST list of items
    i.e. joints,
    members }
```

General format to print location of cg:

**PRINT CG**

General format to print analysis results:

```
PRINT {
    (JOINT) DISPLACEMENTS
    (MEMBER) FORCES
    ANALYSIS RESULTS
    (MEMBER) SECTION FORCES
    (MEMBER) STRESSES
    ELEMENT (JOINT) STRESSES (AT 1, 2)
    ELEMENT (JOINT) STRESSES SOLID
    MODE SHAPES
} List-spec
```

```
List-spec = { (ALL)
    LIST list of items-joints,
    members or elements }
```

General format to print support reactions:

### PRINT SUPPORT REACTIONS

General format to print entire steel table:

### PRINT ENTIRE (TABLE)

#### Description

Note that the list of items are not applicable for PRINT ANALYSIS RESULTS, PRINT SUPPORT REACTIONS, and PRINT MODE SHAPES command.

The PRINT JOINT COORDINATES command prints all interpreted coordinates of joints.

The PRINT MEMBER INFORMATION command prints all member information, including member length, member incidences, beta angles, whether or not a member is a truss member and the member release conditions at start and end of the member (1=released, 0 = not released).

The PRINT ELEMENT INFORMATION command prints all incident joints, element thicknesses, and Poisson ratios for Plate/Shell elements. The PRINT ELEMENT INFORMATION SOLID command prints similar information for Solid elements.

The PRINT MEMBER PROPERTIES command prints all member properties including cross sectional area, moments of inertia, and section moduli in both axes. Units for the properties are always INCH or CM (depending on FPS or METRIC) regardless of the unit specified in UNIT command.

The following designation is used for member property names:

AX - Cross section area

AY - Area used to compute shear deformation in local Y-axis  
 AZ - Area used to compute shear deformation in local Z-axis  
 IZ - Moment of Inertia about the local Z-axis  
 IY - Moment of Inertia about the local Y-axis  
 IX - Torsional constant  
 SY - Smallest section modulus about the local Y-axis  
 SZ - Smallest section modulus about the local Z-axis

The PRINT MATERIAL PROPERTIES command prints all material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members. This command is available for members only.

The PRINT SUPPORT INFORMATION command prints all support information regarding their fixity, releases and spring constant values, if any. The LIST option is not available for this command.

The PRINT ALL command is equivalent to last five print commands combined. This command prints joint coordinates, member information, member properties, material properties and support information, in that order.

The PRINT CG command prints out the coordinates of the center of gravity of the structure. Only the selfweight of the structure is used to calculate the C.G. User defined joint loads, member loads etc. are not considered in the calculation of C.G.

The PRINT (JOINT) DISPLACEMENTS command prints joint displacements in a tabulated form. The displacements for all six directions will be printed for all specified load cases. The length unit for the displacements is always INCH or CM (depending on FPS or METRIC unit) regardless of the unit specified in UNIT command.

The PRINT (MEMBER) FORCES command prints member forces (i.e. Axial force (AXIAL), Shear force in local Y and Z axes (SHEAR-Y and SHEAR-Z), Torsional Moment (TORSION),

Moments about local Y and Z axes (MOM-Y and MOM-Z) in a tabulated form for the listed members, for all specified load cases.

PRINT ANALYSIS RESULTS command is equivalent to the last three commands combined. With this command, the joint displacements, support reactions and member forces, in that order, are printed.

The PRINT (MEMBER) SECTION FORCES command prints member forces at the intermediate sections specified with a previously input SECTION command. The printing is done in a tabulated form, by member, for all specified load cases.

The PRINT (MEMBER) STRESSES command tabulates member stresses at the start joint, end joint and all specified intermediate sections. These stresses include axial (i.e. axial force over the area), bending-y (i.e. moment-y over section modulus in local y-axis), bending-z (i.e. moment-z over section modulus in local z-axis), shear stresses in both local y and z directions (FY/AY and FZ/AZ) and combined (absolute combination of axial, bending-y and bending-z) stresses.

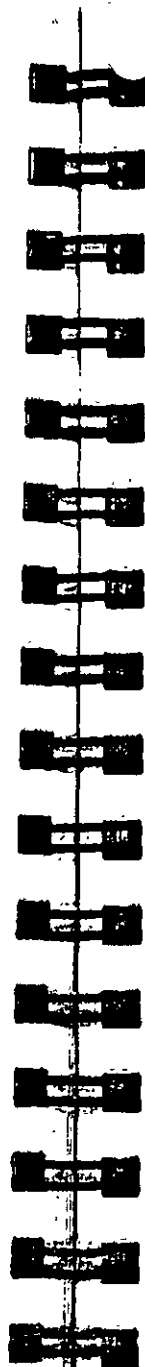
For PRISMATIC sections, if AY and/or AZ is not provided, the full cross-sectional area (AX) will be considered in shear stress calculations.

For TAPERED sections, the values of AY and AZ are those for the location where the stress is printed. Hence if the stress is printed at the location 0.0, the AY at AZ are based on the dimensions of the member at the start node.

AY = Total depth \* Thickness of web.

AZ = 2/3 area of both flanges put together.

The PRINT ELEMENT STRESSES command must be used to print stresses (FX, FY, FXY, QX, QY), moments per unit width (MX, MY, MXY) and principal stresses (SMAX, SMIN, TMAX) for plate/shell elements. Typically, the stresses and moments per unit width at the centroid will be printed. The Von Mises stresses (VONT, VONB) as well as the angle (ANGLE) defining the orientation of the principal planes are also printed.



The variables that appear in the output are the following. See Figure 1.13 in Section 1 of this Reference Manual for more information regarding these variables.

- QX = Shear stress on the local X face in the Z direction
- QY = Shear stress on the local Y face in the Z direction
- MX = Moment per unit width about the local X face
- MY = Moment per unit width about the local Y face
- MXY = Torsional Moment per unit width in the local X-Y plane
- FX = Axial stress in the local X direction
- FY = Axial stress in the local Y direction
- FGY = Shear stress in the local XY plane
- VONT = Von Mises stress on the top surface of the element
- VONB = Von Mises stress on the bottom surface of the element
- SMAX = Maximum in-plane Principal stress
- SMIN = Minimum in-plane Principal stress
- TMAX = Maximum in-plane Shear stress
- ANGLE = Angle which determines direction of maximum principal stress with respect to local X axis

If the JOINT option is used, forces and moments at the nodal points are also printed out in addition to the centroid of the element.

The AT option may be used to print element forces at any specified point within the element. The AT option must be accompanied by  $f_1$  and  $f_2$ . Note that  $f_1$  and  $f_2$  are local X and Y coordinates (in current units) of the point where the stresses and moments are required. For detailed description of the local coordinate system of the elements, refer to Section 1.6 of this manual.

The PRINT ELEMENT (JOINT) STRESS SOLID command enables printing of stresses at the center of the SOLID elements. The variables that appear in the output are the following.

- Normal Stresses : SXX, SYY and SZZ
- Shear Stresses : SXY, SYZ and SZX
- Principal Stresses : S1, S2 and S3.
- Von Mises Stresses : SE

Direction cosines - 6 direction cosines are printed following the expression DC, corresponding to the first two principal stress directions.

The JOINT option will print out the stresses at the nodes of the solid elements.

The PRINT MODE SHAPES command prints joint displacements for all calculated modes

The PRINT SUPPORT REACTIONS command prints support reactions in a tabulated form, by support, for all specified load cases. The LIST option is not available for this command.

The PRINT ENTIRE TABLE command may be used to obtain a print-out of the contents of the steel table from which member properties are being read. This command must be provided following the specification of all member properties.

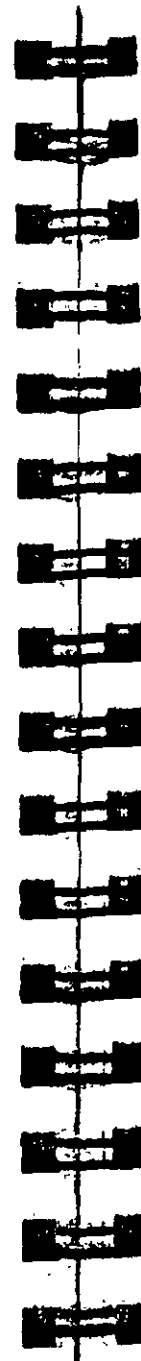
**Example**

```

PERFORM ANALYSIS
PRINT ELEMENT JOINT FORCES
PRINT ELEMENT FORCES AT 0.5 0.5 LIST 1 TO 10
PRINT SUPPORT REACTIONS
PRINT JOINT DISPLACEMENTS LIST 1 TO 50
PRINT MEMBER FORCES LIST 101 TO 124
    
```

**Notes**

- 1) The output generated by these commands are based on the current unit system. The user may wish to verify the current unit system and change it if necessary.
- 2) Results may be printed for all joints/members/elements or based on a specified list.



## 5.42 Print Section Displacement

**Purpose**

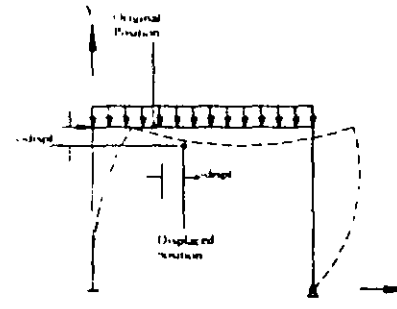
This command is used to calculate and print displacements at sections (intermediate points) of frame members. This provides the user with deflection data between the joints.

**General format:**

```

PRINT SECTION (MAX) DISPL (NSECT I) (SAVE a) {
  NOPRINT
  ALL
  LIST memb-list
}
    
```

**Description**



This command prints displacements at intermediate points between two joints of a member. These displacements are in global coordinate directions (see figure). If the MAX command is used, the program prints only the maximum local displacements among all load cases.

- i = number of sections to be taken. Defaults to 12 if NSECT is not used and also if SAVE is used.
- a = File name where displacement values can be stored and used by STAADPL graphics program. If NOPRINT command is used in conjunction with SAVE command, the program writes the data to file only and does not print them in output. The PC version does not need a filename and if one is provided, it will be ignored.



**Example**

**PRINT SECTION DISPL SAVE**  
**PRINT SECTION MAX DISP**

*See Section 1.19.3*

SECTION DISPLACEMENTS are measured in GLOBAL COORDINATES. The values are measured from the original (undeflected) position to the deflected position. See figure above.

The maximum local displacement is also printed. First, the location is determined and then the value is measured from this location to the line joining start and end joints of the deflected member.

**Notes**

- 1) The section displacement values are available in Global Coordinates. The undeflected position is used as the datum for calculating the deflections.
- 2) This is a secondary analysis command. An analysis must be performed before this command may be used.



## 5.43 Print Force Envelope Specification

**Purpose**

This command is used to calculate and print force/moment envelopes for frame members. This command is not available for finite elements.

**General format:**

**PRINT** { **FORCE** } **ENVELOPE (NSECTION i) list-spec.**  
           { **MAXFORCE** }

list-spec = { **LIST** }  
                   { **(ALL)** }

**Description**

Where i is the number of equally spaced sections to be considered in printing maximum and minimum force envelopes. If the NSECTION i command is omitted, i will default to 12. MAXFORCE command produces maximum/minimum force values only of all sections, whereas the FORCE command prints maximum/minimum force values at every section as well as the max/min force values of all sections. The force components include FY, MZ, FZ, and MY. Note that the SECTION command (as described in section 5.40) does not define the number of sections for force envelopes. For the sign convention of force values, refer to Section 2.19.

*See Section 1.19.5*

**Example**

```
PRINT FORCE ENV  
PRINT MAXF ENV NS 15  
PRINT FORCE ENV NS 4 LIST 3 TO 15
```

**Notes**

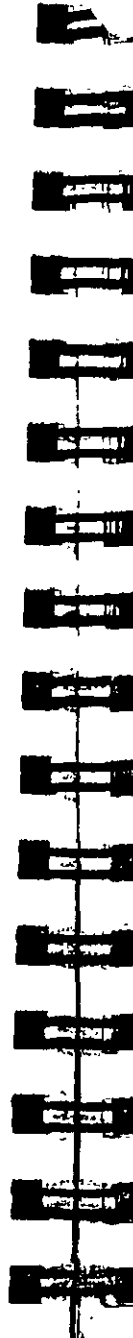
This is a secondary analysis command and should be used after analysis specification.

**5.44 Post Analysis Printer Plot Specifications**

Printer plots of the analysis results may be generated after the analysis has been performed. Refer to Section 5.29 for the various commands that can be provided in the input file after the PERFORM ANALYSIS command.

## 5.45 Post Analysis Graphics Display

This part describes input style to create plot files which can be accessed by STAADPL for graphics display on screen or pen plotters.



### 5.45.1 Display Deflected Shapes

#### Purpose

This command creates a file with data containing the joint displacements. This file must be created in the STAAD analysis/design run so that the deflected shape may be viewed in the Post Processing graphics display in a later session.

#### General format:

**PLOT DISPLACEMENT FILE**

#### Description

This command creates the data required to display the deflected shape for any load case at any associated scale factor. The deflected shapes are displayed by drawing the displaced positions of joints and connecting them with straight lines.

#### Notes

- 1) It is not necessary to have this command in the STAAD analysis run if the deflected shape is viewed in the same session. STAAD's live "concurrent database" stores the displacement data from the latest analysis run which is automatically utilized by the Post Processing module.
- 2) This is a post-analysis facility. Thus this command must be used after the analysis specification.

## 5.45.2 Display Section Displacements

### Purpose

This command creates a file with data containing the section displacements. This file must be created in the STAAD analysis/design run so that the section displacements may be viewed in the Post Processing graphics display in a later session.

### General Format:

**PLOT SECTION FILE**

### Description

Using the Section Displacement command, the user can display the section displacement for any load case with any associated scale factor.

### Notes

- 1) It is not necessary to have this command in the STAAD analysis run if the section displacement is viewed in the same session. STAAD's live "concurrent database" stores the displacement data from the latest analysis run which is automatically utilized by the Post Processing module.
- 2) This is a post-analysis facility. Thus this command must be used after the analysis specification.



## 5.45.3 Display Bending Moment/Shear Force Diagrams

### Purpose

This command creates a file with data containing the bending moment and shear force values. This file must be created in the STAAD analysis run so that the bending moment and shear force diagrams may be viewed in the Post Processing graphics display in a later session.

### General Format:

**PLOT BENDING FILE**

### Description

Using this command, the user can create necessary data to display the moment and shear force diagrams for either one/two members or all members at a time.

### Notes

- 1) It is not necessary to have this command in the STAAD analysis run if the diagrams are viewed in the same session. STAAD's live "concurrent database" stores the necessary data from the latest analysis run which is automatically utilized by the Post Processing module.
- 2) This is a post-analysis facility. Thus this command must be used after the analysis specification.

### 5.45.4 Display Mode Shapes

**Purpose**

This command creates a file with data containing the mode shapes. This file must be created in the STAAD analysis/design run so that the mode shapes may be viewed in the Post Processing graphics display in a later session.

**General Format:**

PLOT MODE FILE

**Description**

Using this command the user can create the data required to display the mode shape for any number of modes with a corresponding scale factor.

**Notes**

- 1) It is not necessary to have this command in the STAAD analysis run if the mode shapes are viewed in the same session. STAAD's live "concurrent database" stores the displacement data from the latest analysis run which is automatically utilized by the Post Processing module.
- 2) This is a post-analysis facility. Thus this command must be used after the analysis specification.



### 5.45.5 Display Stress Contours

**Purpose**

This command creates a file with data containing the stress values. This file must be created in the STAAD analysis/design run so that the stress contours may be viewed in the Post Processing graphics display in a later session.

**General Format:**

PLOT STRESS FILE

**Description**

Using this command the user can display the stress contours for any load case with a corresponding scale factor.

**Notes**

- 1) It is not necessary to have this command in the STAAD analysis run if the stress contours are viewed in the same session. STAAD's live "concurrent database" stores the stress data from the latest analysis run which is automatically utilized by the Post Processing module.
- 2) This is a post-analysis facility. Thus this command must be used after the analysis specification.

## 5.46 Size Specification

### Purpose

This command provides an estimate for required section properties for a frame member based on certain analysis results and user requirements.

### General Format:

```

SIZE { WIDTH      f1 } { MEMBER member-list }
      { DEFLECTION f2 }
      { LENGTH     f3 }
      { BSTRESS    f4 }
      { SSTRESS    f5 }
      { ALL        }
    
```

where,

$f_1$  = Maxm. allowable width

$f_2$  = Maxm. allowable (Length/Maxm. local deflection) ratio

$f_3$  = Length for calculating the above ratio.

Default = actual member length.

$f_4$  = Maxm. allowable bending stress.

$f_5$  = Maxm. allowable shear stress

Note that the values must be provided in the current unit system

### Description

This command may be used to calculate required section properties for a member based on analysis results and user specified criterias. The user specified criterias may include Member width, Allowable (Length/Maxm. Deflection) Ratio, Maxm. allowable bending stress and Maximum allowable shear stress. Any number of these criterias may be used simultaneously. The output includes required Section Modulus (about major axis), required Shear Area (for shear parallel to minor axis), Maxm. moment capacity (about major axis), Maxm. shear capacity (for shear parallel to minor axis) and Maxm. (Length/local maxm. deflection) ratio.



### Example

```

SIZE WID 12 DEFL 300 LEN 240 BSTR 36 ALL
SIZE DEFL 450 BSTR 42 MEMB 16 TO 25
    
```

### Note:

It may be noted that sizing will be based on only the criteria specified by the user in the relevant SIZE command.

In the first example above, sizing will be based on user specified member width of 12, Length/Deflection ratio of 300 (where Length= 240) and max. allowable bending stress of 36.

In the second example, sizing will be based on Length/Deflection ratio of 450 (where Length= actual member length) and max. allowable bending stress of 42.

### Notes

This is a post-analysis facility and must be used after the analysis specifications.

## 5.47 Steel Design Specifications

This section describes all the specifications necessary for structural steel design.

Section 5.47.1 discusses specification of all the parameters that may be used to control the design. Sections 5.47.2 and 5.47.3 describes the CODE CHECKING and MEMBER SELECTION options respectively. Member Selection by optimization is discussed in 5.47.4. STAAD also provides facilities for Weld Design which are described in 5.47.5.



### 5.47.1 Parameter Specifications

**Purpose**

This set of commands may be used to specify the parameters required for steel design

**General format:**

PARAMETER

CODE

- AASHTO
- AISC
- AUSTRALIAN
- BRITISH
- CANADIAN
- FRENCH
- GERMAN
- INDIA
- JAPAN
- LRFD
- NORWAY

{ parameter-name  $f_1$  }  
 { PROFILE a<sub>1</sub>, (a<sub>2</sub>, a<sub>3</sub>) }

{ MEMBER memb-list }  
 { ALL }

**Description**

Parameter-name refers to the "PARAMETER NAME" (s) listed in the parameter table contained in the Steel Design section.

For a list of parameters for AISC Allowable Stress Design, see Table 3.1 of this manual. For a list of parameters for AISC LRFD Design, see Table 3.2. For steel design per other codes, refer to the relevant documentation.

$f_1$  = Value of the parameter.

The user can control the design through specification of proper parameter

The PROFILE parameter is available for the AISC ASD code only. The user can specify up to three profiles ( $a_1$ ,  $a_2$  and  $a_3$ ). Profile, as described in Table 3.1, is the first three letters of an AISC steel table name, e.g. W8X, W12, C10, L20 etc. The PROFILE parameter-name is used only for member selection where members are selected from each of those profile names. The PROFILE for a T-section is a W-shape. Also, the shape specified under PROFILE has to be the same as that specified initially under MEMBER PROPERTIES. Note that the PROFILE command can only be used for the AMERICAN steel table. CODE parameter lets you choose the type of steel code to be checked for design. The default steel code depends on the country of distribution.

**Example**

```
PARAMETERS
CODE AISC
KY 1.5 MEMB 3 7 TO 11
NSF 0.75 ALL
PROFILE W12 W14 MEMB 1 2 23
RATIO 0.9 ALL
```

**Notes**

- 1) All unit sensitive values should be in the current unit system.
- 2) For default values of the parameters, refer to the appropriate parameter table.
- 3) PROFILE command is available with the American AISC ASD code only. It is not available with the LRFD or AASHTO codes.



### 5.47.2 Code Checking Specification

**Purpose**

This command may be used to perform the CODE CHECKING operation.

**General format:**

```
CHECK CODE { MEMBER memb-list }
                ALL
```

**Description**

This command checks the specified members against the specification of the desired code. Refer to Section 2 of this manual for detailed information.

See Section 2.5

**Notes**

The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.



### 5.47.3 Member Selection Specification

**Purpose**

This command may be used to perform the MEMBER SELECTION operation.

**General format:**

```

SELECT { MEMBER memb-list }
      { ALL }
    
```

**Description**

This command instructs STAAD to select specified members based on the parameter value restrictions and specified code. The selection is done using the results from the most recent analysis and iterating on sections until a least weight size is obtained. Refer to Section 2 for more details.

**Notes**

- 1) The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.
- 2) Member selection can be done only after an analysis has been performed. Consequently, the command to perform the analysis has to be specified before the SELECT MEMBER command can be specified.



### 5.47.4 Member Selection by Optimization

**Purpose**

This command performs member selection using an optimized technique based on multiple analysis/design iterations.

**General format:**

```

SELECT OPTIMIZED
    
```

**Description**

The program selects all members based on a state-of-the-art optimization technique. This method requires multiple analyses as well as iteration of sizes until an overall structure least weight is obtained. This command should be used with caution since it will require longer processing time.

**Notes**

- 1) The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.
- 2) This command may require multiple iterations involving analysis/design cycles and therefore may be time consuming.

## 5.47.5 Weld Selection Specification

### Purpose

This command performs selection of weld sizes for specified members.

### General format:

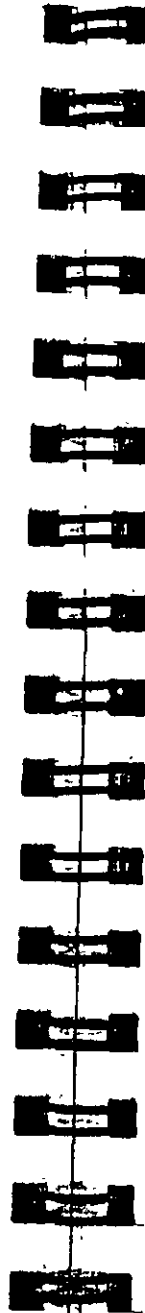
```
SELECT WELD (TRUSS) { MEMBER memb-list
                     ALL }
```

### Description

By this command, the program selects the weld sizes of the specified members at start and end. The selections are tabulated with all the necessary information. If the TRUSS command is used, the program will design welds for angle and double angle members attached to gusset plates with the weld along the length of members.

### Notes

The weld size calculation may also be controlled parametrically. For information on available parameters, refer to the appropriate Steel Design section.



## 5.48 Group Specification

### Purpose

This command may be used to group members together for analysis and steel design.

### General format:

```
(FIXED GROUP)
GROUP prop-spec MEMB memb-list (SAME AS i1)
```

prop-spec =  $\begin{cases} AX \\ SY \\ SZ \end{cases}$  = Cross-section area  
 = Section modulus in local y-axis  
 = Section modulus in local z-axis

### Description

This command enables the program to group specified members together for analysis based on their largest property specification. If member number  $i_1$  is provided in SAME AS command, the program will group the members based on the properties of  $i_1$ . This command is usually used after the member selection, where the members can be grouped for further processing. If the FIXED GROUP option is used, the specified grouping will be retained in memory by the program and will be used in subsequent member select operations and the results of the "GROUPing" will not be seen unless a "SELECT MEMBER" operation is performed.

### Example 1

```
GROUP SZ MEMB 1 3 7 TO 12 15
GROUP MEMB 17 TO 23 27 SAME AS 30
```

In this example, the members 1, 3, 7 to 12, and 15 are assigned the same properties based on which of these members has the largest section modulus. Members 17 to 23 and 27 are assigned the same properties as member 30, regardless of whether member 30 has a

smaller or larger cross-sectional area. AX is the default property upon which grouping is based.

### Example 2

```
FIXED GROUP
GROUP MEMB 1 TO 5
SELECT OPTIMIZED
```

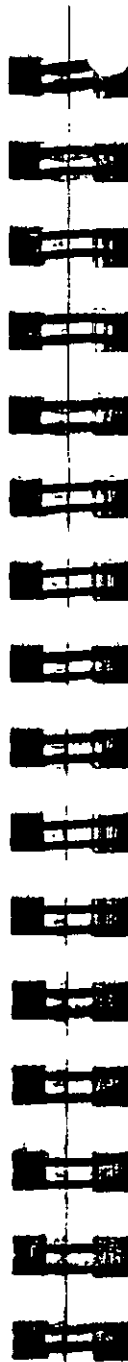
In the above example, the usage of the FIXED GROUP command is illustrated. The SELECT OPTIMIZED command involves the 3 stage process of

- 1) MEMBER SELECTION
- 2) GROUPING of members 1 TO 5
- 3) ANALYSIS

a multiple number of times until a convergence in member properties is attained between adjacent cycles. The FIXED GROUP command is required for execution of step 2 in the cycle

### Notes

This command is typically used after the member selection for further analysis and design. This facility may be very effectively utilized to develop a practically oriented design where several members need to be of the same size



## 5.49 Steel Take Off Specification

### Purpose

This command may be used to obtain a summary of all steel sections being used along with their lengths and weights.

### General format:

**STEEL (MEMBER) TAKE ( OFF )**

### Description

This command provides a complete listing of all steel table sections used in the structure. The tabulated listing will include total length of each section name and its total weight. This can be helpful in estimating steel quantities.

The MEMBER option list each member length, weight, section number, profile-type, length and weight.

### Notes

This facility may be very effectively used to obtain a quick estimate of the structural steel quantity.

## 5.0 Timber Design Specifications

This section describes the specifications required for timber design. Detailed description of the timber design procedures is available in Section 4.

Section 4.50.1 describes specification of parameters for timber design. Sections 4.50.2 and 4.50.3 discuss the code checking and member selection facilities respectively.



### 5.50.1 Timber Design Parameter Specifications

**Purpose**

This set of commands may be used for specification of parameters for timber design.

**General Format:**

PARAMETER

CODE TIMBER

parameter-name  $f_1$  { MEMBER member-list  
ALL }

**Description**

$f_1$  = the value of the parameter.

Note that the parameter-name refers to the parameters described in Section 4.

**Notes**

- 1) All values must be provided in the current unit system.
- 2) For default values of parameters, refer to Section 4.

## 5.50.2 Code Checking Specification

### Purpose

This command performs code checking operation on specified members based on the American Institute of Timber Construction (AITC) codes.

### General Format:

```
CHECK CODE { MEMBER member-list }
                ALL
```

### Description

This command checks the specified members against the requirements of the American Institute of Timber Construction (AITC) codes. The results of the code checking are summarized in a tabular format. Examples and detailed explanations of the tabular format are available in Section 5.

### Notes

The output of this command may be controlled by the TRACK parameter. Two levels of details are available. Refer to Section 5 for detailed information on the TRACK parameter.

## 5.50.3 Member Selection Specification

### Purpose

This command performs member selection operation on specified members based on the American Institute of Timber Construction (AITC) codes.

### General Format:

```
SELECT { MEMBER member-list }
          ALL
```

### Description

This command may be used to perform member selection according to the AITC codes. The selection is based on the results of the latest analysis and iterations are performed until the least weight member satisfying all the applicable code requirements is obtained. Parameters may be used to control the design and the results are available in a tabular format. Detailed explanations of the selection process and the output are available in Section 5.

### Notes

The output of this command may be controlled by the TRACK parameter. Two levels of details are available. Refer to Section 5 for detailed information on the TRACK parameter.

# 1 Concrete Design Specifications

This section describes the specifications for concrete design. The concrete design procedure implemented in STAAD consists of the following steps

- 1) Initiating the design
- 2) Specifying parameters.
- 3) Specifying design requirements
- 4) Requesting quantity take-off.
- 5) Terminating the design

Section 5.51.1 describes the design initiation command. Section 5.51.2 discusses the specification of parameters. Design requirement specifications are described 5.51.3. The CONCRETE TAKE OFF command is described in 5.51.4. Finally, the design termination command is described in 5.51.5.



## 5.51.1 Design Initiation

### Purpose

This command is used to initiate the concrete design.

### General format:

**START CONCRETE DESIGN**

### Description

This command initiates the concrete design specification. With this, the design parameters are automatically set to the default values (as shown on Table 4.1) Without this command, none of the following concrete design commands will be recognized.

### Notes

This command must be present before any concrete design command is used.

### 5.51.2 Concrete Design-Parameter Specification

**Purpose**

This set of commands may be used to specify parameters to control the concrete design.

**General format:**

```

CODE      {
  ACI
  BRITISH
  CANADIAN
  FRENCH
  GERMAN
  INDIA
  JAPAN
  NORWAY
}

parameter-name f1 {
  MEMBER memb/elem list
  ( ALL )
}
    
```

**Description**

Parameter-name refers to the concrete parameters described in Table 4.1

f<sub>1</sub> = is the value of the parameter. Note that this value is always input in current units. The UNIT command is also accepted during any phase of concrete design.

**Notes**

- 1) All parameter values are provided in the current unit system.
- 2) For default values of parameters, refer to Section 3.



### 5.51.3 Concrete Design Command

**Purpose**

This command may be used to specify the type of design required. Members may be designed as BEAM, COLUMN or ELEMENT.

**General format:**

```

DESIGN    {
  BEAM
  COLUMN
  ELEMENT
} memb-list
    
```

**Description**

Members to be designed must be specified as BEAM, COLUMN or ELEMENT. Note that members, once designed as beam, cannot be redesigned as a column again, or vice versa.

**Notes**

Only plate elements may be designed as ELEMENT.

### 5.1.4 Concrete Take Off Command

**Purpose**

This command may be used to obtain an estimate of the total volume of the concrete, reinforcement bars used and their respective weights

**General Format:**

CONCRETE TAKE OFF

**Description**

This command can be issued to print the total volume of concrete and the bar numbers and their respective weight for the members designed.

**SAMPLE OUTPUT:**

\*\*\*\*\* CONCRETE TAKE OFF \*\*\*\*\*  
(FOR BEAMS AND COLUMNS DESIGNED ABOVE)

TOTAL VOLUME OF CONCRETE = 87.50 CU FT

BAR SIZE NUMBER	WEIGHT (in lbs)
4	805.03
6	91.60
8	1137.60
9	653.84
11	818.67
*** TOTAL	= 3506.74

**Notes**

This command may be used very effectively for quick quantity estimates.



### 5.51.5 Concrete Design Terminator

**Purpose**

This command must be used to terminate the concrete design.

**General format:**

END CONCRETE DESIGN

**Description**

This command terminates the concrete design, after which normal STAAD commands resume.

**Example**

```
START CONCRETE DESIGN
CODE ACI
FYMAIN 40.0 ALL
FC 3.0 ALL
DESIGN BEAM 1 TO 4 7
DESIGN COLUMN 9 12 TO 16
DESIGN ELEMENT 20 TO 30
END
```

**Notes**

Without this command, further STAAD commands will not be recognized.



## 5.52 Footing Design Specifications

### Purpose

This set of commands may be used to specify footing design requirements. Sections 5.52.1 through 5.52.4 describe the process of design initiation, parameter specification, design command and design termination.

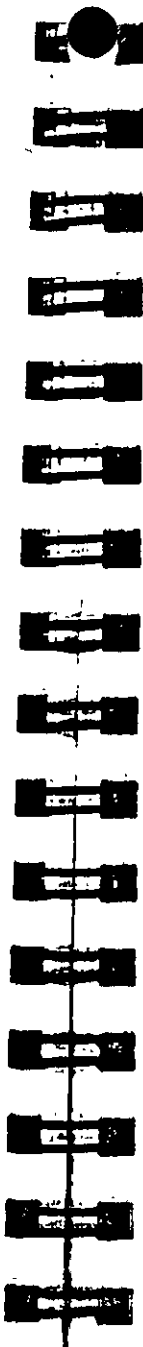
### Description

This facility may be used to design isolated footings for user specified support joints. Once the support is specified, the program automatically identifies the support reaction(s) associated with the joint. All active load cases are checked and design is performed for the support reaction(s) that requires the maximum footing size. Parameters are available to control the design. Dowel bars and development lengths are also calculated and included in the design output.

### Design Considerations

The STAAD isolated footing design is based on the following considerations.

- 1) The design reaction load may include concentrated load and biaxial moments.
- 2) The vertical reaction load is increased by 10% to account for the selfweight of the footing.
- 3) Footing slab size is rectangular. The ratio between the length and the width of the slab may be controlled by the user through a parameter.
- 4) Optional pedestal design is available.
- 5) Footings cannot be designed at supports where the reaction causes an uplift on the footings.



### Design Procedure

The following sequential design procedure is followed:

- 1) Footing size is calculated on the basis of the load directly available from the analysis results (support reactions) and user specified Allowable Soil Pressure. No factor is used on the support reactions.
- 2) The footing size obtained from 1) and the FACTORED LOAD is utilized to calculate soil reactions.

$$\text{FACTORED LOAD} = \text{ACTUAL REACTION} \times \text{Parameter FFAC}$$

Note that the user may provide a desired value for parameter FFAC.

- 3) Footing depth and reinforcement details are based on soil reactions calculated per 2) above
- 4) Dowel bar requirements and development lengths are calculated and reported in the output.

Following parameters are available for footing design.

### Design Parameters

Parameter Name	Default Value	Description
FY	60,000 psi	Yield strength for reinforcement steel.
FC	3,000 psi	Compressive Strength of Concrete
CLEAR	3 0 in.	Clear cover for slab reinforcement.
REINF	Number 9 bar	Main reinforcement bar size for slab design.
FFAC	1.0	Load factor for concrete design.
BC	3000 psf	Soil bearing capacity.
RATIO	1 0	Ratio between slab sides.

**Design Parameters Cont.**

Parameter Name	Default Value	Description
<u>TRACK</u>	1.0	1 0 = only numerical output is provided 2 0 = numerical output and sketch provided
<u>DEPTH</u>	Calculated by the program	The min depth of the footing base slab. Program changes this value if required for design.
<u>S1, S2</u>	Calculated by the program	Size of the footing base slab - S1 and S2 correspond to column sides YD and ZD respectively. Either S1 or S2 or both can be specified. If one is provided, the other will be calculated based on RATIO. If both are provided, RATIO will be ignored.
<u>EMBEDMENT</u>	0.0	The depth of the footing base from the support point of the column.
<u>PEDESTAL</u>	0.0	0 0 = no pedestal design 1 0 = pedestal design with program calculating pedestal dimensions. X1 X2 = pedestal design with user provided pedestal dimensions. X1 and X2 are pedestal dimensions corresponding to slab sides S1 and S2 respectively.



**5.52.1 Design Initiation**

**Purpose**

This command must be used to initiate the footing design.

**General Format:**

**START FOOTING DESIGN**

**Description**

This command initiates the footing design specifications. Without this command, no further footing design command will be recognized.

**Notes**

No footing design specification will be processed without this command.

### 5.52.2 Footing Design Parameter Specification

**Purpose**

This command is used to specify parameters that may be used to control the footing design.

**General Format:**

<u>CODE</u>	AMERICAN BRITISH CANADIAN FRENCH GERMAN INDIA JAPAN NORWAY
parameter-name $f_1$	{ <u>JOINT joint-list</u> ( <u>ALL</u> )             }

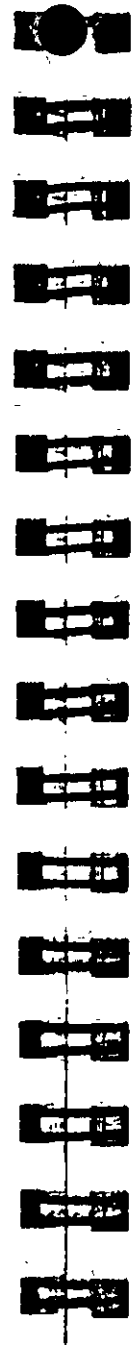
**Description**

Parameter-name refers to the parameters described in Section 5.52

$f_1$  is the value of the parameter. Note that this value should be in the current units. The UNIT command is also accepted during any phase of footing design.

**Notes**

- 1) All parameter values must be provided in the current unit system.
- 2) For default values of parameters, refer to the parameter table in Section 5.52.



### 5.52.3 Footing Design Command

**Purpose**

This command must be used to execute the footing design.

**General Format:**

DESIGN FOOTING joint-list

**Description**

This command may be used to specify the joints for which the footing designs are required.

**Notes**

The output of this command may be controlled by the TRACK parameter (see Section 5.52). If TRACK is set to the default value of 1.0, only numerical output will be provided. If TRACK is set to 2.0, graphical output will be provided in addition.

**EXAMPLE**

```

START FOOTING DESIGN
CODE AMERICAN
UNIT KIP INCH
FY 45.0 JOINT 2
FY 60.0 JOINT 5
FC 3 ALL
RATIO 0.8 ALL
TRACK 2.0 ALL
PEDESTAL 1.0 ALL
UNIT KIP FEET
CLEAR 0.25
BC 5.20 JOINT 2
BC 5.00 JOINT 5
DESIGN FOOTING 1 2 3 5
END FOOTING DESIGN
    
```



### 5.52.4 Footing Design Terminator

**Purpose**

This command must be used to terminate the footing design.

**General Format:**

**END FOOTING DESIGN**

**Description**

This command terminates the footing design.

**Notes**

If the footing design is not terminated, no further STAAD command will be recognized.

## 5.53 Miscellaneous Special Commands

Following are two special commands that may be used to control the precision characteristics of the stiffness matrix elements and memory requirements for a particular problem.



### 5.53.1 Precision Command

#### Purpose

This command may be used to control the precision checking characteristics of the structural stiffness matrix.

#### Description

The global stiffness matrix of a correctly modeled structure is always a symmetric matrix with positive elements on the main diagonal. Negative or very small positive elements are usually a result of incorrect modeling resulting in an ill-conditioned stiffness matrix. In STAAD, all stiffness coefficients numerically less than 0.01 are identified as possible sources of instability and a warning message is issued cautioning the user. In certain special situations, these stiffness coefficients may represent accurate numbers. To eliminate the warning message in these circumstances, the PRECISION command may be used to reset the value against which the coefficients are checked. This command should be positioned near the beginning of the input file.

#### General format:

**PRECISION**  $f_1$

where,  $f_1$  = number against which stiffness coefficients are checked.  
(default = 0.01)

#### Notes

This command should be specified before the JOINT COORDINATES specifications.

## 5.53.2 Input Memory Command

### Purpose

This command may be used to control the memory requirement in the case of a CORE OVERFLOW problem.

### Description

This command may be used in a CORE OVERFLOW situation to decrease memory demand by the problem. A default value of 16000 is preset in the program. A number lower than this may decrease memory requirements. Problems requiring more memory possibly can be solved using this command. Use of this command will result in slower execution. Thus, caution should be exercised in using this command. Increasing the number may result in faster execution if resources are available. This command should be placed near the beginning of input.

### General Format:

**INPUT MEMORY  $f_1$**

where  $f_1$  = number representing memory available  
(default=16000 for PC version)

### Notes

- 1) This command should be specified before the JOINT COORDINATES specifications.
- 2) The value of  $f_1$  may be specified as zero. The command INPUT MEMORY 0 will result in minimum possible core memory demand by the problem. However, the user is cautioned in its use as this will reduce the speed of execution. Thus, one should consider using this command only when a CORE OVERFLOW message has appeared and all other means of reducing the core memory demand have been exhausted.

## 5.54 Save Specification

### Purpose

This command may be used to save run data and analytical results for further processing in a later run.

### General format:

**SAVE ( $a_1, a_2$ )**

### Description

- $a_1$  = Name of the file in which the analytical results and data will be saved. Any name up to twelve characters, starting with an alphabetic character, is accepted.
- $a_2$  = Name of a second file in which the load data will be saved. Any name, different from the first up to twelve characters, starting with an alphabetic character, is accepted.

### Notes

If the file names are not provided, data will be saved in files called TEM1 and TEM2 respectively.

## 5.55 Restore Specification

### Purpose

This command may be used to restore the run data saved by the SAVE command.

### General format:

**RESTORE** ( $a_1, a_2$ )

### Description

$a_1$  and  $a_2$  are the same filenames as specified in the SAVE specification. This command is used to restore a previous STAAD run which was saved with the SAVE command. If the names  $a_1$  and  $a_2$  are left out, the program will assume the names TEM1 and TEM2 respectively. So a run saved with no filename can be restored in the same manner by providing no filenames with the RESTORE command.

### Example

```
STAAD SPACE TITLE
RESTORE FILE1 FILE2
```

### Notes

- 1) This command will also recognize the default file names TEM1 and TEM2.
- 2) Operations involving finite elements cannot be performed with the restored file.

## 5.56 End Run Specification

### Purpose

This command must be used to terminate the STAAD run.

### General format:

**FINISH**

### Description

This command should be provided as the last input command. This terminates a STAAD run.



## STARDYNE Advanced Analysis

### Section 6

#### 6.1 Introduction

STAAD/Pro offers two analysis engines – STAAD and STARDYNE. The STAAD analysis engine has been described in section 1 of this manual. The STARDYNE analysis engine is discussed in this section.

STARDYNE was the first commercially available finite element analysis software for dynamic analysis and has been in use throughout the world for over 30 years. STARDYNE offers the structural analyst a reliable and easy to use method to solve any problem in static and dynamic analysis.

This section of the manual contains a general description of the analysis capabilities available in STARDYNE. Detailed command formats and other specific user information is the same as in STAAD engine (see Section 5). In addition, several analysis-related options may be specified just before the Analysis is performed. For explanation of these options, please refer to Section 3 of the *STAAD Graphical Environment* manual.

The objective of this section is to familiarize the user with the basic principles involved in the implementation of the various analysis capabilities.



## 6.2 Input Generation – STARDYNE

Input is generated through the STAAD/Pro model generation facilities. Please refer to the *STAAD Graphical Environment* manual for detailed description.

## 6.3 Types of Structures – STARDYNE

A STRUCTURE can be defined as an assemblage of elements. STARDYNE is capable of analyzing structures consisting of beam elements, plate/shell elements, solid elements, and/or matrix elements. Almost any type of structure can be analyzed by STARDYNE.

## 6.4 Unit Systems – STARDYNE

The STARDYNE analysis may be conducted in any consistent uniform unit system. STARDYNE intermediate output files, if inspected, will be in a consistent unit system chosen in any consistent uniform unit system. STAAD/Pro provides a seamless translation to and from the analysis modules so that the user need not be concerned with the unit system used in by the translator

## 6.5 Structure Geometry and Coordinate Systems–STARDYNE

A structure is an assembly of individual physical components such as beams, columns, slabs, plates etc.. In STARDYNE, the finite element model will consist of those physical components being subdivided into one or more finite elements per component. Typically, modeling of the structure geometry consists of two steps:

- A. Identification and description of nodes.
- B. Modeling of elements through specification of connectivity (incidences) between nodes.

Note that subdividing individual physical components, such as an I-beam, into several beam finite elements may improve the dynamic response and sometimes improve static response as well. So the user should not think that there must be a one for correspondence between physical components and the finite elements.

STARDYNE uses two types of coordinate systems to define the structure geometry and loading patterns. The GLOBAL coordinate system is an arbitrary coordinate system in space which is utilized to specify the overall geometry & loading pattern of the structure. A LOCAL coordinate system is associated with each element and is utilized in element output or local load specification.

### 6.5.1 Global Coordinate System

The Conventional Cartesian Coordinate System is used for specification of the structure geometry. This coordinate system is a rectangular coordinate system (X, Y, Z) which follows the orthogonal right hand rule. This coordinate system may be used to define the node locations and loading directions. The translational degrees of freedom are denoted by  $u_1$ ,  $u_2$ ,  $u_3$  and the rotational degrees of freedom are denoted by  $u_4$ ,  $u_5$  &  $u_6$ .

### 6.5.2 Local Element Coordinate System

A local coordinate system is associated with each element. The local  $x$  axis is from the first node to the second node for each element type. A third node defines the plane containing the orthogonal local  $y$  axis and its positive direction. The local  $z$  axis is orthogonal to  $x$  and  $y$  and positive by the right hand rule.

The beta angle method of specifying the beam element local system is defined in the Beam Element topic

## 3 Finite Element Information – STARDYNE

STARDYNE includes beam, plate/shell, spring, and matrix finite elements. The features of each is explained below

### 3.6.1 Plate/Shell Element

The Plate/Shell finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (3-4 node, isoparametric, hybrid, axisymmetric, composite) (quadrilateral). If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangular elements. "Surface structures" such as walls, slabs, plates and shells may be modeled using finite elements. The user may also use the element for PLANE STRESS action only

#### Plate Geometry Modeling Considerations

The following geometry related modeling rules should be kept in mind while using the plate/shell element:

- 1) While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise
- 2) Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1
- 3) Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90 degrees and never larger than 180

#### Plate Element Load Specification

Following load specifications are available:

- 1) Linearly varying pressure on element surface in local directions.



- 2) Temperature load due to uniform increase or decrease of temperature.
- 3) Temperature load due to difference in temperature between top and bottom surfaces of the element.
- 4) Also, Density needs to be provided if selfweight is included in any load case.

#### Triangle Plate Theoretical Basis

Two types of triangular plates are available:

- 1) A linear curvature compatible triangle which simulates thin plate behavior (without consideration of transverse shear effects) for non-sandwich structures;
- 2) the "Martin" element which simulates thin or thick plate behavior in a sandwich or homogeneous (solid) structure. Sandwich Core thickness and G for the transverse shear are additional input options. The "Martin" plate must be used when transverse shear effects and/or transverse shear stresses are desired in a 3 node element. A denser nodal mesh is desirable with this element.

#### Assumptions:

- Linear, elastic, and homogeneous or sandwich.
- Small deformations.
- Constant strain (linear variation of displacement) for in-plane (both TYPES) and "Martin" plate bending element.
- Normal stress ( $\sigma_z$ ) is neglected. (Thin shell theory.)
- No coupling of in-plane and bending action.
- The displacement of a triangular plate element is defined by three translations and two rotations at each corner of the plate. 5 equations (dof) per node.
- Orthotropic properties with axis angle from principal directions to local axes.

#### Quadrilateral Plate Theoretical Basis

Several types of quadrilateral plates are available

- 1) QUADB, QUADS A standard 3-4 node isoparametric inplane element with "bubble" functions.

- 2) QUADA. A standard 3-4 node isoparametric with "bubble" functions or hybrid axisymmetric element. The entire model must consist of QUADA elements if used and the model defined in the X-Y plane with X being radial (positive only) and Y being longitudinal. 2 DOF per node.
- 3) QUADB. A bending plate based on 4 linear curvature compatible triangles around a centroidal node with 11 dof per triangle in bending which simulates thin plate behavior (without consideration of transverse shear effects).
- 4) QUADS. A hybrid method Mindlin thick plate for bending with transverse shear energy included.
- 5) QUADH. A hybrid method thin plate for in-plane and bending.
- 6) QUADC and QUADCM. A hybrid method thick plate for in-plane and bending of composite orthotropic multi-ply elements. QUADCM formulation suppresses the internal kinematic modes of deformations in the element. The Tsai-Wu failure theory criterion can be calculated. Up to 100 layers (plys) may be defined.

**Assumptions:**

- Linear, elastic, orthotropic
- Small deformations.
- Normal stress ( $\sigma_z$ ) is neglected.
- No coupling of in-plane and bending action.
- The displacement of a quadrilateral plate element is defined by three translations and two rotations at each corner of the plate. 5 equations (dof) per node.
- Orthotropic properties with axis angle from principal directions to local axes.

**Plate Element Local Coordinate System**

From node JA to node JB is the local x-axis with origin at JA. The JC node lies in the x-y plane and is on the positive y side of the x axis. The local z-axis is found by the right hand rule from x and y.



**Plane Strain Analysis**

If the user wishes to use STARDYNE in an in-plane only, isotropic, plane strain analysis, the material properties (E, POIS12, ALPHA1) specified for the in-plane triangles and quadrilaterals must be modified as follows:

$$E_i = \frac{E_a}{(1 - PR_a)}$$

$$PR = \frac{PR_a}{(1 - PR_a)}$$

$$ALPHA_i = ALPHA_a * (1 + PR_a)$$

Where:

$E_a$  = Actual Modulus of Elasticity

$PR_a$  = Actual Poisson's Ratio

$ALPHA_a$  = Actual Coefficient of Thermal Expansion.

and

$E_i$ ,  $PR_i$ , and  $ALPHA_i$  are the values to be entered in the material property table for the plane strain analysis.

**Output of Plate Element Forces**

Element outputs are available at the following locations:

- A. Center of the element.
- B. All corner nodes of the element.

Following are the items included in the ELEMENT FORCE output.

QX, QY	Shear stresses (Force/ unit len./unit thk.)
FX, FY, FXY	Membrane stresses (Force/unit len./unit thk)
MX, MY, MXY	Bending moments per unit width (Moment/unit len.)
SMAX, SMIN	Principal stresses (Force/unit area)
TMAX	Maxim. shear stress (Force/unit area)
VONMISES	stress (Force/unit area) =
* SQRT[ 0.5 * [(S1 - S2)**2 + (S2 - S3)**2 + (S3 - S1)**2 ] }	

S1, S2, S3 are principal stresses.  
 ANGLE Orientation of the principal plane (Degrees)

Notes:

- 1 All element force output is in the local coordinate system.
- 2 Principal stresses (SMAX & SMIN), the maximum shear stress (TMAX) and the orientation of the principal plane (ANGLE) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.

**1.2 Solid Element**

Solid elements enable the solution of structural problems involving general three dimensional stresses. There is a class of problems such as stress distribution in concrete dams, soil and rock strata where finite element analysis using solid elements provides a powerful tool.

**Solid Element Theoretical Basis**

The solid element used in STAR DYNE is an eight node isoparametric type or optionally hybrid type. The isoparametric type is the same as in STAAD with the addition of 3 "bubble functions" to improve shear behavior. These elements have three translational degrees-of-freedom per node. By collapsing various nodes together, an eight node solid element can be degenerated to an element with four to seven nodes.

**Solid element Local Coordinate System**

The local coordinate system used in solid element is the same as the global system, however stresses may be presented as if JA-JB-JC nodes defined a local system (see plate local coordinate system).

**Solid element Properties and Constants**

Full 3-D orthotropic properties may be used. Also, Density needs to be provided if selfweight is included in any load case.

**Output of Solid Element Stresses**



Element stresses may be obtained at the center and at the nodes of the solid element. The items that are printed are:

- Normal Stresses: SXX, SYY and SZZ
- Shear Stresses: SXY, SYZ and SZX
- Principal stresses: S1, S2 and S3.
- Von Mises stresses: SE
- Direction cosines: 6 direction cosines are printed, following the expression DC, corresponding to the first two principal stress directions.

**Hybrid Element Formulation, Plate or Solid (STAR DYNE)**

In the classical displacement formulation, simple polynomials are used to interpolate nodal variables internal to the element. For example, a unit nodal displacement in the x direction, induces displacements ( $\leq 1.0$ ) in the x direction inside of the element. From these interpolated displacements the strains can be found. Then, using the material constants, the element nodal stiffness coefficients and stress matrices can be computed.

In the hybrid formulation two sets of interpolation functions are used. The first to interpolate displacements along the element boundary and the second to interpolate stress fields inside of the element. Consequently, there is a rather complex relationship between the nodal displacements and stresses inside of the element. STAR DYNE offers a choice between the classical displacement and the hybrid elements. The QUADH header selects the hybrid quads and CUBEH selects the hybrid cubes.

Hybrid elements, in general, yield slightly better accuracy if they are only slightly distorted from rectangular or parallelogram shapes. If there are areas in the model where severely distorted elements must be used, select the QUADB and CUBEG types.

The classical displacement elements converge to the correct answers "from below". This means that as the finite element mesh gets denser the model gets more flexible and approaches to the correct answers (assuming that loads and boundary conditions are

unchanged). The hybrid elements, on the other hand, can converge either from above or below. All of the finite elements in STARDYNE are convergent.

### 3 Matrix (General Spring) Element

Matrix elements are direct additions to the Global Stiffness matrix or Mass matrix. The user specifies the node incidences and the matrix values.

#### Matrix element Local Coordinate System

The local coordinate system used in the matrix element is the same as the global system, however stresses may be presented as if JA-JB-JC nodes defined a local system.

#### Matrix element Properties and Constants

None needed.

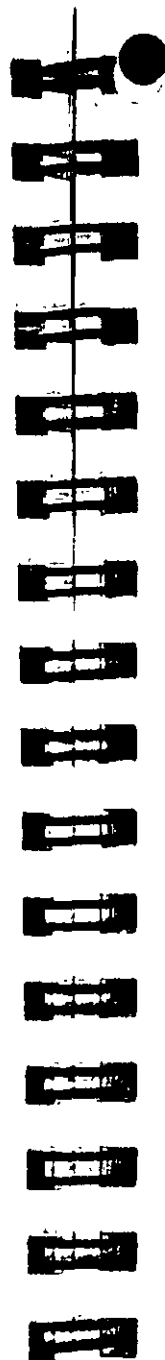
#### Output of Solid Element Stresses

Element forces may be obtained at the nodes of the matrix element.

### 4 Beam Element

The beam element is assumed to be straight 2 node element of constant doubly symmetric cross-section. The beam is capable of resisting axial force, bending moments about the two principal axes (coincident with the beam local axes), and twisting moment about the centroidal axis. The stiffness properties for a uniform beam element are derived from the differential equations for beam displacements in the engineering beam theory. Lateral deflection is the sum of displacement due to bending strain and the displacement due to shearing strain (computed using effective shear area and G).

The beam element is really the sum of 4 uncoupled elements, axial, torsion, shear & bending in x-y, and shear & bending in x-z. Any combination can be created via member releases, or setting cross-section properties to zero, or TRUSS, etc. options.



#### Assumptions

- Linear, elastic, homogeneous.
- Small strains and displacements.
- Plane sections remain plane.
- No coupling of axial, torque and bending.
- Shear deformation are included
- Geometric and elastic properties constant along length.

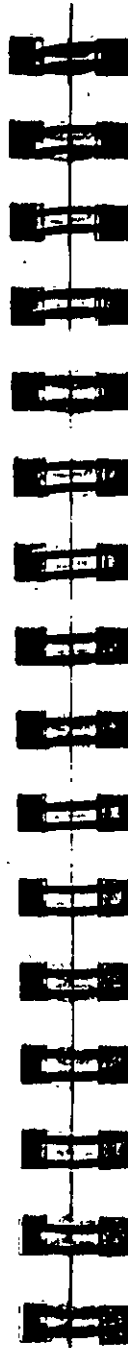
**Beam Local Coordinate System Definition** - The beam element local system are specified the following two methods:

#### STAAD Beam element Beta Angle

The relationship between the local and global coordinate system is defined by an angle measured in the following specified way. This angle will be defined as the beta ( $\beta$ ) angle. When the local x-axis is parallel to the global Y-axis, as in the case of a column in a structure, the beta angle is the angle through which the local z-axis has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis. When the local x-axis is not parallel to the global Y-axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the local z-axis parallel to the global X-Z plane and the local y-axis in the same positive direction as the global Y-axis.

#### Beam element Reference Point

An alternative to providing the beam element orientation is to input the coordinates of an arbitrary reference point located in the element x-y plane but not on the axis of the element. From the location of the reference point, the program automatically calculates the orientation of the element x-y plane.



#### 6.4.1 Beam Element Prismatic Properties

Same as STAAD Analysis engine.

#### 6.4.2 Tapered Beam Elements

Properties of tapered beam elements may be specified. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis. Tapered beam elements may not have beam element loads applied other than selfweight lumped at ends.

#### 6.4.3 Pipe and Elbow Beam Elements

Properties of pipe, pipe tee, and elbow beam elements may be specified. Given key section dimensions, the program is capable of calculating cross-sectional properties which are subsequently used in analysis. Pipe tee and elbow beam elements may not have beam element loads applied other than selfweight lumped at the ends and enclosed liquid weight lumped at the ends. The elbow may optionally allow the calculation of bending flexibility increase due to thin wall ovalization (while considering the opposing effect of internal pressure) to be computed.

#### 6.4.4 Beam Element End Releases

STARDYNE allows releases for beam element end forces and moments. Beam elements are assumed to be rigidly framed into nodes in accordance with the structural type specified. When this full rigidity is not applicable, individual force components at either end of the element can be set to zero with element release statements. By specifying release components, individual element degrees of freedom are removed from the analysis. Release components are given in the local coordinate system for each element.

#### 6.6.4.5 Tension - Only Elements

STARDYNE does not have a tension only element.

#### 6.6.4.6 Cable Elements

STARDYNE does not have a cable element.

#### 6.6.4.7 Beam Element Offsets

Some beam elements may not be concurrent with the incident nodes thereby creating offsets. This offset distance is specified in terms of global coordinate system (i.e. global X, Y and Z distance from the incident node). Secondary forces induced, due to this offset connection, are taken into account in analyzing the structure and also to calculate the individual element forces. The new offset centroid of the beam element can be at the start or end incidences and the new working point will also be the new start or end of the beam element. Therefore, any reference from the start or end of that element will always be from the new offset points.

### 6.7 Rigid System of Nodes – STARDYNE

STARDYNE provides for rigid systems similar to STAAD's master/slave option with all degrees of freedom (Fx, Fy, Fz, Mx, My and Mz) provided as constraints. The nodes of a rigid system will be assumed to be rigidly connected. Rigid systems are defined using beams with the rigid beam property specified.

The Rigid Beam can be used to model those elements of a structure which are very stiff in relation to the total structure. One or more rigid beams form a Rigid System.

The highest numbered node in each rigid system (independent node) will remain a part of the elastic structural model. The remaining nodes in each rigid system (dependent nodes) will not be represented in the stiffness matrix for the elastic model.

However, if any static loads are applied to these dependent nodes, they will be correctly transferred to the elastic model.

In the nodal equilibrium check, forces and moments will appear at those nodes contained in a rigid system. A free-body analysis of the rigid system will indicate that these forces and moments will be in static equilibrium with the external loads acting on the rigid system.

## Material Constants – STARDYNE

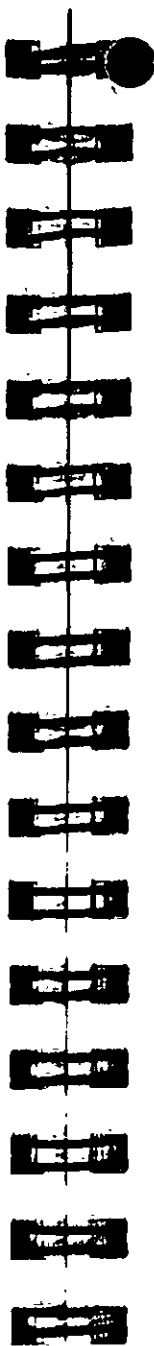
The material constants are: moduli of elasticity (E1, E2, E3); weight density, Poisson's ratio (POIS12); coefficients of thermal expansion (ALPHA1,2,3), G12, G23, G31, composite damping coefficient, and composite element failure criterion constants.

Weight density is used in static analysis when selfweight of the structure is to be taken into account and in dynamics to calculate the nodal masses. Poisson's ratio (POIS12) may be used to calculate the shear modulus (G12), if G12 is left blank, by the formula,

$$G12 = 0.5 \times E1 / (1 + POIS12)$$

Coefficients of thermal expansion (ALPHA1,2,3) are used to calculate the expansion of the elements if temperature loads are applied. The temperature unit for temperature load and ALPHAs have to be the same.

The material constant directions are assumed to be principal directions. For beams the material axes are the beam local axes. For solid elements the material axes are the global axes. For plate elements, the material axes are the same as the local plate axes unless an axis angle is specified for a plate.



## 6.9 Supports – STARDYNE

STARDYNE allows specifications of supports that are parallel as well as inclined to the global axes. Each of the possible six nodal support directions may be restrained or released. Boundary node support dof locations must be specified in order to impose known displacements at those locations in a static analysis. The actual displacement is specified in each load case input.

## 6.10 Loads (STARDYNE)

Loads in a structure can be specified as node load or temperature, and element load or temperature. STARDYNE can also generate the self-weight of the structure and use it as uniformly distributed element loads in analysis. Any multiple of this self-weight can also be applied in any desired direction.

### 6.10.1 Node Load

Node loads, forces & moments or temperature, may be applied to any free node of a structure. These loads act in the global coordinate system of the structure. Positive forces act in the positive coordinate directions. Any number of loads may be applied on a single node, in which case the loads will be additive on that node.

It does not matter which type or types of elements are connected to the node except that if the elements cannot carry loads in that direction, the load in the unsupported direction will be ignored. Loads at supports will not contribute to deformations or element results.

## 6.2 Element Loads

### Beam Element Load

Three types of loads may be applied directly to beam elements in a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a beam element. Concentrated loads act at any intermediate, specified point. Linearly varying loads act over the full length of a beam element. Trapezoidal linearly varying loads act over the full or partial length of a beam element. Up to 10 load descriptions may be specified to act upon a beam element in any independent loading condition. Beam element loads can be specified in the element coordinate system or the global coordinate system. Uniformly distributed element loads provided in the global coordinate system may be specified to act along the full or projected element length. Positive forces act in the positive coordinate directions, local or global, as the case may be.

Temperatures may be specified on 4 points of cross-section and are assumed constant along the length. The difference between the specified temperature and the ambient temperature will cause initial axial strain and/or bending strain in proportion to the thermal expansion coefficient, alpha, for the material and the element geometry.

### Plate Element Load

On Plate/Shell elements, the types of loading that are permissible are:

- 1) Pressure loading which consists of loads which act perpendicular to the surface of the element. The pressure loads can be of uniform intensity or trapezoidally varying intensity over the entire surface of the element.
- 2) Temperature loads which may be constant across the depth of the element (causing only in-plane elongation / shortening) or may vary across the depth of the element causing bending and elongation on the element. The



coefficient of thermal expansion, alpha, for the material of the element must be provided for computation of these effects. Orthotropic properties permitted. Temperatures may vary from node to node.

- 3) The self-weight of the elements can be applied using the SELFWEIGHT loading condition. The density of the elements has to be provided for computation of the self-weight.

### Solid Element Load

On Solid elements, the loading types available are pressure variation normal to each face, temperatures at the nodes, and the self-weight using the SELFWEIGHT loading condition. The alpha and/or density of the elements has to be provided for computation of these loads. Three-dimensional orthotropic properties permitted.

## 6.10.3 Node Temperature Load

Temperatures may be specified at nodes or on element surfaces. Node temperatures are transferred to element temperatures before processing. The difference between the specified temperature and the ambient temperature will cause initial axial strain and/or bending strain (or bi-axial or tri-axial strain for 2d and 3d elements) in proportion to the thermal expansion coefficients, alphas, for the material and the element geometry. Orthotropic material properties may be used. The program calculates the strains (elongation and shrinkage) due to the temperature differences. From this it calculates the induced forces in the element and the analysis is done accordingly. Nodal temperatures will not induce bending initial strain in beams or plates.



## 6.11.1 Support Displacement Specification

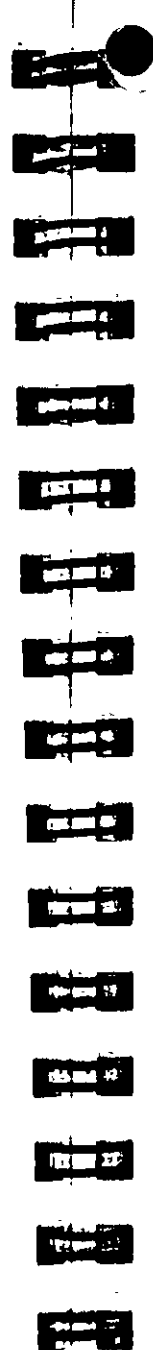
Displacements can be specified at boundary support nodes in specified directions. Displacement can be translational or rotational. The rotational displacements are in radians. Note that displacements can be specified only in directions in which the boundary support node directions are defined.

## 6.11.2 STARDYNE Analysis Capabilities

Currently available analysis capabilities include

- 1) Static Analysis
  - Stiffness Modeling
  - Applied Load Modeling
  - Static Equation Solution
- 2) Second Order Static Analysis:
  - P-Delta Analysis
  - Stress Stiffened static and eigensolution
  - Buckling
- 3) Dynamic Response Analysis (Modal):
  - Mass Modeling
  - Natural frequencies and modes analysis
  - Response Spectrum analysis
  - Linear Time History analysis

Key features of each type of analysis are discussed in the following sections. Detailed theoretical treatments of these features are available in standard structural engineering textbooks.



## 6.11.1 STARDYNE Static Analysis

**STIFFNESS MODELING** - The general solution procedure consists of formulating a nodal stiffness matrix from the finite elements (beams, plates, and solids) and performing one or more of the following operations: static analysis, eigenvalue/eigenvector extraction, and dynamic response analysis. The static analysis and modal extraction phases are based on the "Stiffness Method" also known as the "Displacement Method". The assumptions and formulations are consistent with "Small Displacement Theory".

Each finite element contributes stiffness to the nodes to which it is connected. Each element type has an assumed form of displacement in a manner which satisfies the force equilibrium and displacement compatibility at the nodes. The nodal stiffness matrices of the individual finite elements are first computed and then transformed from the local element coordinate system to the global coordinate system. Finally, the individual stiffnesses associated with each nodal point are systematically summed to obtain the total (global) stiffness matrix  $[K]$ . This square symmetric matrix has up to 6 equations per node.

### Assumptions of the Stiffness Method

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions:

- 1) The structure is idealized into an assembly of beam, plate, solid, spring, and matrix type elements joined together at their vertices (nodes). The assemblage is loaded and reacted by concentrated loads acting at the nodes. These loads may be both forces and moments which may act in any specified direction.
- 2) A beam element is a longitudinal structural element having a constant, doubly symmetric or a linearly tapered cross section along its length. Beam elements may carry axial forces, shear and bending in two arbitrary perpendicular planes, and are also subjected to torsion.

- 3) A plate element is a three or four node flat element having constant thickness and orthotropic properties; or composite layered properties; or may be axisymmetric.
- 4) A solid element is a four to eight node three dimensional element having uniform orthotropic properties.
- 5) A general matrix element may be a spring or generated element matrices or assembled stiffness / mass matrices from an external source.
- 6) Beams have 1 to 6 degrees of freedom (DOF) at each node; plates have 5 DOF; and solids have 3 DOF. Because of this mismatch in DOF, the connection between solid elements and plates or beams needs special modeling.
- 7) The solved internal element forces and the applied external loads acting on each node are in equilibrium except at supports.
- 8) Two types of coordinate systems are used in the generation of the required matrices and are referred to as local and global systems. Local coordinate axes are assigned to each individual element and are oriented such that computing effort for element stiffness matrices are generalized and minimized. The nodal coordinate input data are by definition in the Global coordinate axis system. The assembled forces and stiffness and the solved displacements are in this Global coordinate system.

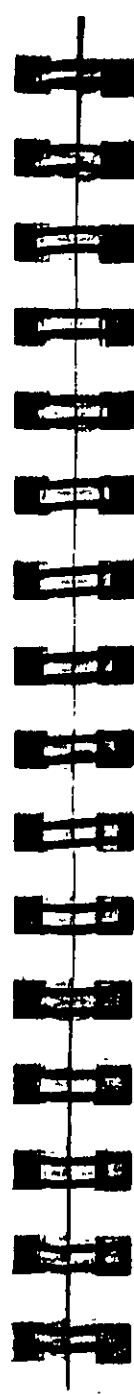
**APPLIED LOAD MODELING** - Loads may be applied in the form of distributed loads on the element surfaces or as concentrated loads at the nodes, element thermal gradients, pressures, inertia (self-weight) loads and imposed nodal displacements. Using standard finite element methods the loadings are assembled into a nodal force vector.

**STATIC EQUATION SOLUTION** - During a static analysis, this matrix equation is solved:

$$[K] \cdot [\delta] = [P]$$

Where

$[K]$  = the stiffness matrix



$[\delta]$  = the resulting nodal displacement vectors  
 $[P]$  = the applied nodal force vectors

Static equation solution uses one of two methods:

- 1) Preconditioned conjugate gradient solver (TurboSolver). This solver is much faster than method 2 and uses much less disk. The TurboSolver is used whenever possible [no inclined supports (ASYSG), spring supports, or rigid systems (master/slave) allowed]; otherwise a
- 2) Modified Gauss, LDL<sup>T</sup>, variable bandwidth solver is used. This solver is always used in the eigensolutions.

#### Consideration of $[K]$ Matrix Bandwidth

Method 2 (LDLT) is more efficient when the equations are reordered to have the non-zero  $[K]$  matrix terms form a narrow band about the diagonal. STARDYNE automatically computes a nearly optimum node order. For this type of matrix fewer calculations are required due to the fact that terms outside the band are all equal to zero. STARDYNE takes full advantage of the variable bandwidth during the solution.

**Independent "Disjoint" Structures (Structural Integrity)** - Unlike STAAD two or more independent structures may be solved. Check the "message file" for disjoint structure messages to see if independent structures were detected. Solving multiple structures in one analysis is not recommended and is usually the result of a modeling error.

**Numerical Instability Problems - Singularities:**  
 Instability problems can occur due to two primary reasons.

- 1) Modeling problem - There are a variety of modeling problems which can give rise to instability conditions. They can be classified into two groups.

- a) Local instability - A local instability is a condition where the fixity conditions at an element node are such as to cause an instability in the element about one or more degrees of freedom. Examples of possible local instability are:
- (i) Beam Release: Beam elements released at both ends for any of the following degrees of freedom (FX, FY, FZ and MX) may be subjected to this problem.
  - (ii) TRUSS or axial only beams have no capacity to transfer shears or moments from the superstructure to the supports. Depending upon the load carrying capacity of the resulting structure there may be a local instability.
  - (iii) STARDYNE plates do not have the 6<sup>th</sup> dof (rotation about the normal to the surface direction) at each node.

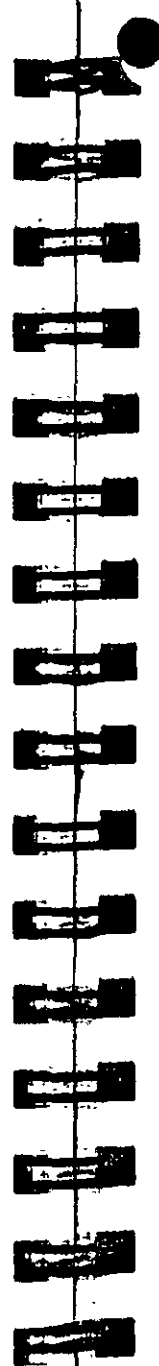
**NOTE 1:**

Zero Stiffness at a node in one or more directions is OK. Loads at those directions will be ignored

**NOTE 2:**

For the LDLT solver, Node Point Singularities are corrected (but not for the TurboSolver). This singularity occurs when there is a zero stiffness direction other than a global direction.

- b) Global Instability - These are caused when the supports of the structure are such that they cannot offer any resistance to motion, sliding or overturning of the structure in one or more directions. An example is that of a structure with all the supports released for FX, FY or FZ.



- 2) Math precision - A math precision error is caused when numerical instabilities occur in the matrix decomposition process. When a very "stiff" element is adjacent to a very "flexible" element, a math precision error may occur. Replace stiff elements with a rigid system or artificially soften the element.

The loads and stresses in the elements are computed using the computed nodal displacement vector.

## 6.11.2 STARDYNE Second Order Static Analysis

### Stress Based Stiffness Modeling (Star)

STARDYNE can compute additional bending stiffness for beams and plates given the axial or inplane stresses. These matrices are known as geometric stiffness matrices [Kg]. When a stress stiffened related analysis is chosen (static, eigen solution, buckling, or P-Delta) then a standard static analysis is performed first to compute the plate & beam stresses. From those stresses and the element geometry, the additional bending stiffness is computed. Then the analysis is performed with the additional stiffness included.

Buckling analyses require LANCZOS (DYNAMIC entry) not HQR on the second run. Stress "stiffened" natural frequencies may use either eigenvalue method.

All quad plate elements (QUADB, QUADH, QUADS, QUADC) including composites can be accommodated, although the geometric stiffness matrix will be the same for all types. All triangular plate types may also be accommodated including 3-node membrane quad plates.

Bending only plates and/or beams, or solid elements, or matrix elements produce no stress stiffening and therefore do not contribute to buckling. Positively stressed elements become stiffer and negatively stressed elements become more flexible.

The following STARDYNE features should not be used, or should be considered carefully when used in stress "stiffened" or buckling analyses: ASYSG; AXIS ANGLE; Solid Elements can be used but no stress stiffening will result from them and they will not enter into any buckling calculations. offset beams, elbows, pipet, and pinned beams are treated as standard straight beams connected with 6 DOF/node at each end of the beam

**P-Delta (P-Δ) or Nonlinear (Star)**

**WHAT:** This is an iterative, geometrically non-linear static analysis. It is often used as part of an analysis of a building to account for the moments induced after lateral displacement of the original applied forces.

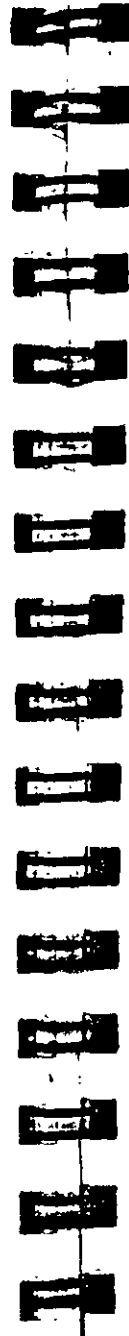
**WHY:** Primarily used as part of an analysis of a building to account for the moments induced after lateral displacement of the original applied forces.  
**WHY STARDYNE:** Two options: (1) updates node coordinates with each iteration; (2) solve incrementally

**HOW:** Select only one static loading case then select P-Delta analysis specifying which option, and number of iterations.

Note that the full load case must be solved simultaneously for this type of analysis, i.e. the lateral loading must be present concurrently with the vertical loading for proper consideration of the P-Delta effect. Other analyses that this procedure may be used for are:

- a) problems with larger displacements while the elements are undergoing small strain;
- b) portions or all of the structure are near buckling or are into simple post-buckling like snap through

You first specify the number of iterations or load steps (default is 4) There is no check for convergence. Next, choose between the following options: 2a. ORIGINAL - Apply the full load on each iteration and add the deformation to the original node coordinates to use as the node coordinates for the next iteration 2b.



**CURRENT** - Solve incrementally. Apply the loads in increments (applied loads divided by (# of increments+1)). Solve #-of-increment times with coordinates and deformations accumulated.

The defaults of 4 iterations and original coordinates (2a), should be adequate for P-Δ effects in buildings. For other, more complex analyses, then more iterations and option 2b may be necessary.

Large deformations could invalidate beam local coordinate definitions and warp plate elements. The local beam coordinates in the intermediate increments are not moving with the beam but are based on either moving nodes which probably are not moving with the beam or on an angle of the beam to the fixed global axes. Deformed and displaced plates often become warped or distorted which degrades the accuracy of the element.

The 'current' coordinates option (which is an incremental load with updated coordinates after each iteration) should have good final displacements. The displacements, element loads and stresses, and nodal equilibrium check results output are the summation of the results from each of the incremental iterations. The element and nodal equilibrium results are approximate to inaccurate depending on the amount of displacement and rotation. Better results for the elements would be obtained by applying the final displacements as a static load case on the model with the original coordinates.

**Stress Stiffened Static and Eigensolution (Star)**

**WHAT:** This is a load case specific analysis. Compression stresses in beam and plate elements are used to determine a global additional stiffness matrix ([Kg]) to add to the global stiffness matrix in either a static or eigensolution.

**WHY:** Primarily used when the tension or compression in beam or plate elements will contribute significantly to those elements bending stiffness and that the increase/decrease in stiffness is important to the solution.

**HOW:** Select only one static loading case then select Stress Stiffened Static or Eigensolution. Other inputs are the same as normal analyses.

The static global matrix equation becomes:

$$[K + K_g] \cdot \{\delta\} = \{P\}$$

The eigensolution global matrix equation becomes:

$$\omega^2 [m] \{q\} - [K + K_g] \{q\} = 0$$

Stress "stiffened" natural frequencies may use either eigenvalue method.

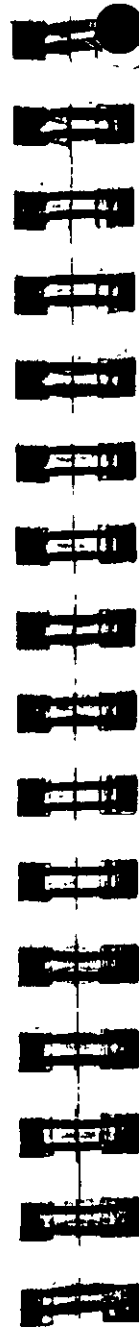
### Buckling (Star)

**WHAT:** Euler buckling calculation based on a single static load distribution. Compression stresses in beam and plate elements are used to determine a global stability matrix ( $[K_g]$ ) to use in place of a mass matrix in an eigensolution. The  $\omega^2$  results are the buckling factors.

**WHY:** Primarily used for stability analysis of parts of structures or mechanical parts.

**HOW:** Select only one static loading case then select buckling analysis.

The process of calculating the system buckling factor and buckled shape involves first solving the static load case for displacements and beam axial stresses and plate inplane stresses. From these stresses, additional beam and plate bending stiffness can be



computed,  $[K_g]$ . The solution of the following eigenproblem will result in the amount that the applied load case would have to be factored (BF) to be at the critical Euler buckling load. The buckled shape is normalized to a maximum displacement of one. To get the actual displacements and stresses at the critical buckling load, repeat the static analysis with the original loads multiplied by BF.

$$BF [K_g] \{q\} - [K] \{q\} = 0$$

Where

$[K_g]$  = the geometric stiffness matrix (banded symmetric)

BF = Buckling Factor

$\{q\}$  = the normalized buckled shape

For buckling analysis, the Lanczos eigenvalues are critical buckling factors. The critical buckling loads are (all of the loads that you have applied) times (the first buckling factor). A positive buckling factor that is less than one indicates the applied load is greater than the buckling load. A negative buckling factor indicates that the applied load case is in the opposite direction of the loading that would cause buckling.

Note that all loads are considered to be factored. If self weight and compression are applied, then the buckling factor would be for both the compression and the self weight.

Buckling analyses require LANCZOS, not HQR.

## 6.11.3 STARDYNE Dynamic Response Analysis

Currently available dynamic analysis capabilities include solution of the free vibration problem (eigenproblem) and several forced vibration excitation analysis types: response spectrum analysis, linear time history analysis, and steady state harmonic analysis: all utilizing modal superposition methods. The dynamic response results are presented as structural deformations (displacements, velocities, or accelerations) and as internal element loads and stresses.

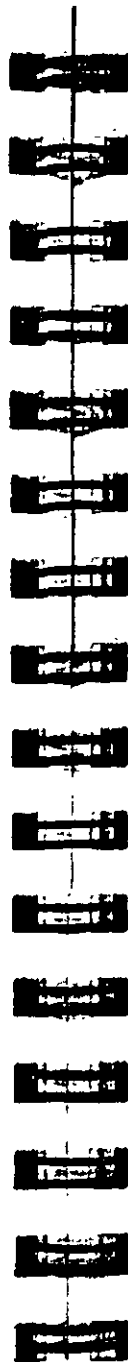
### Mass Modeling (Star)

The natural frequencies and mode shapes of a structure are the primary parameters that affect the response of a structure under dynamic loading. The free vibration problem is solved to extract these values. Since no external forcing function is involved, the natural frequencies and mode shapes are direct functions of the stiffness and mass distribution in the structure. Results of the frequency and mode shape calculations may vary significantly depending upon the mass modeling. This variation, in turn, affects the dynamic analysis results. Thus, extreme caution should be exercised in mass modeling in a dynamic analysis problem. Active masses should be modeled as loads. All masses that are capable of moving should be modeled as loads applied in all possible directions of movement. In response spectrum analysis, as a bare minimum, all masses that are capable of moving in the direction of the spectrum, must be provided as loads acting in that direction.

It is strongly recommended that unit self weight be applied in the X, Y, and Z directions in all STARDYNE Eigenproblem/dynamic analyses even for 2-D frames or structures loaded in one or two global directions. This will generate nodal masses based on density and element volumes. As in STAAD the additional masses at a point can be specified as forces.

### **GUYAN SUPPORT DOF**

If the HQRH-Guyan eigensolution method is chosen, then you should select some mass dof (Guyan dof) to be retained in the eigensolution. The other mass dof will be statically redistributed (transformed) to add to the selected mass dof resulting in a full mass matrix (non-diagonal). This method is directly analogous to the static condensation of stiffness at interior nodes to stiffness at boundary nodes used in substructures or in some finite elements. No stiffness is lost or changed but the kinetic energy of the mass is modified. However if N reasonably distributed Guyan dof are chosen, then  $N/2$  of the lowest eigenvalues will be accurate. This procedure is only used by engineers who know how to use Guyan dof to eliminate unwanted localized modes.



### Damping Modeling (Star)

In the modal dynamic response method, damping is entered as a damping ratio (as a fraction of critical damping; 5% damping is entered as .05) for each eigenvalue/mode used in the response.

An alternative to the user specifying a value for each mode is the composite modal damping method of assigning damping for each mode. This method is based on the concept of different materials having different damping ratios and that a particular mode's damping ratio should be a weighted average of the strain energies \* damping of the materials used.

### Solution of the Eigenproblem (Star)

The eigenproblem is solved for structure frequencies and mode shapes considering a mass matrix lumped at the nodes (See Mass Modeling above). Masses at all active d.o.f. may be considered.

Any structural model which consists of a stiffness matrix and N mass degrees of freedom contains N normal modes of vibration. Each normal mode occurs at a specific frequency of vibration known as a natural frequency (or eigenvalue). At these frequencies, the nodal displacements, known as the mode shape (or eigenvector), oscillate from positive to negative about 0.0. The advantage of modal methods are that only a small number of the low frequency modes are necessary to obtain good dynamic response results.

The process of calculating the system eigenvalues and eigenvectors is known as Modal Extraction and is performed by solving the equation:

$$\omega^2 [m] \{q\} - [K] \{q\} = 0$$

Where

$[m]$  = the mass matrix (assumed to be diagonal, i.e., no mass coupling)

$\omega$  = the natural frequencies (eigenvalues)

$\{q\}$  = the normalized mode shapes (eigenvectors)

Frequency (HZ) =  $\omega/2\pi$

Two solution methods are available: the Lanczos ("Lan Chô's") method (recommended) and the HQR11-Guyan method.

Please note that various nomenclature is used to refer to the normal modes of vibration. (Eigenvalue, Natural Frequency, Modal Frequency and Eigenvector, Generalized Coordinate, Mode Shape, Modal Vector, Normal Modes, Normalized Mode Shape.

#### GENERALIZED WEIGHT AND GENERALIZED MASS

Each eigenvector  $\{q\}$  has an associated generalized mass defined by

$$\text{Generalized Mass (GM)} = \{q\}^T [M] \{q\}$$

$$\text{Generalized Weight (GW)} = \text{GM} * g$$

**PARTICIPATION FACTORS** - A participation factor ( $Q_i$ ) is computed for each eigenvector for each of the three global ( $X_i$ ) translational directions.  $N$  is the number of nodes.

$$Q_i = \frac{\sum_{j=1}^N (q_{ij})(w_{ij})}{\text{GW}}$$

**MODAL WEIGHTS** - The summation of modal weights for all modes in a given direction is equal to the Base Shear which would result from a one g base acceleration. The modal weight for each mode =  $(\text{GW})(Q_i^2)$  The sum of the modal weights for the



computed modes may be compared to the total weight of the structure (that has not been lumped at supports). The difference is the amount of mass missing from a dynamic, base excitation, modal response analysis. If too much is missing, then rerun the eigensolution asking for a greater number of modes or read about how to employ the Missing Mass Correction method.

**Lanczos Method and Free-Free Structures** - An unrestrained, free-free model has zero eigenvalues that may be solved in LANCZOS. However you must know that zero eigenvalues are expected and select the Free-Free option. Internally the program will compute a shift frequency,  $\omega^2$ , which will be used to create a matrix,  $[K_s]$ , to be added to the stiffness matrix,  $[K]$ , prior to the eigensolution ( $[K_s] = \omega^2 [M]$ ). The eigenvectors of the shifted eigenproblem are exactly the same and the frequencies are shifted by  $\omega^2$  from that of the actual eigenproblem. So we simply add  $\omega^2$  to each computed frequency and we have completed an otherwise unsolvable singular eigenproblem. If, however, a singularity is still detected during decomposition, it is an indication that there is a portion of the structure without mass that is free to move statically relative to the rest of the structure. Such a structure could not be solved statically either. For example, a free-free beam which has Area, I2 and I3 properties, but has masses in only the transverse directions does not have mass in the axial direction and consequently will be singular in that direction. The remedy would be to set Area=0.0 or restrain the axial direction, since without mass, no axial modes can exist.

### 6.11.3.1 STARDYNE Response Spectrum Analysis (Dynre4)

**WHAT:** Approximate modal analysis which computes the "Modal Responses" due to a given directional, base excitation, response spectrum. Each mode's response to a given excitation direction are then combined, using one of the many methods available, to form the total response for that input direction. If there is more than one direction of input, the total responses of each direction are then combined by the SRSS method.

**WHY:** Primarily used for seismic structural engineering of buildings.  
**WHY STARDYNE:** supports "missing mass", multiple base excitation; distributed force excitation; the SRSS, CQC, Gupta, Double Sum, NRL, Ten Percent, closely spaced group, etc. methods. In addition, a more realistic method is available: weighted average of CQC (or Gupta) at low frequencies (~3Hz) and algebraic sum at higher frequencies (~20 Hz).

**HOW:** First perform the Mass Modeling step carefully, noting that a revised node/element model with additional mid-span nodes with masses may give better dynamic and element results. Next, decide if Missing Mass modes are to be computed. Then select the number of eigenmodes and/or frequency range required. Next, perform the eigensolution and ensure that the desired frequencies were computed. Now you are ready for the Response Spectrum Analysis Step.

Response Spectrum Analysis allows the user to analyze the response of multi-degree-of-freedom linear elastic structure models subjected to an arbitrarily oriented foundation shock response spectrum input. You may enter Response Spectra versus Frequency tables or select some ratio of the averaged response spectra shapes computed from the 1940 El Centro, 1934 El Centro, 1949 Olympia, and 1952 Taft earthquakes for any of the directions of motion.



For any supplied response spectrum (acceleration or velocity or displacement vs. frequency) the following may be calculated: node displacement, velocity, and acceleration; element forces and stresses, and support reactions. Results of the response spectrum analysis may be combined with the results of the static analysis to perform subsequent design. To account for reversibility of seismic activity, load combinations can be created to include either the positive or negative contribution of seismic results

The Response Spectrum Analysis appendix contains: a complete mathematical description of the 20 modal combination methods; a description of the User furnished force or multi-base excitation analysis procedures; and the spectra for the NRC SSE Safe Shutdown Earthquake.

### 6.11.3.2 STARDYNE Linear Time History Response Analysis (Dynre1)

**WHAT:** Modal dynamic analysis which computes the "Modal Responses" versus time due to forces and/or ground motions that may vary with time. Each uncoupled mode's time history responses are summed to form the total structure response versus time.

**WHY:** Used for any time varying force-motion analysis including seismic, forced vibration, etc.  
**WHY STARDYNE:** supports "missing mass"; multiple base excitation; solution method may be more accurate for high frequency input forces like earthquakes, whole structure response at specific times using nodal forces ( $F = Ma$ ) rather than displacement, select times for whole structure response based on times that specific stresses or displacements reach peak values.



**HOW:** First perform the Mass Modeling step carefully, noting that a revised node/element model with additional mid-span nodes with masses may give better dynamic and element results. Next, decide if Missing Mass modes are to be computed. Then select the number of eigenmodes and/or frequency range required. Next, perform the eigensolution and ensure that the desired frequencies were computed. Now you are ready for the Linear Time History Analysis Step.

STARDYNE has the capability to perform a linear modal response history analysis on a structure subjected to time varying forcing function loads at the nodes; ground motion at its base; initial displacements and velocities; and multiple base accelerations. Output consists of nodal displacements, velocities, accelerations, element loads and stresses. (Please see the Non-Linear Modal Time History Response Analysis if you need to model simple nonlinearities with your analysis. There is also a "direct integration" method time history method if you have a specialized problem requiring that method).

The Linear Time History analysis is performed using the modal superposition method. Hence, all the active masses should be included in the determination of the mode shapes and frequencies. Please refer to the section above on "mass modeling" for additional information on this topic.

In the mode superposition method, it is assumed that the structural response can be obtained from "p" modes (usually the lowest frequency modes). The equilibrium equations are written as:

$$[m]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = \{p_i\} \quad (1)$$

$$\{x\} = \sum_{i=1}^p \{\phi_i\} \eta_i \quad (2)$$

Equation (1) reduces to "p" separate uncoupled equations of the form:

$$\ddot{q}_i + 2\xi\omega\dot{q}_i + \omega^2q_i = R_i(t)$$

where  $\xi$  is the modal damping ratio and  $\omega$  the natural frequency for the  $i$ th mode.

These uncoupled modal equations of motion are solved by the LaPlace transform method from one solution-time-point to the next solution-time-point (this is not a numerical integration method like the Wilson -  $\theta$  method). There is a solution-time-point at each time that any load curve changes slope as well as at the requested output times. With the assumption that the load curves are linear between solution-time-points this method is exact and stable. Unlike in STAAD, this method does not require an integration time step for solution accuracy but does need the user to specify a sufficient number of output times to allow linear X-Y plots to reasonably show the peaks and shape of the response curves. There is no advantage in having thousands of output time points however.

Output consists of "relative" and "absolute" nodal displacements, velocities, accelerations, element loads and stresses. For selected element stress locations and/or selected node-direction displacements or accelerations, that response value may be displayed as a function of time. "Relative" response is (for the case of base ground motion) the nodal response relative to the base motion (basically the elastic part of the response).

The user may also select times (and/or have the program select peak times) at which the response of the complete structure is computed. The results and displays that are then made possible are similar to those in a static solution. The user may select to use either nodal forces ( $F = Ma$ ) (recommended method) or displacements at the selected times for the calculation of element forces and stresses and the support reactions.

Less frequently used features including: enforced nodal motions, individual modal results, large mass and relative motion methods



for multi-base excitation; are described in the Linear Time History appendix

### 6.11.3.3 STARDYNE Steady State Harmonic Frequency Response Analysis (Dynre2)

**WHAT:** Modal analysis which computes the "Modal Responses" due to a given directional, base excitation or distributed forces with their magnitude varying over the selected frequency range. Each forcing direction can be out of phase with the other directions by a phase lag angle. Each mode's response is computed and then summed to form the total response for each frequency.

**WHY:** Used to understand the response of a structure to a broad range of frequencies. Could be used as a basis to tune a structure to minimize response or to modify the frequency of the forces to avoid resonance.

**WHY STARDYNE:** supports "missing mass"; multiple base excitation, more accurate and straightforward than a sine wave input to a time history analysis.

**HOW:** First perform the Mass Modeling step carefully, noting that a revised node/element model with additional mid-span nodes with masses may give better dynamic and element results. Next, decide if Missing Mass modes are to be computed. Then select the number of eigenmodes and/or frequency range required. Next, perform the eigensolution and ensure that the desired frequencies were computed. Now you are ready for the Steady State Harmonic Frequency Response Analysis.

Steady state harmonic frequency response to steady state sinusoidal dynamic loadings using the modal superposition method. This is a "sine sweep" approach where the response is calculated for the given loadings applied at a sequence of frequencies. Naturally, for lightly damped structures, the responses will peak at the natural (modal) frequencies of the structure.

Input forcing functions may be in the form of distributed forces, base excitations (displacements, velocities or accelerations) and unit sinusoidal excitations (displacements, velocities, accelerations or forces) at specified nodes. Output consists of "relative" and "absolute" nodal responses and element loads and stresses versus frequency (X-Y plot displays of selected node responses or element forces/stresses). The response of the complete structure at particular point in the overall cycle may be computed as well. The results and displays that are then made possible are similar to those in a static solution.

The use of Frequency spacing criteria and output frequency input to improve the quality of the results versus frequency graphs and the probability of identifying the peak response frequencies is described in the Harmonic response appendix.

### 6.11.3.4 STARDYNE Missing Mass Method

**Correction Modes** - In structural dynamic analysis by modal superposition it is sometimes necessary to have a method to account for the static effects of those higher frequency modes which were not calculated in the eigenvalue analysis and therefore will be omitted from the response solution. These omitted higher frequency modes should have natural frequencies well above the range of dynamic input so they would act in a static manner.

The missing mode approach exactly accounts for the static effect of all missing modes and will exactly calculate the correct results in all modal dynamic response programs as long as the missing mass modes are truly well above the forcing frequency range.

The missing mass method described here is forcing function specific. That is, there is one additional missing mass mode for each different (does not have the same spatial distribution pattern of force; proportional forcing functions are not different) forcing function of interest. The method is quite simple. First, the user calculates as many modes as desired. Then, in the same analysis, a static analysis of each forcing function is calculated. Each Static displacement is orthogonalized to the computed modes to

mine the missing mass mode (that part of the static displacement that cannot be represented by the computed modes) for this forcing function.

Then, run the dynamic analysis by selecting all real computed modes and desired damping as usual while also selecting which missing mass modes are associated with each forcing function. Note that the missing mass modes are not normalized to one as would be the case for an eigenmode shape

Additional details of the Missing Mass method are in the Missing Mass appendix.

## MISSING MASS APPENDIX

**"DYNAMIC" and "STATIC" Missing Mass Methods** - The "dynamic" method uses a pseudo dynamic mode approach. The missing mass mode is given an eigenvalue. Then the generalized weight and participation factors are computed from the missing mass mode shape. Depending on the eigenvalue/ forcing function/ spectra/ damping given, the dynamic response is computed for this mode as if it were an eigenmode. These modes may be dynamically amplified as well as statically amplified. In some cases the "dynamic" method improves the dynamic portion of the response but should not be a substitute for including enough eigenmodes to describe the dynamic acceleration and velocity response to the input forcing functions. The "dynamic" method is not documented in the literature so it is not known whether the method always has the effect of improving the dynamic response.

The primary function of missing mass modes should be to describe the additional detailed static deformation and stresses of the omitted eigenmodes (i.e. modes that would have no dynamic response to the forcing functions). The missing mass modes usually provide a localized but sometimes significant improvement in displacements, element forces and stresses; usually near the supports.

In the classic "static" missing mass method the extra static residual vectors may only be amplified statically. In this method only displacements, element forces and stresses are changed. This method is well documented in the literature and does (in the time history solutions) exactly supply the missing static "mass" effect that the lower eigenmodes selected do not supply.

**Time History Response (Dynre1)**- The missing mass modes can be referenced by any of the first 300 forcing functions. A missing mass mode number may be referred to by more than one forcing function. However each forcing function may only refer to one missing mass mode. The "dynamic" missing mass method is the default method.

**Steady State Harmonic Frequency Response (Dynre2)** - This program only uses the "dynamic" missing mass method.

**Response Spectrum Response (Dynre4)** - The "dynamic" missing mass method will be used as the default. The Missing mass method will generally improve the Response spectrum solution. However (unlike time history solutions) it will not give the same deformations and element results as a modal solution using all of the possible eigenmodes. The Missing mass method should not be used with User Furnished Force or Multi-base Input.

If you are running only 1 global shock direction with one missing mass mode and while using the "dynamic" missing mass method; then there is no need to indicate that the mode is a missing mass mode.

In the "static" method the spectra will be chosen at the ZPA frequency. In CQC, DBLSUM, or GUPTA, the coupling of the missing modes to other modes and to themselves is ignored.

For the "static" method or for multiple shock directions it is necessary to select each missing mass mode separately and to indicate which directions the missing mass mode is not to be used (e.g. for X direction shock mode: IMX=0, IMY=1, IMZ=1).

Alternatively a new option allows you to set IMX=2 and leave IMY=IMZ= blank.

For multiple shock directions, the RSS=14 to 20 combination methods should work properly with the multiple missing mass modes.

For RSS=1 to 5, only one direction is used by taking the indicated vector sum of the base shocks before the modal combinations. For missing mass, please run your static 1g run in the same non-global direction to create the one missing mass mode. Then select all the modes without declaring any mode to be a missing mass mode.

For RSS=6 to 13, multiple independent shock directions should be possible. However RSS=2 to 13 are untested and therefore not recommended for use with missing mass modes.

## 6.12 References (STARDYNE)

The basic theory of the Finite Element Method is presented in many textbooks. The following publicly available documents describe many of the technical aspects of the STARDYNE finite elements.

- **Beam element**

Przemieniecki, J. S. Theory of Matrix Structural Analysis. McGraw-Hill, New York, USA, 1968.

- **Triangular Plate**

Clough, R. W. and Tocher, J. L. Finite Element Stiffness Matrices for the Analysis of Plate Bending. Conference on Matrix Methods in Structural Mechanics, Wright Patterson, AFB, 1965 (thin plate)

Martin, H. C. Stiffness Matrix for a Triangular Sandwich Element in Bending. JPL No. 32-1158, Jet Propulsion Laboratory, Pasadena, California, October, 1967 (thick plate and sandwich option)

### Quadrilateral (QUADB)

Clough, R. W. and Felippa, C. A. A Refined Quadrilateral Finite Element for the Analysis of Plate Bending. AD 703-685, Proceedings of the 2nd Conference on Matrix Methods in Structural Mechanics, 1968 (thin plate)

- **Solid Hexahedron**

Irons, B. and Ahmad, S. Techniques of Finite Elements. Ellis Horwood, Sussex, UK, 1980.

Wilson, E. L., Taylor, R. L., Doherty, W. P., and Ghaboussi, T. Incompatible Displacement Models. Numerical and Computer Methods in Structural Mechanics, pp 43-57, ed. S. Fenves, et al., Academic Press, 1973. (8-node cube can be made more flexible by the incompatible modes-- 'bubble functions').

Taylor, R. L., Beresford, P.J., Wilson, E.L. A Non-Conforming Element for Stress Analysis. International Journal of Numerical Methods in Engineering, 10, 1976 (makes an element with incompatible modes to pass the patch test).

- **Hybrid Cube and Hybrid Quadrilateral Plate**

Pian, T.H.H. Hybrid Models. Numerical and Computer Methods in Structural Mechanics, pp58-78, ed. S. Fenves, et al., Academic Press, 1973.

Irons, B. and Ahmad, S. Techniques of Finite Elements. Ellis Horwood, Sussex, UK, 1980 (Chapter 15).

Loikkanen, M. Hybrid Membranes, Solids and Plates. System Development Corporation, Santa Monica, California.

- **Piping**

ASME Boiler and Pressure Vessel Code, Section III., 1971

**Recommended Overall Reading**

Cook, R D., et.al. Concepts and Applications of Finite Element Analysis. John Wiley, 1989, Third Edition.

Bathe, K.-J. Finite Element Procedures in Engineering Analysis. Prentice-Hall, 1982

Hughes, T. J. R. The Finite Element Method. Prentice-Hall, 1987.

Zienkiewicz, O.C. The Finite Element Method. McGraw-Hill, 1977, Third Edition.

