FACULTAD DE INGENIERÍA UNAM
DIVISIÓN DE EDUCACIÓN CONTINUA

CURSOS ABIERTOS

STAAD-PRO

PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

CA 003

TEMA

INSTRUCTIVO PARA LA UTILIZACIÓN DEL MÓDULO STAAD DEL PROGRAMA DE COMPUTADORA STAAD/PRO

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERÍA
ENERO DEL 2003

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

CONTENIDO

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 caracteristicas 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, mamposteria, 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 tecnologia 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).

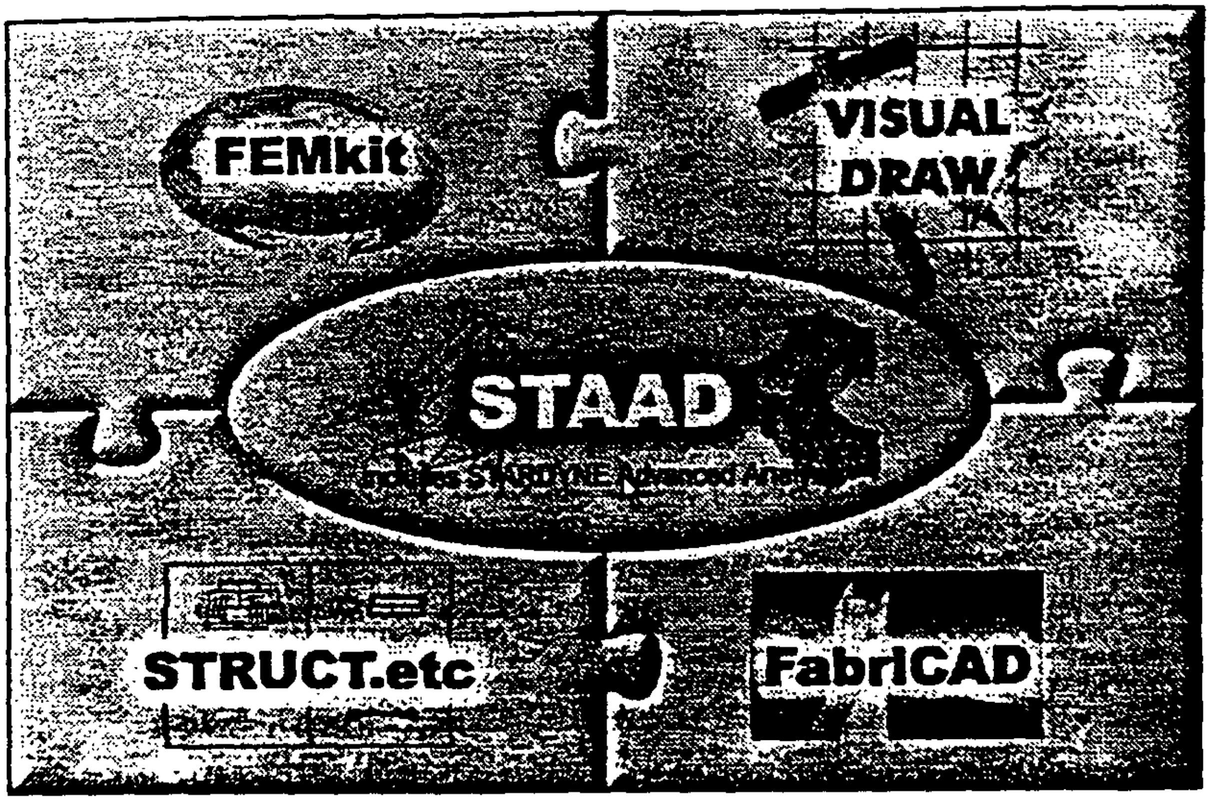


Figura 1.1 STAAD/Pro, programa principal y sistemas que lo integran.

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 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.

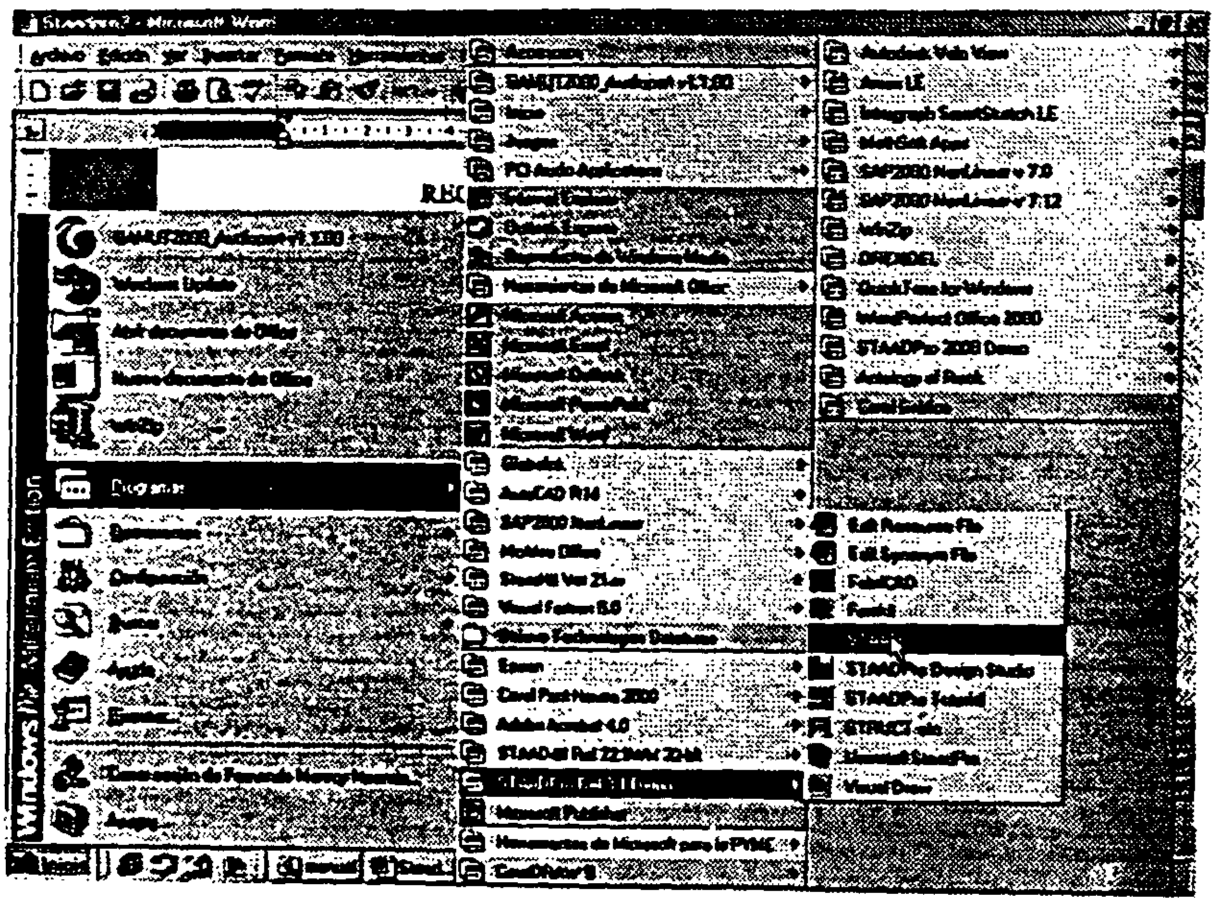


Figura 2.1 Ejecución del programa **STAAD/Pro.**

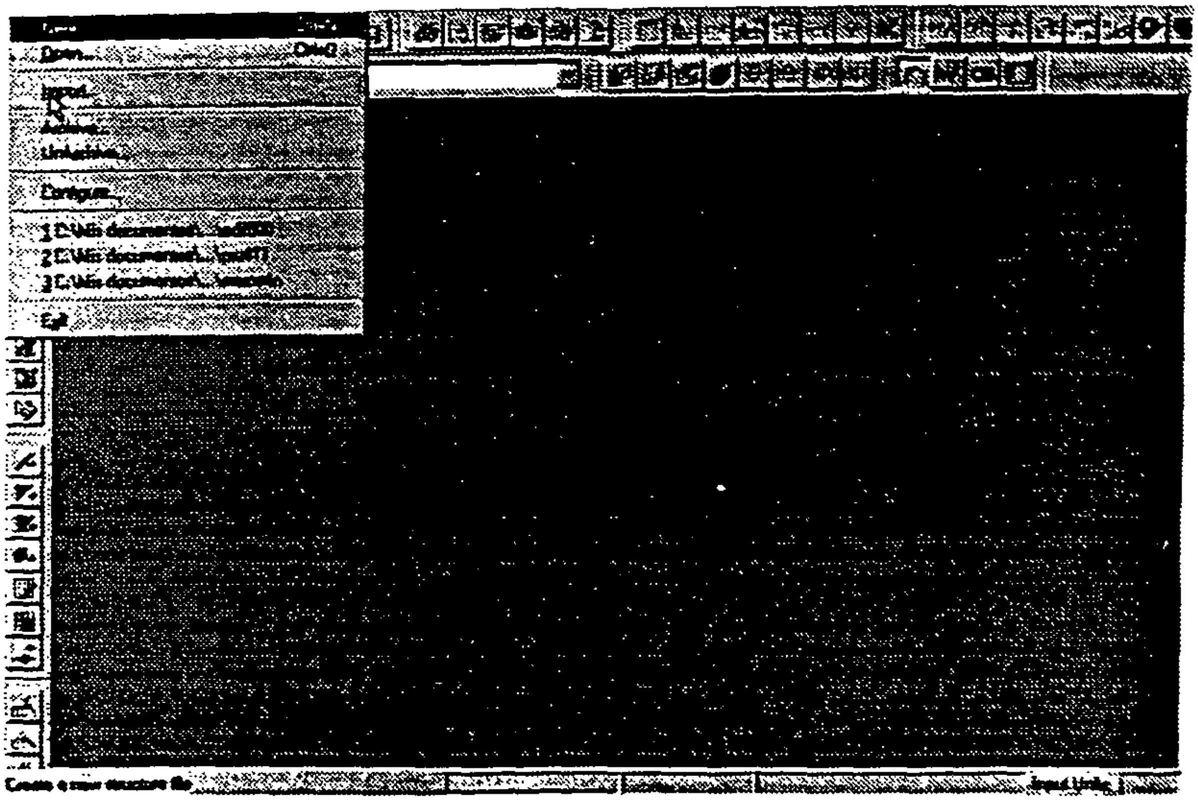


Figura 2.2 Inicio del programa STAAD.

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 New del menú File (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.

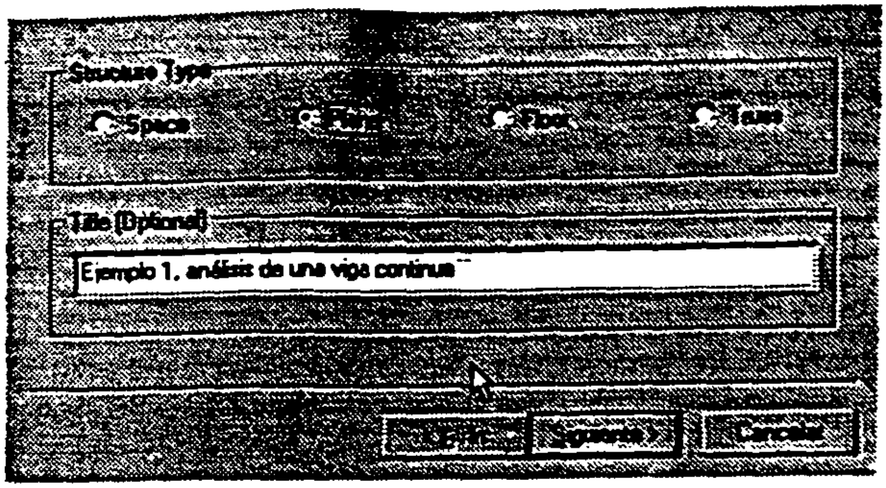


Figura 2.3 *Datos generales al inicio del programa* **STAAD**.

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 (reticulas) 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 rigido).

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.)

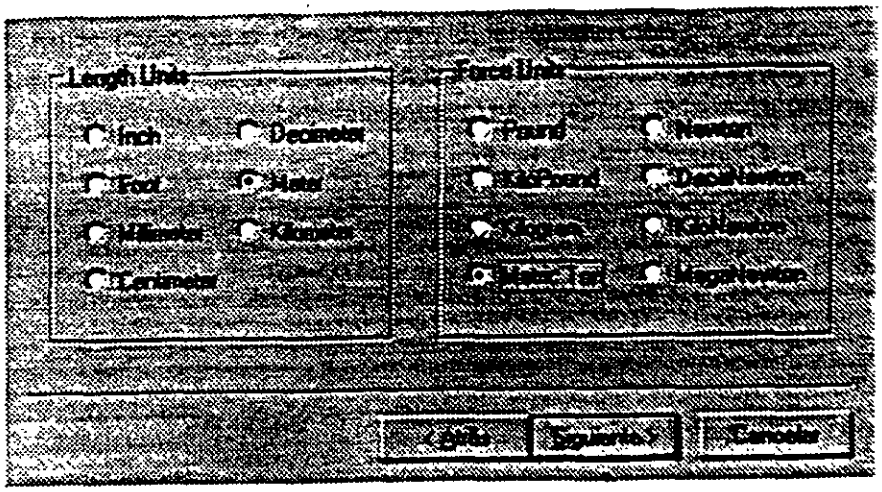


Figura 2.4 Datos de unidades al inicio del programa **STAAD**.

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**.

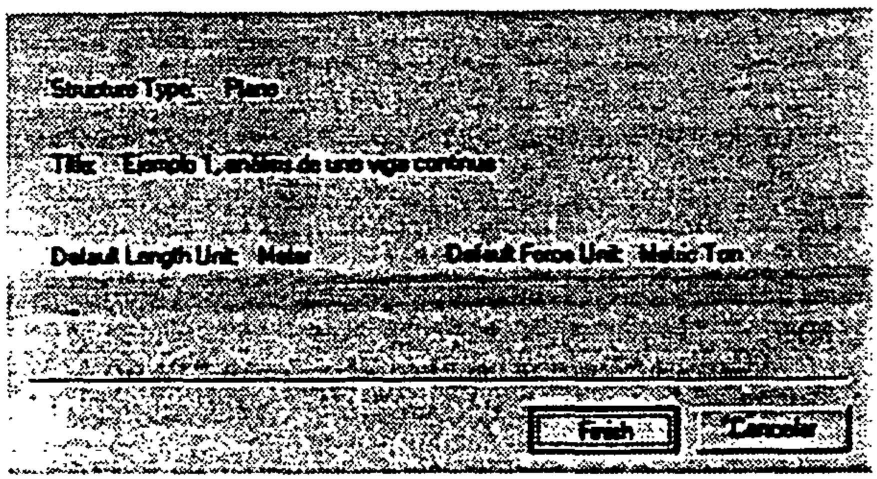


Figura 2.5 Datos seleccionados por el usuario al inicio del programa **STAAD**.

Obsérvese que en esta ventana (figura 2.6), en general, están contenidos algunos elementos tipicos 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 categorias (Job, Setup, Geometry, etc.), seguidas por un área con fondo blanco que se utilizará para desplegar gráficamente la geometria y algunas caracteristicas 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.).

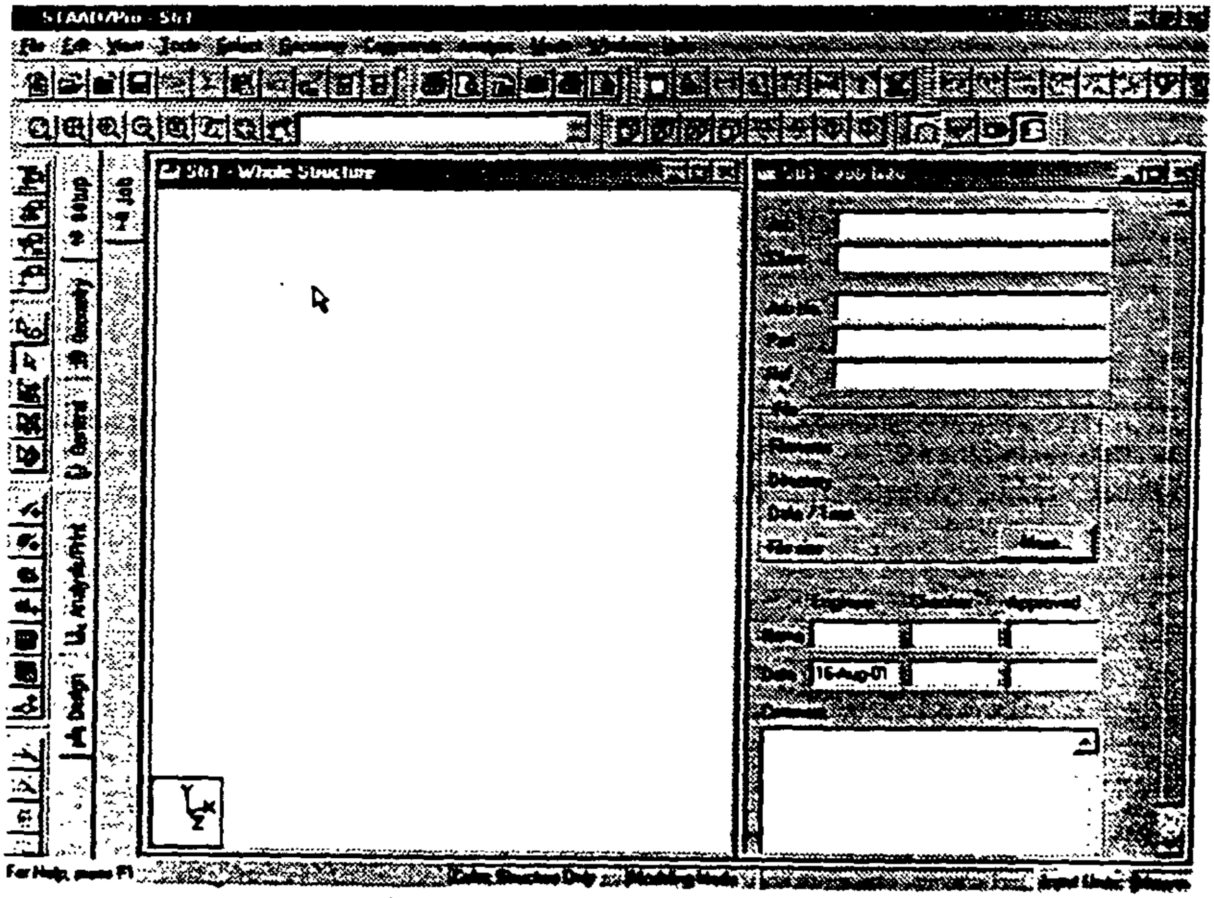


Figura 2.6 Ventana completa del programa STAAD.

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).

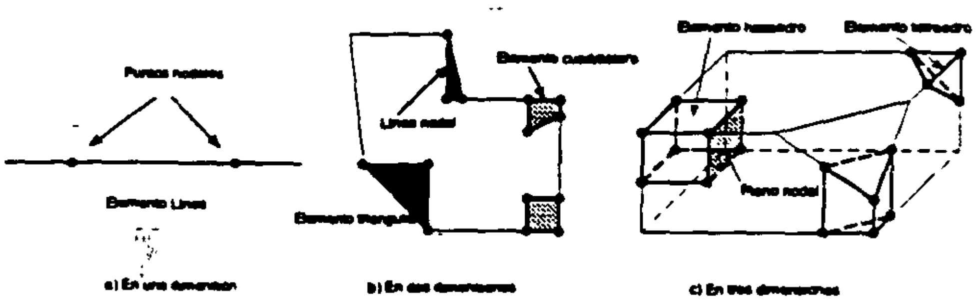


Figura 2.7 Tipos de nudos.

2.4 Definición de las propiedades geométricas de los elementos

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

a) Prismáticos (rectangular, circular, etc.).
b) Elementos estándar de acero.
c) Elementos de acero definidos por el usuario.
d) Sección I de peralte variable.
e) 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.

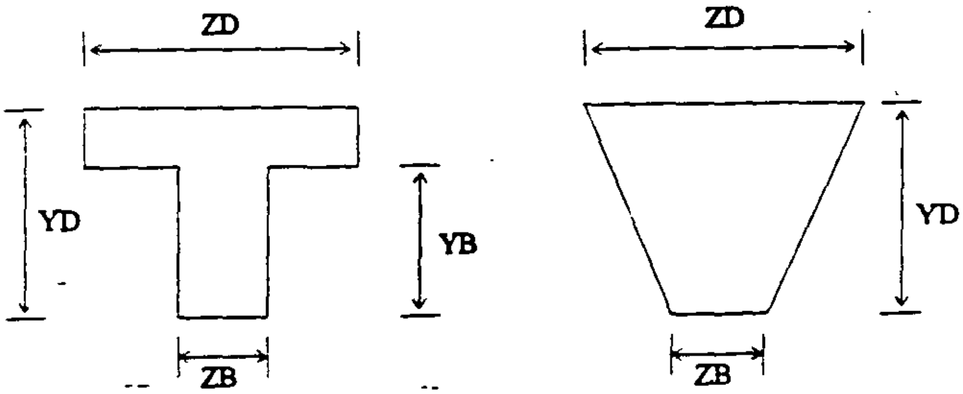


Figura 2.8 Características de secciones T y trapecial

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.

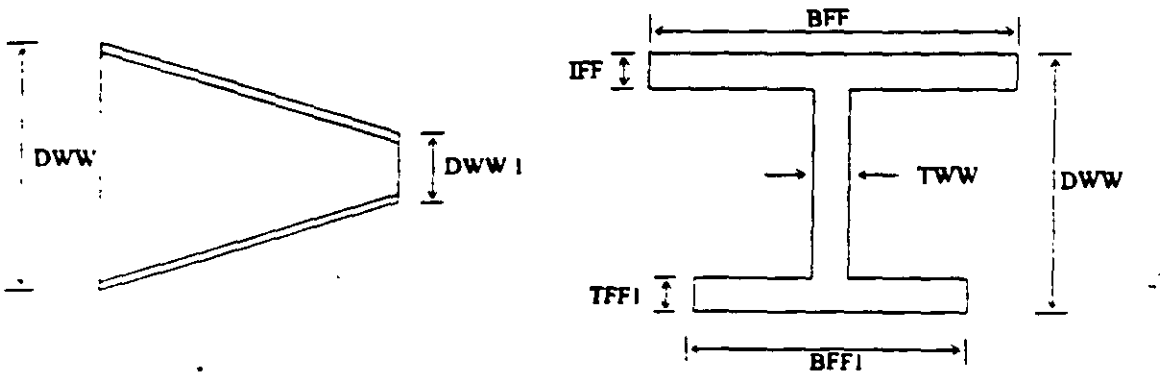
Dependiendo del tipo de estructura, en la tabla 2.1 se muestran las propiedades geométricas mínimas que es necesario proporcionar para que el análisis se pueda realizar.

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.



$$DWW \geq DWW1$$

Figura 2.9 Características de la sección I de perálte variable.

Al programa se le pueden dar instrucciones para que, de manera automática, maneje a los elementos con secciones de formas especificas (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 1er orden y también de 2do orden, en el segundo caso se consideran efectos P-Δ, 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, fy, 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 μ (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 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(HS2O, HS15, H2O, HI5) 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-Δ, análisis de segundo orden (especificando el número de iteracciones) y análisis dinámico.

- Y otras opciones más.

3.2 Descripcion general

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

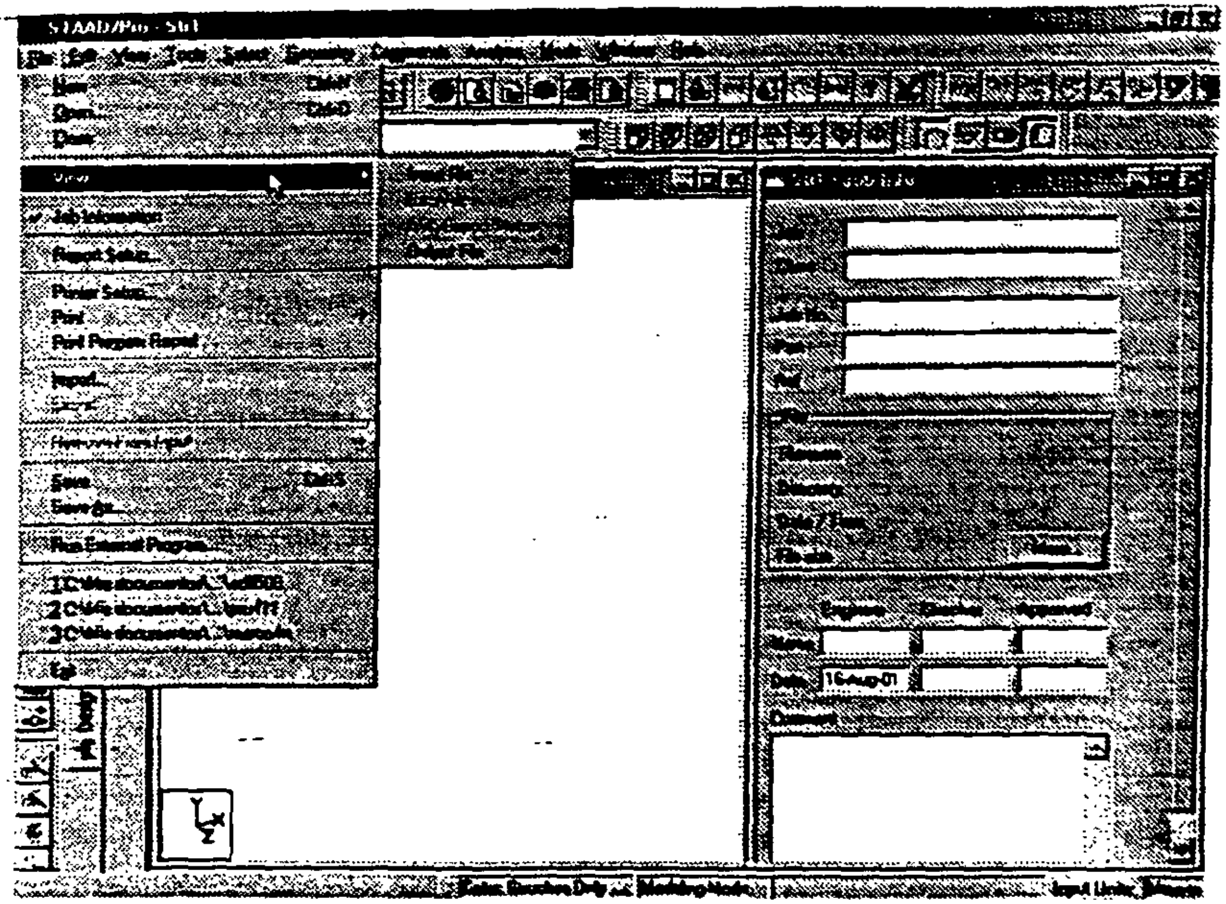


Figura 3.1 Menú File de STAAD.

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 Edit

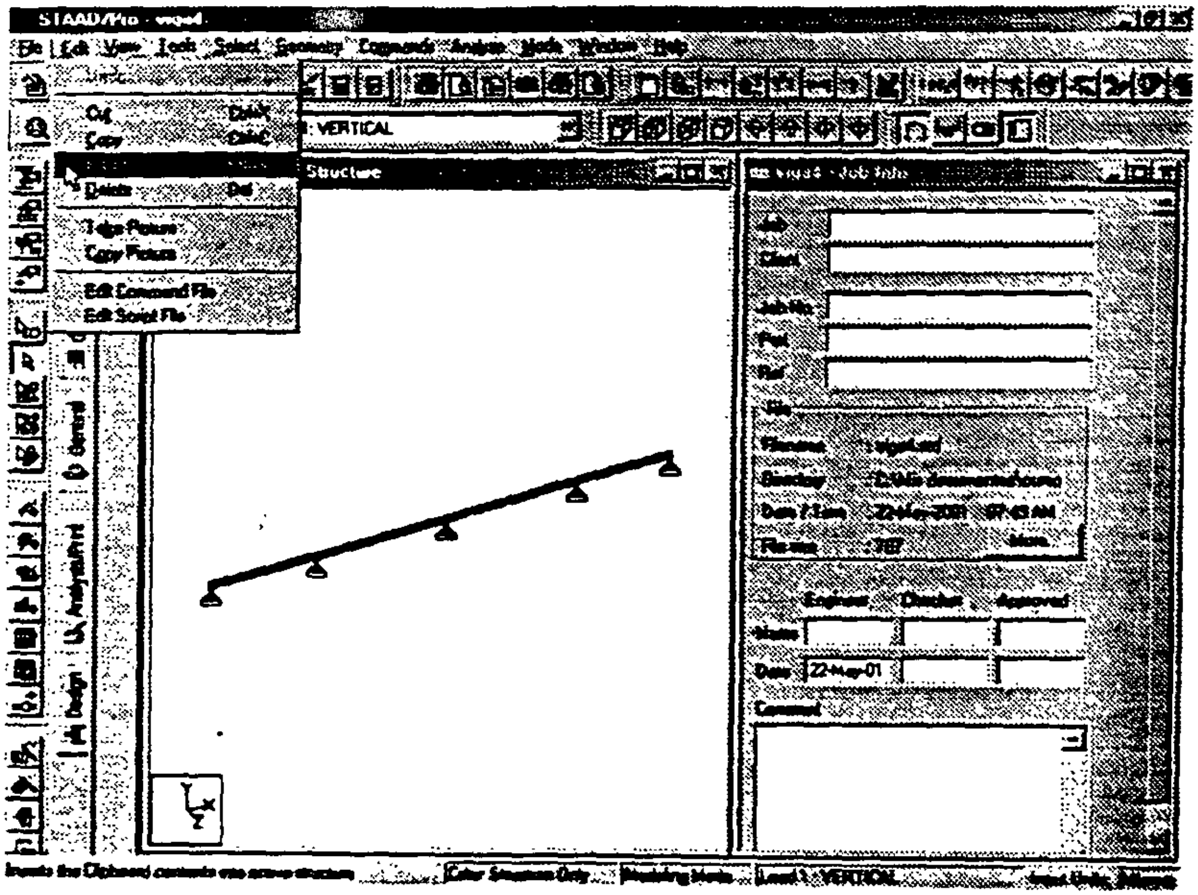


Figura 3.2 Menú **Edit** del módulo **STAAD.**

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 realizársele cambios (adicionar comandos o datos, suprimir o modificar parte de la información etc.).

Job Information

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

Structure Type	SPACE FRAME

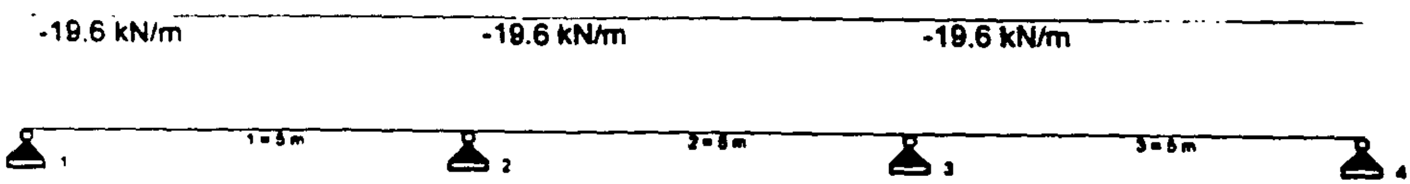
Number of Nodes	4	Highest Node	4
Number of Elements	3	Highest Beam	3

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

Included in this printout are data for:

All	The Whole Structure

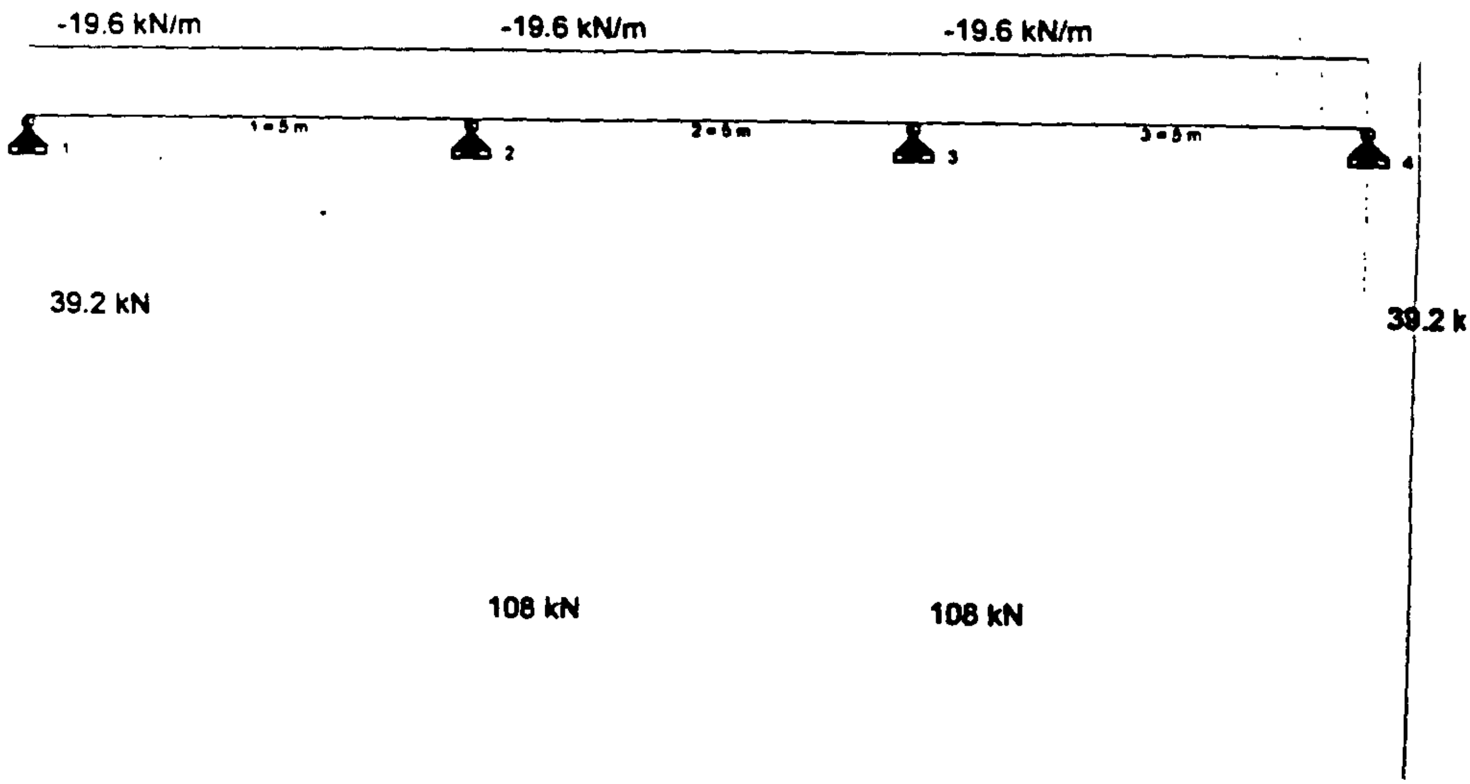
Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 04-Oct-00	Chd
Client	File V3ciapro.std	Date/Time 23-Sep-2001 22:26	

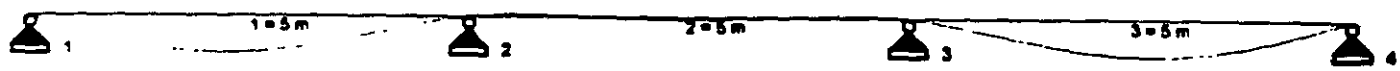


```
STAAD SPACE EJEMPLO
START JOB INFORMATION
ENGINEER DATE 04-Oct-00
END JOB INFORMATION
INPUT WIDTH 79
UNIT METER MTON
JOINT COORDINATES
1 0 0 0; 2 5 0 0; 3 10 0 0; 4 15 0 0;
MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4;
MEMBER PROPERTY AMERICAN
1 TO 3 PRIS YD 0.5 ZD 0.25
UNIT METER KN
CONSTANTS
E 2.5e+007 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

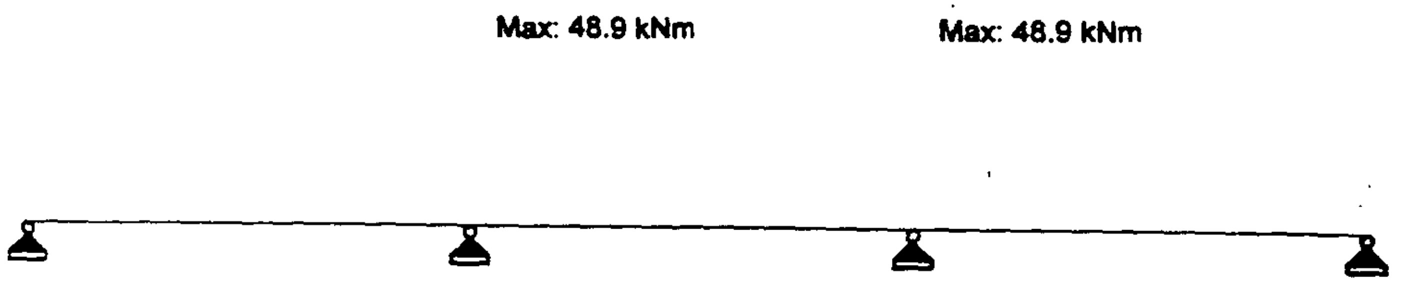
Job Title	Job No	Sheet No 1	Rev
	Part		
	Ref		
	By	Date 04-Oct-00	Chd
Client	File V3clspro.std	Date/Time 23-Sep-2001 22:41	



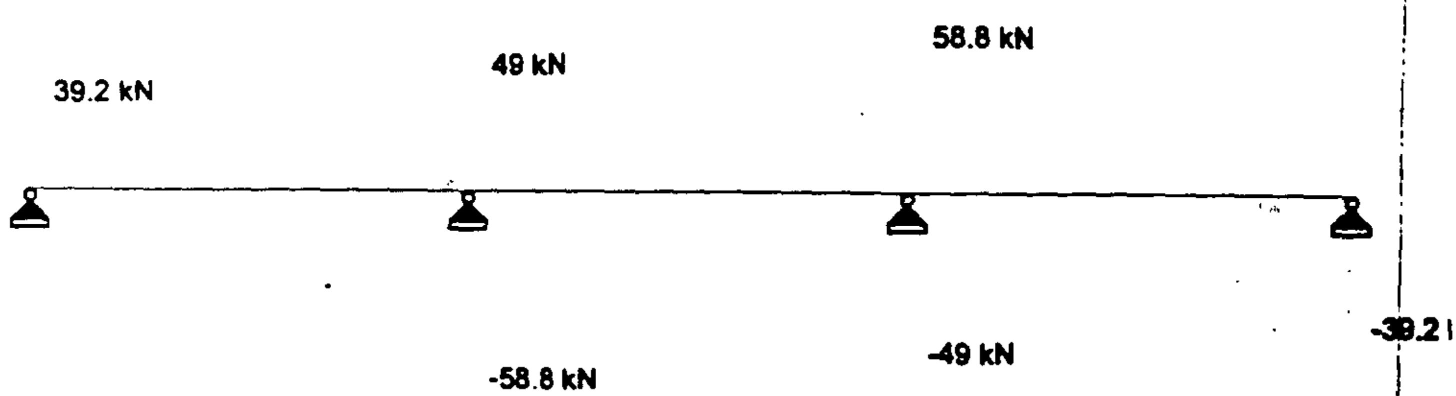


1 = 5 m
2 = 5 m
3 = 5 m
1
2
3
4

Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 04-Oct-00	Chd
Client	File V3clapro.std	Date/Time 23-Sep-2001 22:41	



Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 04-Oct-00	Chd
Client	File V3clapro.std	Date/Time 23-Sep-2001 22:41	



```
**************************************************
*                                                *
*            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-OCT-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 INTÈRNAL 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

   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  *
     ****************************************************
```

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

Beams

Beam	Node A	Node B	Length (m)	Property	β 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^2)	I_{yy} (m^4)	I_{zz} (m^4)	J (m^4)	Material
1	Rect 0.80X0.40	0.320	0.004	0.017	0.012	-

Materials

Mat	Name	E (kN/mm^2)	G (kN/mm^2)	v	Density (kg/m^3)	α ($1/^oK$)
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	26.800	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

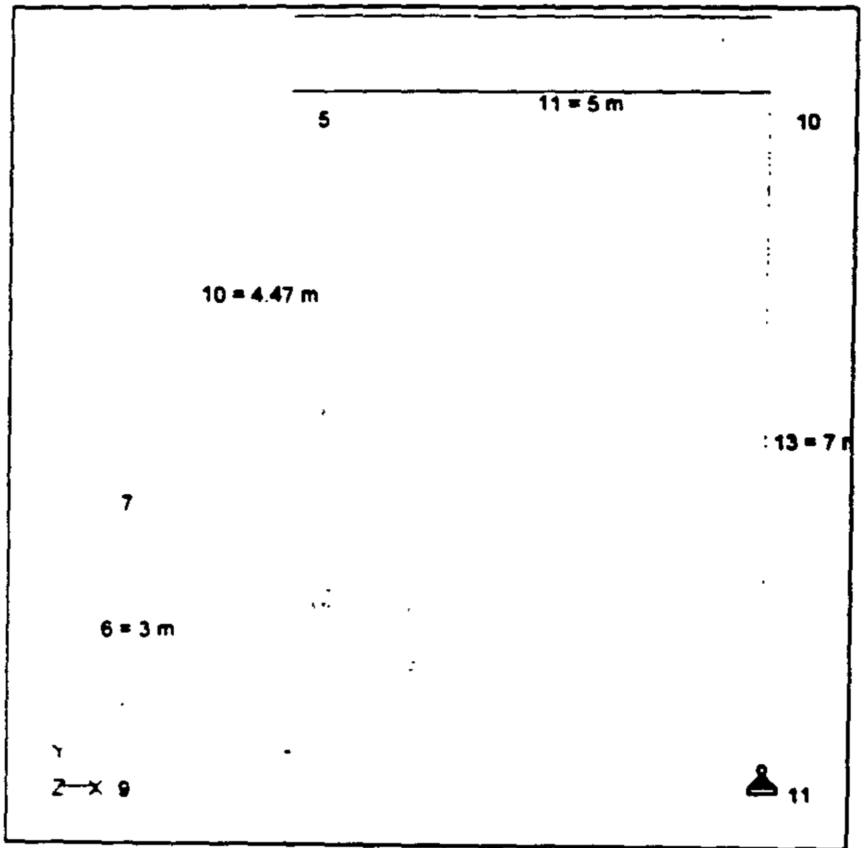
There is no data of this type.

Basic Load Cases

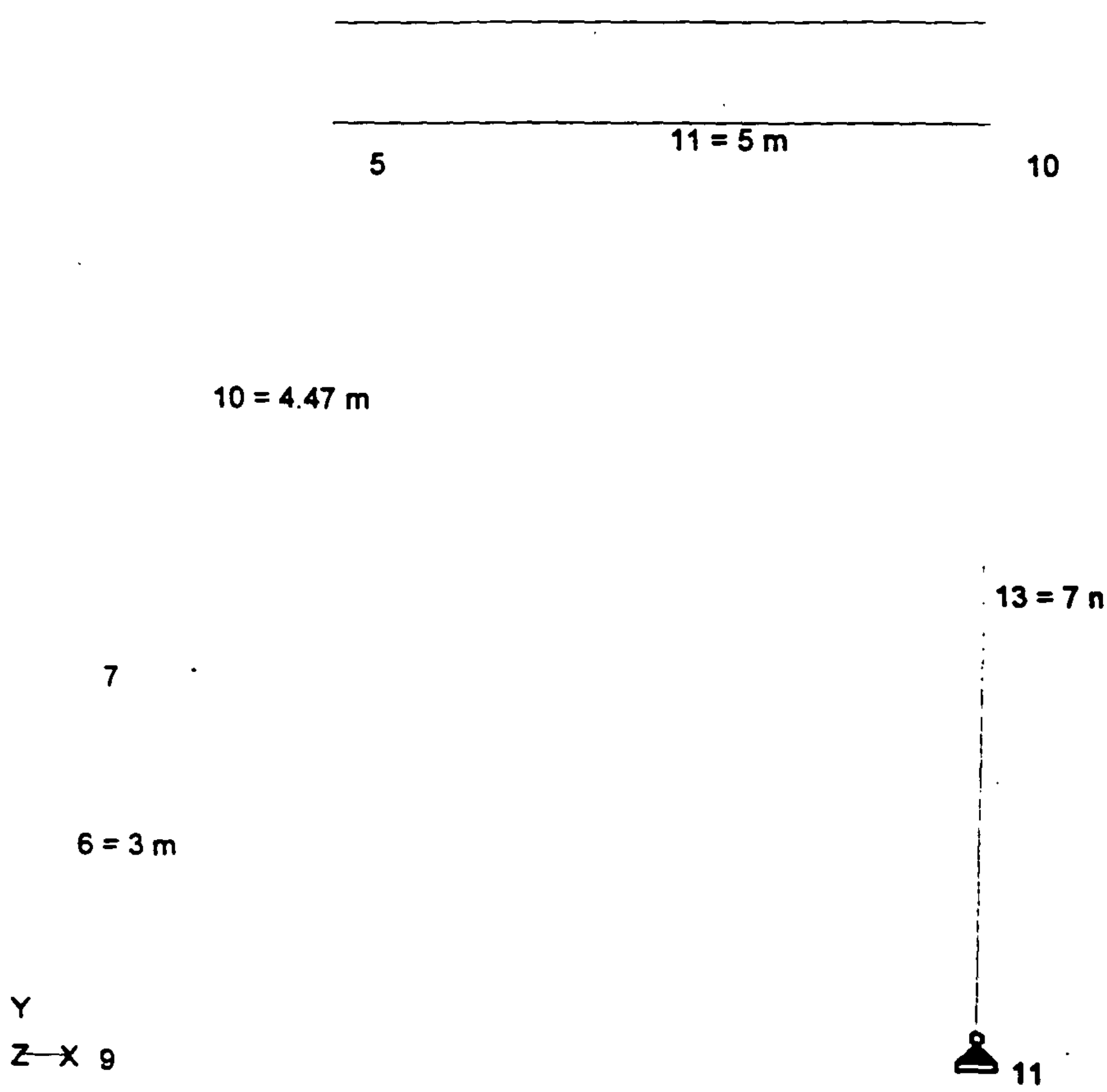
Number	Name
1	VERTICAL

Combination Load Cases

There is no data of this type.

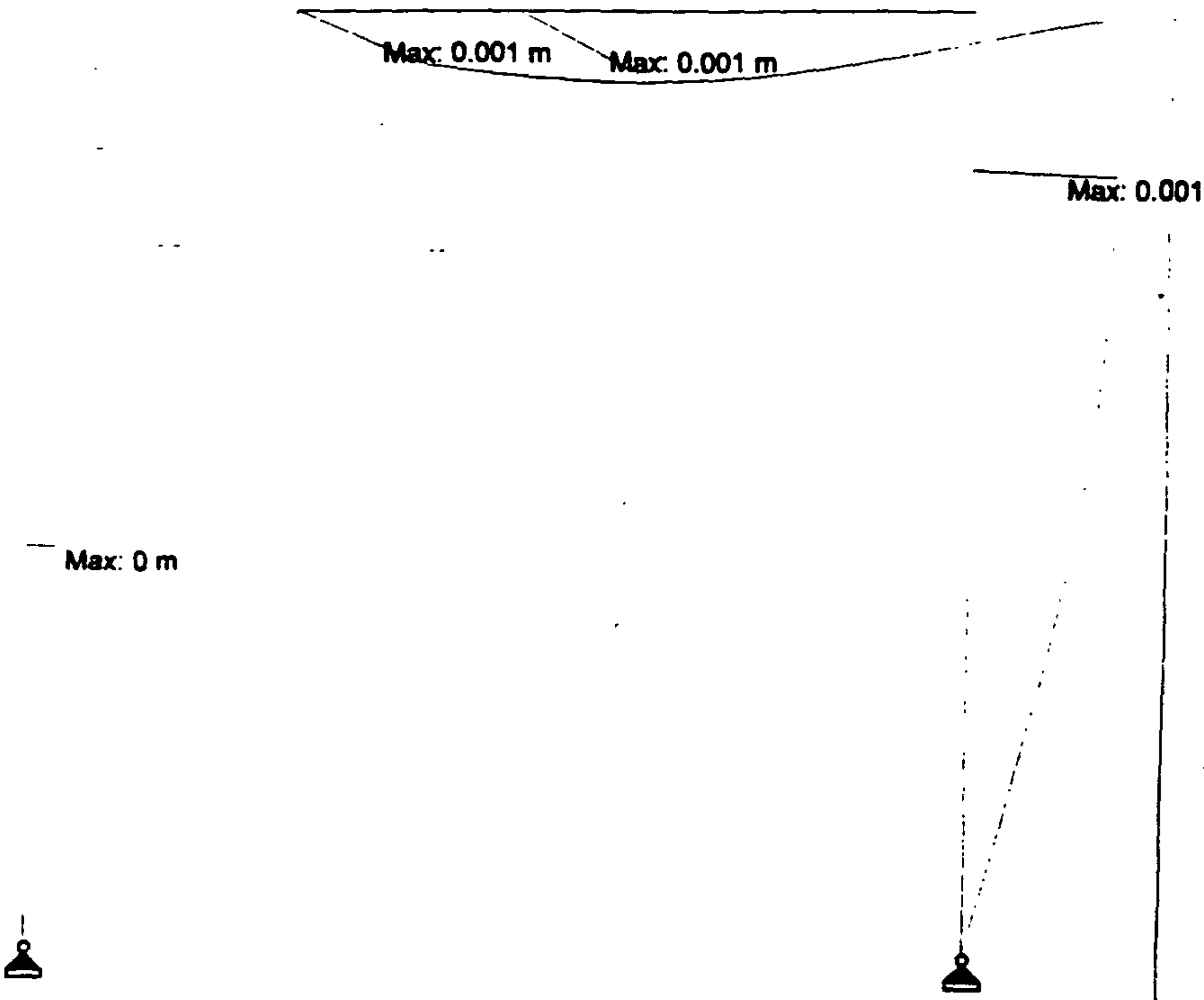


Whole Structure Loads 5kN:1m 1 VERTICAL

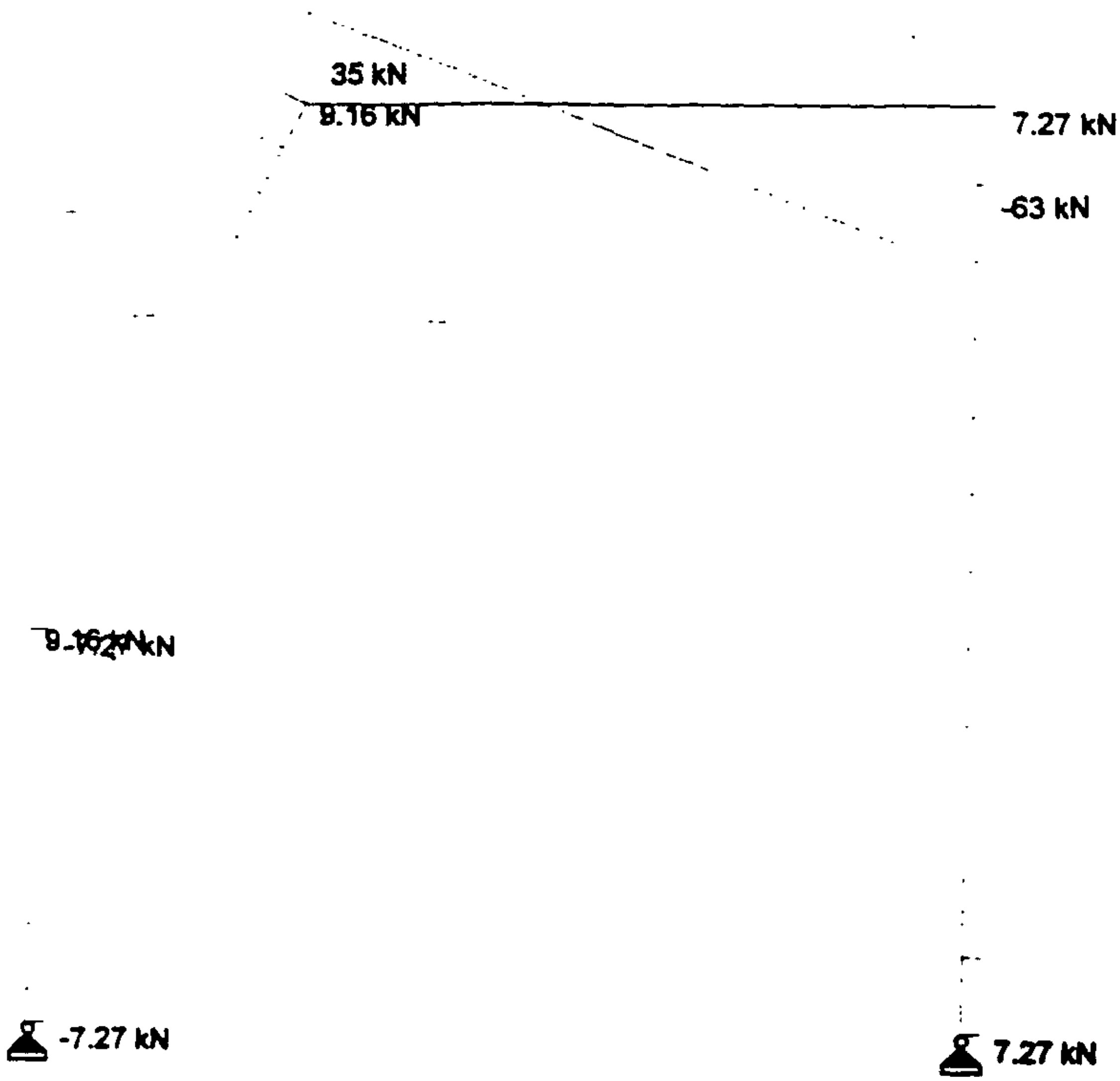


Whole Structure Loads 5kN:1m 1 VERTICAL

Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 18-May-01	Chd
Client	File marco1.std	Date/Time 18-May-2001 02:11	

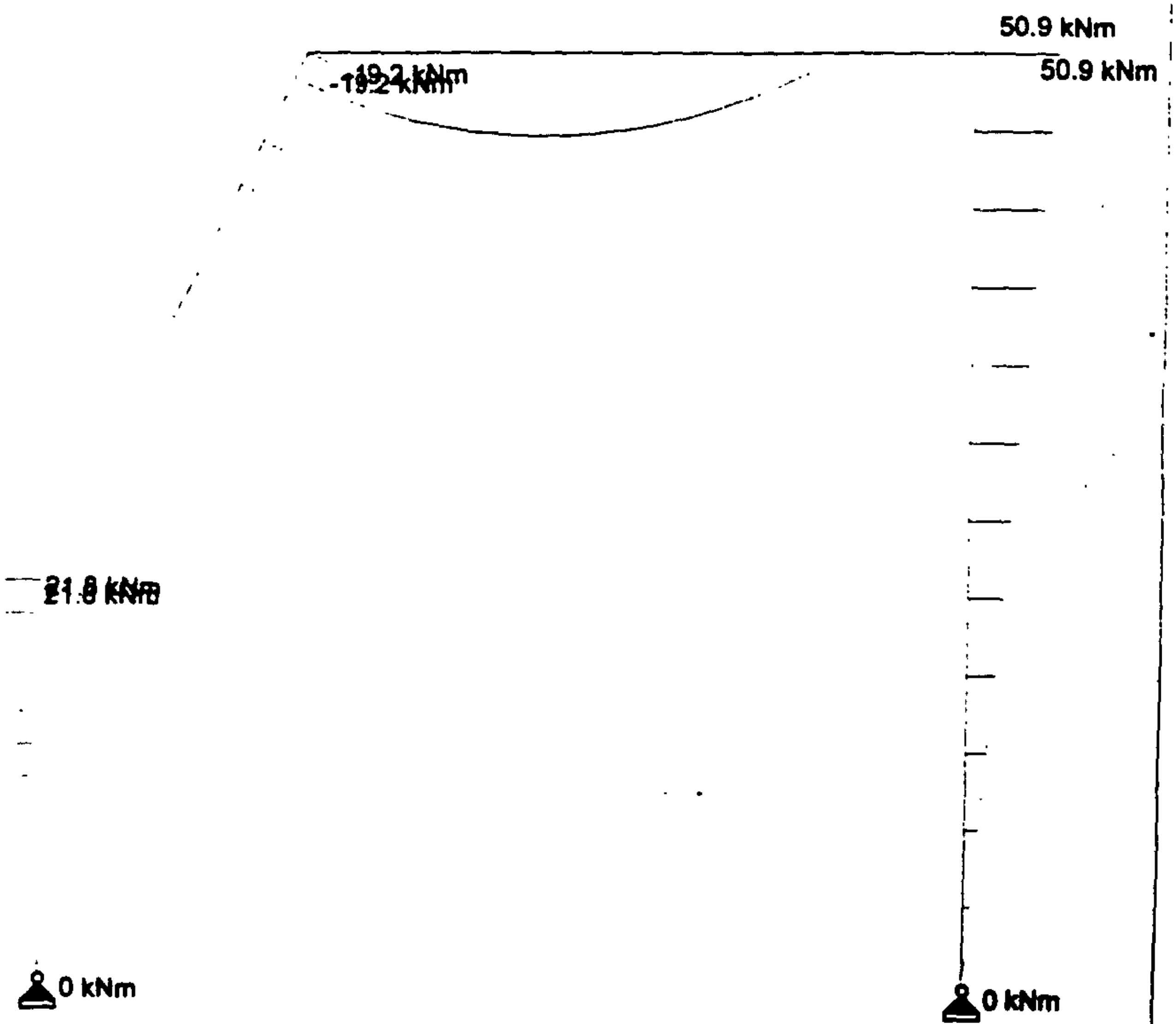


Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 18-May-01	Chd
Client	File marco1.std	Date/Time 18-May-2001 02	



Software licensed to Unknown User

Job Title	Job No		Sheet No 1		Rev
	Part				
	Ref				
	By		Date 18-May-01		Chd
Client	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 DISPLACMENTS ALL
PRINT MEMBER FORCES ALL
FINISH
```

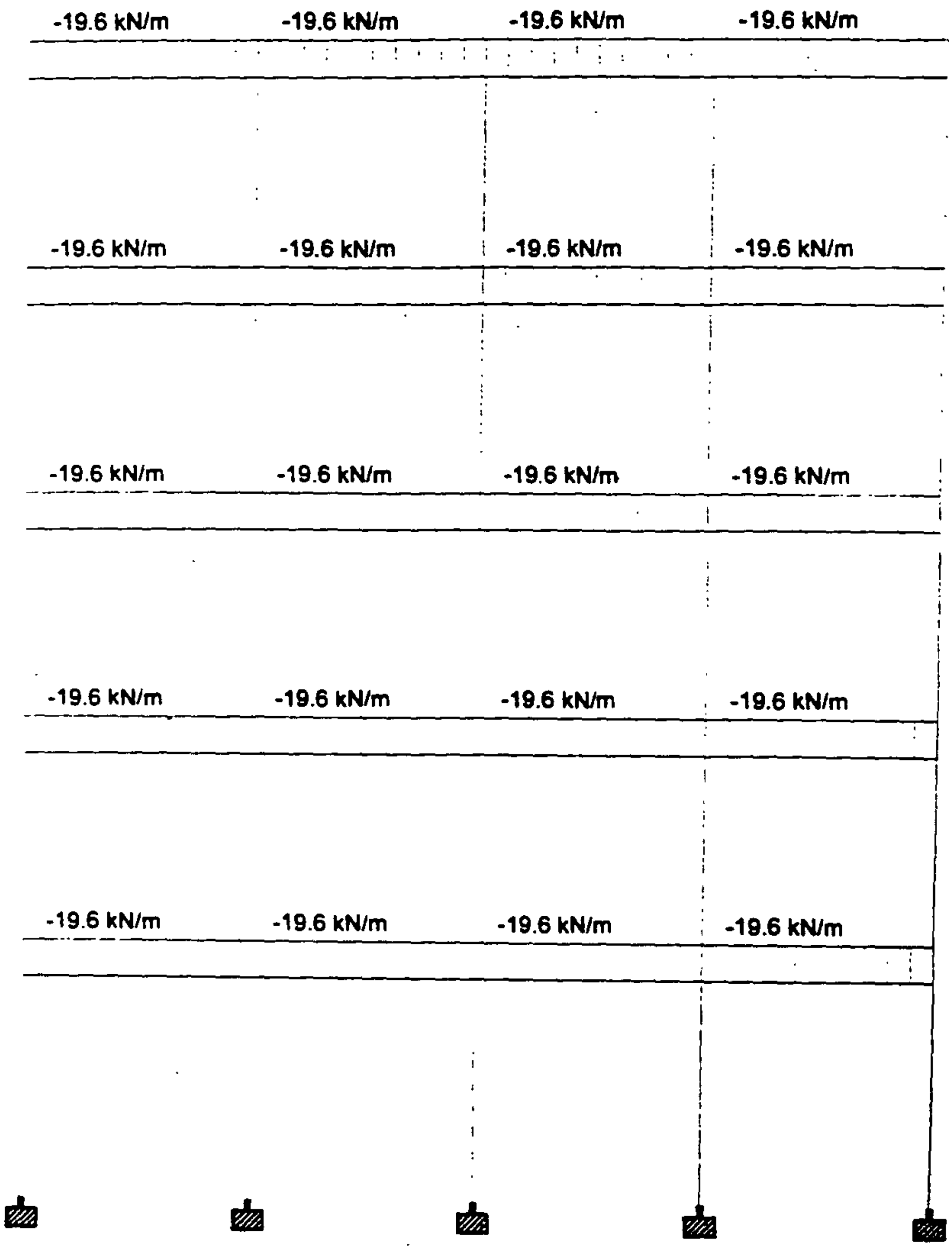
```
**************************************************
*                                                *
*            STAAD/Pro STAAD-III                 *
*            Revision 3.1                        *
*            Proprietary Program of              *
*            RESEARCH ENGINEERS, Inc.            *
*            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
```

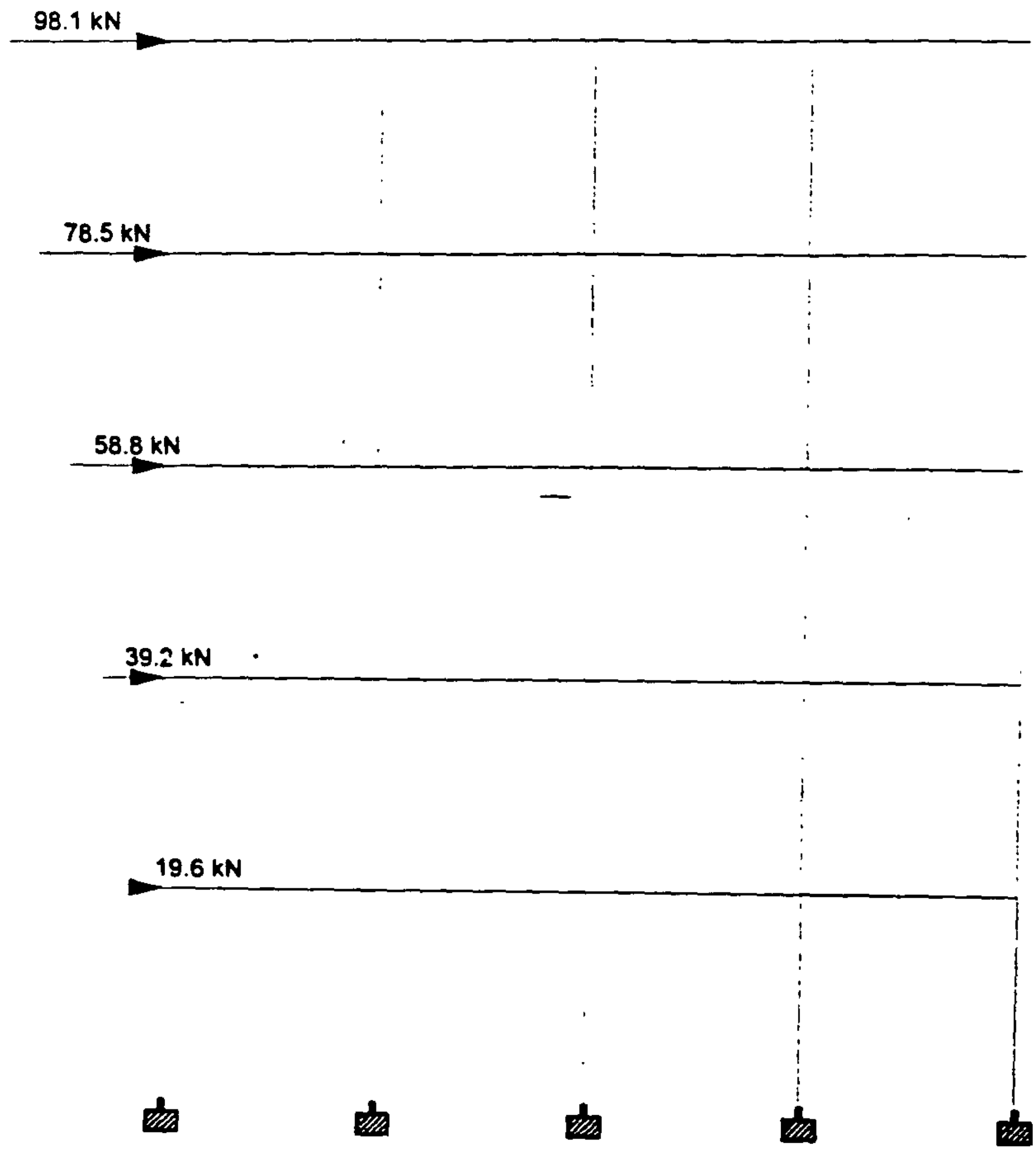
```
************** END OF LATEST ANALYSIS RESULT **************

 27. PRINT JOINT DISPLACMENTS 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  *
      ****************************************************
```

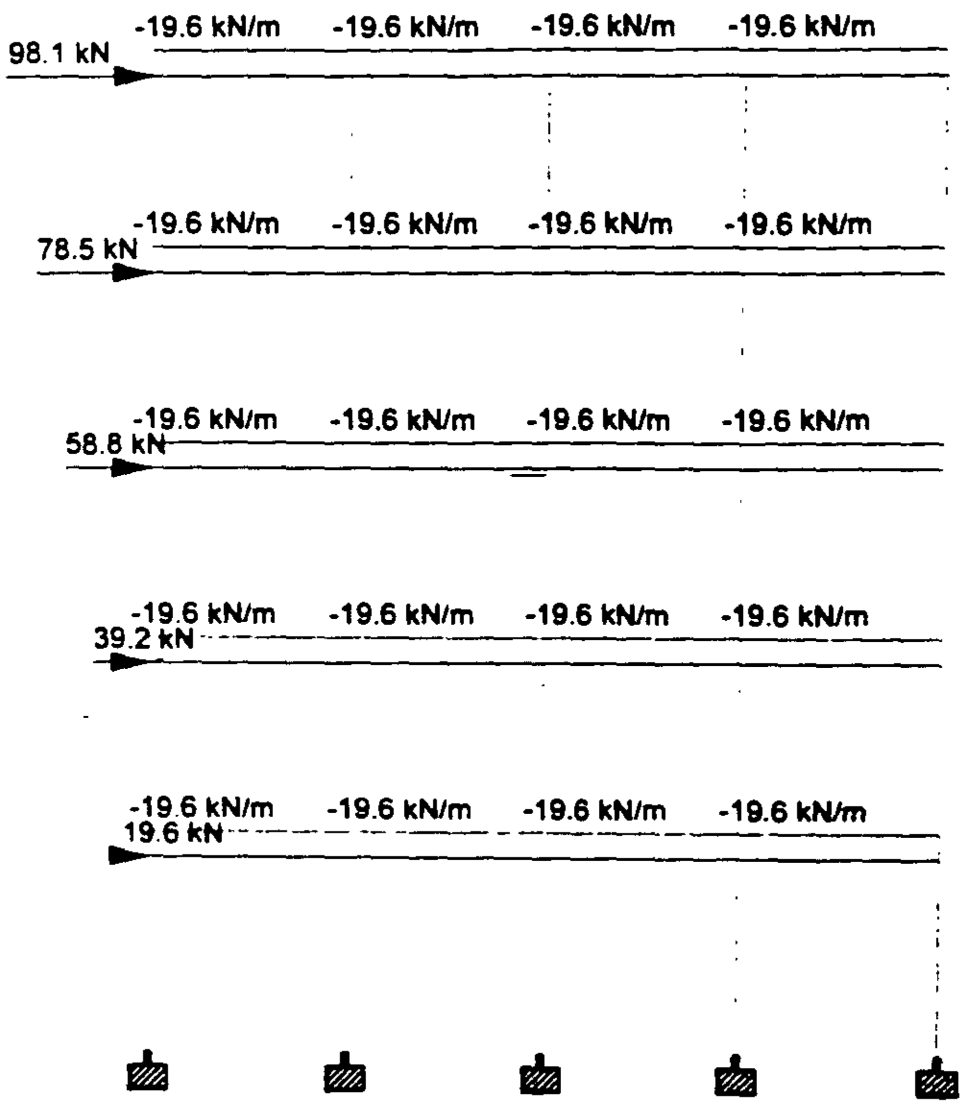
Software licensed to Unknown User

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



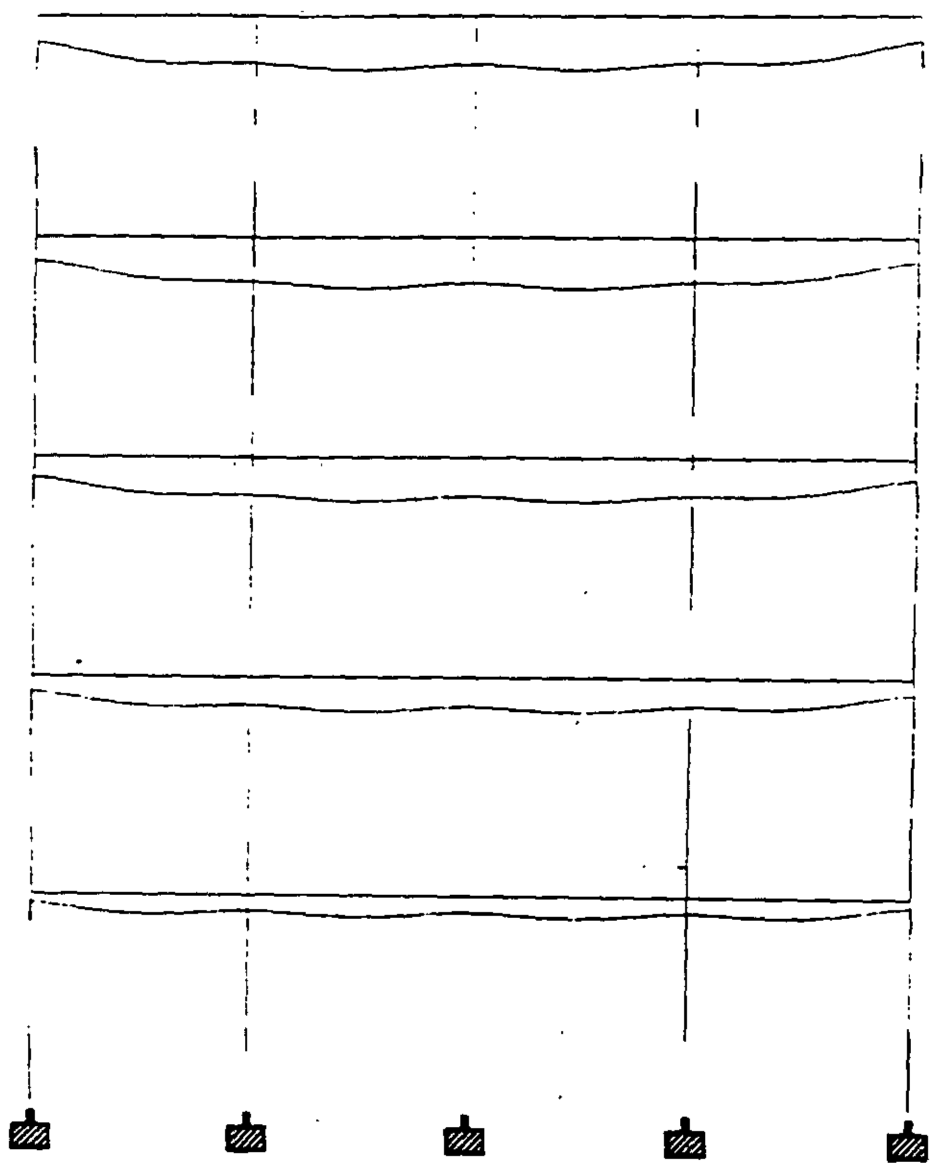
	Job No	Sheet No 1	Rev
Software licensed to Unknown User	Part		
Job Title	Ref		
	By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 31-May-2001 22:39	




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

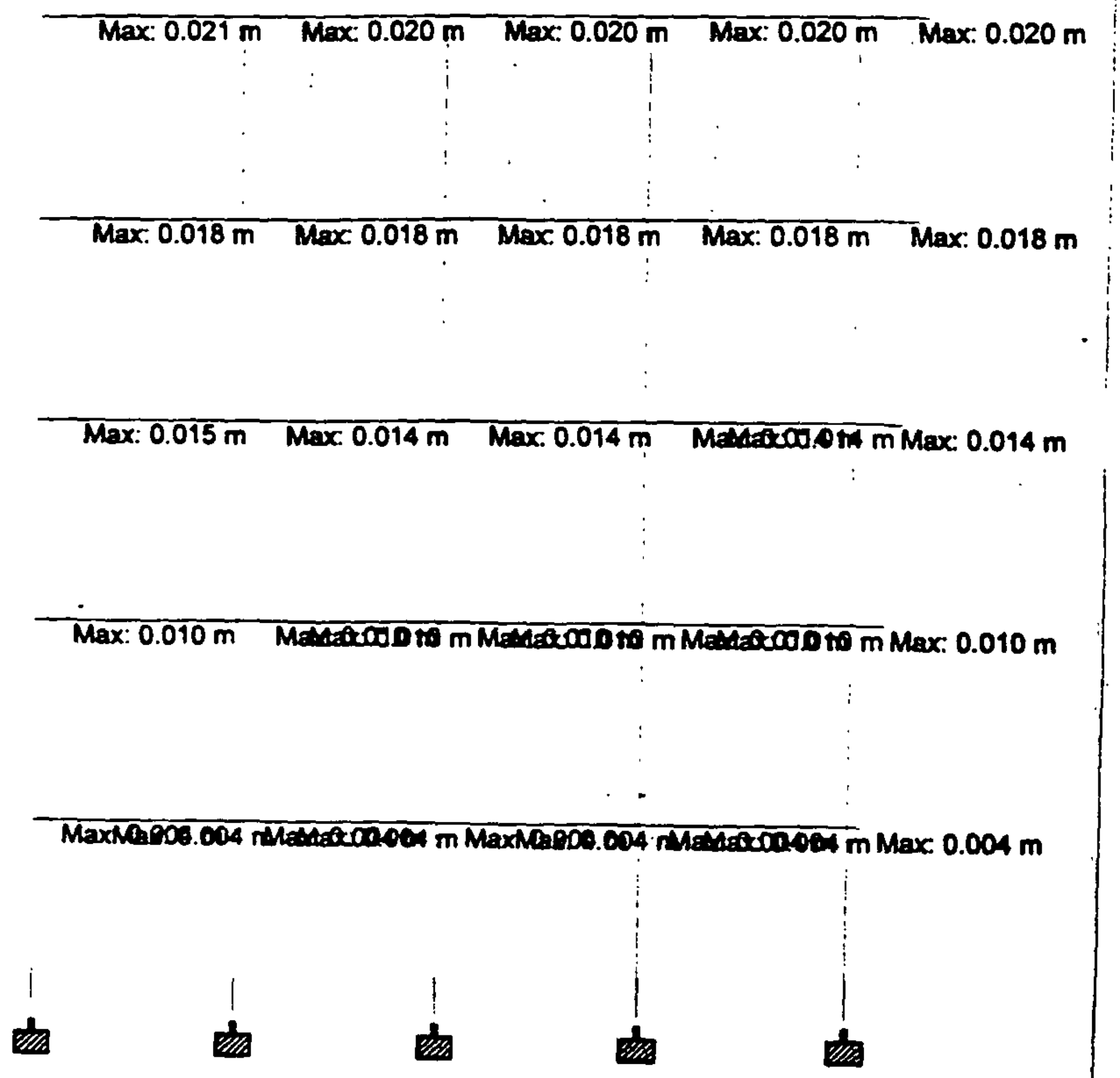
Software licensed to Unknown User

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

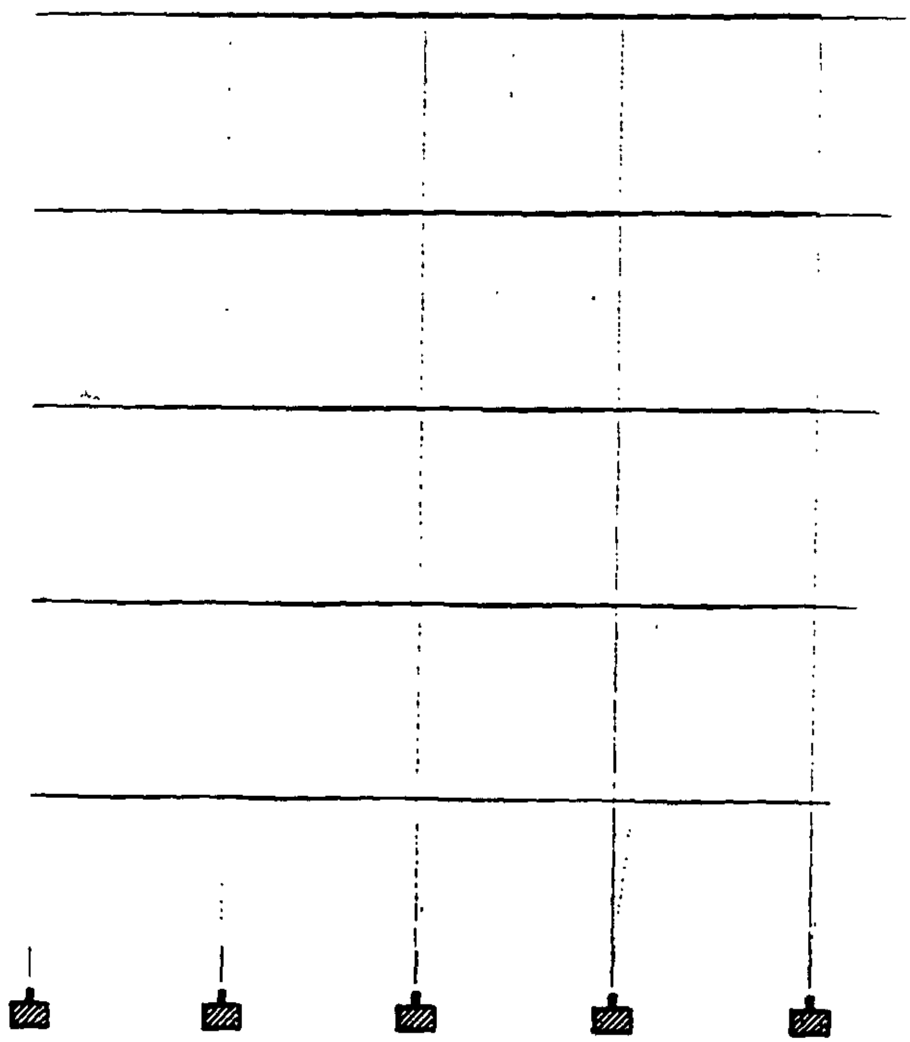


Software licensed to Unknown User

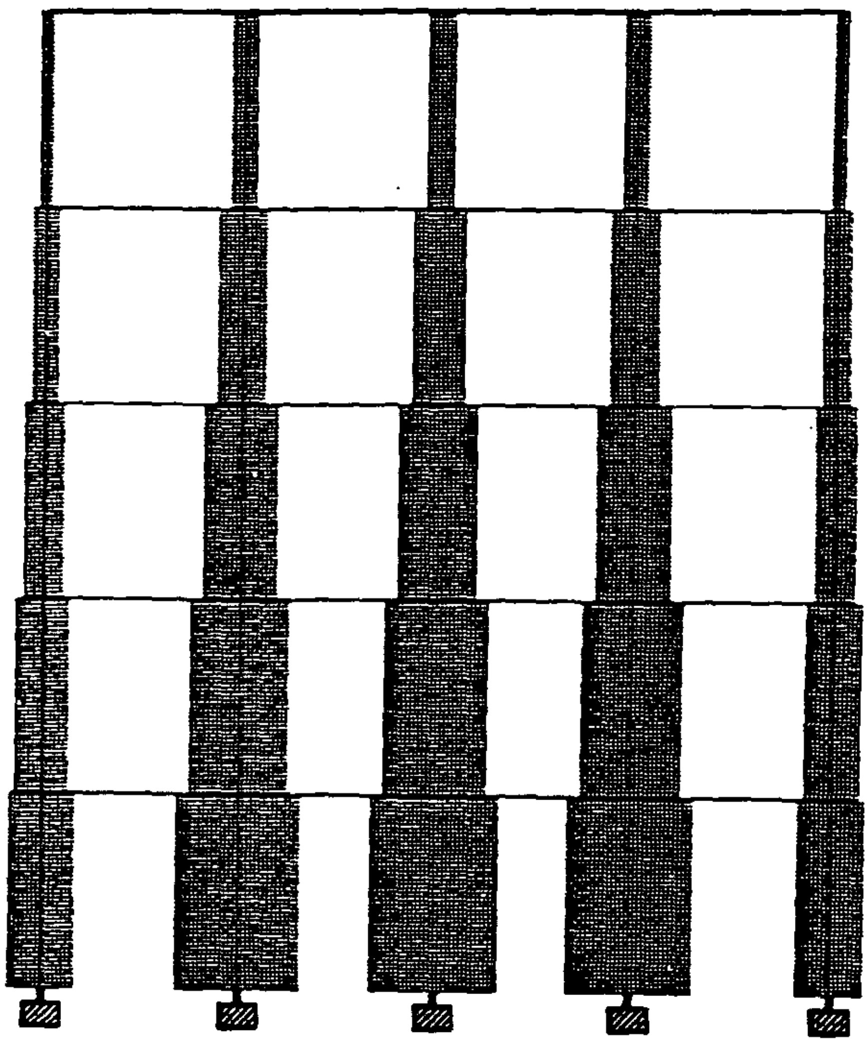
Job Title	Job No	Sheet No 1	Rev
	Part		
	Ref		
	By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55	



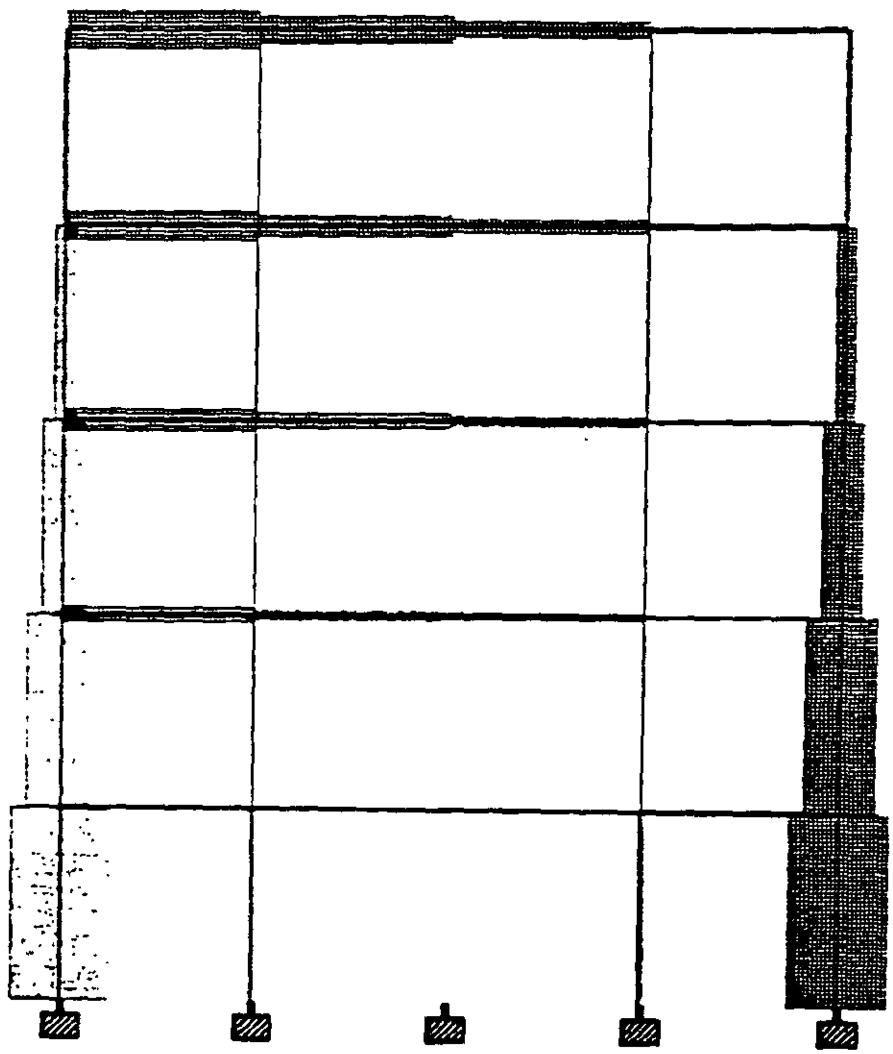
	Job No	Sheet No 1	Rev
Software licensed to Unknown User	Part		
Job Title	Ref		
	By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 24-Sep-2001 01:41	



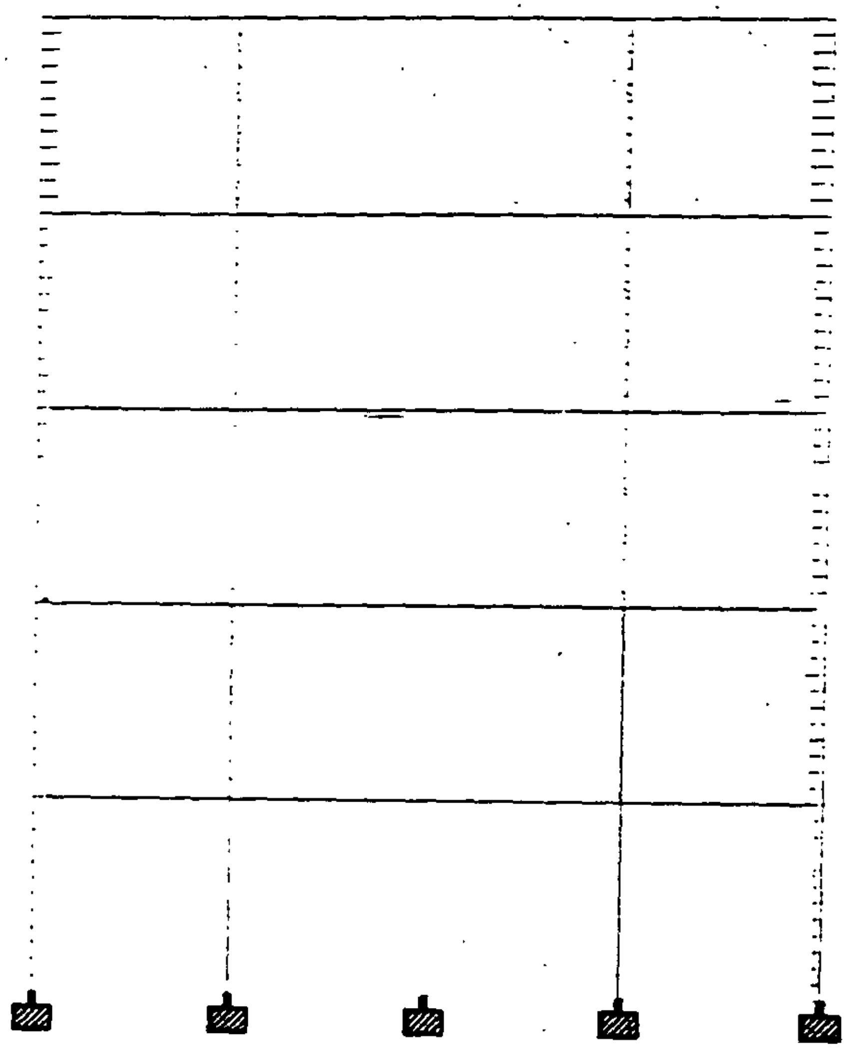
Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55	



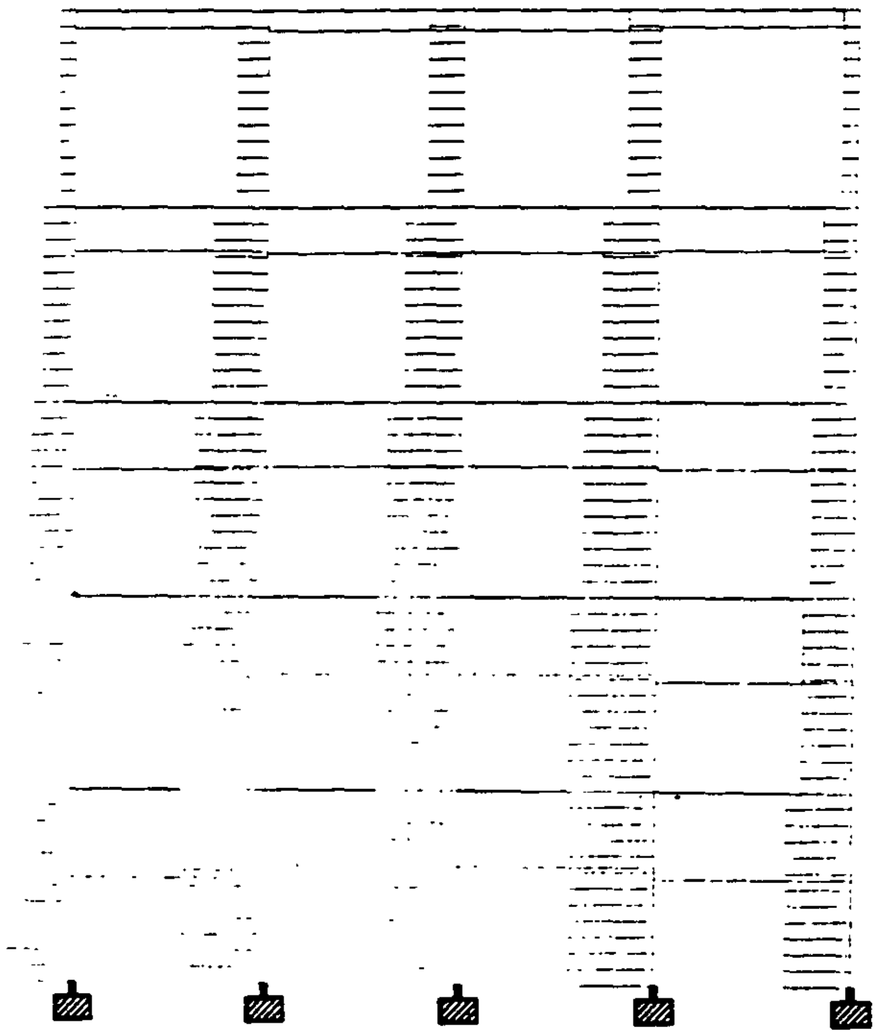
Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55	

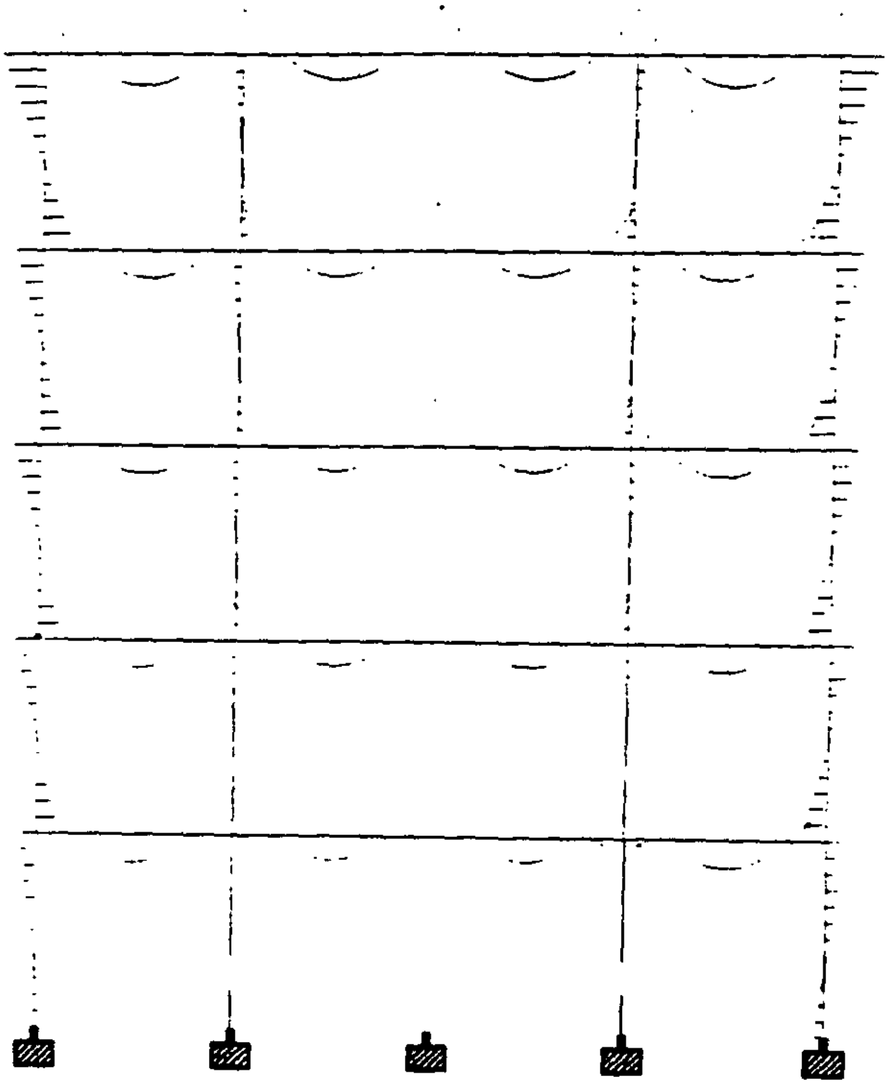


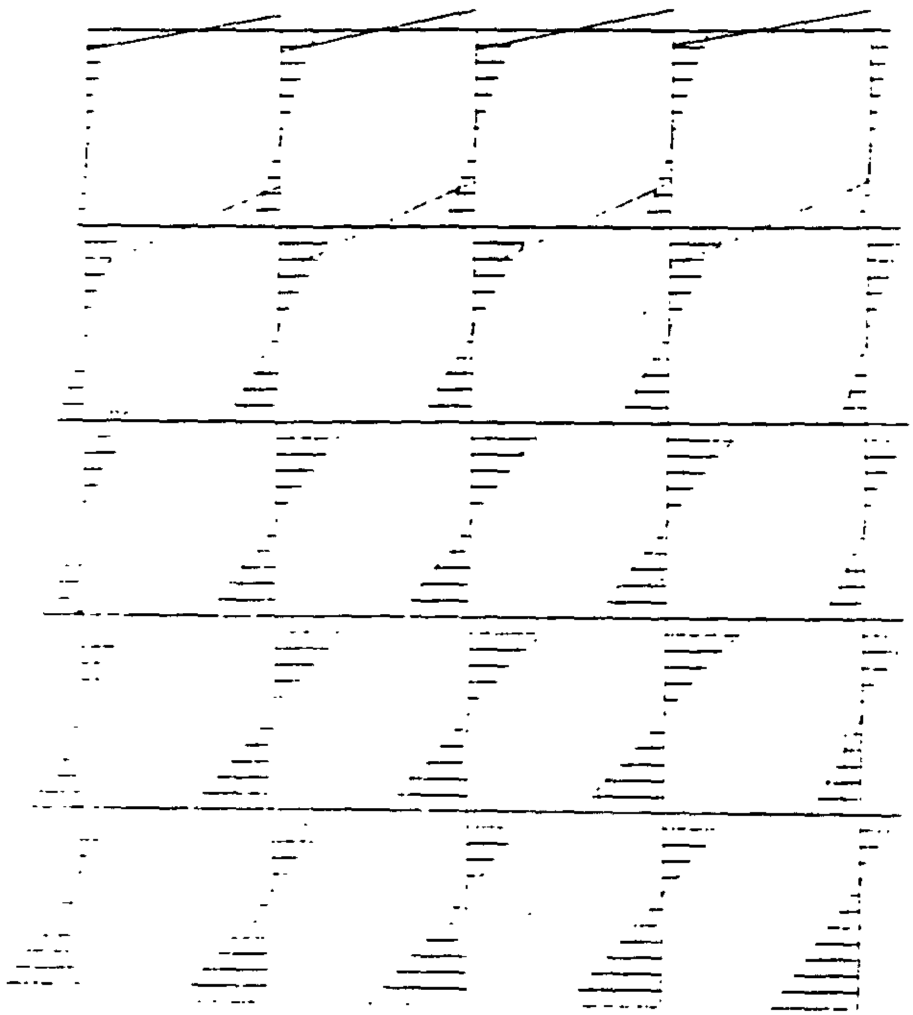
		Job No	Sheet No 1	Rev
Software licensed to Unknown User		Part		
Job Title		Ref		
		By	Date 31-May-01	Chd
Client		File marco4n.std	Date/Time 24-Sep-2001 00:55	



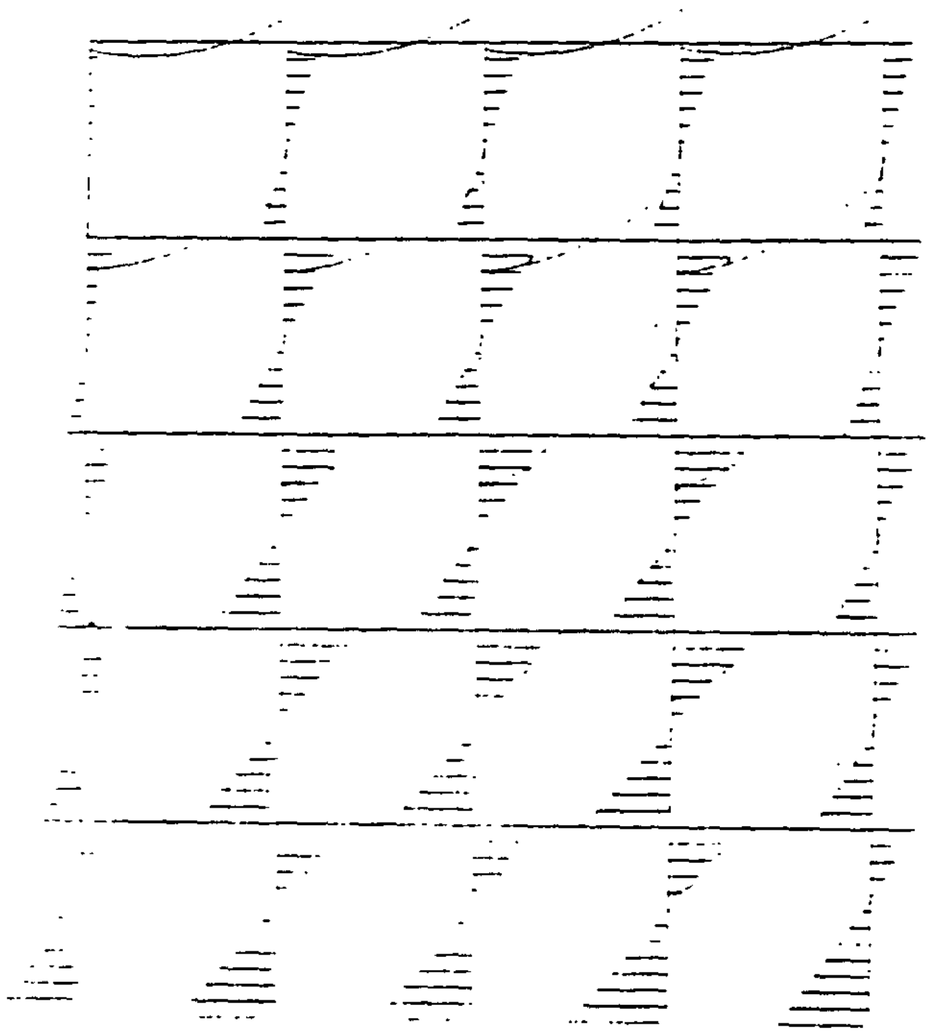
Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 31-May-01	Chd
Client	File marcob4n.std	Date/Time 24-Sep-2001 00:55	

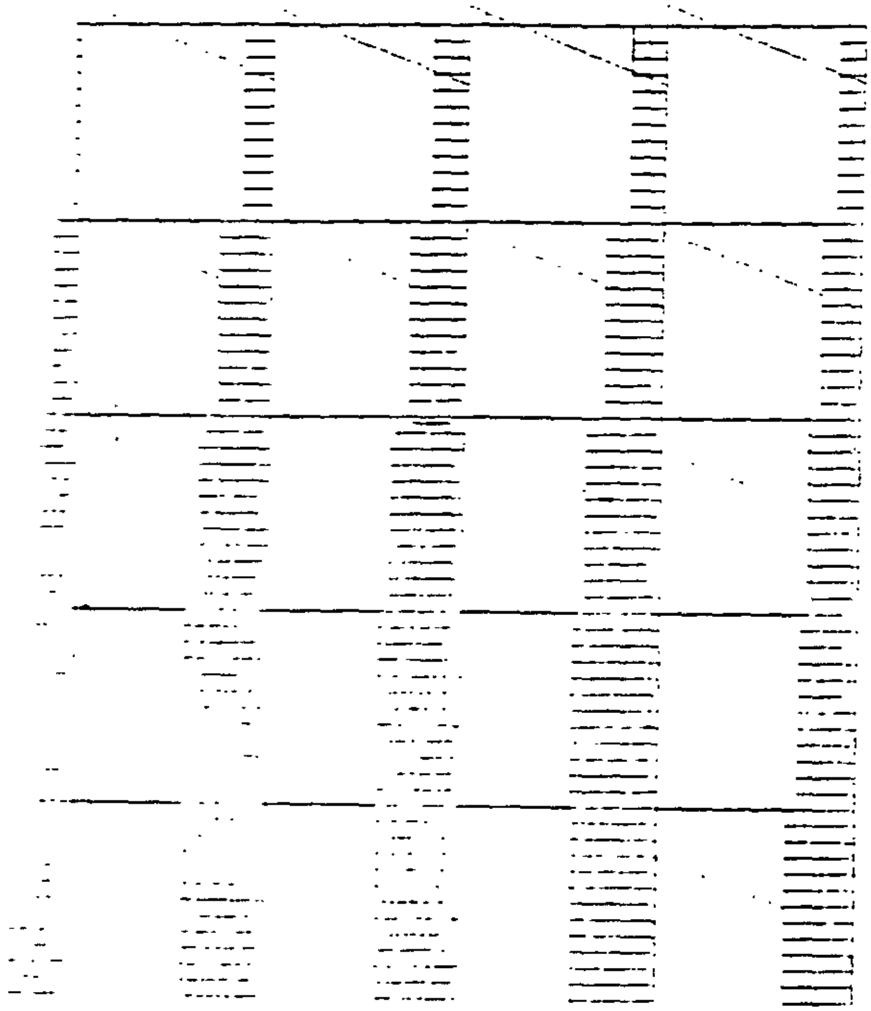






Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 31-May-01	Chd
Client	File marco4n.std	Date/Time 24-Sep-2001 00:55	





```
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
```

C:\Mis documentos\cursos\stadpro\ejemplos\marco4n.std

```
                                                                  PAGE NO.    1
               ****************************************************
               *                                                  *
               *           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)

    VIGA OCHO CLAROS                                          -- PAGE NO.    2
```

```
42. 1 1.0 2 1.0
43. 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 =    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
```



```
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


```
***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

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

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

VIGA OCHO CLAROS -- PAGE NO. 7

LOAD COMBINATION NO. 3
COMBINACIÓN (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

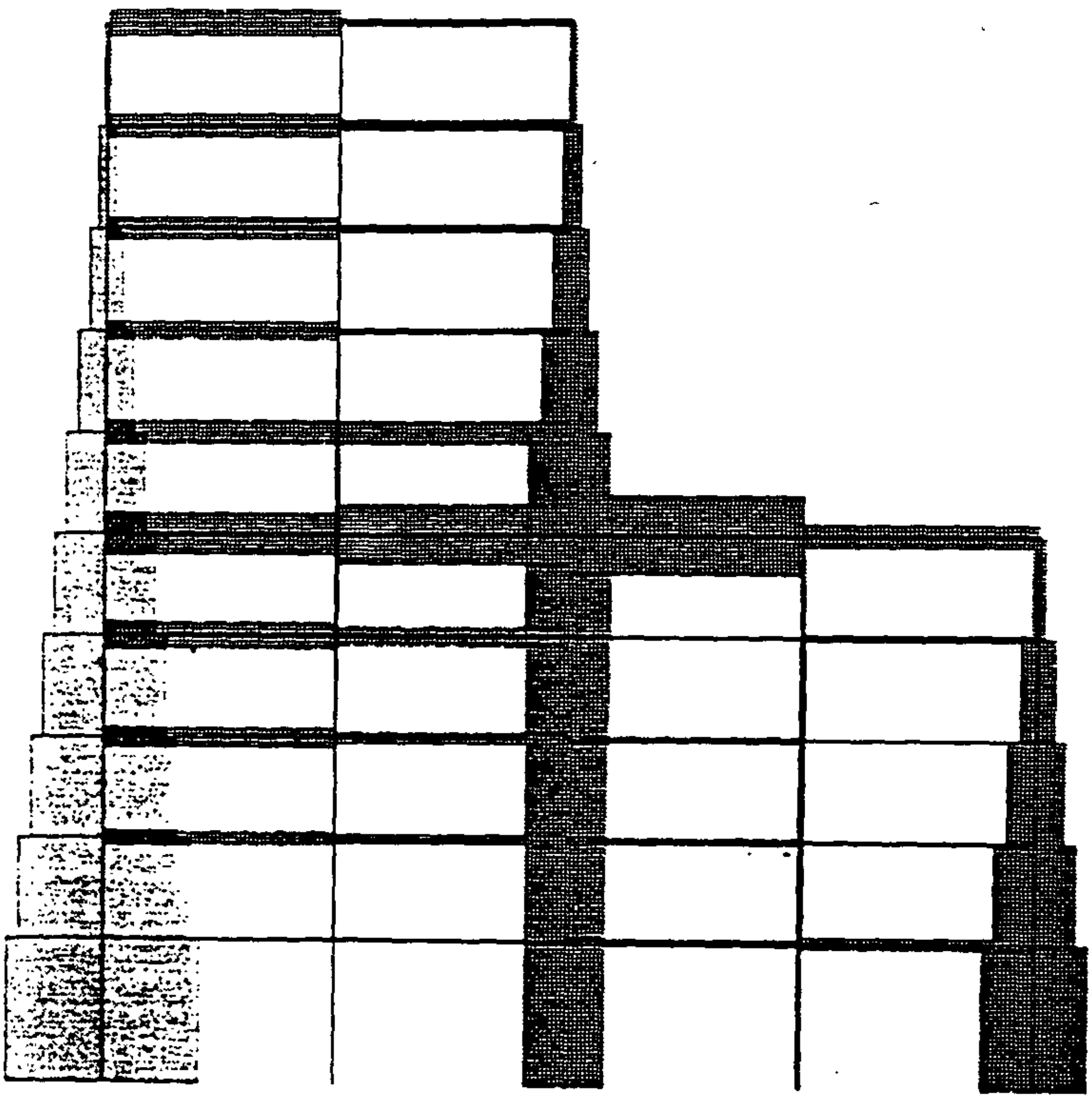
JOINT	LOAD	FORCE-X	FORCE-Y	FORCE-Z	MOM-X	MOM-Y	MOM Z
26	1	0.27	15.62	0.00	0.00	0.00	-0.27
	2	-5.13	-23.71	0.00	0.00	0.00	10.02
	3	-4.86	-8.09	0.00	0.00	0.00	9.75
27	1	0.01	29.30	0.00	0.00	0.00	-0.02
	2	-6.64	1.38	0.00	0.00	0.00	11.49
	3	-6.63	30.67	0.00	0.00	0.00	11.48
28	1	0.00	30.17	0.00	0.00	0.00	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 **************

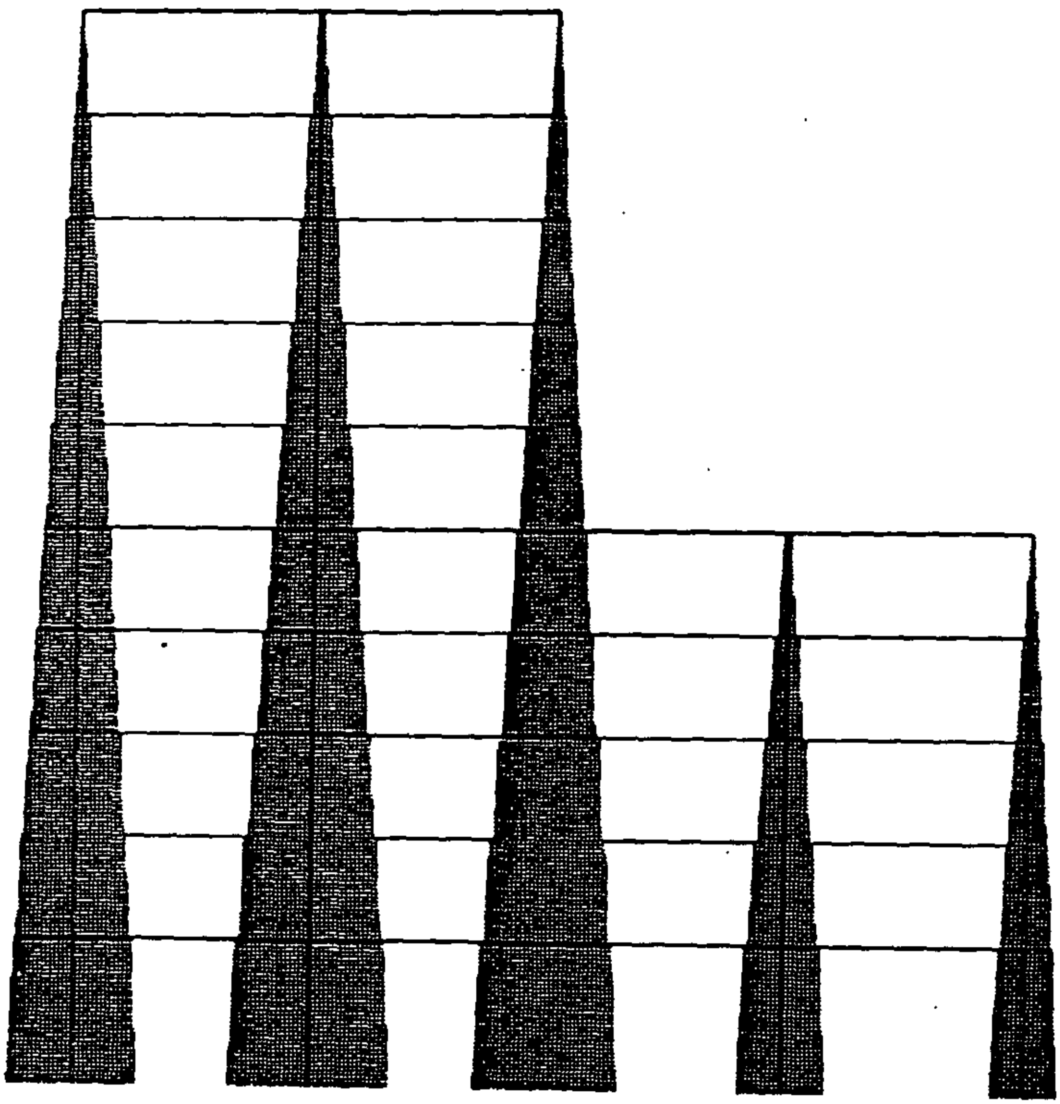
45. FINISH

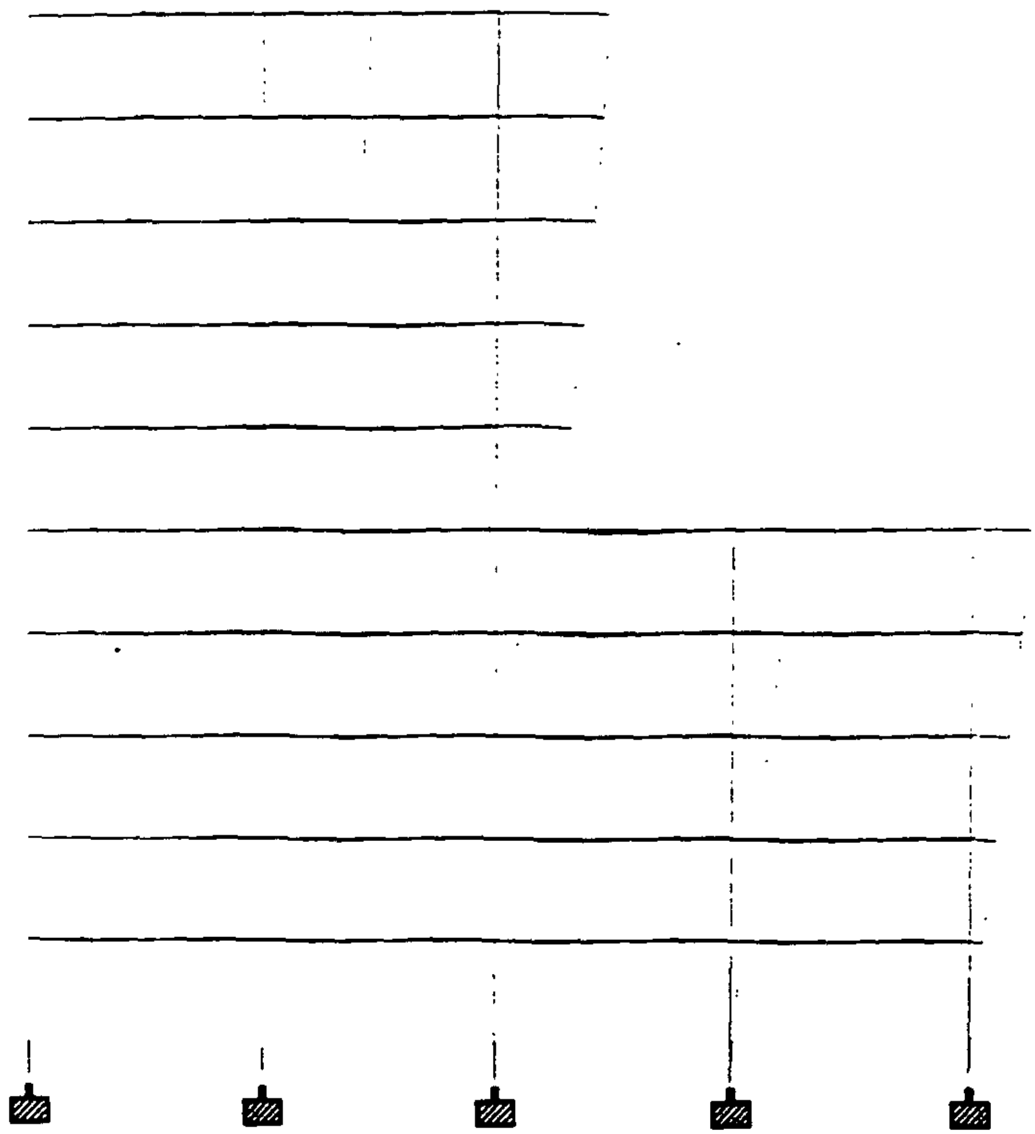
*************** END OF STAAD-III ***************
**** DATE= SEP 24,2001 TIME= 1:29:31 ****

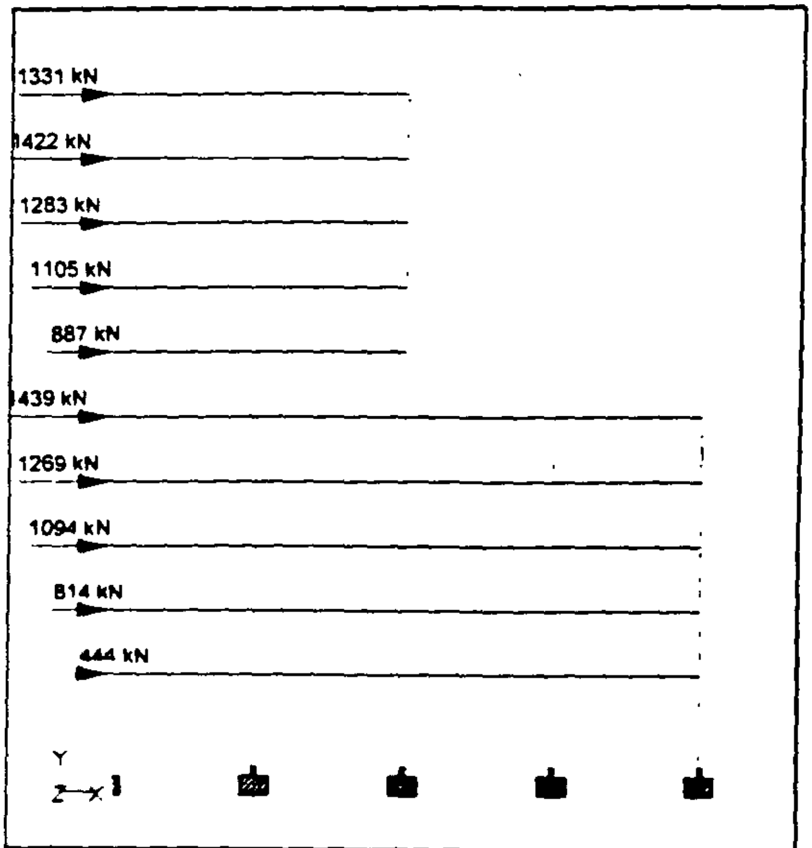
```
*********************************************************
*  FOR  QUESTIONS REGARDING THIS VERSION OF PROGRAM     *
*              RESEARCH ENGINEERS, Inc at               *
*  West Coast: Ph- (714) 974-2500  Fax- (714) 921-2543  *
*  East Coast: Ph- (978) 688-3636  Fax- (978) 685-7230  *
*********************************************************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



Software licensed to Unknown User	Job No	Sheet No 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	







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

Section Properties

Prop	Section	Area (m^2)	I_{yy} (m^4)	I_{zz} (m^4)	J (m^4)	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^2)	G (kN/mm^2)	v	Density (kg/m^3)	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.

Beams Cont...

Beam	Node A	Node B	Length (m)	Property	β 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

Beams

Beam	Node A	Node B	Length (m)	Property	β 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

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

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

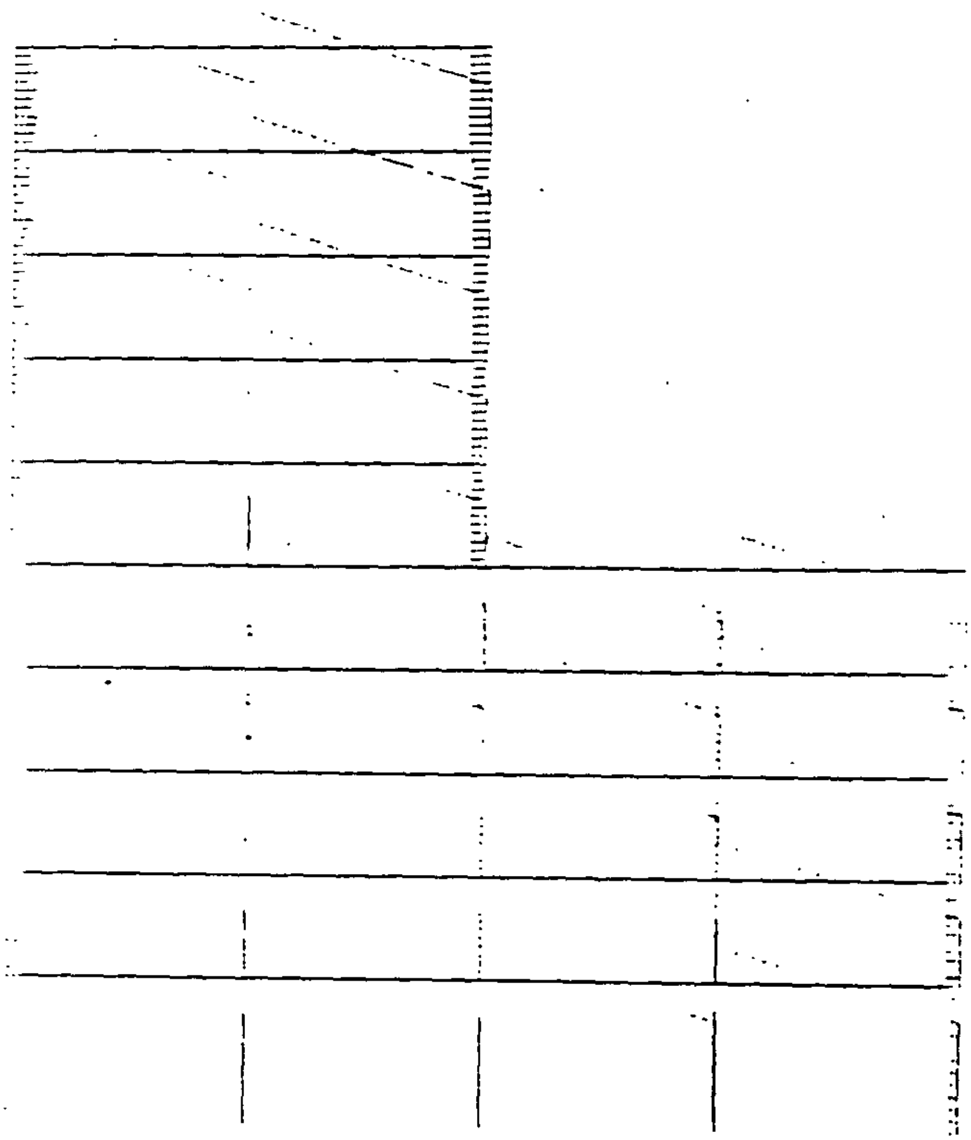
Included in this printout are results for load cases:

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 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	



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

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

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
36	112.70	0.00	0.00	0.00	0.00	0.00
	-112.70	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
41	130.85	0.00	0.00	0.00	0.00	0.00
	-130.85	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
46	145.00	0.00	0.00	0.00	0.00	0.00
	-145.00	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
51	135.70	0.00	0.00	0.00	0.00	0.00
	-135.70	0.00	0.00	0.00	0.00	0.00
52	0.00	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	0.00	0.00	0.00	0.00
53	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 DISPLACMENTS ALL

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

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	0.00	0.00	0.00	0.00	0.00
	195.75	845.84	0.00	0.00	0.00	-838.75
2	0.00	0.06	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

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

P R O B L E M   S T A T I S T I C S

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

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 RIGIDECES

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

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                       *
****************************************************
STAAD PLANE MARCOS TIPO EN X (PARA TORSIN, 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
```

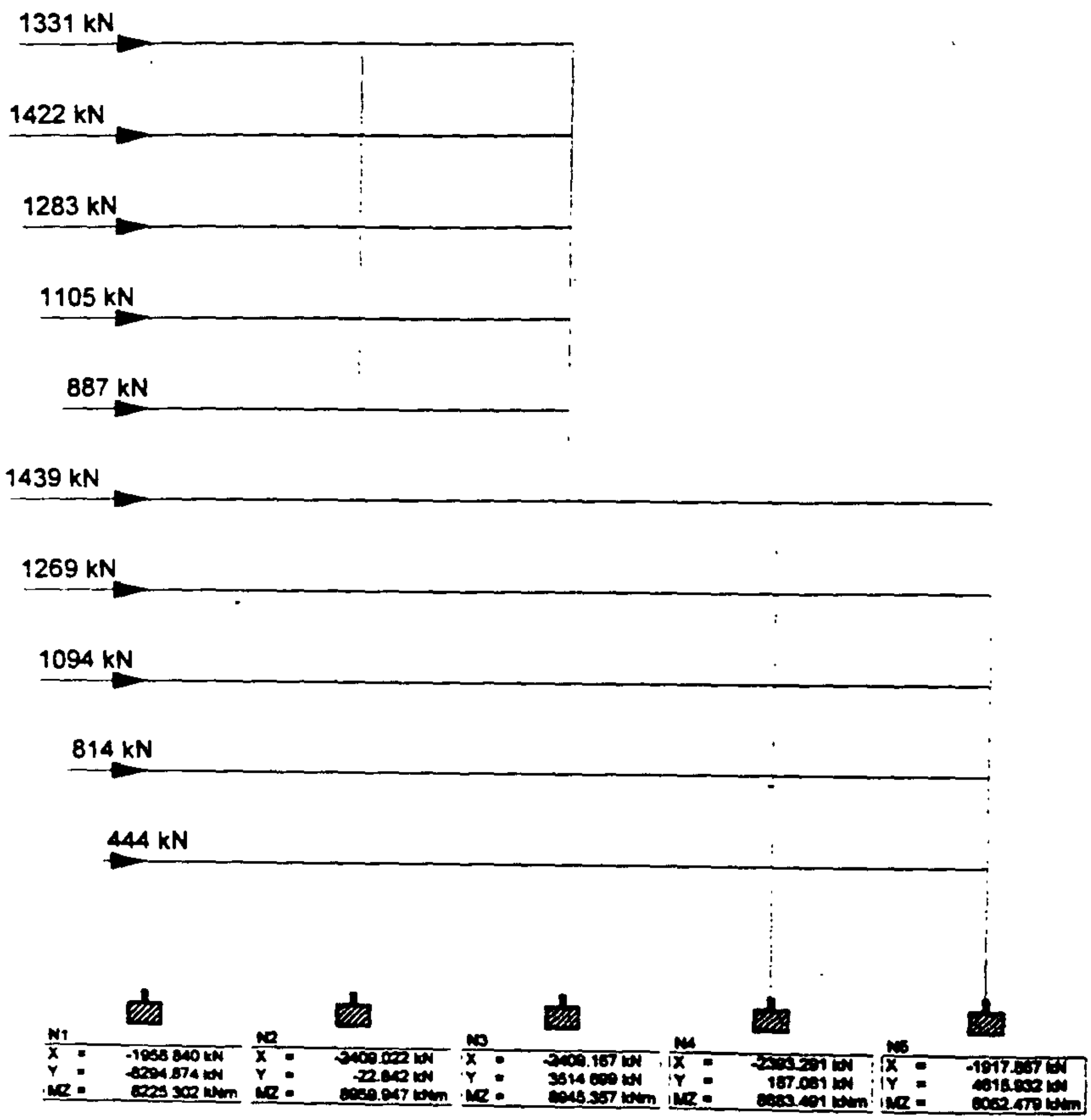


```
SELFWEIGHT Y -1
LOAD 2 FUERZAS LATERALES PARA RIGIDECES
JOINT LOAD
51 FX 135.7
46 FX 145
41 FX 130.85
36 FX 112.7
31 FX 90.45
26 FX 146.7
21 FX 129.4
16 FX 111.6
11 FX 83
6 FX 45.25
PERFORM ANALYSIS PRINT ALL
```

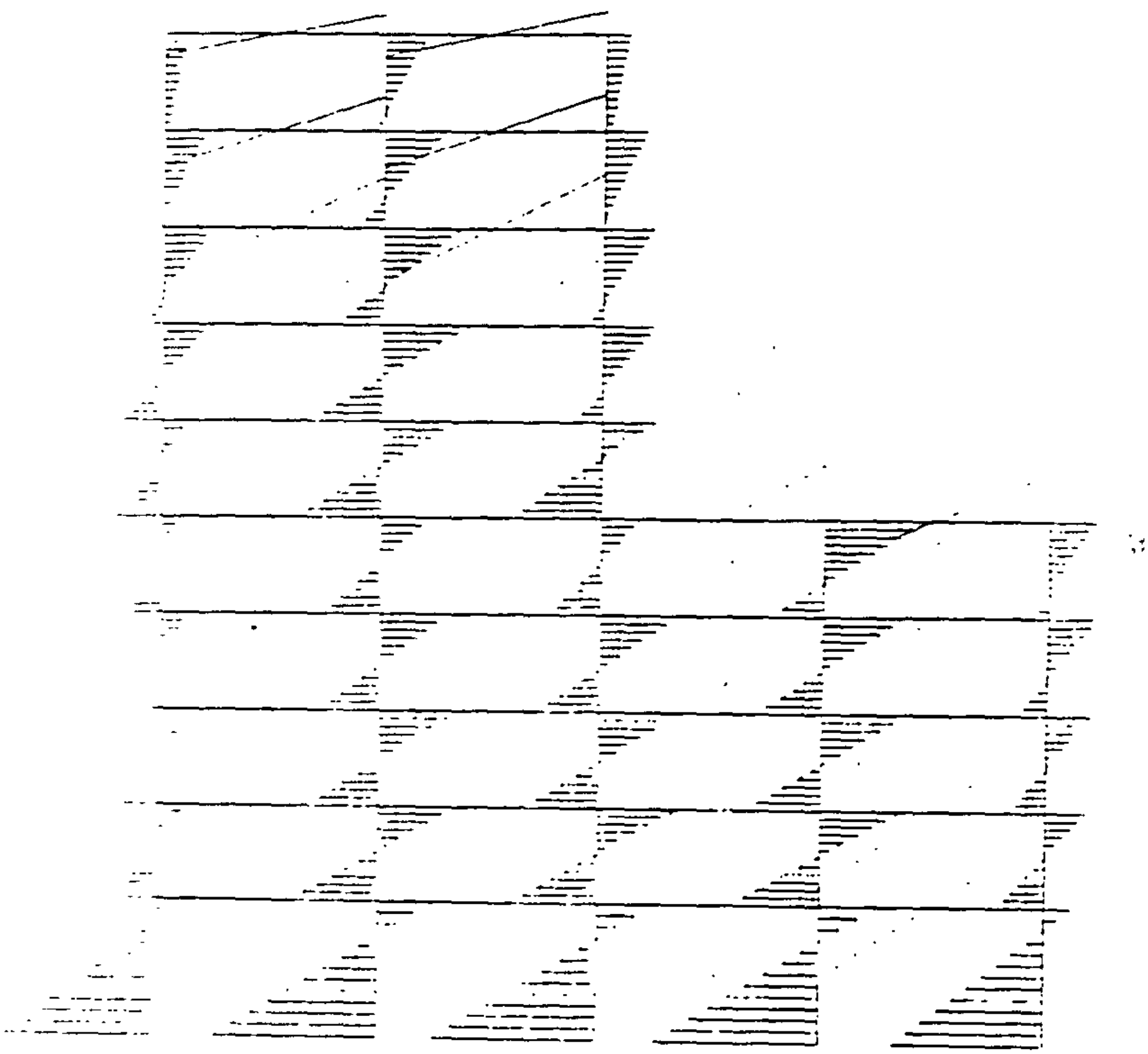
```
STAAD PLANE MARCOS TIPO EN X (PARA TORSIÓN, FMM)
START JOB INFORMATION
ENGINEER DATE 24-Sep-01
END JOB INFORMATION
INPUT WIDTH 72
UNIT METER MTON
JOINT COORDINATES
1 0 0 0; 2 8 0 0; 3 16 0 0; 4 24 0 0; 5 32 0 0; 6 0 5 0; 7 8 5 0;
8 16 5 0; 9 24 5 0; 10 32 5 0; 11 0 8.5 0; 12 8 8.5 0; 13 16 8.5 0;
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 RIGIDECES
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 DISPLACMENTS ALL
PRINT SUPPORT REACTION
LOAD LIST 2
PRINT JOINT DISPLACMENTS 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
```

C:\Mis documentos\cursos\staadpro\ejemplos\mar3-26-10a.std

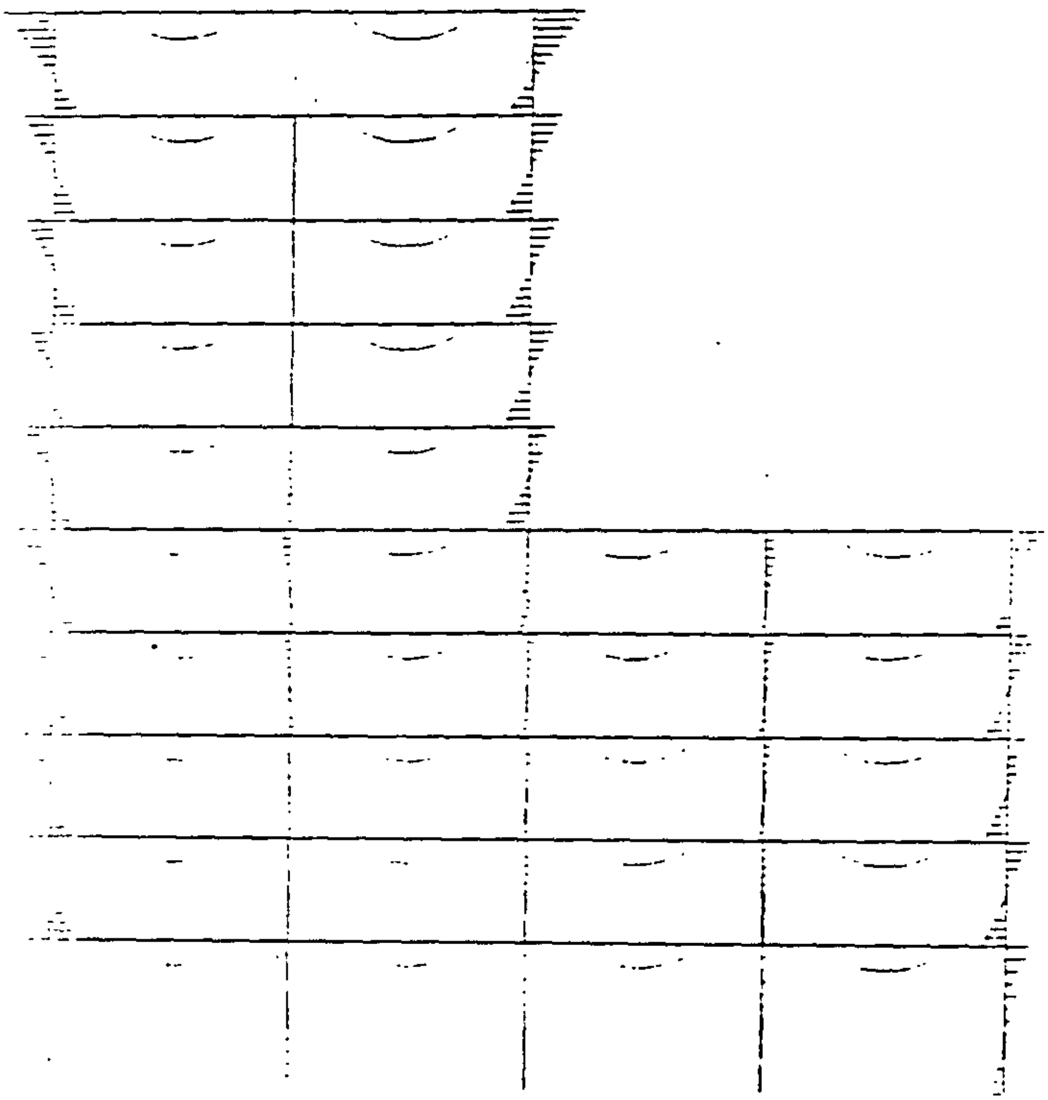
Software licensed to Unknown User	Job No	Sheet No 1		Rev
Job Title	Part			
	Ref			
	By	Date 24-Sep-01	Chd	
Client	File mar3-2d-10n.std		Date/Time 24-Sep-2001 16:07	



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	

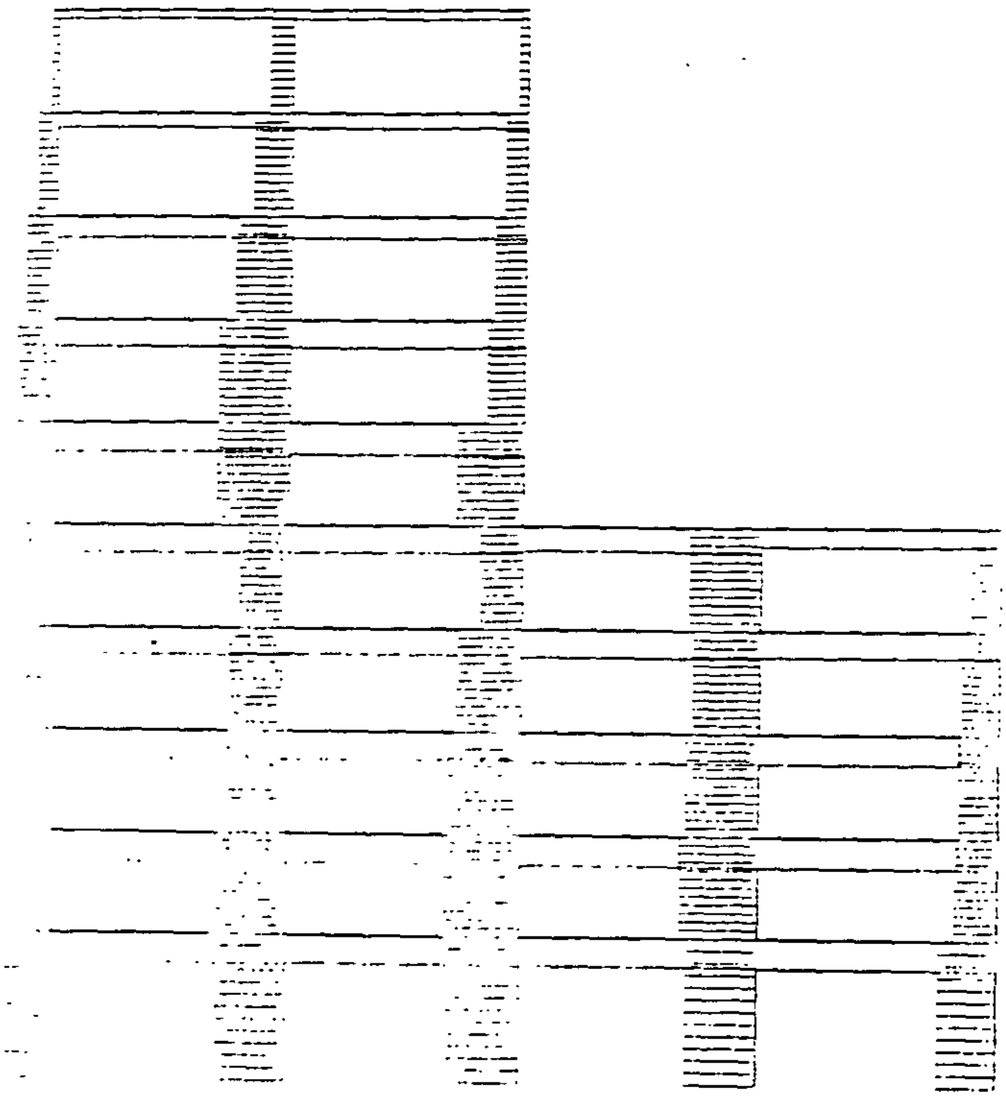


Software licensed to Unknown User	Job No	Sheet No 1	Rev
	Part		
Job Title	Ref		
	By	Date 24-Sep-01	Chd
Client	File mar3-2d-10n.std	Date/Time 24-Sep-2001 16:07	



Software licensed to Unknown User

Job Title	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	



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

CURSOS ABIERTOS

STAAD-PRO

PARA ANÁLISIS Y DISEÑO ESTRUCTURAL

CA 003

TEMA

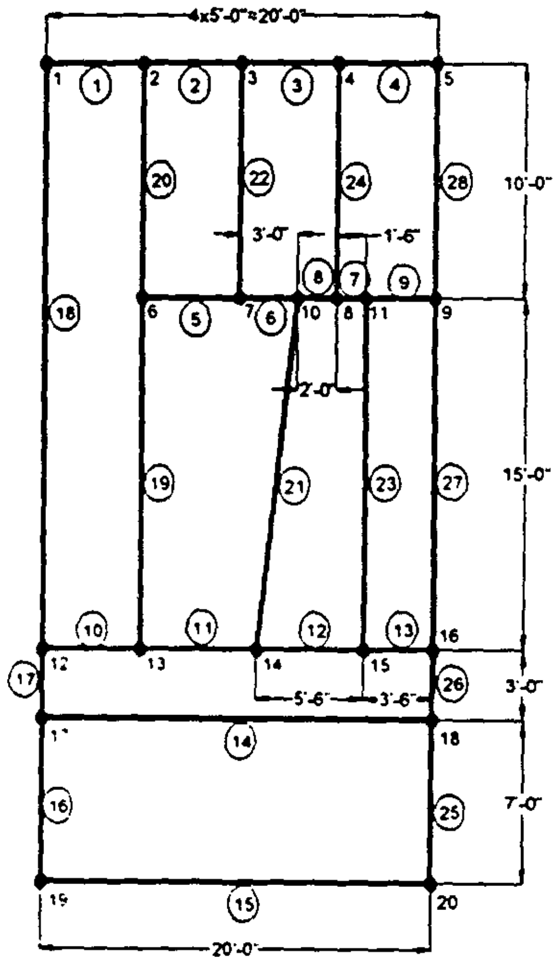
EXAMPLE PROBLEM No. 2

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERÍA
ENERO DEL2003

Palacio de Minería Calle de Tacuba 5 primer piso Deleg. Cuauhtemoc 06000 México, D.F. Tels: 521-40-20 y 521-73-35 Apdo. Postal M-2285

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.

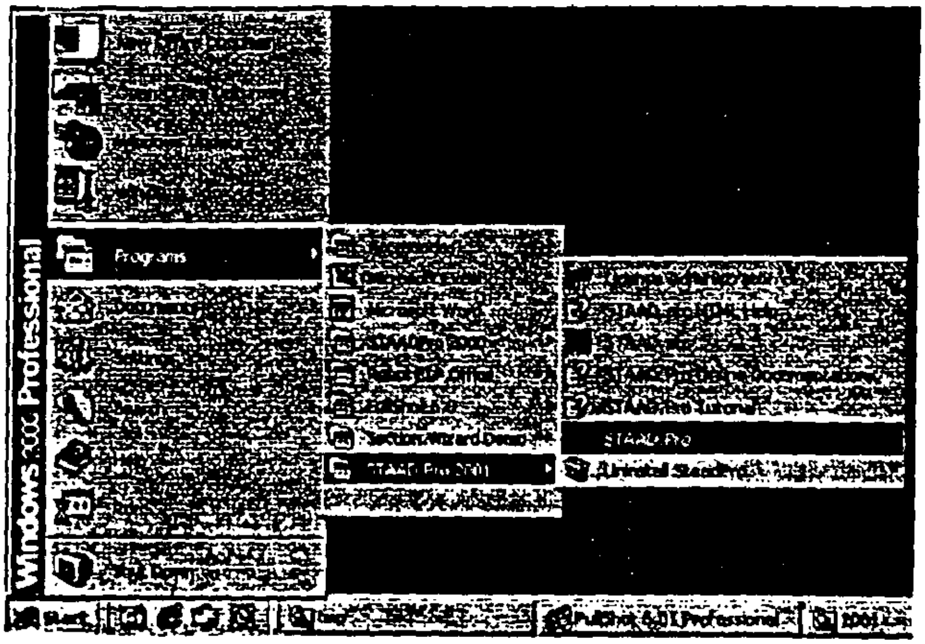


Figure 2. 1

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.

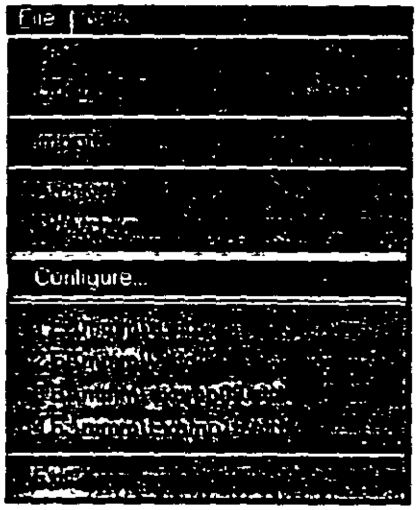


Figure 2. 2

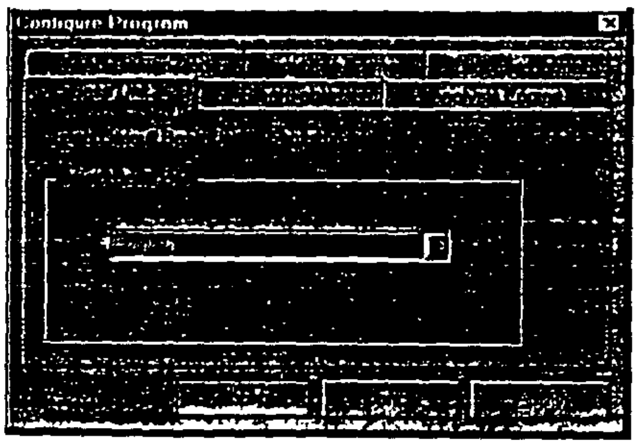


Figure 2. 3

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).

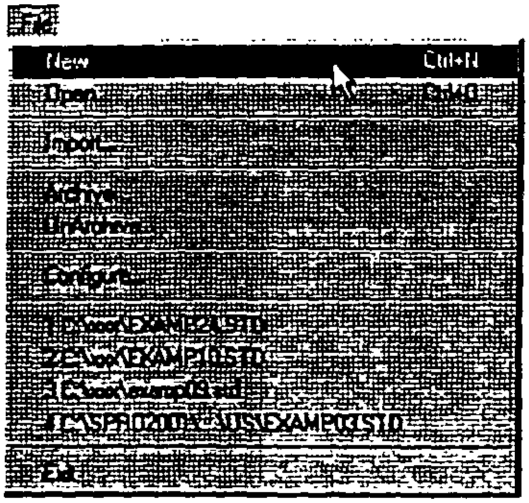


Figure 2. 4

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.

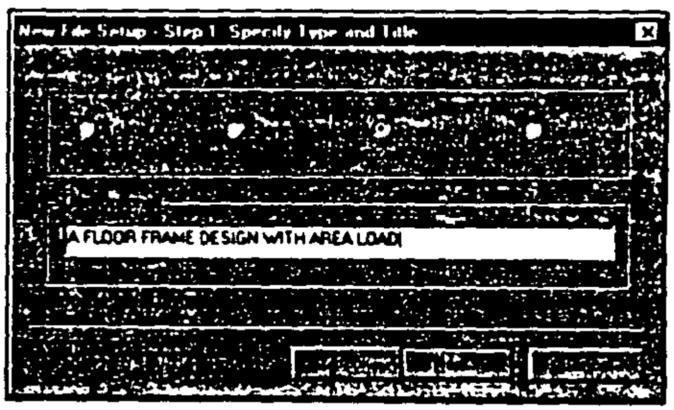


Figure 2. 5

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.

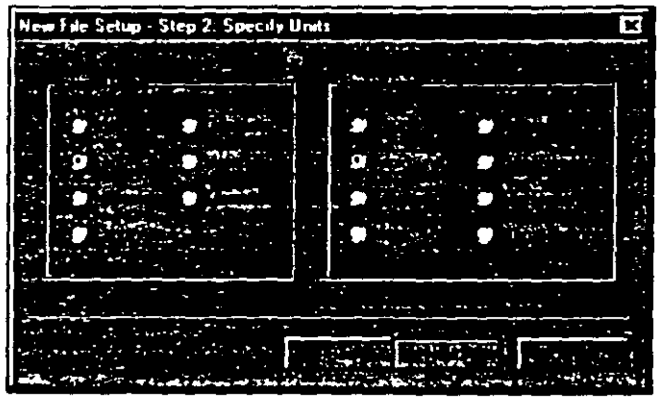


Figure 2. 6

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

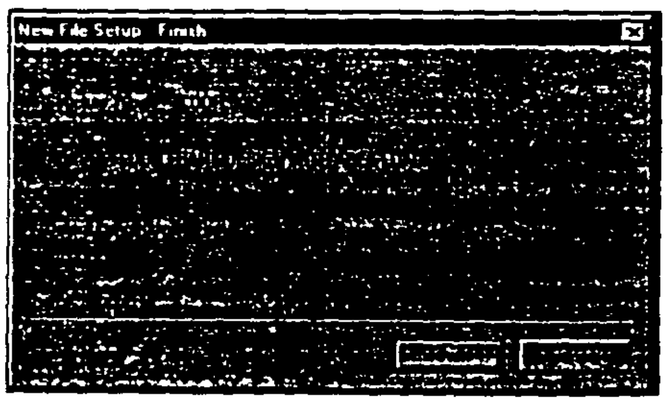


Figure 2. 7

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

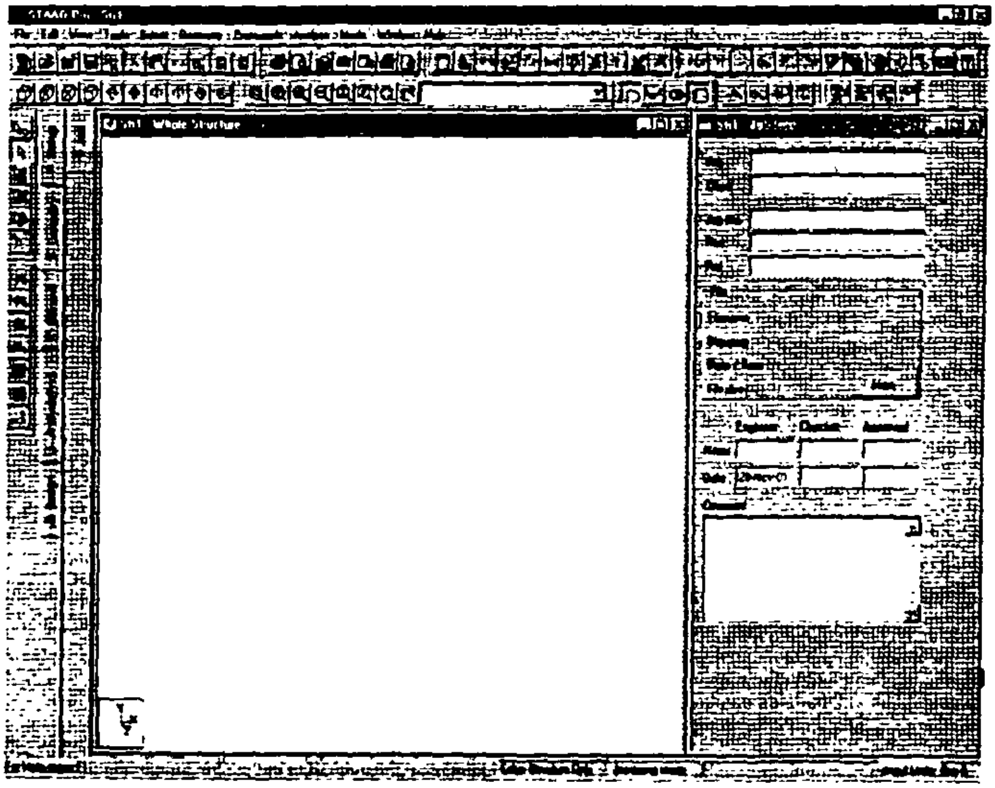


Figure 2. 8

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 *1* ft. (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.

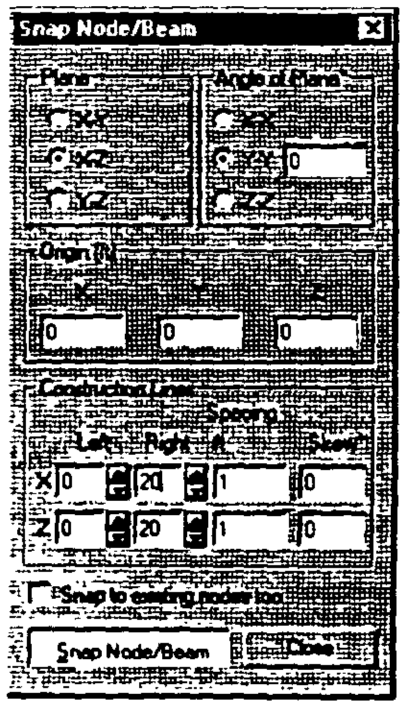


Figure 2. 9

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.

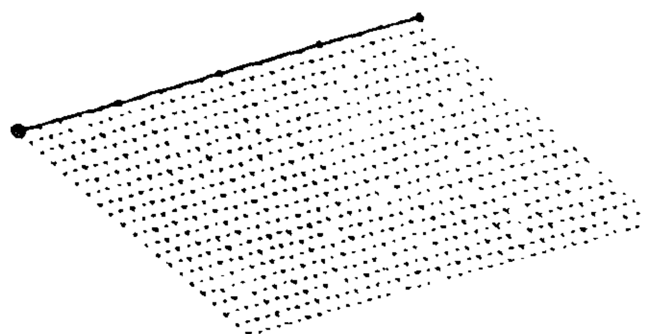


Figure 2. 10

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

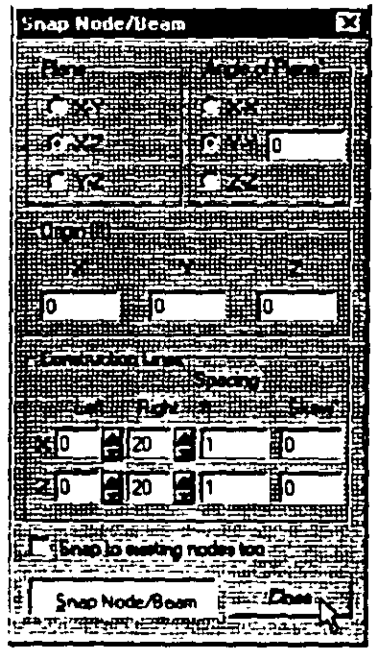


Figure 2. 11

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.

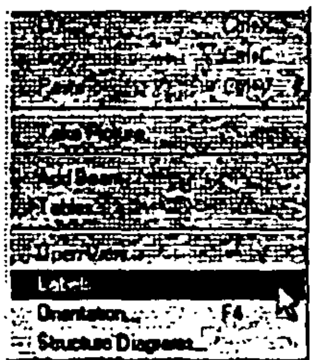


Figure 2. 12

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.

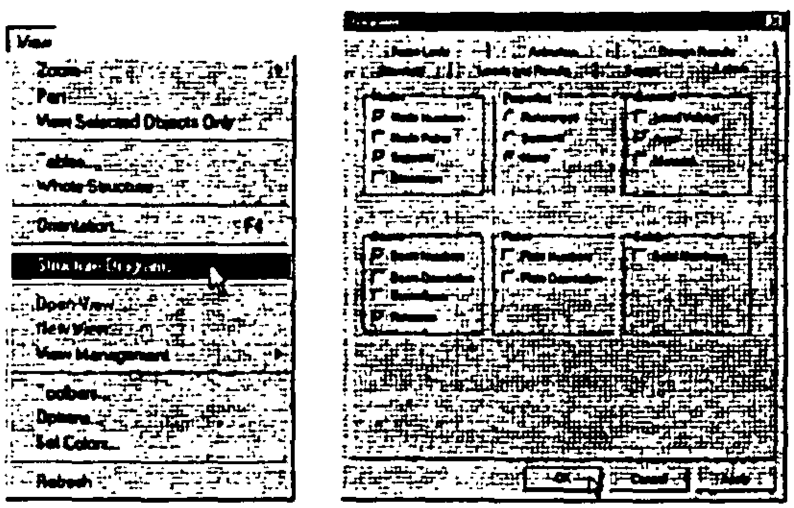


Figure 2. 13

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

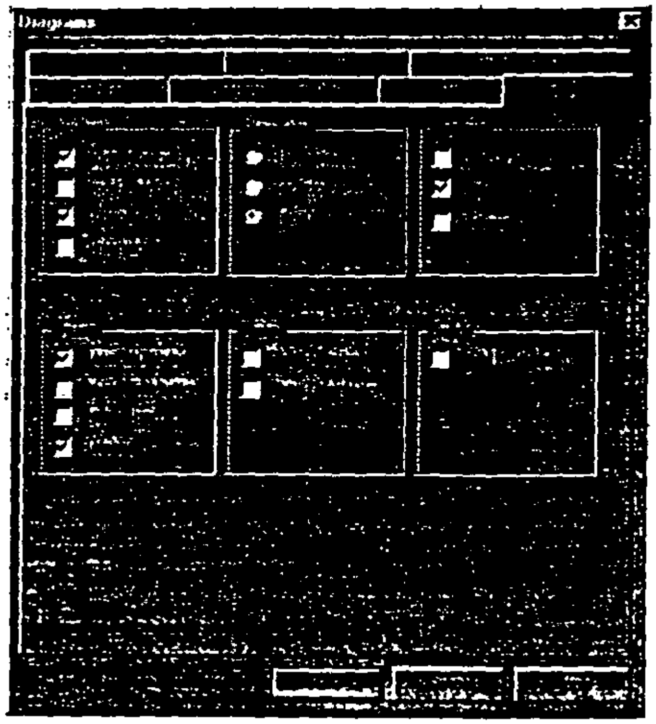


Figure 2. 14

The nodes and beams are now labeled on the drawing.

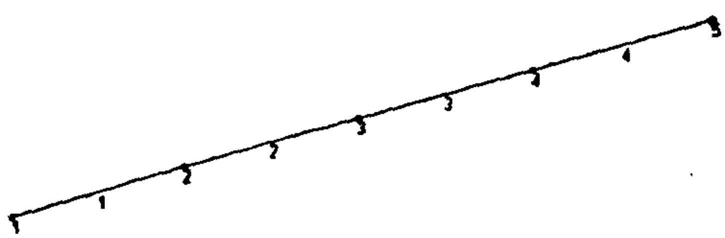


Figure 2. 15

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 = 10ft. 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.

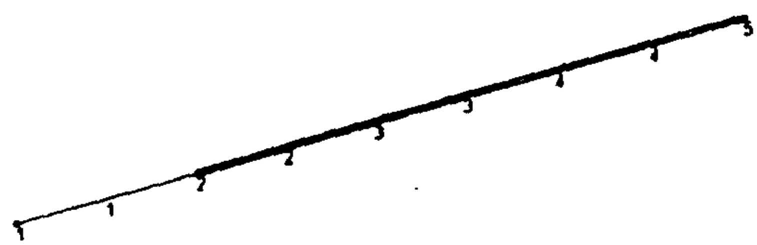


Figure 2. 16

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

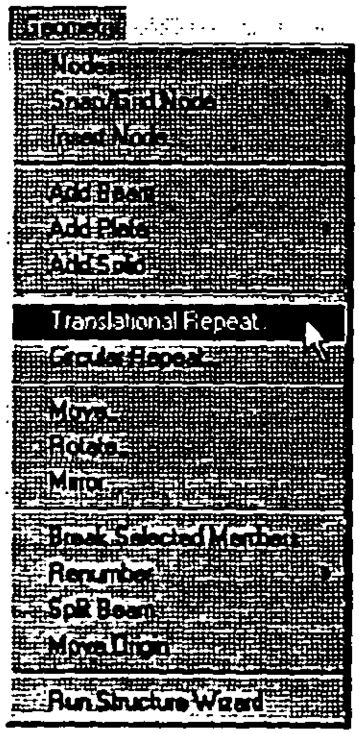


Figure 2. 17

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*.

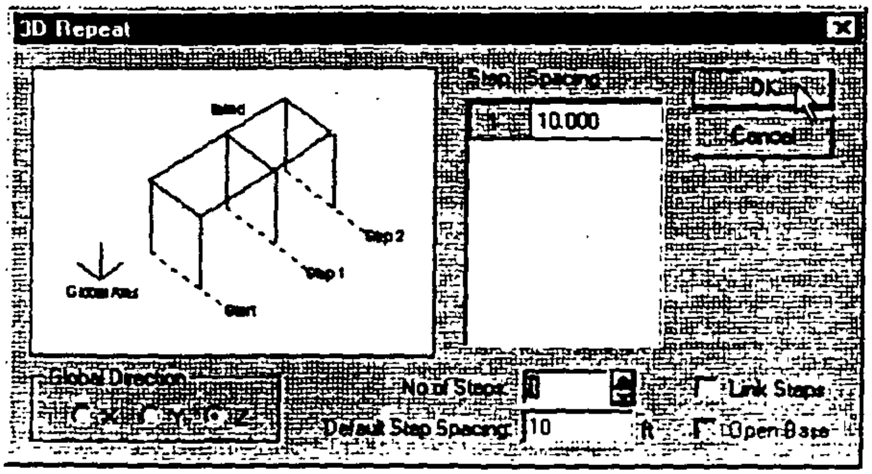


Figure 2. 18

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

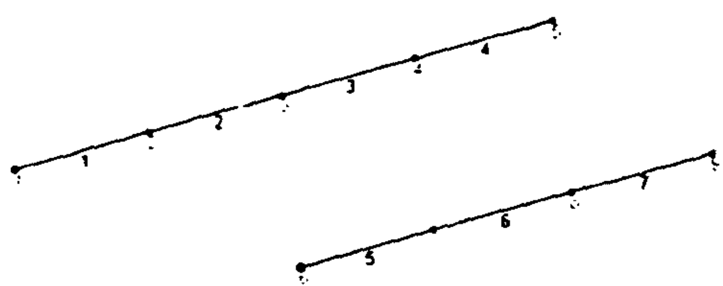


Figure 2. 19

14. Next, let us spilt 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.

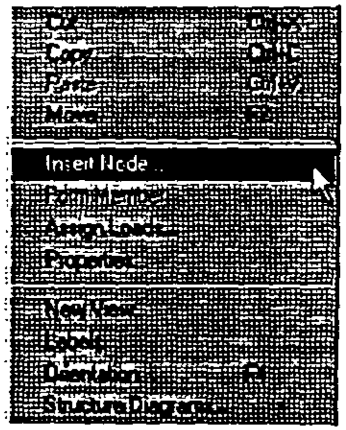


Figure 2. 20

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

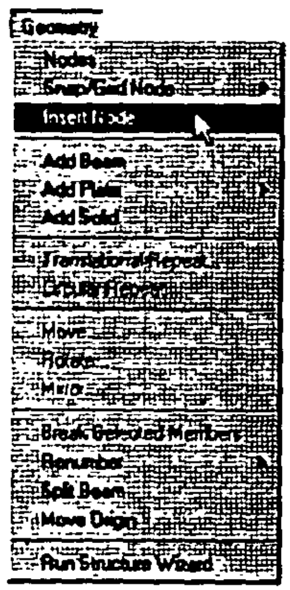


Figure 2. 21

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

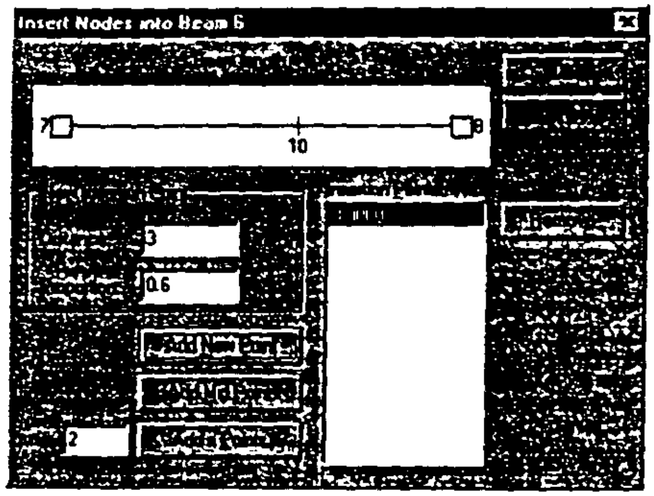


Figure 2. 22

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

Before insertion

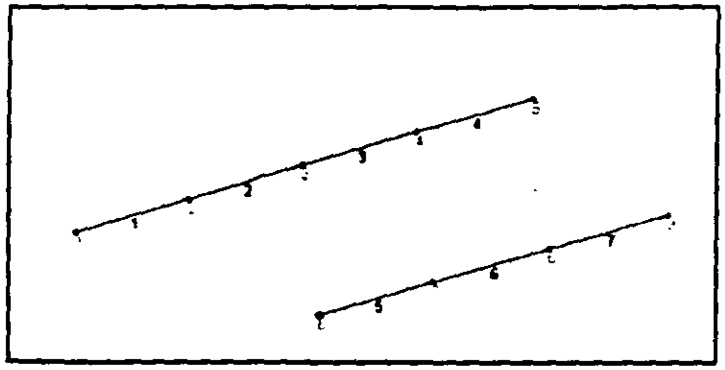


Figure 2. 23

After insertion

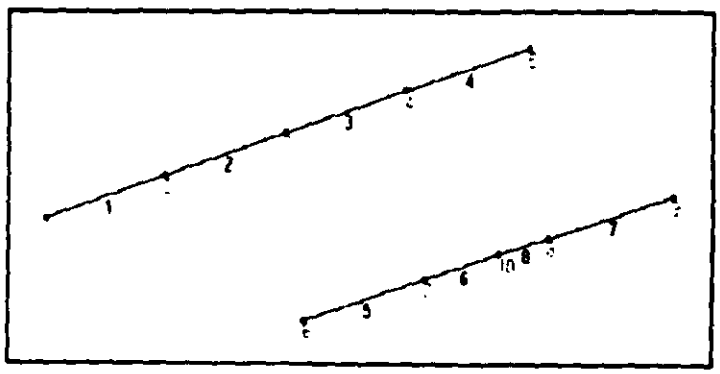


Figure 2. 24

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*.

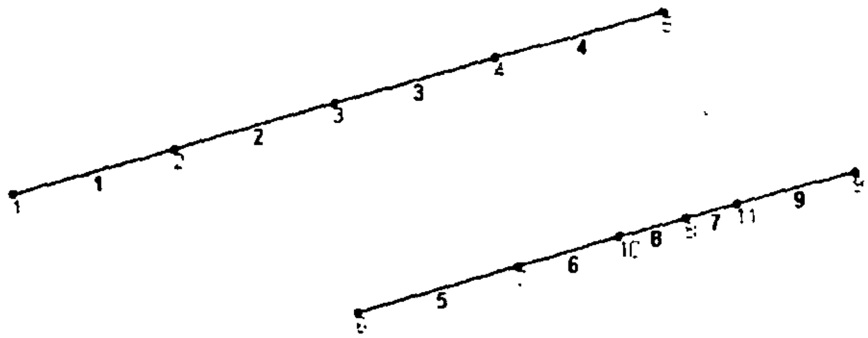


Figure 2. 25

Creating Members 10 to 13

17. To create the X direction beams at Z = 25ft, 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.

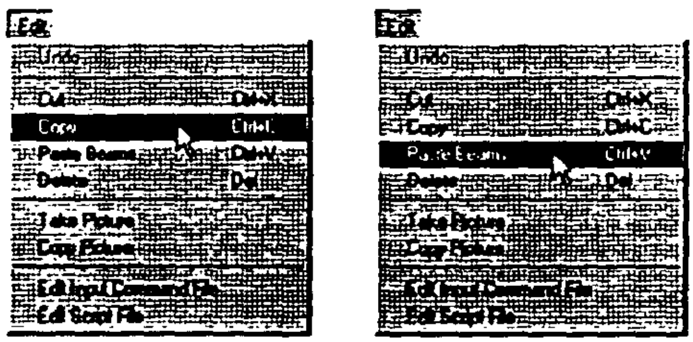


Figure 2. 26

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

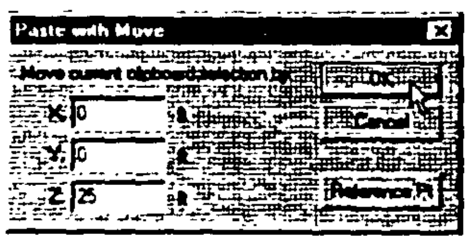


Figure 2. 27

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.

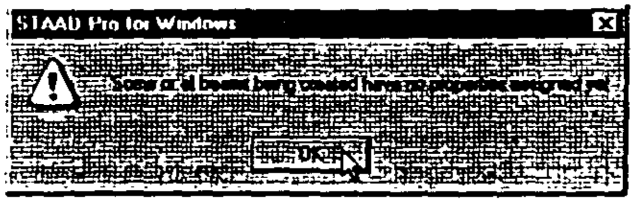


Figure 2. 28

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

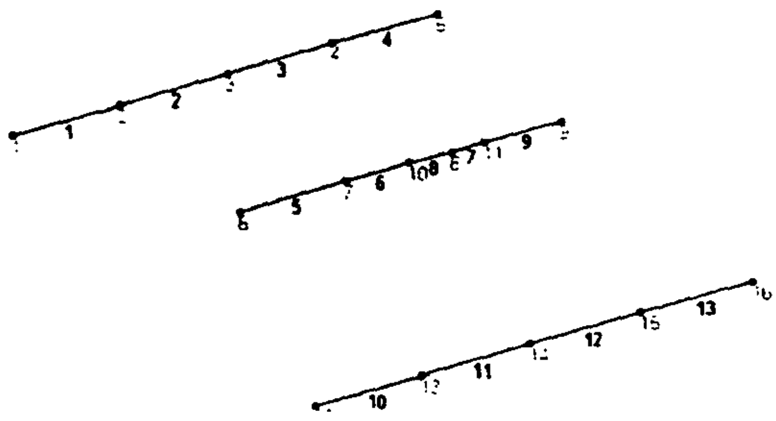
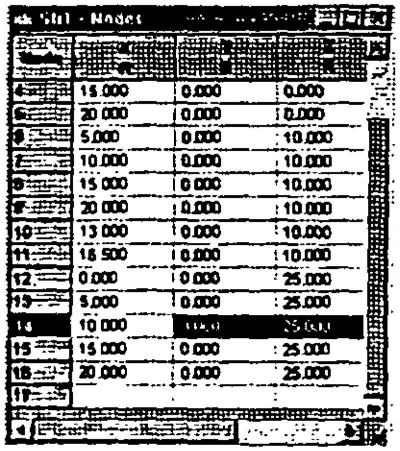


Figure 2. 29

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.

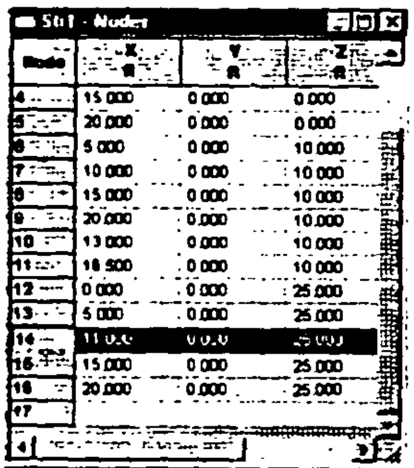


5t1 - Nodes

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	18.500	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*.



5t1 - Nodes

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	18.500	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*.

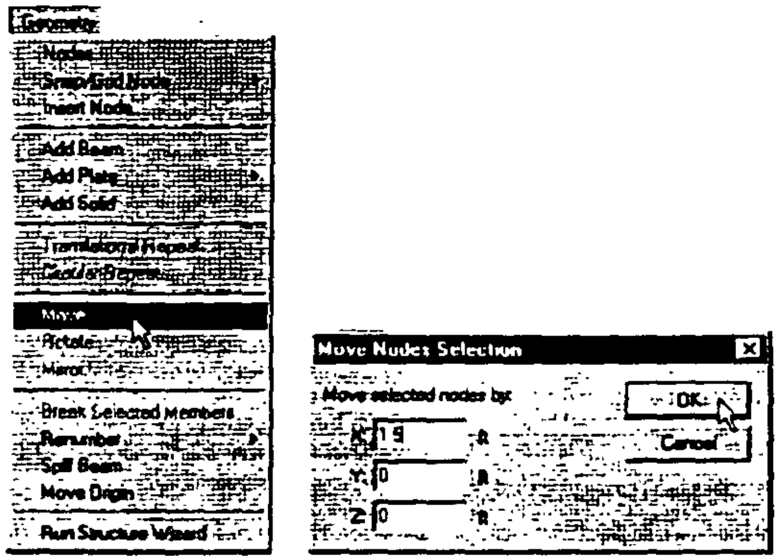


Figure 2. 32

Creating Member 14

23. We now have to create the X direction beam at Z = 28ft. 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.

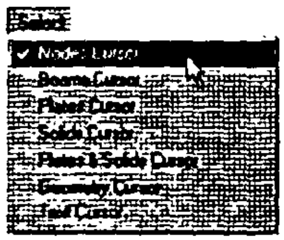


Figure 2. 33

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 Ctrl+C or select *Copy* from the *Edit* menu. To paste, type Ctrl+V or select *Paste Nodes* from the *Edit* menu.

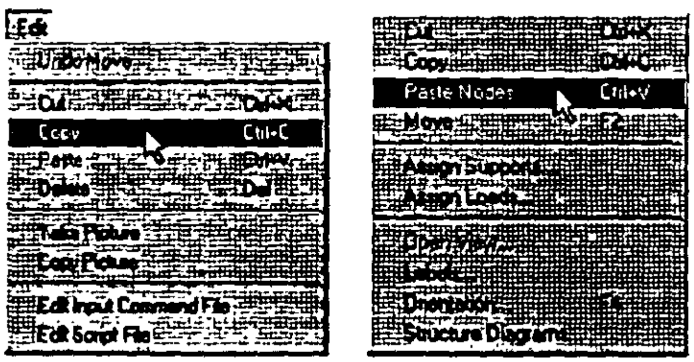


Figure 2. 34

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.

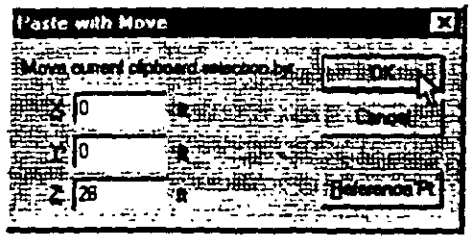


Figure 2. 35

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

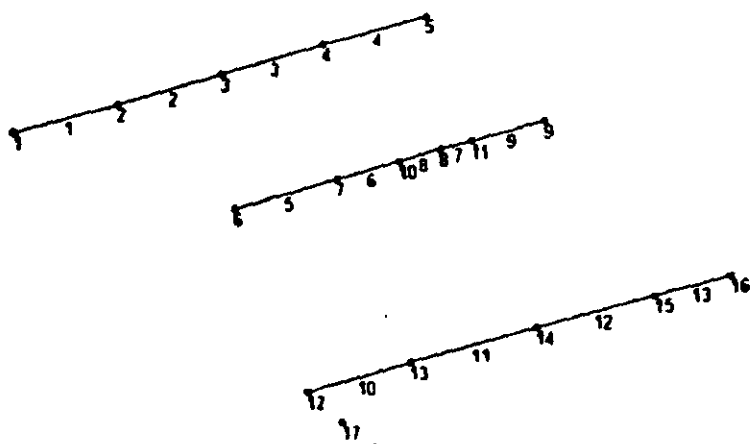


Figure 2. 36

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

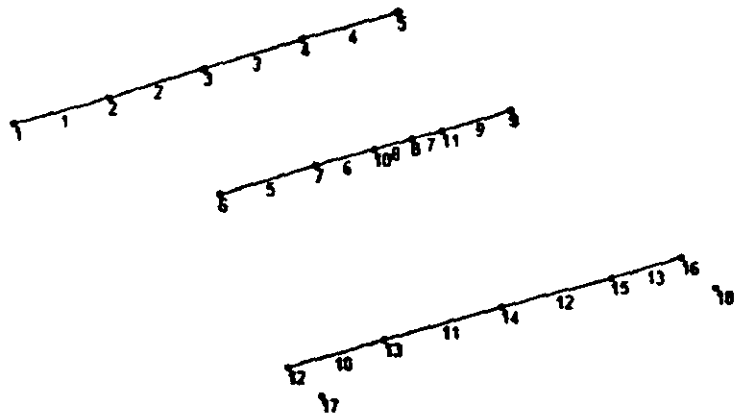


Figure 2. 37

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.

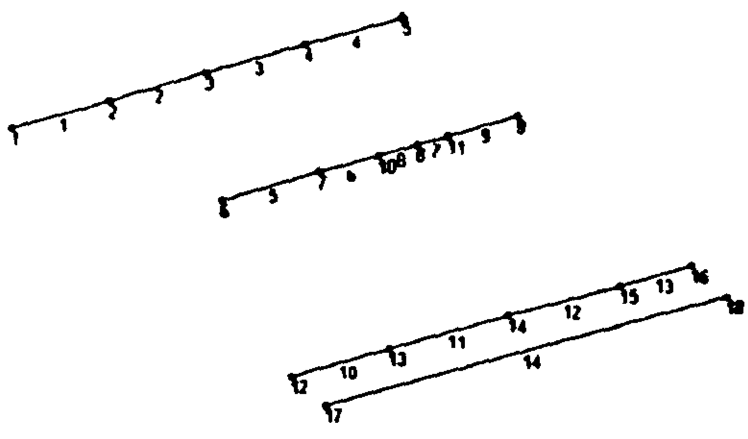


Figure 2. 38

Creating Member 15

26. To create the beam at Z = 35ft, 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 = 7ft. 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.

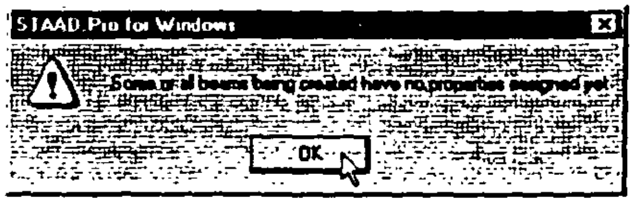


Figure 2. 39

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

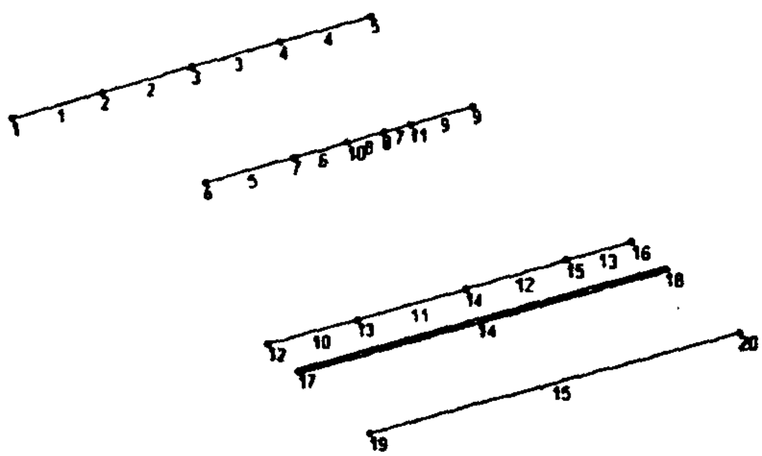


Figure 2. 40

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*.

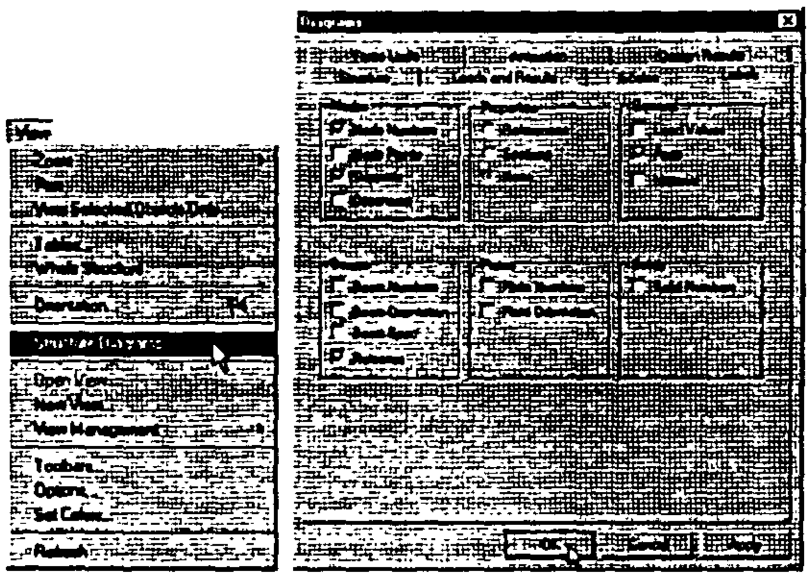


Figure 2. 41

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

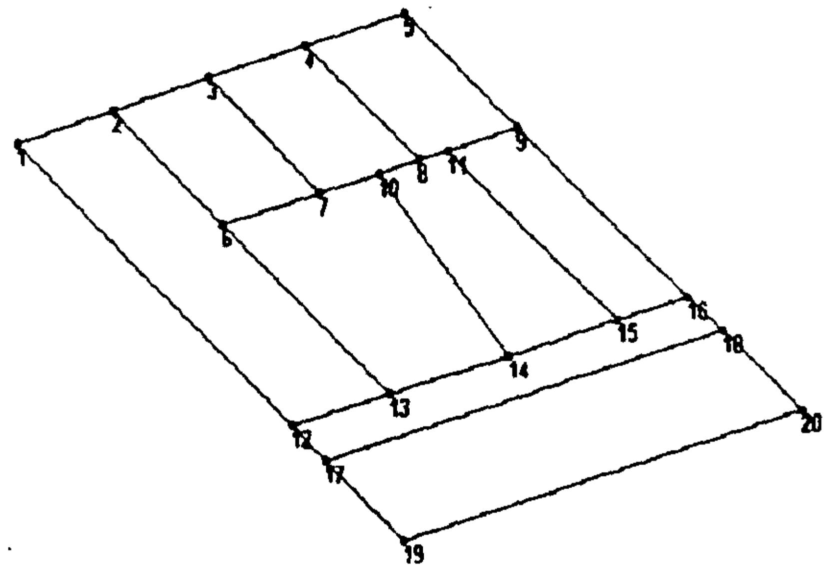


Figure 2. 42

Plan View

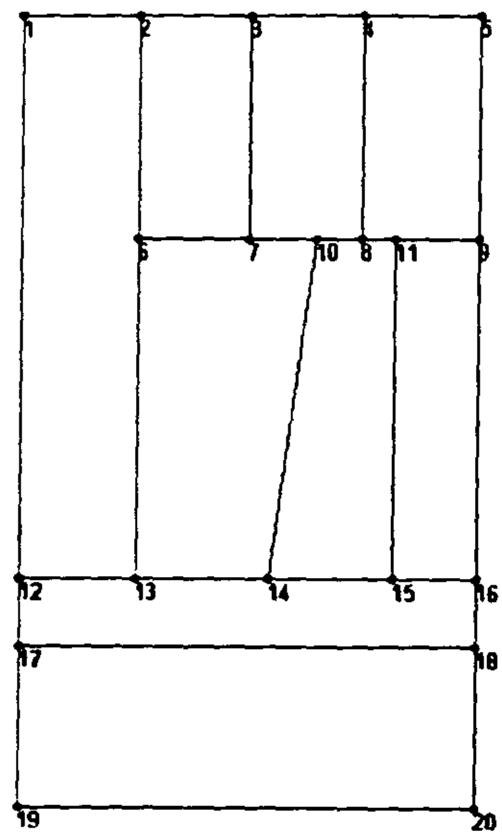


Figure 2. 43

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.

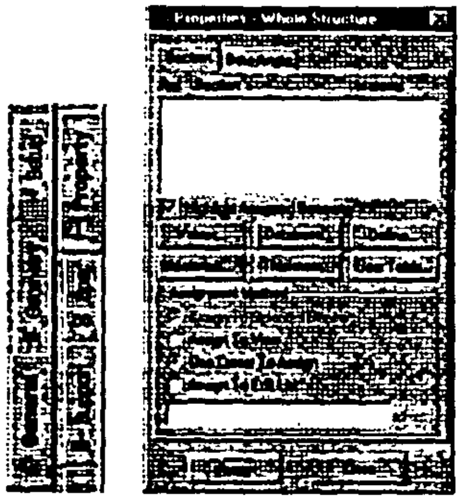


Figure 2. 44

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*.

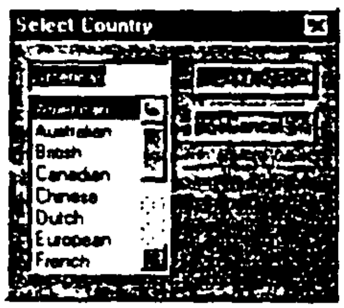


Figure 2. 45

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.

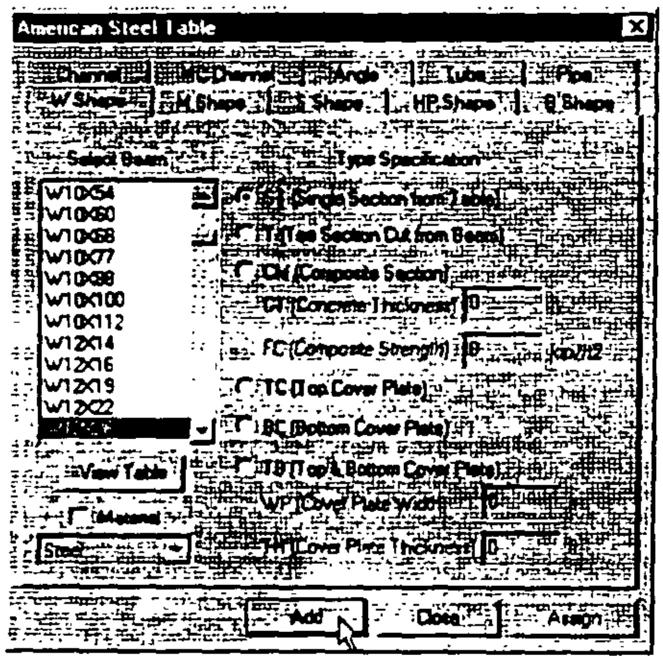


Figure 2. 46

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.

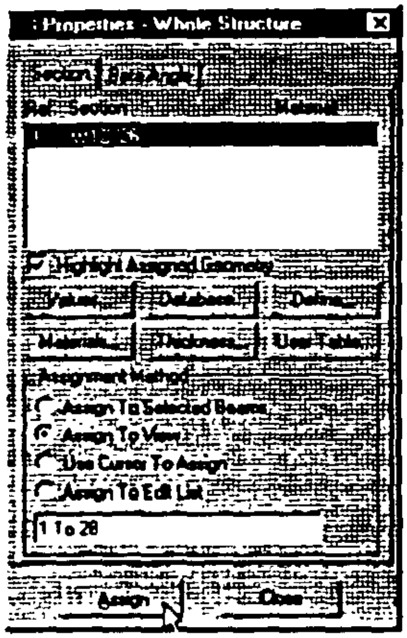


Figure 2. 47

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*.

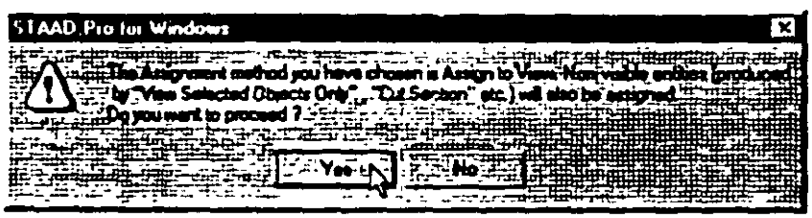


Figure 2. 48

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.

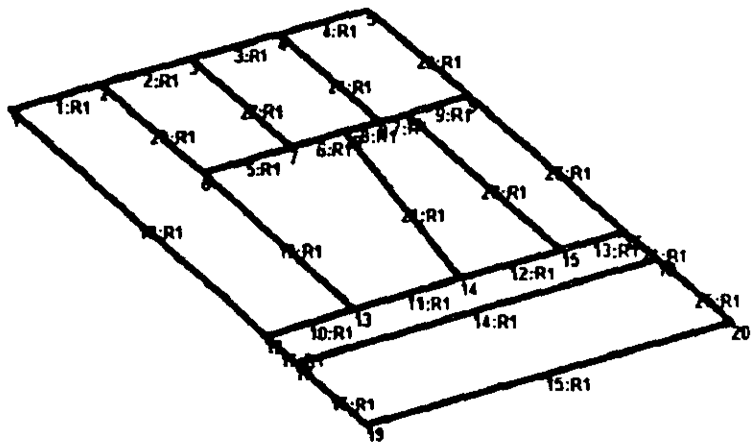


Figure 2. 49

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

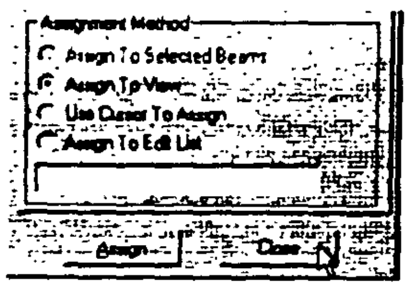


Figure 2. 50

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.

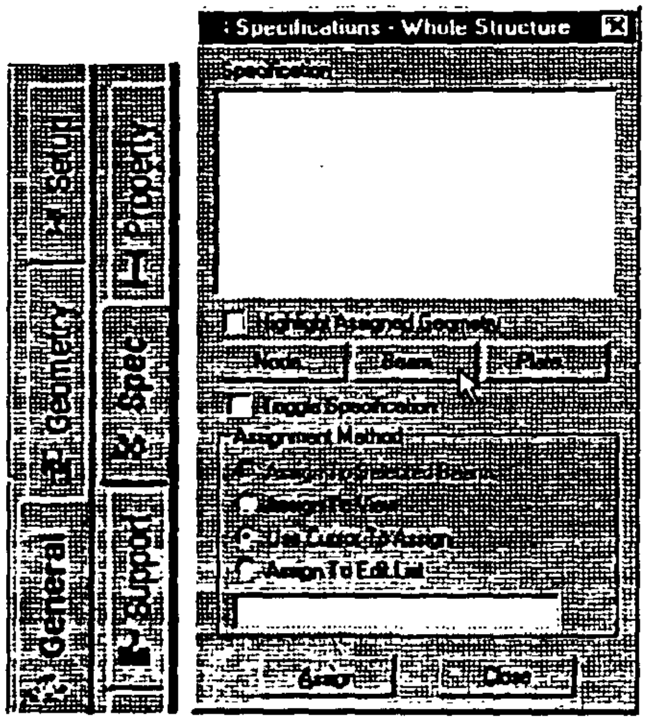


Figure 2. 51

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.

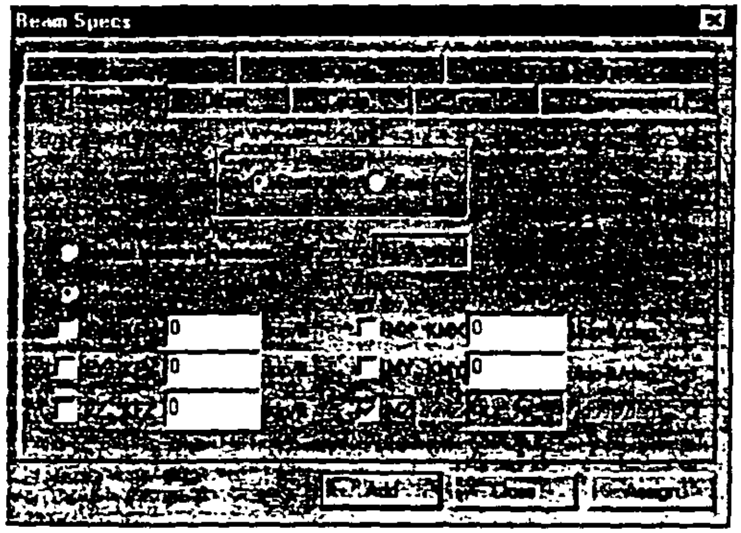


Figure 2. 52

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.

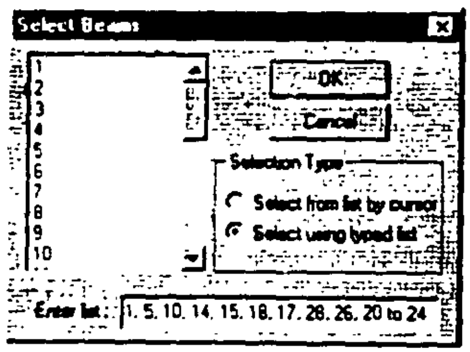


Figure 2. 53

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

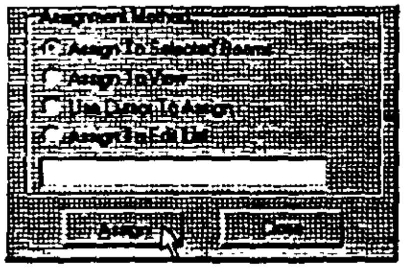


Figure 2. 54

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*.

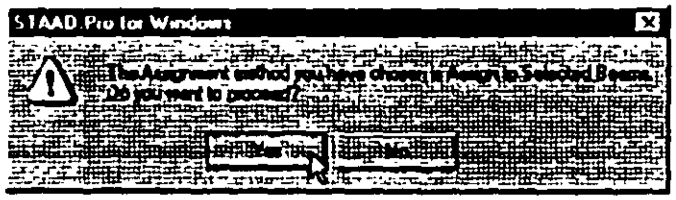


Figure 2. 55

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:

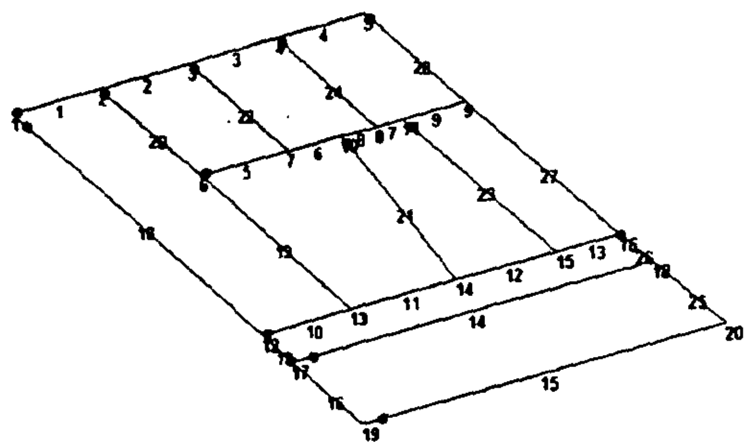


Figure 2. 56

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:

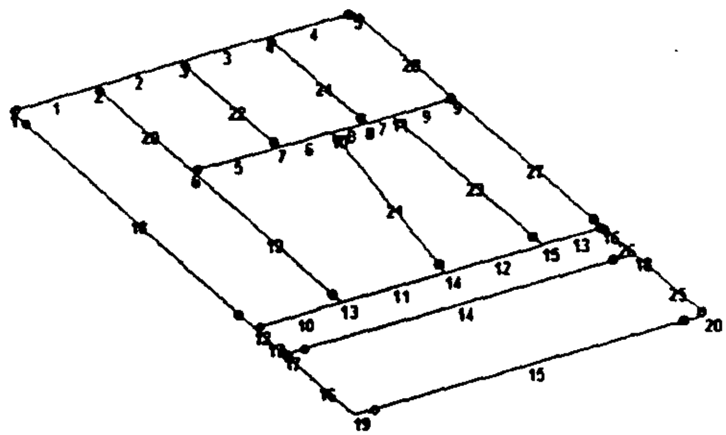


Figure 2. 57

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.

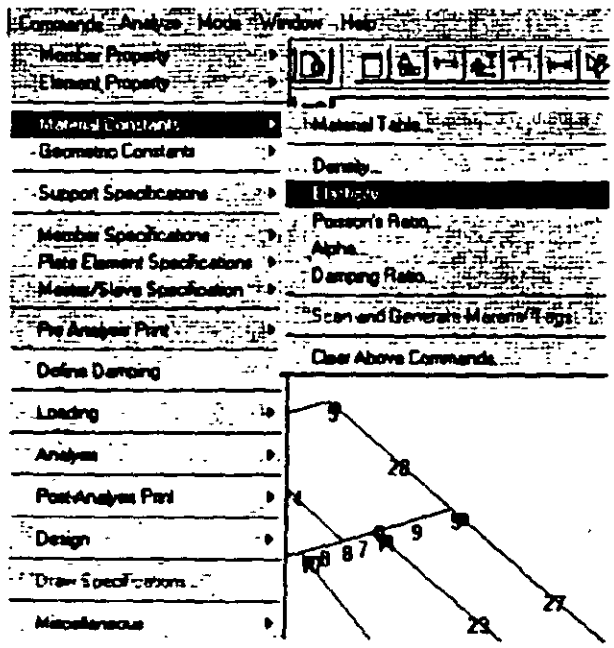


Figure 2. 58

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*.

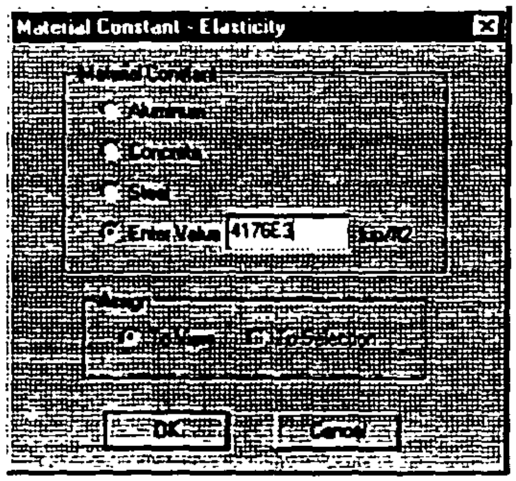


Figure 2. 59

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.

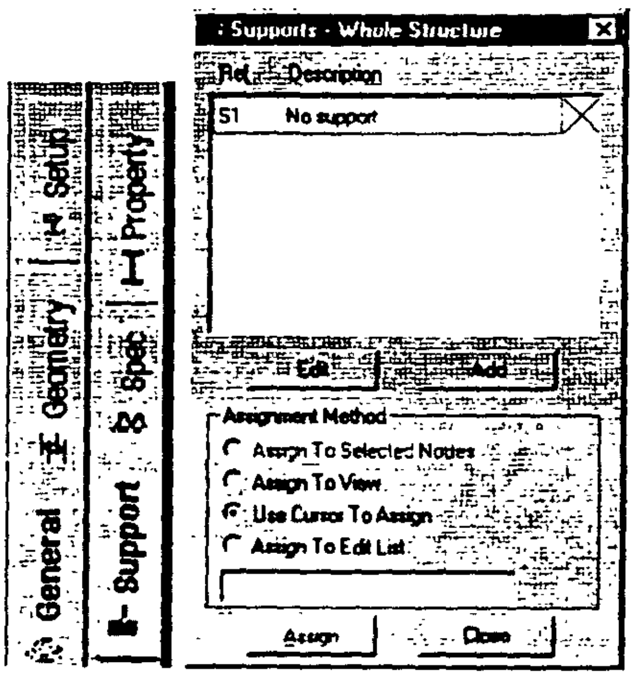


Figure 2. 60

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.

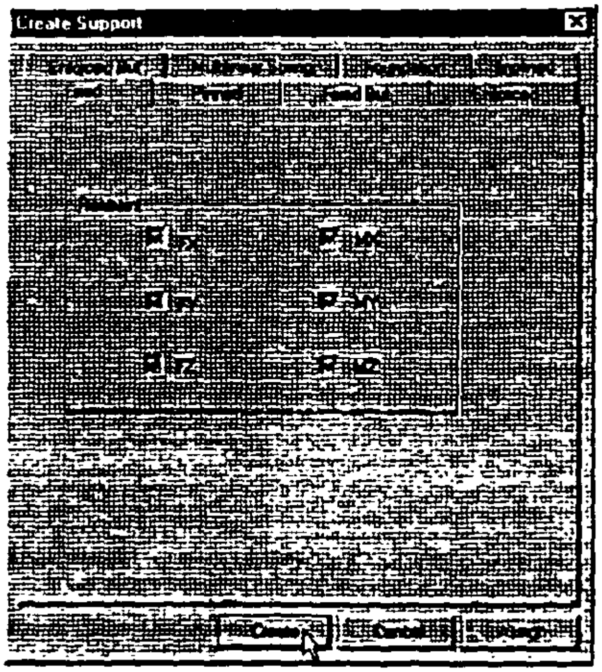


Figure 2. 61

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

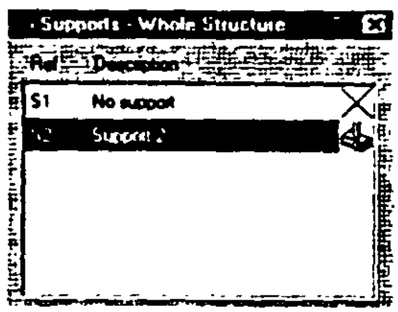


Figure 2. 62

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.

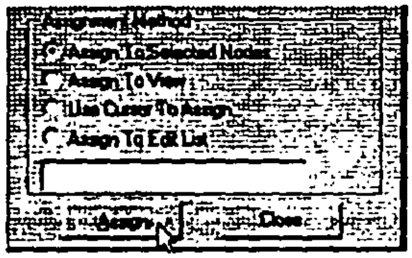


Figure 2. 63

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*.

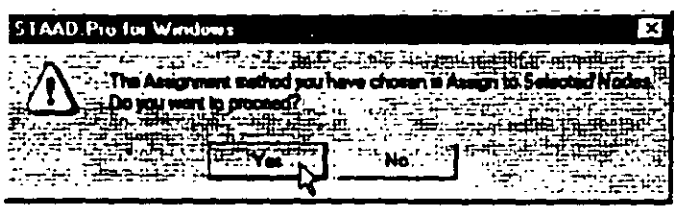


Figure 2. 64

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

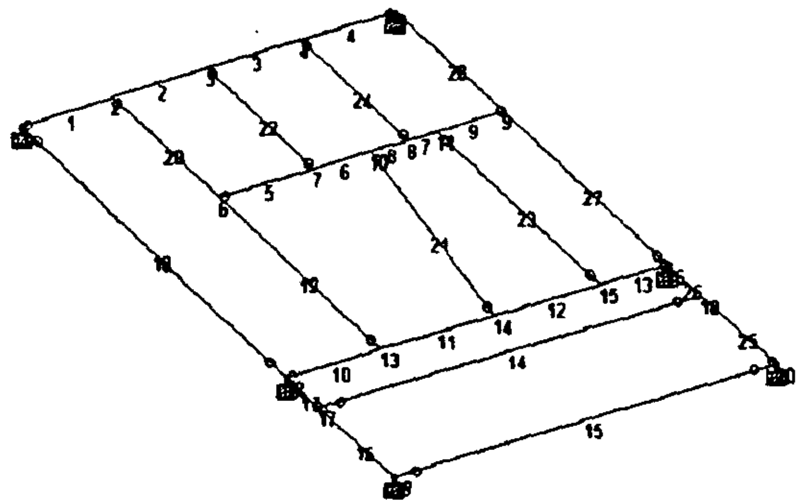


Figure 2. 65

Specifying Loads

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

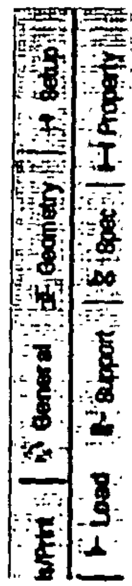


Figure 2. 66

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*.

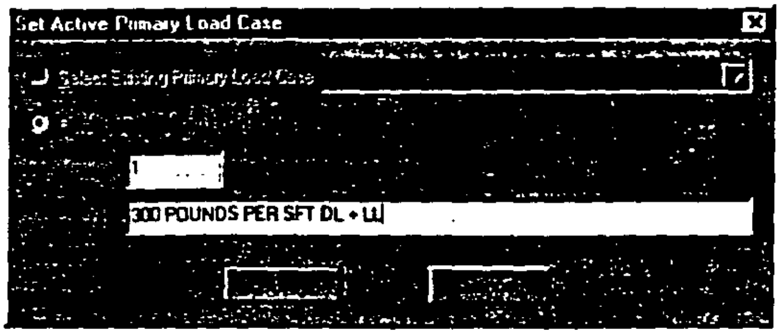


Figure 2. 67

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

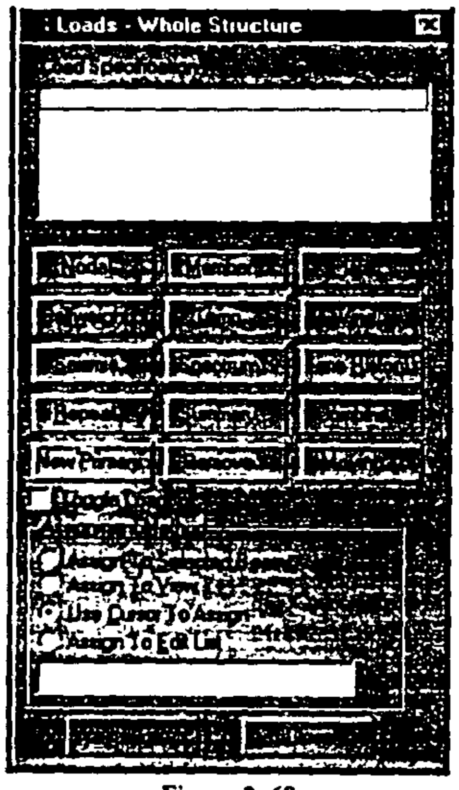


Figure 2. 68

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.

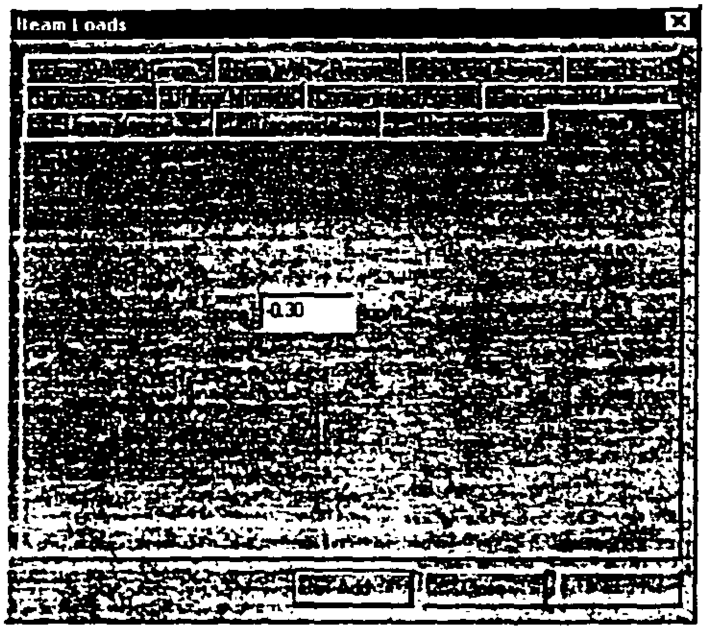


Figure 2. 69

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*.

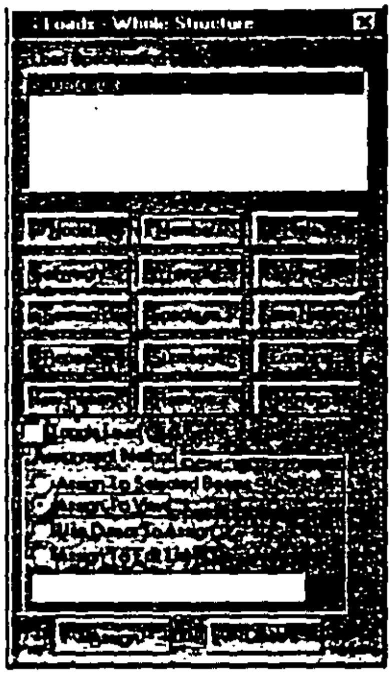


Figure 2. 71

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*.

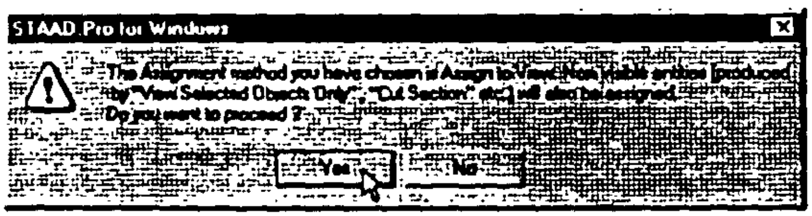


Figure 2. 72

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.

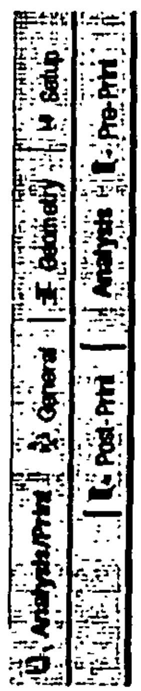


Figure 2. 73

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

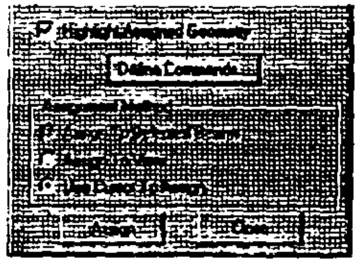


Figure 2. 74

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.

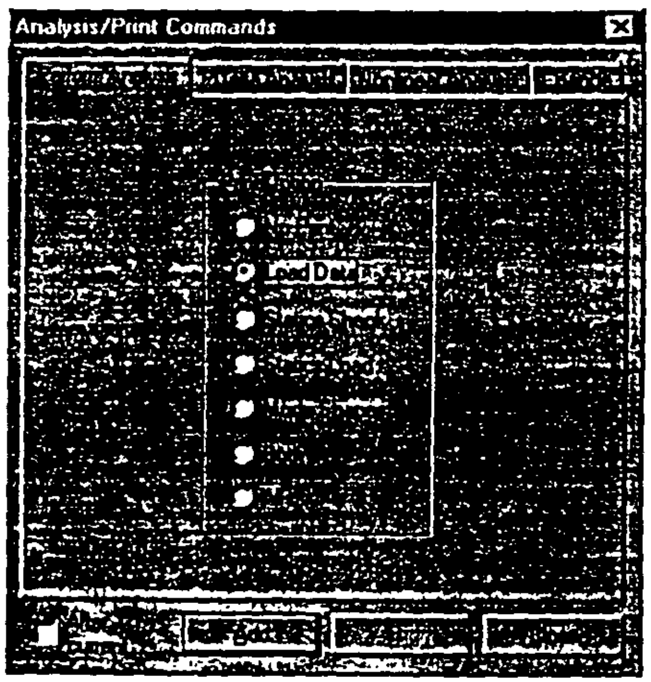


Figure 2. 75

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.

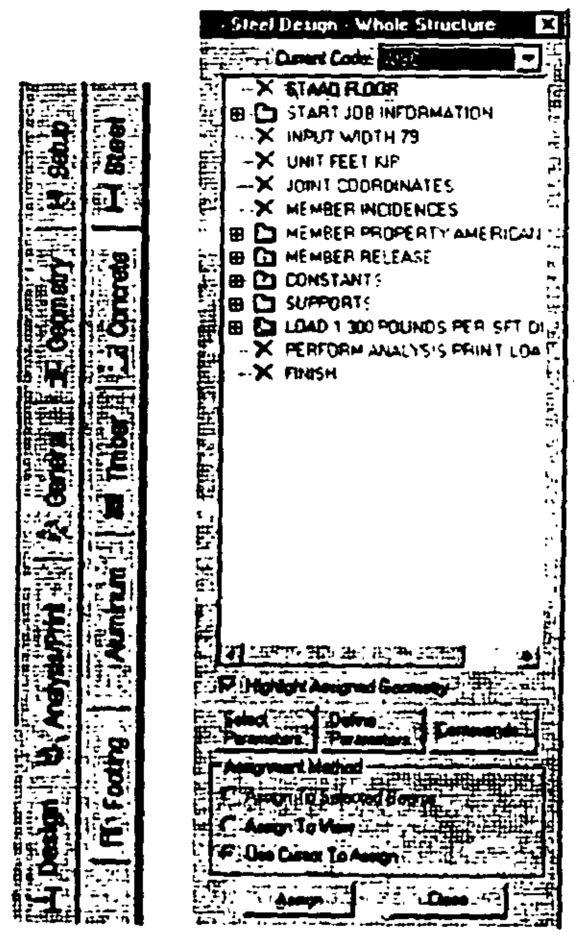


Figure 2. 76

The commands we wish to generate are:

BEAM 1 ALL
DMAX 2.0 ALL
DMIN 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.

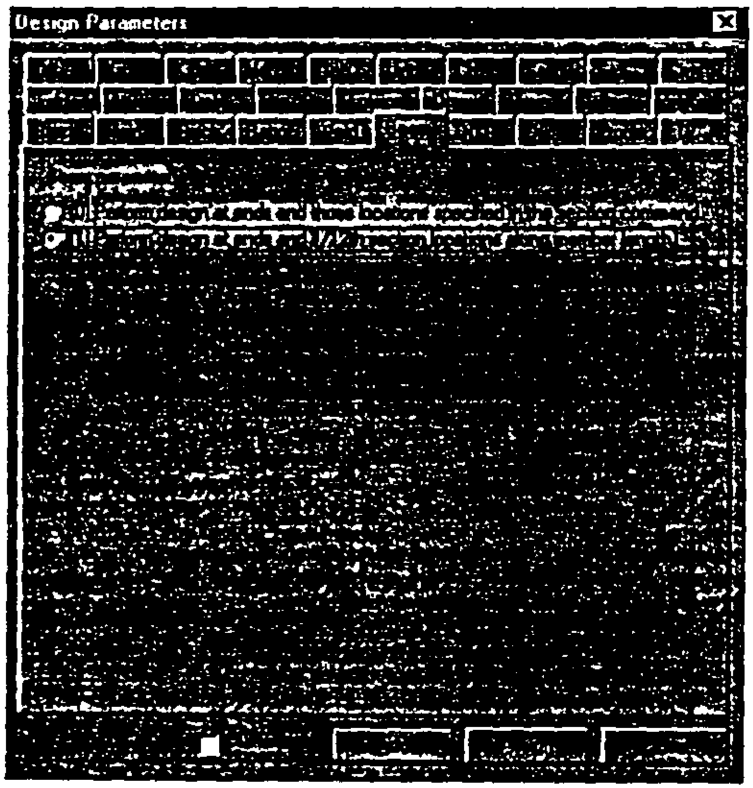


Figure 2. 77

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.

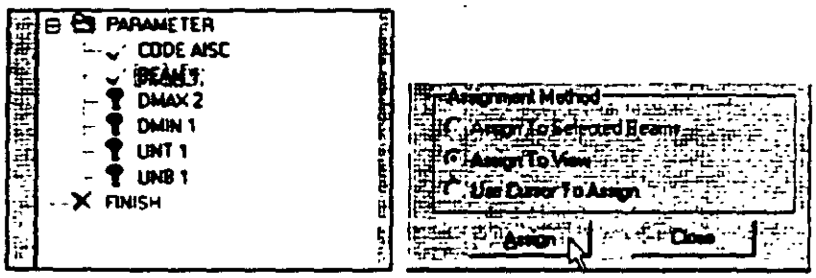


Figure 2. 78

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*.

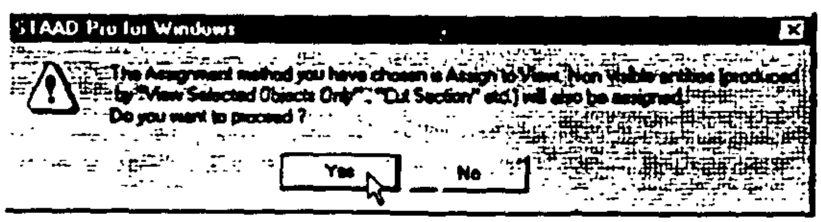


Figure 2. 79

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

Before After

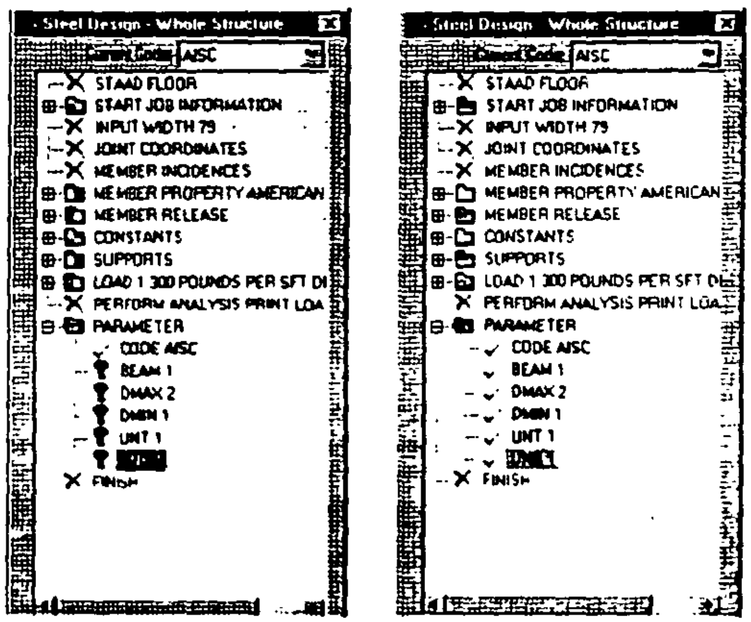


Figure 2. 80

Specifying The SELECT MEMBER Command

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

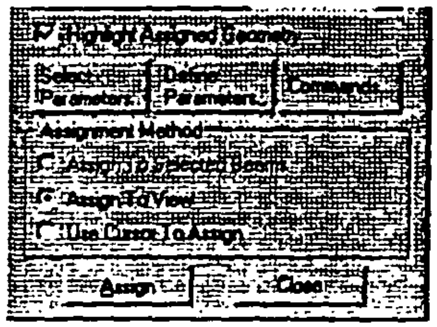


Figure 2. 81

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

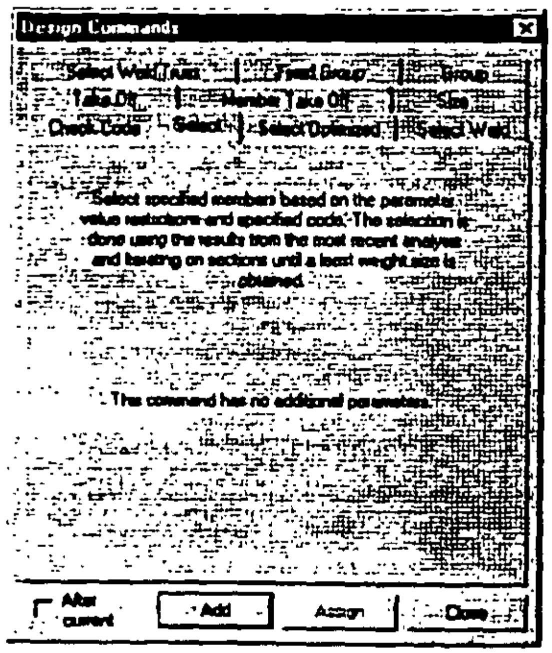


Figure 2. 82

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

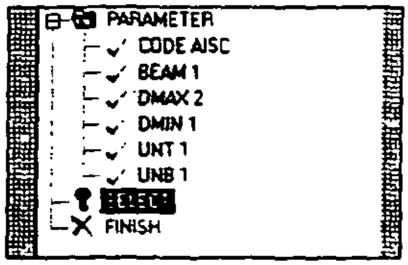


Figure 2. 83

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.

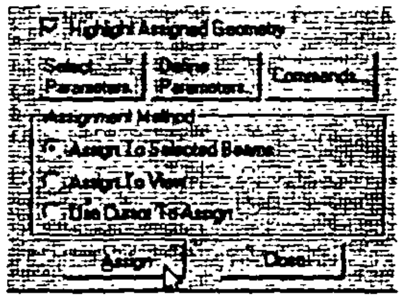


Figure 2. 84

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*.

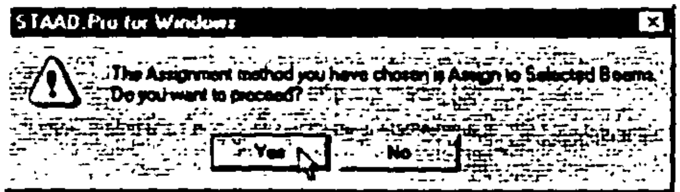


Figure 2. 85

The structure will now look as shown below.

Isometric View

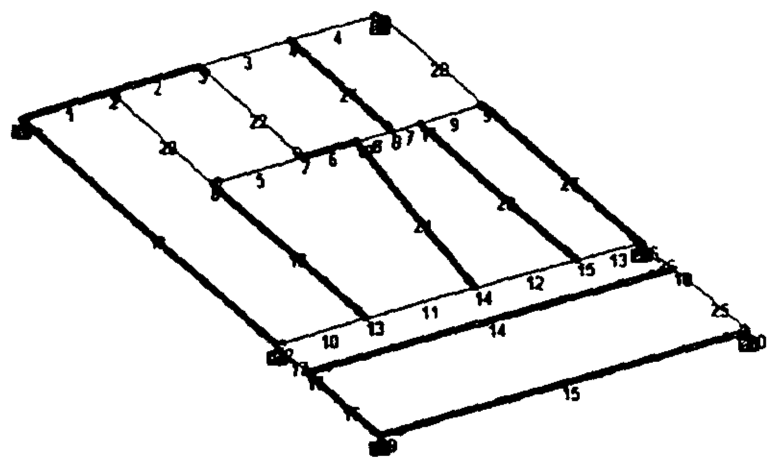


Figure 2. 86

Plan View

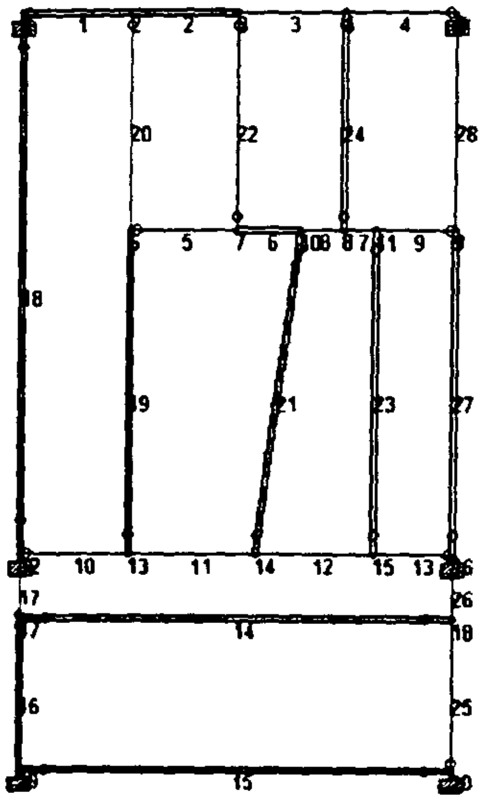


Figure 2. 87

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

CURSOS ABIERTOS

STAAD-PRO
PARA ANÁLISIS Y DISEÑO ESTRUCTURAL
CA 003

TEMA

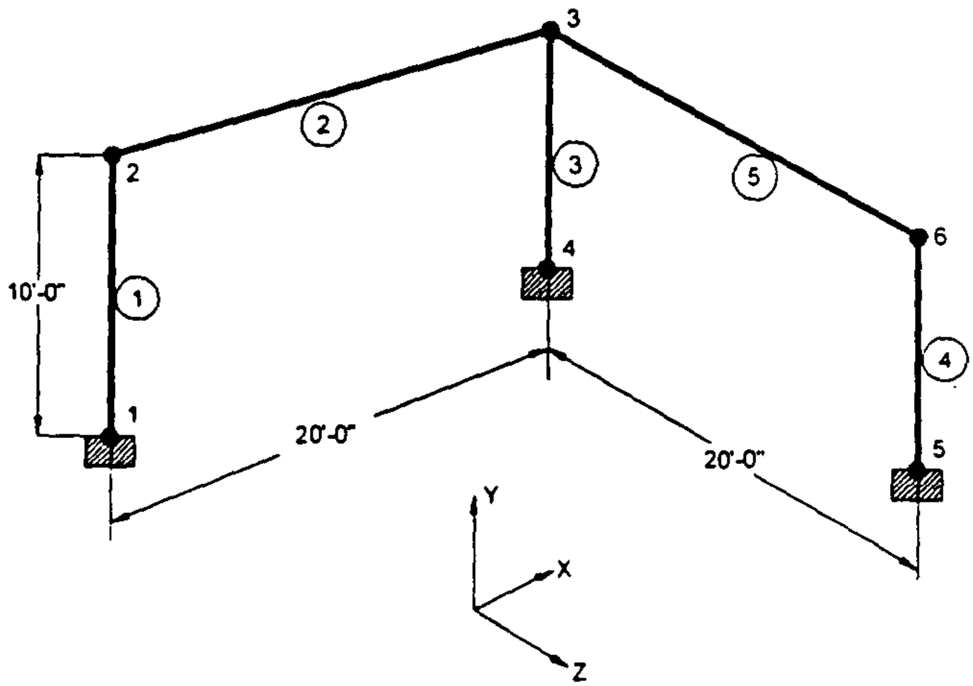
EXAMPLE PROBLEM No. 5

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERÍA
ENERO DEL2003

Palacio de Minería Calle de Tacuba 5 primer piso Deleg. Cuauhtémoc 06000 México, D.F. Tels: 521-40-20 y 521-73-35 Apdo. Postal M-2285

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.

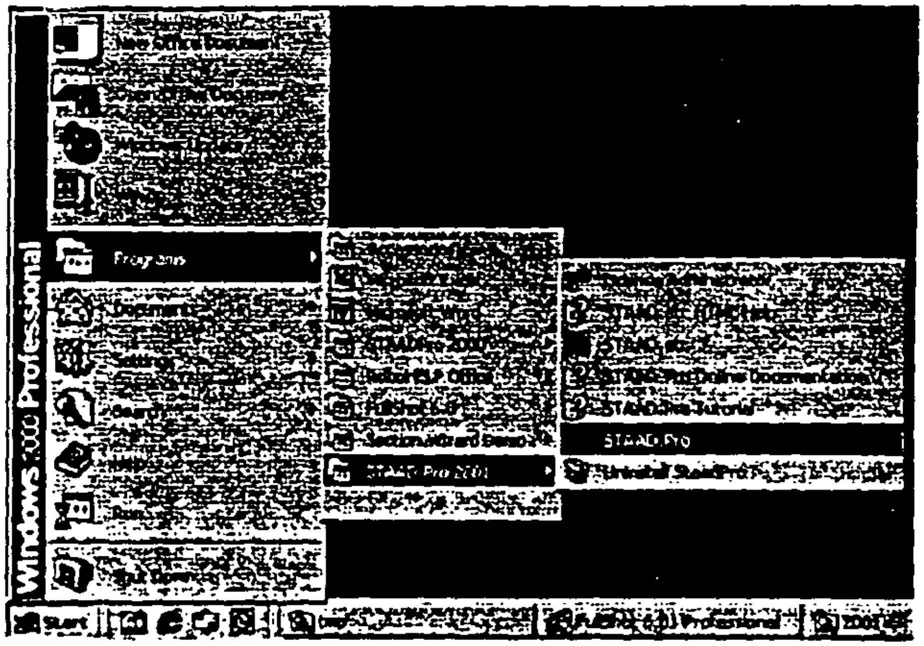


Figure 5. 1

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.

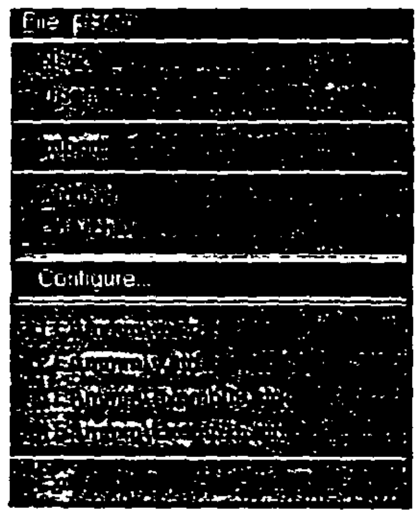


Figure 5. 2

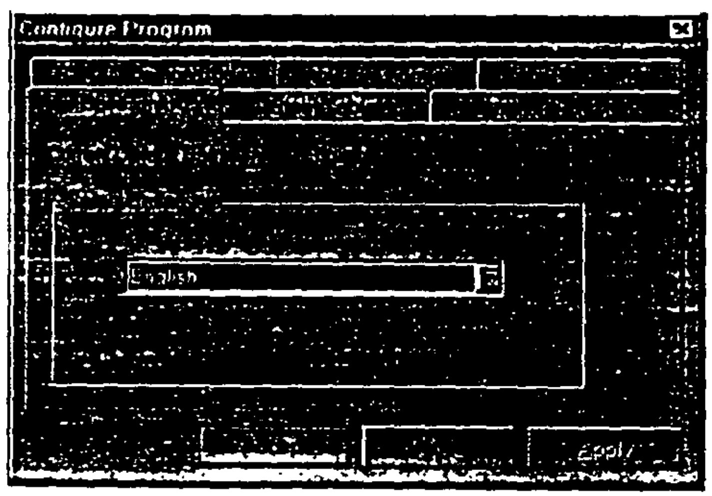


Figure 5. 3

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).

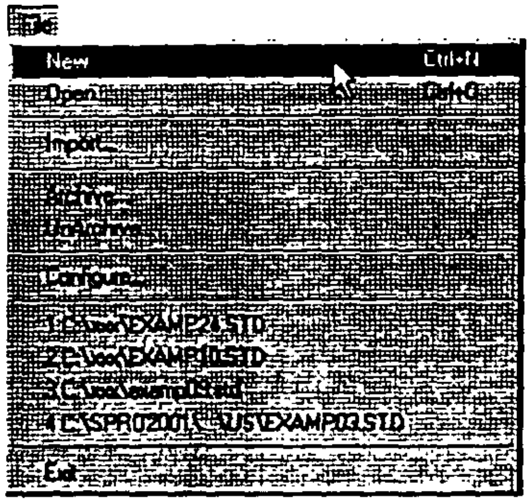


Figure 5. 4

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.

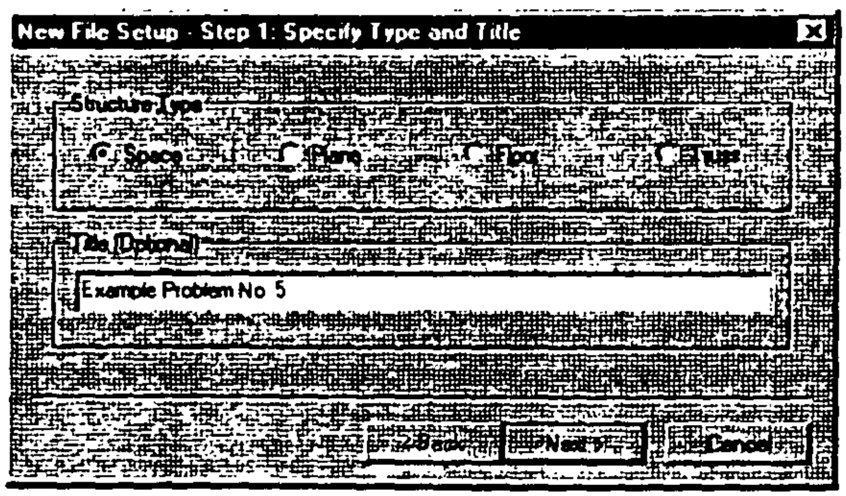


Figure 5. 5

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.

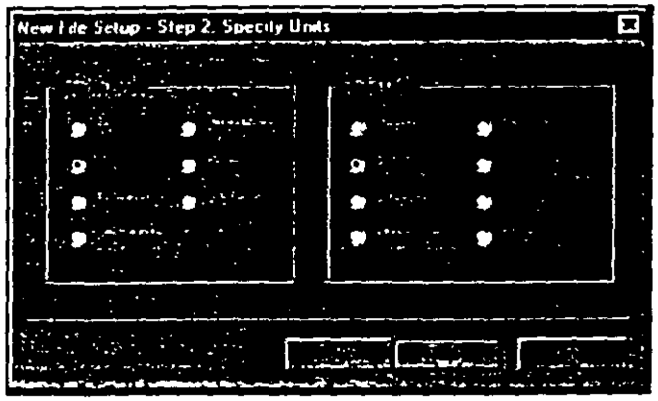


Figure 5. 6

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

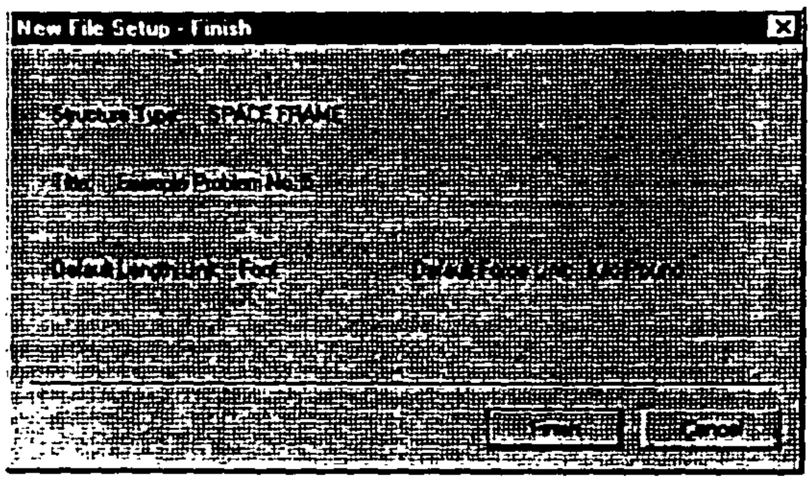


Figure 5. 7

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

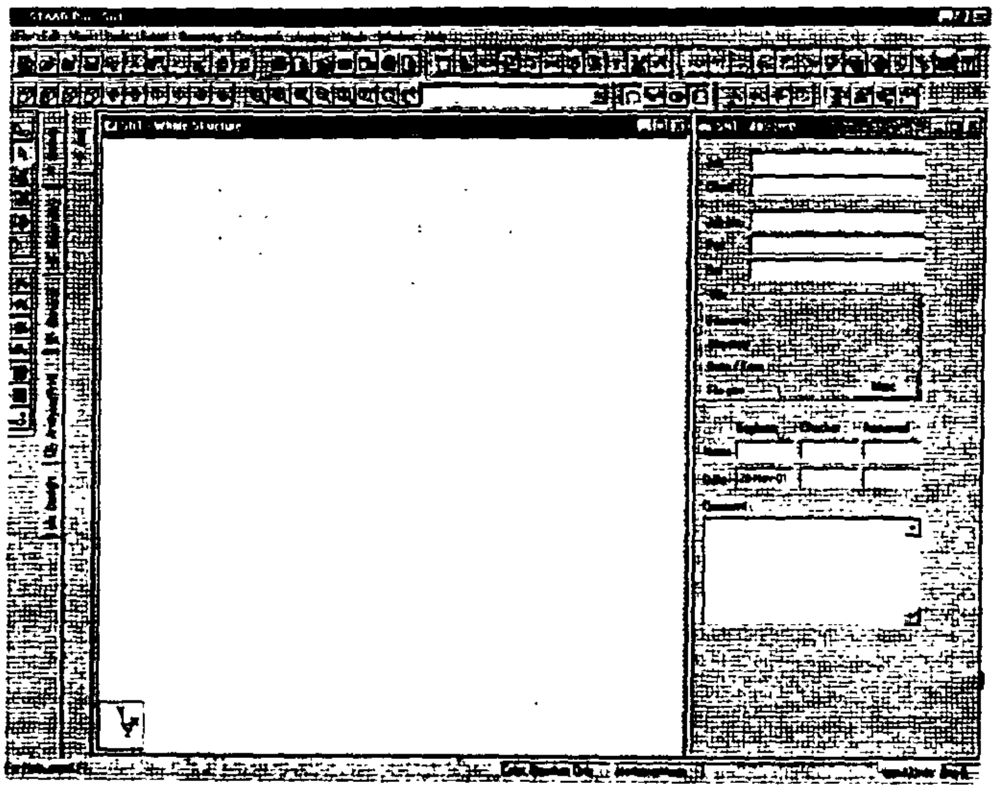


Figure 5. 8

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 *1*ft. (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.

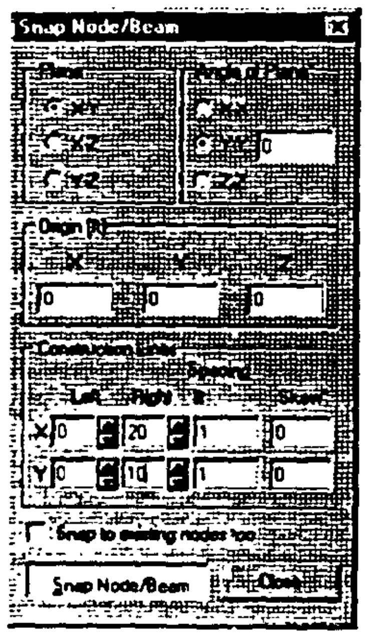


Figure 5. 9

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.

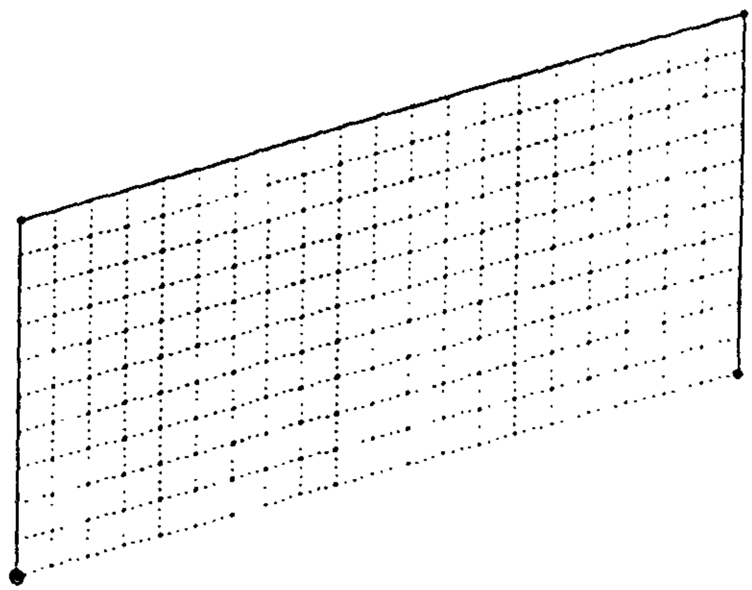


Figure 5. 10

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

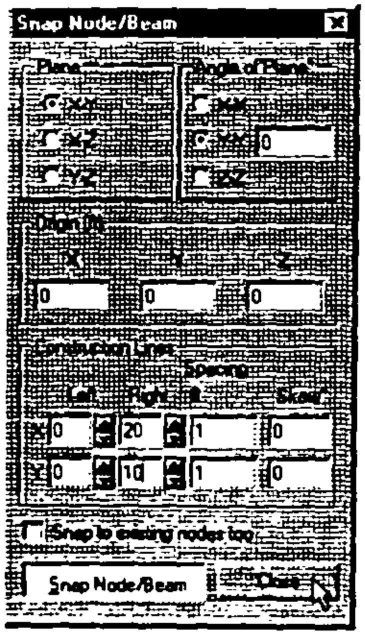


Figure 5. 11

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.

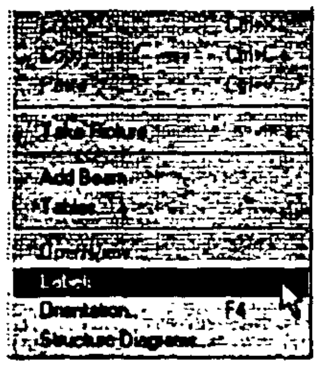


Figure 5. 12

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)

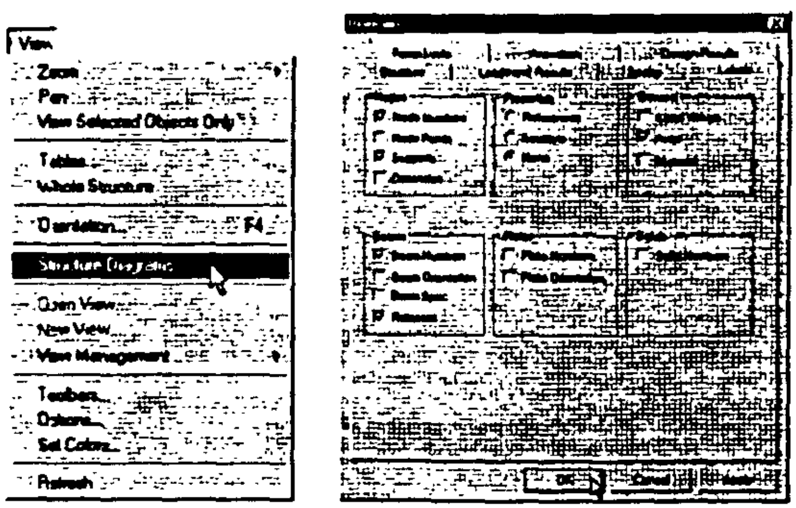


Figure 5. 13

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

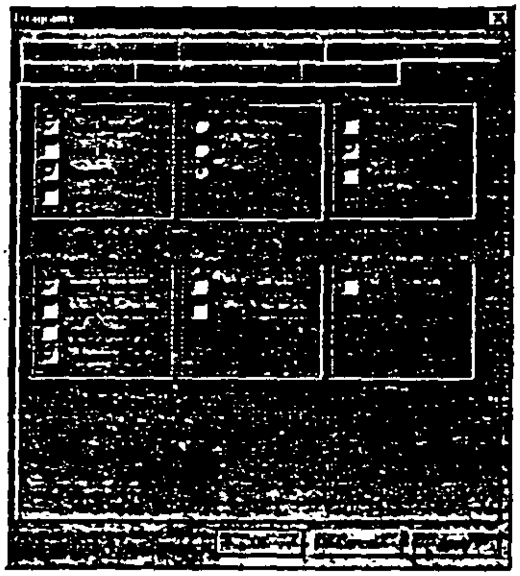


Figure 5. 14

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

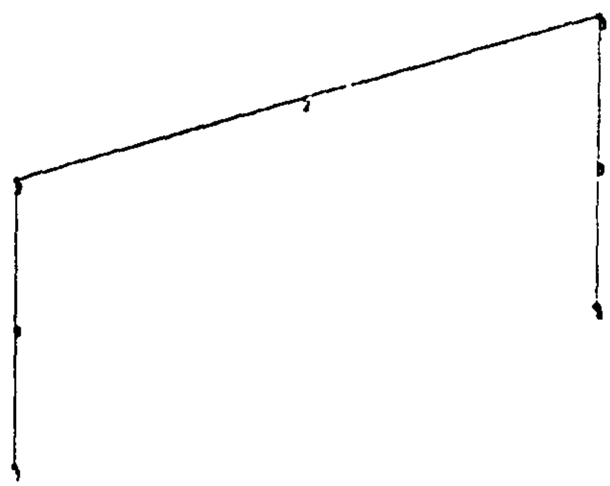


Figure 5. 15

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.

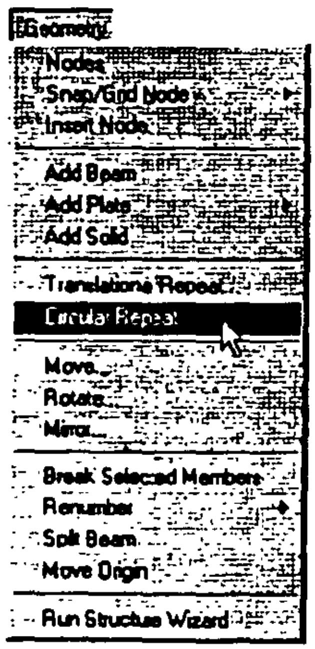


Figure 5. 16

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 <u>unchecked</u> and click on the *OK* button. (see figure below)

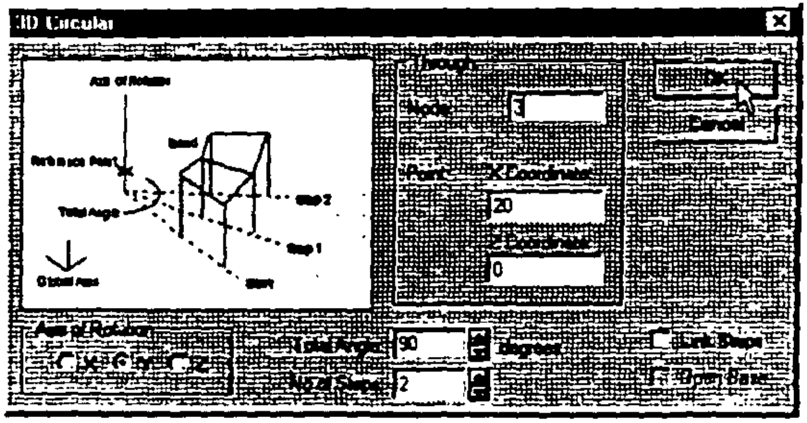


Figure 5. 17

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

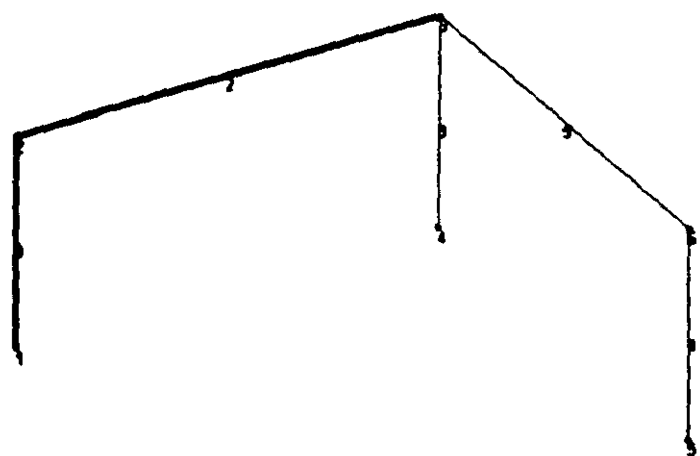


Figure 5. 18

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.

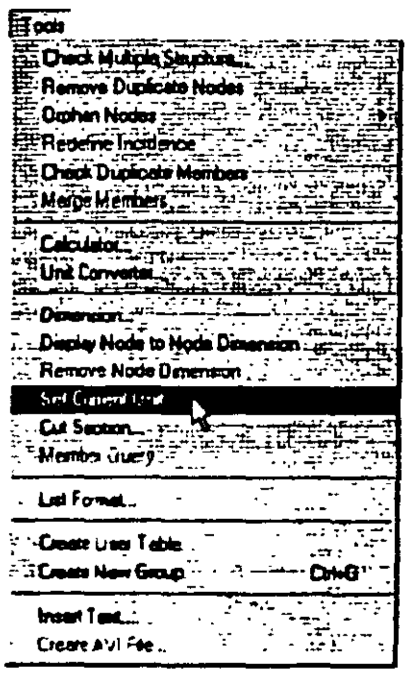


Figure 5. 19

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

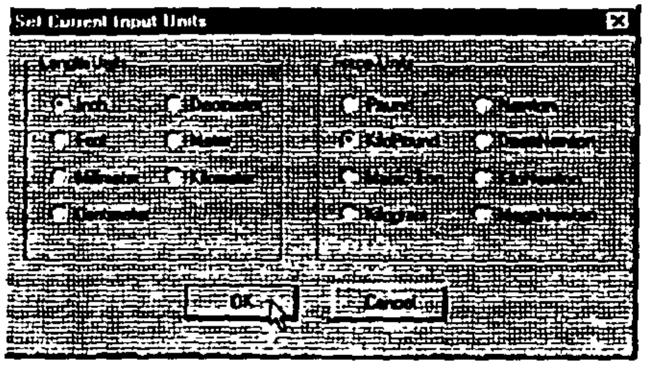


Figure 5. 20

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.

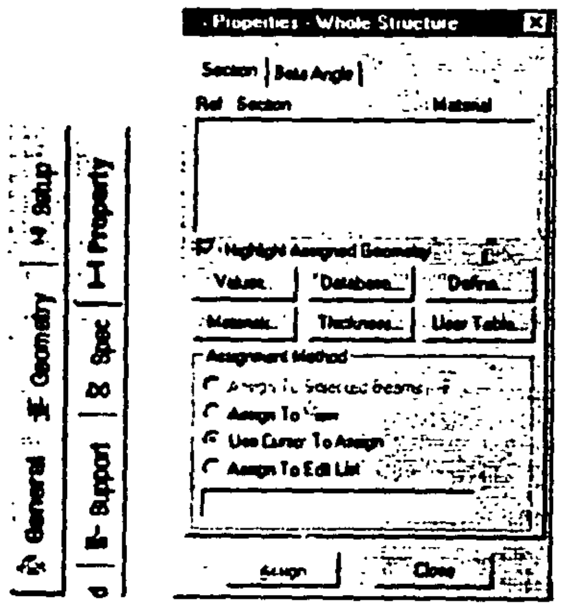


Figure 5. 21

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.

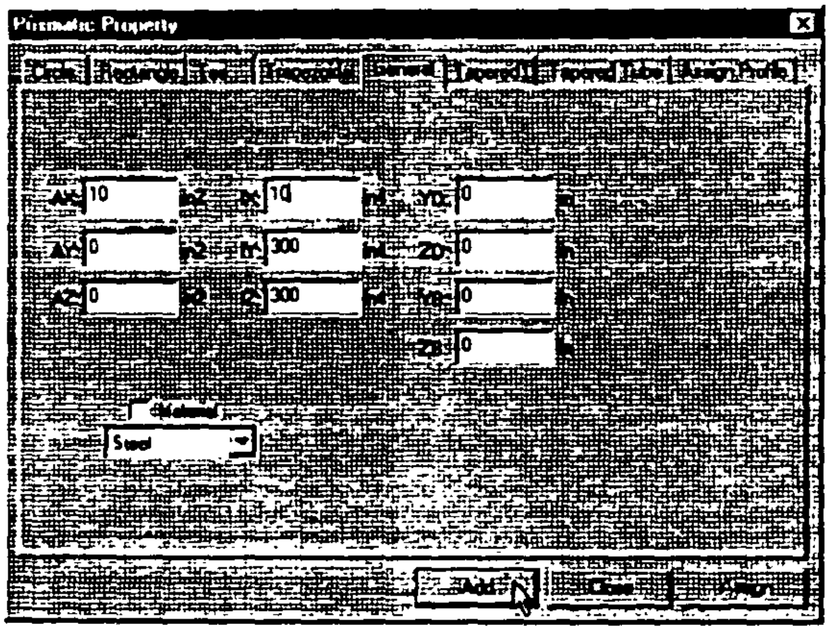


Figure 5. 22

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.

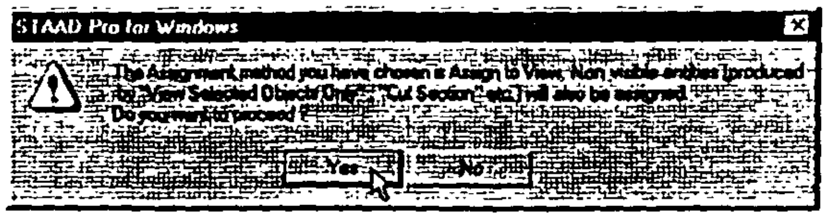


Figure 5. 23

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

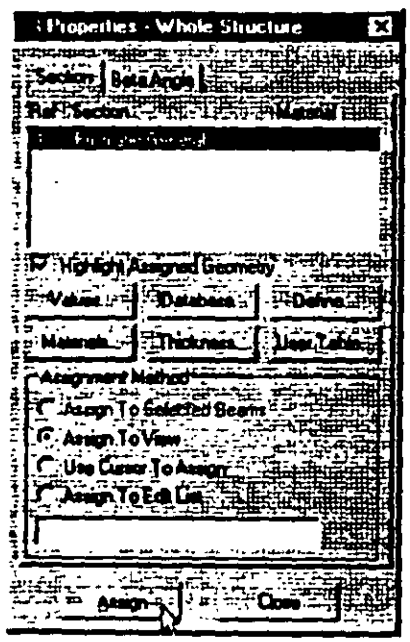


Figure 5. 24

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

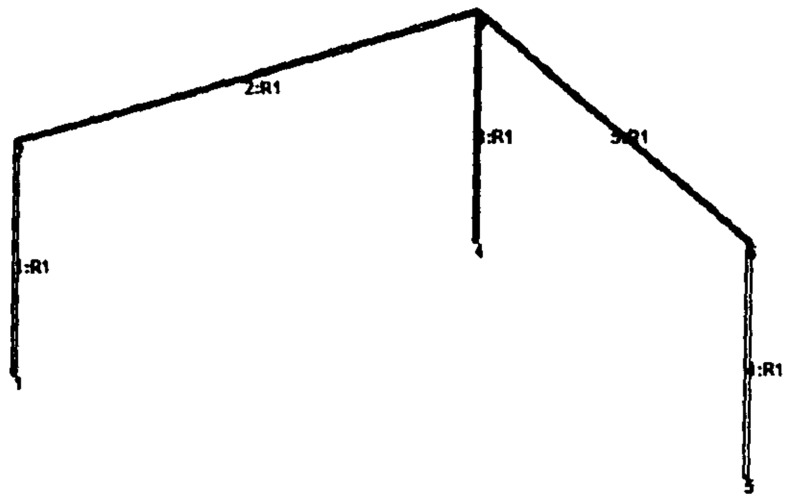


Figure 5. 25

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.

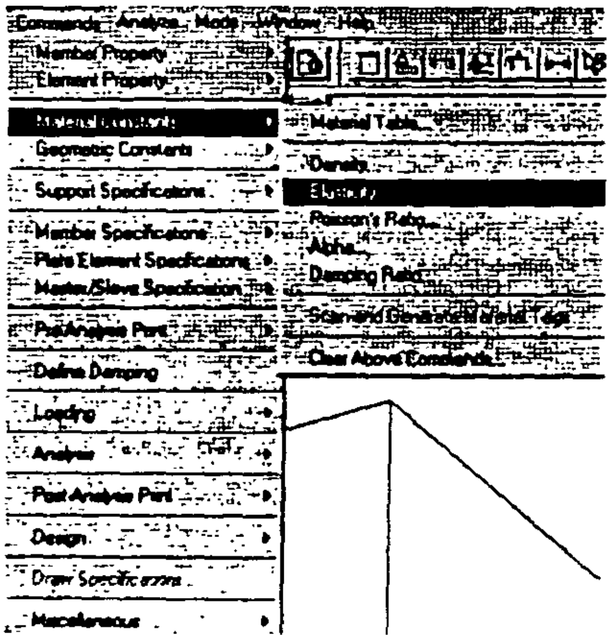


Figure 5. 26

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)

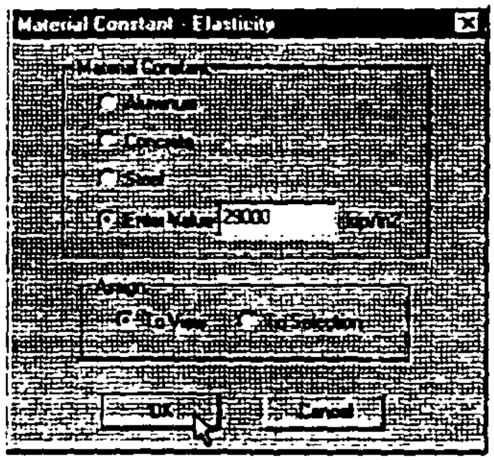


Figure 5. 27

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:

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)

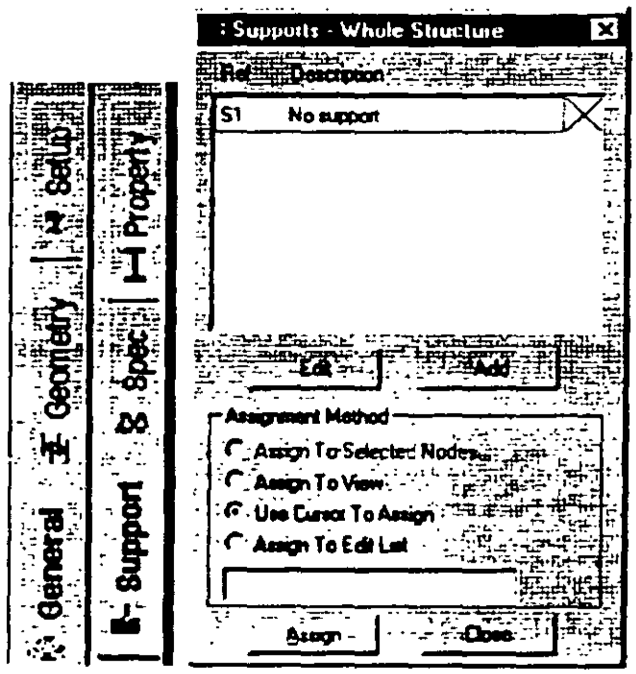


Figure 5. 28

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.

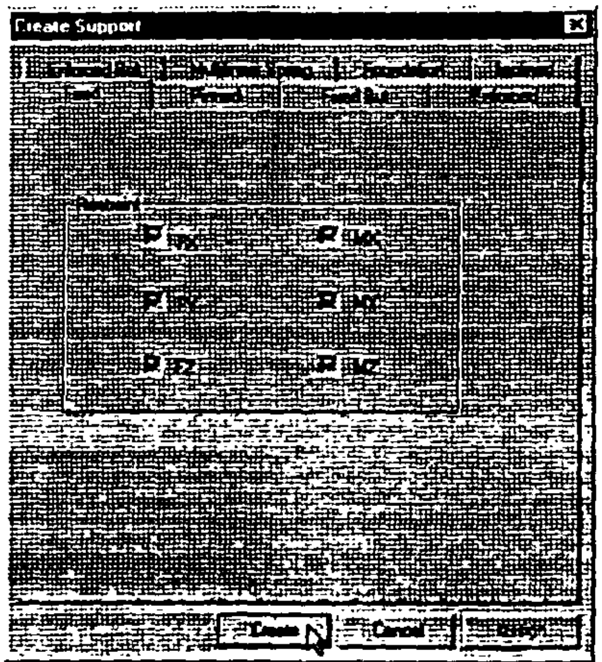


Figure 5. 29

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.

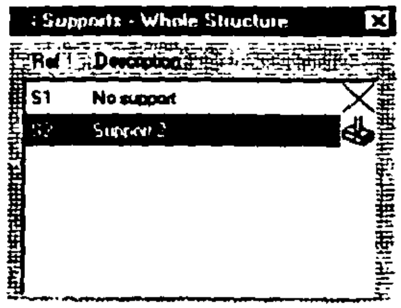


Figure 5. 30

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)

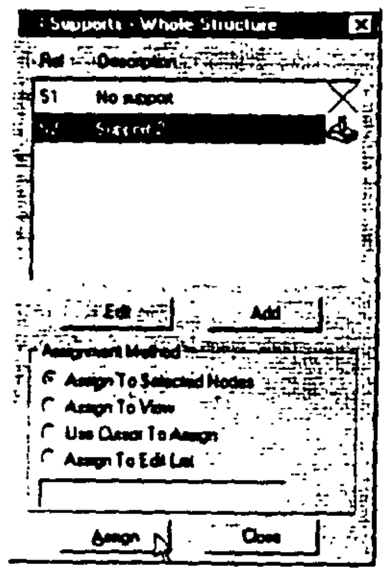


Figure 5. 31

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*.

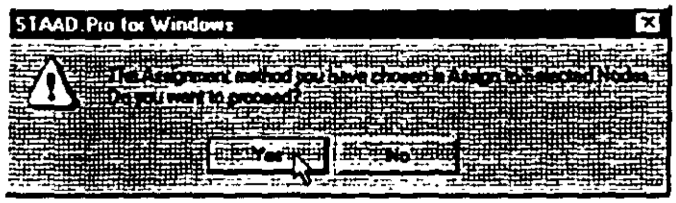


Figure 5. 32

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

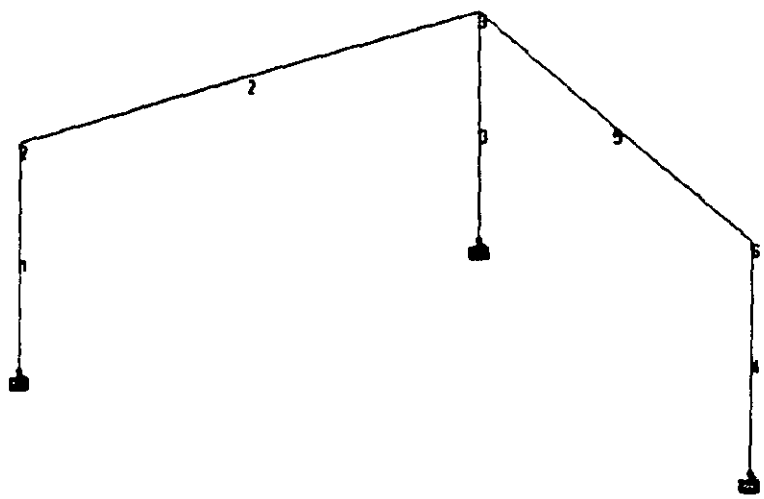


Figure 5. 33

Specifying Loads

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

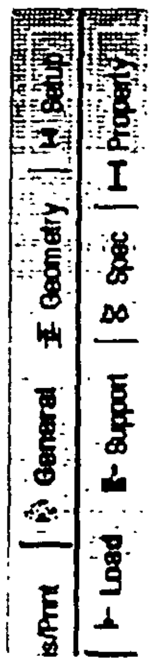


Figure 5. 34

For load case 1, we wish to apply a ½ 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)

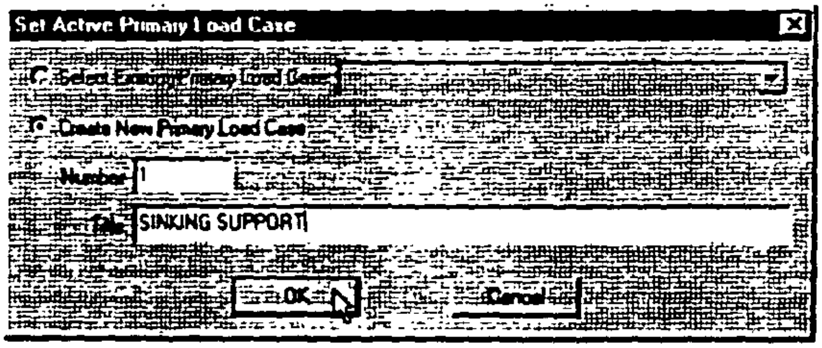


Figure 5. 35

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)

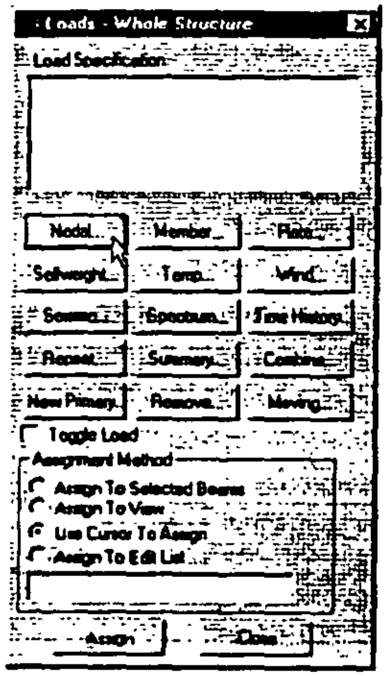


Figure 5. 36

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

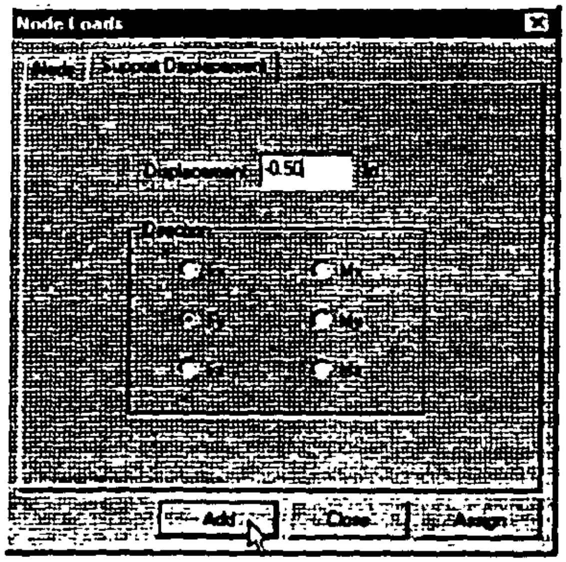


Figure 5. 37

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).

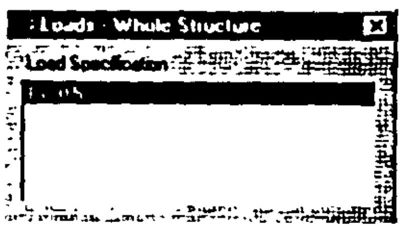


Figure 5. 38

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)

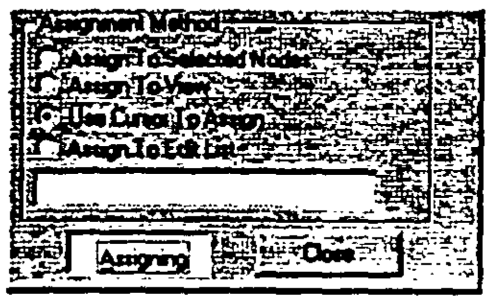


Figure 5. 39

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.

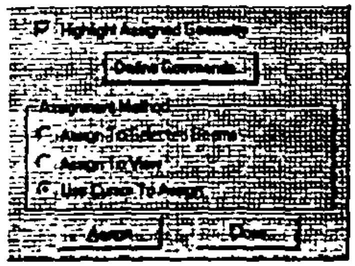


Figure 5. 41

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.

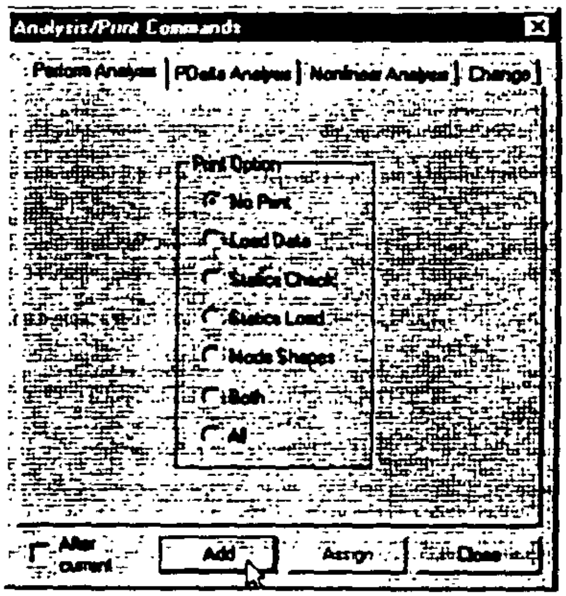


Figure 5. 42

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

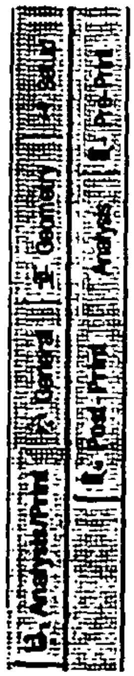


Figure 5. 43

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

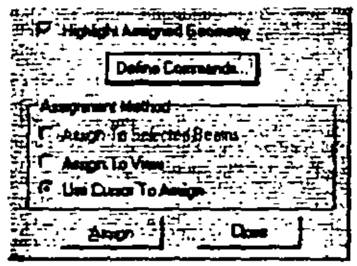


Figure 5. 44

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.

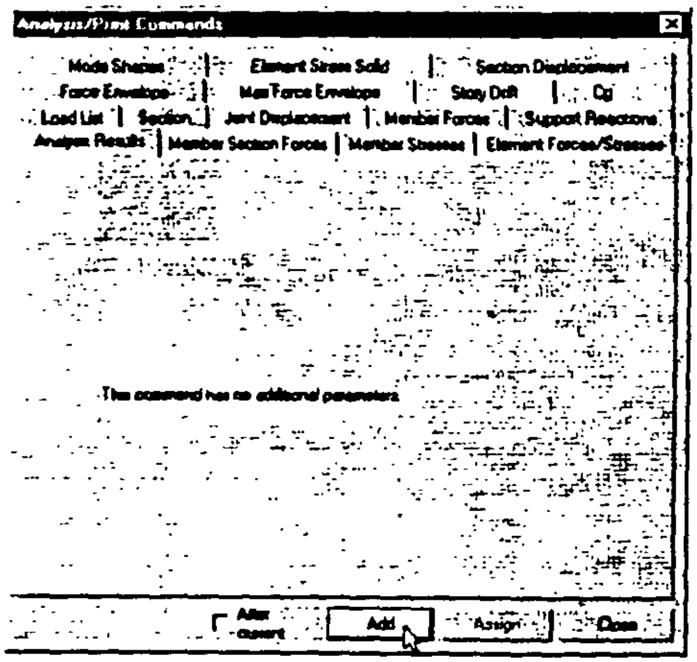


Figure 5. 45

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.

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

CURSOS ABIERTOS

STAAD-PRO
PARA ANÁLISIS Y DISEÑO ESTRUCTURAL
CA 003

TEMA

STAAD. Pro 2002 - GETTING STARTED TUTORIAL

EXPOSITOR: ING. FERNANDO MONROY MIRANDA
PALACIO DE MINERÍA
ENERO DEL2003

STAAD.Pro 2002 – Getting Started Tutorial

A portal frame type steel structure is sitting on concrete footings. The soil is to be considered as an elastic foundation. The value of soil subgrade reaction is known from which spring constants are calculated by multiplying the subgrade reaction by the tributary area of each modeled spring.

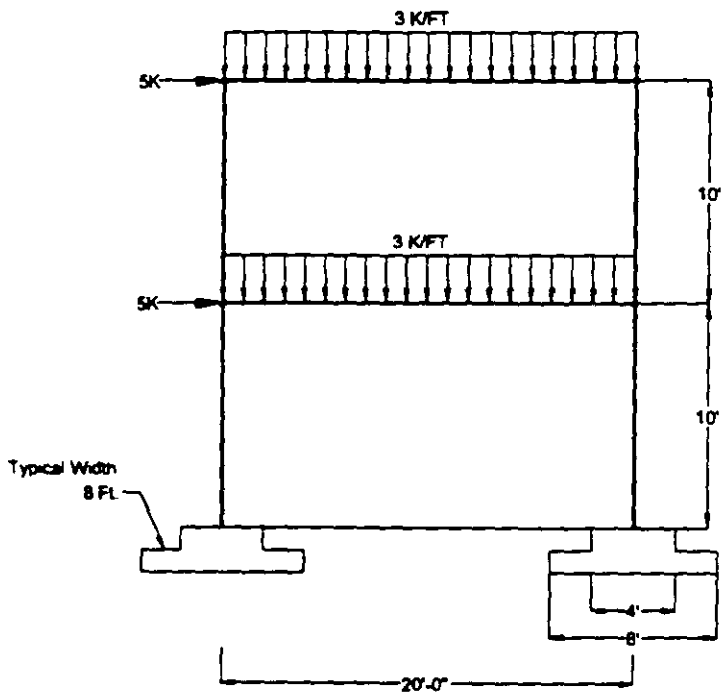


Figure A

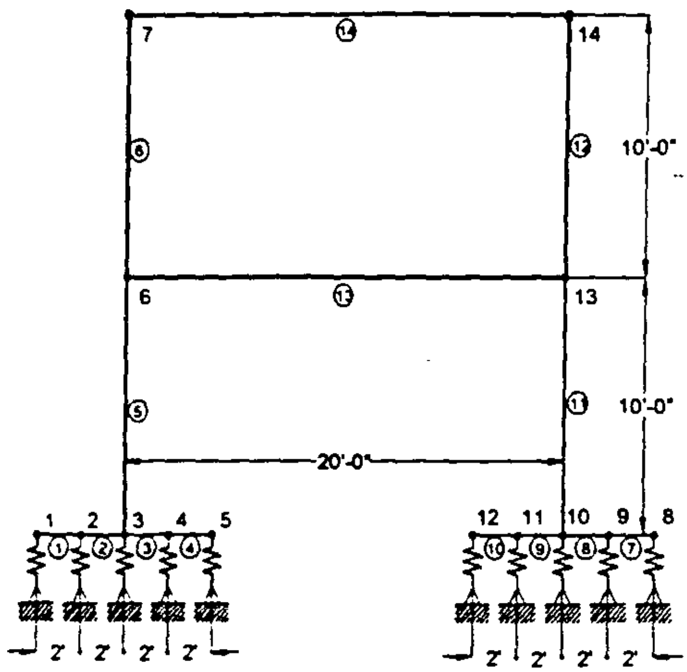


Figure B

Columns* 5, 6, 11 and 12 - W10X33 from AISC Table

Beams 13 and 14* - W12X26 from AISC Table

*All member (columns and beams) numbers are encircled while node numbers are not.

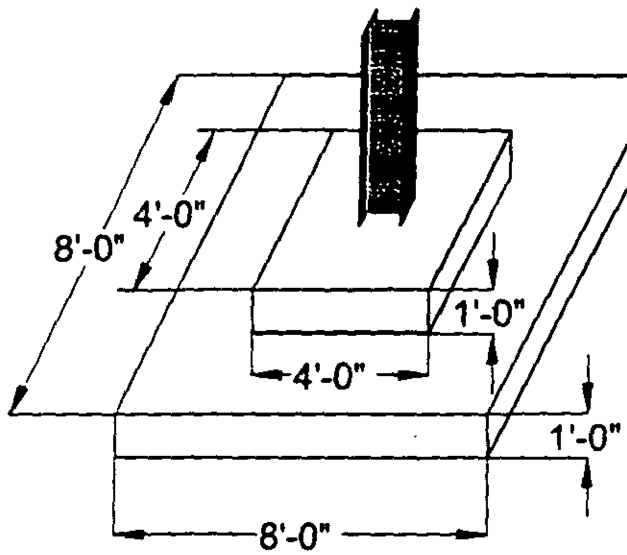


Figure C

Magnified view of the column-footing junction
This layout represents beams 1-4 and 7-10 in the main model.

NOTE:

1) All dimensions are in feet.
2) Soil Subgrade Reaction - 250 kips/ft^3.

<u>Spring constant calculation</u>

Spring of joints 1, 5, 8 & 12 = 8 x 1 x 250 = 2000 kips/ft

Spring of joints 2, 3, 4, 9, 10 & 11 = 8 x 2 x 250 = 4000 kips/ft

There are two methods of creating the structure data:

a) using the graphical model generation mode, or graphical user interface (GUI) as it is usually referred to.
b) using the command file.

Both methods are explained in this chapter also. The graphical method is explained first from section 1 onwards. Section - describes how to view the file using the STAAD.Pro Editor.

1. Starting The Program

Steps:

1. If you have already started the STAAD.Pro 2002 program, you can skip this step. Select the *STAAD.Pro* icon from the STAAD.Pro 2002 program group.

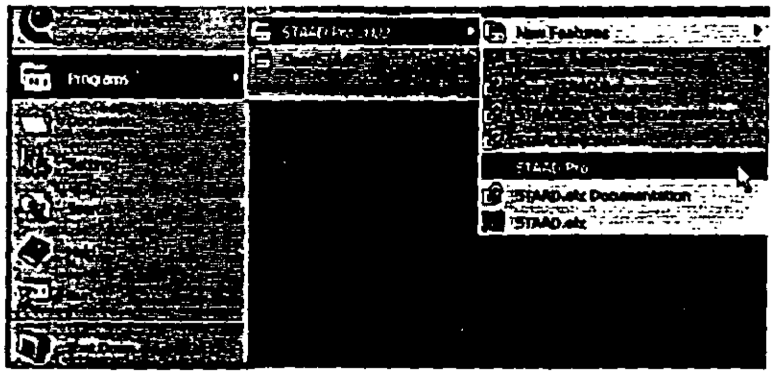


Figure 3. 1

The STAAD.Pro Graphical Environment will be invoked.

The File New dialog box will prompt you to enter certain information about the model. The explanations for each of these input items are as follows:

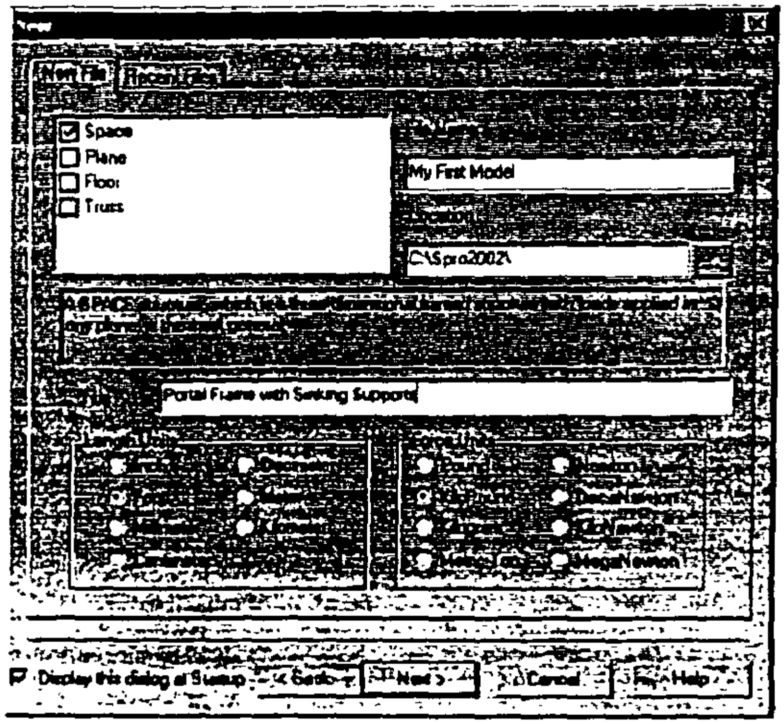


Figure 3. 2

Structure Type	The explanations for each of the structure types are shown in the space below the checkboxes. In our example, we will choose "Space"	Yes
File Name	The name of your STAAD file This is what the file will be saved as We will	Yes

	choose "My First Model"	
Location	The directory (path) in which this file will be saved. You can use the  button to browse for a directory.	Yes
Title	A brief comment about your model in which to identify your structure at a future time. We will use "Portal Frame with Sinking Supports"	No
Length Units	Select the units of length you wish to use in the model. By default, the units are those that we chose when installing the program. We will keep the default – "Foot"	Yes
Force Units	Select the units of force you wish to use in the model. By default, the units are those that we chose when installing the program. We will keep the default – "KiloPound" (kips)	Yes
Display this Dialog at Startup	If checked, this dialog box will always popup when STAAD.Pro is started. If it is unchecked, you will have to go to File | New or File | Open from the top menu bar to access a file.	No

In the File New box in Figure 3.2, there is another tab labeled "Recent Files". When you come back into STAAD.Pro 2002, you can select this tab and choose from a list that includes the last 20 files opened. In the Recent Files tab-page, there is also another button labeled "Other" which will allow you to open another STAAD file from anywhere on your computer.

Configure Units

The units in which we wish to create this example model are the English units (feet, kip, etc.). The default unit system setting is whatever we chose during the installation of the program. Thus, the File New box in Figure 3.2 will always start with the default English units. If you had chosen Metric at the time of installation and want to change it to English, go to the File | Configure menu option, click on the Base Unit tab (see Figure 3.3) and choose the appropriate unit system. Then, click on the *Accept* button.

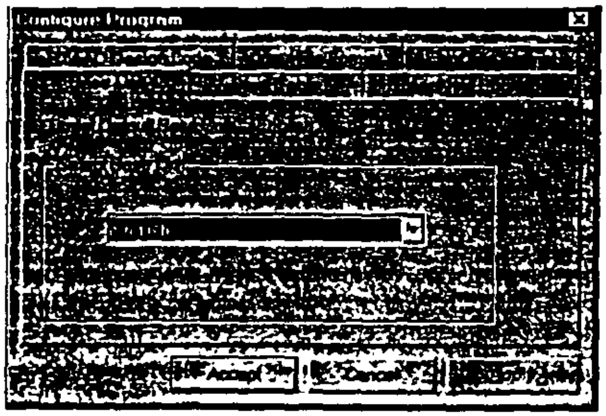
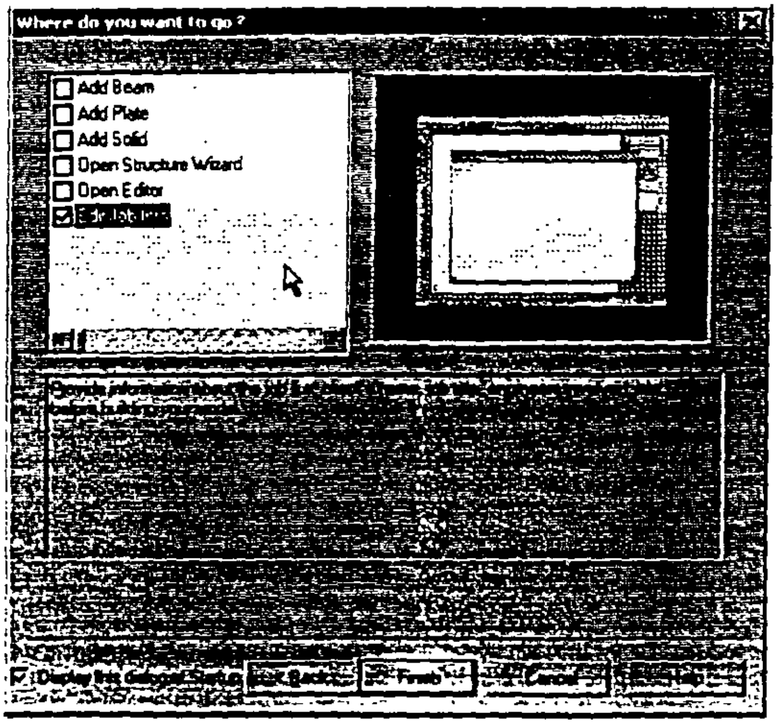


Figure 3. 3

Once your input matches that of Figure 3.2, click on the "Next" button to go to the next page.

In the next page of input, the program will prompt you on how or where you would like to start using the program (see Figure 3.4)



. **Figure 3. 4**

A description of each of the choices is displayed below the picture. We will select "Edit Job Info" so we can input some information about the project including the client, engineers working on the project, etc. Click "Finish" to enter the STAAD.Pro2002 environment in the Job Info page (see Figure 3.5).

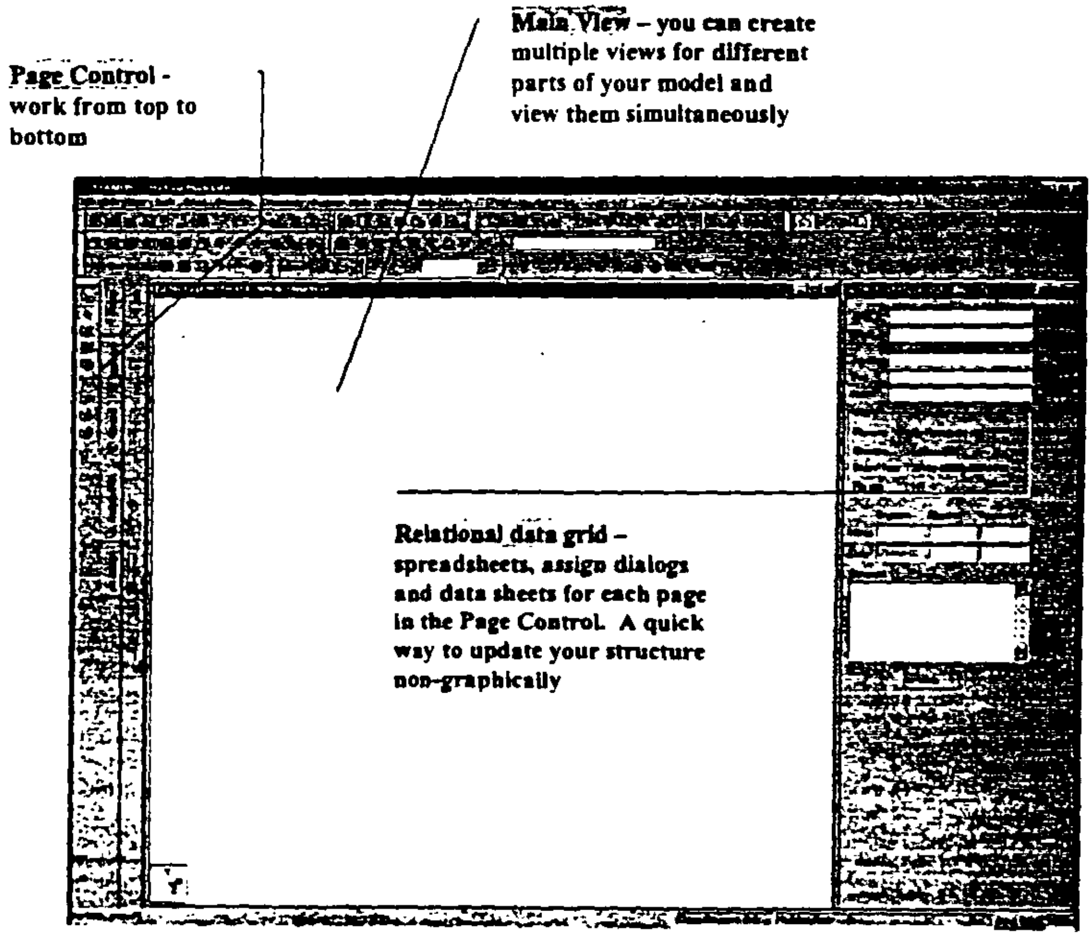

Main View – you can create multiple views for different parts of your model and view them simultaneously
Page Control - work from top to bottom
Relational data grid – spreadsheets, assign dialogs and data sheets for each page in the Page Control. A quick way to update your structure non-graphically

Figure 3. 5

2. Editing the Job Information Page

Topics: Adding information to your job, finding out the status of your model

1. Once you have entered the STAAD.Pro graphical environment, you can begin inputting information about your model in the Job Info page. The Job Info information is optional and is used in the header when printing your final reports. The information can be updated at any time by simply clicking on the Setup button from the Page Control tab (vertical buttons on the left-hand side of the screen).

Figure 3.6

The Job Info page (Figure 3.7) provides useful information about the structure including when it was created, the size of the model and what parameters are missing from the model (the latter two can be found by clicking on the More... button from the Job Info page or the icon from the toolbars on the top) The Info dialog box is shown in Figure 3.8.

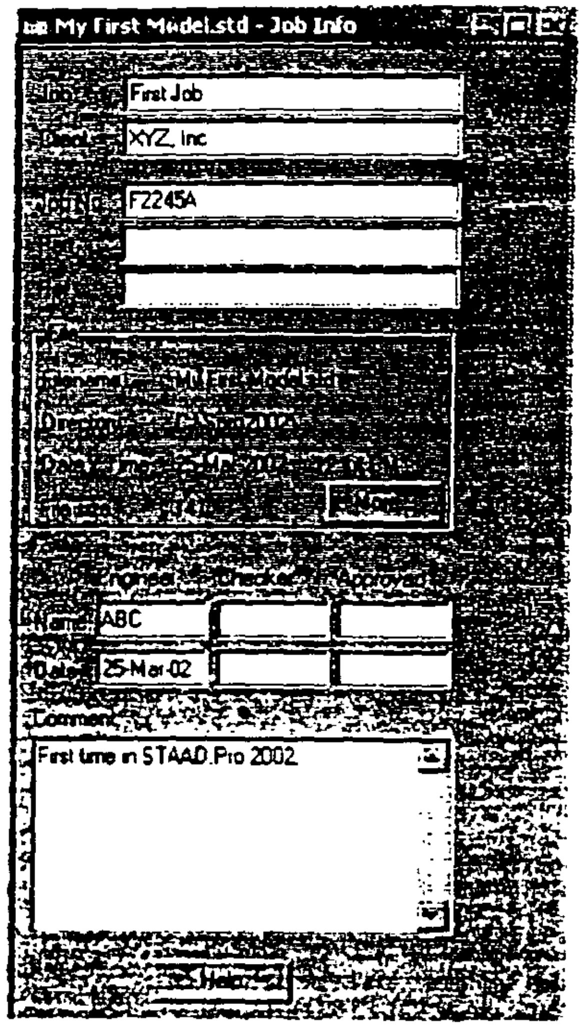

My First Model.std - Job Info
First Job
XYZ, Inc
FZ245A
ABC
25-Mar-02
First time in STAAD.Pro 2002

Figure 3.7

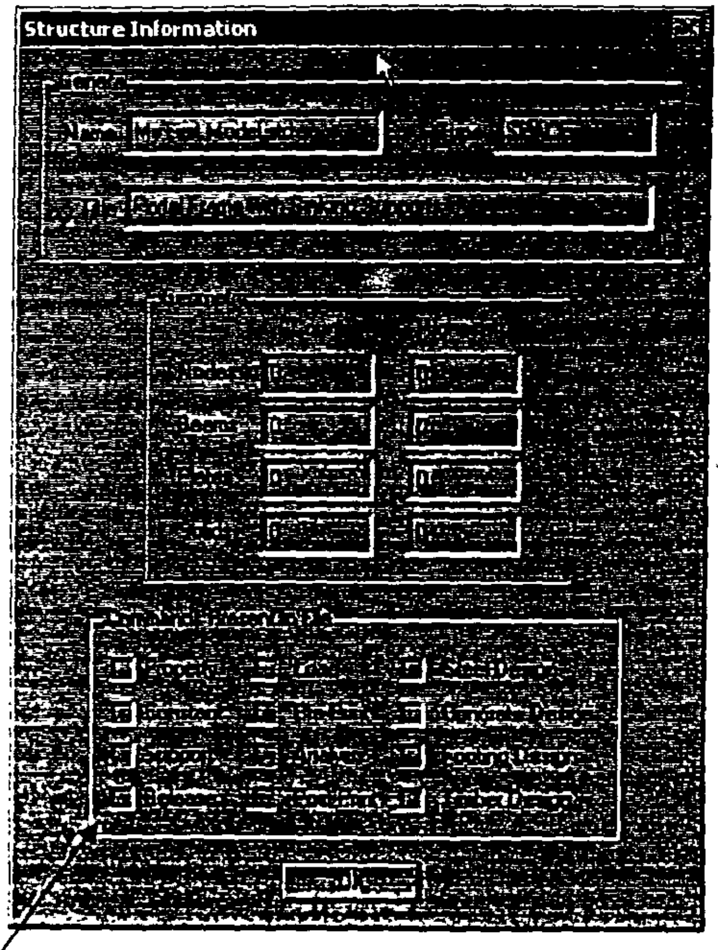


Figure 3.8

If a box is not checked, that specific command has not been added to your model.

3. Creating Nodes 1 to 5 And Members 1 to 4

Topics: Using the construction grid lines to add nodes and beams, roaming grid labels

1. From the page control tab on the left-hand side (vertical buttons), select *Geometry* | *Beam* to begin adding beams to your model (by default, this will also add nodes). You may also notice that facilities for adding in Plates and Solids also exist. The following steps can easily be applied to the creation of these other entities as well.

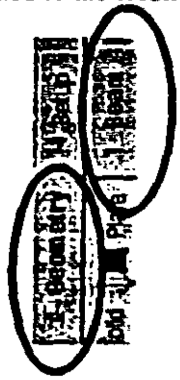
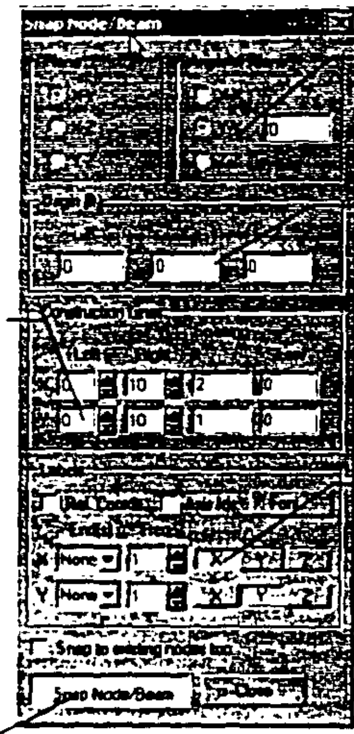


Figure 3.9

2. In the *Snap Node/Beam* dialog box that appears in the Data Grid area (on the right side of the screen), choose *X-Y* as the *Plane* Members 1 to 4 are located in the range of X = 0 to 8 and Y = 0 to 0. The construction lines, spaced 1ft apart, and numbering 10 along the X direction (0 to the left of the origin, and 10 to the right of the origin), and 10 along the Y direction (0 below the origin, 10 above the origin) already allow us to draw a structure which spans 10ft each in X and Y. So, there is no need to change the number of construction lines horizontally or vertically. But, since members 1 to 4 are each 2ft long, it will be convenient to draw them if the grid lines along X are set to 2ft apart. (It is interesting to note that, by doing so, 11 construction lines - the vertical line passing through the origin counts as one line - at 2ft spacing between adjacent lines

allow for a 20ft span along X.) Hence, set the *Spacing* of grid lines along *X* to *2ft*. (see Figure 3.10)



Skew the angle of the construction grid plane about an axis

Change the origin of the grid line

Change the number of gridlines to the left and right of the origin. To add in coordinates with negative values (assuming origin is at 0,0,0), increase the number of gridlines to the left.

Add in roaming gridlines so the gridlines have the coordinates labeled on the grid.

Figure 3. 10

Click off (not depressed) to turn off automatic snapping and creating of

3. Using your left mouse button, click at the origin (0, 0) (marked by a circle) to create the first node. Then, click on the following points to create nodes 2-5 (see schematic of original structure in Figure A).

   (2, 0), (4, 0), (6, 0) and (8, 0).

   We will notice that beams are automatically created between successive nodes. Also, notice that the status bar (Figure 3.11) located at the bottom right-hand corner of the window continuously updates the X, Y, and Z coordinates of the current cursor position when the mouse is moved over the drawing area. You can also turn on the roaming grid labels (Figure 3.10) to see the coordinates on the gridlines.

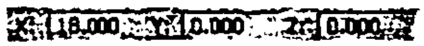


Figure 3.11

The "Hot" Node

The node highlighted in red is the "hot" node. The next click on the grid will automatically create a beam from the "hot" node to the next click. Each click will create a node at that point. To reset the "hot" node, simply hold your "Ctrl" key down and click on another point. After the nodes and beams have been created, the structure will look as shown below
The construction gridlines along with the Snap/Grid Node Beam dialog box can be turned on/off at anytime through the icon from the toolbar or from *Geometry Snap Grid/Node Beam*.

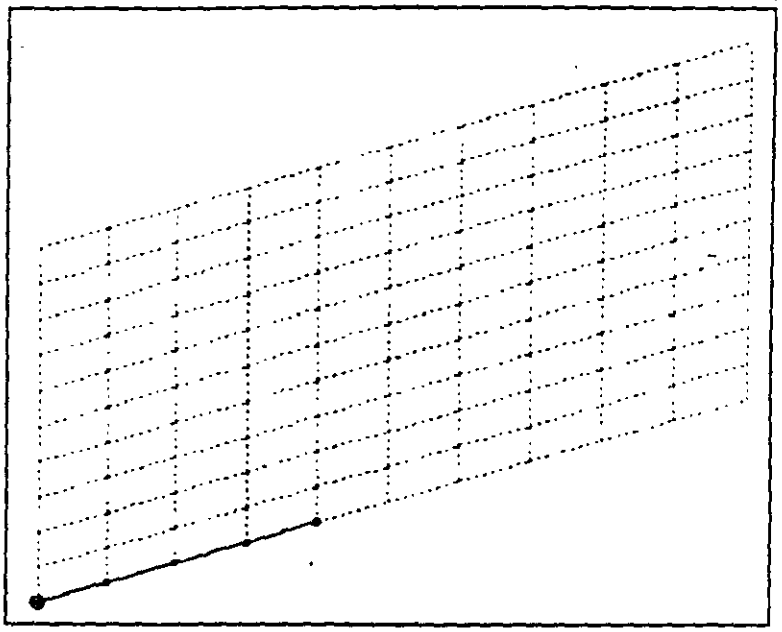


Figure 3. 12

4. To temporarily switch off the drawing mode, click on the *Snap Node | Beam* button once again to deactivate the snap option.

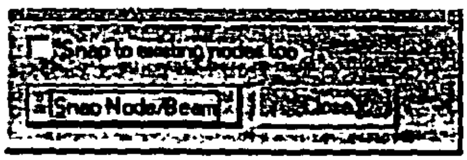


Figure 3. 13

With "Snap Node/Beam" button in its dormant state, we will be prevented from creating a member in the drawing area resulting from accidental clicks of the mouse.

5. Let us save the file by going to the *File | Save* menu option.

3. Switching On Node and Beam Labels

Topics: Turning on labels, changing their look and modifying display units

1. Nodes and beam labels are a way of identifying the entities we have drawn on the screen. In order to display the *node* and *beam numbers*, right click anywhere within the structure. In the dialog box that opens, choose *Labels* (as shown in the figure below). Alternatively, one may access this option by selecting the *View* menu from the top menu bar followed by *Structure Diagrams*. Select the *Labels* tab of the dialog box that opens. Most of the widely-used functions can be accessed by right-clicking your mouse in the main view.

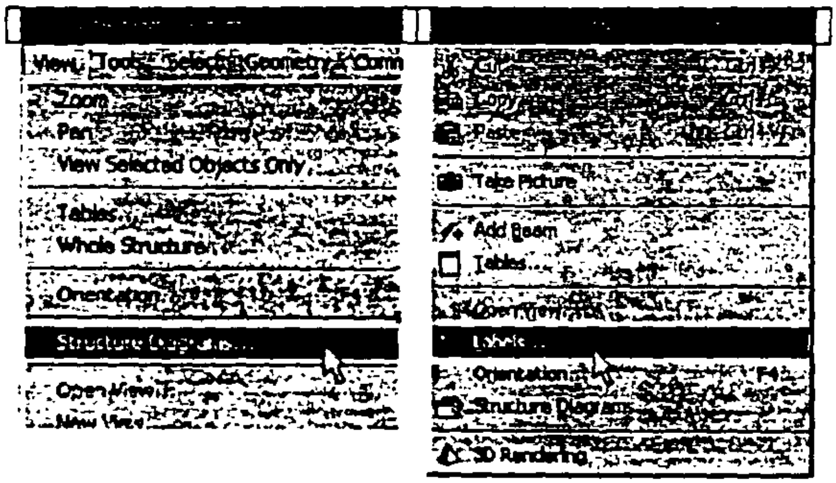


Figure 3.14

2. In the *Diagrams* dialog box that appears, turn the *Node Numbers* and *Beam Numbers* on and then click on *OK*. (see figure below)

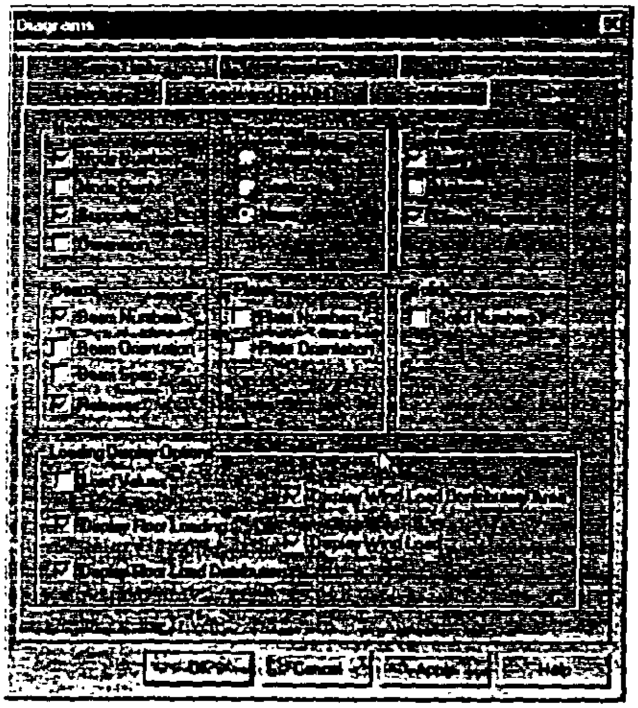


Figure 3.15

The following figure demonstrates the node and beam numbers displayed on the structure.

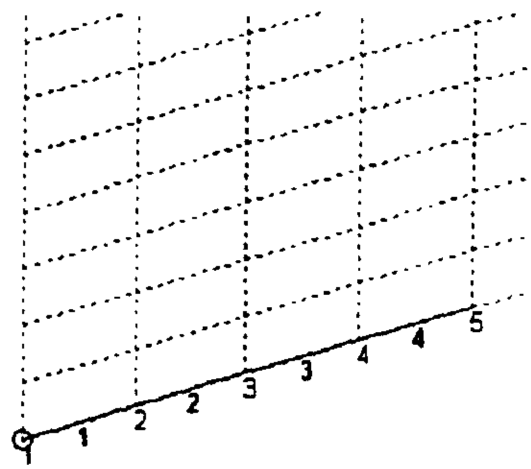


Figure 3.16

By default, all beam, node, plate and solid labels are colored in black. To change the colors and the positions of these labels, go to *View | Options* from the main menu and click on the appropriate tab (Node Labels for nodes, etc.)

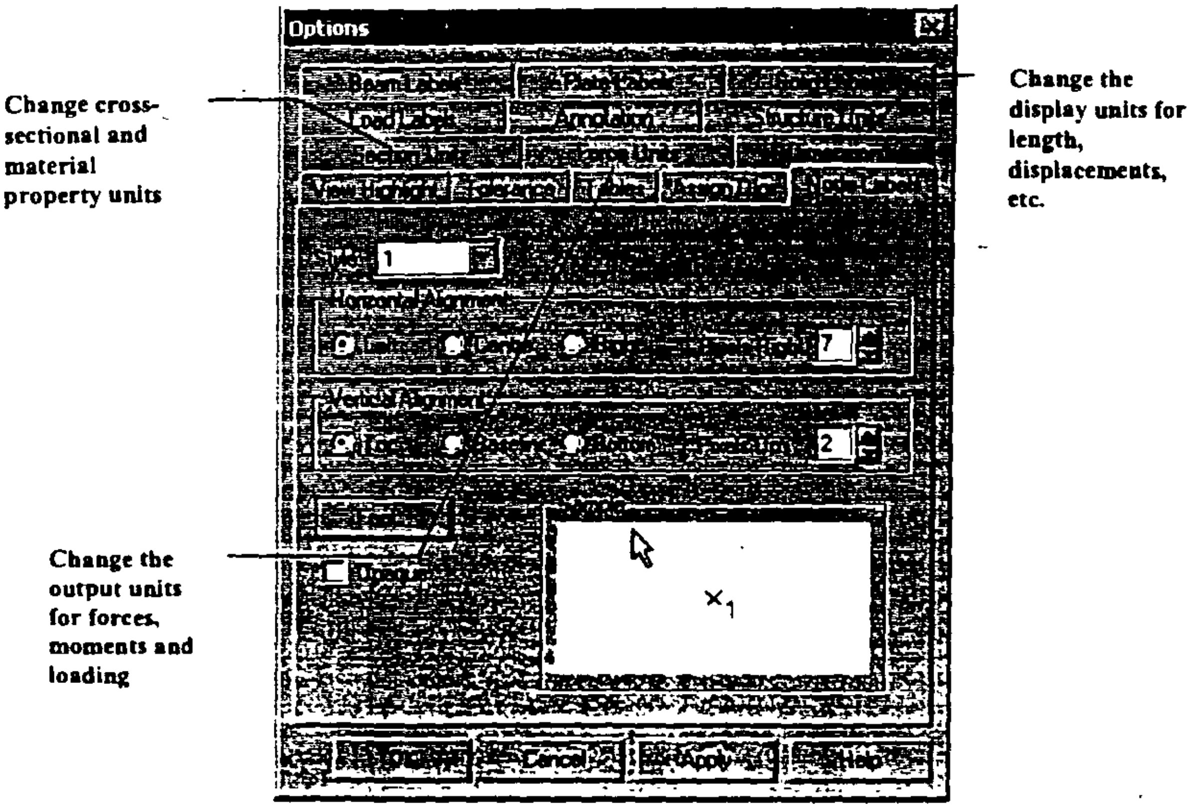


Figure 3.17

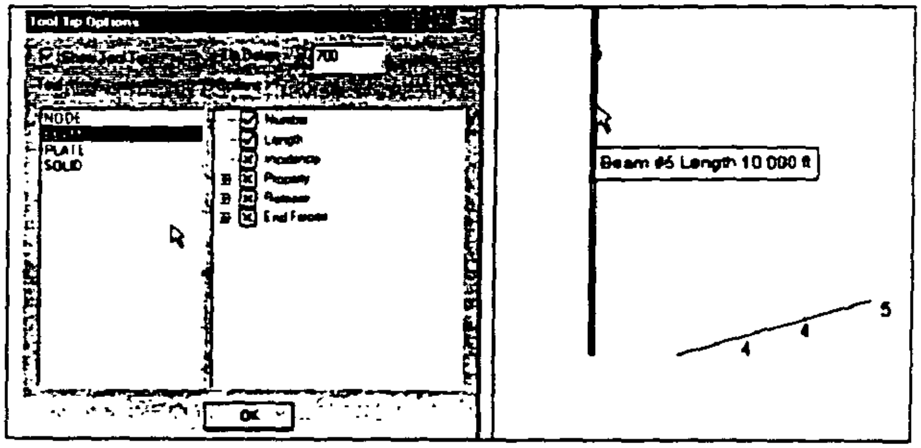
In this example, we can offset the node labels by 7 pixels to the right and 2 pixels up. Click on the "Font" button to change the color to navy. Hit "OK" to have the settings take effect.

Turning On/Off Some Labels

To turn on/off the labels for some nodes or beams, you must first create a group by selecting the entities you want in the group, creating a new group (*Tools* . *Create New Group*) and then going to *Select* . *By Group* to turn the group on or off.

Structural Tool Tips

You can quickly find out critical information about a particular entity (node, beam, plate, etc.) by selecting the selection cursor for that entity (please refer to the 'Task Reference' section at the end of this chapter to learn about selecting entities) and then simply hovering your mouse over the entity you want more information about. You can customize what you want to see in the tool tip by going to *View | Structural Tool Tip Options*.



4. Creating Member 5 and Node 6

Topics:

1. To create member 5, reactivate the drawing mode by clicking on the *Snap Node/Beam* button again.

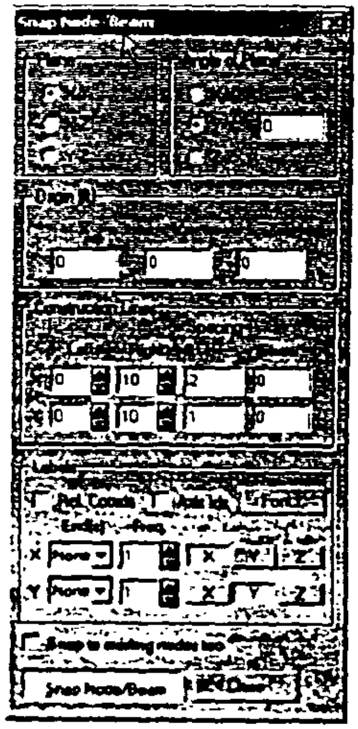


Figure 3.18

2. Then, click at node 3. Click again at the point (4, 10). We will see that member 5 has been created as shown in Figure 3.19. In this example, the roaming grid labels were turned on from Snap Node/Beam dialog box so the point (4, 10) can be readily identified.

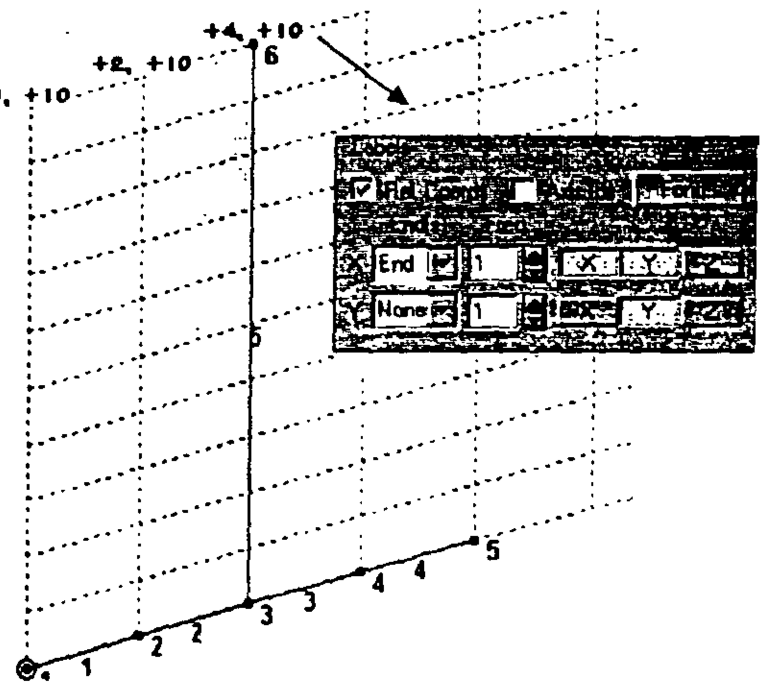


Figure 3.19

3. At this point, let us close the *Snap Node/Beam* dialog box by clicking on the "Close" button.
4. Let us also save the model we have created so far by going to the *File* | *Save* menu option.

5. Creating Member 6 and Node 7

Topics: Selecting members, copy/paste

1. To create member 6, we shall utilize the *Copy-Paste* facility available either under the *Edit* menu, right-clicking your mouse button (Figure 3.20) or using the shortcut keys "Ctrl + C" and "Ctrl + V". To apply this method, first select member 5. (Please refer to the 'Task Reference' section at the end of this chapter to learn about selecting members.) Click the right mouse button and choose *Copy* from the pop-up menu (or press "Ctrl + C"). Once again, click the right mouse button and select *Paste Beams* (or press "Ctrl + V") as shown below. It is important to note that STAAD.Pro follows the Microsoft convention for all of its standard shortcuts (CTRL+C, +V, +X, +Z, +Y and +A)

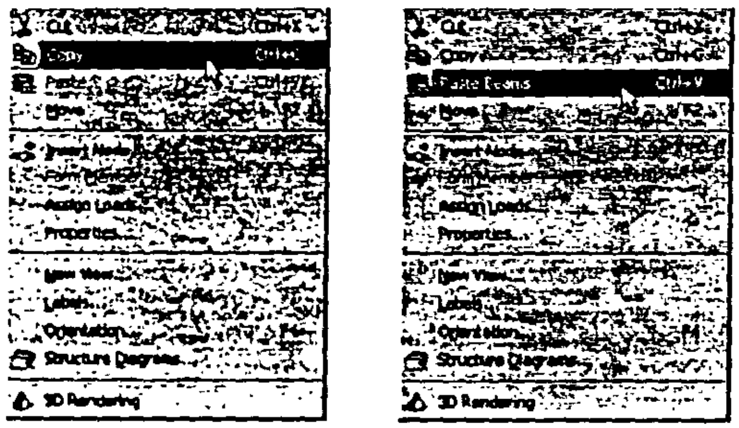


Figure 3.20

Quick Methods to Select Objects

Selection icons (for nodes, for beams, for plates, for solids and for all geometry)	Simply click on an entity (described by the selection cursor) and click again to deselect. Hold the "Ctrl" key down to perform multiple selections.	When you want to pick a few entities at a time
Rubber band window	Hold your left-mouse button down and drag your mouse to create a window. Everything within that window will be selected.	When you want to select a lot of entities at one time.
Ctrl + A	Selects everything on the screen	When you want to select everything.
Select a row from the grid table	Selects the entity corresponding to that row	When you want to find an entity without having to turn on labels.

You can also go to the Select menu to find other ways of selecting objects.

2. Since the two ends of member 6 are 10ft above the corresponding nodes of beam 5, provide 0, 10, and 0 for X, Y and Z respectively and click on the *OK* button. (see figure below). You could also use the "Reference Point" button to graphically select the new position to paste to.

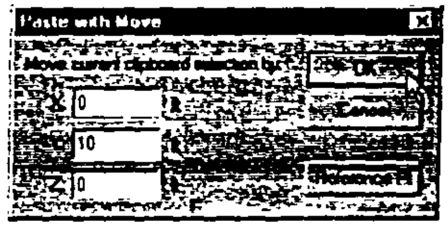


Figure 3.21

The directions for X, Y and Z are respect to the global axis system. The global axis system is displayed in the lower-left hand corner of your main view. As we click on the *OK* button, the following message box comes up. This is only a reminder that we need to subsequently assign the required properties to these entities somewhere down the line. Click on the *OK* button to continue.

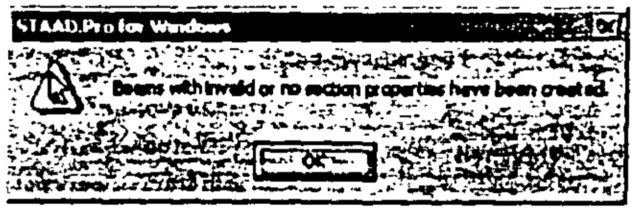


Figure 3.22

The following figure shows the structure with the newly created member 6.

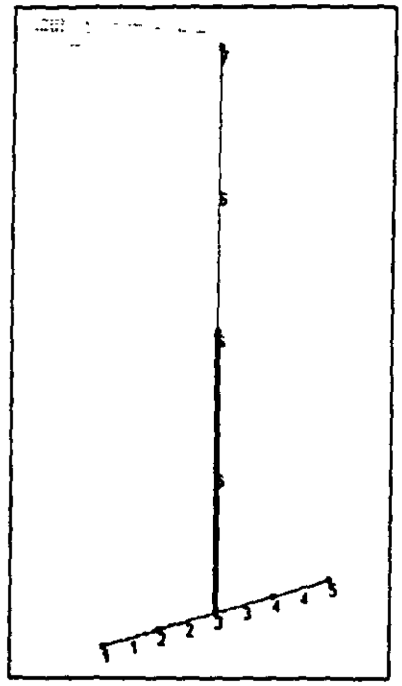


Figure 3.23

3. Let us once again save the model by going to the *File* | *Save* menu option.

Unlimited Undo and Redo and Deleting

You can Undo/Redo any action by simply clicking "Ctrl + Z" to undo or "Ctrl + Y" to redo. Each action item that can be undone or redone is listed in the main toolbar.

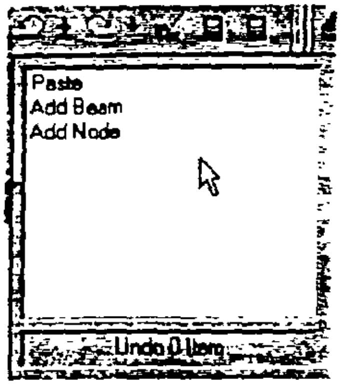


Figure 3.24

You can also delete an entity by either selecting it with the proper selection cursor or by highlighting the row that corresponds to the entity (on a data grid table) or hitting the "Delete" key on your keyboard.

6. Creating Members 7 To 12 and Nodes 8 To 14

Topics: Mirror, Dimensioning

1. Looking at Figure A, it is apparent that the model is symmetric about a vertical line passing through the points (14, 0, 0). Utilizing the *Mirror* facility available under the *Geometry* menu, we can mirror the first 6 members to create members 7 to 12.

   Select the 6 existing members by rubber-banding a window around them using the mouse. (Please refer to the 'Task Reference' section at the end of this chapter to learn more about selecting members.) Then, go to *Geometry* *Mirror* menu option as shown below.

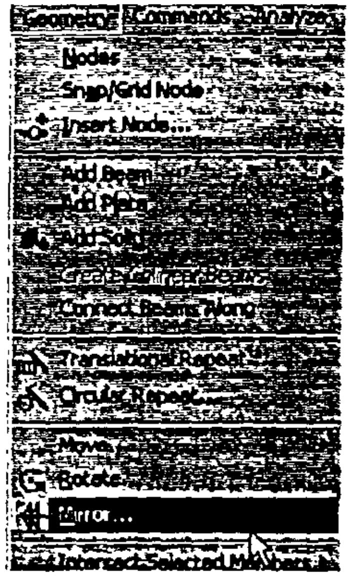


Figure 3.25

2. In the *Mirror* dialog box that comes up, specify the *Mirror Plane* as *Y-Z*, the *Distance to Origin* as *14ft* and the *Generate Mode* as *Copy*. Then, click on the *OK* button.

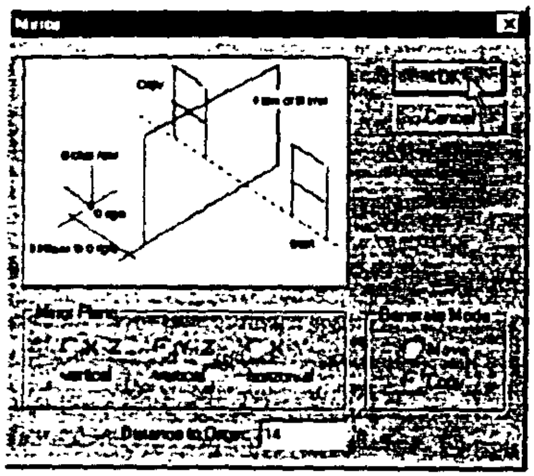


Figure 3.26

The following figure shows the structure after the mirroring has been done.

Before | After

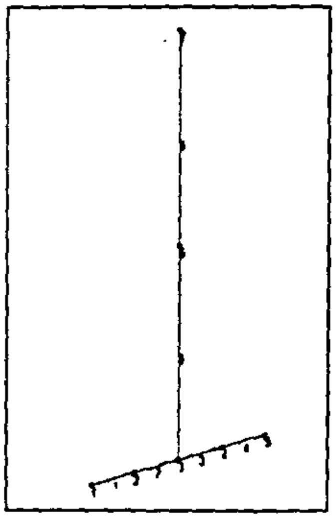


Figure 3.27

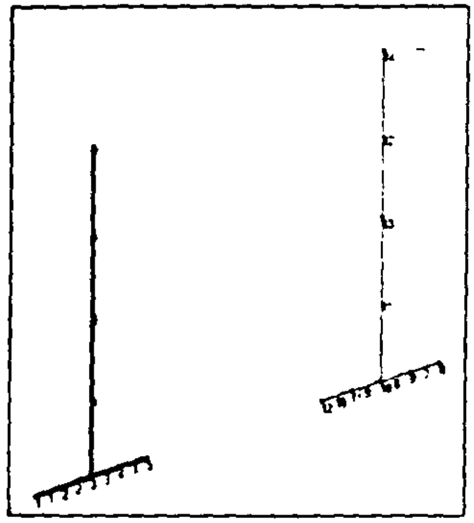


Figure 3.28

3. Let us save our structure again by using the *File : Save* menu option.

Verifying Distances

You can find out the distances between any two points by using the dimension icon from the toolbars. Simply click on the starting point and then the ending point and a dimension line will be drawn between the two points. The Dimension icon will label all or some of the member lengths in the model. You can also double-click on a member to find information about its geometry as well.

7. Creating Members 13 and 14

Topics: Adding Beams in 3D, Viewing, Rotating

1. To complete the structure, we need to add beams between joints (6 and 13) and (7 and 14). To do that, either select the *Add Beams* icon  from the left side of the screen or, select the *Add Beam* option under the *Geometry* menu from the top menu bar.

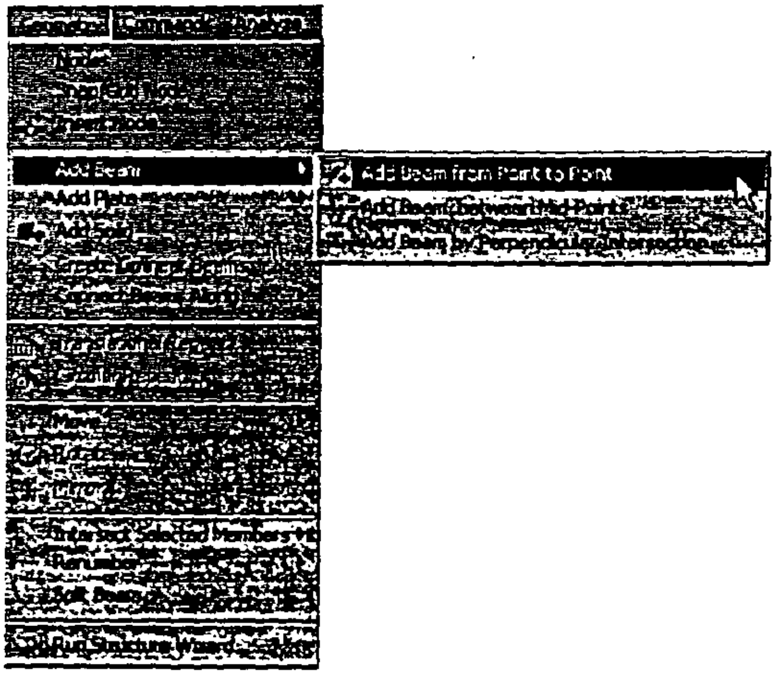


Figure 3.29

Notice that as we select the *Add Beam* option, the cursor changes as shown below.

2. Click on the two nodes (6 and 13) in succession and notice that a new beam (no. 13) has been created. Once you click on node 6, a line will follow your cursor until you click on node 13. Repeat this procedure by clicking between nodes 7 and 14 to create member 14. Once these are created, switch off the *Add Beam* option by clicking on the *Add Beams* icon once again.

The structure will now look as shown below. (Please refer to the 'Task Reference' section at the end of this chapter to learn more about viewing the structure from different angles.)

Isometric View

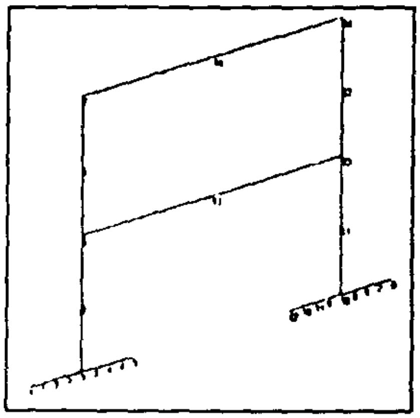


Figure 3.30

Side View from +X

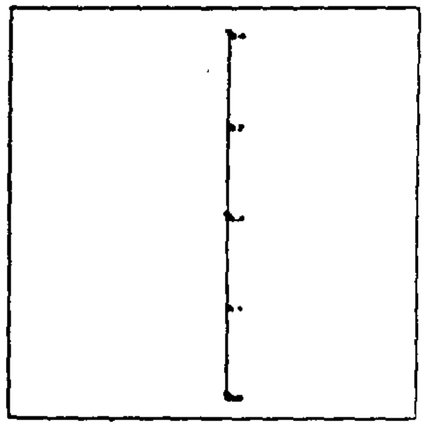


Figure 3.31

3 Select beams 1-4 and 7-10 by rotating the structure to its side position (Figure 3.31) You can use the icon in the toolbar to accomplish this. Use the rubber band window to select the bottom beams by just creating a box around the bottom node (Figure 3.32).

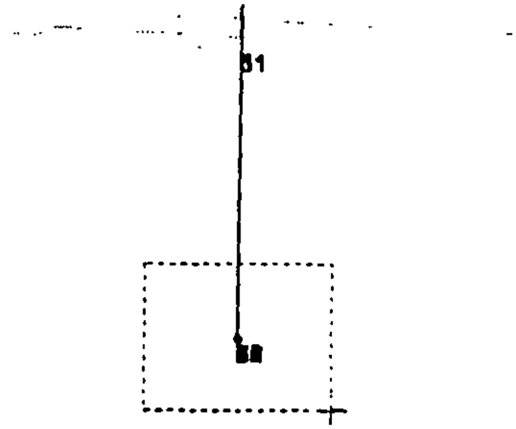


Figure 3.32

4. Go the isometric view  to verify that all the bottom beams have been selected.

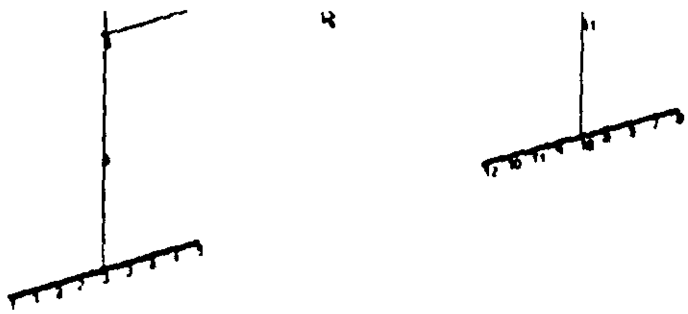


Figure 3.33

5. Right-click your mouse button and select the New View option to create a window with just those beams in it.

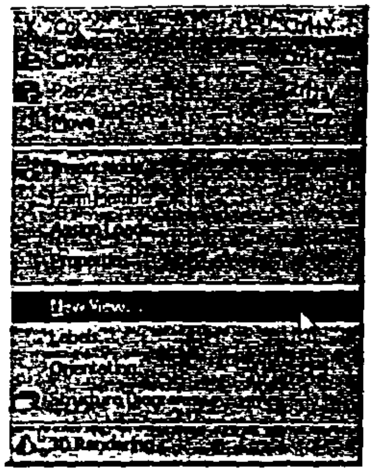


Figure 3.33

6. From the dialog box that pops up, select "Create a new window for the view" to make a new window separate from the main view but visible at the same time.

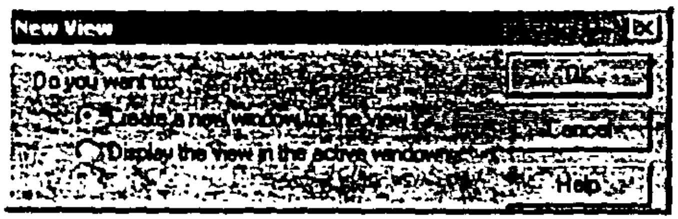


Figure 3.34

7. Go to View | View Management | Save View and supply the name of the view as "Footings"

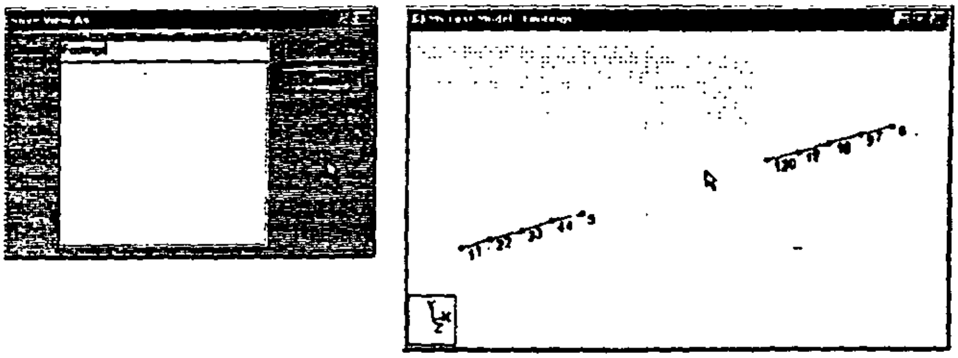


Figure 3.35

8. This view has the same functionality as the original main view. You can turn on the Snap/Node grid, add beams, turn on labels, etc. It allows you the flexibility to work on a smaller part of the model.
9. Close the new view (by clicking on the button on the view).
10. Let us save our model again by going to the *File* | *Save* menu option.

The reason for creating a new view for this example is so that we can look at the post-processing results for just the nodes/members in this view. It is a great way to isolate a part of the structure and see how it behaves.

Looking at the model in different views

The easiest way to rotate your model is to use the Up/Down and Left/Right arrow keys on your keyboard. This will spin the structure around very quickly.

8. Assigning Member Properties and Material Constants

Topics: Properties, Materials, 3D Rendering, Assigning Props/Materials

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

```
1 4 7 10 PRIS YD 1.0 ZD 8.0
2 3 8 9 PRIS YD 2.0 ZD 8.0
5 6 11 12 TABLE ST W10X33
13 14 TA ST W12X26

CONSTANTS
E 29000. MEMB 5 6 11 TO 14
E 3150. MEMB 1 TO 4  7 TO 10
DEN 0.283E-3 MEMB 5 6 11 TO 14
DEN 8.68E-5 MEMB 1 TO 4  7 TO 10
POISSON 0.3 MEMB 5 6 11 TO 14
POISSON 0.17 MEMB 1 TO 4  7 TO 10
```

These are the textual commands that get saved in your STAAD input file. We will create these commands graphically. In the case of constants, the values listed above for E, Poisson, Density, etc. also happen to be the default values built into the program for steel and concrete. Section 5.6.2 of the STAAD Technical Reference Manual reinforces this fact. The advantage of this is that we can use the property dialog boxes for assigning properties as well as constants simultaneously, instead of assigning them in separate operations. **Once you learn how to use the Assign dialog box, assigning loads, member specifications, supports, etc. are done in a similar manner.**

2. To do this, select *General* | *Property* Page from the page control buttons on the left side of the screen.

Figure 3.36

Alternatively, you can click on the icon from the toolbar to open the same page. The shortcut icons allow you to open up multiple assign boxes (properties, supports, loads, etc.) at the same time. The page control action (Figure 3.36) changes the layout of your interface by opening the accompanying spreadsheets (data grids) for that page and automatically turns on labels associated with that page.

3. The rectangular cross sections, characterized by the YD and ZD options, for members 1 to 4 and 7 to 10 are created from the dialog boxes available under the *Define* button in the *Properties* dialog box as shown below. This is where you would go when you need to create non-standard shapes (rectangles, circles, trapezoids, general, etc.)

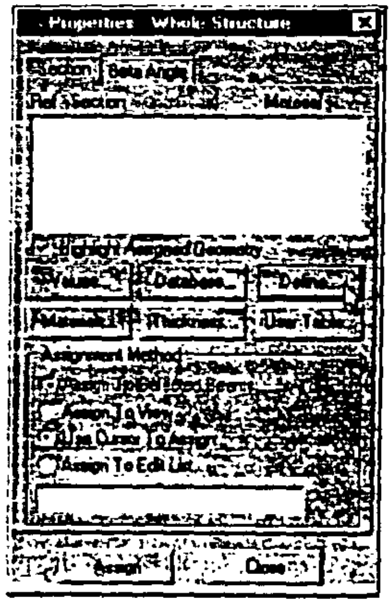


Figure 3. 37

4. In the *Prismatic Property* dialog box that comes up, select the *Rectangle* tab. Notice that the field called *Material* is presently on the <u>**checked**</u> mode. If we keep it that way, the material properties of concrete (E, Poisson, Density, Alpha, etc.) will be assigned along with the cross-section name. The material property values assigned will be the program defaults. Since we wish to go with the defaults for concrete, we will leave that box as it is, namely, <u>**checked**</u> Enter *1.0* for *YD* and *8.0* for *ZD*. Finally, click on the *Add* button as shown below

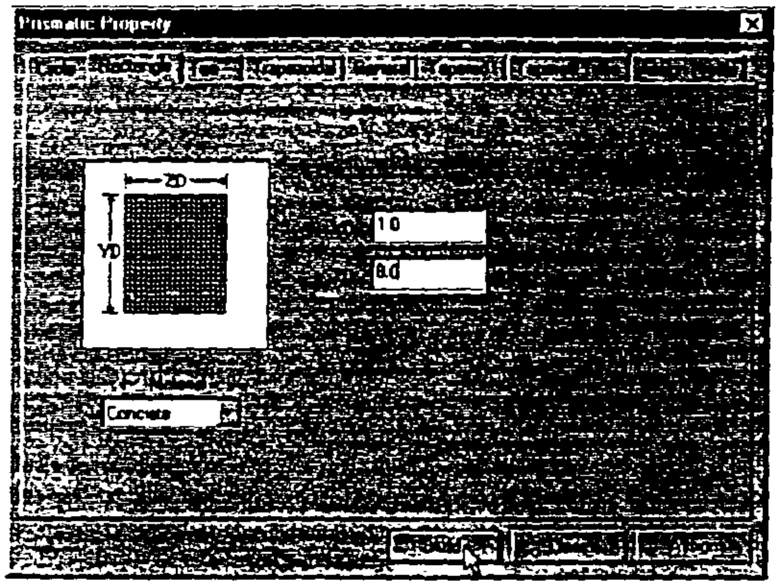


Figure 3.38

5. To create the rectangular section that will eventually be assigned to members 2, 3, 8 and 9, repeat the Step 4 and enter *2.0* for *YD* and *8.0* for *ZD*. Close the dialog box by clicking on the "Close" button.

Before you can actually assign properties to members, you must create them first. This is what steps 4 and 5 are accomplishing.

Inputting values in any unit system

A quick way to add in values in any unit system in almost any dialog box is to hit the "F2" button on your keyboard when your mouse is positioned in the edit box (i.e. YD or ZD boxes for Figure 3.38).

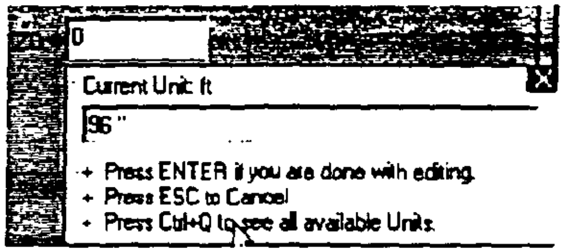


Figure 3.39

You can even type in fractions (5 ¼") and use feet and inch symbols. Press "Ctrl + Q" to see a list of unit options.

6. The next property type we wish to create is the W shape from the AISC table. This is available under the *Database* button in the *Properties* dialog box as shown below.

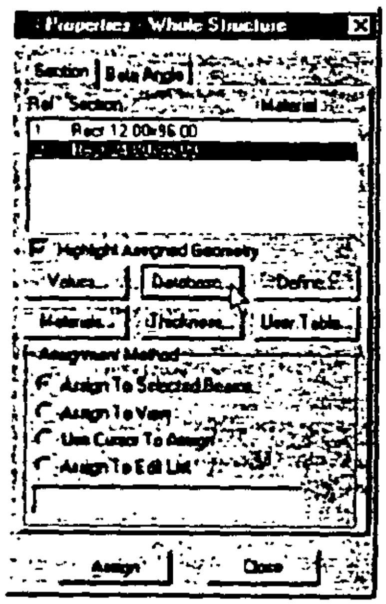


Figure 3.40

7. 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*.

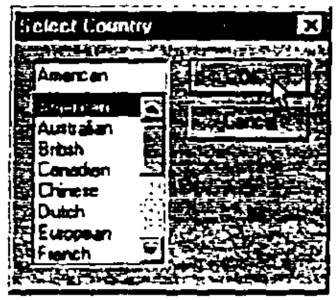


Figure 3.41

8. In the *American Steel Table* dialog box, select the *W Shape* tab. Once again, notice that the *Material* box is checked. Let us keep it that way because it will enable us to subsequently assign the material constants E, Density, Poisson, etc. along with the cross-section since we want to assign the default values as explained in step 4.

   Choose *W10X33* as the beam size, *ST* as the section type and click on the *Add* button as shown in the figure below. Detailed explanation of the terms such as ST, T, CM, TC, BC, etc. is available in Section 5 of the STAAD Technical Reference Manual.

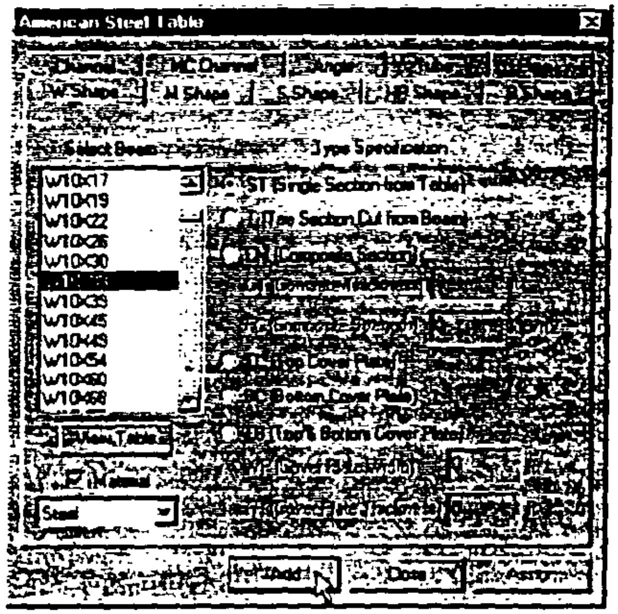


Figure 3.42

9. Repeat this procedure and select *W12X26* as the beam size. This is the section we intend to assign to members 13 and 14. Click on the "Close" button to close the property selection dialog box.

At this point, the *Properties* dialog box will look as shown below:

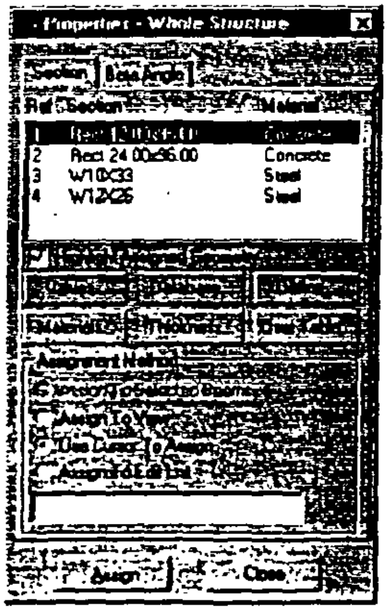


Figure 3.43

10. The next step is to associate or apply these created properties to the appropriate members. To do that, using the mouse, select the first property type in the *Properties* dialog box.

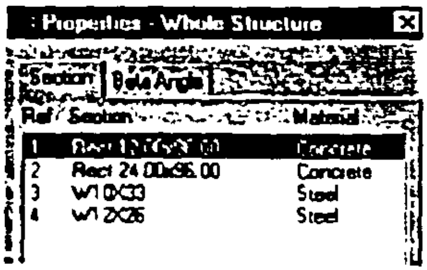


Figure 3.44

11. Open the view we created previously containing the bottom footing members by going to *View* | *Open View* and select the view called "Footings". Make sure the option "Create a new window for the view" is selected. Click on "OK" to open this view.

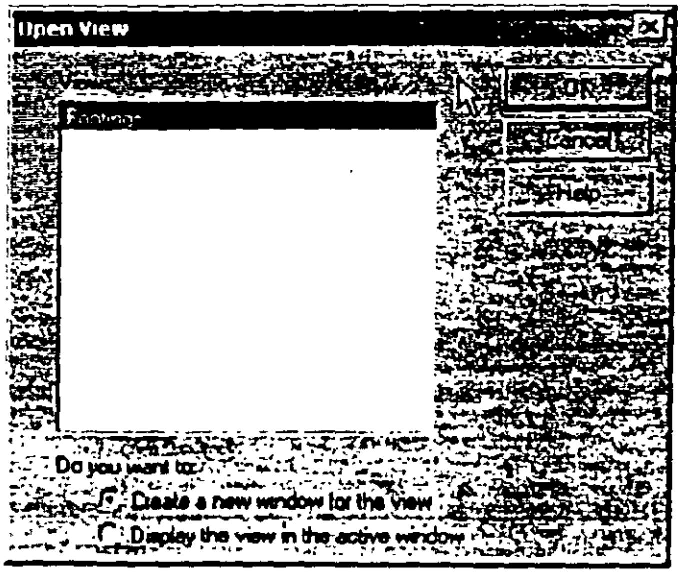


Figure 3.45

Then, select members 1, 4, 7 and 10 using either the *Beams Cursor* or, by going to *Select* | *By List* | *Beams* menu option and specifying the beam numbers in the *Enter List* box. (Please refer to the 'Task Reference' section at the end of this chapter to learn more about selecting members or Section 5 of this tutorial.)

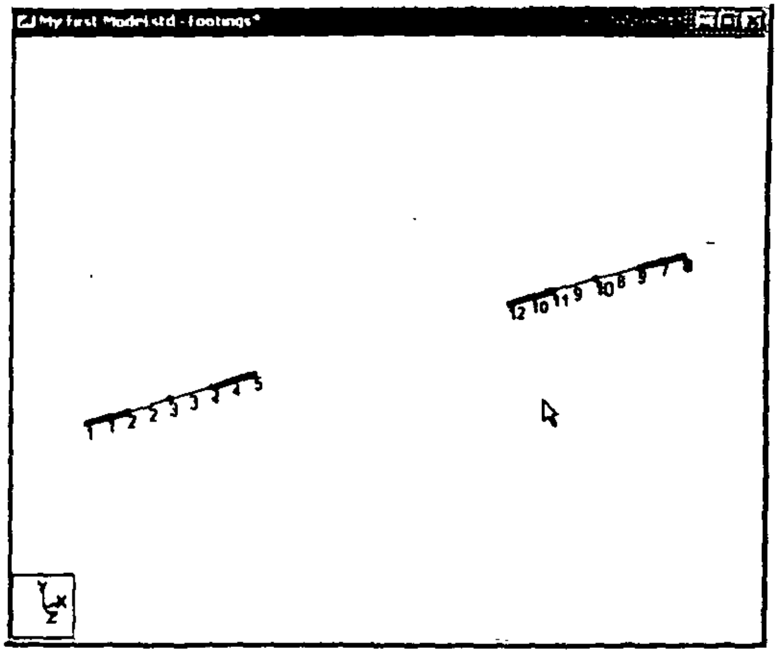


Figure 3.46

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

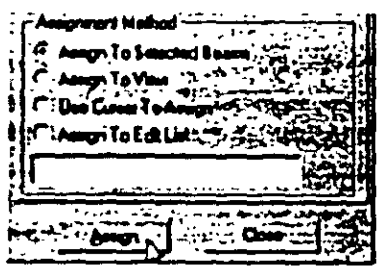


Figure 3.47

After the property has been assigned, the model will look as shown below.

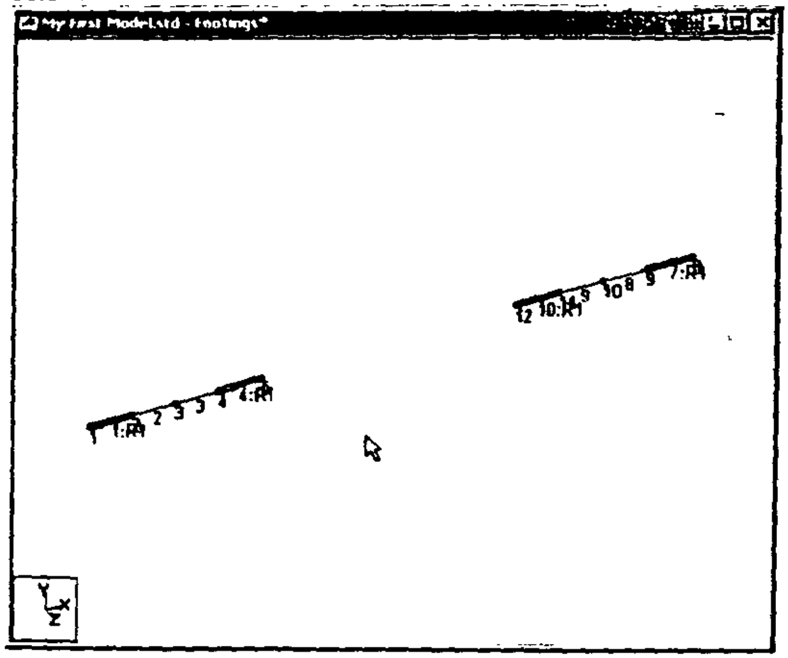


Figure 3.48

Click anywhere in the drawing area to un-highlight the members. You can display the property names on the section itself by right-clicking your mouse button, choose Labels and click on the Sections option under Properties. Please note that turning on labels on one view does not automatically turn them on for other views. You must handle each view separately.

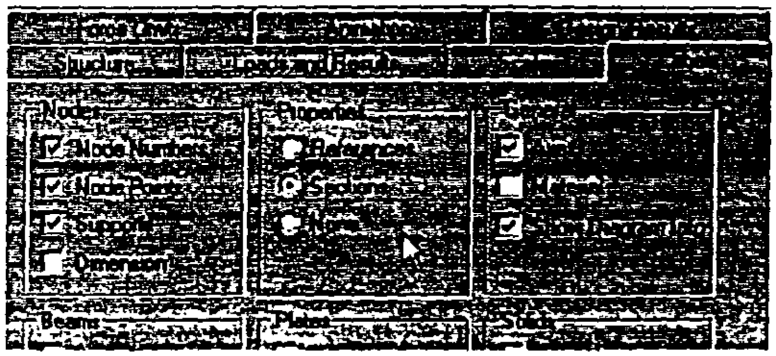


Figure 3.49

12. To demonstrate another way of assigning properties, choose "Use Cursor to Assign" from the Assign properties dialog box. Select the property "Rect 24.00 x 96.00" and click on the "Assign" button.

    Your cursor will change to look like this: . Click on members 2,3,8 and 9 (in the view labeled "Footings") to assign the properties. Assign the remaining properties to the members as listed below. You can use the method listed either Step 11 or Step 12.

Property Name	Members To Be Assigned
ST W10X33	5, 6, 11, 12
ST W12X26	13, 14

Click anywhere in the drawing area to un-highlight the members.

After all the properties have been assigned, the model will look as shown below.

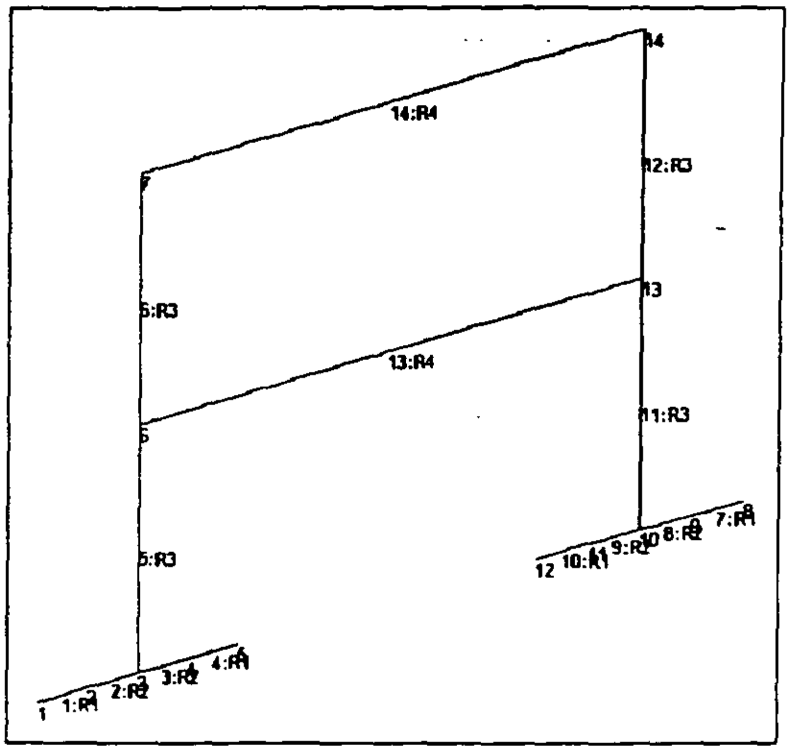


Figure 3.50

Let us close the *Properties* dialog box as shown below.

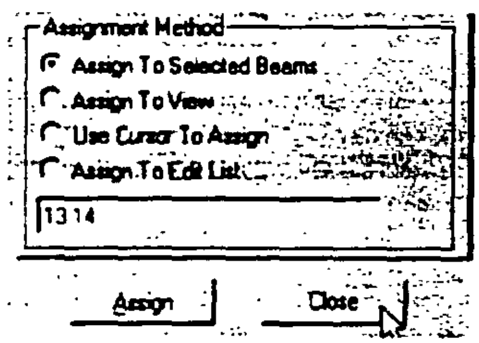


Figure 3. 4

13. Let us again save the work we have done so far. From the *File* menu, select *Save*, to save the file. You can close the "Footings" view as well.
14. Turn on the 3D section view of the model by right clicking your mouse button and selecting "Structure Diagrams"

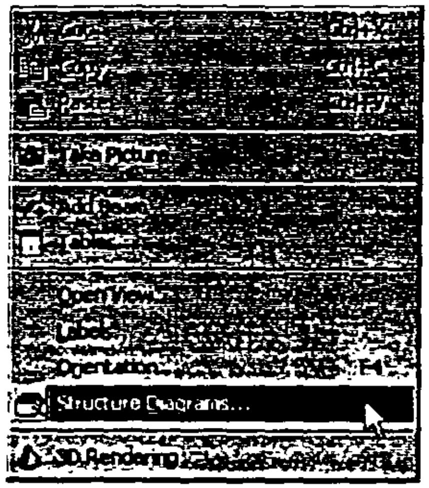


Figure 3.51

Click on the option "Full Sections" under 3D sections to draw the 3D sections. You can also change the color of the sections by clicking on the "Section Outline" color button under the Colors section. Click on "OK".

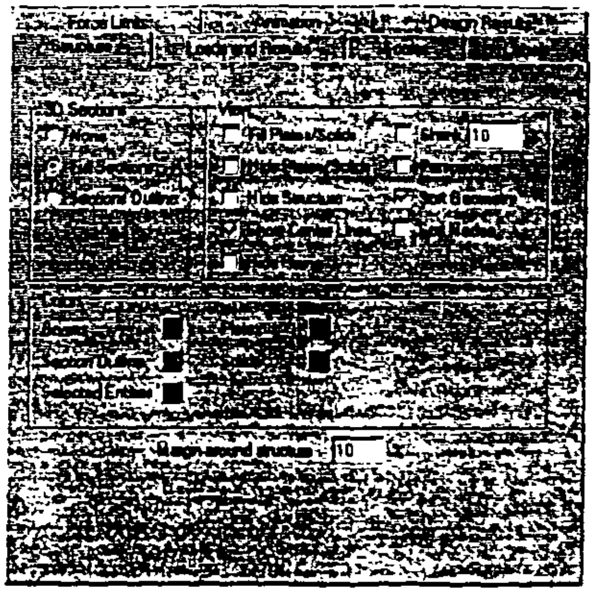


Figure 3.52

The resulting diagram is shown in Figure 3.53 below.

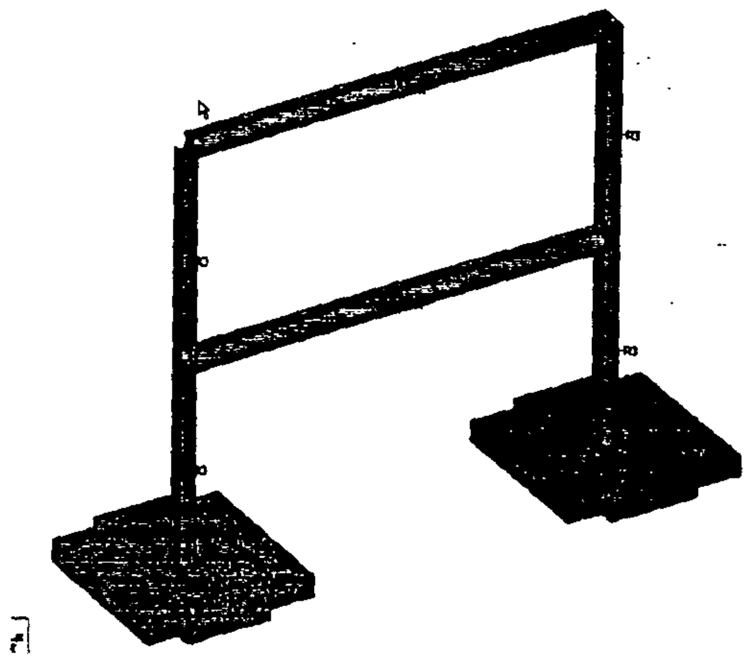


Figure 3.53

Some notes on properties

1. You can change a property in the assign dialog box by simply double-clicking on it and updating the values. You can also delete it by clicking on the "Delete" button on your keyboard.
2. You can see what properties have been assigned to what members by checking the "Highlight Assigned Geometry" box in the assign dialog.
3. A true 3D rendering with lighting and shading can be viewed by clicking on the icon from the toolbar or right-clicking your mouse button.

9. Assigning Supports

Topics: Creating and assigning spring supports

1. The commands we wish to generate are:

   2 TO 4 9 TO 11 FIXED BUT MZ KFY 4000.
   1 5 8 12 FIXED BUT MZ KFY 2000.

   These are the textual commands that get saved in your STAAD input file. We will create these commands graphically. To access the dialog boxes for choosing the appropriate type of support, we select the *General | Support* Page from the left side of the screen or by clicking on the icon from the toolbars. In the *Supports* dialog box that subsequently pops up, click on the *Add* button to create a support type like we created a property in Section 8.

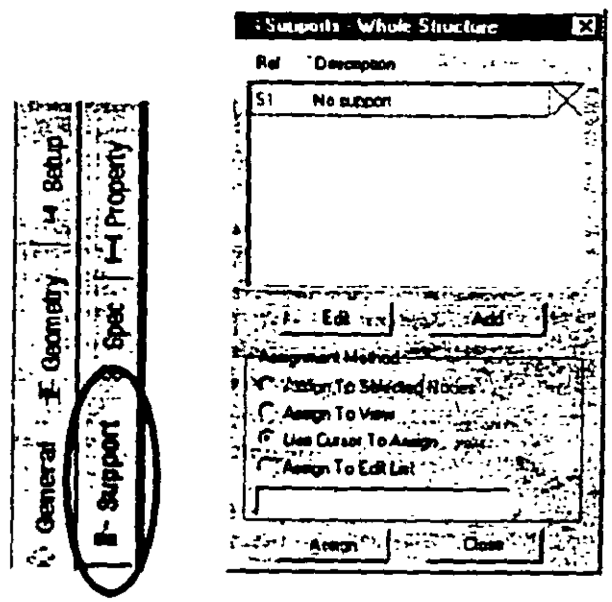


Figure 3.54

2. In the *Create Support* dialog box that opens, select the *Fixed But* tab. Select *MZ* under *Release* and enter *4000 kip/ft* as the spring constant for *KFY* under *Define Spring*. (This creates a FIXED type of support for all degrees of freedom except a) MZ and b) the translational degree of freedom FY is not fully restrained but represented by a spring having a spring constant of 4000 kip/ft.) Click on the *Create* button.

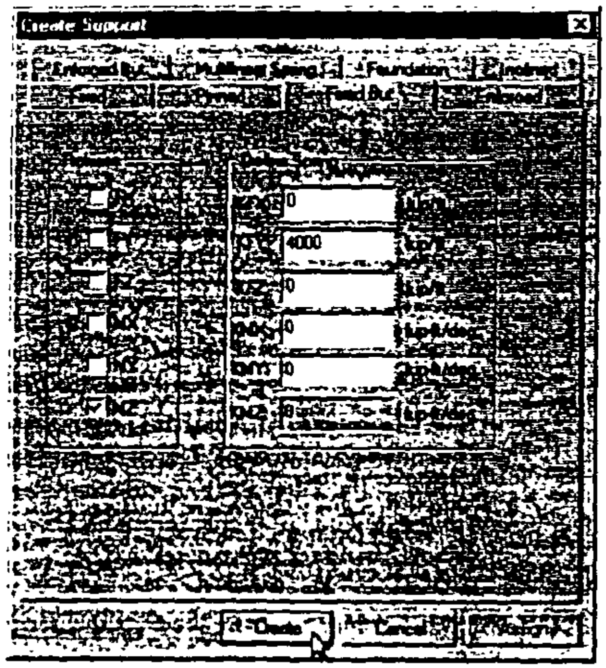


Figure 3.55

3 To create the second spring that will eventually be assigned to nodes 1, 5, 8 and 12, repeat the above procedure but specify *2000 kip/ft* for *KFY* instead of 4000.

4. The next step is to associate these created supports with specific joints. Click on the first support specification (Support 2) in the *Supports* dialog box.

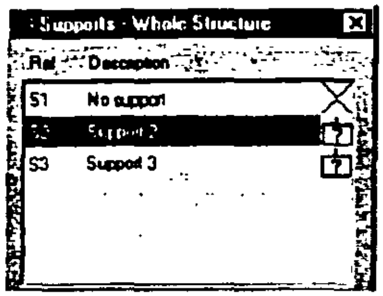


Figure 3.56

5. Then, select the following nodes:

   2, 3, 4, 9, 10, 11

   To select these nodes, click on these nodes (holding the "Ctrl" key down) on the drawing or use the rubber band window using the *Nodes Cursor*. Alternatively, go to *Select | By List | Nodes* menu option, and in the *Select Nodes* dialog box, type the node numbers in the *Enter list* box. (Please refer to the 'Task Reference' section at the end of this chapter to learn more about selecting nodes or Section 5 of this tutorial.)

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

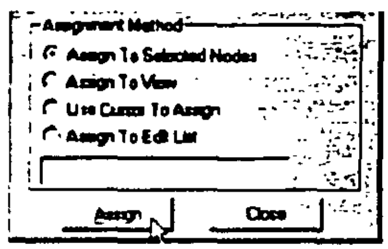


Figure 3.57

The assignment box works and behaves the same way as the Property Assignment box in Section 8.

6. In a similar fashion, assign the second support (Support 3) to nodes 1, 5, 8 and 12.

   After both the supports have been assigned, the structure will look as shown below:

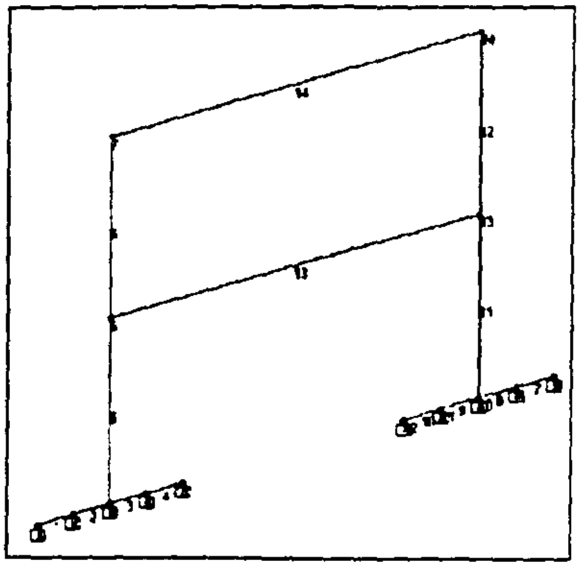


Figure 3.58

7. Let us save our model again by going to the *File | Save* menu option.

Quick methods to graphically change supports

1. You can select the "Support Edit Cursor" from the toolbar or from *Select | Support Edit Cursor* from the main menu and double-click on the support directly to modify it.

2. You can change a support in the assign dialog box by simply double-clicking on it. This will change all nodes that have been assigned this support.
3. Using the Supported Nodes spreadsheet, you can change the support type for a node by changing the support reference in the "Support" column.

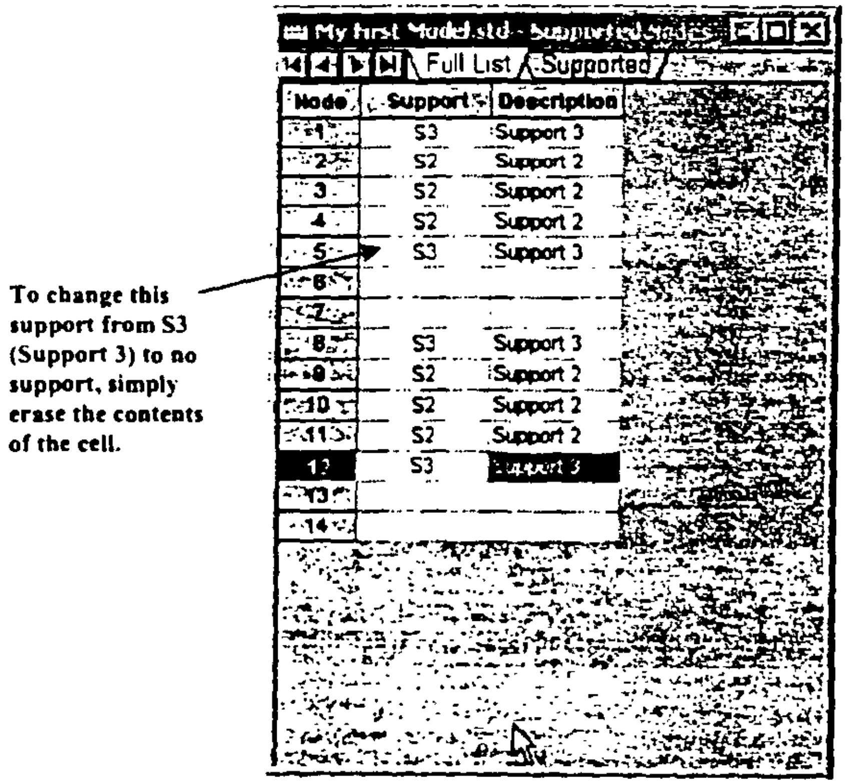


Figure 3.59

10. Specifying Loads

Topics: Creating and assigning selfweight loads, nodal loads, member loads; viewing load values; creating load combinations

1. Load assignments are done from the *General | Load* Page as shown below. You can also access this page by clicking on the  icon from the toolbars.

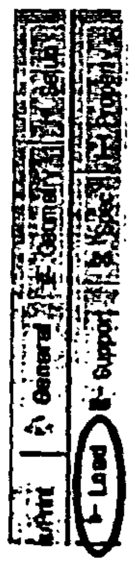


Figure 3.60

For this example, we wish to generate the following load data using the graphical environment.

```
LOADING 1 Self Weight
SELF Y -1.0
LOADING 2 Dead Load
MEMBER LOAD
13 14 UNI GY -3.0
```

```
LOADING 3 Wind Load
JOINT LOAD
6 7 FX 5.0
LOAD COMB 4 Combination of 1+2+3
1 1.0 2 1.0 3 1.0
```

In the *Set Active Primary Load Case* dialog box that comes up, enter Self Weight as the title for *Load Case 1* and click on *OK*.

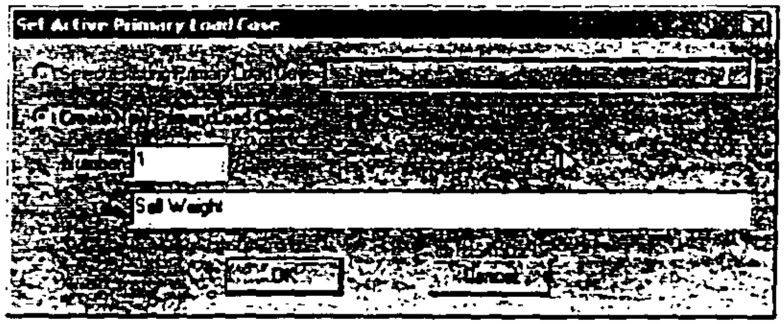


Figure 3.61

2. The *Loads* dialog box will now appear in the data area on the right hand side of the screen. To generate and assign the selfweight load type, click on the "*Selfweight*" button.

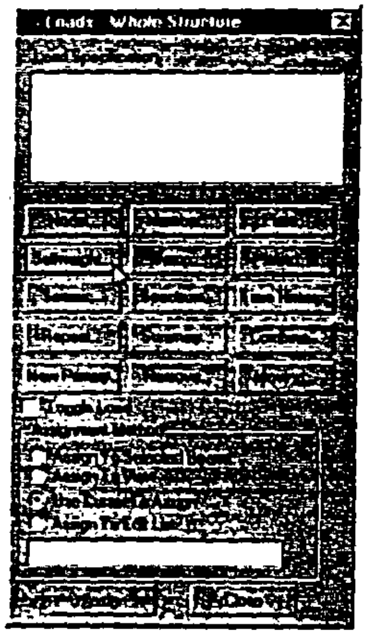


Figure 3.62

3. In the *Selfweight Load* dialog box, specify the *Direction* as *Y*, and enter the *Factor* as *-1.0*. Since the selfweight of the structure is to be applied on the entire model, click on the *Assign* button. If you don't want to specify a selfweight for certain members, you can assign a density of zero to those members through *Commands* | *Material Constants* | *Density*.

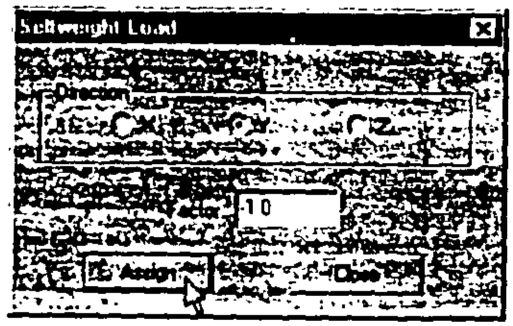


Figure 3.63

4. Create another Primary Load Case by clicking on the "New Primary.." button in the assign loads dialog box. Provide the title of this load case (Load Case 2) as "Dead Load" as shown in Figure 3.64 below.

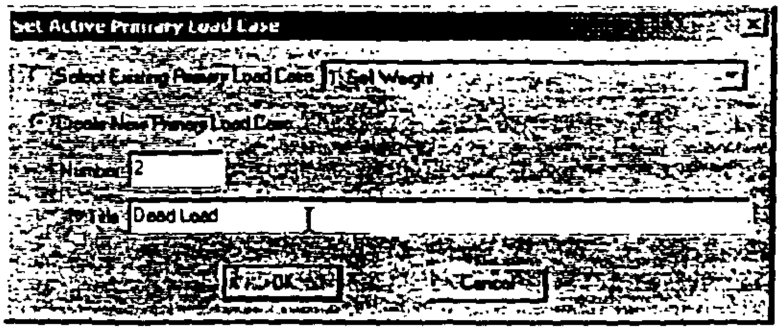


Figure 3.64

5. To create the member load, click on the *Member* button in the *Loads* dialog box.

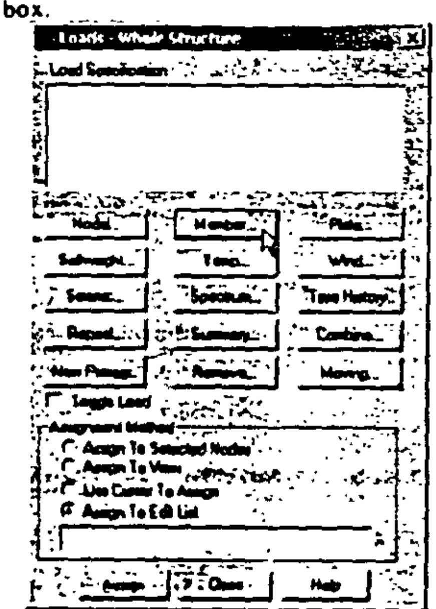


Figure 3.65

We could either adopt the method of creating the member load first and then applying it to the members using the "Assign Cursor" or selecting the beams first and then using the "Assign to selected beams" option. We will choose the former.

6. In the *Beam Loads* dialog box that comes up, select the *Uniform Force* tab. Specify *GY* as the *Direction* and enter *-3.0* as the *Force*. Then, click on the *Add* button.

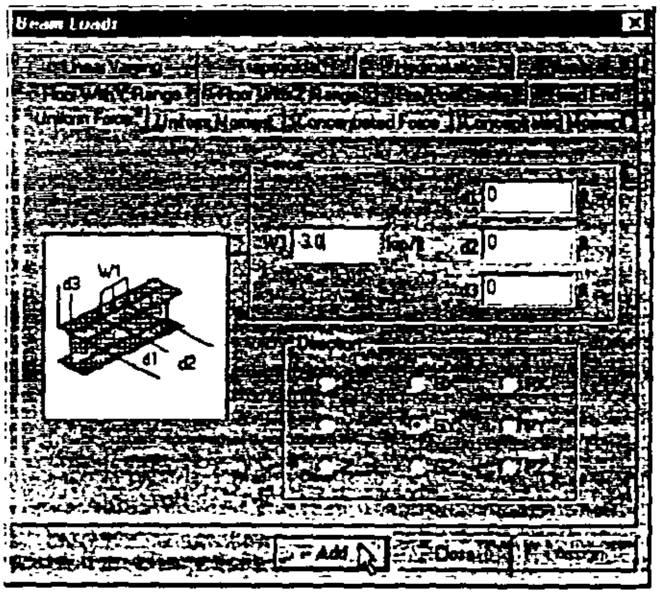


Figure 3.66

At this point, the *Loads* dialog box will look as shown below.

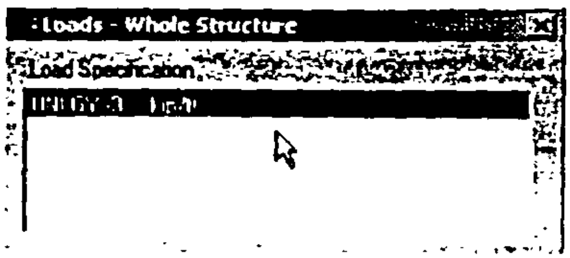


Figure 3.67

7. The next step is to associate the member load with members 13 and 14. We shall use a method of assignment called *Use Cursor to Assign*.

   Notice from the previous figure that the member load (UNI GY -3) in the *Loads* selection box is already highlighted. If it is not highlighted, click on it to highlight it.

   Click on the "Use Cursor To Assign" option and click on the "Assign" button. Your cursor should turn into this: 

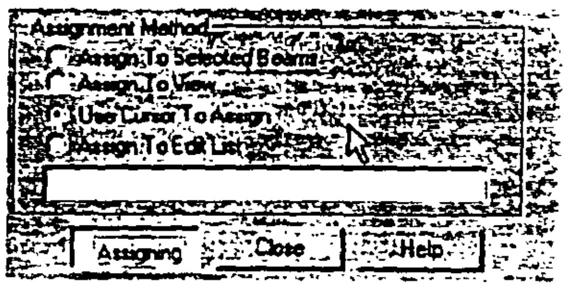


Figure 3.68

8. Click on members 13 and 14 to assign the member loads to those members. Turn on the load values on the main view by right-clicking your mouse button and choosing "Labels". Check the "Load Values" option under the Loading Display Options.

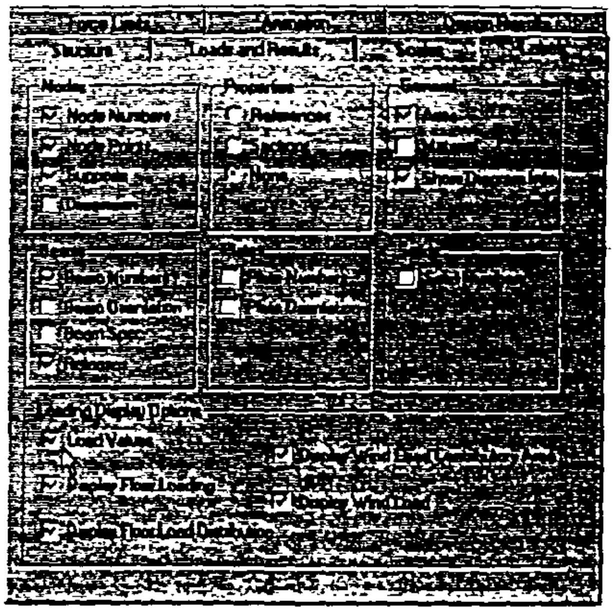


Figure 3.69

Your structure (for Load Case 2) will look like Figure 3.70.

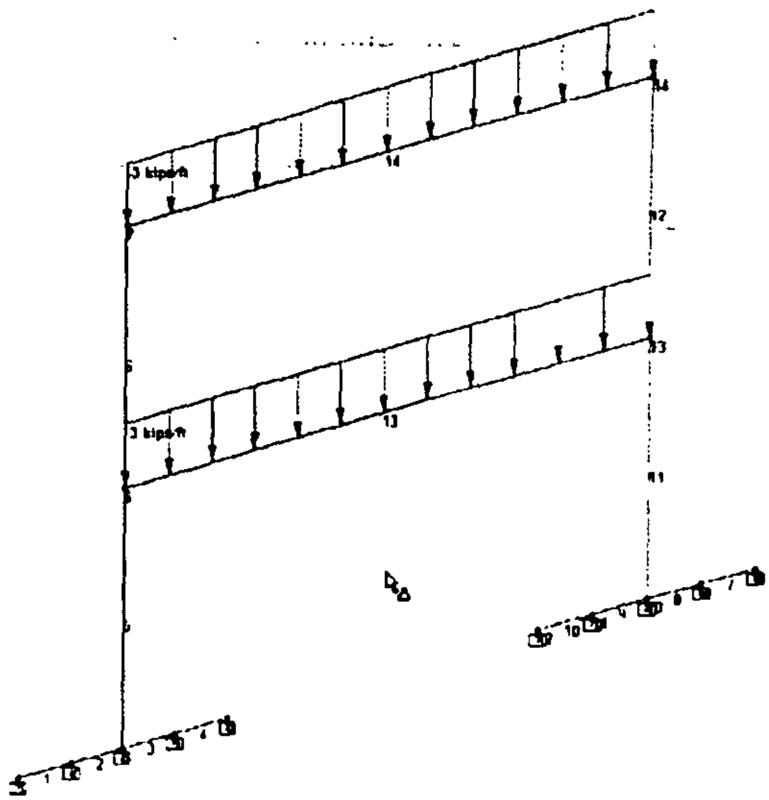


Figure 3.70

9. Create another Primary Load Case by clicking on the "New Primary. " button in the assign loads dialog box. Provide the title of this load case (Load Case 3) as "Wind Load" as shown in Figure 3.71 below

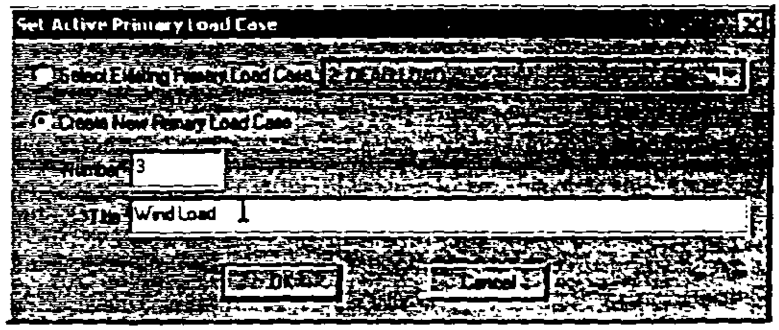


Figure 3.71

10. *Joint loads* are created through the dialog boxes available under the *Nodal* type of loads.

    Since we know that these loads are to be assigned to nodes 6 and 7, let us first select those nodes prior to creating the load itself. To select these nodes, click on the nodes on the drawing using the *Nodes Cursor*. Alternatively, go to *Select | By List | Nodes* menu option, and in the *Select Nodes* dialog box, type the node numbers 6 and 7 in the *Enter list* box. (Please refer to the 'Task Reference' section at the end of this chapter to learn more about selecting nodes or Section 5 of this tutorial.)

11. After selecting the nodes, click on the "*Nodal*" button.

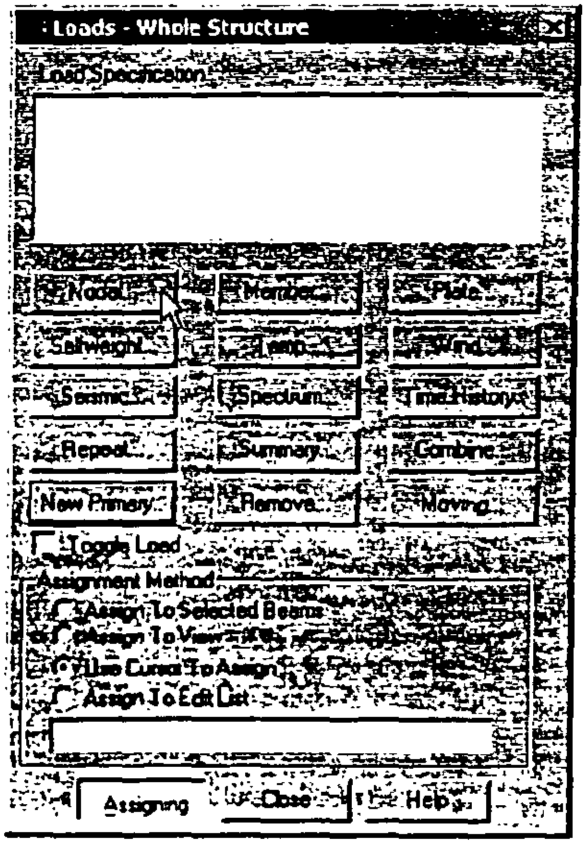


Figure 3.72

12 In the *Node Loads* dialog box that comes up, enter *5.0* for *Fx*. We can straightaway click on the *Assign* button to apply these loads on the selected joints. Remember, you can also use the other assignment choices listed in Figure 3.72 depending on the situation.

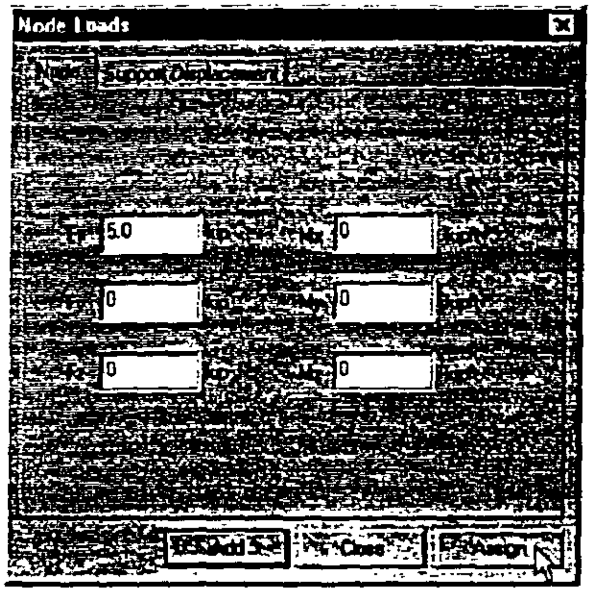


Figure 3.73

The dialog box will be automatically closed and the structure for Load Case 3 will now look as shown below.

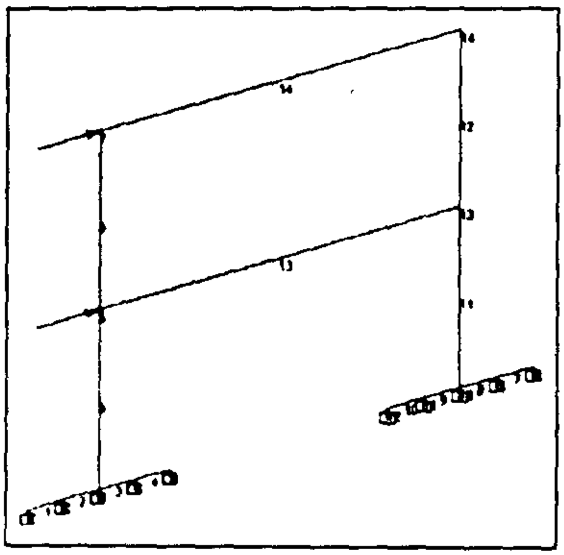


Figure 3.74

To view the loads for different load cases, simply choose the load case from the drop-down combo-box in the toolbars.

Figure 3.75

13. The final load case we need to create is a load combination. Click on the "Combine..." button from the load assign box.

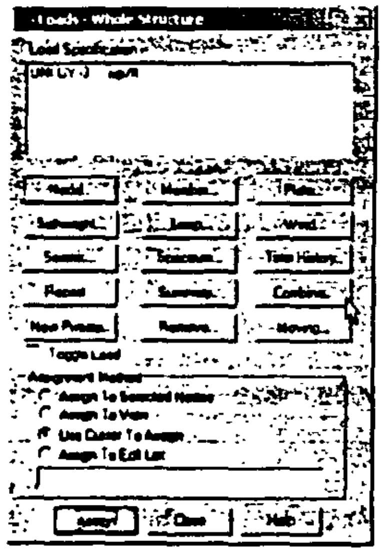


Figure 3.76

14. Once the Define Combinations dialog box pops up, click on the "New" button to create a new load combination.

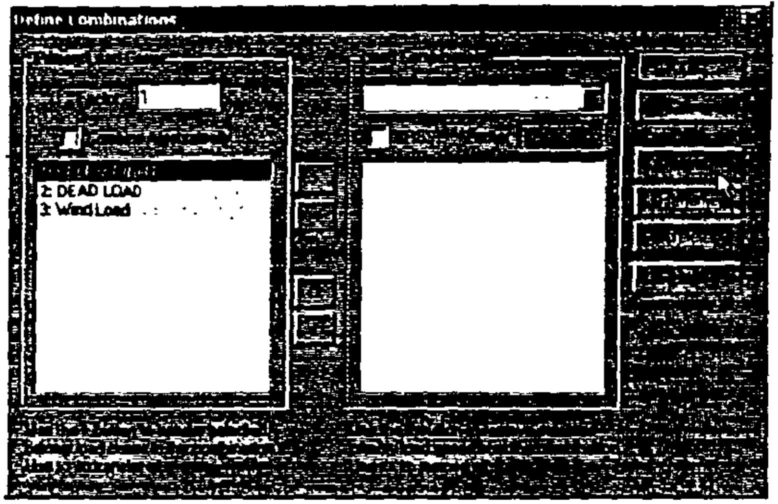


Figure 3.77

15. Provide the name "Combination of 1+2+3" and click on "OK".

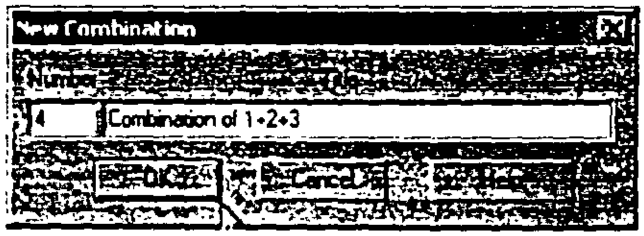


Figure 3.78

16. Select all the load cases from the left-hand side and click on the  button to create a combination which applies a factor of 1.0 to each case and adds them up. Click on "OK".

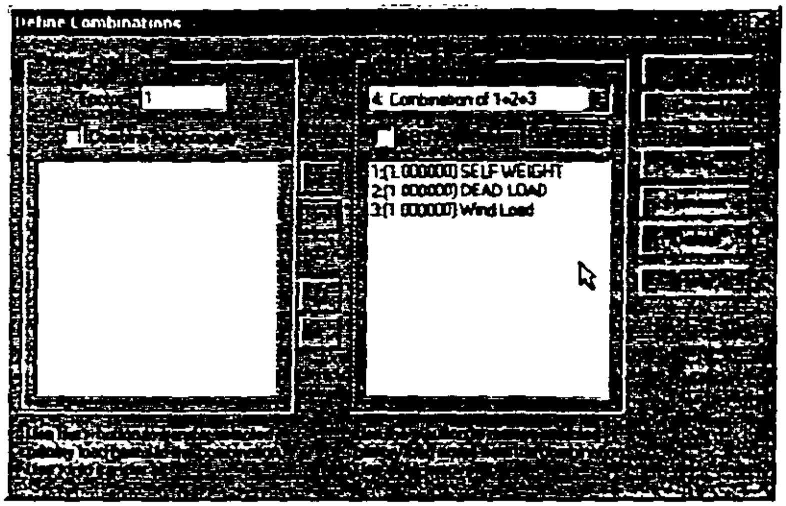


Figure 3.79

You can view the load values for the load combination on the structure from the drop-down combo-box in the toolbars.

17. Click anywhere on the screen to un-highlight the members. Let us also save our model again by going to the *File | Save* menu option.

Quick methods to graphically change loads

1. You can select the "Load Edit Cursor" from the toolbar or *Select | Load Edit Cursor* from the main menu and simply double-click on a load in the view window to change the values of the load
2. You can double-click on the load definition from the load assign dialog box
3. Check the "Toggle Load" box when assigning loads to assign and un-assign specific loads to nodes, members, plates or solids.
4. Use the spreadsheets'

11. Specifying the Analysis Command

Topics: Adding analysis commands; selecting results to be included in the STAAD output file

1. The next step is to assign the commands to perform the analysis and report the analysis results. We wish to generate the following commands in the graphical environment:

   PERFORM ANALYSIS PRINT STATICS CHECK
   PRINT ANALYSIS RESULTS

   The "PERFORM ANALYSIS" command tells STAAD.Pro to do a normal linear-static analysis. The "PRINT STATICS CHECK" performs and prints out an equilibrium check. The "PRINT ANALYSIS RESULTS" prints displacements, reactions and member forces in the output file. 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.

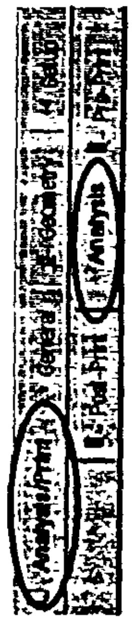


Figure 3.80

2 Click on the *Define Commands* button in the data area on the right hand side of the screen.

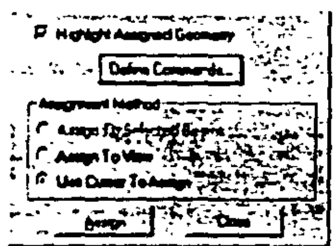


Figure 3.81

3 In the *Analysis/Print Commands* dialog box that appears, select the *Statics Check* print option. Then, click on the *Add* button followed by the *Close* button. This will add the command "PERFORM ANALYSIS PRINT STATICS CHECK" to the input file.

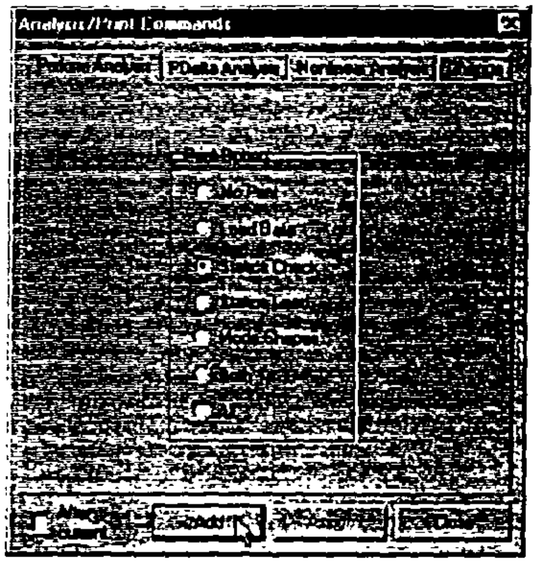


Figure 3.82

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

Figure 3.83

5. Click on the *Define Commands* button in the data area on the right hand side of the screen.

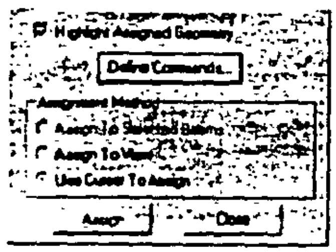


Figure 3.84

6. 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.

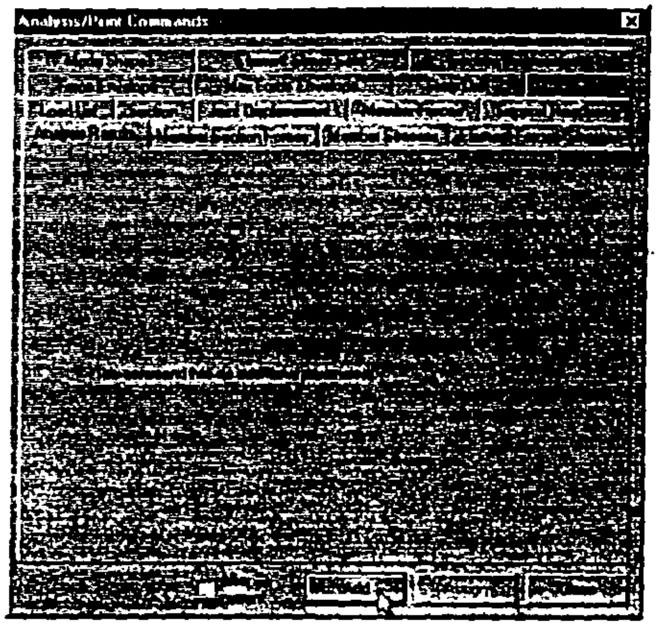


Figure 3.85

This concludes the task of assigning all the data to the structure. Let us *Save* the file one final time.

12. Viewing the Input Command File

Topics: STAAD input file (.STD file); STAAD Editor; manipulating the model using STAAD script commands

1. Let us now take a look at the data that has been written into the file that we just saved above. The contents of the file can be viewed either by clicking on the *STAAD Editor* icon or, by going to the *Edit* menu and choosing *Edit Input Command File* as shown below.

Figure 3.86

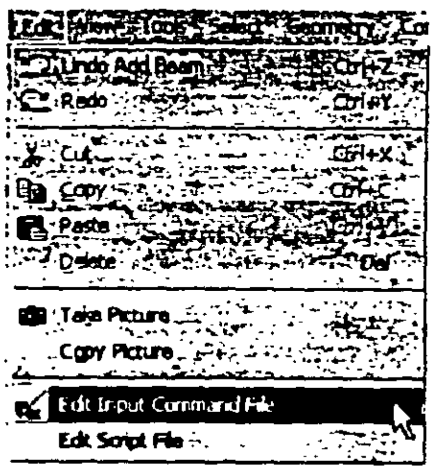


Figure 3.87

A new window will open up with the data listed as shown here:

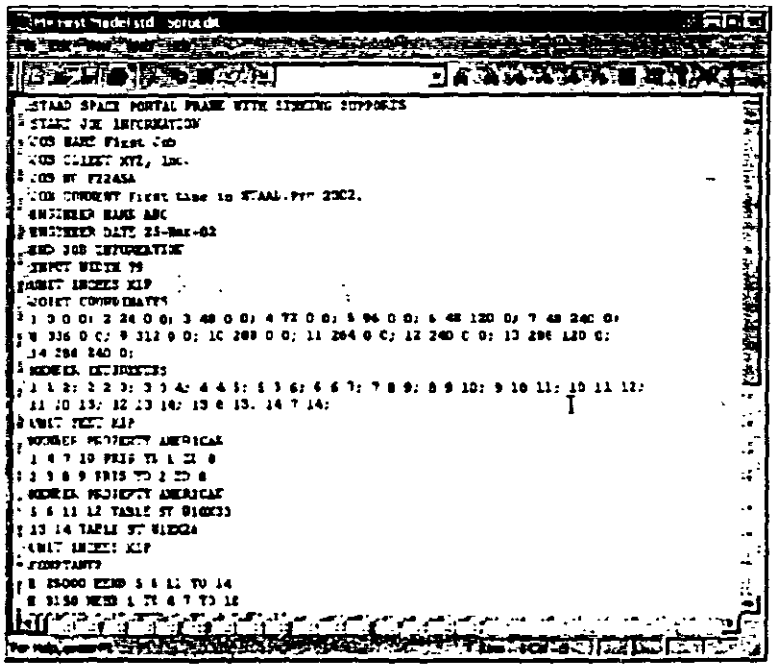


Figure 3.88

This window and the facilities it contains is known as the *STAAD Editor*.

We could make modifications to the data of our structure in this *Editor* if we wish to do so. Let us *Exit* the *Editor* without doing so by selecting the *File | Exit* menu option of the editor window (not the File | Exit menu of the main window behind the editor window).

If you understand the STAAD command language, you can create the part of or the entire model by typing the commands in the Editor (or any other Editor of your choice), instead of using the graphical method explained in the previous sections. The Editor provides a quick way of making changes to your model.

13. Performing the Analysis and Design

STAAD.Pro performs analysis and design simultaneously. In order to perform the analysis and design, select the *Run Analysis* option from the *Analyze* menu or press "Ctrl + F5" on your keyboard.

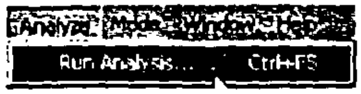


Figure 3.89

If the structure has not been saved after the last change was made, you should save the structure first by using the *Save* command from the *File* menu.

When you select the *Run Analysis* option from the *Analyze* menu, the following dialog box appears:

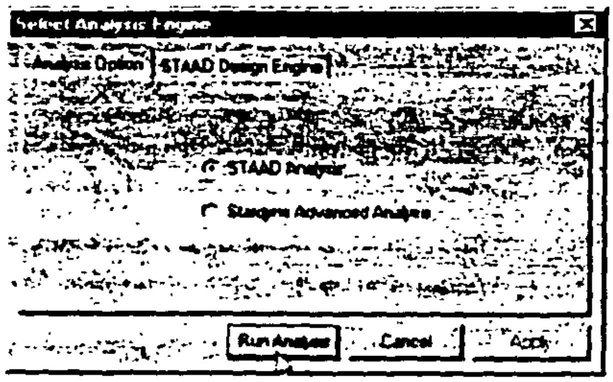


Figure 3.90

We are presented with the choice of 2 engines: the STAAD engine and the STARDYNE Advanced Analysis engine. The STARDYNE Analysis engine is suitable for advanced problems such as Buckling Analysis, Modal Extraction using various methods, fatigue analysis, etc. However, if the calculations call for steel or concrete design, UBC load generation, etc., we have to select the STAAD engine. Most of your analysis will probably be done through the STAAD engine.

Let us set the radio button on the STAAD engine. The *STAAD Design Code* tab in Figure 3.90 is used to select the design code to be used, if you have installed multiple design codes. Also, if you purchased the standard package, and no additional codes, you should select the design engine that was supplied with the standard package – such as, US for American buyers, British for British buyers, and so on.

Click on that tab and make sure the standard US concrete and steel codes are selected.

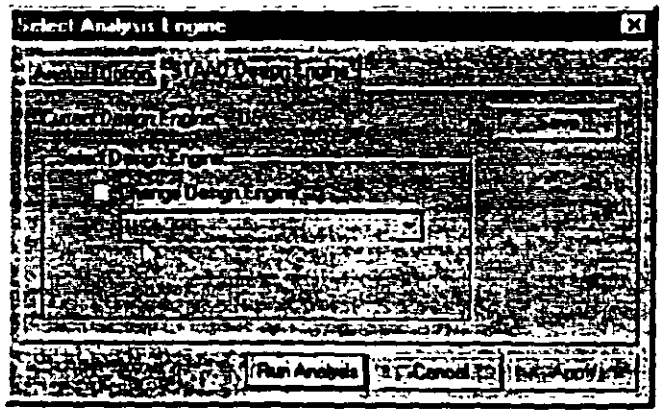


Figure 3.91

Click on the *Run Analysis* button.

As the Analysis progresses, several messages appear on the screen as shown in the figure below.

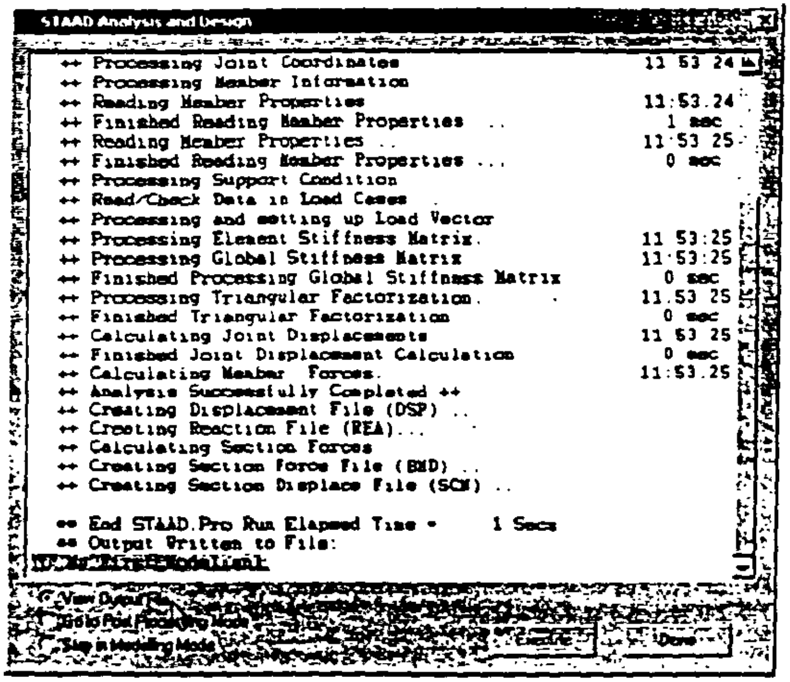


Figure 3.92

You are presented with three options. The "View Output File" option will invoke the STAAD Viewer with all of the analysis results presented in a textual format. The "Go to Post Processing Mode" option will take you to the STAAD.Pro Post-processor where you can graphically view your results. The "Stay in Modeling Mode" option will keep you in the Pre-processor or modeling environment. Select the first option ("View Output File") and Click on the "*Done*" button.

14. Viewing the Output File

During the analysis process, STAAD.Pro creates an Output file. This file provides important information on whether the analysis was performed properly. For example, if STAAD.Pro encounters an instability problem during the analysis process, it will be reported in the output file.

If you successfully followed Section 13, the output file should be displayed. We can also access the output file at any time by selecting *File | View | Output File | STAAD Output* option from the top menu or by clicking on the  icon. The STAAD.Pro output file for the problem we just ran is shown in the next few pages.

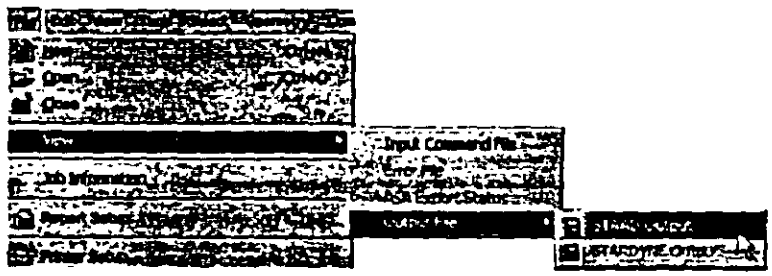


Figure 3.93

The STAAD.Pro output file is displayed through a file viewer called *SproView*. This viewer allows us to set the text font for the entire file and print the output file to a printer. Use the appropriate *File* menu option from the *SproView* menu bar.

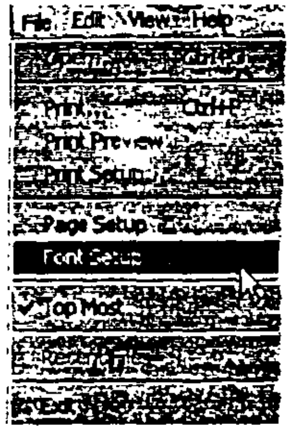


Figure 3.94

By default, the output file contains a listing of the entire Input also. You may choose not to print the echo of the Input commands in the Output file. Please select *Commands | Miscellaneous | Set Echo* option from the menu bar and select the *Echo Off* button.

It is quite important that we browse through the entire output file and make sure that the results look reasonable, that there are no error messages or warnings reported, etc. **Errors encountered during the analysis and design can disable access to the Post-processing mode – the graphical screens where results can be viewed graphically**. All errors in the model will be reported in the STAAD output file. The information presented in the output file is a crucial indicator of whether or not the structure satisfies the engineering requirements of safety and serviceability.

```
           P R O B L E M   S T A T I S T I C S
           -----------------------------------

    NUMBER OF JOINTS/MEMBER+ELEMENTS/SUPPORTS =     14/    14/
10
    ORIGINAL/FINAL BAND-WIDTH=     7/     3/     20 DOF
    TOTAL PRIMARY LOAD CASES =    3, TOTAL DEGREES OF FREEDOM
=    44
    SIZE OF STIFFNESS MATRIX =       1 DOUBLE  KILO-WORDS
    REQRD/AVAIL. DISK SPACE  =    12.0/ 14278.8 MB,  EXMEM =
76.0 MB

□
   PORTAL FRAME WITH SINKING SUPPORTS                          --
PAGE NO.    3



  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     1 )
       SUMMATION FORCE-X =          0.00
       SUMMATION FORCE-Y =        -31.16
       SUMMATION FORCE-Z =          0.00

     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=            0.00  MY=             0.00  MZ=          -
436.19

  ***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING     1 )
       SUMMATION FORCE-X =          0.00
       SUMMATION FORCE-Y =         31.16
       SUMMATION FORCE-Z =          0.00

     SUMMATION OF MOMENTS AROUND THE ORIGIN-
     MX=            0.00  MY=             0.00  MZ=
436.19

  MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING      1)
           MAXIMUMS    AT NODE
     X =  6.42833E-05     7
     Y = -1.23773E-02    14
     Z =  0.00000E+00     0
     RX=  0.00000E+00     0
     RY=  0.00000E+00     0
     RZ=  4.58081E-05    14

  ***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     2 )
       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=          -
1680.00
```

```
***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING      2 )
    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=
1680.00


  PORTAL FRAME WITH SINKING SUPPORTS
PAGE NO.    4

 MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING       2)
          MAXIMUMS    AT NODE
   X = -7.42318E-03      14
   Y = -8.42445E-02      14
   Z =  0.00000E+00       0
   RX=  0.00000E+00       0
   RY=  0.00000E+00       0
   RZ=  5.28974E-03      14

***TOTAL APPLIED LOAD ( KIP  FEET ) SUMMARY (LOADING     3 )
    SUMMATION FORCE-X =       10.00
    SUMMATION FORCE-Y =        0.00
    SUMMATION FORCE-Z =        0.00

   SUMMATION OF MOMENTS AROUND THE ORIGIN-
   MX=            0.00  MY=            0.00  MZ=
150.00

***TOTAL REACTION LOAD( KIP  FEET ) SUMMARY (LOADING      3 )
    SUMMATION FORCE-X =      -10.00
    SUMMATION FORCE-Y =        0.00
    SUMMATION FORCE-Z =        0.00

   SUMMATION OF MOMENTS AROUND THE ORIGIN-
   MX=            0.00  MY=            0.00  MZ=
150.00

 MAXIMUM DISPLACEMENTS ( INCH /RADIANS) (LOADING       3)
          MAXIMUMS    AT NODE
   X =  6.34271E-01       7
   Y =  1.90142E-02       1
   Z =  0.00000E+00       0
   RX=  0.00000E+00       0
   RY=  0.00000E+00       0
   RZ= -2.35485E-03       6


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

  51  PRINT ANALYSIS RESULTS
```

15. Post-Processing

STAAD.Pro offers extensive result verification and visualization facilities. These facilities are accessed from the *Post-Processing* Mode. The *Post-Processing* mode is used to verify the analysis and design results and generate reports.

For this session, we shall perform the following tasks:

- Display deflection diagrams
- Annotate Displacements
- Change the display units for displacement values shown in the tables
- Switch between load cases for viewing deflection diagrams
- Review the Nodes Displacement Table
- Animate the sectional displacements

15.1 Going To The Post-Processing Mode

Topics: Invoking the Post-processor

1. To access the *Post-Processing* mode, we can either click on the *Post-Processing* icon from the top toolbar or select it from the *Mode* menu as shown in the figures below.

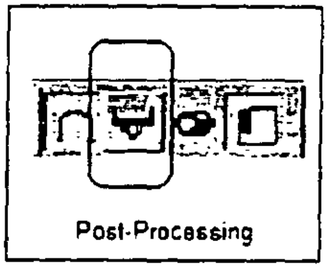


Figure 3.95

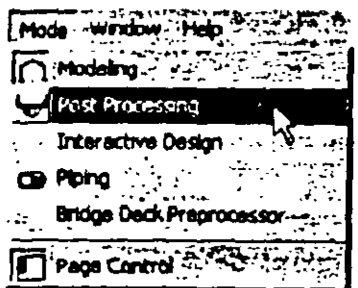


Figure 3. 96

2. The *Results Setup* dialog box appears as shown below. Here we can select the load cases for which to display the results. Let us select the load case from the *Available* selection box and click on the  button. Then, click on the *OK* button. The dialog box will look like the one shown below.

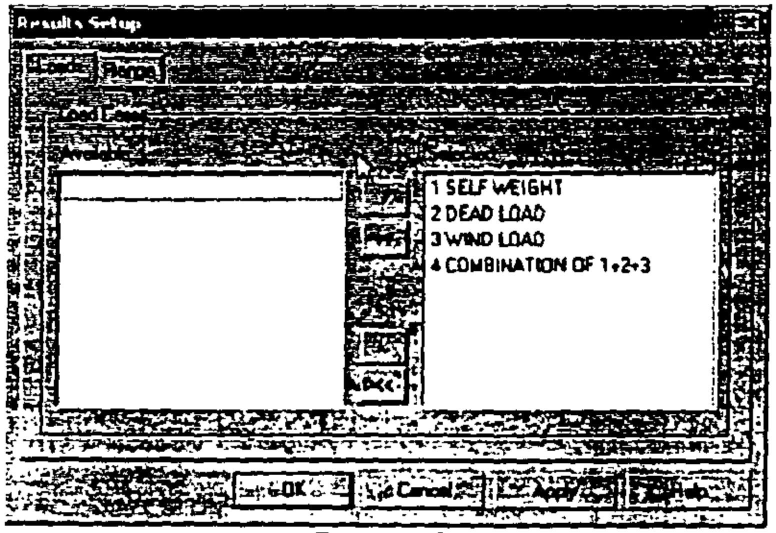


Figure 3.97

15.2 Viewing The Deflection Diagram

The screen will now look like the figure shown below.

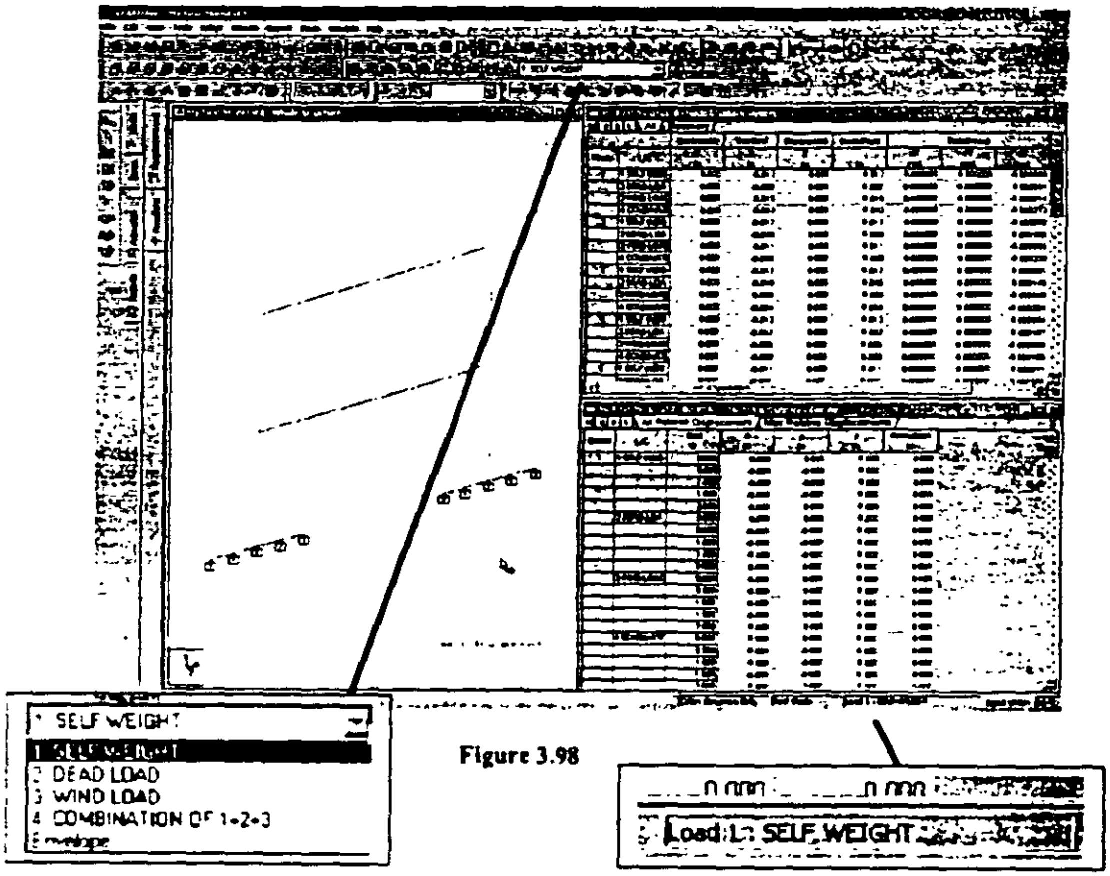


Figure 3.98

The diagram currently on display is the node deflection diagram for load case 1 (SELFWEIGHT). The title at the bottom of the diagram is indicative of that aspect

If you, let's say, wandered off into any other result diagram, and wanted to get back to the deflection diagram, just select the *Node | Displacement* tab along the page control area on the left side.

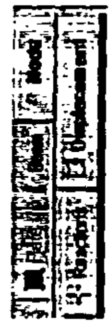


Figure 3.99

The option for selecting the deflection diagram is also available from another facility - the *Results | Deflection* menu option - as shown below.

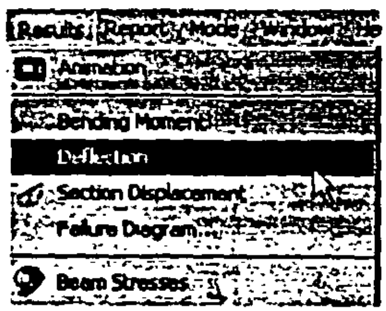
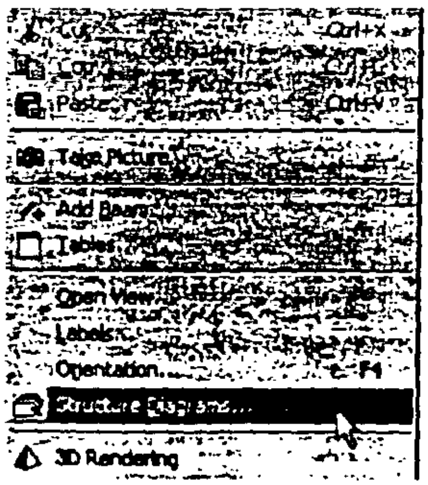


Figure 3.100

15.3 Switching Between Load Cases For Viewing The Deflection Diagram

Topics: Methods to graphically view results from different load cases

1. We can change the load case for which to view the deflection diagram by selecting a load case in the list box called *Active Load*. See Figure 3.98 Alternatively, either click on the *Symbols and Labels* icon or right-click your mouse and select *Structure Diagrams* as shown below.



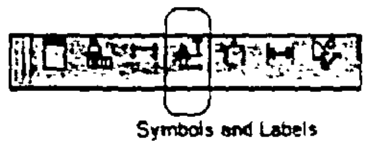


Figure 3.101

2. In either case, the *Diagrams* dialog box comes up. Select the *Loads and Results* tab and choose the desired load case from the *Load Case* list box. You can also turn on the load icons as well as other force diagrams by simply checking on the desired boxes. Click on *OK* once you have made your selections.

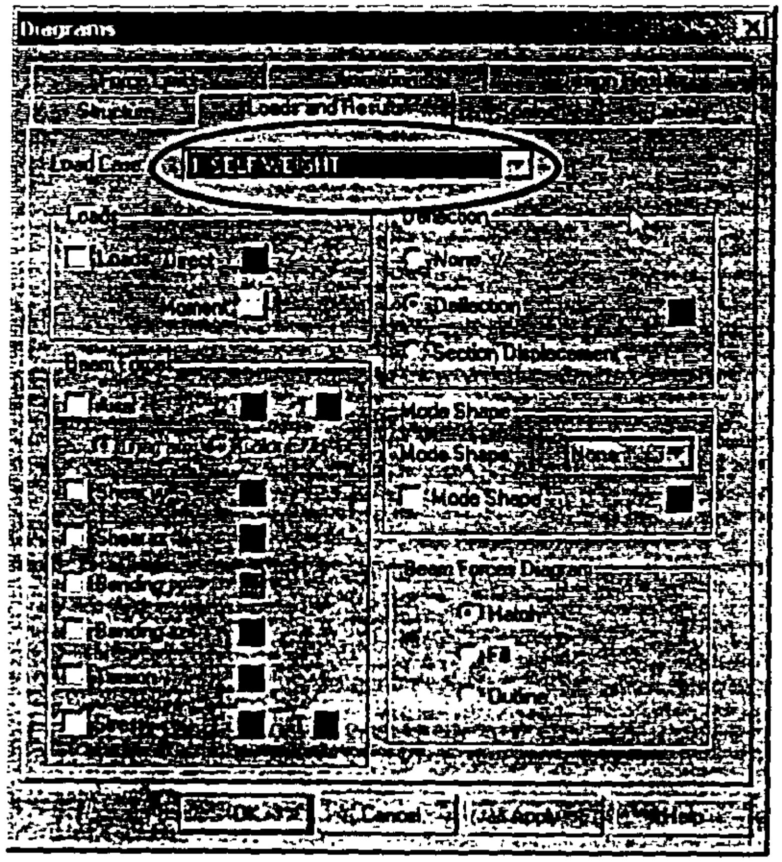


Figure 3.102

15.4 Changing The Size Of The Deflection Diagram

Steps: Scaling diagrams

1. If the diagram appears too imperceptible, it may be because it may be drawn to too small a scale. To change the scale of the deflection plot, you may

   a) click on the *Scale* icon from the toolbars or

Figure 3.103

   b) choose *Scale* from the *Results* menu or

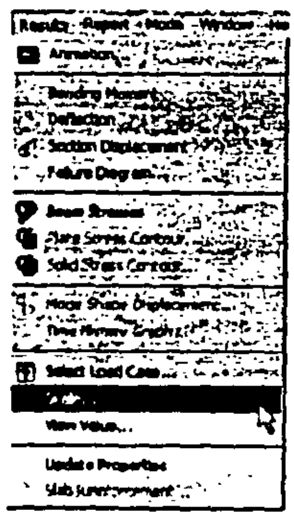


Figure 3.104

   c) right-click your mouse and choose *Structure Diagrams* and then select the "*Labels*" tab.

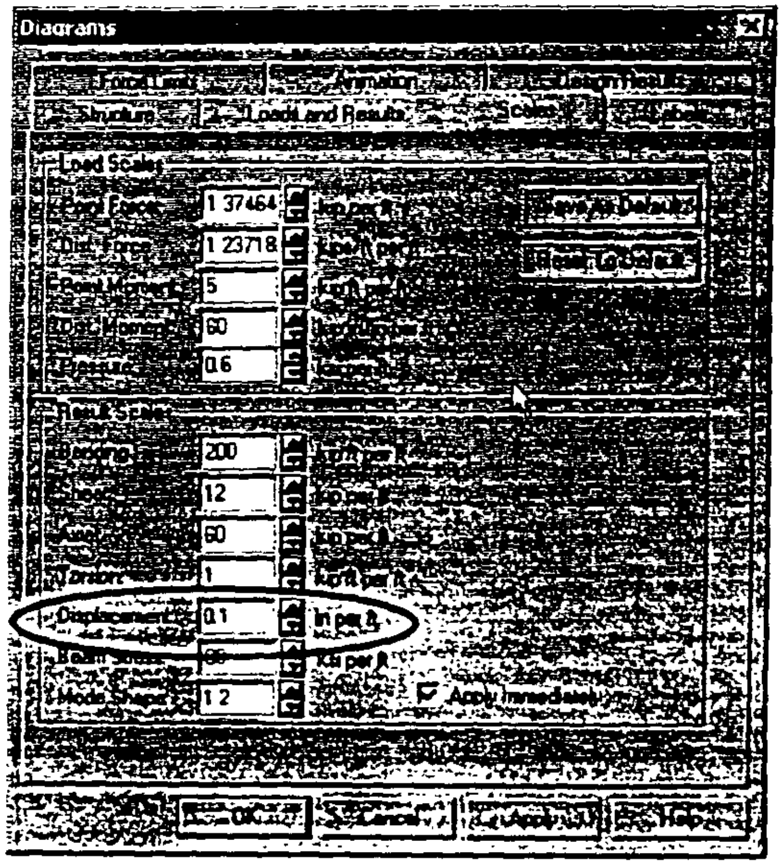


Figure 3.105

2. Check on the "*Apply immediately*" box to see the effects of the scaling immediately. In the *Displacement* field, specify a smaller number than what is currently listed. The deflection diagram should now be larger. In Figure 3.105, the number provided is 0.1 in per ft. This means that for every 0.1 inches of deflection, the program will scale it to 1 ft. Click on *OK* to close the box.

15.5 Annotating Displacements

Annotation is the process of displaying the displacement values on the screen.

Topics: Annotating values

1. Select the *View Value* option from the *Results* menu.

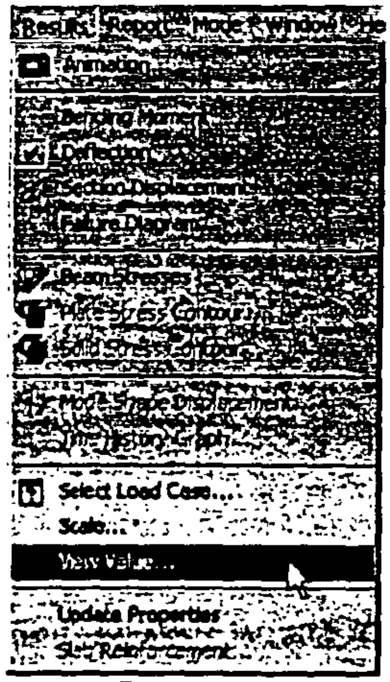


Figure 3.106

2. The following dialog box comes up. From the *Ranges* tab, select *All* nodes. If you wish to annotate deflection for just a few nodes, specify the node numbers in the node list. You can only use commas and hyphens as separators (i.e. 1,3,5 or 1-6,8)

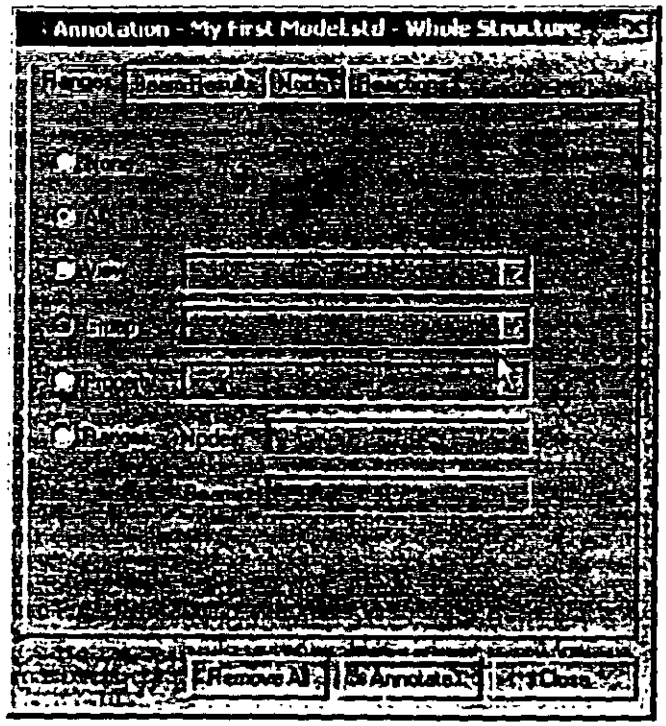


Figure 3.107

We will annotate the results for all nodes. So, keep the button on *All*.

3. From the *Node* tab, check the ***Resultant*** option. Resultant stands for the square root of sum of the squares of the values of X, Y and Z displacements. Click the ***Annotate*** button to view the values on the structure. Click the *Close* button to close the dialog box.

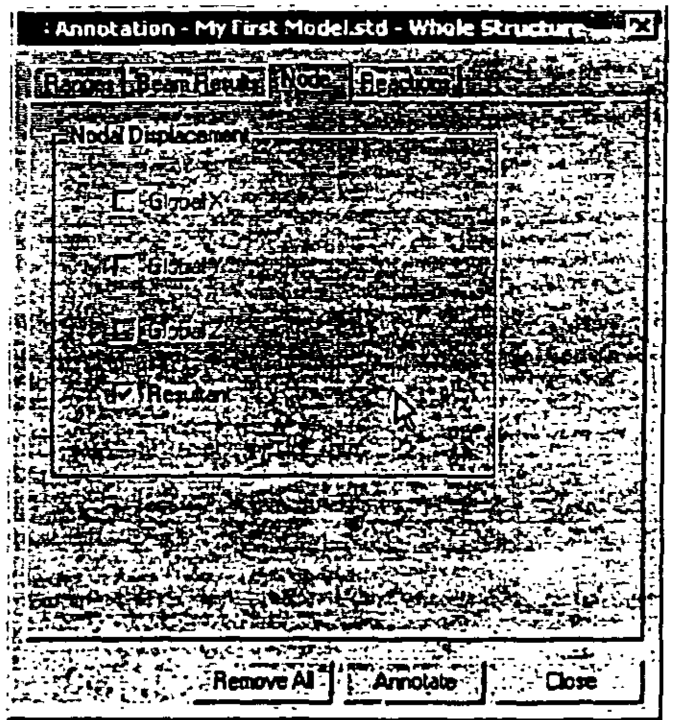


Figure 3.108

The following figure shows the annotated deflection diagram for load case 3 (Wind Load)

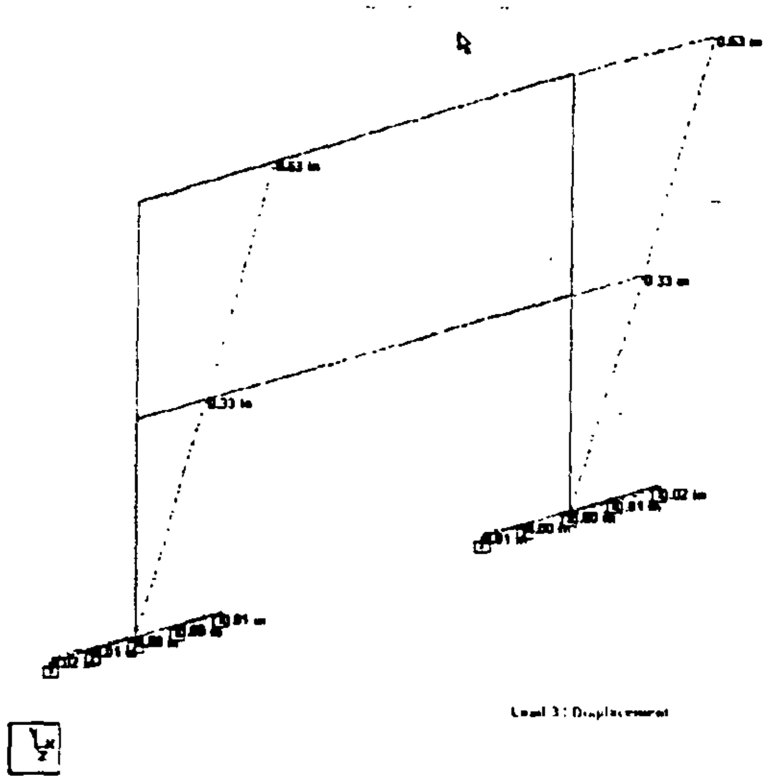


Figure 3.109

I can't see my annotations!!!

You can only annotate the values for a particular result if and only if the result diagram is displayed. For example, if you want to annotate the maximum bending moment values about the Z-axis, you must turn on the M_z diagram.

15.6 Changing the Units in which Displacement Values are Annotated

The units in which displacement values are displayed in the post-processing mode are referred to as the display units.

Topics: Changing Display Units

1. Display units may be modified from the *Tools* | *Set Current Display Unit* menu option as shown below.

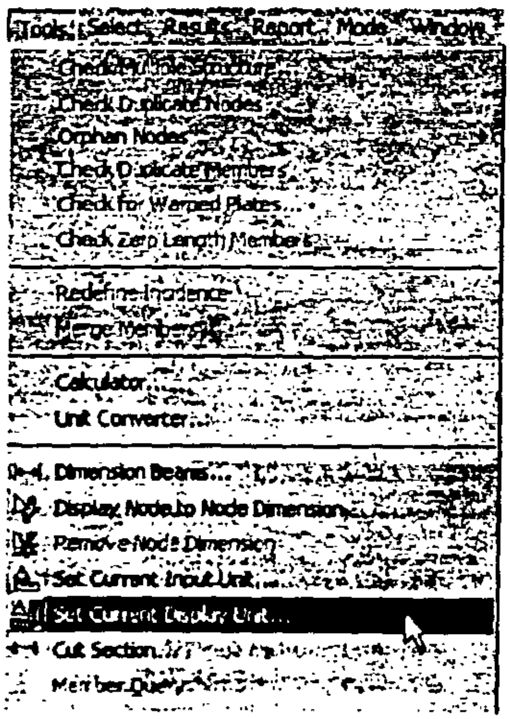


Figure 3.110

2. In the *Options* dialog box that comes up, select the *Structure Units* tab. Change the *Dimensions* of *Displacement* from *feet* to say, *cm* or *inches* or anything else you desire, and select *OK*. The diagram will be updated to reflect the new units.

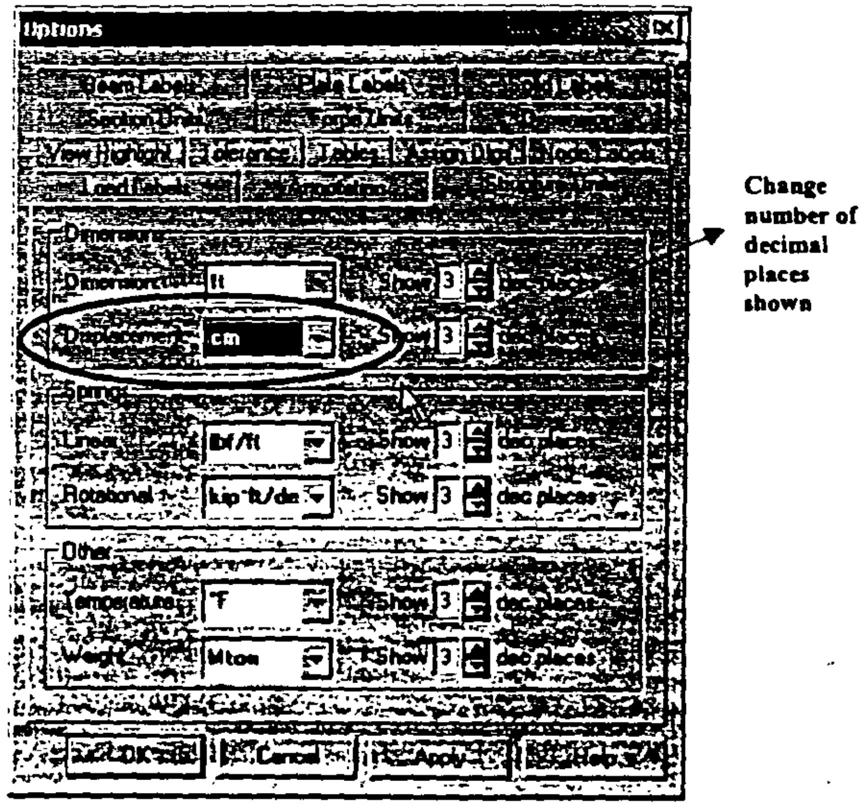


Figure 3.111

You can also change the number of decimal places to be shown for each value as well

The following figure shows the annotations of the deflection diagram in centimeters.

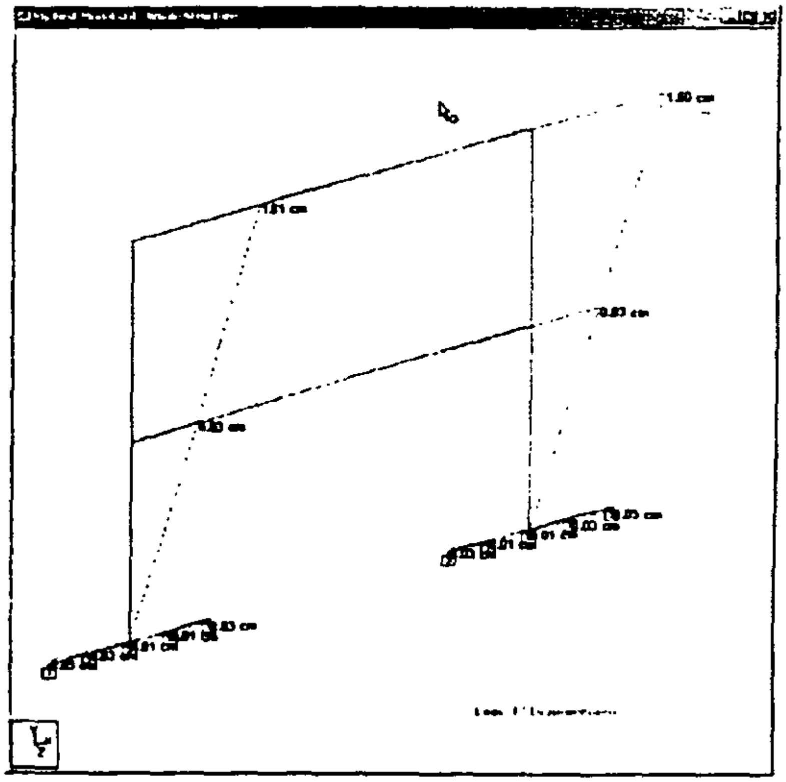


Figure 3.112

15.7 The Node Displacement Table

Upon entering the Post-Processing mode, the first screen that we came across is shown below.

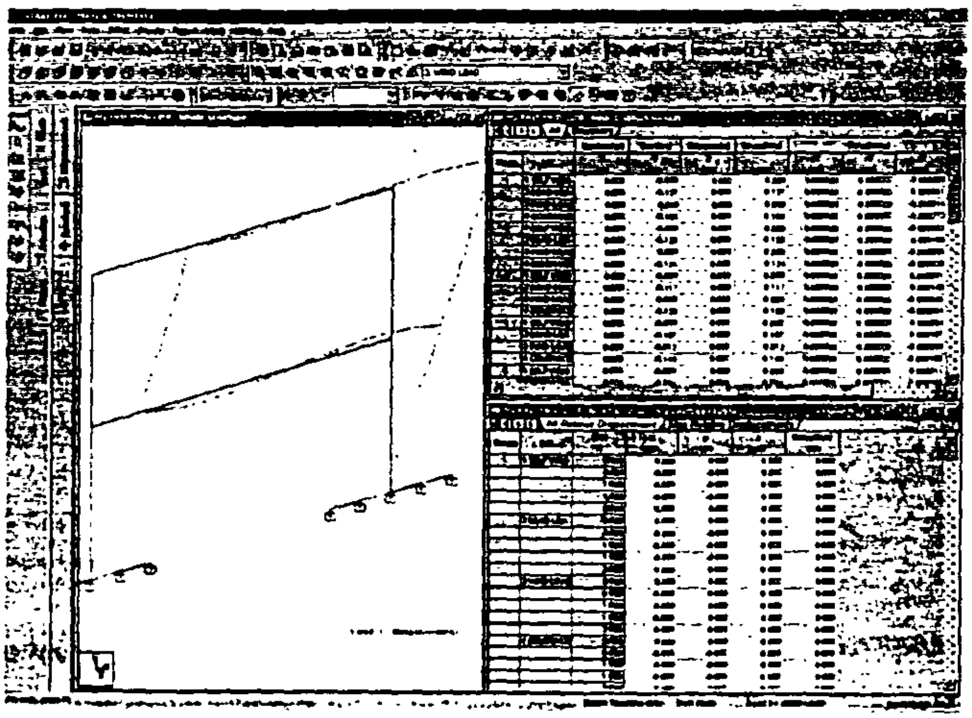


Figure 3.113

For the *Node | Displacement* page on the left side, notice that there are 2 tables displayed along the right side. The upper table, called the *Node Displacements Table*, lists the displacement values for every node for every selected load case. Load cases may be selected or de-selected for the purpose of this table from the *Results | Select Load Case* menu. The lower table is called the *Beam Relative Displacement Table*.

If you happen to close down any of these tables, you can restore them from the *View* | *Tables* menu or by right-clicking your mouse button and selecting *"Tables"*

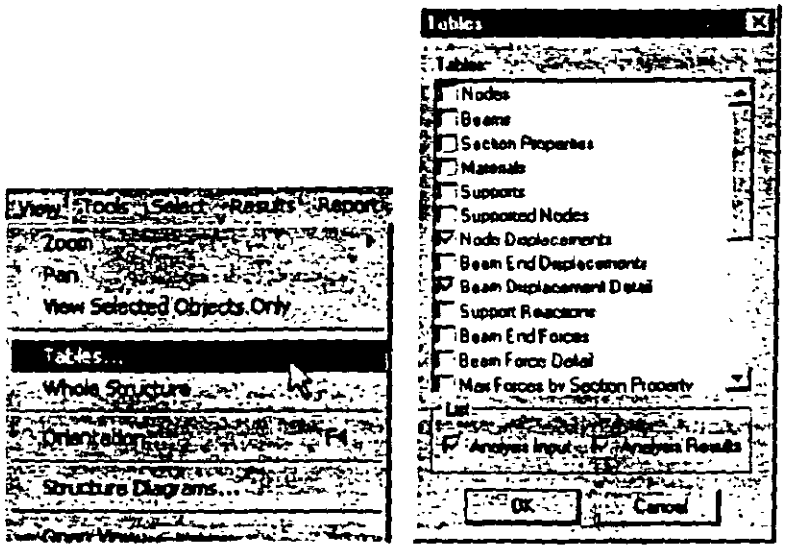


Figure 3.54

The *Node Displacement* table window has two tabs: *All* and *Summary* (see figure below).

Figure 3.115

All - This tab presents all nodal displacements in tabular form for all load cases and all degrees of freedom.

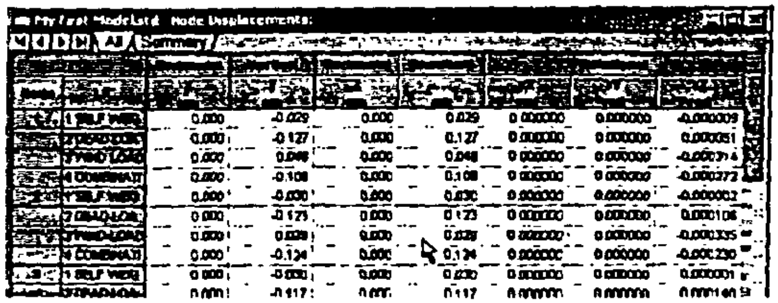


Figure 3.116

Summary - This tab, shown in the figure below, presents the maximum and minimum nodal displacements (translational and rotational) for each degree of freedom. All nodes and all Load Cases specified during the Results Setup are considered. Maximum values for all degrees of freedom (in bold type) are presented with the corresponding Node of occurrence and Load Case number (L/C).

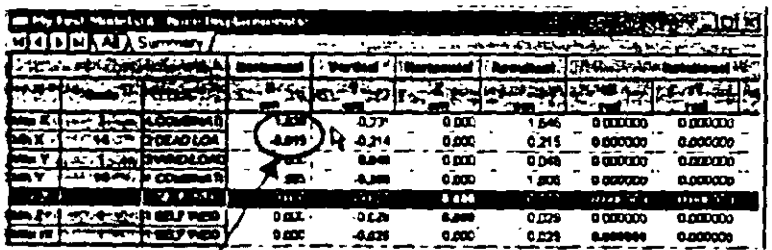


Figure 3.117

Maximum values are in bold

For the Beam Relative Displacement table, the details are as follows:

All - The *All* tab presents the displacements of members at intermediate section points. All specified members and all specified load cases are included. The table shows displacements along the local axes of the members, as well as their resultants.

Max Displacements - The *Max Displacements* tab presents the summary of maximum sectional displacements (see figure below). This table includes the maximum displacement values and location of its occurrence along the member, for all specified members and all specified load cases. The table also provides the ratio of the span length of the member to the resultant maximum section displacement of the member.

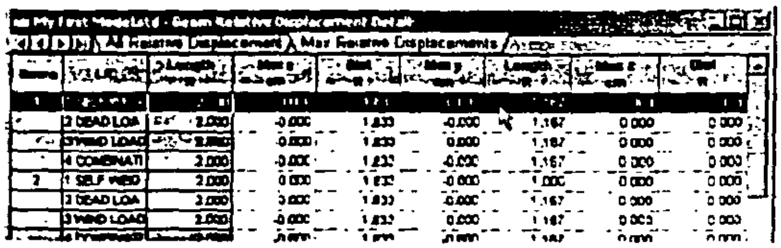


Figure 3.118

The sub-pages under the *Node* page are described below in brief.

Node	Displacement	Displays nodal displacements along with tabular results for Node-Displacements and sectional Beam displacements.
	Reactions	Displays support reactions on the drawing as well as in a tabular form.
	Modes	Displays mode shapes for the selected Mode shape number. The eigenvectors are simultaneously displayed in tabular form This Page appears only for dynamic analyses cases, namely, response spectrum, time history, and if modal calculations are requested
	Time History	Displays Time history plots, for time history analysis. This sub-page too will appear only if time history analysis is performed

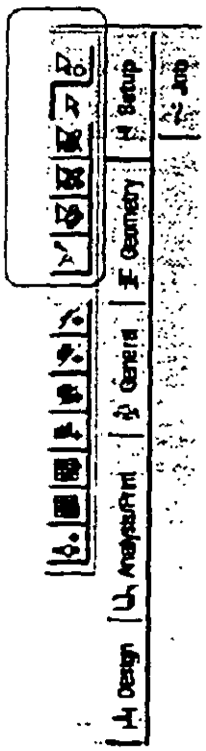
Other methods to view nodal results

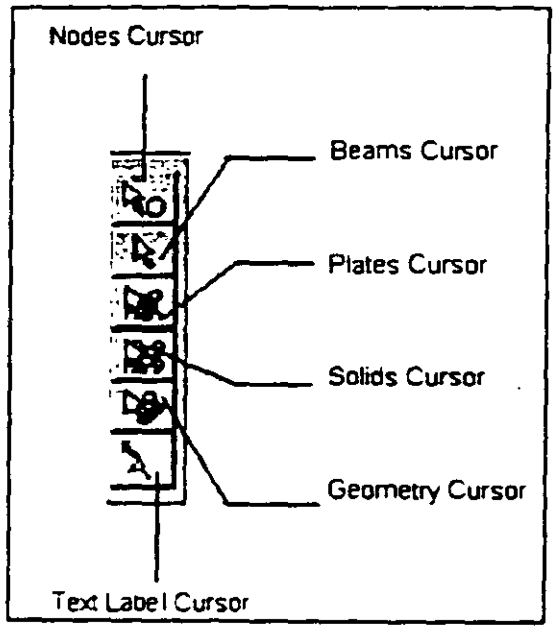
1) Select the nodes cursor and double-click on any node. You can simply double-click from node to node without having to close the dialog box.

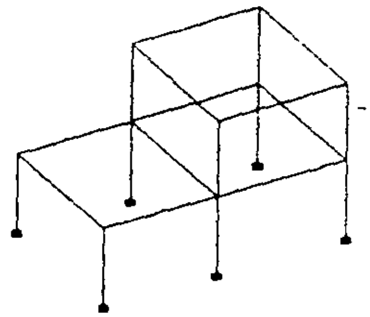
2) Go to View | Structural Tool Tip Options... and select the node displacements you want to appear when the mouse is placed over any node. Make sure you have selected the nodes cursor before placing your mouse over a node.

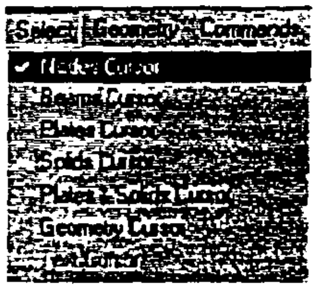
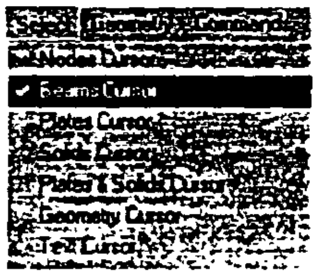
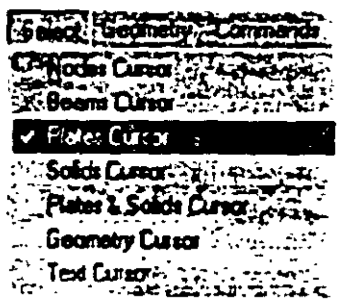
A. Selecting nodes, beams, plates, etc.

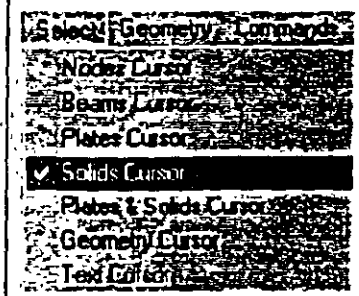
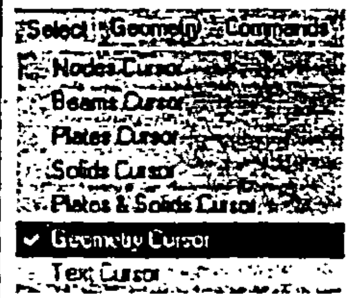
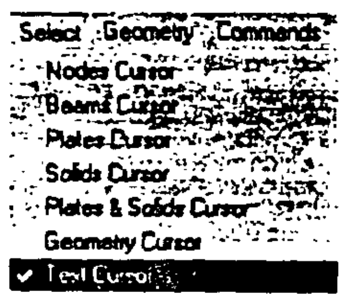
The Selection Toolbar





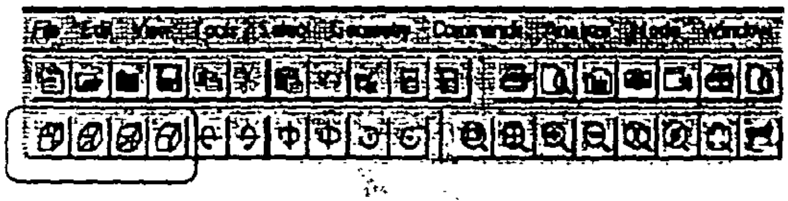
Icon	Description	Example
 Isometric View	Displays the structure in the isometric view.	

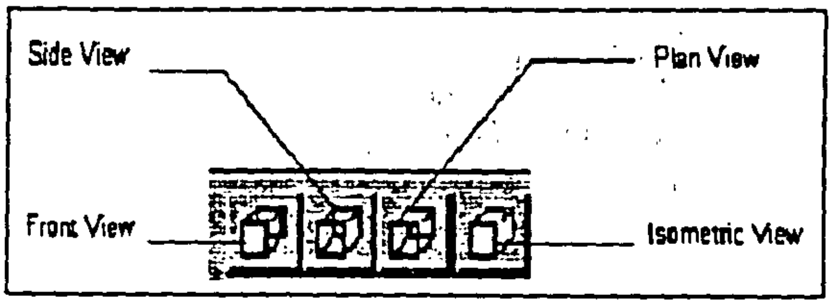
con	Corresponding Menu/ ub-menu options	urpose	Des ption
 Nodes Cursor	Select | Nodes Cursor 	Used to select nodes graphically	First, select the Nodes Cursor. Then, click on the nodes you wish to select. To select multiple nodes, hold down the Control key while selecting.
 Beams Cursor	Select | Beams Cursor 	Used to select nodes graphically	First, select the Beams Cursor. Then, click on the members you wish to select. To select multiple members, hold down the Control key while selecting.
 Plates Cursor	Select | Plates Cursor 	Used to select plates graphically	First, select the Plates Cursor Then, click on the plates you wish to select To select multiple plates, hold down the Control key while selecting.

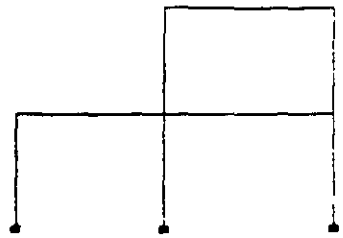
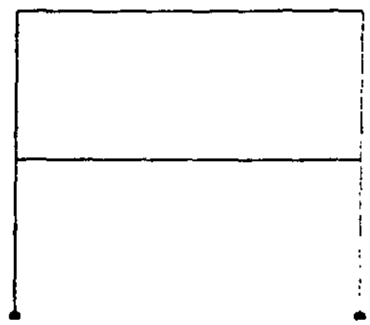
Icon	Corresponding Menu/ Sub-menu options	Purpose	Description
 Solids Cursor	Select | Solids Cursor 	Used to select solids graphically	First, select the Solids Cursor. Then, click on the solids you wish to select. To select multiple solids, hold down the Control key while selecting.
 Geometry Cursor	Select | Geometry Cursor 	Used to select any geometry graphically. It is a mechanism for selecting nodes, beams, plates and solids, or, any combination of these, simultaneously	First, select the Geometry Cursor. Then, click on the entity you wish to select. To select multiple entities, hold down the Control key while selecting
 Text Label Cursor	Select | Text Cursor 	Used to enter the mode for editing pre-created text labels	To edit any pre-created text, first, select the Text Label Cursor. Then, double-click on the text that you wish to modify.

B. Viewing the structure from different angles

The Rotation Toolbar





Icon	Description	Example
 Front View	Displays the structure as seen from the front. When the global Y axis is vertical, this is the X-Y plane view.	
 Side View	Displays the structure as seen from the side. When the global Y axis is vertical, this is the Y-Z plane view	
 Plan View	Displays the structure as seen from the top When the global Y axis is vertical, this is the X-Z plane view