



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

### **A LOS ASISTENTES A LOS CURSOS**

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

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

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

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

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

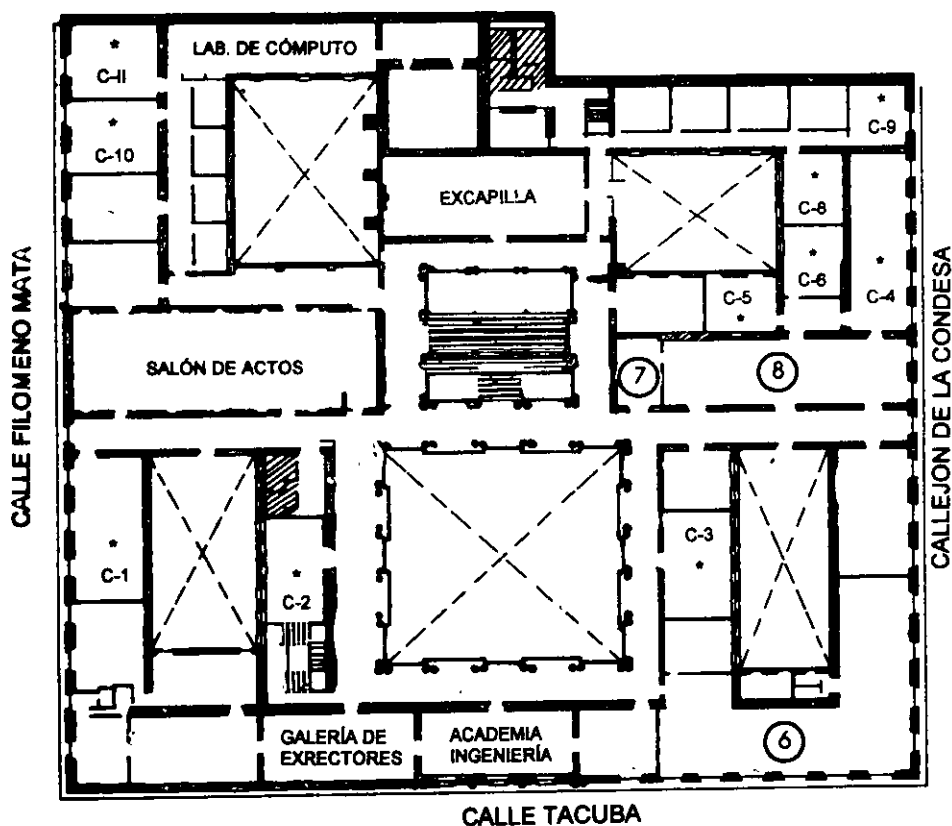
Con el objeto de mejorar los servicios que la División de Educación Continua ofrece, al final del curso deberán entregar la evaluación a través de un cuestionario diseñado para emitir juicios anónimos.

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

**Atentamente**

**División de Educación Continua.**

# PALACIO DE MINERIA



## GUÍA DE LOCALIZACIÓN

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

SANITARIOS

\* AULAS

**1er. PISO**

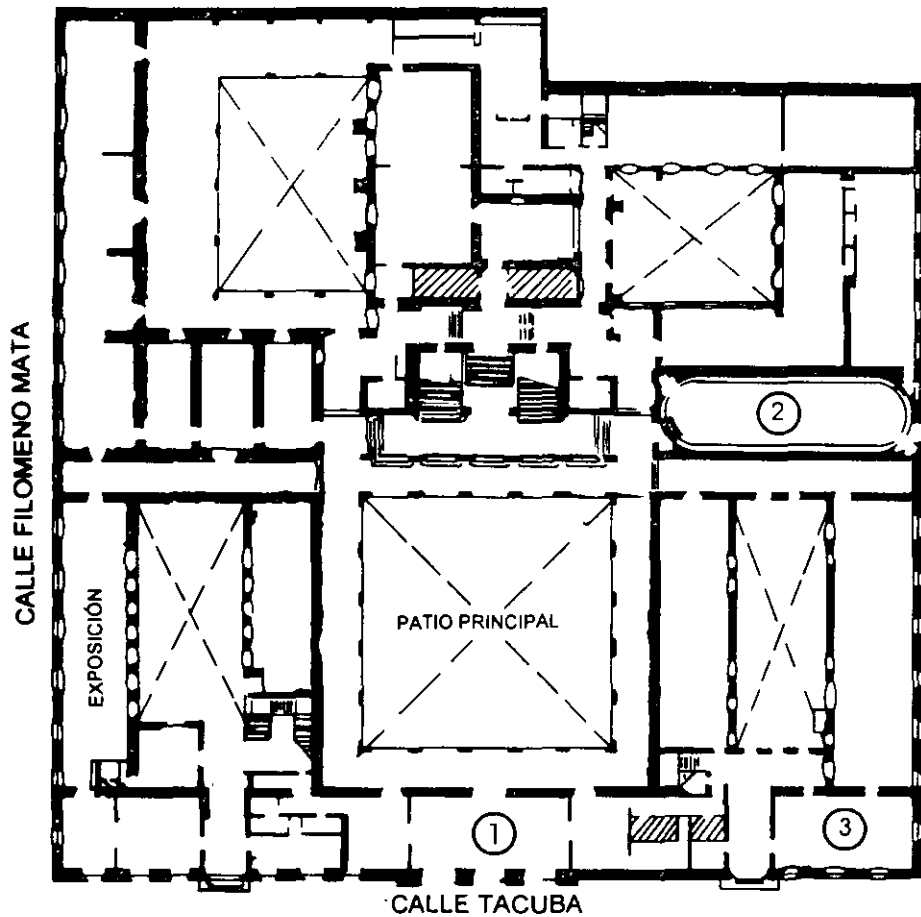


DIVISIÓN DE EDUCACIÓN CONTINUA  
FACULTAD DE INGENIERÍA U.N.A.M.  
CURSOS ABIERTOS

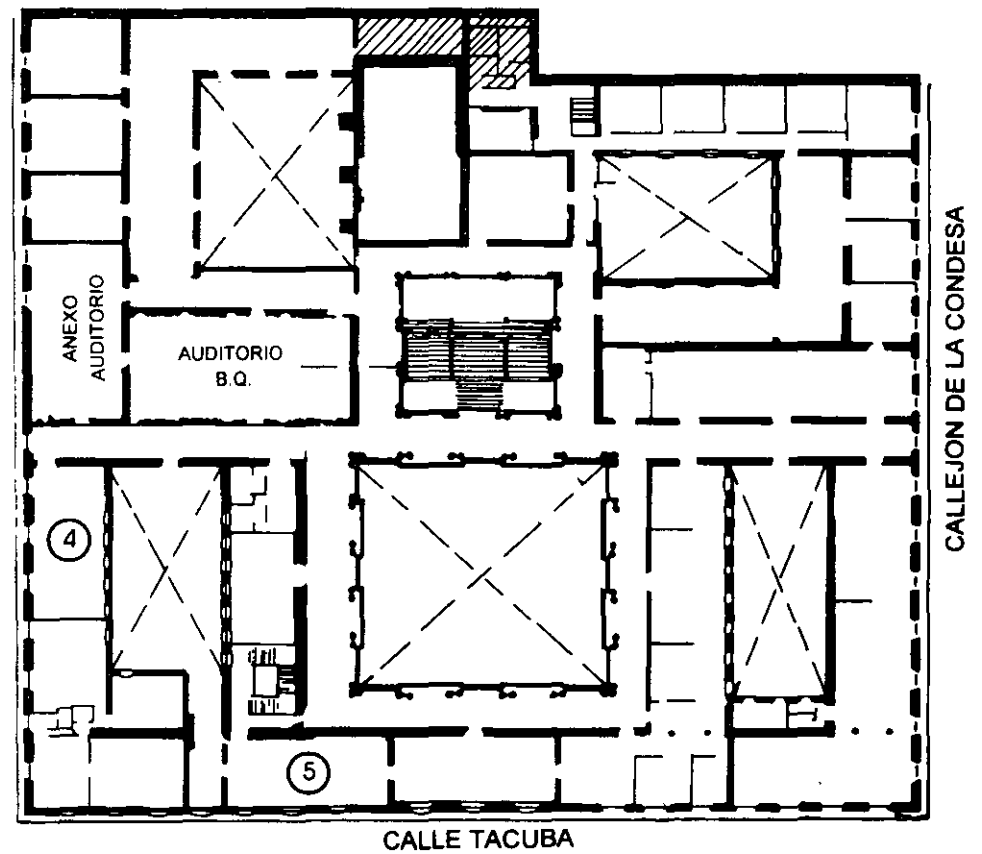
DIVISIÓN DE EDUCACIÓN CONTINUA



# PALACIO DE MINERIA



PLANTA BAJA



MEZZANINNE



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

## **CURSOS ABIERTOS**

### **CA-007 ETABS PARA ANÁLISIS Y DISEÑO TRIDIMENSIONAL DE EDIFICIOS**

**TEMA  
INSTRUCTIVO PARA LA UTILIZACIÓN DEL  
PROGRAMA DE COMPUTADORAS ETABS  
ANÁLISIS TRIDIMENSIONAL Y DISEÑO  
ESTRUCTURAL DE EDIFICIOS POR  
COMPUTADORA**

**ING. FERNANDO MONROY MIRANDA  
PALACIO DE MINERIA  
MARZO DEL 2004**

UNIVERSIDAD NACIONAL AUTÓNOMA DE MÉXICO  
FACULTAD DE INGENIERÍA

***INSTRUCTIVO PARA LA UTILIZACION DEL  
PROGRAMA DE COMPUTADORA ETABS***

*Análisis Tridimensional y Diseño  
Estructural  
de Edificios por Computadora*

FERNANDO MONROY MIRANDA

DIVISION DE INGENIERIA CIVIL TOPOGRÁFICA Y GEODESICA  
DEPARTAMENTO DE ESTRUCTURAS

# PRÓLOGO

El programa **ETABS** es uno de los programas mas conocidos en el campo de la Ingeniería Estructural, ha sido utilizado por un buen numero de ingenieros en nuestro país y en muchas partes del mundo, cuenta con respaldo y soporte tecnico al que tiene derecho el usuario autorizado así como los manuales respectivos

Por lo anterior el Departamento de Estructuras considero conveniente iniciar una serie de cursos para enseñar a los alumnos de la carrera de Ingeniero Civil a utilizar el programa **ETABS**, para ello el contar con un instructivo que permita introducir al usuario de una manera facil a ese programa facilitara el objetivo anterior

En este instructivo se pretende describir algunos de los principales elementos que interviene en el uso del programa de computadora para Analisis Tridimensional y Diseño Estructural de Edificios **ETABS**, cuya principal utilizacion sera para los alumnos de la materia "Diseño Estructural" de la carrera de Ingeniero Civil de la Facultad de Ingeniería de la UNAM

Se ha procurado realizar este instructivo de una manera sencilla y resumida para que el usuario no emplee demasiado tiempo en leerlo y pueda resolver su problema en lo que respecta al Analisis Tridimensional y Diseño Estructural de Edificios utilizando el programa **ETABS**

Se recomienda que si algunos de los elementos no son descritos ampliamente se consulten los manuales respectivos, las demostraciones animadas o la ayuda en linea incluida en el programa y se observen los ejemplos que se desarrollan al final del instructivo. Se supone que el usuario esta familiarizado con la nomenclatura y terminología utilizada en el Analisis y Diseño Estructural y que cuenta con conocimientos basicos de computación en lo que respecta a manejo de informacion (archivos) y ejecución de programas en ambiente Windows (98, 2000 o XP)

El autor agradece al M. en I. Octavio García Domínguez, Jefe del Departamento de Estructuras el apoyo para el desarrollo de este tipo de actividades, por las facilidades otorgadas para la realización de este trabajo así como la revisión del presente instructivo

**FERNANDO MONROY MIRANDA**

# CONTENIDO

## PRÓLOGO

## CAPÍTULO 1 EL PROGRAMA ETABS

### 1.1 INTRODUCCIÓN AL PROGRAMA ETABS

## CAPÍTULO 2 RECOMENDACIONES PARA EL USO DEL PROGRAMA

### 2.1 Paso 1 Tipos de Estructura

### 2.2 Paso 2 Definición de la Geometría

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

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

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

### 2.6 Paso 6 Elección del tipo de análisis y resultados

### 2.7 Paso 7 Diseño Estructural de Elementos

## CAPÍTULO 3 MÓDULOS DEL PROGRAMA DESCRIPCIÓN GENERAL

### 3.1 Ejecución del programa módulos que lo componen

### 3.2 Descripción General

## CAPÍTULO 4 GENERACIÓN DE LA ESTRUCTURA

### 4.1 Introducción

### 4.2 Descripción General

### 4.3 Generación de la Geometría

- 4.4 Asignación de propiedades geométricas
- 4.5 Definición y Asignación materiales
- 4.6 Condiciones de Frontera. tipos de apoyo
- 4.7 Asignación de Fuerzas y combinaciones
- 4.8 Opciones de Análisis y Diseño. seleccion de resultados

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

## **CAPÍTULO 6 VER RESULTADOS**

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

## **CAPÍTULO 7 LOS MÓDULOS COMPLEMENTARIOS**

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

## **CAPÍTULO 8 EJEMPLOS E INTERPRETACIÓN DE RESULTADOS**

- Ejemplo No 1 Marco Plano
- Ejemplo No 2 Marco Plano con diagonales y muros
- Ejemplo No 3 Edificio Tridimensional (Análisis Estático)
- Ejemplo No 4 Edificio Tridimensional (Características Dinámicas)
- Ejemplo No 5 Edificio Tridimensional (Análisis Dinámico Modal Espectral)
- Ejemplo No 6 Edificio Tridimensional (Interacción Suelo-Estructura)

## **CAPÍTULO 9 COMENTARIOS FINALES**



# CAPÍTULO 1

## INTRODUCCIÓN AL PROGRAMA ETABS

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

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

El sistema **ETABS** es un programa escrito para computadoras personales IBM o compatibles mediante el cual puede realizarse el Análisis Tridimensional de Edificios y Diseño Estructural bajo uno o más sistemas de carga formados por un conjunto de fuerzas estáticas y/o dinámicas aplicadas a la estructura.

**ETABS** 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 muros y losas. Consta básicamente de una serie de módulos (Véase Figura 1) de ellos, en este instructivo se describirá solo el módulo **ETABS**, en este módulo 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.

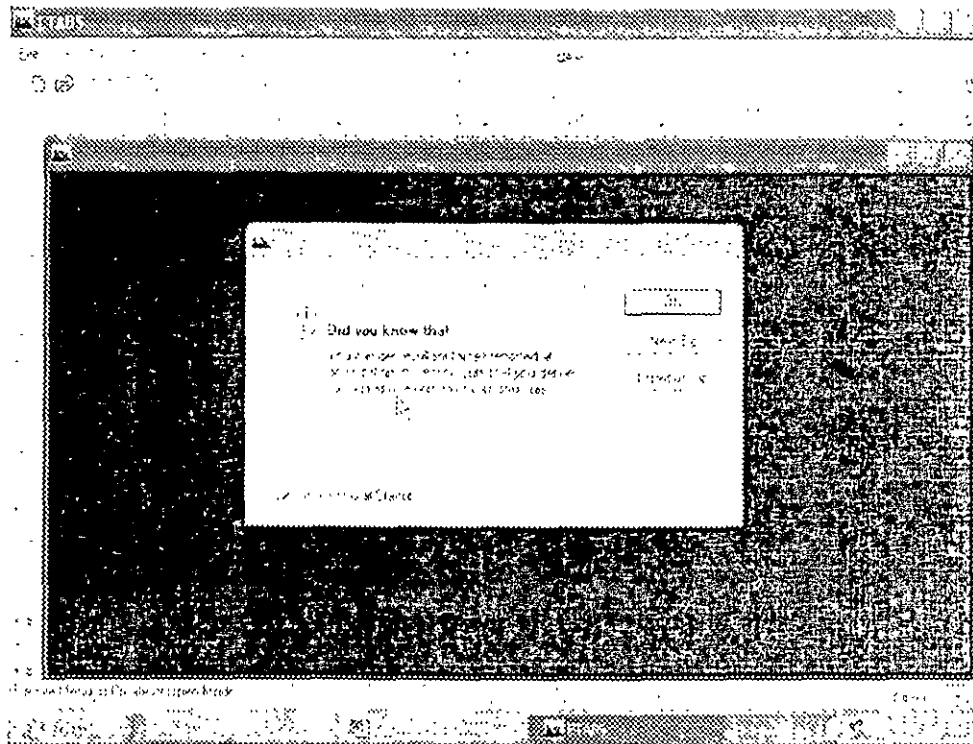


Figura 1.1 ETABS, programa principal

Una de las principales características del programa es la interacción que se puede establecer entre éste y el usuario 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. **ETABS** es un programa orientado a eventos (seleccionar un elemento con el ratón, elegir una opción, activar/desactivar sucesos, etc.) y no siempre solicita textualmente los elementos (datos) que se vayan requiriendo para la ejecución completa de ese módulo, además es necesario saber las convenciones de signos empleadas, los sistemas de referencia utilizados así como algunas recomendaciones para su uso, éstas y algunas características más son descritas en los capítulos posteriores.

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



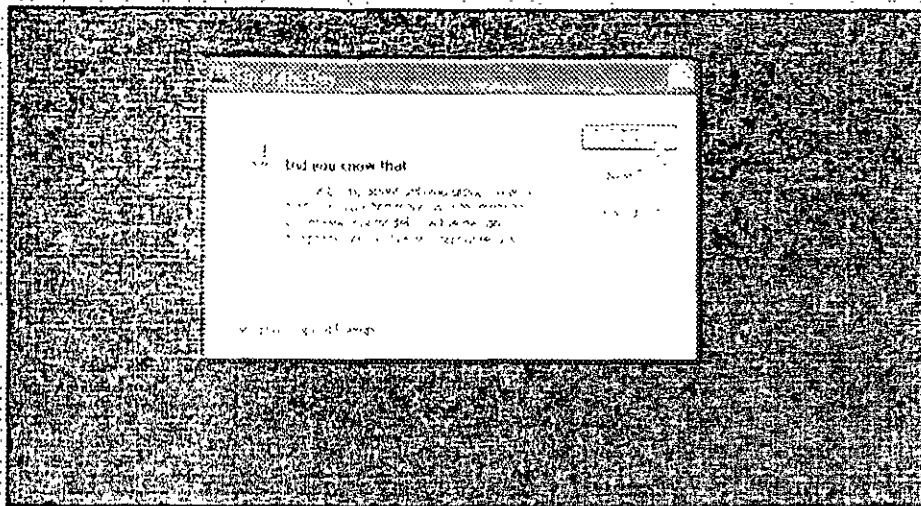


Figura 2.2 Inicio del programa ETABS

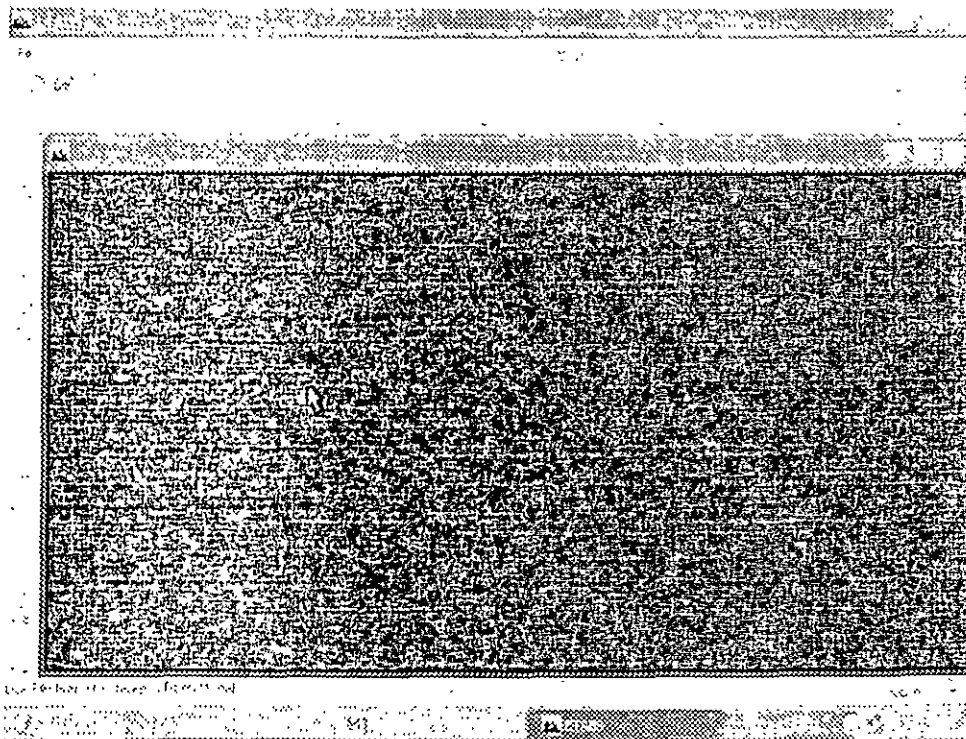


Figura 2.3 Programa ETABS. ventana principal.



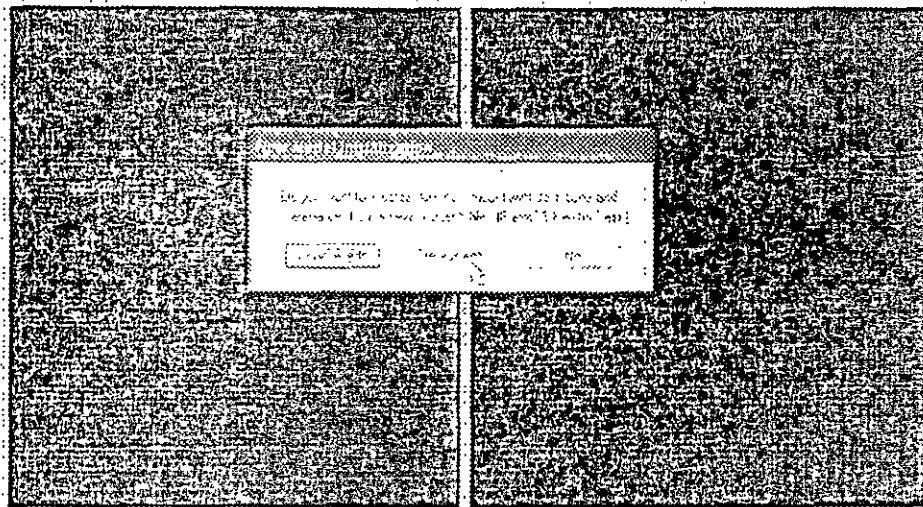


Figura 2.5 Definición de un nuevo modelo

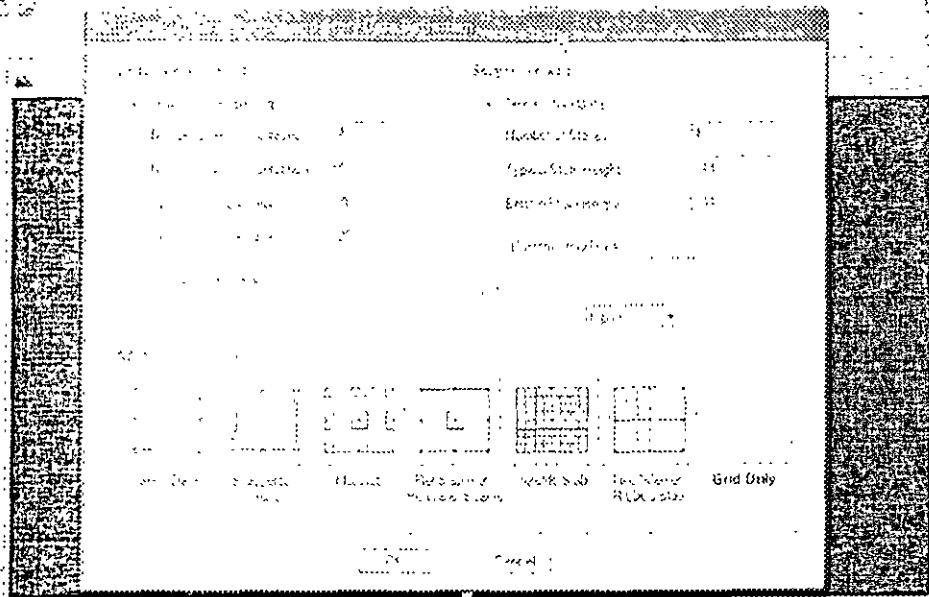
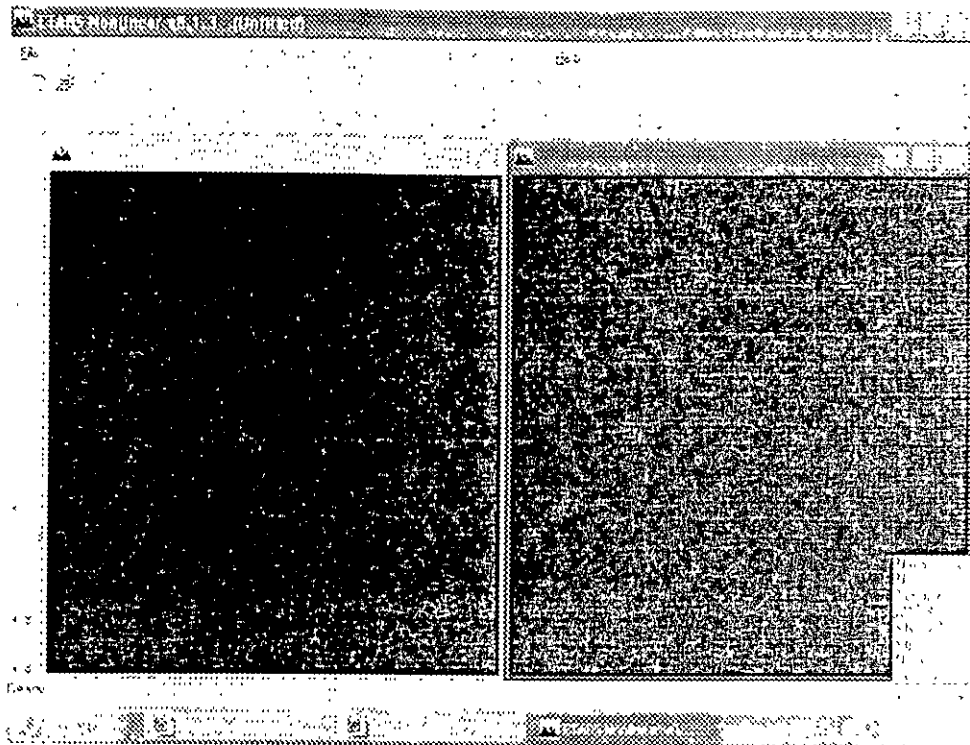


Figura 2.6 Datos generales para iniciar un modelo.



**Figura 2.7** Datos de unidades al inicio del programa **ETABS**

**ETABS** permite analizar estructura plana o en 3 dimensiones (3D). Obsérvese que en la ventana principal del programa **ETABS**, en general, están contenidos algunos elementos típicos de varios programas desarrollados para ambiente o plataforma **Windows**, es decir, una barra de título (extremo superior de la ventana), una de menús desplegables (File, Edit, View, etc.), barras de iconos (algunos típicos, y otros propios del programa **ETABS**), una barra de estado en el extremo inferior de la ventana (Ready). En el extremo izquierdo del área con fondo negro se muestran algunos iconos, en esa área se desplegará gráficamente la geometría y algunas características de la estructura (apoyos, cargas, etc.).

### **2.3 DEFINICIÓN DE LA GEOMETRÍA**

Antes de iniciar la ejecución del programa **ETABS** 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 se podrán modelar como elementos barra (trabes, columnas, diagonales) y elementos finitos placa (losas, muros y rampas), todos esos elementos estarán unidos en puntos comunes (nudos), algunos nudos estarán completamente o parcialmente restringidos (apoyos), en uno o varios grados de libertad.

La definición de los elementos (barra y placa) 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) queda definida la ubicación del elemento

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

## 2.4 DEFINICIÓN DE LAS PROPIEDADES GEOMÉTRICAS DE LOS ELEMENTOS

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

- 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 sus propiedades referidas a ejes locales y centroidales de la barra, algunas de esas propiedades son

- $A_x$  = Área de la sección transversal
- $I_x$  = Constante de torsión
- $I_y$  = Momento de inercia al rededor del eje y
- $I_z$  = Momento de inercia al rededor del eje z
- $A_v$  = Área de cortante en dirección y
- $A_z$  = Área de cortante en dirección z
- $J$  = Constante de torsión

Para secciones específicas (rectangular, circular, etc ) las propiedades son obtenidas por el programa solo 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 )

El programa **ETABS** permite asignar las propiedades 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

Al programa se le pueden dar instrucciones para que de manera automática trabaje a los elementos con secciones específicas (columna, viga, sección T, o formada por ángulo (s) etc )



Para el caso de los elementos placa será necesario proporcionar el espesor de la placa

## **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, sólido) como son  $E$  (Modulo elástico) y  $\nu$  (relación de Poisson) Para incluir el peso propio es necesario proporcionar el peso volumétrico. si se consideran efectos de temperatura sera necesario especificar el coeficiente lineal de dilatacion 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 analisis ( $P$  ej peso propio, carga viva, sismo, viento, etc ) y para cada condicion de carga contar con las características de las fuerzas (tipo, magnitud, dirección, sentido y punto de aplicacion) que forman parte de 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 simulan, por ejemplo el peso de tanques, etc. Otra condicion de carga puede ser el sismo que puede ser representado o por una serie de fuerzas estaticas (sismo estático) aplicadas a determinados nudos. Una condicion mas puede ser la carga viva, idealizada como una fuerza por unidad de area actuando en una determinada zona de la estructura ( $P$  ej azotea, entrepiso, escaleras, etc )

Los sistemas de carga independientes pueden ser utilizados para formar sistemas de carga dependientes, es decir combinaciones, si lo anterior se desea, es necesario saber de antemano el número de combinaciones a incluir en el análisis y para cada combinación, las condiciones de carga que se incluirán así como su participacion 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 sera 1.5 de la carga muerta + 1.5 de la carga viva máxima, por lo que el factor de carga o participación de las condiciones anteriores 1 y 2 es 1.5 siendo 1 y 2 las condiciones de carga respectivas (1 la carga muerta y 2 la viva)

## **2.7 ELECCIÓN DEL TIPO DE ANÁLISIS Y LOS RESULTADOS**

**ETABS** permite realizar un análisis elástico lineal de 1er Orden, dinámico y considerar efectos P- $\Delta$ , 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 especificar cuales se requeriran, por ejemplo, desplazamientos, elementos mecánicos, graficas y resultados de diseño (revisión), graficas de la deformada de algún marco o de toda la estructura, etc

## **2.8 DISEÑO DE ELEMENTOS**

**ETABS** permite diseñar elementos de acero, concreto y madera por lo que será necesario especificar un código a utilizar (ACI, AISC, LRFD, ASSTHO, etc ), así como proporcionar los valores de los parametros a utilizar (p. ej.  $f_c$ ,  $f_y$ , etc ), e indicar los elementos que se diseñaran y el criterio a seguir para su diseño.

# ETABS<sup>®</sup>

## Integrated Building Design Software

### An Introduction to ETABS



**Computers and Structures, Inc.**  
Berkeley, California, USA

Version 8  
January 2002

# Copyright

The computer program ETABS and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

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

Phone: (510) 845-2177

FAX: (510) 845-4096

e-mail: [info@esiberkeley.com](mailto:info@esiberkeley.com) (for general questions)

e-mail: [support@esiberkeley.com](mailto:support@esiberkeley.com) (for technical support questions)

web: [www.esiberkeley.com](http://www.esiberkeley.com)

© Copyright Computers and Structures, Inc. 1978-2000

The CSI Logo is a registered trademark of Computers and Structures, Inc.

ETABS is a registered trademark of Computers and Structures, Inc.

Windows is a registered trademark of Microsoft Corporation.

Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

# DISCLAIMER

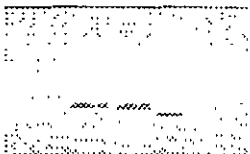
CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABS. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED IN USING THE PROGRAM. HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY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

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

# An Introduction to ETABS

<b>1</b>	<b>Introduction</b>	
	Using This Manual	1-1
	Help Available within the Program	1-2
	Overview of the Program	1-2
	Fundamental Concept	1-3
	Variety of Options	1-3
	Numerical Methods	1-3
	Advanced Capabilities	1-4
	Data Sharing with Other Software Programs	1-4
<b>2</b>	<b>An Example Model</b>	
	The Project	2-2
Step 1	Begin a New Model	2-2
	Define an Auto Select Section List	2-5
Step 2	Add Line Objects	2-7
	Set Up to Add Objects to Multiple Stories Simultaneously	2-7
	Draw Column Objects	2-8
	Save the Model	2-13
	Draw the Lateral Force-Resisting Beam Objects	2-13

	Draw the Secondary (Infill) Beam Objects	2-15
Step 3	Add Area Objects	2-18
	Draw the Floor Area Objects	2-18
	Add Dummy Area Objects used for Wind Load Application	2-21
	Add Objects in Elevation View	2-21
	Add Objects in Plan View	2-22
Step 4	Define Static Load Cases	2-25
Step 5	Assign Gravity Loads	2-29
Step 6	Define a Developed Elevation	2-34
Step 7	Assign Wind Loads	2-37
Step 8	Review Database Display of Input Data	2-42
Step 9	Run the Analysis	2-44
Step 10	Graphically Review the Analysis Results	2-44
Step 11	Design the Composite Beams	2-47
Step 12	Design the Steel Frame	2-55



## Chapter 1

# Introduction

## Using this Manual

This manual introduces you to ETABS Version 8. The step-by-step instructions guide you through development of your first model. The intent is to demonstrate the fundamentals and to show how quickly and easily a model can be created using this program.

ETABS is an extremely versatile and powerful program with many features and functions. This manual does not attempt to fully document all of those features and functions. Rather, we briefly show how to work with the program, providing some commentary along the way. To grasp the full value of ETABS, you should use this introductory manual in conjunction with the other ETABS documentation, such as the graphical user interface reference manual and the steel, concrete, shear wall, and composite floor design manuals.

We hope you enjoy your first experience with ETABS. It is the first step in changing the way you work, for the better.





## Help Available within the Program

The program includes extensive on-line help that is available any time the graphical interface is open. The documentation is accessible in two forms: a standard Windows style of help file, and an extensive Documentation library.

Access the Windows style of help by clicking on the **Help** menu and selecting **Search For Help On...**, or by pressing the F1 function key on the keyboard. If the F1 key is pressed while a form is open, context-sensitive help related to that form will display. The Windows style of help provides guidance with respect to entering data into the various forms used in the program. It often also clarifies the meaning of the data entered into the forms.

The Documentation library is a series of .pdf files that can be viewed or printed using Adobe® Acrobat Reader®. Access the Documentation library using the **Help menu > Documentation and Tutorials** command, which will bring up the ETABS Documentation form. This form displays the various categories of documentation available.

Double click (left click) on a category name to display a list of the individual documents that are available in .pdf format. Note that some categories also have subcategories.

Right click on the name of an individual document to pop up summary information about the document contents, size and date.

Double click (left click) on the name of an individual document or highlight it and click the **Display Selected Document** button to launch Adobe® Acrobat Reader® and display the selected document. Note that many files contain hyperlinks to facilitate cross referencing among the documents.

## Overview of the Program

ETABS is a stand-alone finite-element-based structural analysis program with special purpose features for structural design and analysis of building systems. Embedded beneath the simple, intuitive user interface are



very powerful numerical methods, design procedures and international design codes that allow you to be versatile and productive, whether you are designing a simple 2-dimensional frame or performing a dynamic base isolation analysis of a complex high-rise.

## **Fundamental Concept**

ETABS works off of an integrated database. The basic concept is that you create only one model consisting of the floor systems and the vertical and lateral framing systems to analyze and design the whole building. Everything you need is integrated into one versatile analysis and design system with one user interface. There are no external modules to maintain and no worries about data transfer between modules. The effects on one part of the structure from changes in another part are instantaneous and automatic.

## **Variety of Options**

The analysis methods include a wide variety of Static and Dynamic Analysis Options. The integrated model can include, among others, complex Composite Floor Framing Systems with Openings and Overhangs, Steel Joist Systems, Moment Resisting Frames, Complex Shear Wall Systems, Rigid and Flexible Floors, Sloped Roofs, Ramps and Parking Structures, Mezzanine Floors, Trussed Systems, Multiple Tower Buildings and Stepped Diaphragm Systems.

## **Numerical Methods**

The numerical methods used to analyze the building allow modeling of steel deck floors and concrete floor systems that can automatically transfer their loads to main girders. The automated finite element meshing of complex floor systems with automated displacement interpolation at mismatched mesh transitions, coupled with Ritz analysis for dynamics, makes inclusion of diaphragm flexibility effects in the analysis very practical.

Vertical Dynamic Analysis options allow you to include the effects of vertical ground motion components in your earthquake analysis. It also



allows you to perform detailed evaluations of vertical floor vibration problems in addition to the traditional empirical methods that are also built into the software.

Special problems associated with building type structures have been addressed with customized numerical techniques that allow you to include their effects into your analysis effortlessly. The special problems include, among others, Calculation of Centers of Rigidity, Global and Local P-Delta Effects, Inclusion of Joint Panel Zone Deformations, Effects of Joint Rigid Zone Ends, and Member End Offsets due to Cardinal Points of a section.

### **Advanced Capabilities**

More advanced numerical methods include sophisticated options for modeling Nonlinear Dampers, Pushover Analysis, Base Isolation, Construction Sequence Loading, Structural Pounding and Uplift.

### **Data Sharing with Other Software Programs**

A wide variety of export options allow you to transfer information from the ETABS database for use with other software packages. Some uses of these export options, among others, are framing plans and elevations using AutoCAD, foundation and slab analysis using SAFE, and information for detailing packages using CIS/2 Step files.

---

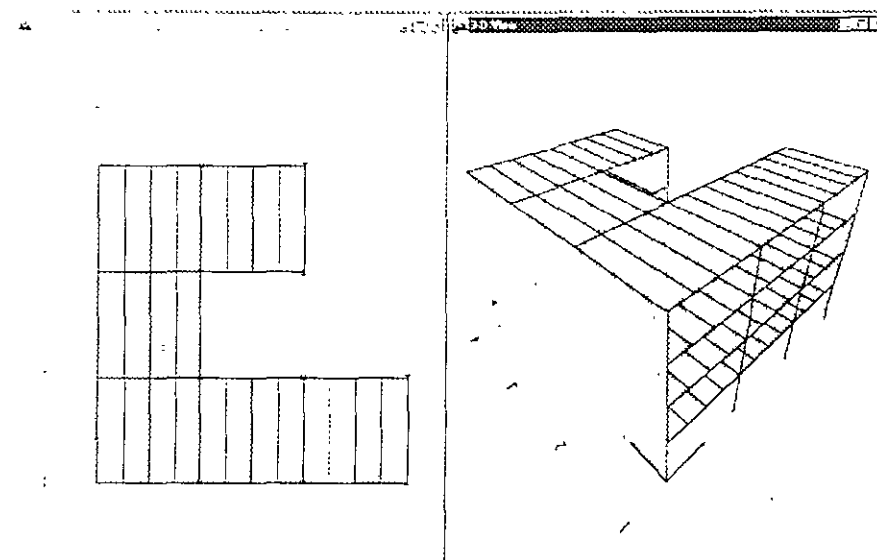
## Chapter 2

---

### An Example Model

This chapter provides step-by-step instructions for building a basic LTABS model. Each step of the model creation process is identified, and various model construction techniques are introduced. At the completion of this chapter, you will have built the model shown in Figure 1.

*Figure 1  
An Example  
of a Model*



## The Project

2


The example project is an irregularly shaped four-story building. The first story is 15 feet high and stories 2, 3, and 4 are each 12 feet high. The bays are 24 feet in the X and Y directions.

The lateral force resisting system consists of intersecting moment frames. The floors consist of 3 inches of concrete over a 3-inch-deep metal deck. The secondary (mfld) beams are designed as composite beams. The lateral-force resisting beams that connect the columns are designed as non-composite beams.

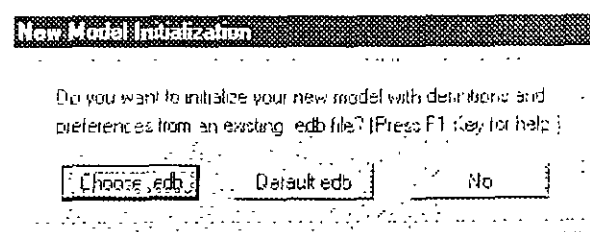
The architect for the building has requested that the maximum beam depth not exceed that of a W18 beam to allow sufficient clearance for ductwork running beneath the beams.

### Step 1 Begin a New Model

In this Step, the dimensions and story height are set. Then a list of sections that fit the parameters set by the architect for the design are defined.

- A. If the units shown in the drop-down box in the lower right-hand corner of the ETABS window are not Kip-in, click the drop-down box to set the units to Kip-m.
- B. Click the **File menu > New Model** command or the New Model button . The form shown in Figure 2 will display.

*Figure 2  
The New  
Model  
Initialization  
form*



**Figure 3**  
The Building  
Plan Grid  
System and  
Story Data  
Definition  
form



**Note:**

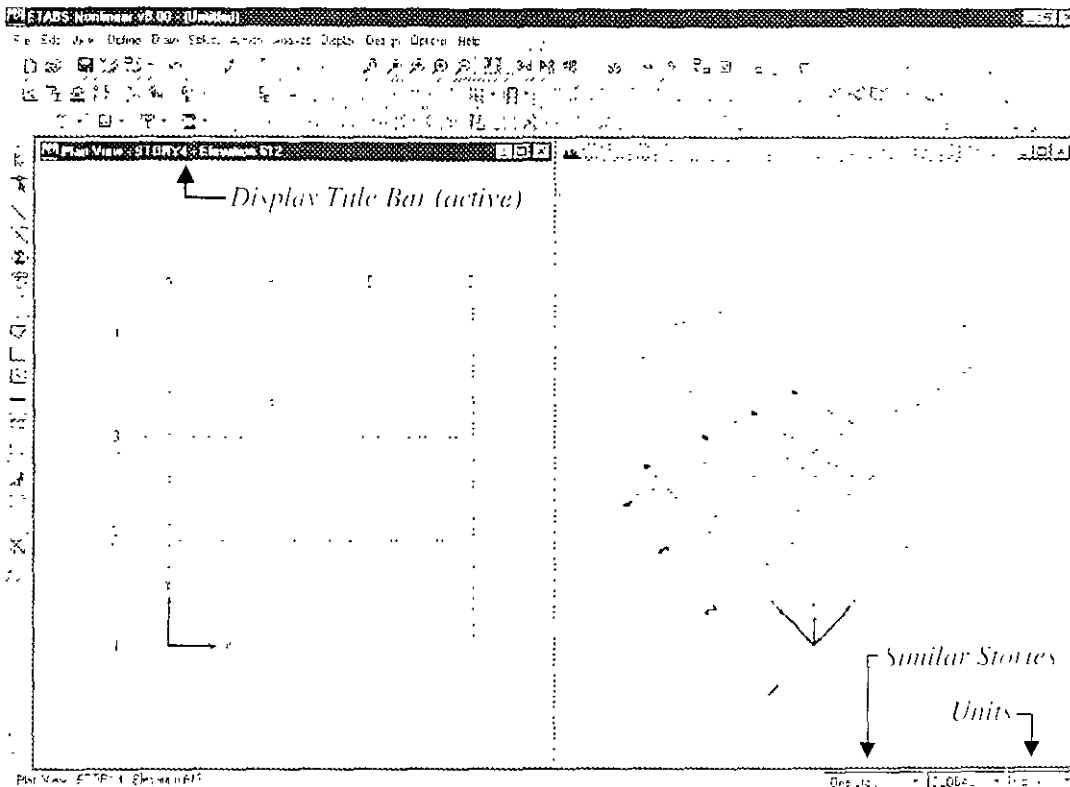
Templates are more fully explained in the File Menu chapter of the graphical user interface reference manual

- C Select the **No** button on that form and the form shown in Figure 3 will display.

The Building Plan Grid System and Story Data form is used to specify horizontal grid line spacing, story data, and, in some cases, template models. Template models provide a quick, easy way of starting your model. They automatically add structural objects with appropriate properties to your model. We highly recommend that you start your models using templates whenever possible. However, in this example, the model is built from scratch, rather than using a template.

- D Set the number of stories in the Number of Stories edit box to 4.
- E Type **15 ft** into the Bottom Story Height edit box and press the Enter key on your keyboard. Notice that the program automatically converts the 15 ft to 180 because the current units are kips and inches (15 feet = 180 inches).
- F Select the **Grid Only** button.

2



**Figure 4**  
The ETABS  
main window

G Click the **OK** button to accept your changes

When you click the **OK** button, your model appears on screen in the main ETABS window with two view windows tiled vertically, a Plan View on the left and a 3-D View on the right, as shown in Figure 4. The number of view windows can be changed using the **Options menu > Windows** command.

Note that the Plan View is active in Figure 4. When the window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window. If you change the views, return to the default described in the previous paragraph, with the Plan View active, before continuing with the next step in this manual.

## Define an Auto Select Section List

An auto select selection list is simply a list of sections, for example, W18X35, W18X40, W21X44, W21X50 and W24X55. Auto select section lists can be assigned to frame members. When an auto select selection list is assigned to a frame member, the program can automatically select the most economical, adequate section from the auto select section list when it is designing the member.

2

The program has several built-in auto select section lists. Some of those lists will be used later in these instructions. Because the architect requested that the beams be no deeper than W18, it is useful to create an auto select section list that contains W16 and W18 beams now

- A. Click the **Define menu > Frame Sections** command, which will display the Define Frame Properties form shown in Figure 5

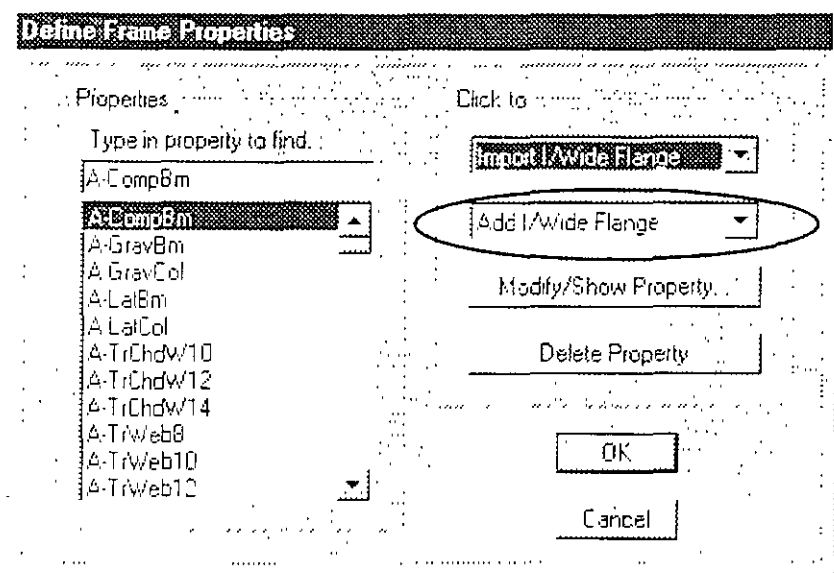
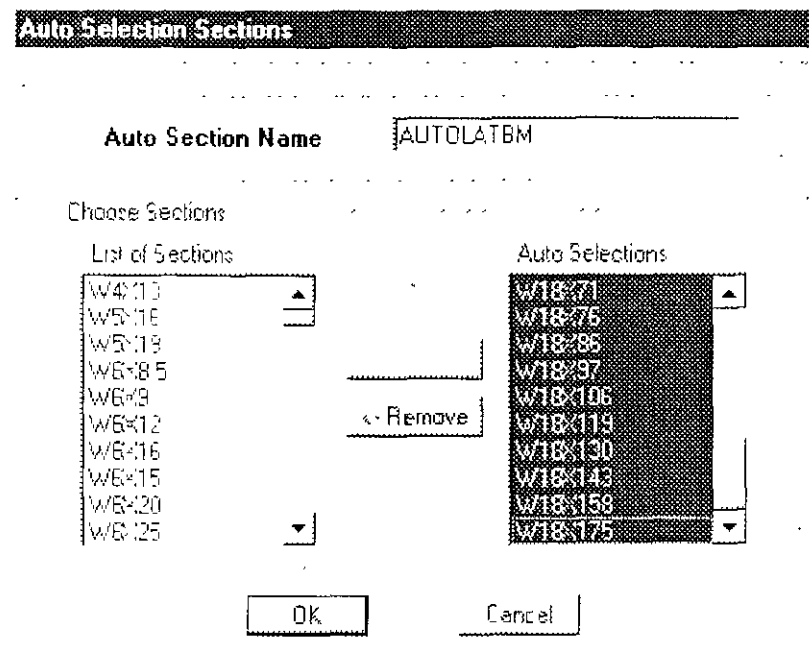


Figure 5  
The Define  
Frame  
Properties  
form

- B. Click the drop-down box that reads "Add I/Wide Flange" in the Click To area of the Define Frame Properties form. Scroll down the resulting list of potential Add sections until you find *Add Auto Select List*. Double click on it. The Auto Selection Sections form shown in Figure 6 appears
- C. Type **AUTOLATBM** in the Auto Section Name edit box.



2



**Figure 6**  
Auto Select  
Sections form

- D Scroll down the list of beam sections in the List of Sections to find the W16X26 beam. Click once on that beam to highlight it.
- E Scroll further down the list of beam sections in the List of Sections to find the W18X175 beam. Press the Shift key on your keyboard and then click once on the W18X175 beam. You should now have all of the beams between the W16X26 and the W18X175, inclusive, highlighted.
- F Click the **Add** button to add the selected beams to the Auto Selections list on the right side of the form.
- G Click the **OK** button and then click the **OK** button in the Define Frame Properties form to accept your changes.

## Step 2 Add Line Objects

In this Step, the program is set up to add objects to multiple stories simultaneously. Then the structural objects are added to the model.

### Set Up to Add Objects to Multiple Stories Simultaneously

Make sure that the Plan View is active. To make a window active, move the cursor, or mouse arrow, over the view and click the left mouse button. When a view is active, the Display Title Bar is highlighted. The location of the Display Title Bar is indicated in Figure 4.



**Note:**

*The Similar Stories feature is only active when you are working in a plan view.*

- A. Click the drop-down box that reads "One Story" at the bottom right of the Main window, which is shown in Figure 4.
- B. Highlight **Similar Stories** in the list. This activates the Similar Stories option for drawing and selecting objects.
- C. To review the current Similar Story definitions, click the **Edit menu > Edit Story Data > Edit Story** command. The Story Data form shown in Figure 7 appears. Note the Master Story and Similar Story columns in the form.

With the Similar Stories option active, as additions or changes are made to a story—for example, Story 4—those additions and changes will also apply to all stories that have been designated as Similar To Story 4 on the Story Data form. By default, the program has defined Story 4 as a Master story and, as shown in Figure 7, Stories 1, 2 and 3 are Similar To Story 4. This means that, with Similar Stories active, any drawing or selection performed on any one story will apply to all of the other stories. A story can be set as Similar To NONE so that additions or changes will not affect it.

- D. We will not make any changes to the form, so press the **Cancel** button to close the form.

2

**Story Data**

	Label	Height	Elevation	Master Story	Similar To	Splice Point	Splice Height
5	STORY4	144	612	Yes		No	0
4	STORY3	144	468	No	STORY4	No	0
3	STORY2	144	324	No	STORY4	No	0
2	STORY1	180	180	No	STORY4	No	0
1	BASE		0				

Reset Selected Row:

Height:

Master Story:

Similar To:

Splice Point:


Splice Height:

Units:

Figure 7  
Story Data form

### Draw Column Objects

Make sure that the Plan View is active.

- A. Click the **Create Columns in Region or at Clicks**  button or use the **Draw menu > Draw Line Objects > Create Columns in Region or at Clicks** command. The Properties of Object pop-up box for columns shown in Figure 8 will appear.

**Properties of Object**

Property	A-LatCol
Angle	90.
Plan Offset X	0
Plan Offset Y	0


Figure 8  
Properties of Object box

If the Properties of Object box is covering any part of the model in either of the displayed views, drag it out of the way. To move the

form, click in the Properties of Object box title bar, hold down the left mouse button, and drag the form out of the way.

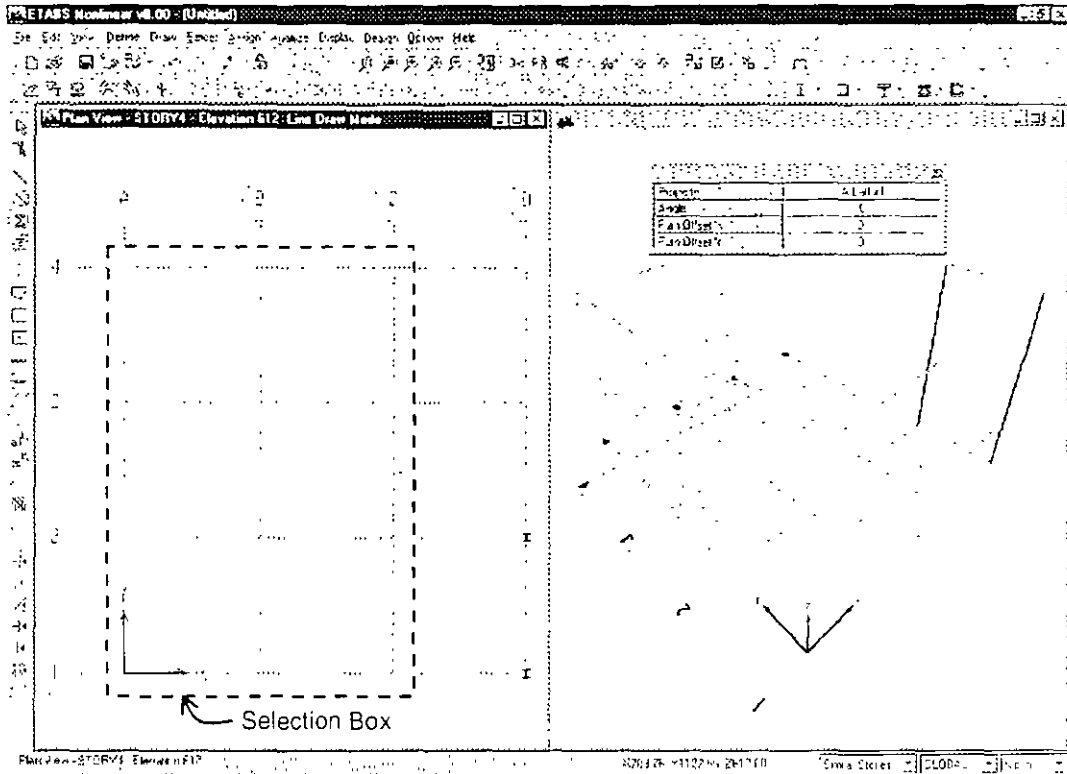
- B. Make sure that the Property item on the Properties of Object form is set to A-LatCol. If it is not, click once in the edit box opposite the Property item to activate the drop-down box and then select A-LatCol from the resulting list. A-LatCol is a built-in auto select section list intended to be used for lateral force resisting columns.

2

If you want to review sections included in A-LatCol, or any of the other auto select section lists, (1) click the **Define menu > Frame Sections** command or click the **Define Frame Sections** button . The Define Frame Properties form will appear. (2) Highlight A-LatCol in the Properties drop down list (3) Click the **Modify/Show Property** button. The Auto Selection Sections form appears, the sections included in the A-LatCol auto select section list are listed in the Auto Selections area of the form. (4) Click the **Cancel** button to close the form.

- C. Click in the Angle edit box on the Properties of Object form and set the angle to 90. This means that the columns will be rotated 90 degrees from their default position.
- D. To draw the first column, left click once in the Plan View at the intersection of grid lines D and 1. An I-shaped column should appear at that point in the Plan View. Also, in the 3D View, note that the column is displayed extending over all story levels even though the column was drawn at only one story level. This occurs because the Similar Stories feature is active. Note that the Similar Stories feature only applies when additions or changes are made to the model in the Plan View. The Similar Stories feature does not apply when additions or changes are made in the Elevation or 3D views.
- E. Click once in the Plan View at the intersection of grid lines D and 2 to draw the second column.
- F. Now change the angle item in the Properties of Object form from 90 to 0.


2



**Figure 9**  
*Drawing column objects in a windowed region*

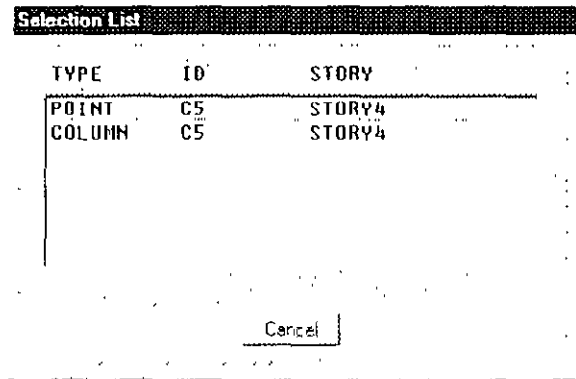
- G Now draw the remaining columns in one action by "windowing" around the grid intersections as shown in Figure 9 To "window," click the left mouse button above and to the left of grid intersection A-4 and then, while holding the left mouse button down, drag the mouse until it is below and to the right of grid intersection C-1. A selection box similar to that shown in Figure 9 should expand around the grid line intersections as the mouse is dragged across the model. Release the left mouse button and the program will draw the column objects at the grid line intersections.

Note that these columns appear rotated 90 degrees from the first two.

- H. Click the **Select Object** button, , to change the program from Draw mode to Select mode.
- I. Hold down the Ctrl key on your keyboard and left click once in the Plan View on column A-2. A selection list similar to the one shown

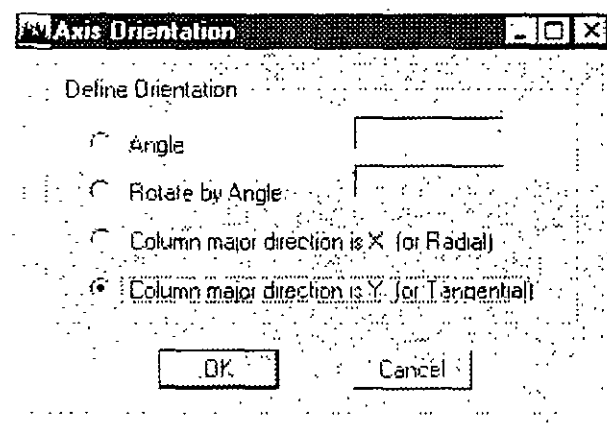
in Figure 10 pops up because multiple objects exist at the location that was clicked. In this example, a point object and a column object exist at the same location. Note that the selection list will only appear when the Ctrl key is used with the left click.

Figure 10  
Selection List  
form



- J. Select the column from the list by clicking on it. The column at A-2 is now selected. It is selected over its entire height because the Similar Stories feature is active. Note that the status bar in the bottom right-hand corner of the main ETABS window indicates that 4 lines have been selected.
- K. Repeat the selection process at B-2, A-3, C-3 and C-4. The status bar should indicate that 20 lines have been selected.
- L. Click the **Assign menu > Frame/Line > Local Axes** command to bring up the form shown in Figure 11.

Figure 11  
Axis Orientation  
form for  
columns





**Note:**

When the local axes are displayed, the color coded arrows are red, white and blue, corresponding to the 1, 2 and 3 axes respectively, always

**2**

- M Click the Column Major Direction is Y option in the form and then click the **OK** button. The selected columns rotate 90 degrees

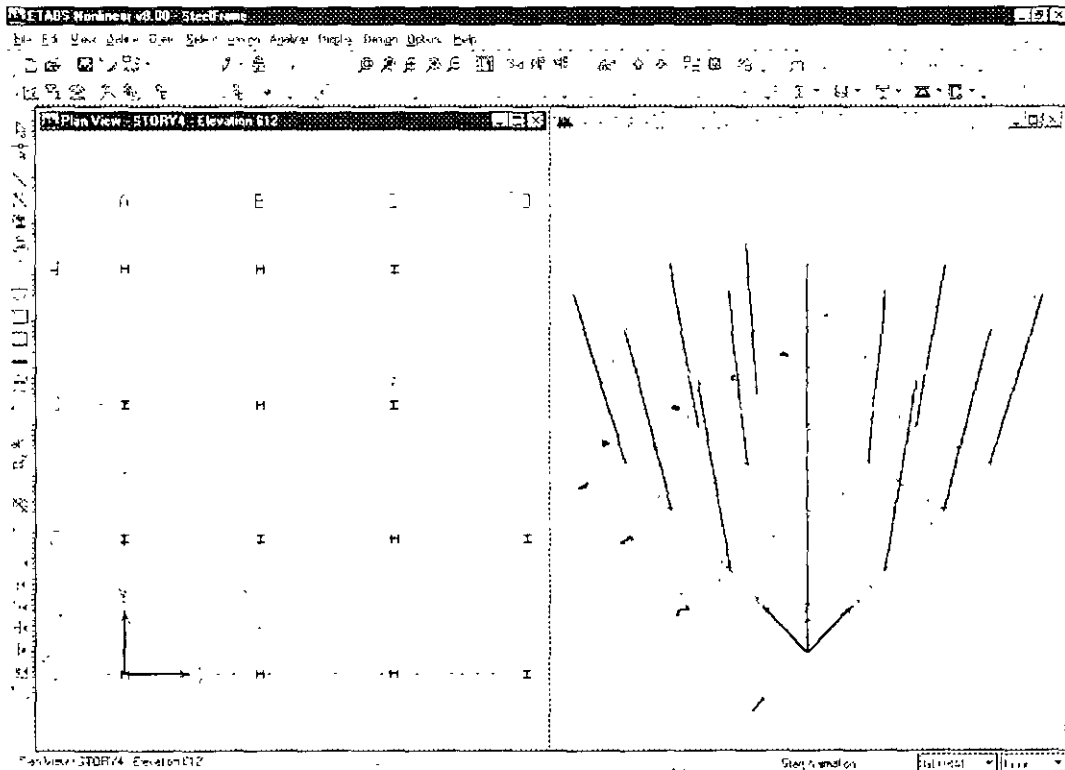
Notice the colored arrows associated with each column. Those arrows indicate the local axes directions. The red arrow is always in the local 1 direction, the white arrow is in the local 2 direction and the blue arrow is in the local 3 direction. Currently, the red arrow is not visible because it (and thus the column local 1-axis) is perpendicular to the screen

An easy way to remember the color coding for the local axes is to think of the American flag. The American flag is red, white and blue. The color coding for the local axes is red = 1, white = 2 and blue = 3.

**Figure 12**  
The example model with the columns drawn

Click the **Assign menu > Clear Display of Assigns** command to clear the display of the arrows

The model should now appear as shown in Figure 12






*Note:*  
*'Save your model often'*

## Save the Model


During development, save the model often. Although typically you will save it with the same name, thus overwriting previous models, you may occasionally want to save your model with a different name. This allows you to keep a record of your model at various stages of development.

- A Click the **File menu > Save** command, or the **Save** button, , to save your model. Specify the directory in which you want to save the model and, for this example, specify the file name SteelFrame

2

## Draw the Lateral Force-Resisting Beam Objects

Make sure that the Plan View is active. Draw the beams between the columns using the following Action Items.

- A Click the **Create Lines in Region or at Clicks** button, , or the **Draw menu > Draw Line Objects > Create Lines in Region or at Clicks** command. The Properties of Object pop-up box for line objects shown in Figure 13 will appear.

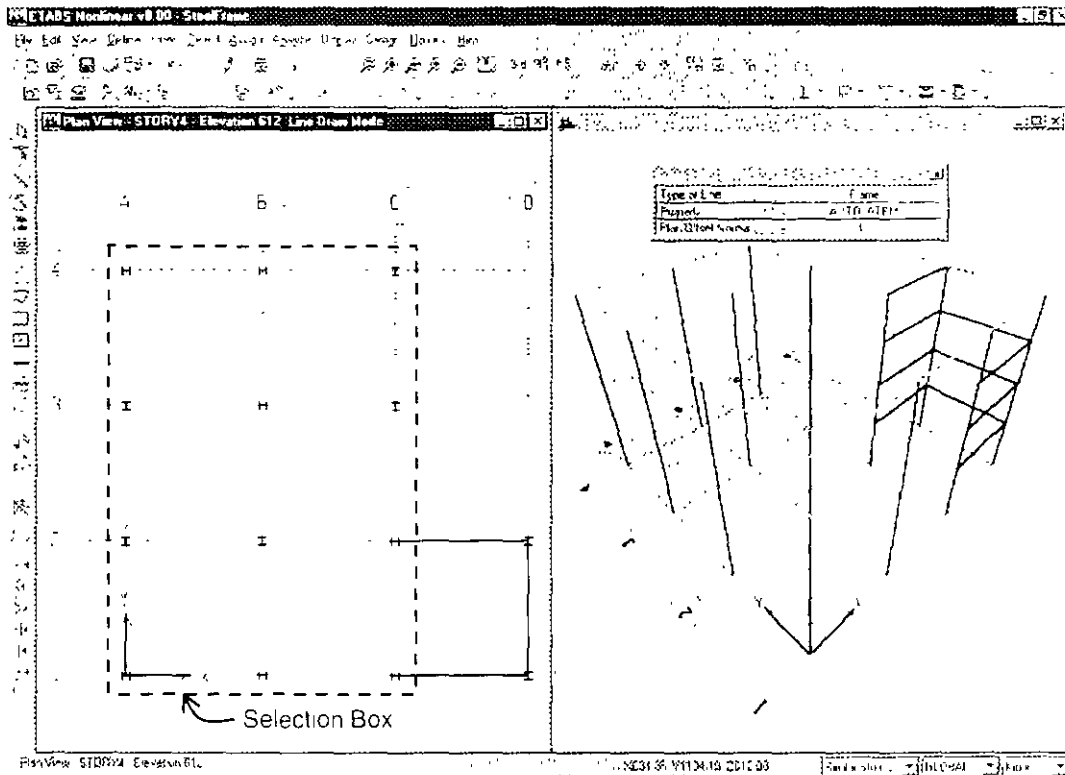
*Figure 13*  
*Properties of*  
*Object box*

Properties of Object	
Type of Line	Frame
Property	AUTOLATBM
Plan Offset Normal	0


- B Click once in the edit box opposite the Property item to activate the drop-down box and then scroll down to select AUTOLATBM in the resulting list. Recall that AUTOLATBM is the auto select section list created in Step 1
- C Left click once in the Plan View on grid line D between grid lines 1 and 2. A beam is drawn along the selected grid line. Because the Similar Stories option is active, beams are created at all levels.
- D In a similar manner, left click once on grid line 1 between grid lines C and D and then left click once on grid line 2 between grid lines C and D to draw beams in two more locations.




2




**Figure 14**  
Drawing lateral force-resisting beam objects in a windowed region

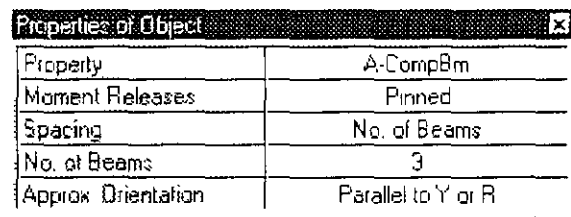
- E Now draw the remaining lateral force-resisting beams in one action by windowing around the grid lines to add beams between the columns drawn in Step 1, as shown in Figure 14. To window, click the left mouse button above and to the left of grid intersection A-4 and then, while holding the left mouse button down, drag the mouse until it is below and to the right of grid intersection C-1. A selection box will expand around the grid line intersections as the mouse is dragged across the model. Release the left mouse button to draw the beams.
- F. Click the **Select Object** button, , to change the program from Draw mode to Select mode.
- G Left click once on the beam along grid line C between grid lines 2 and 3 to select it. Press the Delete key on your keyboard or click the **Edit menu > Delete** command to delete the selection because no beams should connect points C-3 and C-2 in the model.

- H Click the **File menu** > **Save** command, or the Save button, , to save your model.

### Draw the Secondary (Infill) Beam Objects

Make sure that the Plan View is active. Now draw the secondary beams that span between girders using the following Action Items.

- A. Click the **Create Secondary Beams in Region or at Clicks** button, , or the **Draw menu** > **Draw Lines Objects** > **Create Secondary Beams in Region or at Clicks** command. The Properties of Object pop-up box for beams shown in Figure 15 will appear.



Property	
Property	A-CompBm
Moment Releases	Pinned
Spacing	No. of Beams
No. of Beams	3
Approx. Orientation	Parallel to Y or R

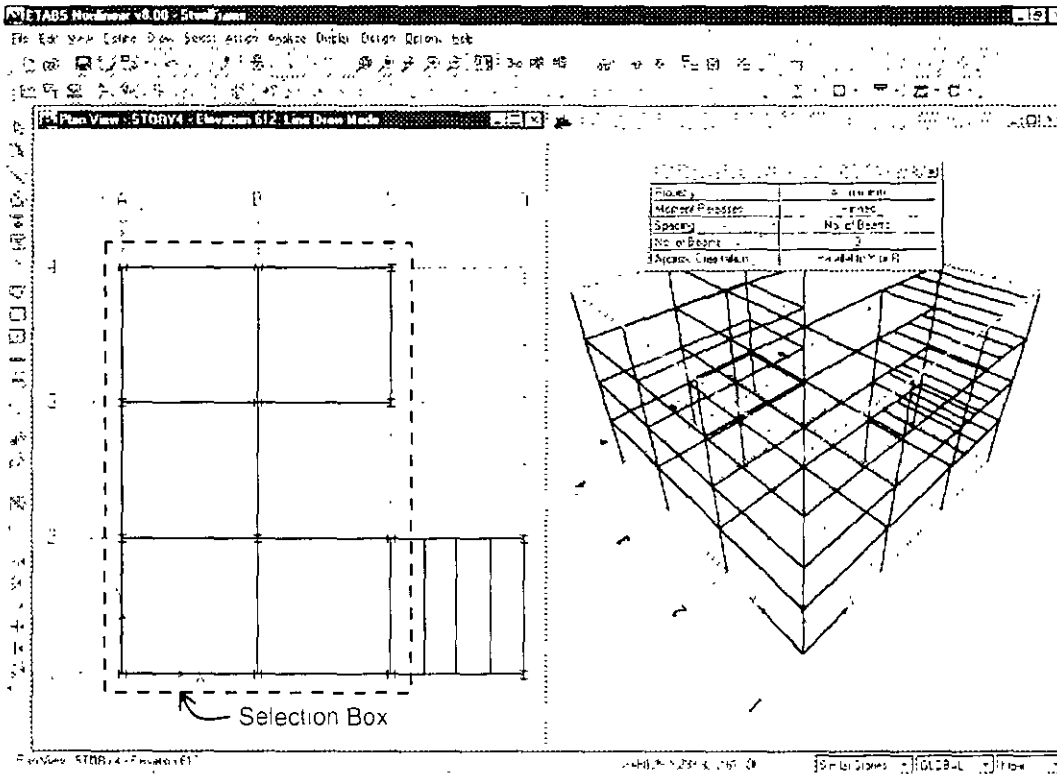
Figure 15  
Properties of  
Object box

Make sure that the Property item in this box is set to A-CompBm. If it is not, click once in the edit box opposite the Property item to activate the drop-down box and then select A-CompBm from the resulting list. A-CompBm is a built-in auto select section list intended to be used for composite secondary beams. To review the sections included in the A-CompBm auto select list, (1) click the **Define menu** > **Frame Sections** command. (2) Highlight A-CompBm in the drop down list. (3) Click the **Modify/Show Property** button. (4) When finished, click the **Cancel** button to close the form.

Make sure that the Approx. Orientation item in the Properties of Object box is set to Parallel to Y or R.



- B. Left click once in the bay bounded by grid lines C, D, 1 and 2 to draw the first set of secondary beams.
- C. Draw the remaining secondary beams in one action by windowing around the bays where secondary beams are to be added, as shown in Figure 16. To window, click the left mouse button above and to the left of grid intersection A-4 and then, while holding the left mouse

2



**Figure 16**  
Drawing secondary beam objects in a windowed region

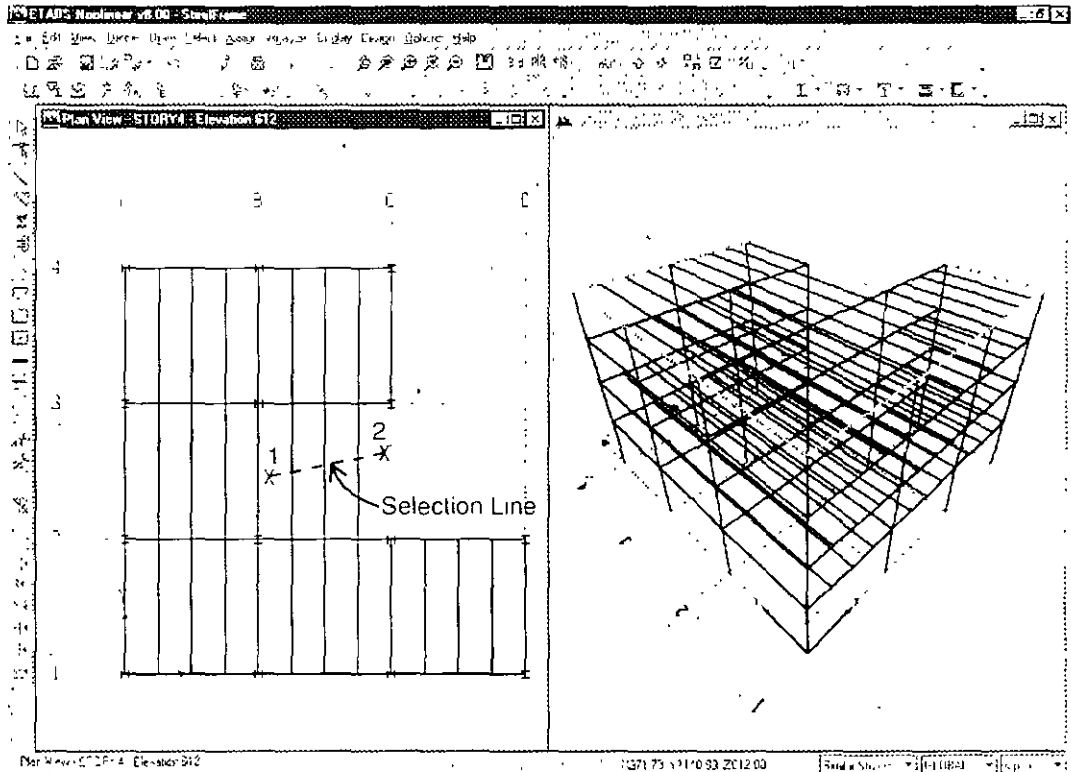
button down, drag the mouse until it is below and to the right of grid intersection C-1. A selection box similar to that shown in Figure 16 will expand as the mouse is dragged across the model. Release the left mouse button to draw the secondary beam objects.


- D. Click the **Select Object** button, , to change the program from Draw mode to Select mode
- E. Click the **Select Using Intersecting Line** button, , or click the **Select menu > Using Intersecting Line** command to put the program in intersecting line select mode.

In intersecting line selection mode, left click the mouse once to start a line. Then, while holding the left mouse button down, drag the mouse to another location, thus creating a selection line. When the left mouse button is released, all objects that are crossed by the selection line are selected.

- F. Refer to Figure 17. Left click the mouse in the Plan View between grid lines 2 and 3 just to the right of grid line B at the point labeled 1 in the figure. Holding down the left mouse button, drag the mouse pointer to the point labeled 2 in the figure. The selection line should be crossing the unwanted secondary beams in the bay bounded by grid lines 2, 3, B and C. Release the left mouse button to select the beams.

*Figure 17*  
Selection using an intersecting line




- G. Press the Delete key on your keyboard or click the **Edit menu > Delete** command to delete the selected beams from the model.
- H. Click the **File menu > Save** command, or the Save button, , to save your model.

## Step 3 Add Area Objects

In this Step, floors are added to the model and a "dummy" area is created to which wind load can be assigned in Step 7.

### Draw the Floor Area Objects

Make sure that the Plan View is active. Now draw an area object to represent the floor using the following Action Items.

- A. Click the **Draw Areas** button, , or select the **Draw menu > Draw Area Objects > Draw Areas** command. The Properties of Object pop-up box for areas shown in Figure 18 will appear

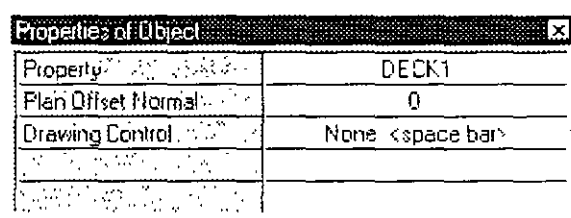



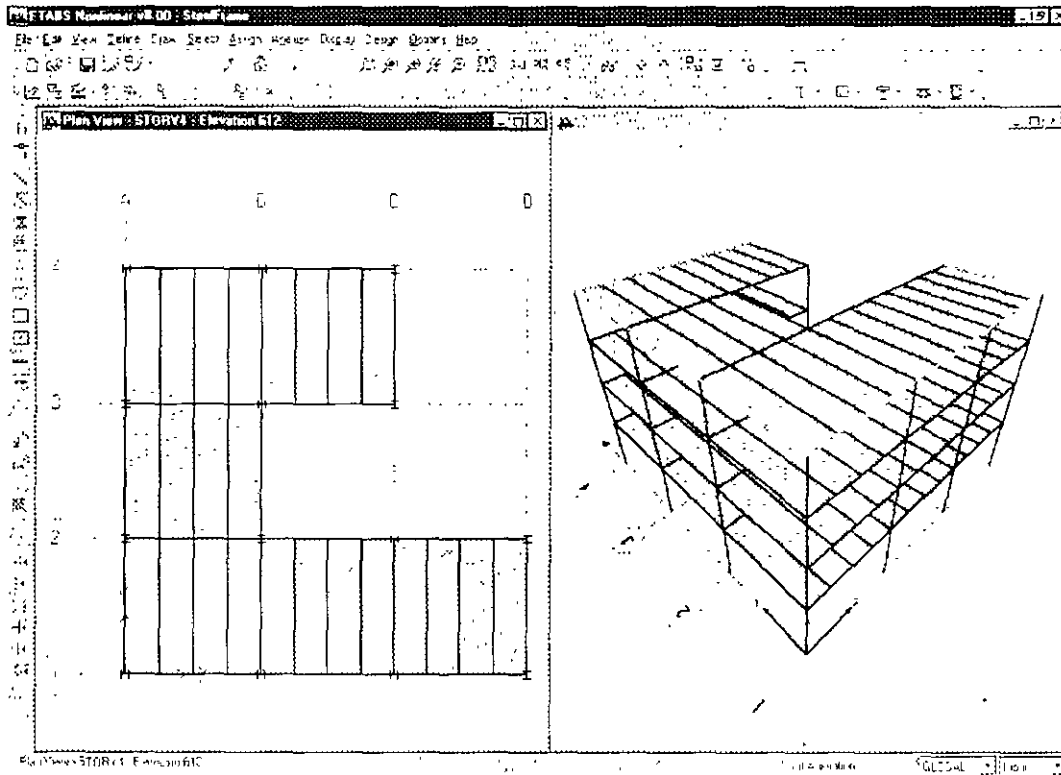
Figure 18  
Properties of  
Object box

Make sure that the Property item in this box is set to Deck1. If it is not, click once in the edit box opposite the Property item to activate the drop-down box and then select Deck1 in the resulting list. Deck1 is a built-in deck section property. The deck properties are reviewed in a subsequent Action Item of this step.

- B. Check that the **Snap to Grid Intersections and Points** command is active. This will assist in accurately drawing the area object. This command is active when its associated button  is depressed. Alternatively, use the **Draw menu > Snap To > Grid Intersections and Points** command to ensure that this command is active. By default, this command is active.




2



**Figure 20**  
Model after  
the floor area  
objects have  
been added

The model now appears as shown in Figure 20

- F. Review the Deck1 property that was assigned to the deck section. Click the **Define menu > Wall/Slab/Deck Sections** command to bring up the Define Wall/Slab/Deck Sections form.
  1. Highlight the Deck1 section and click the **Modify/Show** button. The Deck Section form shown in Figure 21 appears.
  2. Set the Slab Depth (tc) item to 3 to indicate that the slab depth above the metal deck is 3 inches
  3. Click the **OK** button and then click the **OK** button in the Define Wall/Slab/Deck Sections form to accept your changes
- G. Click the **File menu > Save** command, or the Save button, , to save your model

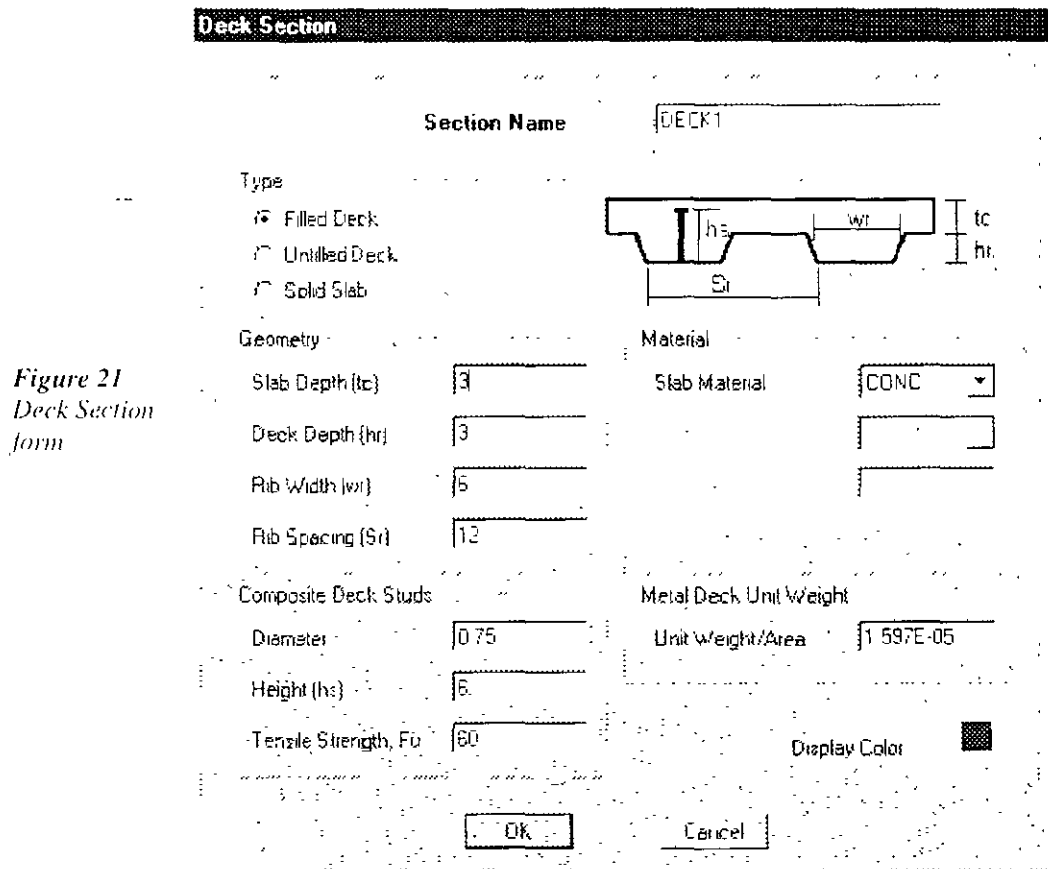



Figure 21  
Deck Section  
form

### Add Dummy Area Objects used for Wind Load Application

Some "dummy" area objects that have no mass or stiffness will be added to the model. The areas will be used in Step 7 to apply wind load to the building.


#### Add Objects in Elevation View

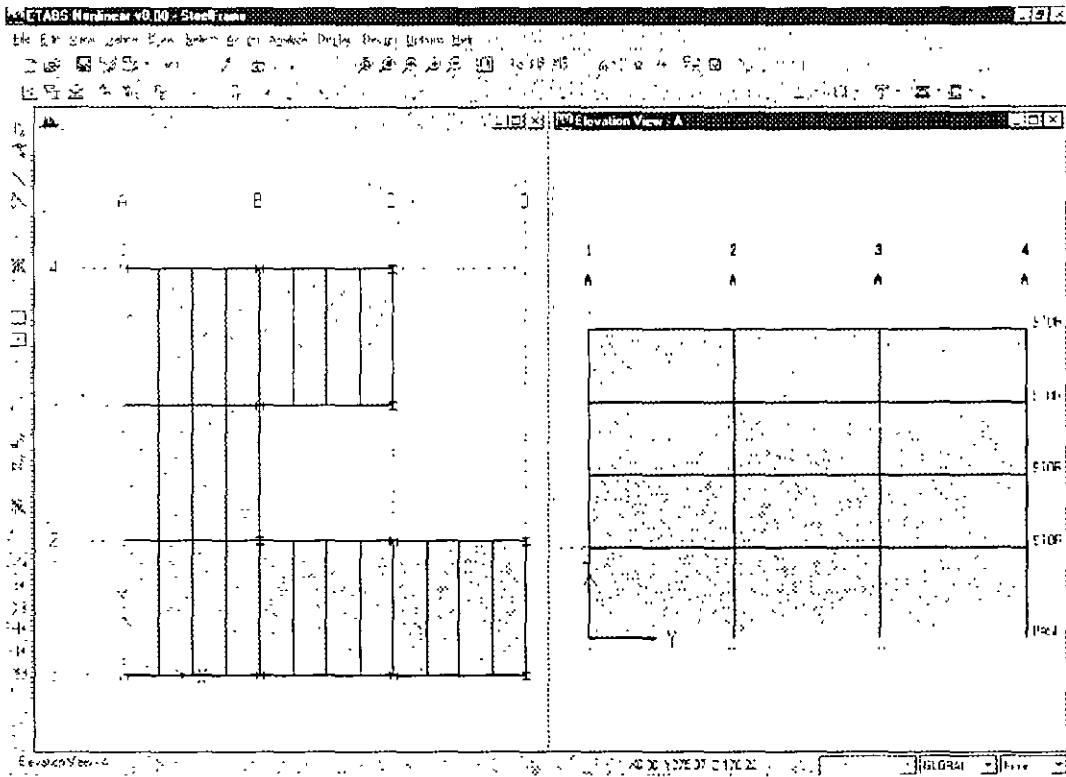
- A. Set the 3D View active by clicking in it. The view is active when its title bar is highlighted.
- B. Click the **Elevation View** button  and select A (i.e., grid line A) from the Set Elevation View form. click the **OK** button. The 3D View changes to an Elevation View of grid line A.



2



**Figure 22**  
*Model after dummy area objects have been added in Elevation View along line A*

- C. Click the **Create Areas at Click** button, , or click the **Draw menu > Draw Area Objects > Create Areas at Click** command. A Properties of Object box for area objects will appear. Click the Property edit box and select NONE from the resulting drop-down box.
- D. Click once in each of the bays shown in this elevation view to add the dummy area elements. Figure 22 shows the model with the dummy wall-type objects that have no mass or stiffness added along line A.

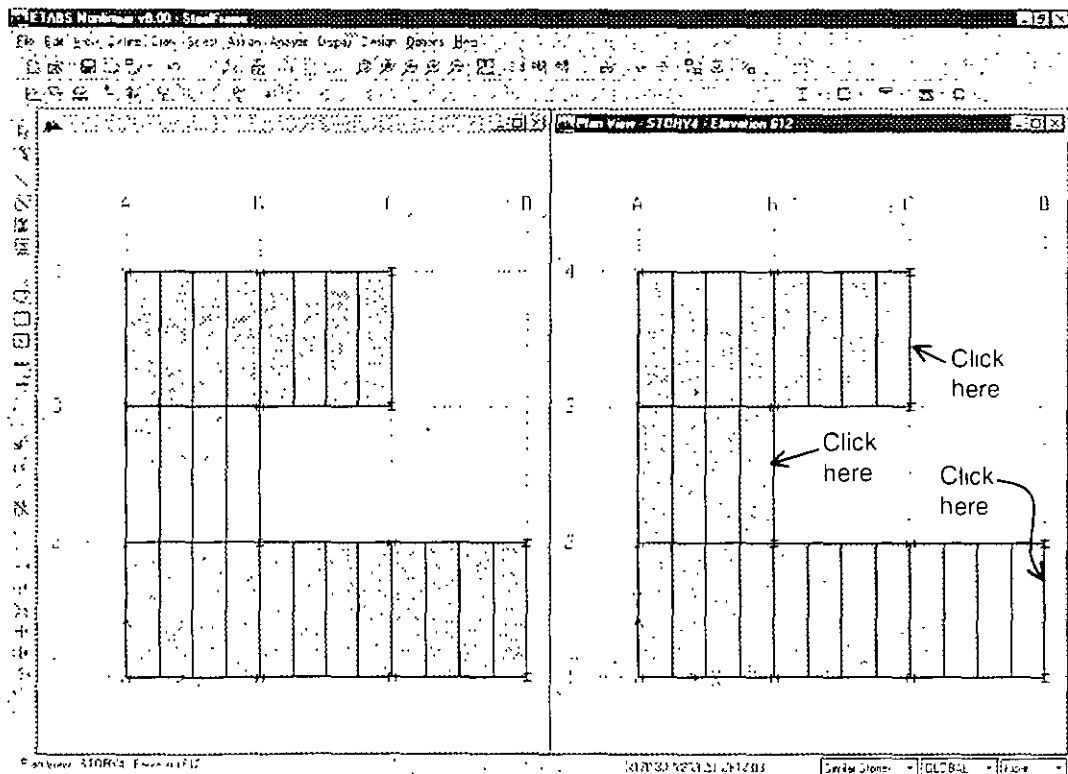


**Add Objects in Plan View**




- A. Make sure the Elevation View is active. The view is active when its title bar is highlighted

- B. Click the Plan View button  and select Story 4 from the Select Plan Level form.
- C. Click the **Create Walls in Region or at Click** button, , or click the **Draw menu > Draw Area Objects > Create Walls in Region or at Clicks** command. A Properties of Object box will appear for area objects. Click the Property edit box and select NONE from the resulting drop-down box.
- D. Click once on grid line C between grid lines 3 and 4; click once on grid line B between grid lines 2 and 3, and click once on grid line D between grid lines 1 and 2, as shown in Figure 23. Dummy-type walls with no stiffness or mass are added to the model at all levels because the Similar Stories feature is active and the command was executed in Plan View.

**Figure 23**  
Adding  
dummy wall-  
type objects  
in Plan View

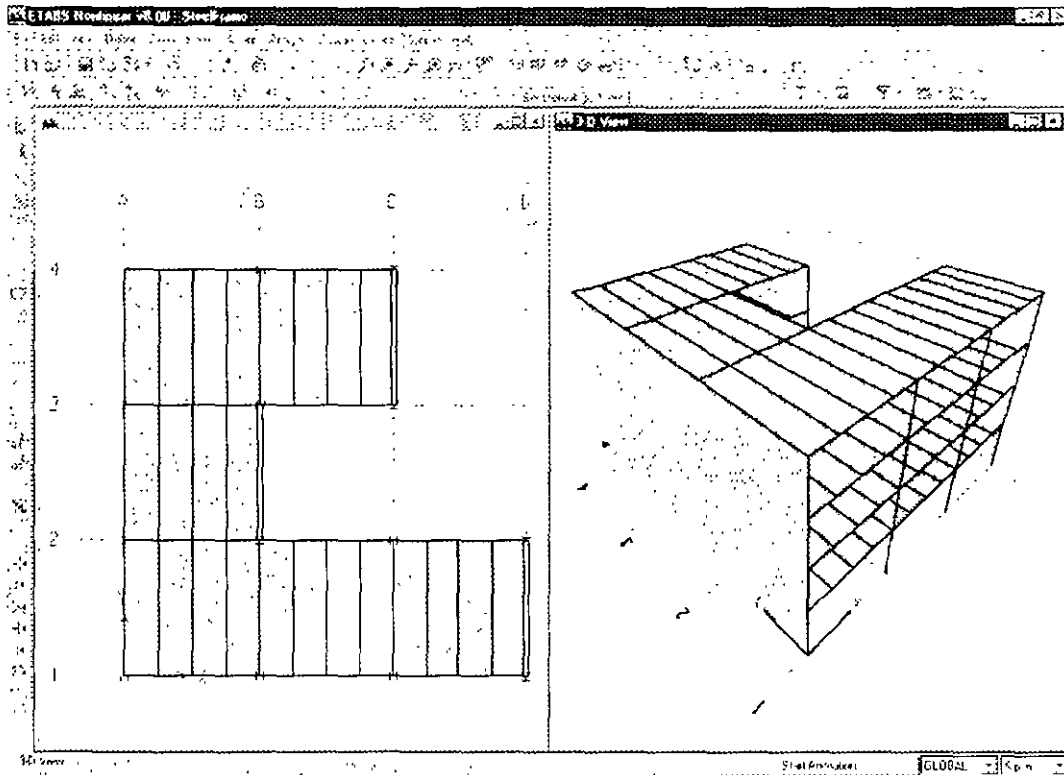


2

- E. Click the **Select Object** button, , to change the program from Draw mode to Select mode.
- F. Make sure the right-hand Plan View is active. Click on the Set Default 3D View button, , to change the Plan View to a 3D View
- G. Click the **File menu** > **Save** command, or the Save button, , to save your model.

*Figure 24 Model after all dummy wall-type objects have been added*

Your model now appears as shown in Figure 24.



## Step 4 Define Static Load Cases

The static loads used in this example consist of the dead, live, earthquake and wind loads acting on the building.

For this example building assume that the dead load consists of the self weight of the building structure, plus 35 psf (pounds per square foot) additional dead load applied to the floors and 250 plf (pounds per linear foot) additional dead load applied to the beams around the perimeter of the building. The 35 psf additional dead load applied to the floors accounts for items such as partitions, ceiling, mechanical ductwork, electrical items, plumbing, and so forth. The 250 plf additional dead load around the perimeter accounts for the cladding.

The live load is taken to be 100 psf at each story level. This live load is reducible for steel frame and composite beam design.




**Note:**

*There is no limit to the number of static load cases that can be defined in ETABS.*

Note that realistically those loads would probably vary at some of the different floor levels. However, for the purposes of this example, we have chosen to apply the same load to each story level.

This example also applies a UBC97 static earthquake load to the building and an ASCE 7-98 wind load to the building. The forces that are applied to the building to account for the earthquake and wind load are automatically calculated by the program.

- A. Click the **Define menu > Static Load Cases** command or click the **Define Static Load Cases** button,  to bring up the Define Static Load Cases form shown in Figure 25. Note that there are two default load cases defined. They are DEAD, which is a dead load case, and LIVE, which is a live load case.

Note that the self weight multiplier is set to 1 for the DEAD case. This indicates that this load case will automatically include 1.0 times the self weight of all members.

- B. Click on LIVE to highlight the row as shown in Figure 25. Select REDUCE LIVE from the Type drop-down menu. Click the **Modify**

2

Figure 25  
Define Static  
Load Case  
Names form

Load	Type	Self Weight Multiplier	auto Lateral Load
LIVE	LIVE	1	
DEAD	DEAD	1	
LIVE	LIVE	0	

Buttons: Add New Load, Modify Load, Delete Load, OK, Cancel

**Load** button to change the load type from live to reducible live. We will apply live load to the structure later.

- C. Click in the edit box for the Load column. Type the name of the new load; in this case, type **SDEAD**. Select a Type of load from the Type pull down menu, in this case, select **SUPERDEAD**. Make sure that the self weight multiplier is set to zero. Self weight should be included in only one load case; otherwise, self weight might be double counted in the analysis. In this example, self weight has been assigned to the **DEAD** load case. Click the **Add New Load** button to add the **SDEAD** load to the Load list.
- D. Repeat Action Item C to add a **SUPERDEAD**-type load named **CLADDING**. We will apply superimposed dead load to the structure later.
- E. To define the UBC97 earthquake load, click in the edit box for the Load column again and type **EQY**. Select **QUAKE** for the Load Type. Make sure the Self Weight Multiplier is zero. Use the Lateral Load pull-down menu to select **UBC97**, with this option selected, ETABS will automatically apply static earthquake load based on the 1997 UBC code requirements. Click the **Add New Load** button.
- F. With the **EQY** load highlighted, click the **Modify Lateral Load** button. This will bring up the 1997 UBC Seismic Loading form (the UBC97 form comes up because the Auto Lateral Load type was set to UBC97 in item E). On this form, click the **Y Dir** option at the top.

**1997 UBC Seismic Loading**

Direction and Eccentricity  
 X Dir  
 Y Dir  
 Z Dir  
 Dir + Eccen Y  
 Y Dir + Eccen X  
 X Dir + Eccen Y  
 Y Dir + Eccen X

Time Period  
 Method  
 Program Calc  $\Omega (R) = 0.035$   
 User Defined

Soil Profile Type: SC  
 Seismic Zone Factor: U-40

Near Source Factor  
 Per Code  
 User Defined

Seismic Source Type: B  
 Dist. to Source (km): 15

Other Factors  
 Importance Factor I: 1

Story Range  
 Top Story: STORY4  
 Bottom Story: BASE

Factors  
 Overstrength Factor, R: 8.5

OK Cancel

Figure 26  
1997 UBC  
Seismic  
Loading form

2

of the form, as shown in Figure 26. Click the **OK** button. The Define Static Load Case Names form reappears.

- G. To define the ASCE7-98 wind load, click in the edit box for the Load column again and type WINDX. Select WIND as the Type. Select ASCE7-98 from the Lateral Load pull down menu. Click the **Add New Load** button.
- H. With the WINDX load highlighted, click the **Modify Lateral Load** button. This will bring up the ASCE 7-98 Wind Loading form shown in Figure 27 (the ASCE 7-98 form comes up because the Auto Lateral Load type was set to ASCE 7-98 in item G.) Select the Exposure from Area Objects option. Notice that the form changes.

The Exposure from Area Objects option means that the wind load is being defined from user-specified wind pressure coefficients applied to the dummy vertical elements that were drawn earlier in Step 3.

2


Figure 27  
ASCE 7-98  
Wind Loading  
form

Type 100 into the edit box for Wind Speed, as shown in Figure 27 and then click the **OK** button. The Define Static Load Case Names form reappears.

The Define Static Load Case Names form should now appear as shown in Figure 28. Click the **OK** button in that form to accept all of the newly defined static load cases.

Figure 28  
The Define  
Static Load  
Case Names  
form after all  
static load  
cases have  
been defined


Load	Type	Self Weight Multiplier	Auto Lateral Load
WIND	WIND	1	ASCE 7-98
DEAD	DEAD	0	
LIVE	REDUCE LIVE	0	
SDEAD	SUPER DEAD	0	
CLADDING	SUPER DEAD	0	
EQU	DUAKE	0	UBC 97
WIND	WIND	1	ASCE 7-98

- I. Click the **File menu > Save** command, or the **Save** button, , to save your model


## Step 5 Assign Gravity Loads

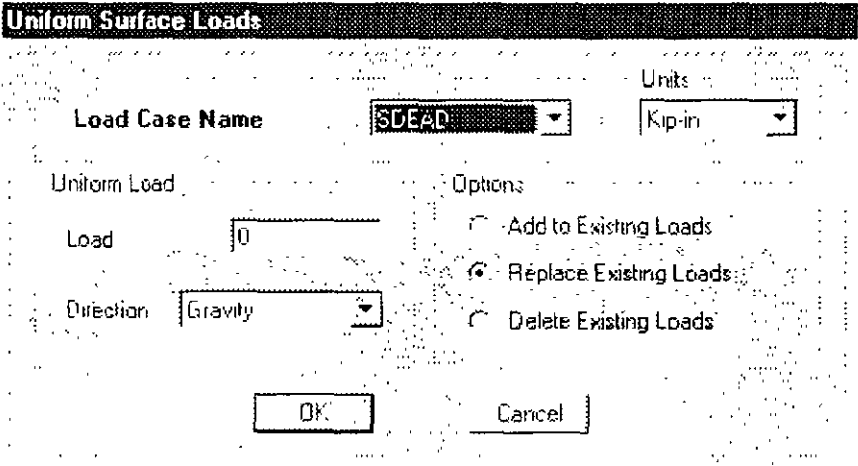
2

In this Step, the superimposed dead and live gravity loads will be applied to the model. Make sure that the Similar Stories feature is enabled and that the Plan View is active.

- A. Click anywhere on the deck (but not on a beam) to select the deck. A dashed line should appear around the perimeter of the deck. This dashed line indicates that the deck has been selected. If you make a mistake in selecting, press the **Clear Selection** button, , and try again.

The status bar in the lower left-hand corner of the Main ETABS window should indicate that four area objects have been selected because the Similar Stories feature is active.

- B. Click the **Assign > Shell/Area Loads > Uniform** command or click the **Assign Uniform Load** button, . This brings up the Uniform Surface Loads form. Select SDEAD from the Load Case Name drop-down box, as shown in Figure 29.



The screenshot shows the 'Uniform Surface Loads' dialog box. It has a title bar 'Uniform Surface Loads'. Inside, there are several fields and options:

- Load Case Name:** A dropdown menu with 'SDEAD' selected.
- Units:** A dropdown menu with 'Kip-in' selected.
- Uniform Load:** A text box containing '0'.
- Direction:** A dropdown menu with 'Gravity' selected.
- Options:** Three radio buttons:
  - Add to Existing Loads
  - Replace Existing Loads
  - Delete Existing Loads
- At the bottom, there are two buttons: 'OK' and 'Cancel'.

*Figure 29  
The Uniform  
Surface Loads  
form*



2

Note that the Direction specified for the load is Gravity. The gravity load direction is downward; that is, in the negative Global Z direction.

1. Hold down the Shift key and double click in the Load field to display the Calculator form, shown in Figure 30. This built-in calculator has a number of functions that are useful for assigning loads. In this case, it will be used to convert units and assign the superimposed dead load in lb-ft.

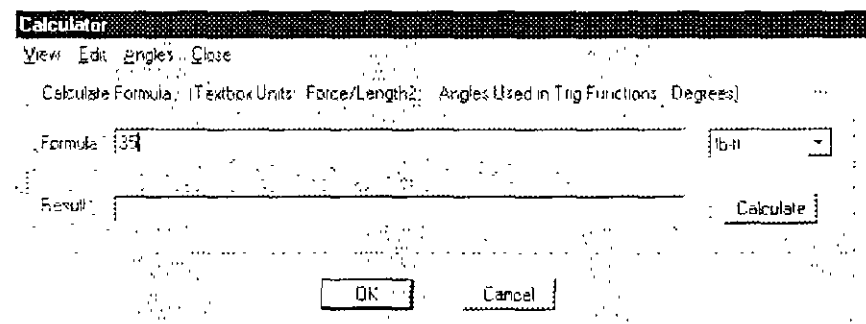



Figure 30  
The Calculator form

Note that the calculator displays that the Load text box has units of Force over Length squared (Force/Length<sup>2</sup>)

Select lb-ft from the drop down list in the Calculator form and then type **35** in the Formula edit box. Be sure to set the units before typing 35.

Click the **OK** button on the Calculator form; ETABS automatically converts the lb-ft input into Kip-inch, displaying the result, 2.43055555555556E-04 kips/in<sup>2</sup>, on the Uniform Surface Load form.

2. Click the **OK** button on the Uniform Surface Load form to accept the superimposed dead load.
- C. Click anywhere on the deck (but not on a beam) to select the deck.
- D. Click the **Assign > Shell/Area Loads > Uniform** command or click the **Assign Uniform Load** button, . This brings up the Uniform


Surface Loads form. Select LIVE from the Load Case Name drop-down box

1. Set the units drop-down box in the form to lb-ft and then enter 100 in the Load edit box. The Uniform Surface Loads form should now appear as shown in Figure 31.

2

**Figure 31**  
The Uniform  
Surface Loads  
form

2. Click the **OK** button on the Uniform Surface Load form to accept the live load

- E. Check that the **Snap to Grid Intersections and Points** command is **not** active. This will make it easier to select the perimeter beams. This command is active when its associated button  is depressed. Thus, make sure the button is *not* depressed. You can also toggle the snap feature using the **Draw menu > Snap To > Grid Intersections and Points** command.




**Note:**

*We strongly recommend that you apply perimeter cladding loads to the spandrel beams objects, not to the deck objects*

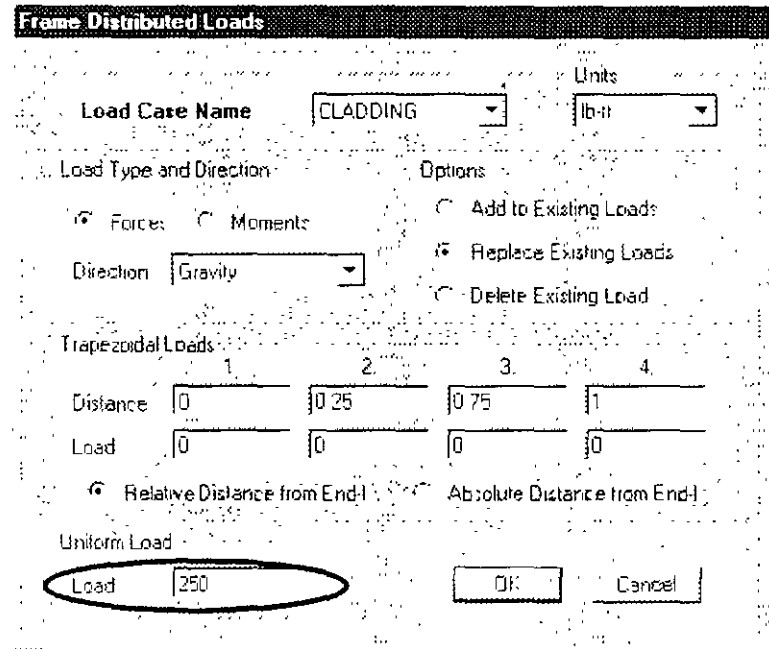
- F. Select the perimeter beam along grid line A between grid lines 1 and 2 by left clicking on it once in Plan View. Notice that the status bar in the lower left-hand corner of the main ETABS window indicates that four lines have been selected because the Similar Stories feature is active. Also note that the selected lines appear dashed.
- G. Select the other thirteen perimeter beams in a similar manner. When you have selected all perimeter beams, the status bar should indicate

that 56 lines have been selected (14 beams times 4 stories = 56 beams).


**2**

- H Click the **Assign > Frame/Line Loads > Distributed** command or click the **Assign Frame Distributed Load** button, . This brings up the Frame Distributed Loads form shown in Figure 32. Select CLADDING from the Load Case Name drop-down box

*Figure 32  
The Frame  
Distributed  
Loads form*



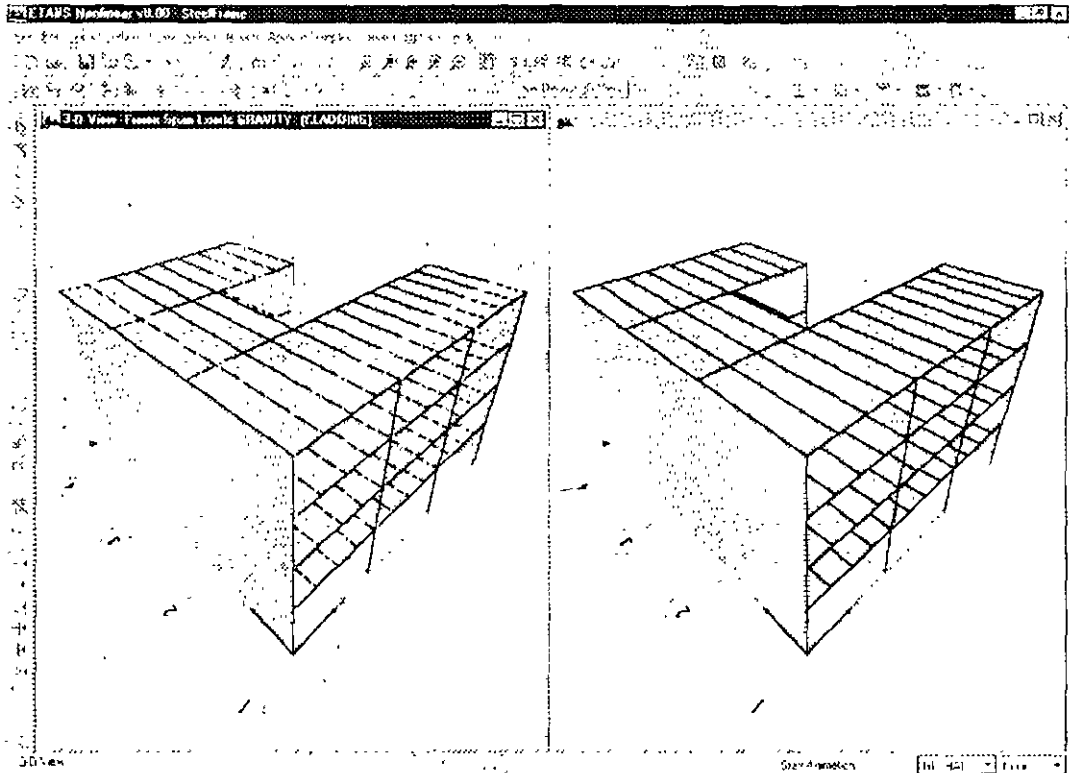
1. Set the units drop-down box in the form to lb-ft and then enter 250 in the Load edit box that is located in the Uniform Load area of the form.
2. Click the **OK** button on the Frame Distributed Loads form to accept the uniform superimposed dead load that is applied to the perimeter beams to represent the cladding

Note that the Frame Distributed Loads form also has a Delete Existing Loads check box. To delete a load assignment, select the beam(s) and use the **Assign > Frame/Line Loads > Distributed** command or the **Assign Frame Distributed Load** button,  to access this form. In the Load Case Name drop-down box, locate the load to be re-



moved, check the Delete Existing Loads check box and click the **OK** button.

- I Make sure the Plan View is active. Click on the Set Default 3D View button, 3-d, to change the Plan View to a 3D View. You should now be able to graphically see the load applied to the perimeter beams, as illustrated in Figure 33.

2



*Figure 33*  
Frame distributed loads applied to the perimeter beams

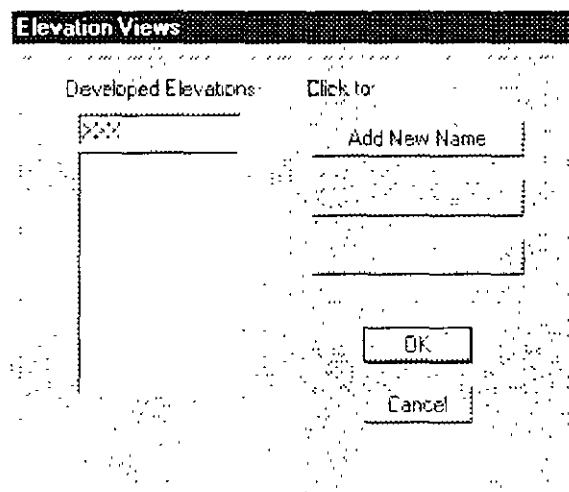
- J Click the **Assign menu > Clear Display of Assigns** command to clear the display of the assigned loads.
- K. Make sure the left-hand 3D view is active. Click the Plan View button  and select Story 4 from the Select Plan Level form.
- L Click the **File menu > Save** command, or the **Save** button, , to save your model.

## Step 6 Define a Developed Elevation


2

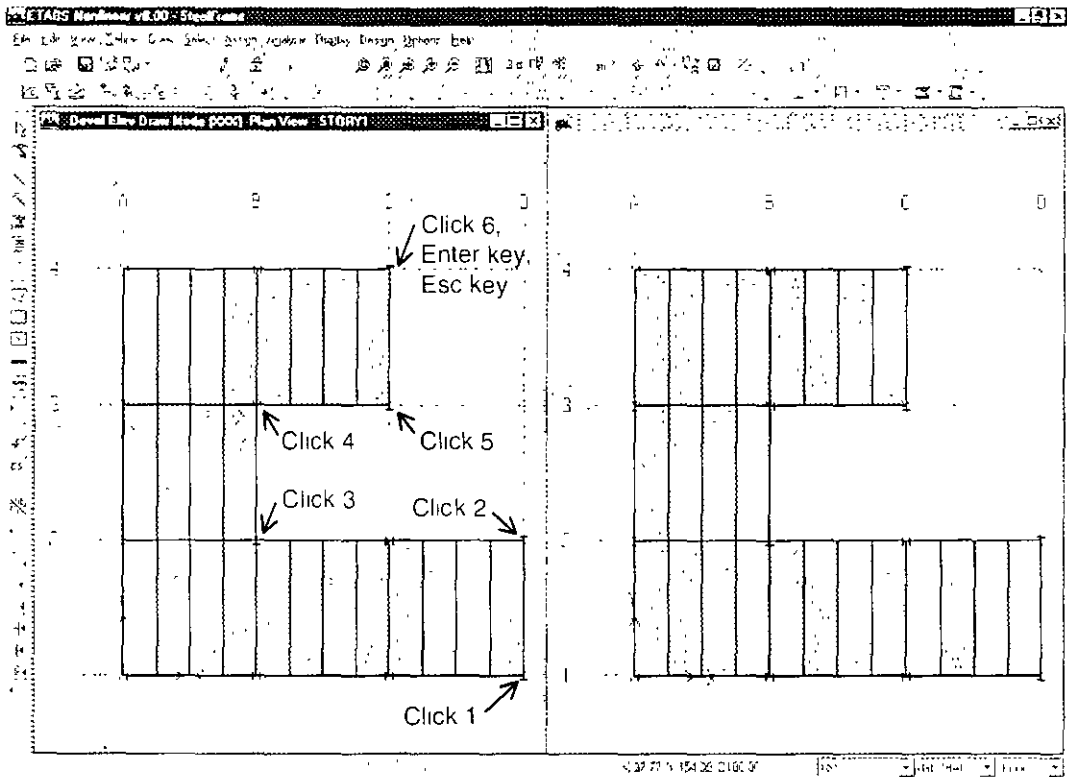
In this Step, a Developed Elevation View of the right-hand side of the building will be defined so that the wind load can be assigned to it in Step 7

- A Click the **Draw menu > Draw Developed Elevation Definition** command to bring up the Elevations Views form shown in Figure 34.




*Figure 34  
The Elevation  
Views form*

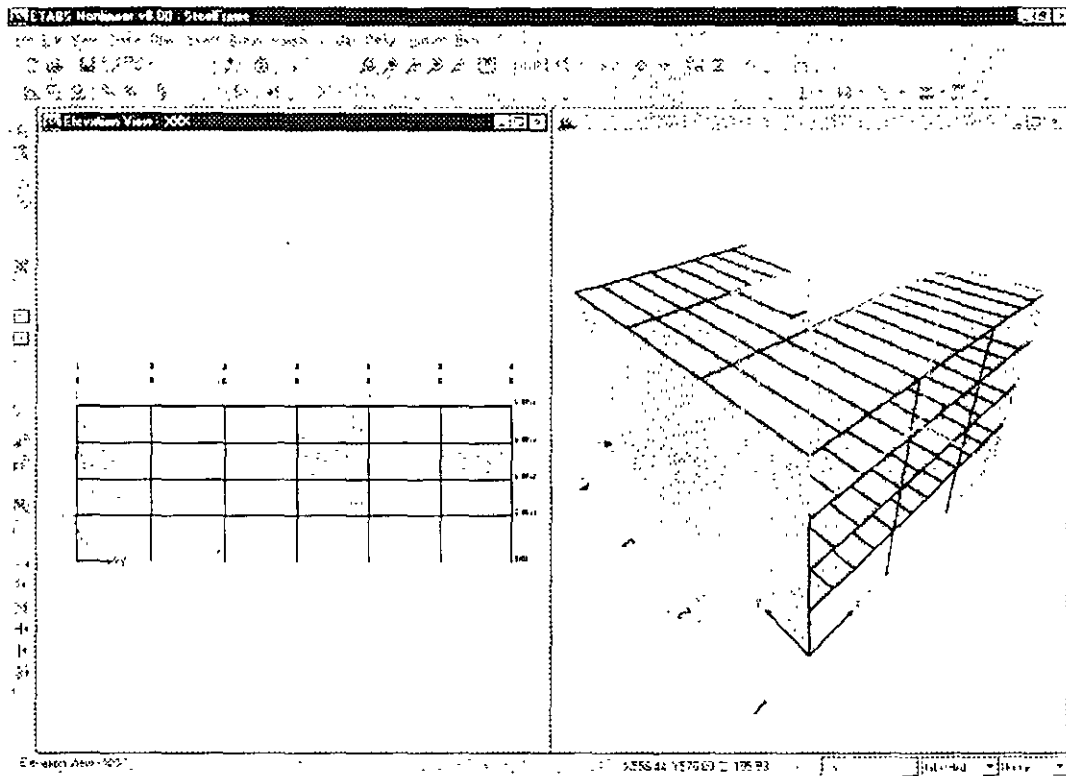
1. Type **XXX** in the Developed Elevations edit box, as shown in the figure. This will be the name of the Developed Elevation.
  2. Click the **Add New Name** button and then click the **OK** button. Note that both views are now Plan Views and that the Developed Elevation draw mode is active. The model appears as shown in Figure 35
- B. Check that the **Snap to Grid Intersections and Points** command is active. This will assist in accurately drawing the developed elevation definition. This command is active when its associated button  is depressed. Alternatively, use the **Draw menu > Snap To > Grid Intersections and Points** command to ensure that this command is active. By default, this command is active.



*Figure 35*  
*Developed*  
*elevation*  
*draw mode*

- C. Working in the left-hand Plan View (note that this can be completed in either Plan View), left click once at Grid D-1. Then moving counterclockwise around the building, left click once at D-2, B-2, B-3, C-3 and C-4 in that order. The sequence of clicks is illustrated in Figure 35
- D. When all of the points have been clicked, press the Enter key on your keyboard to finish drawing the developed elevation definition
- E. Press the Esc key on your keyboard to exit the Developed Elevation draw mode. Notice that the views return to those that existed before enabling the Developed Elevation draw mode.
- F. Make sure the Plan View is active. Click the **Elevation View** button  and select XXX (the developed elevation just defined) from the Set Elevation View form; click the **OK** button. The Plan View changes to a Developed Elevation View, as illustrated in Figure 36

2





*Figure 36  
Developed  
Elevation  
View*

The developed elevation is an "unfolded" view of the newly defined elevation. The scope of the developed elevation is illustrated by the cyan-colored lines in the 3D View.

As many developed elevations as desired can be defined. Note however, that a developed elevation can not cross itself and it can not close on itself. Either of those situations would require that the same point occur in two different locations within the developed elevations, which is not allowed

After a developed elevation has been defined, it can be viewed, objects can be drawn on it, assignments can be made to objects in it, and so forth, similar to any other Elevation View. The XXX Elevation View will be used in the next step.

G. Make sure the Developed Elevation View (i.e., Elevation View XXX) is active. Click the Plan View button  and select Story 4 from the Select Plan Level form.

H. Click the **File menu** > **Save** command, or the **Save** button, , to save your model.


2

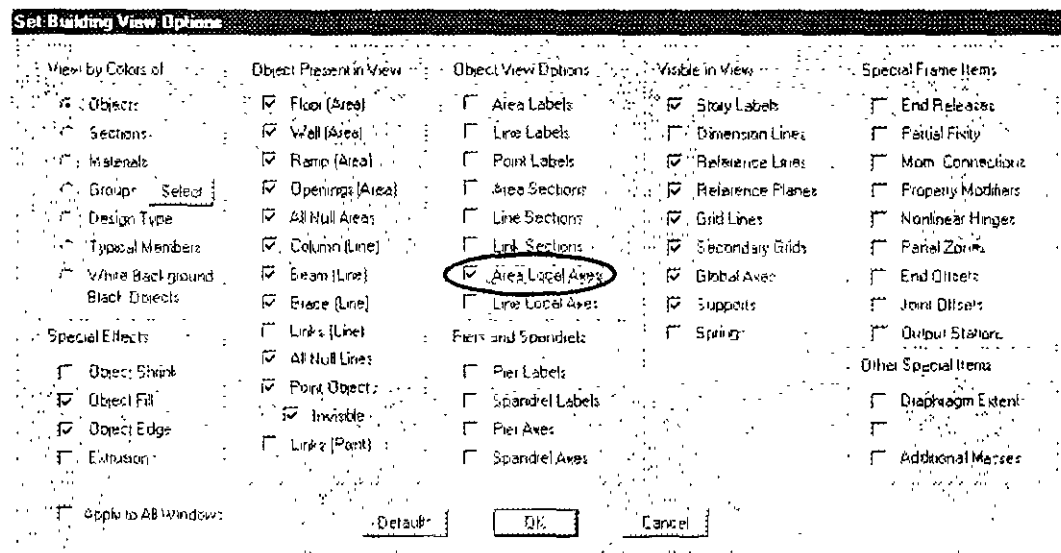
## Step 7 Assign Wind Loads

In this Step, wind loads are assigned to the Developed Elevation View defined in Step 6. Typically, wind pressure coefficients are applied to the vertical surface of an area object. In such cases, and in this example, a positive wind pressure coefficient applies wind load in the positive local 3-axis direction of the area object. A negative wind pressure coefficient applies wind load in the negative local 3-axis direction of the area object.

A. Click in the 3D View window to make that window active.

*Figure 37*  
*Set Building*  
*View Options*  
*form*

B. Click the **View menu** > **Set Building View Options** command or click the **Set Building View Options** button, , to bring up the Set Building View Options form shown in Figure 37.



The screenshot shows the 'Set Building View Options' dialog box with the following sections and options:

View by Colors of	Object Present in View	Object View Options	Visible in View	Special Frame Items
<input checked="" type="checkbox"/> Objects	<input checked="" type="checkbox"/> Floor (Area)	<input type="checkbox"/> Area Labels	<input checked="" type="checkbox"/> Story Labels	<input type="checkbox"/> End Releases
<input type="checkbox"/> Sections	<input checked="" type="checkbox"/> Wall (Area)	<input type="checkbox"/> Line Labels	<input type="checkbox"/> Dimension Lines	<input type="checkbox"/> Partial Fixity
<input type="checkbox"/> Materials	<input checked="" type="checkbox"/> Ramp (Area)	<input type="checkbox"/> Point Labels	<input checked="" type="checkbox"/> Reference Lines	<input type="checkbox"/> Mom. Connections
<input type="checkbox"/> Groups <input type="button" value="Select"/>	<input checked="" type="checkbox"/> Openings (Area)	<input type="checkbox"/> Area Sections	<input checked="" type="checkbox"/> Reference Planes	<input type="checkbox"/> Property Modifiers
<input type="checkbox"/> Design Type	<input checked="" type="checkbox"/> All Hull Areas	<input type="checkbox"/> Line Sections	<input checked="" type="checkbox"/> Grid Lines	<input type="checkbox"/> Nonlinear Hinges
<input type="checkbox"/> Typical Members	<input checked="" type="checkbox"/> Column (Line)	<input type="checkbox"/> Link Sections	<input checked="" type="checkbox"/> Secondary Grids	<input type="checkbox"/> Partial Zoning
<input type="checkbox"/> White Background	<input checked="" type="checkbox"/> Beam (Line)	<input checked="" type="checkbox"/> Area Local Axes	<input checked="" type="checkbox"/> Global Axes	<input type="checkbox"/> End Releases
<input type="checkbox"/> Black Objects	<input checked="" type="checkbox"/> Brace (Line)	<input type="checkbox"/> Line Local Axes	<input checked="" type="checkbox"/> Supports	<input type="checkbox"/> Joint Releases
<input type="checkbox"/> Special Effects	<input type="checkbox"/> Links (Line)	Piers and Spandrels	<input type="checkbox"/> Springs	<input type="checkbox"/> Output Stations
<input type="checkbox"/> Object Shrink	<input checked="" type="checkbox"/> All Hull Lines	<input type="checkbox"/> Pier Labels		<input type="checkbox"/> Other Special Items
<input checked="" type="checkbox"/> Object Fill	<input checked="" type="checkbox"/> Point Objects	<input type="checkbox"/> Spandrel Labels		<input type="checkbox"/> Diaphragm Extent
<input checked="" type="checkbox"/> Object Edges	<input checked="" type="checkbox"/> Invisible	<input type="checkbox"/> Pier Axes		<input type="checkbox"/> Additional Masses
<input type="checkbox"/> Extrusion	<input type="checkbox"/> Links (Point)	<input type="checkbox"/> Spandrel Axes		
<input type="checkbox"/> Apply to All Windows				

Buttons:

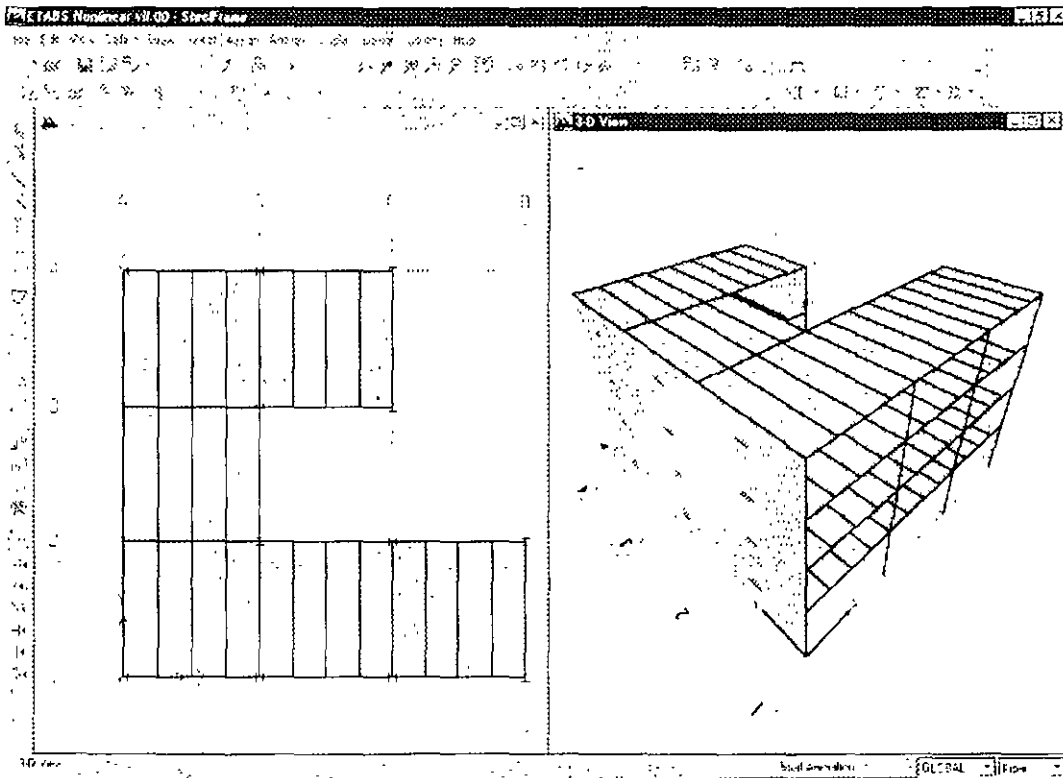



2

- 1 Check the Area Local Axes check box to turn on the area local axes and then click the **OK** button to exit the form. Red, white and blue arrows appear defining the area object local axes. Recall that Red = 1 axis, White = 2 axis and Blue = 3 axis.




The building appears as shown in Figure 38. Notice that for the vertical dummy area objects along grid line A, the blue arrows representing the local 3-axes point to the right in the positive global X direction. Note the global axes that are located at the origin of the model.

*Figure 38  
Area object  
local axes*



- C Click the Rotate 3D View button, , and then left click once in the 3D View, and while holding down the left mouse button, drag the mouse to the left. Notice that a dashed bounding box shows how the view is being rotated.

Rotate the view such that you can see the other vertical dummy area objects located on grid lines B, C and D. Confirm that the local 3 axes for those elements are also pointing in the positive global X direction.

- D. When you have confirmed that all vertical area objects have their local 3 axes pointing in the positive global X direction, click the **View menu > Set Building View Options** command or click the **Set Building View Options** button, , to bring up the Set Building View Options form. Uncheck the Area Local Axes check box to turn off the area local axes display and then click the **OK** button to exit the form.
- E. Make sure the 3D View is active and then click on the **Set Default 3D View** button, , to reset to the default 3D view.
- F. With the 3D View active, click on the **Elevation View** button, , and select A to reset the view to an elevation of grid line A.
- G. Click the left mouse button and drag the mouse to draw a "rubber band" selection box window around all of the panels in this elevation view, as shown in Figure 39.
- H. Click the **Assign menu > Shell/Area Loads > Wind Pressure Coefficient** command, which brings up the Wind Pressure Coefficient form shown in Figure 40.
  1. Select WINDX from the Wind Load Case Name drop-down box. Set the Coeff. Cp to 0.8, and check the Windward (varies) option.

Selecting the Windward option means that the wind load applied to these dummy panels will vary over the height of the building in accordance with the building code specified when the wind load was defined, in this case, ASCE 7-98.

2

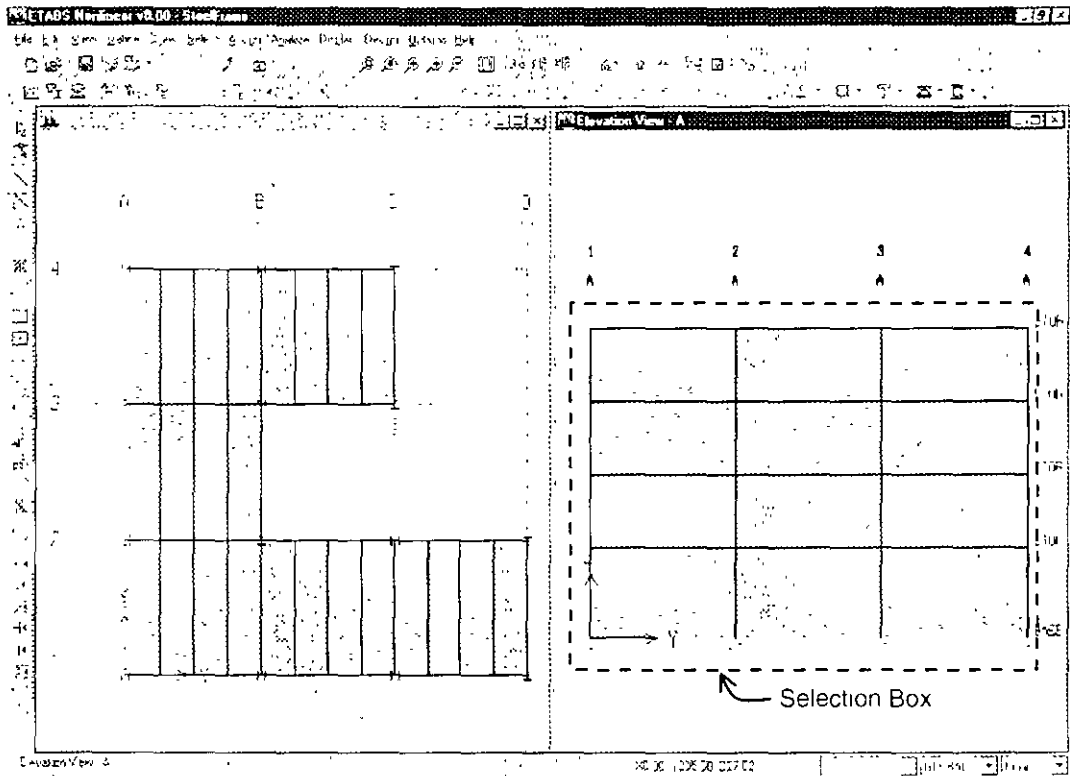
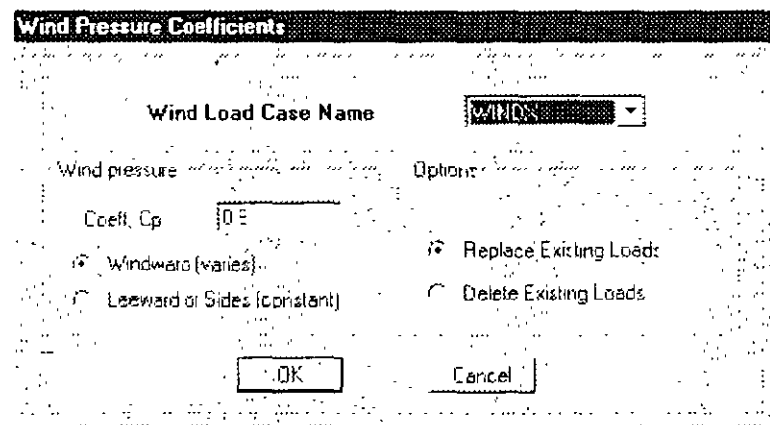



Figure 39  
Selecting  
vertical area  
objects in an  
elevation view

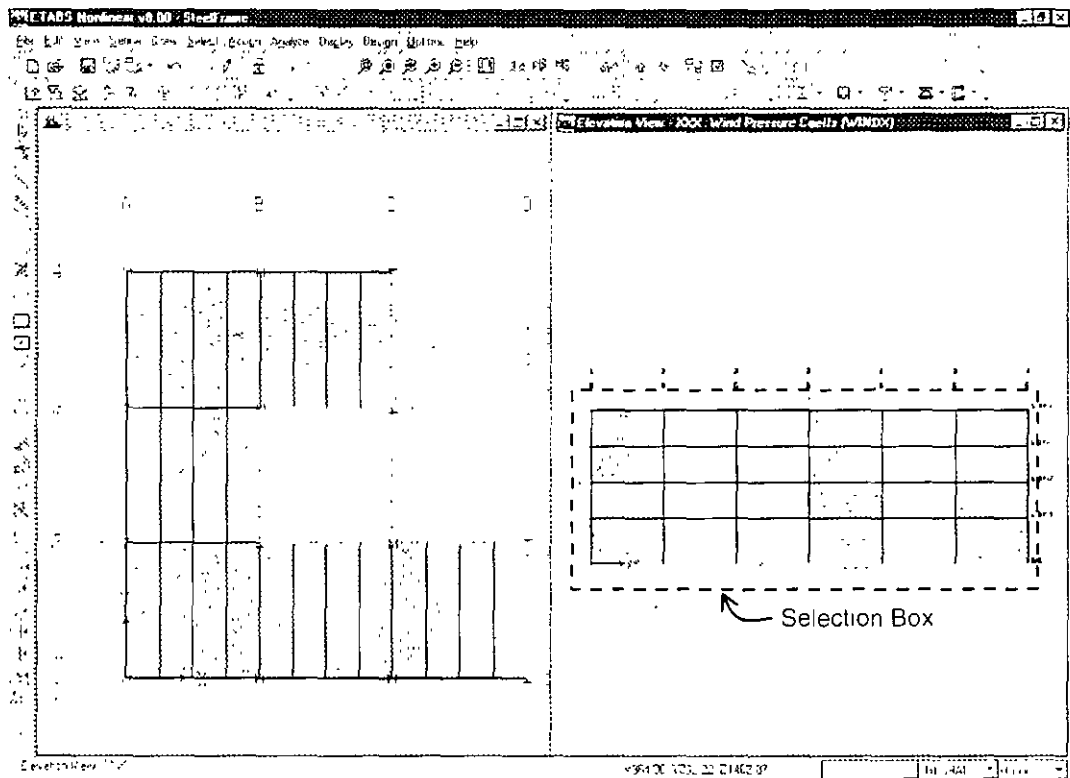
Figure 40  
The Wind  
Pressure  
Coefficients  
form



- Click the **OK** button to assign this load. Note that with the positive  $C_p$ , the load will act in the positive global X direction.

- With the Elevation View active, click the **Elevation View** button  and select XXX to display the Developed Elevation View. Click the **OK** button. In the Developed Elevation View, click the left mouse button and drag the mouse to draw a "rubber band" selection box window around all of the panels, as shown in Figure 41



**Figure 41**  
Selecting vertical area objects in a developed elevation view



- Click the **Assign menu > Shell/Area Loads > Wind Pressure Coefficient** command, to bring up the Wind Pressure Coefficient form. Set the Coeff.  $C_p$  to 0.5, and check the Leeward or Sides (constant) option. Click the **OK** button to assign this load. Again, note that with the positive  $C_p$ , the load will act in the positive global X direction

**2**

Selecting the Leeward or Sides option means that the wind load applied to these dummy panels will be constant over the height of the building in accordance with the building code specified when the wind load was defined, in this case, ASCE 7-98. The magnitude of the wind load is based on the elevation of the top of the building

- K. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the wind pressure coefficient assignments.
- L. Make sure the Elevation View is active and then click on the **Set Default 3D View** button, , to reset the view to the default 3D view.
- M. Click the **File menu > Save** command, or the **Save** button, , to save your model.

## Step 8 Review Database Display of Input Data

In this Step, a database display of the wind pressure coefficients that were input in Step 7 will be reviewed.

- A. Click the **Display menu > Set Input Table Mode** command to bring up the Database Input Tables form. Click on the Assignments tab and then check the Wind Pressures check box as shown in Figure 42. Make sure that the Selection Only check box is *not* checked.
  1. Click the **OK** button to display the selected database table. The table shown in Figure 43 is displayed.

Each row in the table corresponds to one area object. Notice that the fifth column in the table, labeled Cp, displays the Cp coefficients that were input for each of the vertical area objects. The next three columns display the global X, Y and Z components of this Cp factor. In this case, all of the load is in the positive global X direction, as desired.
  2. Click the **OK** button to close the database window.
  3. Repeat the process to view other tables if desired.

Figure 42  
The Database  
Input Tables  
form

**Database Input Tables**

Definitions (0 of 18 Active Boxes Checked)      ASSIGNMENTS (11 of 17 Active Boxes Checked)

<input type="checkbox"/> Point Assignments	<input type="checkbox"/> Line Assignments	<input type="checkbox"/> Area Assignments
<input type="checkbox"/> Support (Relevant)	<input type="checkbox"/> Frame Sections	<input type="checkbox"/> Wind/Dir/Dang
<input type="checkbox"/>	<input type="checkbox"/> Releases	<input type="checkbox"/> Local Area
<input type="checkbox"/>	<input type="checkbox"/> Offsets	<input type="checkbox"/> Pier Spandrel
<input type="checkbox"/>	<input type="checkbox"/> Output Station	<input type="checkbox"/> Floor Mesh Option
<input type="checkbox"/>	<input type="checkbox"/> Local Axes	<input type="checkbox"/> Uniform Load
<input type="checkbox"/>	<input type="checkbox"/>	<input checked="" type="checkbox"/> Wind Pressures
<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/> Summary
<input type="checkbox"/> Summary	<input type="checkbox"/>	<input type="checkbox"/>

Check/Uncheck All      Check/Uncheck All      Check/Uncheck All

Select Load

6 of 6 Loads Selected

Ok      Cancel      Check/Uncheck All

Figure 43  
Database  
table for wind  
pressure co-  
efficients

**Area Wind Pressures**

Edit View

Area Wind Pressures


Case	Story	Area	Windward	Cp	XComponent	YComponent	ZComponent
WINDX	STORY4	W1	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY4	W2	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY4	W3	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY4	W4	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY4	W5	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY4	W6	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY3	W1	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY3	W2	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY3	W3	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY3	W4	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY3	W5	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY3	W6	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY2	W1	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY2	W2	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY2	W3	Yes	0.8000	0.8000	0.0000	0.0000
WINDX	STORY2	W4	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY2	W5	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY2	W6	No	0.5000	0.5000	0.0000	0.0000
WINDX	STORY1	W1	Yes	0.8000	0.8000	0.0000	0.0000

Navigation icons: [Back] [Forward] [Home] [End]


## Step 9 Run the Analysis

In this Step, the analysis will be run.

2



- A. Click the **Analyze menu > Run Analysis** command or the Run Analysis button, , and click the **Run** button on the Run Options form

The program will create the analysis model from your object-based ETABS model, and will soon display an "Analyzing, Please Wait" window. Data will scroll in this window as the program runs the analysis. After the analysis has been completed, the program performs a few more bookkeeping actions that are evident on the status bar in the bottom left-hand corner of the ETABS window.

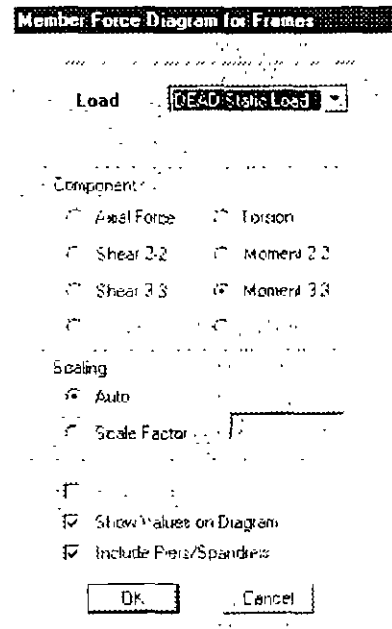
When the entire analysis process has been completed, the model automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the Lock/Unlock Model button, , appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results.

## Step 10 Graphically Review the Analysis Results

In this Step, the analysis results will be reviewed using graphical representation of the results.

- A. Make sure the 3D View is active. Then click on the **Elevation View** button, , and select 1 to reset the view to an Elevation View of grid line 1.
- B. Click the **Show Frame/Pier/Spandrel Forces** button, , or the **Display menu > Show Member Force/Stresses Diagram > Frame/Pier/Spandrel Forces** command to bring up the Member Force Diagram for Frames form shown in Figure 44.

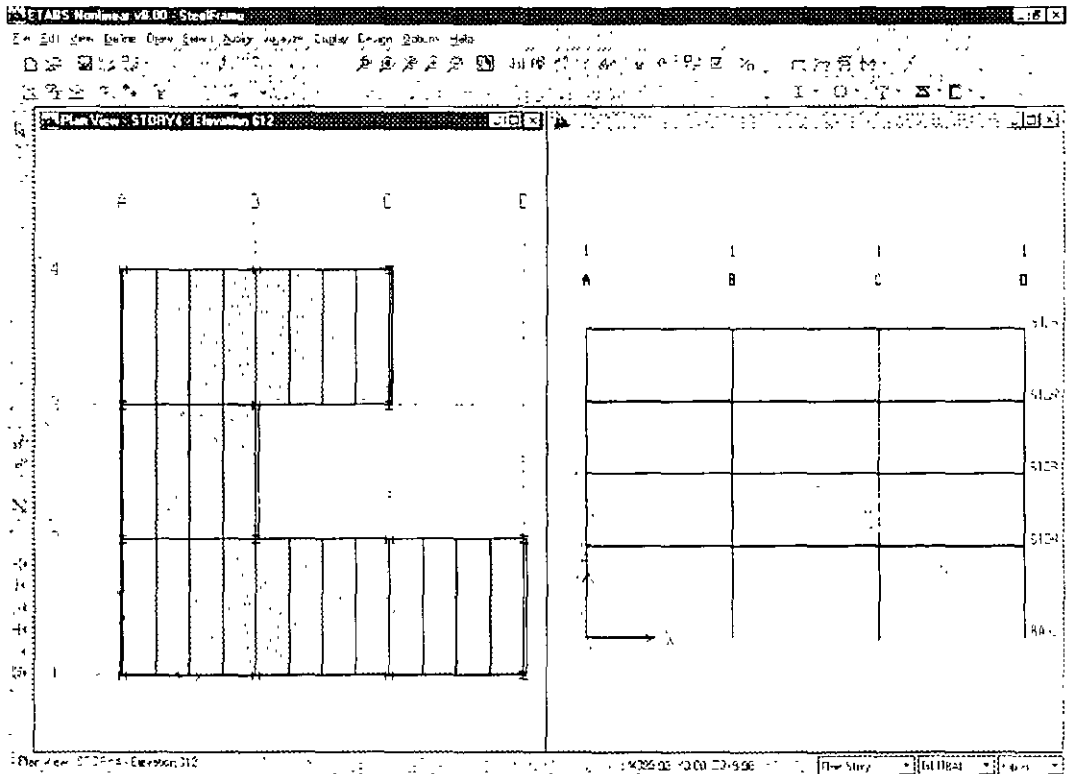
- 1 Select DEAD Static Load from the Load drop-down box.
- 2 Select the Moment 3-3 option
- 3 Uncheck the Fill Diagram if it is checked.
- 4 Check the Show Values on diagram check box
5. Click the **OK** button to generate the moment diagram output shown in Figure 45.



2

*Figure 45*  
M33 moment diagram in an elevation view

*Figure 44*  
Member-Force Diagram for Frames form





2

Note that these moment diagrams are plotted with the positive moment on the tension side of the member. Change this, if desired, using the **Options menu > Moment Diagrams on Tension Side** command toggle.

- C. Right click on the top level beam between grid lines A and B to bring up the Diagram for Beam form shown in Figure 46.

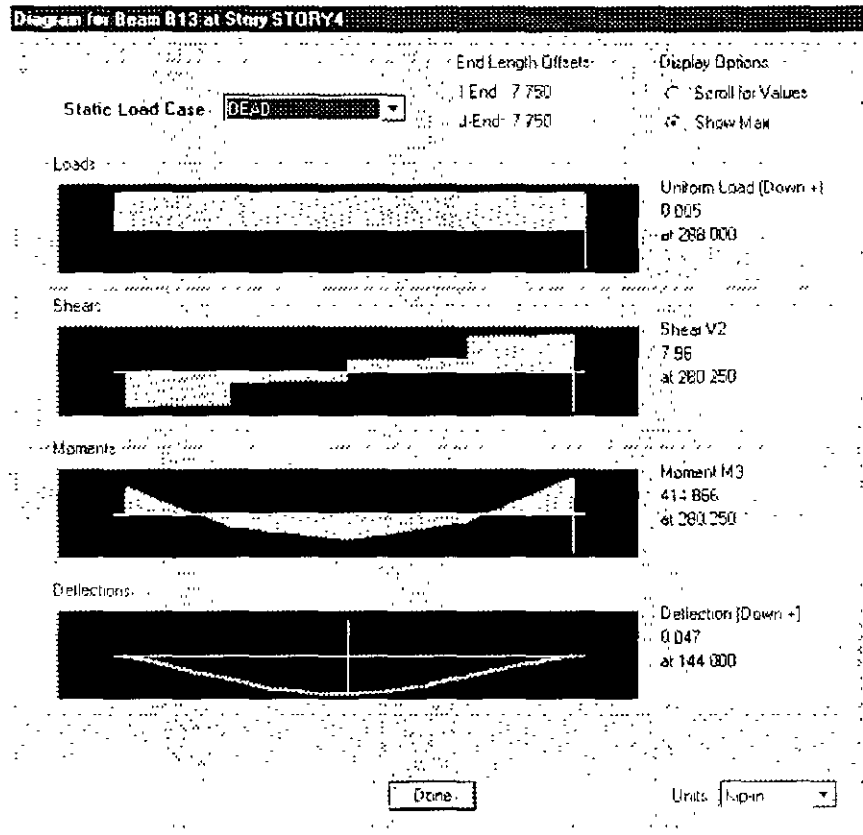




Figure 46  
Force details  
obtained by  
right-clicking  
a beam shown  
in the eleva-  
tion view in  
Figure 45

Note that the applied load, shear moment and deflection are shown for the beam, and the maximum values are identified on the Diagram for Beam form. The Uniform Load shown for the beam is the tributary load from the deck plus the self weight of the beam. Tributary concentrated loads are not shown.

1. Click the Scroll for Values option and a scroll bar appears at the bottom of the form. Drag the scroll bar with your mouse to see values at different locations along the beam.
  2. Set the Units drop-down box at the bottom of the form to kip-ft. Then type 6.5 into the Location edit box. The load, shear, moment and deflection values are displayed at this exact location in kip and feet units.
  3. Click the Static Load Case drop-down box and select CLADDING from the list to display the forces acting on this beam from the superimposed dead load named CLADDING. The Uniform Load (Down +) should display a value of 0.250 klf, which is the cladding load that was applied in Step 5.
  4. Click the **Done** button to close the form.
- D. Make sure the Elevation View is active and then click the **Display menu > Show Undeformed Shape** command or the **Show Undeformed Shape** button, , to clear the display of the moment diagrams in the Elevation View.
- E. Make sure the Elevation View is active and then click on the **Set Default 3D View** button, , to reset the view to the default 3D view.

## Step 11 Design the Composite Beams

In this Step, the composite beams will be designed. Note that the analysis (Step 9) should be run before completing the following Action Items.

- A. In the Plan View, right click on one of the secondary (infill) beams in the bay bounded by grid lines 1, 2, A and B. The Line Information form shown in Figure 47 appears.

2

**Line Information**

Location | Assignments | Loads

Identification

Label: B2B      Line Type: Beam

Story: STORY4      Design Procedure: Composite Beam

Length	288
Start Point (I)	28
Story	STORY4
X	144
Y	0
Delta Z	0
End Point (J)	17
Story	STORY4
X	144
Y	288
Delta Z	0

Units: Kip-in

OK

**Figure 47**  
Line Information form



**Note:**

To change the design procedure for a beam, select the beam and use the **Design menu > Overwrite Frame Design Procedure** command

Note that the form reports that the Design Procedure for this form is Composite Beam. The program assigned this default design procedure to this line object because (1) it lies in a horizontal plane, (2) the ends of the beam are pinned (that is, moment is released at each end of the beam), and (3) it is assigned a steel section that is either I-shaped or a channel. Any steel frame element that does not meet these requirements is given a Steel Frame design procedure.

Review the information available on all three tabs of the Line Information form and then click the **OK** button to close the form.

- B. Click the **Options menu > Preferences > Composite Beam Design** command. The Preferences form shown in Figure 48 appears.

*Figure 48*  
Preferences  
form for com-  
posite beam  
design

Preferences	
Factors	
Beam	Deflection   Vibration   Price
Shored?	No
Middle Range (%)	70
Pattern Live Load Factor	0.75
Stress Ratio Limit	0.95
Reset Tab	
Design Code	AISC-LRFD93
Reset All	OK Cancel

1. Click the Design Code drop-down box near the bottom of the form to see the available design codes. Select the AISC-LRFD93 code.
  2. Review the information available on all five tabs in the Preferences form and then click the **OK** button to accept the Design Code change.
- C. Click on the title bar of the 3D View to activate 3D view.
- D. Click the **Set Building View Options** button . When the Set Building View Options form appears, uncheck the Object Fill check box as shown in Figure 49. This will remove the display of the fill in the area objects.

2

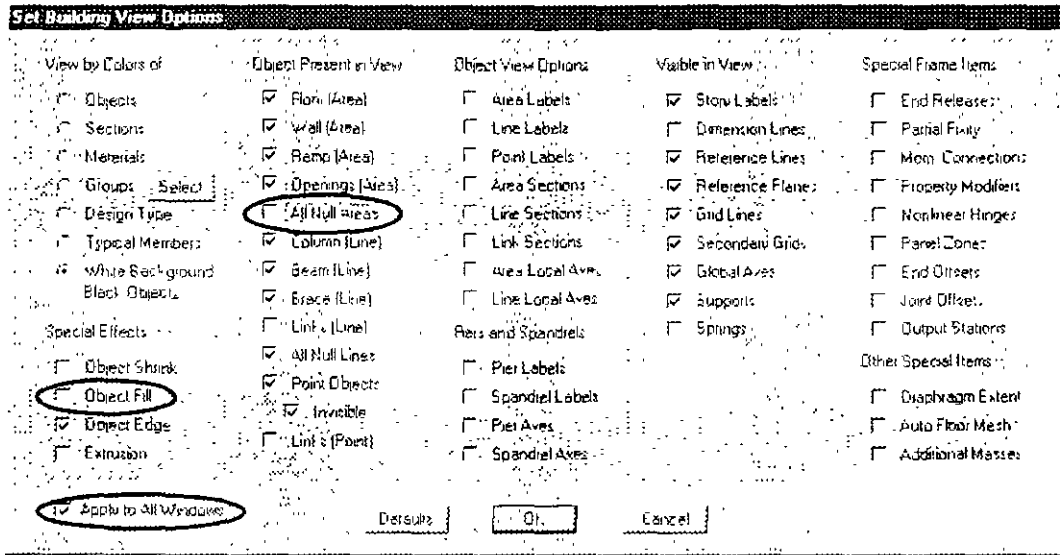


Figure 49  
Set Building  
View Options  
form

1. In the Object Present in View area of the form, uncheck the All Null Areas check box.
2. Check the Apply to All Windows check box and click the **OK** button to accept the changes.



**Note:**  
The **Start Design Without Similarity** command is only available after the analysis has been run.

- E. With the 3D View active, click the **Design menu > Composite Beam Design > Start Design Without Similarity** command to start the design process. The program designs the composite beams, selecting the optimum beam size from the A-CompBm auto select section list that was assigned to them when they were drawn in Step 2.

When the design is complete, the selected sizes are displayed on the model. The model appears as shown in Figure 50.

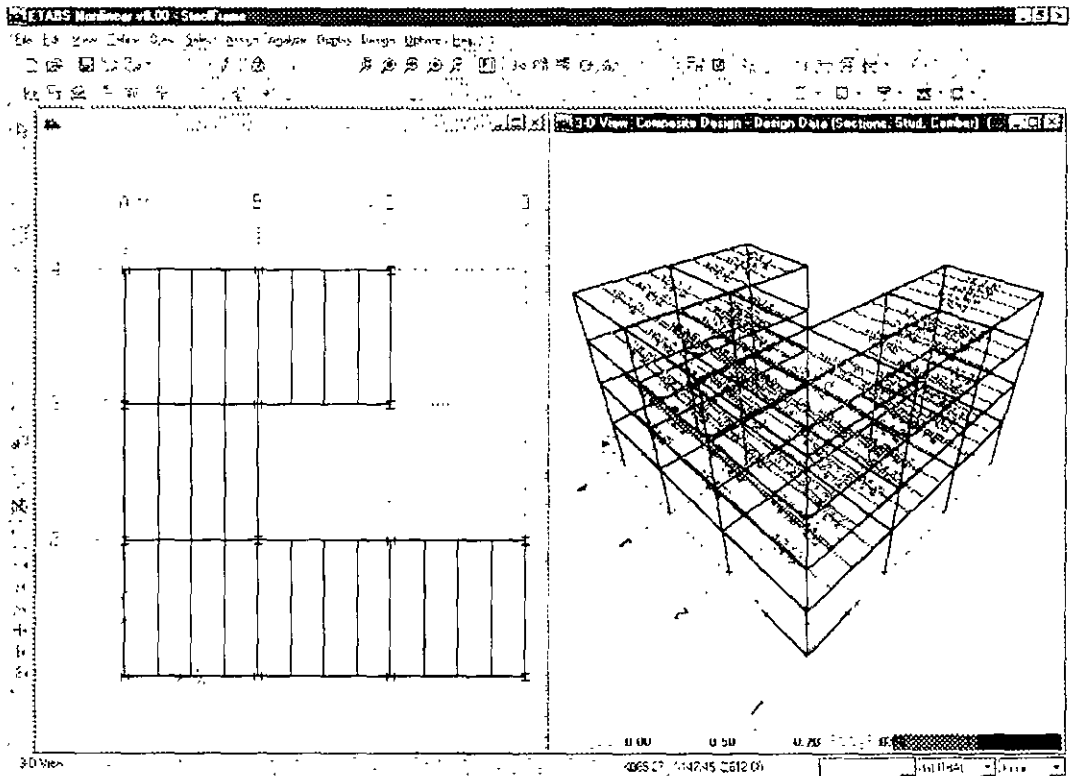
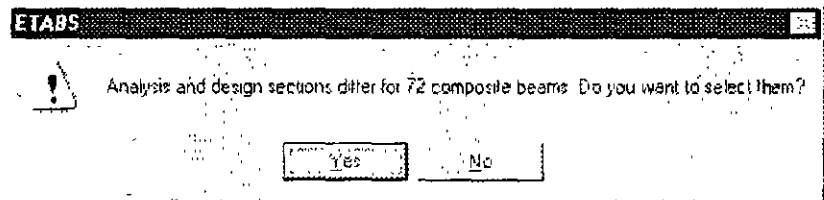


Figure 50 Model after the initial composite design

F Click the **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command. A message similar to the one shown in Figure 51 appears. Click the **No** button to close the form.

Figure 51 Analysis vs Design Section warning message for an incomplete design



In the initial analysis (Step 9), the program used the median section by weight from the A-CompBm auto select section list. During design (this Step), the program selected a W12X19 design section, which differs from the analysis section used. The message in Figure 51 indicates that the analysis and design sections are different.

2

The goal is to repeat the analysis (Step 9) and design (Step 11) process until the analysis and design sections are all the same. Note that when the building is reanalyzed (i.e., Step 9 is repeated), ETABS will use the current design sections (i.e., those selected in Step 11) as new analysis sections for the next analysis run. Thus, in the next analysis of this example, the composite beams will be analyzed using W12X19 analysis sections.

- G. Right click on one of the composite beams in the 3D View shown in Figure 50. The Interactive Composite Design and Review form shown in Figure 52 appears.

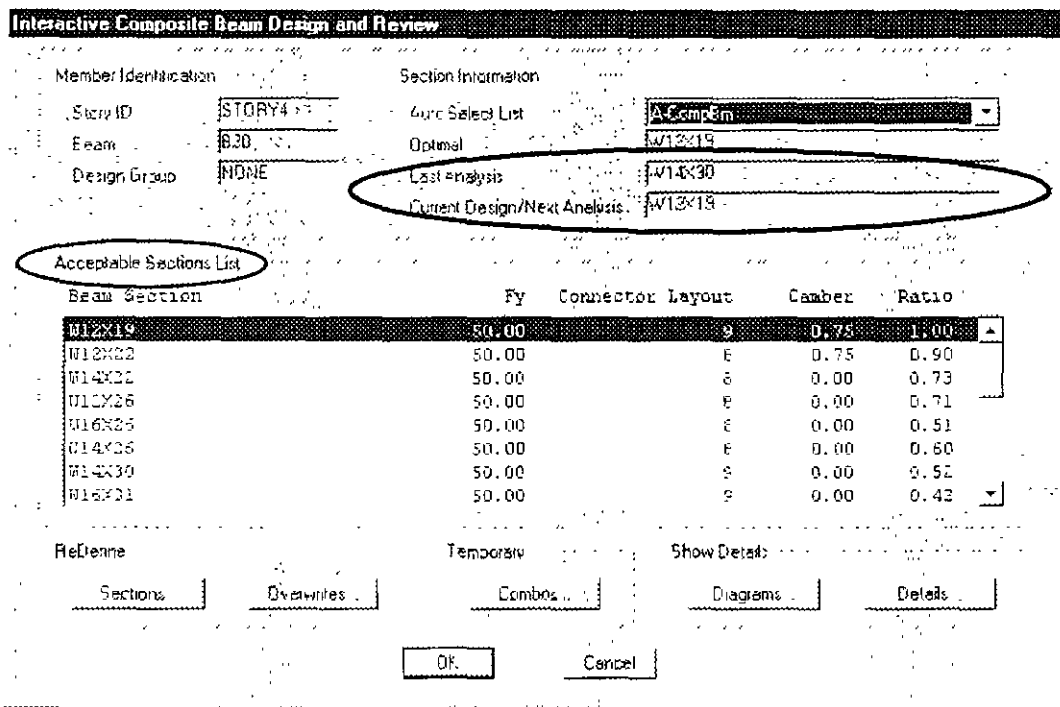


Figure 52  
Interactive  
Composite  
Beam Design  
and Review  
form

Note that the current design section is reported as W12X19 and the last analysis section is reported as W14X30

The Acceptable Sections List shows all of the beams in the A-CompBm auto select section list that are adequate for the design forces

- Click on the **Details** button in the Interactive Composite Beam Design and Review form. The Composite Beam Design Form shown in Figure 53 appears. This form shows detailed design information about the beam. Review the information on each of the four tabs in this form. Then click the X in the upper right-hand corner of the form to close it.

2

AISC-LRFD93 Composite Beam Design			
Beam Label: B30		Beam Label: B30	
Units: Kip-in		Units: Kip-in	
Beam Label: B30	Story: STORY4	Shored: No	Overwrites: No
Group: None	Length: 398.000	Camber: 0.750	b-cp: N/A
Beam: W12x19	Loc X: 72.000	Comparative: \$0.45	t-cp: N/A
Fy: 50.000	Loc Y: 720.000	Stud Diam: 0.750	Fy-cp: N/A
Fu: 80.000	RLLF: 1.000	EQF: 1.000	Consider-cp: No
Stud Layout: 5			
Seg Length: 275.410			
Stud Ratio: 0.091			
Deck Left: DECK1	Deck Right: DECK1	Itam: 668.31	wbar: 14.877
Dir Left: Respondr	Dir Right: Respondr	Ibase: 130.00	ybar: 6.100
b-eff Left: 36.000	b-eff Right: 36.000	lw: 130.00	
Clap Left: 1.000	Clap Right: 1.000	leff: 467.22	yeff: 12.941
Cbot Left: 0.000	Cbot Right: 0.000	PCC: 39.24%	Qr: 25.02

**Figure 53**  
Composite  
Beam Design  
form



**Note:**

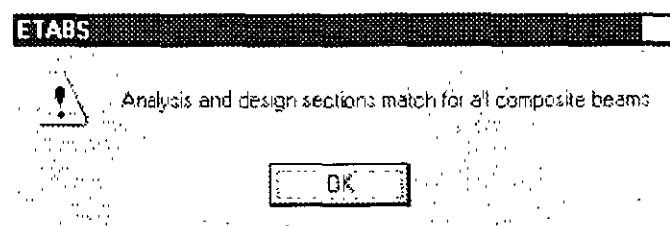
Use the **File menu > Print Tables > Composite Beam Design** command to print summary and detailed composite beam design information.

- Click the **Cancel** button to close the Interactive Composite Design and Review form.
- To rerun the analysis with the new analysis sections for the composite beams, click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, and click the **Run** button on the Run Options form.
  - When the analysis is complete, click the **Design menu > Composite Beam Design > Start Design Without Similarity** command to start the composite beam design process.
  - Click the **Design menu > Composite Beam Design > Verify Analysis vs Design Section** command. The message shown in Figure 54 should appear, indicating that the analysis and design sections are the same for all composite beams. Click the **OK** button.



2

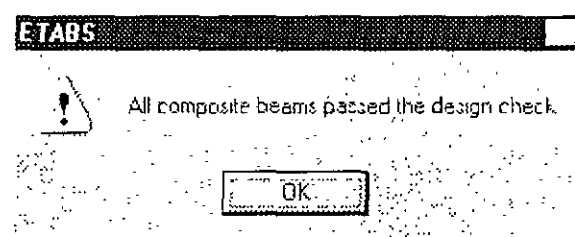
*Figure 54  
Analysis vs  
Design Sec-  
tion warning  
message for a  
complete de-  
sign*



If you do not get this message, repeat Action Items H, I and J until you do get it, before proceeding to the next Action Item.

- K. Click the **Design menu > Composite Beam Design > Verify All Members Passed** command. The message shown in Figure 55 should appear, indicating that all composite beams passed the design check. Click the **OK** button to close the form.

*Figure 55  
Verify All  
Members  
Passed  
warning  
message for  
a complete  
design*



- L. Click the **Select All** button,  $\text{all}^{\text{F5}}$ , or click the **Select menu > All** command, or press the Ctrl and A keys simultaneously on your keyboard to select all objects in the model
- M. Click the **Design > Composite Beam Design > Make Auto Select Section Null** command and click the **OK** button for the resulting message. This removes the auto select section list assignments from the composite beam members and replaces them with their current design sections.
- N. Click the **Assign menu > Clear Display of Assigns** command. Also click the **Clear Section** button,  $\text{cl}^{\text{F5}}$ , to clear the selection.
- O. Click the **File menu > Save** command, or the **Save** button,  $\text{save}^{\text{F5}}$ , to save your model. The composite beam design is now complete.

## Step 12 Design the Steel Frame

In this Step, steel frame design is completed. Note that the analysis (Step 9) should be run before performing the following Action Items.

- A In the Plan View, right click on the beam along grid line A between grid lines 1 and 2. The Line Information form shown in Figure 56 appears. Review the information. Note that the design procedure for this beam is Steel Frame. Click the **OK** button to close the form.

**Line Information**

Location | Assignments | Loads

Identification

Label: B4      Line Type: Beam

Story: STORY4      Design Procedure: Steel Frame


Length	288
Start Point (I)	C4
Story	STORY4
X	0
Y	0
Delta Z	0
End Point (J)	C5
Story	STORY4
X	0
Y	288
Delta Z	0

Units: Kip-in

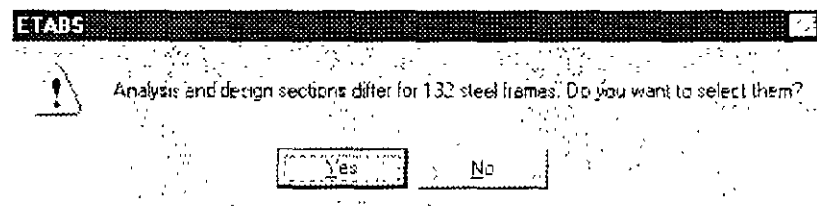
OK

Figure 56  
Line  
Information  
form

2

- B Click on the title bar of the 3D View to activate 3D view. This allows the design results to appear in the 3D View.
- C Click the **Design > Steel Frame Design > Start Design/Check of Structure** command or click the **Start Steel Design/Check of Structure** button.  to start the steel frame design process. The columns and the lateral beams that span between columns are designed
- D When the initial design is complete, a form similar to that shown in Figure 57 appears

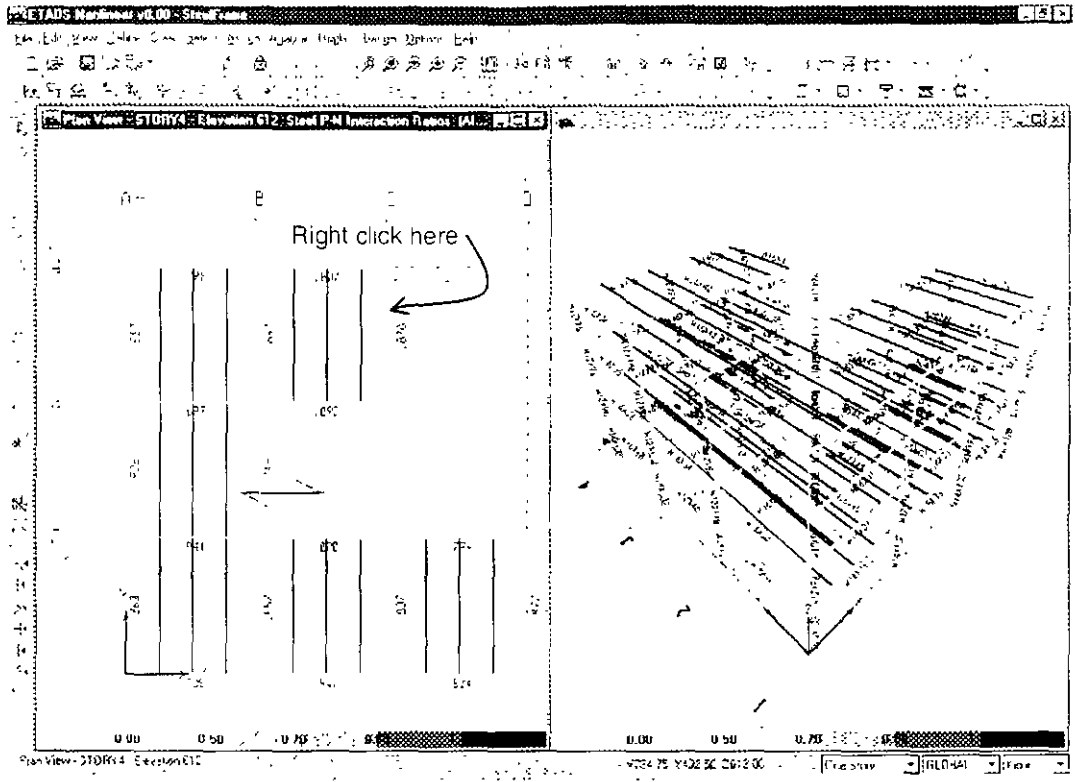
*Figure 57  
Analysis vs  
Design Sec-  
tion warning  
message for  
an incomplete  
design*



Similar to composite design (described in Step 11), in the initial analysis, the program used the median section by weight from the AUTOLATBM and A-LatCol auto select section lists for the analysis. The design sections chosen differ from the analysis sections used. The message in Figure 57 indicates that the analysis and design sections are different.

- I Click the **No** button to close the form.
- E Click on the title bar of the Plan View to activate the view.
- F Click the **Design > Steel Frame Design > Display Design Info** command. The Display Design Results form appears.
  - 1. Make sure that the Design Output option is selected and that P-M Ratio Colors & Values is selected in the Design Output drop-down box. Then click the **OK** button.

Results are displayed in the Plan View and the model appears as shown in Figure 58.



**Figure 58**  
Model after  
the initial  
steel frame  
design

G In the Plan View, right click on the beam along grid line C between grid lines 3 and 4 as indicated in Figure 58. The Steel Stress Check Information form shown in Figure 59 appears. Note that the reported analysis and design sections are different.



**Note:**

Use the **File menu > Print Tables > Steel Frame Design command** to print additional steel frame design information.

The main body of the form lists the design stress ratios obtained at various stations along the beam for each design load combination. Note that the program automatically created code-specific design load combinations for this steel frame design. (It also did this for the composite design.)

Click the **Details** button on the Steel Stress Check Information form. The Steel Stress Check Information AISC-ASD89 form shown in Figure 60 appears. Note that you can print this information using the File menu on the form.

2

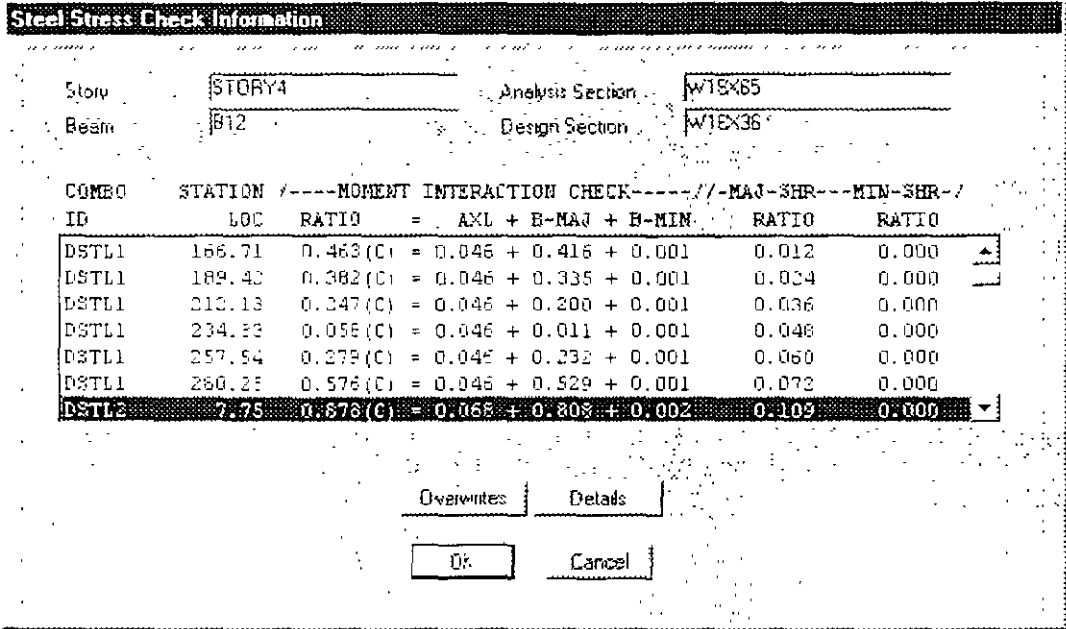
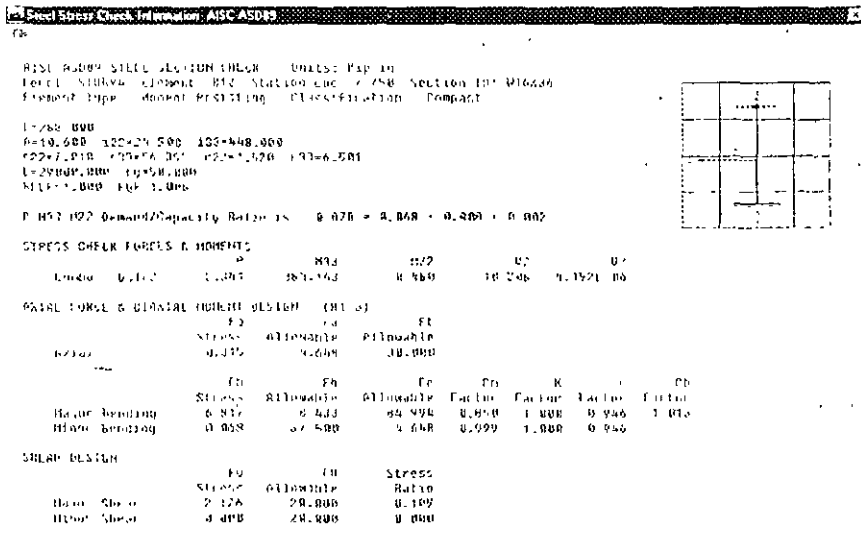


Figure 59  
Steel Stress  
Check Infor-  
mation form

Click the X in the upper right-hand corner of the Steel Stress Check Information AISC-ASD89 form to close it.

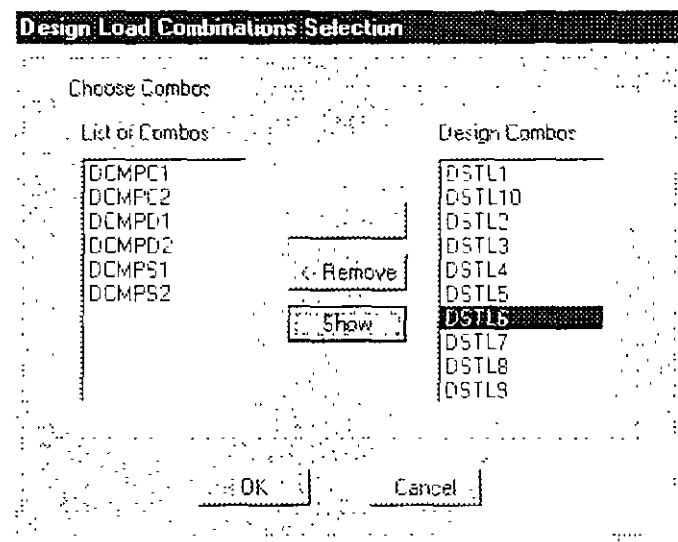
Click the **Cancel** button to close the Steel Stress Check Information form.

Figure 60  
Steel Stress  
Check Infor-  
mation AISC-  
ASD89 form




- H Click the **Design > Steel Frame Design > Select Design Combo** command. The Design Load Combinations Selection form shown in Figure 61 appears.

**Figure 61**  
Design Load  
Combination  
Selection  
form





The Design Combos list identifies the ten default steel frame design load combinations created by the program. Click on DSTL6 to highlight it and then click the **Show** button. The Load Combination Data form shown in Figure 62 pops up, showing how the program defined design load combination DSTL6.

1. Click the **OK** button in the Load Combination Data form to close it. If desired, review other design load combination definitions and then click the **OK** button to close the Data form.
  2. Click the **Cancel** button in the Design Load Combinations Selection form to close it without accepting any changes that may have inadvertently been made.
- I Click on the title bar of the Plan View to activate the view.
  - J Click the **Display menu > Show Undeformed Shape** command or the **Show Undeformed Shape** button, , to clear the display of the stress ratios.

2

Figure 62  
Load Combination Data  
form


Case Name	Scale Factor
CLADDING Static Load	1
DEAD Static Load	1
SDEAD Static Load	1
WINDX Static Load	-1

- K Click on the title bar of the 3D View to activate the 3D view.
- L To rerun the analysis with the new analysis sections for the steel beams, click the **Analyze menu > Run Analysis** command or the **Run Analysis** button, , and click the **Run** button on the Run Options form.
- M When the analysis is complete, a deformed shape will display. Click the **Design > Steel Frame Design > Start Design/Check of Structure** command or click the **Start Steel Design/Check of Structure** button, , to start the steel frame design process.

When the design is complete, a message will pop up indicating how many design sections are different from the analysis sections. Click the **No** button to clear the pop up box.

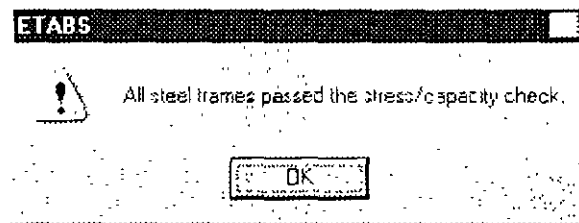
Repeat Action Items L and M until the analysis and design sections are the same, which is indicated when no message pops up at the end

of the design (and the "wait hour glass" has disappeared). This may take five or more iterations for this example


- N. When the analysis and design sections are the same, click the **Select All** button, , or click the **Select menu > All** command, or press the **Ctrl and A** keys simultaneously on your keyboard to select all objects in the model.
- O. Click the **Design > Steel Frame Design > Make Auto Select Section Null** command and click **OK** for the resulting message. This removes the auto select section list assignments from the steel frame members and replaces them with their current design sections.
- P. Click the **Design > Steel Frame Design > Verify All Members Passed** command. A form similar to that shown in Figure 63 should appear indicating that all members passed.

2

*Figure 63  
Verify All  
Members  
Passed  
warning  
message for  
a complete  
design*



Note that members not passing at this stage is an indication of inadequate sections in the auto select list. The program would have used the largest section in the auto select list for both analysis and design, finding it inadequate. In that case, either add more sections to the auto select list or assign a larger sections to the members that did not pass and continue with the design process.

- Q. Click the **File menu > Save** command, or the **Save** button, , to save your model. The steel frame design and this introduction to ETABS Version 8 are now complete