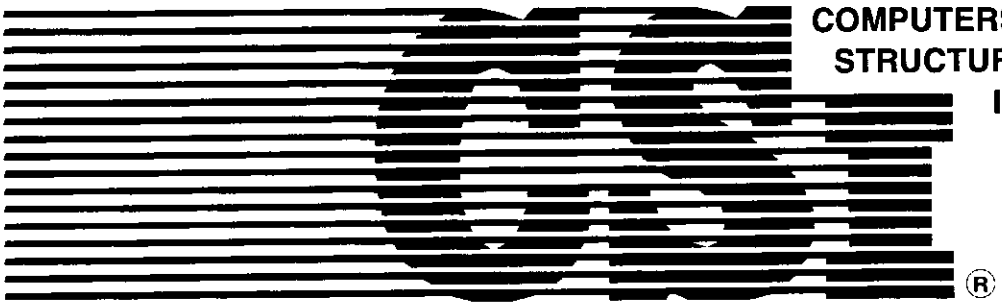


COMPUTERS &
STRUCTURES
INC.

ETABSIN™

An Interactive Graphical Model Builder for ETABS®

USER'S MANUAL

Version 6.2
Revised May, 1997

Developed and written in U.S.A.

COPYRIGHT

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@csiberkeley.com

DISCLAIMER

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

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

Table of Contents

Chapter I

Introduction, Installation and Terminology

Introduction

ETABSIN is an interactive graphical Model Builder for the ETABS building analysis program. ETABSIN allows you to model a building using an intuitive graphical method, while still maintaining the numerical exactness necessary for dimensional and structural elements.

ETABSIN does no analysis, but prepares and edits the Input Files used by program ETABS. An Input File is in standard ASCII text format and may be edited with any ASCII text editor. See the ETABS User's Manual "ETABS Input Data File" for a complete description of the Input File format.

ETABSIN is a Windows program and **requires** Microsoft Windows 95 or later.

Installation

ETABSIN is supplied on a floppy disk as part of the complete ETABS package. The whole of the ETABS set of programs and other associated files must be installed from Windows. Follow the installation procedure given in the ETABS User's Manual.

The setup program adds a Program Group titled ETABS including a Program Icon titled ETABSIN to the Windows Program Manager. No other changes are made to the Windows settings.

During the execution of ETABSIN several scratch files are written. These are written to the Working Directory identified as a property of the ETABSIN Icon. The Working Directory is also the first directory on the path to select the section property database file. The setup program installs the program icon with the Working Directory identified as the directory where the executables are located. You may want to change the Working Directory by editing the properties of the ETABSIN Icon.

Terminology

There are a number of terms used in this manual and in the ETABSIN program that have a special meaning. These terms are described below:

Current File

This is the file shown in the Title Bar at the top of the ETABSIN screen. If you do **Save** from the **File** menu, the current Building will be saved in this file.

Current Building

The current Building is ALL items that can be defined in ETABSIN or read from an Input File. This includes Story definitions, Material and Section Properties, Structural Element definitions, all Frames and Frame Locations, and all Loading. In short, it is everything.

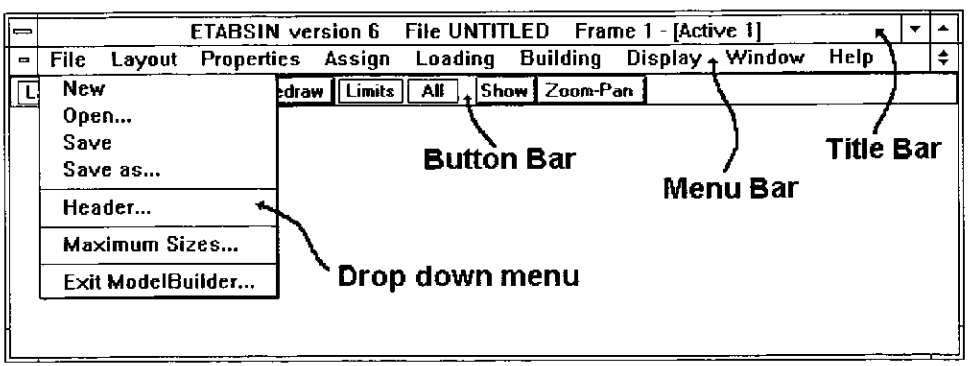
When you do **Save** or **Save as** in the **File** menu, the current Building is written into the Input File. When you do **Open** from the **File** menu, the current Building is erased and the contents of the Opened File become the new current Building.

Current Frame

Your Building may be made up of a single Frame or it may have many Frames, each at one or more Locations. Each Frame contains the dimensional Layout of the Frame along with the Structural Elements and Frame Loading. A Frame does NOT contain Stories, Material and Section Properties, Masses or Building Loading.

When working in ETABSIN, you can only display and work on one Frame at a time, which is the current Frame. If you have additional Frames, they are kept in temporary files on the hard disk. The only exceptions to this are for Masses, since they are applied to the entire Building, and Building Display, which shows all Frames at all Locations, but can not be edited.

ETABSIN Screen



Title Bar

The Title Bar is the top line of the ETABSIN screen. It contains the program Version number (6), the current File (UNTITLED), the current Frame (1) and the window number (1). There can be up to four display windows within ETABSIN. The active window is shown in the Title Bar.

Menu Bar and Drop Down Menu

When you click on an item in the Menu Bar, a drop down menu appears, where you can click on an individual item to select it. Click outside of the drop down menu to close it without doing anything.

The Menu Bar is primarily used to access Spreadsheets and Dialog Boxes in which you deal with the Building using numbers and the keyboard. The menu items are discussed in Chapter V, **Using the Menus**.

Button Bar and Top Windows

The Button Bar contains a number of buttons which you can click on to perform various functions controlling the display and mouse.

The Button Bar is used to deal with the Building in a graphical way using the screen and mouse. Top Windows are windows accessed from the Button Bar. They always stay on top of the ETABSIN screen and allow you to make changes while working on-screen with the mouse. The Button Bar and Top Windows are discussed in Chapter IV, **Using the Button Bar and Top Windows.**

Help

Context-sensitive information is available at any time by pressing F1. Also most windows in ETABSIN contain a **Help** menu item that give access to the help file.

Chapter II

Making a Building

Making a Building means defining and laying out all of the elements that make up a building model. There are several steps involved. Remember, that ETABSIN does not analyze the Building, but only helps in preparing the input data file for analysis by ETABS by allowing you to graphically model the building.

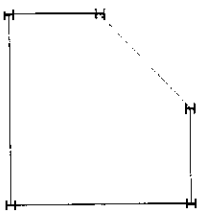
After reading this Chapter, please do the Tutorial, which is the best way to get an overview of the process.

The process of making a Building is described on the following pages. It assumes that the Building is made of a single Frame, at a single Frame Location.

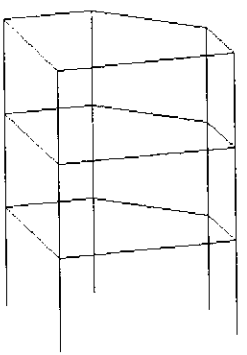
Make the Layout

The Layout allows definition of an **Assign 3D** view on which you can place the Structural elements such as Columns and Beams. The Layout is made by defining Column Lines and Beam Bays on the **Layout 2D** view of the Frame at the Baseline (Elevation 0). When you define Stories, the **Assign 3D** view shows the Column Lines as vertical lines from the Baseline to the top Story, and the Beam Bays as horizontal lines at each Story, connecting the Column Lines.

In the left figure, a **Layout 2D** view at the Baseline, the Column Lines are shown as I sections and the Beam Bays as lines.



In the right figure, an **Assign 3D** view of the same Layout with 3 defined Stories, the Column Lines and Beam Bays are shown as lines.



The Layout can be made in the traditional way, by specifying the location of each item in a Spreadsheet, or in a graphical way, on-screen, using the mouse.

Stories are defined only in a Spreadsheet.

Define Properties

The Structural Elements that you will place on the Layout require specification of their Material and Section properties. The process is as follows:

- Define a set of Materials, each with the desired specifications.
- Use this set of Materials to define a set of Properties for each of the Structural Elements you will be Assigning to the Frame Layout, such as Columns, Beams, Braces, Panels and Floors. Each of the Properties you define will use one of the defined Materials.

The definition of Properties is done by entering values in Spreadsheets.

Assign the Structural Elements

Using the Properties for the type of Structural Element you want to Assign, you can Assign an Element on the **Assign 3D** view by clicking the mouse. You can also Assign elements by specifying the location and Property in a Spreadsheet. Using the Spreadsheet has an advantage in that you can easily Assign multiple elements that all use a single Property.

Specify the Loading

There are three fundamentally different types of Loading:

- The first type is treated exactly like Structural Elements. You first define a set of Load Patterns (the equivalent of Structural Element Properties), each of which has Load magnitude and orientation. Then Assign the Patterns to the Layout using the mouse or Spreadsheet. These Loads are Joint, Beam and Floor Loads, and just like Structural Element Assigns, there can be an entirely different set of Load Patterns and Assigns for each Frame.

- The second type applies Static Lateral Loads to the entire Building and can not be defined graphically. This includes Wind Static loading.

- The third type applies Seismic Static Loads or Dynamic Loads, either as a Response Spectrum or as a ground acceleration Time History. This requires Mass definition, which applies to the entire Building. Mass definition can be done using a Spreadsheet, or using the mouse in a special **Layout 2D** view of the entire Building.

Chapter III

Using the Spreadsheets

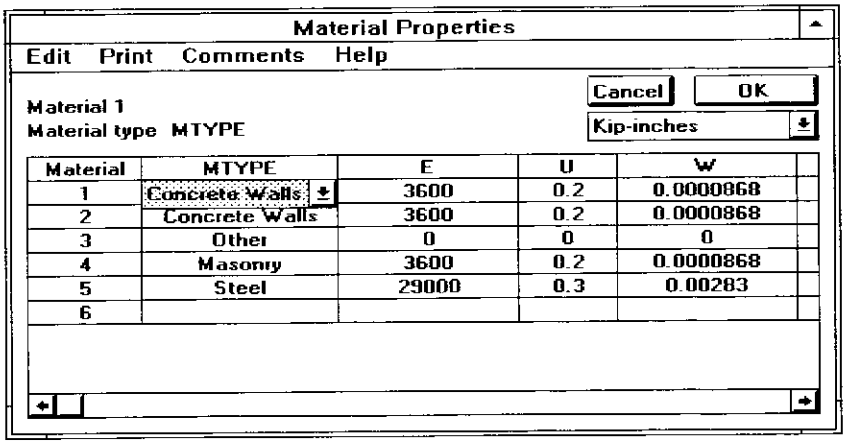
Spreadsheets are used throughout ETABSIN to hold tables of Layout dimensions, Properties, Assigns and Loads. With the Spreadsheets, you can make a Building in the traditional way, using numbers and the keyboard, without ever using the graphical features of ETABSIN.

All Spreadsheets work in a similar way, though they may be different sizes and contain different items.

All Spreadsheets are accessed from items in the menu bar. Only one Spreadsheet can be on-screen at a time, and no changes are made to the display until the Spreadsheet is closed by clicking on the **OK** button.

Each cell in a Spreadsheet has a specific type of item that it can contain, and this can not be changed. Further, no columns can be added to or deleted from a Spreadsheet and no formulas are allowed. Therefore, a Spreadsheet in ETABSIN is not a general purpose Spreadsheet, but is a table of specific items which can be edited.

The following pages describe the use of a typical Spreadsheet, the Material Properties Spreadsheet.



Material	MTYPE	E	U	W
1	Concrete Walls	3600	0.2	0.0000868
2	Concrete Walls	3600	0.2	0.0000868
3	Other	0	0	0
4	Masonry	3600	0.2	0.0000868
5	Steel	29000	0.3	0.00283
6				

This is a Material Properties Spreadsheet with five Materials defined. The selected cell is row 1, column **MTYPE**. All numeric items are shown in the current units, Kip-inches.

Select a cell by clicking on it or using the cursor keys. A description of the contents of the cell will be displayed above the Spreadsheet.

There are two types of cells, List and Number. List cells, once selected, have a Drop down selection list box to allow you to pick one item from a list. Number cells, once selected, allow you to type in a number. The selected cell in this Spreadsheet is a List cell.

Edit a cell by selecting it using the cursor keys and pressing the **Enter** key **or** by clicking on the cell and pressing the **Enter** key **or** by double-clicking on the cell.

The cursor keys now move within the cell. A green check mark and a red cross will appear. When you have selected the value you want in a List cell or have typed in a value in a Number cell, click on the green check mark (or press the **Enter** key) to accept the new value, or click on the red cross mark (or press the **Esc** key) to return to the original value.

Copy or Delete entire rows or groups of rows using the Windows clipboard. Select rows by clicking or dragging in the left (gray) column. The selected rows will be highlighted. Then use the Edit menu to perform the operation.

Cut copies the selected rows to the clipboard and removes them from the Spreadsheet.

Copy copies the selected rows to the clipboard.

Delete removes the selected rows from the Spreadsheet and does not affect the clipboard.

Paste puts the contents of the clipboard into the Spreadsheet ABOVE the currently selected cell.

Undo cancels the last operation done and only works once.

Change units for all items by clicking on the Units box, which drops down a list of available units and allows selection of one set.

Exit the Spreadsheet by clicking on the **OK** button.

Clicking on the **Cancel** button will exit the Spreadsheet and cancel ALL changes made. This returns the model to the state it was in when you first opened the Spreadsheet.

Chapter IV

Using the Button Bar and Top Windows

Using the mouse, you can make the Layout and then Assign Structural Elements and Frame Loads on-screen. The Button Bar and Top Windows are used to control what is shown on the screen and to determine what the mouse does.

When you are working on a Frame, the screen and mouse are in either the **Layout mode** or in the **Assign mode**:

Layout mode displays a 2D view of the Frame at the Baseline. It shows only Layout items, Grids, Column Lines, Beam Bays and Floor Bays. It NEVER shows any assigned Structural Elements. The mouse can be used to create and edit Column Lines, Beam Bays and Floor Bays with the use of Top windows.

Assign mode displays a 3D view of the Frame. It will display all Layout items except Grids, and all Assigned Structural Elements and Loads. The mouse can be used to create and edit any Assigned Structural Elements or Load using the Top windows. The mouse can NOT be used to create or edit any Layout items.

Button Bar

The Button Bar contains only buttons, which all work by clicking on them. The buttons are in two groups as follows.

The **Layout 2D** and **Assign 3D** buttons work together as a pair. If one is selected, then the other is de-selected. The selected button is high-lighted.

If you Click on a selected button, it will either display or hide a Top window. That is, if the **Assign 3D** button is selected, then if you click on **Layout 2D**, Layout will be selected. If you click on **Layout 2D** again, the **Layout** Top window will be displayed. If you click on **Layout 2D** again, the **Layout** Top window will disappear. The **Assign 3D** button works in the same way.

When **Layout 2D** is selected, then the screen and mouse are in **Layout mode**. When **Assign 3D** is selected, then the screen and mouse are in **Assign mode**.

Clicking on **All** draws the entire Frame so it all fits on the screen. Clicking on **Limits** draws the portion of the Frame defined by the **Limits** Dialog Box which is accessed in the **Display** menu. The last selected button, **Limits** or **All**, is high-lighted.

Clicking on **Redraw** redraws the screen to show changes made to the items displayed since the screen was last drawn. **Redraw** is faster than either **Limits** or **All**.

Clicking on **Show** hides or displays the **Show** top window, which allows you to choose which items are displayed. When Show items are changed, click on **Redraw** to display them.

Clicking on **Zoom Pan** hides or displays the **Zoom Pan** top window. Zoom and Pan in the Top window until the view is what you want, and then click on **Redraw**. The maximum Zoom out is the view made by **All** or **Limits**.

The **3D View** button only appears in **Assign mode**. Clicking on the **3D View** button hides or displays the **3D View** Top window. Use the scroll bars in the Top window to select the View angles that you want and then click on **All** or **Limits**.

Top Windows

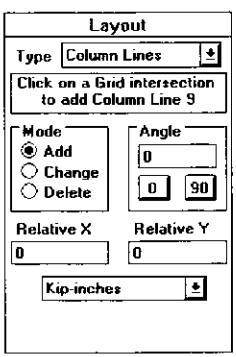
Top Windows ALWAYS stay on Top of the ETABSIN screen. Their purpose is to allow you to make changes while working on-screen with the mouse.

Top Windows are displayed and hidden by clicking on buttons in the Button Bar. The following buttons operate Top Windows:

Layout 2D, **Assign 3D**, **Show**, **Zoom Pan** and **3D View**.

Layout Top Windows

There are 4 Layout Top windows, Column Line, Beam Bay, Floor Bay and Edit. The one you see depends on the setting in the Type drop down selection list box. The selection box in the figure below is set to Column Lines thus the **Column Line** Top window is showing:



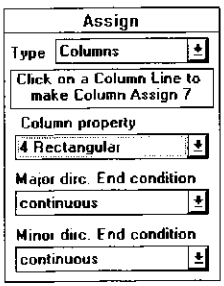
If you have previously defined a Grid, then when this window is visible, you can:

Add a new Column Line by clicking at a Grid intersection, with X,Y and Angle relative to the Grid axes.
Change an existing Column Line at a Grid intersection, by clicking on it and then on its new location.
Delete an existing Column Line by clicking on it.

Switch to one of the other Layout Top windows by using the Drop down selection list box. The other windows are similar, except for Edit. The Edit window is described in Chapter V, **Using the Menus**, under the **Layout** menu, **Layout Edit** item.

Assign Top Windows

There are 10 Assign Top windows, Joint, Column, Beam, Floor, Brace, Panel, Link, Joint Load, Beam Load, and Floor Load. The **Column** Top window is shown here:



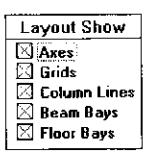
When this window is visible, you can click on any Column Line (the vertical line between two Stories) to Assign a new Column with the Property and End Conditions you have set.

To delete an existing Column, set the property to **0 nothing** and click on the existing Column.

The other Assign Top windows are similar. Use the Type drop down selection list box to select the one needed.

Show Top Window

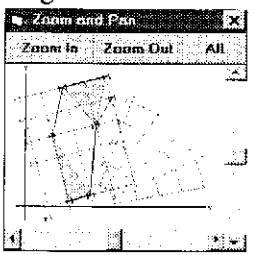
There are two Show Top windows. The one you see depends on whether **Layout 2D** or **Assign 3D** is selected. The **Layout Show** Top window is shown here:



You can select any or all items. When you click on **Redraw**, **Limits** or **All**, the screen will be drawn with these Show settings.

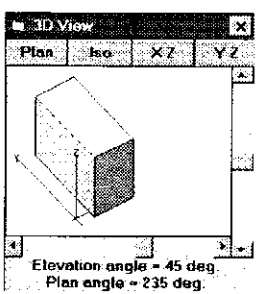
Zoom Pan Top Window

There is only one **Zoom Pan** Top Window, which shows the current screen in a reduced view. Clicking on the **Zoom In** or **Zoom Out** buttons will change the magnification of the reduced view. **All** will show the full structure based on the current limits setting. Clicking on the Scroll bars will scroll the reduced view, if it is not zoomed all the way out (the maximum Zoom Out is similar to All).



When the view is set to your liking, then click on **Redraw** to draw the screen. Do not click on **Limits** or **All** on the button bar as they will return to maximum Zoom Out, both for the screen and the reduced view.

3D View Top Window



There is only one **3D View** Top window, and the **3D View** button and window only appear when **Assign 3D** is selected. This window allows you to select the 3D view angles. Use the Scroll Bars to change the Plan and Elevation angles. When the angles are to your liking, then click on **All** or **Limits**, whichever you did last. You can also click on **Redraw**, but the display may have rotated out of view. The **Plan**, **Iso**, **XZ** and **YZ** buttons provide a quick method of changing the view.

Chapter V

Using the Menus

The menus allow you to Open and Save files, use the various spreadsheets, and control program settings and parameters. The menus do NOT directly control the screen or mouse, which must be done from the Button Bar. Even if you change the Colors using the menus, it does not take effect until the **Redraw** button is clicked.

New erases any existing Building completely.

Open displays the standard Windows File Open dialog box so you can select the File to Open. Before reading in the File, any existing Building is completely erased.

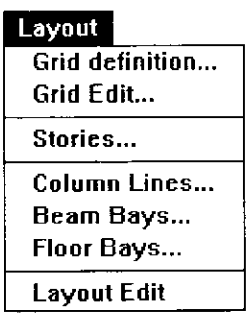
Save writes the existing Building to the File shown in the Title Bar, which overwrites the existing File, destroying its previous contents.

Save as displays the standard Window File Save dialog box so you can select the File you wish to Save, or create a new File. The existing Building is written to the File you select, overwriting the File (if it exists) and destroying any previous contents.

Header displays a dialog box where you can enter two Heading Lines and up to ten comment lines, all of which will be written at the beginning of any Saved File.

Maximum Sizes displays a dialog box where you can change the Absolute Maximum number of Properties, Assigns and Loads in a model. **This should be used sparingly**, as it is meant to change the program capacity to match your available memory. Save your Building before changing Maximum Size, as any existing Building will be erased. The smaller the Maximum Sizes, the faster the program will be when it first starts, files are Opened, or New is done.

Exit ETABSIN closes ETABSIN, but will give you a chance to Save your Building, if any changes have been made.



Grid Definition displays a dialog box allowing you to create new Grids and change existing Grid definitions. Grids are used to place Column Lines on-screen. The advantage of on-Grid Column Lines is that they move with the Grid. All Grids belong to the current Frame.

Grid Edit displays a Spreadsheet where you can edit any defined Grid by specifying its coordinates or grid spaces.

Stories displays a Spreadsheet which allows you to create and edit Stories for the Building. **There is only one set of Stories for a Building** and it is the same for every Frame. Stories should be one of the first things to be done, as addition or deletion of stories for a completed Building can be complicated if Assignments have been made.

Column Lines displays a Spreadsheet allowing you to create and edit off-Grid Column Lines. It also shows on-Grid Column Lines, but they must be edited on-screen, in the way they were created. On-Grid Column Lines may also include Column Lines offset from the Grid.

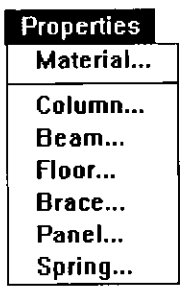
Beam Bays displays a Spreadsheet where you can create and edit Beam Bays as an alternative method to making them on-screen.

Floor Bays displays a Spreadsheet where you can create and edit Floor Bays as an alternative method to making them on-screen.

Layout Edit works in Layout mode only and on-screen only. This menu item simply tells how to do the edit. Layout Edit allows you to select any Grid and the

Layout items on the Grid. The selected items, including the Grid, can be copied to another location or deleted. This allows you to replicate the same structure in several places. Layout Edit also copies any assignments made to copied Column Lines and Bays, as long as they do not connect to another Grid. Whatever is done, all items belong to the current Frame.

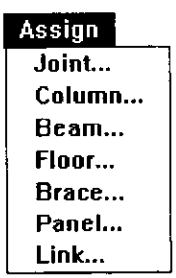
The same effect can be achieved by using multiple Frames, which is more powerful, since Frames can include multiple Grids and off-grid Column Lines and Assignments between Grids. However, the separate Frames must satisfy ETABS criteria for vertical connectivity isolation.



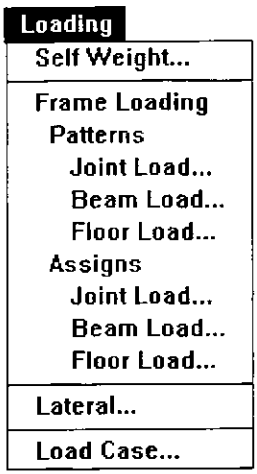
Material displays a Spreadsheet where you can create and edit the Materials used for the other Properties. Create your Materials BEFORE creating other Properties as deletion of a material or addition of a material in the middle of the list can cause problems.

Column, **Beam**, **Floor**, **Brace**, **Panel** and **Spring** all display Spreadsheets that allow you to create and edit the Properties. These Properties are used in Assignments, so create your Properties BEFORE starting to make Assignments as deletion of a property or addition of a property in the middle of the list can cause problems.

There is only ONE set of Properties for a Building, so every Frame uses the same Properties.



Joint, **Column**, **Beam**, **Floor**, **Brace**, **Panel** and **Link** all display Spreadsheets that allow you to create and edit Assignments. The advantage in using the Spreadsheet over on-screen Assignment is that you can generate multiple Assignments over many column Lines and Stories with a single Assignment entry.



Self Weight displays a dialog box where you can specify if Self Weight is to be included in the Loading.

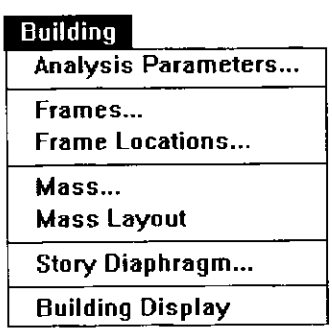
Frame Loading is Loading that can be different for each Frame. It is done in the same way as Structural Elements such as Columns and Beams. First you define Patterns (the equivalent of Properties), and then use those Patterns in making Assignments. Assignments can be made using a Spreadsheet or on-screen. There is one difference between Patterns and Properties, in that there is a different set of Patterns for each Frame, but there is only one set of Properties for the Building.

Joint Load, **Beam Load** and **Floor Load** (under **Patterns**) display Spreadsheets where you can create and edit Patterns. This can not be done on-screen.

Joint Load, **Beam Load** and **Floor Load** (under **Assigns**) display Spreadsheets where you can create and edit Assignments. This can be done on-screen as well. The advantage of the Spreadsheet over on-screen is that you can create multiple Assignments over many Column Lines and Stories with a single entry.

Lateral brings up a dialog box allowing you to define Lateral Loading that is applied to the Building. This includes Static Seismic and Wind Loading as well as Dynamic Response Spectrum and ground acceleration Time History Loading.

Load Case displays a Spreadsheet that allows you to create and edit load combinations from the individual Load Conditions already defined.



Analysis Parameters brings up a dialog box where you can set various parameters that apply to the Building like Rigid End Offset and P-Delta.

Frames displays a dialog box which allows you to create and delete Frames and to select the Current Frame. When you are working on the Model, you are working on the Current Frame. All other Frames are saved in temporary files on the hard disk.

Frame Locations displays a Spreadsheet where you can specify the Location of your Frames. A single Frame can be Located at several Locations, which is the same as duplicating it at these Locations.

Mass displays a Spreadsheet allowing you to create and edit off-grid Masses. It also shows on-Grid Masses, but they must be edited on-screen, in the way they were created.

Mass Layout changes the screen to display Mass Grids and Masses. This display can show the entire Building, where normal Layout mode and Assign mode always show just the Current Frame.
When you are in Mass Layout, use the Button Bar and Top Windows just like when you Layout Column Lines. In this case, you can only Layout Masses, and the display only shows Grids and Masses.
To get out of Mass Layout display, click on **ExitMassLayout**.

Story Diaphragm brings up a Spreadsheet where you can create and edit Diaphragm assignments. The diaphragms can have mass properties and external spring properties. The mass properties could be a Mass already defined in **Mass Layout**.

Building Display switches to a Layout view of the entire Building. This displays ONLY Layout items; Grids, Column Lines, Beam Bays and Floor Bays in a 2D view of the Building at the BaseLine. You can NOT make any changes to the Building. This view is for display only.

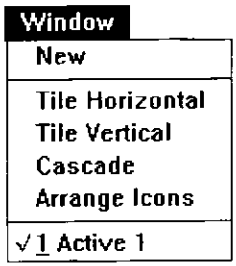
Numbers displays a dialog box which allows you to determine whether or not numbers are displayed for screen items such as Column Lines and some Assigns. This applies to both **Layout mode** and **Assign mode.**

Grids displays a dialog box which allows you to select which Grids are displayed for the current Frame.

Limits brings up a dialog box where you specify limits that are used when you click on the **Limits** button in the Button Bar. Global X and Y limits can be set, which apply to both **Layout and Assign modes.** Story limits can also be set which apply to **Assign mode** only.

Layout Colors and **Assign Colors** bring up dialog boxes allowing you to set the colors for **Layout mode** and **Assign mode.** You can also set printer colors.

Redraw to Printer is exactly like clicking on the Redraw button in the Button Bar, except it will draw on the default printer.



In ETABSIN, it is possible to have up to four views (windows) of the Current Frame at once. **This can be very confusing until you get used to it.** For new users, we recommend that you only use one window, which is always active.

The view windows can be either Layout or Assign, and the windows always remain inside of the ETABSIN main window. The windows can be minimized or maximized. This is a Windows MDI (Multiple Document Interface) application just like the Windows File Manager.

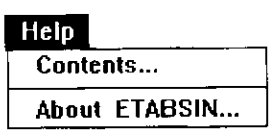
New creates a new view window, if there are less than 4, and makes it the same view as the previously active window.

Tile Horizontal and **Tile Vertical** fit all windows into the ETABSIN window next to each other, so they all show.

Cascade displays all windows in a medium size, so you can see them all (sometimes windows get behind one another, and seem to be lost!).

Arrange Icons arranges the window Icons if any view windows are minimized.

1 Active 1 is a list of the view windows. In this case there is only one window, number 1, and it is active. If there were more windows, they would all be listed here, and clicking on a window makes it active (on Top). You can get this same effect by clicking anywhere inside of an inactive window on-screen. However, sometimes one window will be hidden from view, so you can make it the active window using this menu.



Contents brings up Window Help, set at the ETABSIN Contents. Click on an underlined phrase to jump to that topic. Click on the Back button to go back after a jump. Context-sensitive help is also available at any time by pressing F1.

About ETABSIN brings up a dialog box which has information about ETABSIN, including the location of the files used.

Chapter VI

Tutorial

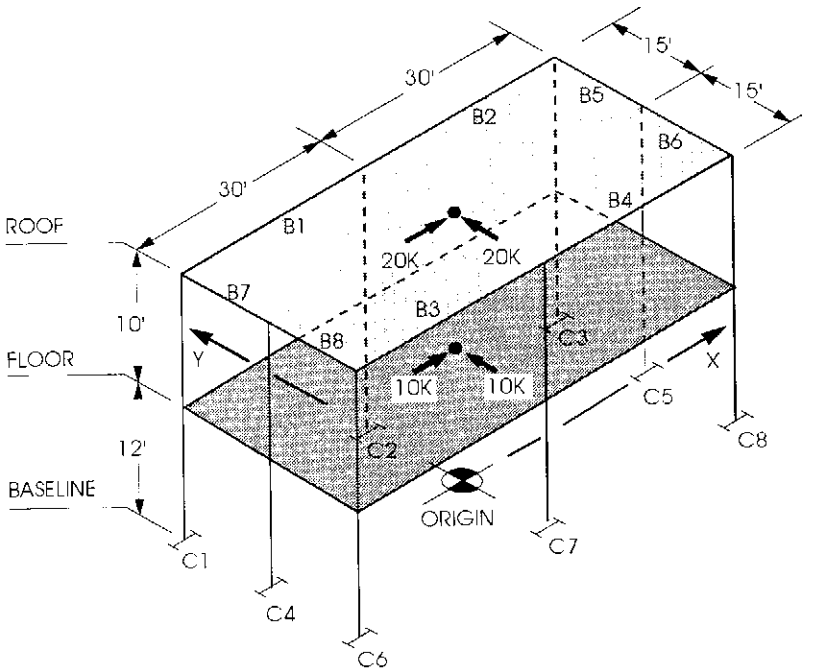
This tutorial defines the step by step procedure to produce an ETABS model for a simple structure. The tutorial will introduce you to the basic options of ETABSIN. It is recommended that you read Chapters III and IV of the ETABS User's Manual to be familiar with the ETABS terminology before trying this tutorial.

The tutorial guides you through the generation of the structural model and the lateral loads for the example structure shown in Figure VI-1. The completion of the tutorial will produce a text file that contains the input data required to execute ETABS for the analysis of the model generated in ETABSIN.

It is assumed that you are familiar with the Windows terminology, starting Windows applications and the use of the mouse and the keyboard in association with Windows.

The exercise assumes that ETABSIN has been installed on your computer according to the instructions given in Chapter I.

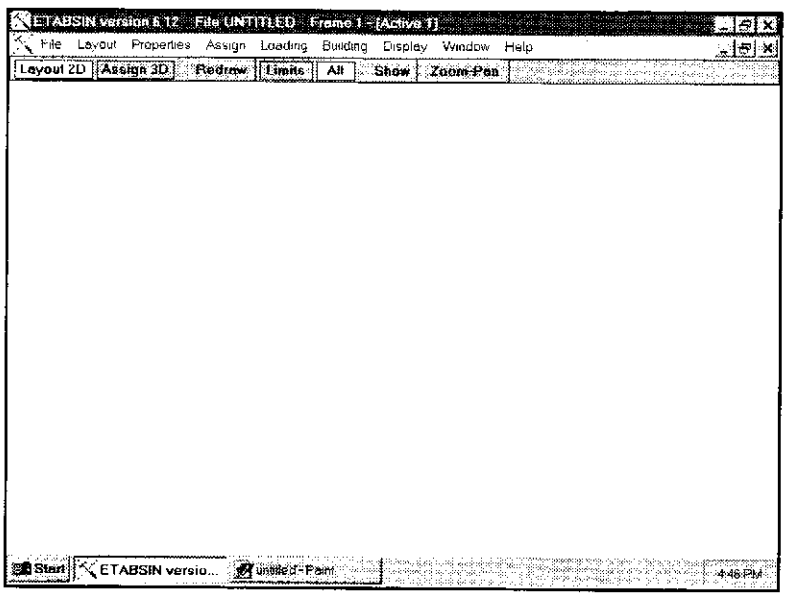
It is possible that the screens shown in the tutorial appear slightly different from what they appear on your screen. This can be due to different screen resolution and font settings on your computer. The working of the program, however, would be the same.



- ALL COLUMNS ARE W14x120
- ALL BEAMS ARE W33x118
- A36 STEEL
- FIXED BASE

EXAMPLE STRUCTURE
Geometry and Lateral Loading
Figure VI-1

In order to generate an ETABS model of the simple structure that we have chosen you will need to first start ETABSIN. The setup program adds a program group called CSI to the Windows Program Manager. The group contains an Icon for ETABSIN. The best way to start ETABSIN is to double-click on the ETABSIN Icon. The screen will initially display the ETABSIN window with an hourglass shape as the program initializes the arrays needed for modeling. If the program is being run for the first time (or if the Working Directory has been changed) the program will give a message that the ETABSIN.INI file cannot be read and that defaults have been used. This is all right, so click on **OK**. After the initialization is complete the mouse pointer (arrow) appears indicating the program is ready for modeling and the following screen will show:



This is the ETABSIN window as indicated by the title bar at the top. The next bar down (the menu bar) displays the ETABSIN main menu items. Each menu item has a corresponding pull down menu that is displayed and accessed by clicking on the menu item. The pull down menu has a list of commands. A particular command from the pull down menu can be selected by clicking on it. The next bar down (the button bar) displays a set of buttons that control the mode which ETABSIN is in and control the display.

Once ETABSIN has been started the following steps are required to interactively generate the model of the structure:

- Define architectural grid
- Define column line coordinates
- Define bay connectivity
- Define story heights and labels
- Define material properties
- Define column section properties
- Define beam section properties
- Assign column properties
- Assign beam properties
- Define lateral loading
- Define load cases (combinations)

The following subsections correspond to the eleven steps specified above. Each subsection defines in detail the procedures required to implement the associated step.

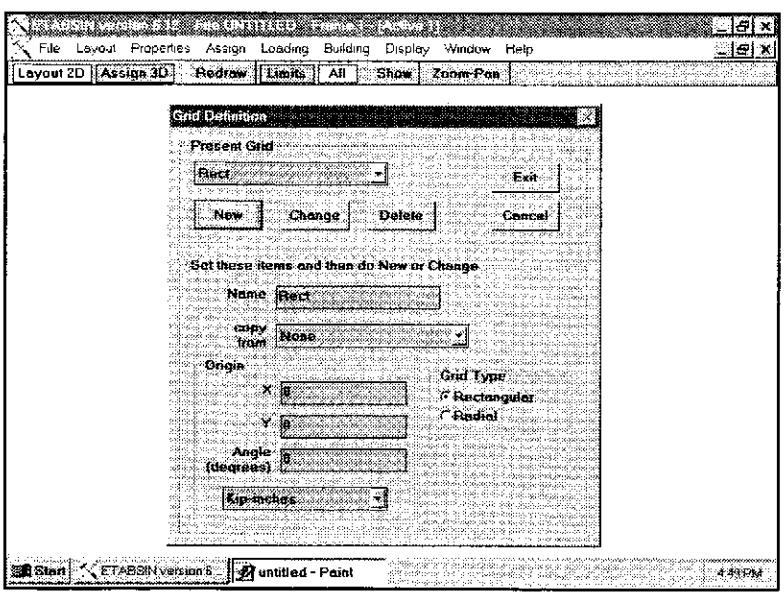
Defining Architectural Grid

Definition of the architectural grid is a two step process. First we need to define the type of the grid system and then we need to define the coordinates or spaces in the grid system. To define the grid do the following:

1. Click on **Layout** on the menu bar. The **Layout** pull down menu will appear.

2. Click on **Grid Definition** in the **Layout** menu. The **Grid Definition** dialog box will appear. It allows you to define a name, type, origin and angle of a grid system with respect to the global origin.

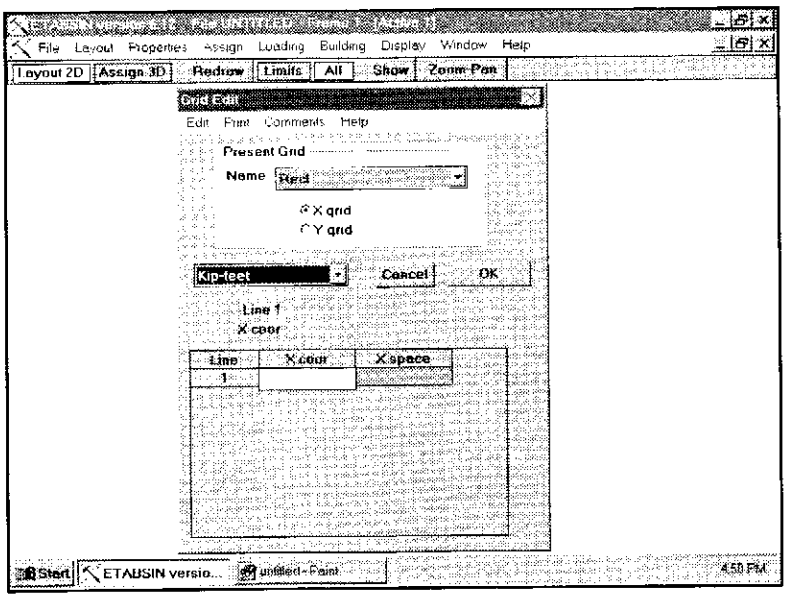
3. Type **Rect** in the edit box labeled **Name**. We will accept the defaults for the type of grid (rectangular), the origin (0,0) and the angle (0), so click on the button labeled **New**. This changes the **Present Grid** to **Rect** and the dialog box will look as follows:



The above steps have added a rectangular grid system Rect. If needed one can define several grid systems to build a model.

4. Now click on **Exit** to leave the dialog box.

5. Click on **Grid Edit** in the **Layout** menu. The following **Grid Edit** dialog box will appear:



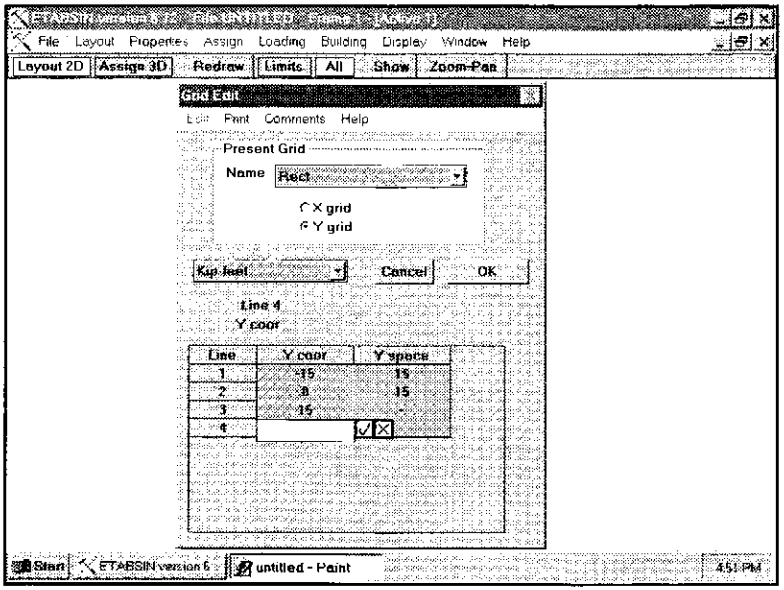
It allows you to define the coordinates and spaces in the X and Y directions.

6. We will input the grid coordinates in feet units. So click on the units box currently showing Kip-inches and select Kip-feet as the units.

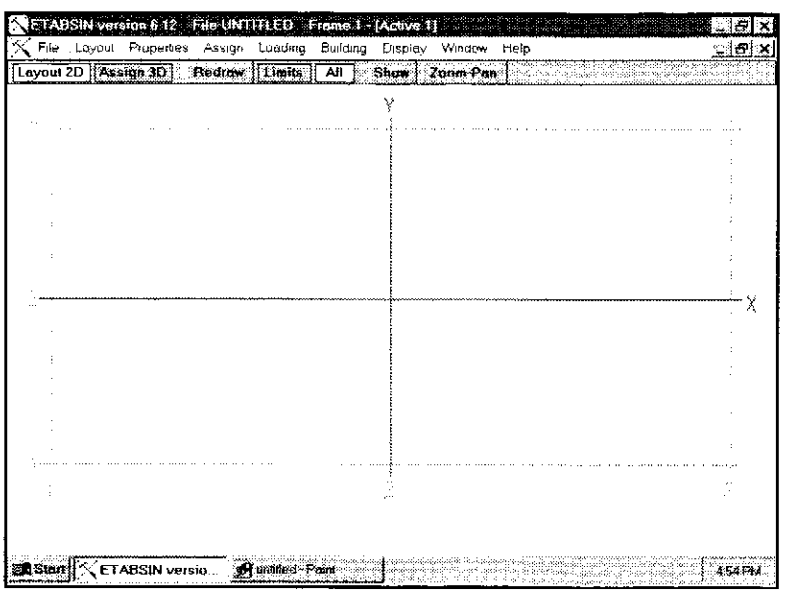
7. The grid coordinates are entered as one would do it in a spreadsheet. Click on the **X coor** cell for Line 1 to put it in the edit mode. Type **-30** as the location of the first X grid line. Hit Enter or select the check mark with the mouse to accept the value. Line 1 is added to the list and Line 2 will now show. Enter **0** for Line 2 and Enter **30** for Line 3. This will complete the X grid editing.

   It should be noted that once the X coordinate of the first grid line is defined, other grid lines can also be defined by entering spaces rather than coordinates.

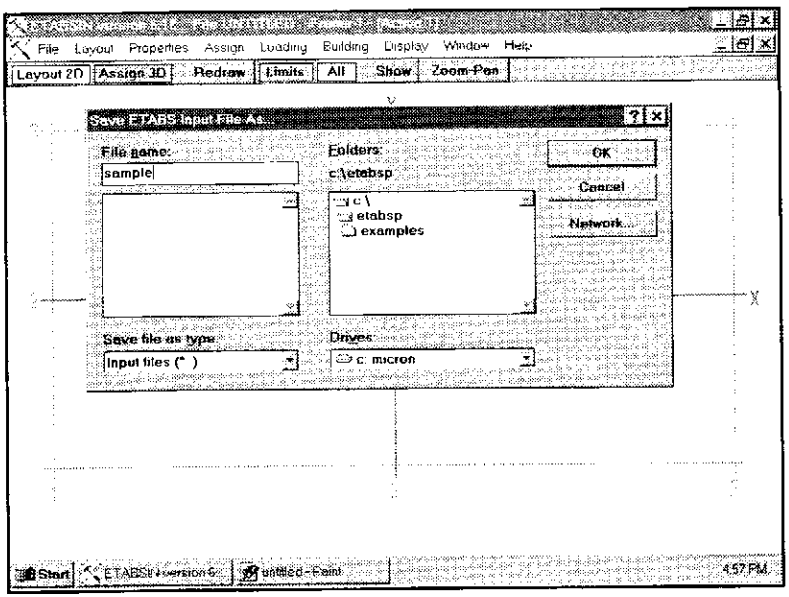
8. To enter the Y coordinates, first click on the radio button labeled **Y grid**, then enter the values similar to the X coordinates. Type **-15**, **0** and **15** for Y grid lines 1,2 and 3, respectively. The screen will now show as follows:



The editing of the grids is now complete. Click on **OK** to accept the values and close the **Grid Edit** dialog box. The program will now draw the X and Y grid and the screen will show as follows:



9. Before proceeding to the next item it is a good idea to save the model as it exists at this time. To save the model click on **Save As** in the **File** menu. The **Save ETABS Input File As** dialog box will appear. We will save the file under the name Sample, so type in **sample** instead of **untitled** showing in the **File Name** box. The screen will now look as follows:



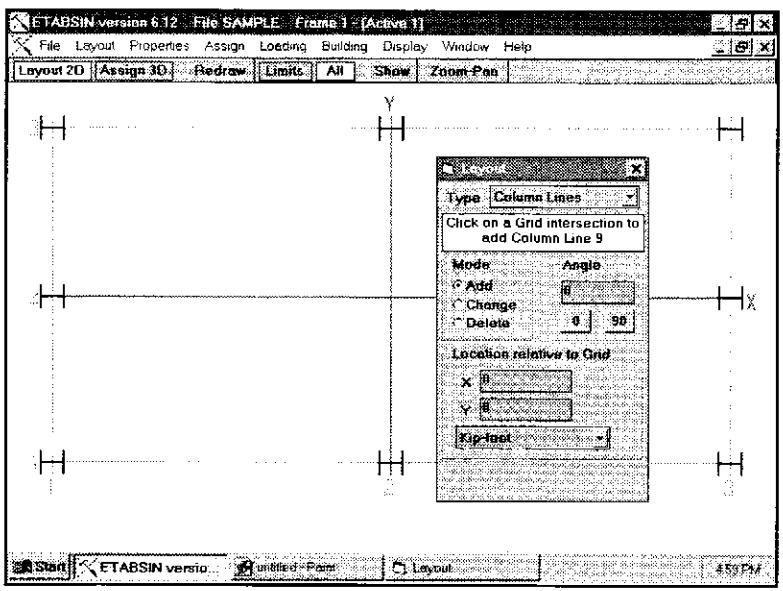
Click on **OK** to close the dialog box. The program will give you a chance to change the units in which the file is to be saved. There is no need to change the units at this time so click on **OK** to accept the selection of units. The program then saves the file with the name and in the directory specified. The title bar now shows this name for the model.

Defining Column Line Coordinates

The definition of column line locations can be done in ETABSIN two different ways - graphically, associated with the grids already defined, or by defining the coordinates in a spreadsheet. When column lines are defined associated with the grids, any change in the grids will automatically be reflected in the column line locations. We will define column line coordinates using the grids. The following steps are involved:

1. Click on **Layout 2D** button on the button bar. The **Layout** top window will appear. The **Layout 2D** button has two functions. If the program is in the assign mode selecting this button will put the program in the layout mode and change the display to a 2D plan view. If the program is already in the layout mode this button acts as a toggle to open and close the **Layout** top window.

2. Make sure the **Type** in the **Layout** top window is showing **Column Lines**. The setting for **Mode** should be **Add**, and the other edit boxes should all be showing **0**. Clicking on any grid intersection now will add a column line at that location. For display purposes the column lines are shown as I-sections, the web having the angle specified with respect to the X axis. So click on the perimeter grid intersections to add 8 column lines. The screen will now show as follows:

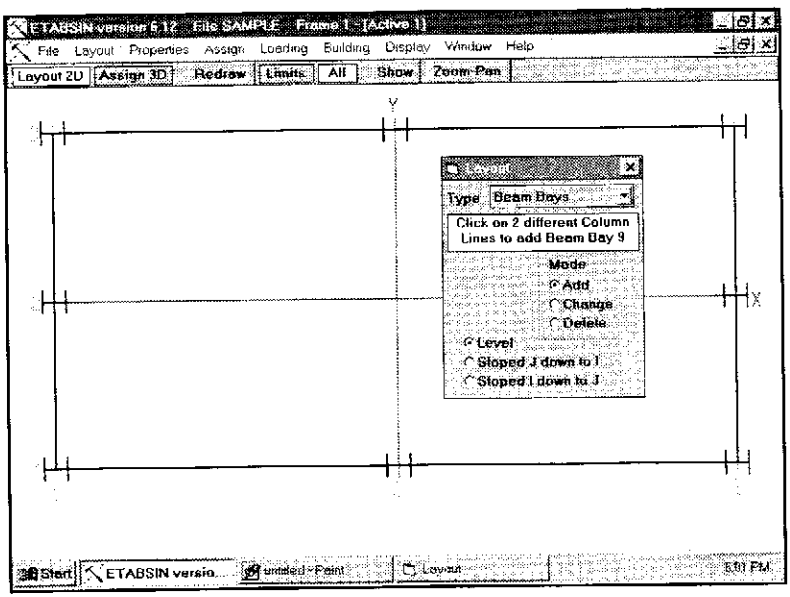


Defining Bay Connectivity

The definition of beam bay connectivity can be done in ETABSIN two different ways - graphically, or by defining the connectivity in a spreadsheet. The bays are always associated with the column lines and move with them if they are relocated. We will define bay connectivities graphically. The following steps are involved:

1. From the **Type** drop down selection list in the **Layout** top window select **Beam Bays**. If the **Layout** top window is not showing, then click on **Layout 2D** button first, to show it.

2. The setting for **Mode** should be **Add**, and the selection for slope should be **Level**. Clicking on any two column lines now will add a beam bay between them. The beam bays are shown between the column lines as they are added. So click on the perimeter column lines to add the 8 beam bays we have. The screen will now show as follows:



It is noted that the column lines and beam bays defined are automatically numbered sequentially. Usually their numbering is not important if the whole model is to be defined graphically, but for large structures it is helpful to be able to generate assignments. It is, therefore, recommended that a definite numbering system be followed. The numbering assigned by the program can be viewed by selecting **Numbers** under the **Display** menu and turning numbers for particular items on.

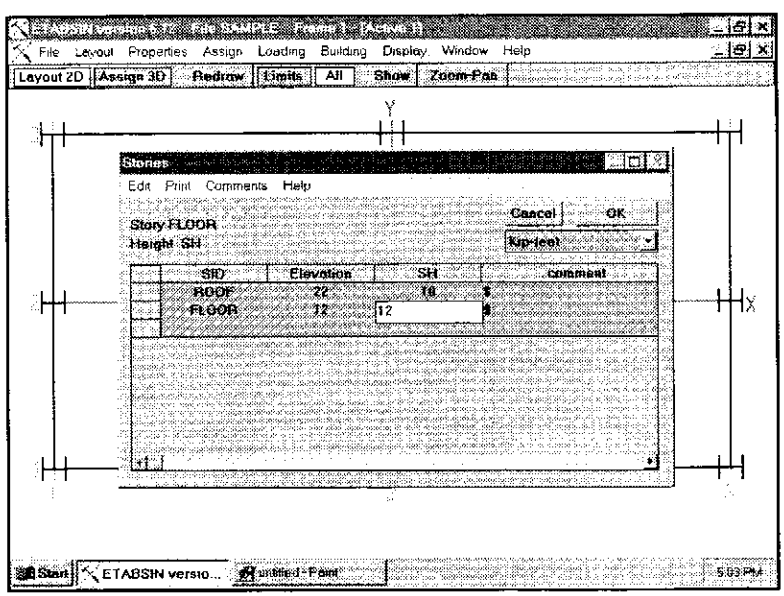
Defining Story Labels and Heights

The definition of story labels and heights in ETABSIN is done through a spreadsheet. To define story heights and labels do the following:

1. Click on **Stories** in the **Layout** menu. The **Story Properties** dialog box will appear. The story labels and heights are entered in its spreadsheet. The story labels are entered from the top going down.

2. Click on the cell labeled **SID** to put it in the edit mode. Type the label **Roof** and hit Enter or click on the check box to accept it. The program will accept this as the label for the top story and put default values for **Elevation** and **SH** (height). Change the **SH** (height) to **10** making sure the units box shows Kip-feet, otherwise change it to the feet unit.

3. Now add the next level down by typing **Floor** in the **SID** cell and hitting Enter. The program will now use the last defined value of height as the new default. Change the value to **12.** The screen will now show as follows:



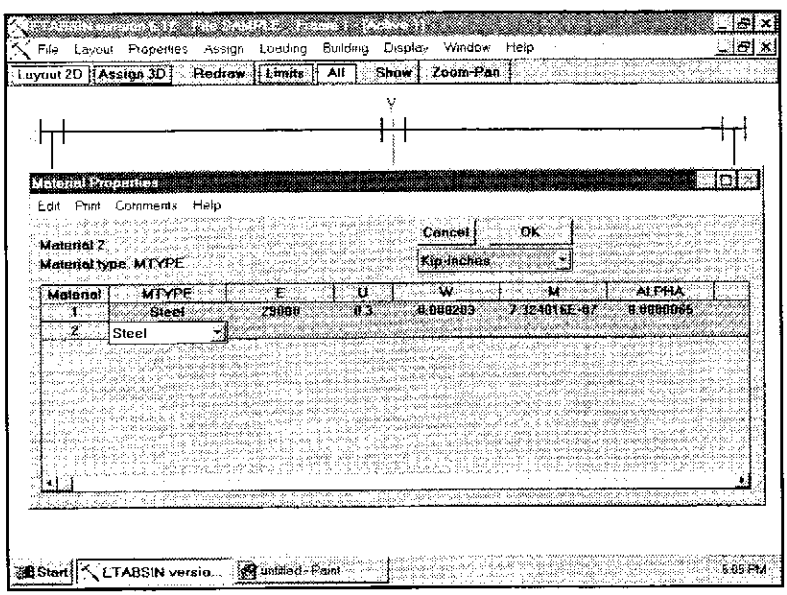
The story data has now been entered. Click on **OK** to close the **Story Properties** dialog box.

Defining Material Properties

The definition of material properties in ETABSIN is done through a spreadsheet. To define material properties do the following:

1. Click on **Material** in the **Properties** menu. The **Material Properties** dialog box will appear. The material properties are entered in its spreadsheet.

2. The materials are entered by selecting a material type, **MTYPE** from the drop down list box showing in the cell under **MTYPE**. Click on the box to show the list and select **Steel**. Accept this selection by either hitting Enter or by clicking on the check box. This will add steel as the first material with default values for all the properties and the program will allow a second material to be entered. If the units are now changed to Kip-inches, the screen will appear as follows:



The default values are built into the program but can be changed by editing the ETABSIN.INI file. Individual values can be changes on the spreadsheet. For our purposes we will accept the default values shown. The material properties data has now been entered. Click on **OK** to close the **Material Properties** dialog box.

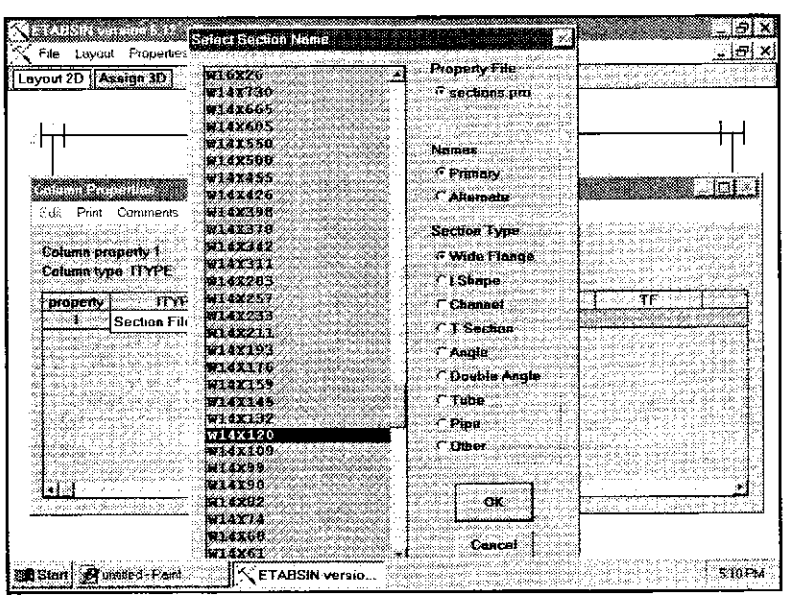
It is a good idea to click on **Save** in the **File** menu to save the model as it exists at this time. The model will be saved in the file called **Sample** and a choice will be given to choose the units. Kip-inch units are fine.

Defining Column Section Properties

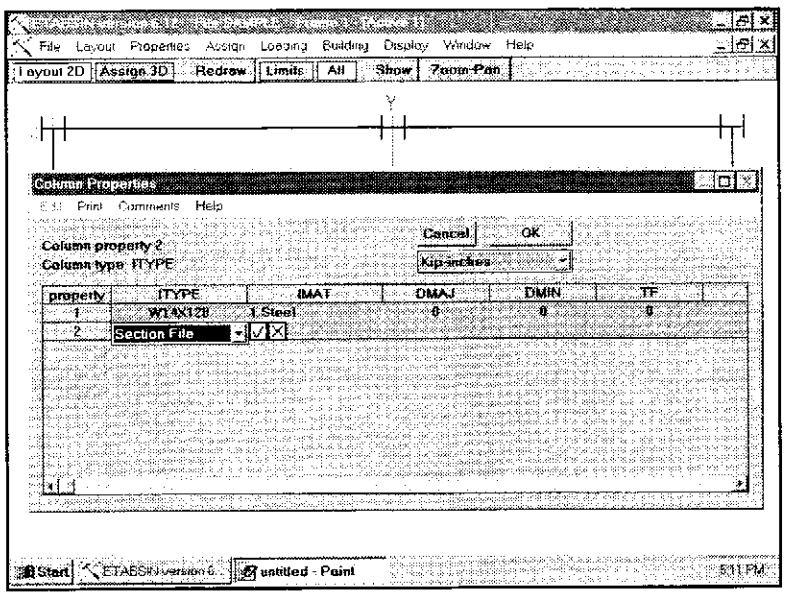
The definition of column section properties in ETABSIN is done through a spreadsheet. To define column section properties do the following:

1. Click on **Column** in the **Properties** menu. The **Column Properties** dialog box will appear. The column properties are entered in its spreadsheet.

2. The column properties are entered by selecting a section type, **ITYPE** from the drop down list box showing in the cell under **ITYPE**. Click on the box to show the list and select **Section File**. Accept this selection by either hitting Enter or by clicking on the check box. This will bring up a **Select Section Name** dialog box. It is possible, depending on what other section property databases (files with .PRO extension) are detected by the program, that a **Select User Section File** dialog box comes up first. Simply click on **None** without selecting a user file for this tutorial. From the **Select Section Name** dialog box select the label **W14x120**. Use the scroll bars if needed to get to this property. The screen should now look as follows:



3. Click on **OK** to select the label and close the **Select Section Name** dialog box. This will add the section **W14x120** as the first column property with a default material property number, **IMAT** of 1 and the program will now allow the entering of a second column section property. The screen will now look as follows:



Since the default material is acceptable, the column properties data has now been entered. Click on **OK** to close the **Column Properties** dialog box.

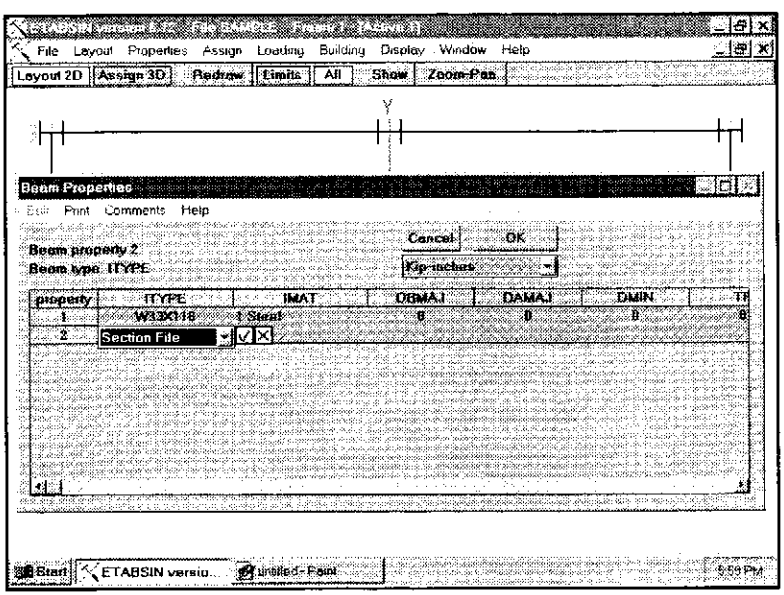
Defining Beam Section Properties

The definition of beam section properties in ETABSIN is done through a spreadsheet and is done identical to the way the column section properties are done. To define beam section properties do the following:

1. Click on **Beam** in the **Properties** menu. The **Beam Properties** dialog box will appear.

2. Click on the cell under **ITYPE** to show the drop down list and select **Section File**. Accept this selection by either hitting Enter or by clicking on the check box. This will bring up the **Select Section Name** dialog box. From the **Select Section Name** dialog box select the label **W33x118**. Use the scroll bars if needed to get to this property.

3. Click on **OK** to select the label and close the **Select Section Name** dialog box. This will add the section **W33x118** as the first beam property with a default material property number, **IMAT** of 1 and the program will now allow the entering of a second beam section property. The screen will now look as follows:

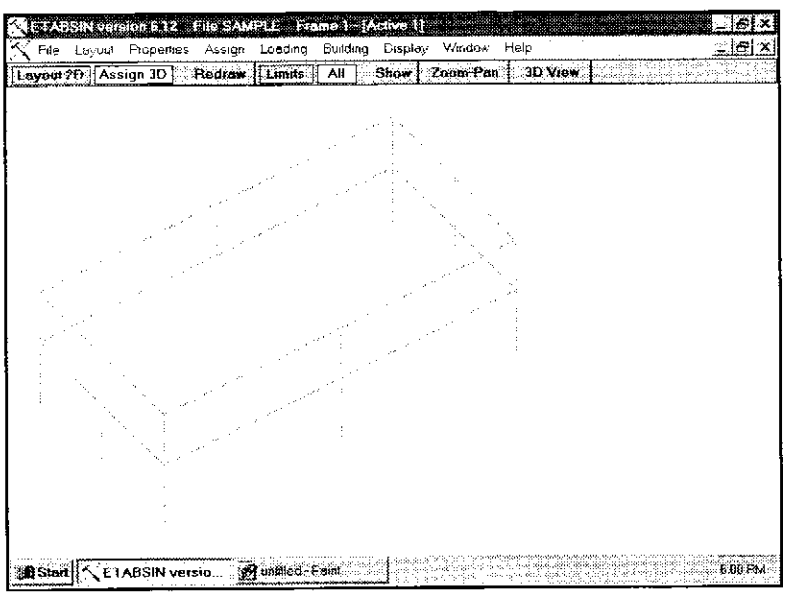


Since the default material is acceptable, the beam properties data has now been entered. Click on **OK** to close the **Beam Properties** dialog box.

Assigning Column Properties

The assigning of column properties to the column lines where columns exist can be done in ETABSIN two different ways - graphically, or by defining the assignments in a spreadsheet. We will define the assignments graphically. The following steps are involved:

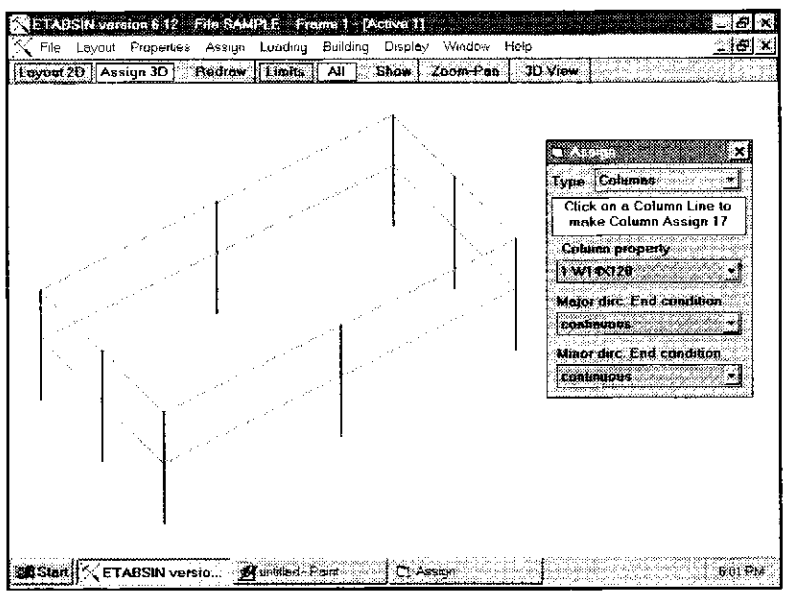
1. Click on **Assign 3D** button on the button bar. The **Assign 3D** button has two functions. If the program is in the layout mode selecting this button will put the program in the assign mode and change the display to a 3D view. If the program is already in the assign mode this button acts as a toggle to open and close the **Assign** top window. In our case the program was in the layout mode so clicking on **Assign 3D** button puts the program in the assign mode and the screen will look as follows:



2. Now click on **Assign 3D** button again so the **Assign** top window appears. From the **Type** drop down list select **Columns**. Assigning columns graphically simply involves clicking on the columns in the model after the correct properties and end conditions have been selected.

3. Click on the **Column property** drop down list in the **Assign** top window to select property **1 W14x120**. We will leave the end conditions as continuous for both the major and minor directions.

4. Once the items are all selected, click on each column to assign this property to it. The assigned columns with the same property will take a Magenta color. The message box in the **Assign** top window lets you see what would be the next assign number. When all assignments have been made the screen will look as follows:



The column assignment is now complete.

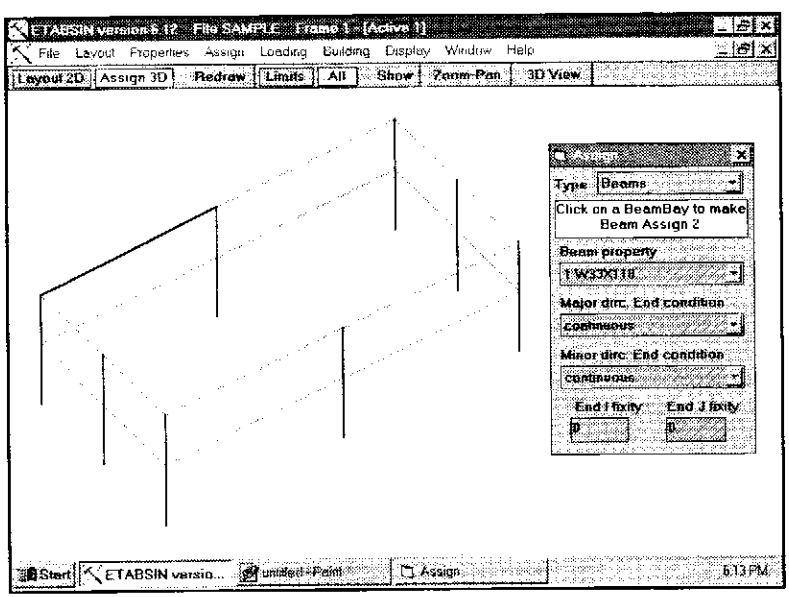
Assigning Beam Properties

The assigning of beam properties to the beam bays where beams exist can be done in ETABSIN two different ways - graphically, or by defining the assignments in a spreadsheet. For most assignments a combination of both methods is useful. We will use this combination for assigning beams. The following steps are involved:

1. From the **Type** drop down selection list in the **Assign** top window select **Beams**. If the **Assign** top window is not showing, then click on **Assign 3D** button first, to show it.

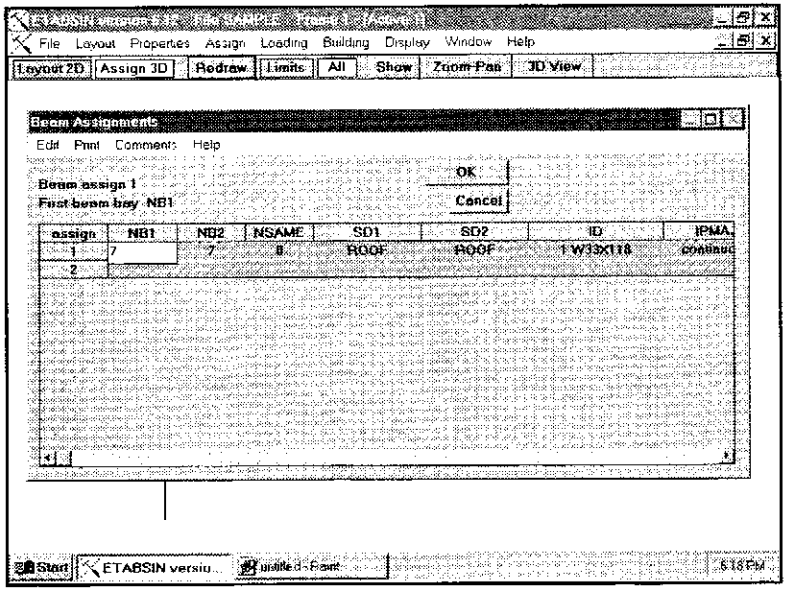
2. Click on the **Beam property** drop down list in the **Assign** top window to select property **1 W33x118**. We will leave the end conditions as continuous for both the major and minor directions.

3. Once the items are all selected, click on just one beam to assign this property to it. The assigned beam will take a Magenta color. The message box in the **Assign** top window lets you what would be the next assign number. The screen will look as follows:

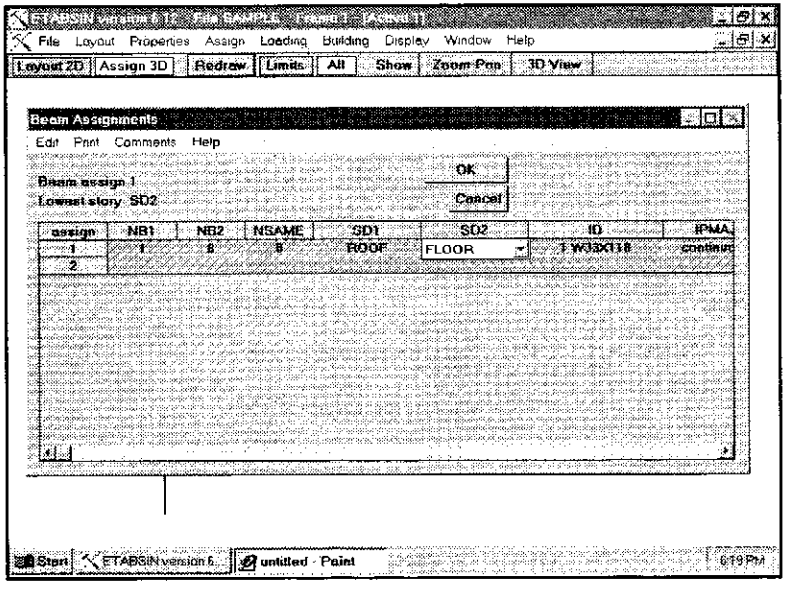


4. To generate assignments for the other beams, click on **Beam** in the **Assign** menu. The **Beam Assignments** dialog box will appear. One assignment

entry is already present. This could also have been entered directly into the spreadsheet. The screen looks similar to the following:

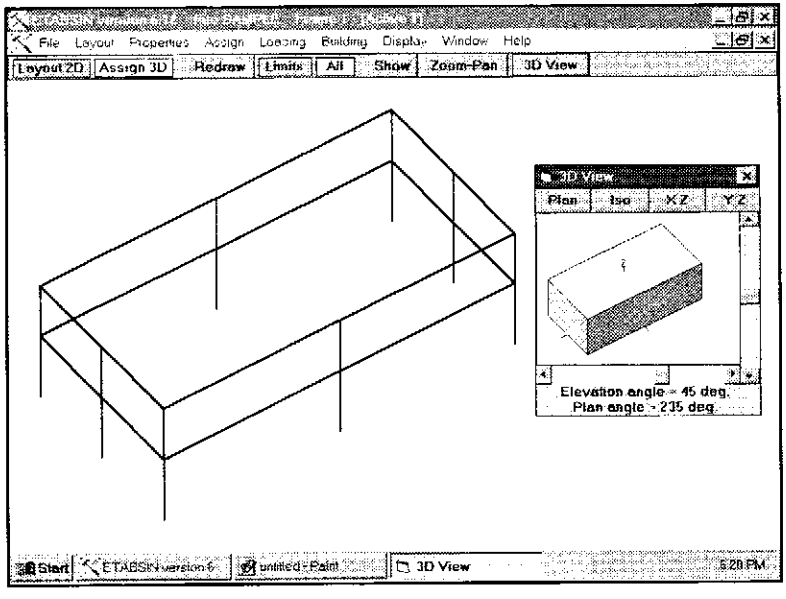


5. We will now modify the spreadsheet to generate all beam assignments. Change the entry for **NB1**, **NB2**, **SD1** and **SD2** to be **1**, **8**, **Roof** and **Floor**. The screen will now look as follows:



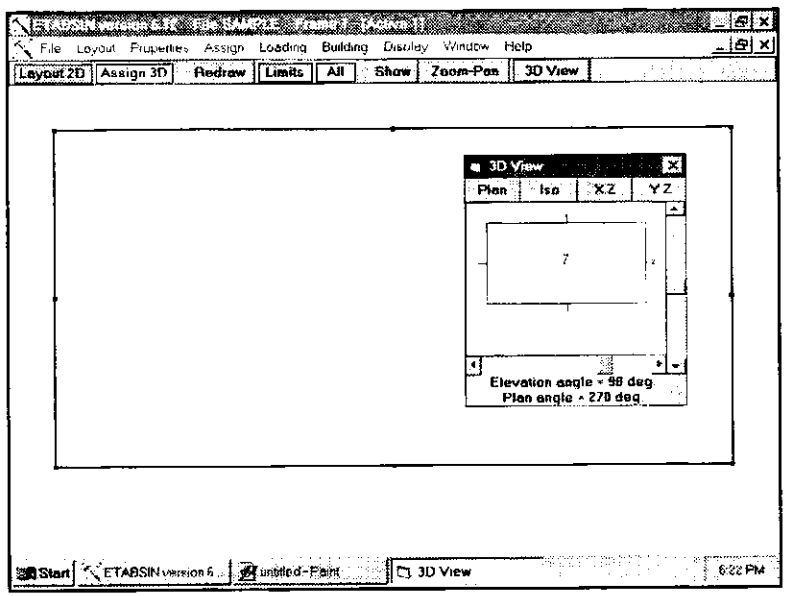
6. Click on **OK** to close the **Beam Assignment** dialog box. The screen will now show all beams as having been assigned.

We have done all assignments on the default view. It is possible to work in other views. To change the view click on the **3D View** button to show the **3D View** top window as follows:

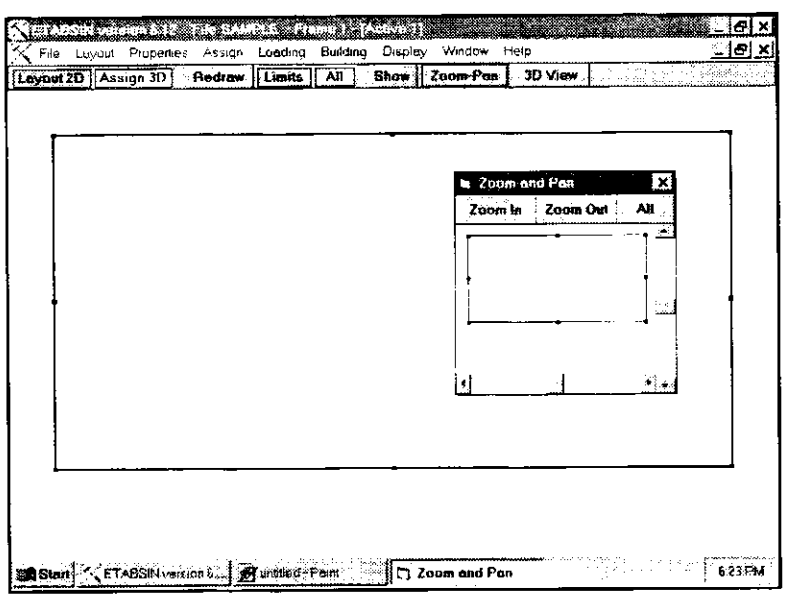


The scroll bars in the **3D View** top window can be used to change the view. Once the view in the top window is as desired, click on the **Redraw** button to redraw the display in the new view. The redraw is done at the current drawing scale. Clicking on the **All** button will rescale the display to show the full model. Similarly, clicking on the **Limits** button will rescale the display to show only the portion of the model within the plan and story limits set in the **Limits** dialog box under the **Display** menu.

If the elevation angle is set to 90 degrees and the plan angle to 270 degrees and then **All** button is clicked the following plan view of the model will appear:



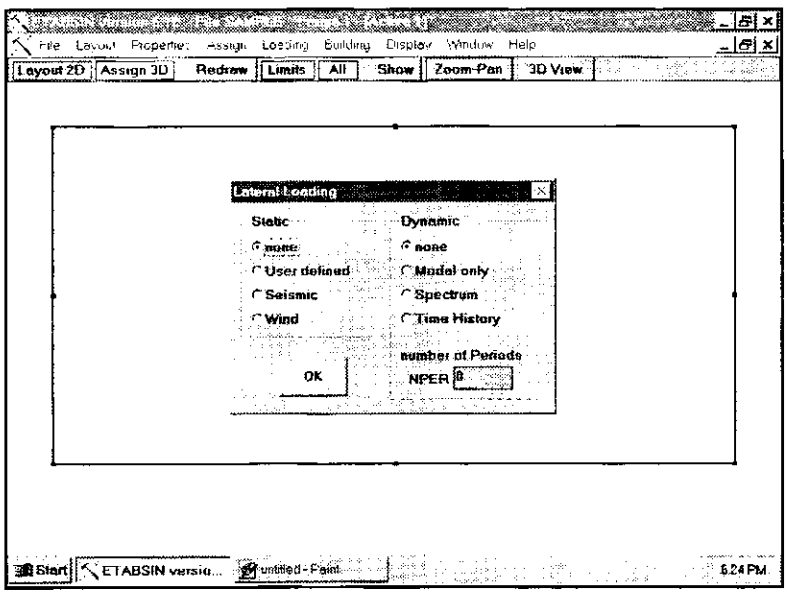
Similar to the **3D View** top window the **Zoom Pan** top window will allow you to set the magnification and pan the model to view different areas. Both of these buttons act as toggles to open or close the top windows. The **Zoom Pan** top window looks as follows:



Defining Lateral Loading

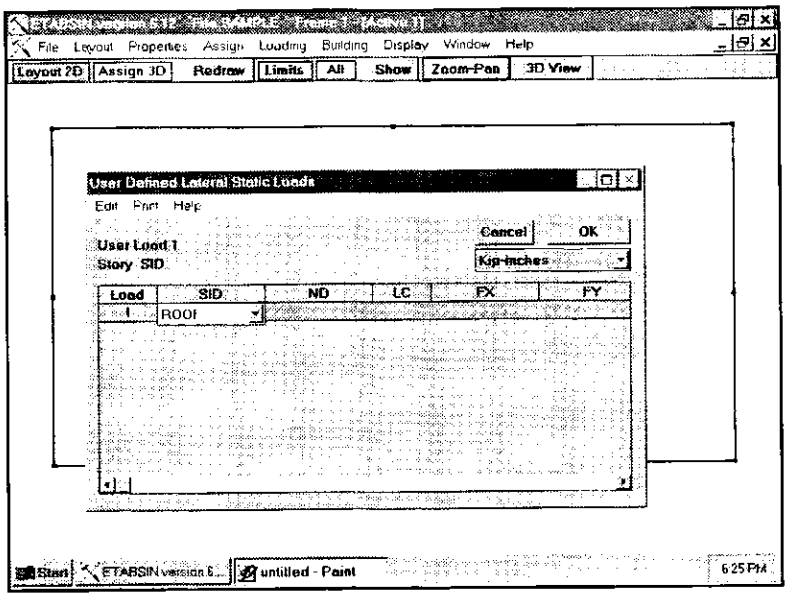
To define Lateral Loading on the structure do the following:

1. Click on **Lateral Loading** in the **Loading** menu. The following **Lateral Loading** dialog box will appear:



The **Lateral Loading** dialog box allows us to select the type of lateral loading to be applied to the structure. One type of static and one type of dynamic loading can be applied at one time. Selection of a type opens an appropriate dialog box to add the loading or select parameters for that type of loading.

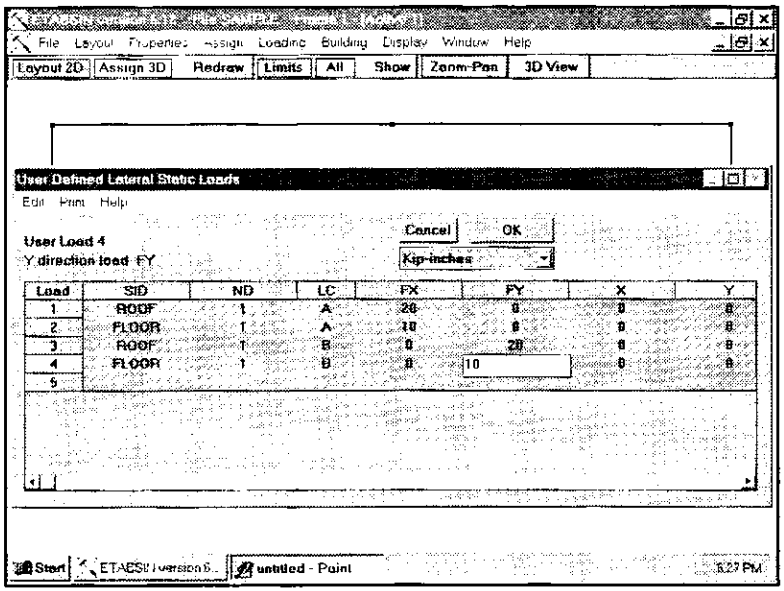
2. We want to specify the loading explicitly, so click on **User defined** under the Static heading. This opens the **User Defined Lateral Static Loads** dialog box and the screen looks as follows:



The lateral loads are to be entered in this spreadsheet.

3. We will enter the X direction loads in Load Condition A first and then the Y direction loads in Load Condition B. Also we will enter the loads from the top to the bottom. The order, however, is not important. So click on the cell under **SID** to drop down the list box showing the story labels. Select **Roof**. This adds a row to the spreadsheet with default values. It is faster to first add all the rows that we will need and then change any of the default values that we want. So add three more rows in the order **Floor**, **Roof** and **Floor**.

4. The items we need to change are the load condition, **LC** for rows 3 and 4 to B, to change the load in the X direction, **FX** in rows 1 and 2 to 20 and 10, respectively, and to change the load in Y direction, **FY** in rows 3 and 4 to 20 and 10, respectively. Once this is done the screen will look as follows:



The lateral load data has now been entered. Click on **OK** to close the **User Defined Lateral Static Loads** dialog box and again on **OK** to close the **Lateral Loading** dialog box.

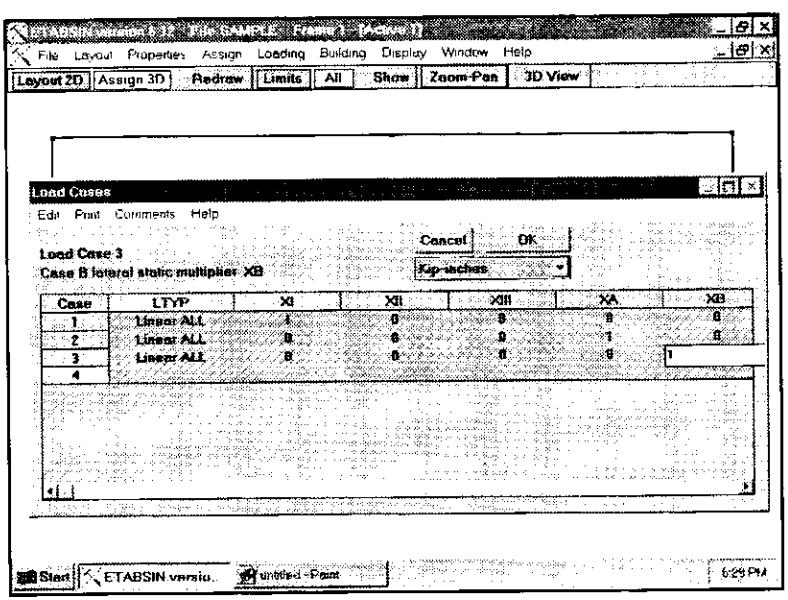
Defining Load Cases

We have provided lateral loads to the program for Load conditions A and B. Also the program defaults to putting self weight of the structure in Load condition I. This can be checked by looking at the **Self Weight** dialog box in the **Loading** menu. We will request data for these three load conditions separately. To define load cases (combinations) to the program do the following:

1. Click on **Load Case** in the **Loading** menu. The **Load Cases** dialog box will appear. The load combination multipliers are entered in this spreadsheet.

2. Click on the cell under **LTYP** to drop down the list of the different load combination types available. We will use Linear combination so select that. The program now adds a row with default values. We will add a total of three rows and edit them later.

3. The items we need to change are the multiplier for Load condition I, **XI** in row 1, the multiplier for Load condition A, **XA** in row 2, and the multiplier for Load condition B, **XB** in row 3 which all need to be changed to **1**. Once this is done the screen will look as follows:



The load case data has now been entered. Click on **OK** to close the **Load Cases** dialog box.

The model is now done. Click on **Save** in the **File** menu to save the model again. The model will be saved in the file called **Sample** and a choice will be given to choose the units. Kip-inch units are fine.

There is now a file named SAMPLE in the ETABS directory. If the file is viewed on the screen or printed it will appear as shown in Figure VI-2.

```
$ Control Data.    File SAMPLE saved 8/2/95 18:29:51 in Kip-inches
ETABS 6.1
Heading Data 1
Heading Data 2
 2 1 1 1 0 3 0 1 1 1 0 0 0 0 1 0 0 1 4 1 1
 386.4 0.0001 0 1
$ Story Data
 ROOF 120 0
 FLOOR 144 0
$ Material Property Data
 1 S 29000 0.3 0.000283 7.324016E-07 0.0000065 36 0 0 0
$ Column Property Data
 1 W14X120 1 0 0 0 0 1 1 1
$ Beam Property Data
 1 W33X118 1 0 0 0 0 0 1 1 1
$ Frame Heading and Control Data

 1 8 8 0 0 0 0 0 0 0 0 0 1
$ Layout Grids
! 1 Rect rect 0 0 0 3 3
!   -360 0 360
!   -180 0 180
$ Layout Column Lines
 1 -360 -180 0 ! 1 1 1 0 0 0
 2 -360 0 0 ! 1 1 2 0 0 0
 3 -360 180 0 ! 1 1 3 0 0 0
 4 0 -180 0 ! 1 2 1 0 0 0
 5 0 180 0 ! 1 2 3 0 0 0
 6 360 -180 0 ! 1 3 1 0 0 0
 7 360 0 0 ! 1 3 2 0 0 0
 8 360 180 0 ! 1 3 3 0 0 0
$ Layout Beam Bays
 1 1 2 0
 2 2 3 0
 3 6 7 0
 4 7 8 0
 5 1 4 0
 6 4 6 0
 7 3 5 0
 8 5 8 0
$ Column Assignment Data
 3 3 0 ROOF ROOF 1 0 0
 3 3 0 FLOOR FLOOR 1 0 0
 2 2 0 ROOF ROOF 1 0 0
 2 2 0 FLOOR FLOOR 1 0 0
 1 1 0 FLOOR FLOOR 1 0 0
 1 1 0 ROOF ROOF 1 0 0
 5 5 0 FLOOR FLOOR 1 0 0
 5 5 0 ROOF ROOF 1 0 0
 4 4 0 FLOOR FLOOR 1 0 0
 4 4 0 ROOF ROOF 1 0 0
 6 6 0 FLOOR FLOOR 1 0 0
 6 6 0 ROOF ROOF 1 0 0
```

File SAMPLE
Figure VI-2

```
 7 7 0 ROOF ROOF 1 0 0
 7 7 0 FLOOR FLOOR 1 0 0
 8 8 0 ROOF ROOF 1 0 0
 8 8 0 FLOOR FLOOR 1 0 0

$ Beam Assignment Data
 1 8 0 ROOF FLOOR 1 0 0 0 0

$ Frame Location Data
 1 0 0 0
$ User Defined Lateral Static Loads
 ROOF 1 A 20 0 0 0 0
 FLOOR 1 A 10 0 0 0 0
 ROOF 1 B 0 20 0 0 0
 FLOOR 1 B 0 10 0 0 0

$ Load Case Data
 1 0 1 0 0 0 0 0 0 0
 2 0 0 0 0 1 0 0 0 0
 3 0 0 0 0 0 1 0 0 0
```

File SAMPLE
Figure VI-2 (continued)

Index

R

S

T

W

Z