ETABSIN™

An Interactive Graphical Input Generator for ETABS®

Version 5.4
Revised July, 1992

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

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABSIN. 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

I.

INTRODUCTION and TERMINOLOGY

The ETABSIN program is an interactive graphical input file generator for the ETABS building design program. ETABSIN allows you to model a building using an intuitive graphical method, while still maintaining the numerical exactness necessary for the dimensional and structural elements.

ETABSIN does no analysis, but prepares and edits the input files used by program ETABS. The files are in standard ASCII text format and may be edited with any text editior, if desired. See the ETABS Users Manual "ETABS Input Data File" for a complete description of the input file format.

There are a number of terms used in this manual and in the ETABSIN program that are not described in the ETABS Users Manual. These terms are described below. When you see the use of "menus", please refer to Chapter IV, which describes the menus and commands in the menus.

Current File

This is the file shown at the top center of the ETABSIN screen enclosed within parentheses. If you do the command **Save** from the **File** menu, the current structure will be saved in this file.

Current Structure

The current structure is all items that have been defined in ETABSIN or read in from a file.These include story and mass definitions, structural element definitions, all frames with their structural element assignments and all building loads. In short, it's everything.

When you do a **Save** or **Save as** command in the **File** menu, the current structure is written into the file. When you do the command **Open** in the **File** menu, any current structure is erased and the contents of the opened file become the new current structure.

Current Frame

The current structure may contain many frames. Each frame contains the frame layout, structural member locations and loading patterns. Only one frame can be seen within ETABSIN at one time, while any other frames are saved in files on the hard disk. You can select the frame that is displayed, and this is the current frame.

Graphical Elements

ETABSIN uses the Microsoft Windows graphical operating environment. There are a numbers of items specific to Windows that are described in the "Windows Users Guide", included with each version of Windows. For convenience, the basic definitions are given below. Please refer to Chapter II for instructions on using the mouse and keyboard.

Windows Version

ETABSIN requires Microsoft Windows 3.0 or later and will NOT run on earlier versions. Also, ETABSIN will NOT run in Windows Real mode as there is not enough memory available. This means that you MUST have an extended memory manager such as HIMEM.SYS in your CONFIG.SYS file so that Windows will run in Standard or 386 Enhanced modes. You can check the mode by clicking on "About Program Manager" under the Help menu in Windows Program Manager. Please refer to the Windows Users Guide.

Starting ETABSIN

There are a number of ways to start ETABSIN, depending on whether Windows is running or not. The suggested methods are listed below:

1. If Windows is not running, change to the Drive and Directory where ETABSIN.EXE is located and start ETABSIN from the DOS prompt by typing:

   WIN ETABSIN

   This requires that Windows be included in the PATH statement in your AUTOEXEC.BAT file.

2. With Windows running, open the Program Manager window, click on File, and then click on Run. Then enter the COMPLETE path followed by ETABSIN. For example:

   E:\ETABS\ETABSIN

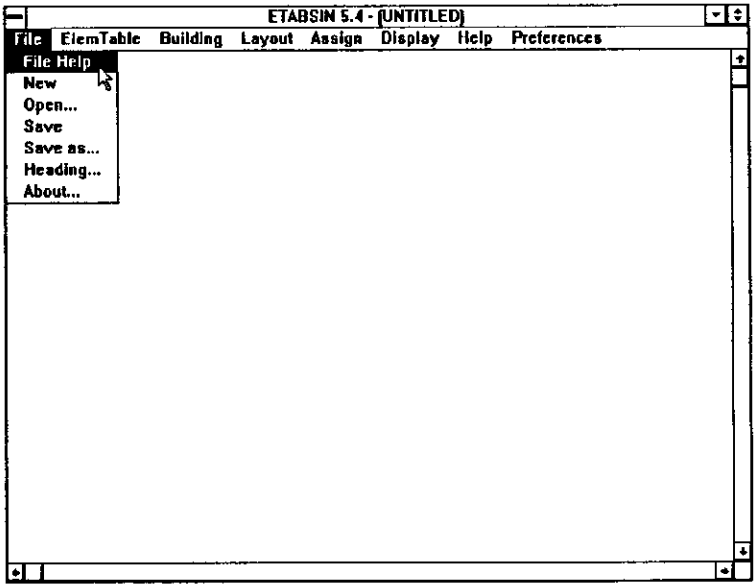
   Then click on OK to start ETABSIN.

3. With Windows running, open the File Manager and select the Drive and Directory containing ETABSIN.EXE. Then double-click on ETABSIN.EXE to start ETABSIN.

4. With Windows running, it is also possible to put ETABSIN.EXE in a Program Manager document icon. Refer to the Windows Users Guide for the procedure.

ETABSIN Screen

When ETABSIN is started, this screen will appear:



This is the basic ETABSIN screen. The current file name is UNTITLED, which means that no file has yet been specified.

Menu Bar

The second line down from the top, containing **File**, **ElemTable**, etc. is the menu bar. Click on one of the words to bring down a menu. The **File** menu is shown.

Menu

The menu is the list of commands that appear when you click in the menu bar. The **File** menu is shown. In the menu, any commands that have ... after them will bring up a dialog box. Commands without ... after them will take immediate action. For example, in the **File** menu, **Save** will immedi-

ately save the current structure in the current file while **Save as** will bring up a dialog box asking for a file name and allow you to cancel the operation.

Close Box

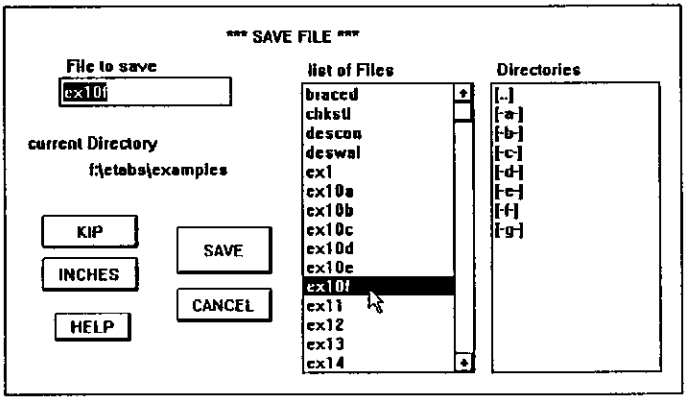
The small box at the upper left corner will close (Stop) ETABSIN if you double-click on it.

Scroll Bars

The bars at the right and bottom of the screen are the scroll bars. They have an arrow at each end. See Chapter II for the use of the scroll bars.

Dialog Box

A dialog box is a box that appears as the result of clicking on a menu command such as **Save as** in the **File** menu. In this case the following dialog box appears:



Dialog boxes usually contain a number of boxes which allow you to enter data, perform actions and see lists of files or other items. These boxes are described below.

List Box

In the dialog box example above, one of the list boxes is labeled List of Files. It contains a list of the files in the Current Directory, but other list boxes may contain directories, lists of beams, or other items. You can scroll through the list using the scroll bar at the right side of the list box. See Chapter II for how to do this.

Edit Box

In the SAVE FILE dialog box example above, the edit box is the rectangular box with the clear background, labeled FILE to SAVE. The purpose of edit boxes is to enter data from the keyboard or to display data from a list box. See Chapter II for a description of entering data. In the example

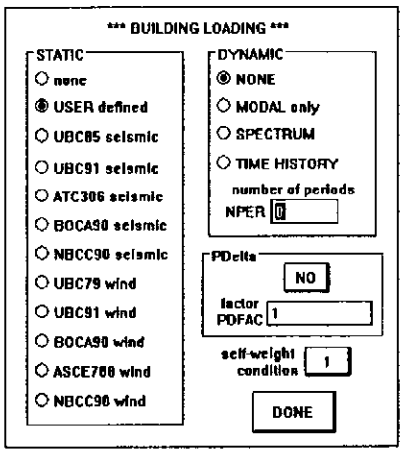
dialog box, clicking on a file in the list box will put its name in the File to Save edit box.

Push Button

The square boxes with a shaded background, labeled SAVE, CANCEL, HELP, KIP and INCHES in the dialog box example above, are all push buttons. Push buttons perform actions when clicked on, but you can NOT enter data in a push button. For example, clicking on the SAVE push button saves the current structure in the File to Save, which is in the Current Directory. Clicking on the KIP push button changes the force units at every click until it returns to KIP.

Radio Button

The following dialog box contains radio buttons:



In the STATIC group, each of the items has a ROUND radio button. Only one of these items can be selected at a time, so that clicking on any radio button in a group turns others in that group off. This is like the station selection buttons in a car radio. Similary in the DYNAMIC group only one

II.

USING the MOUSE and KEYBOARD

ETABSIN follows the conventions given in the Windows Users Guide for the use of mouse and keyboard. This chapter describes the most commonly used conventions. For more details, refer to the Windows Users Guide. Please note that ETABSIN can NOT be run without a mouse, as many of the dialog boxes can not be exited using the keyboard.

Mouse Click

Press and quickly release the LEFT button, while the arrow cursor is pointing to the item you are clicking on.

Mouse Double-click

Click the LEFT button twice rapidly while the arrow cursor is pointing to the item you want to double-click on. If you have a problem with this, it is probably because you are not clicking fast enough. It does take some practice. If you have a continual problem, the double-click speed can be set slower. The procedure is described in the Windows User's Guide.

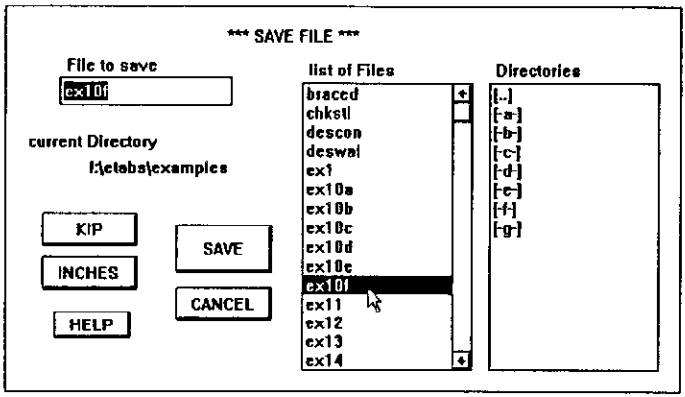
Mouse Assignment

The mouse is used to assign various structural elements to the frame layout. Items that have length or area such as Beams or Panels require the following procedure:

POINT to the starting point with the arrow cursor, PRESS the LEFT button and HOLD it down while moving the cursor to the end point. Then RELEASE the button. See the Assign menu in Chapter IV.

Scroll Bars

The following dialog box contains a scroll bar in the list box labeled List of Files:



The small empty box in the scroll bar shows the present position in the list box. Point to the box, press and hold the left button, move the box as desired and then release the button. If the list of items is short, so that they all show, then no scroll bar will be showing. You can also scroll the contents of the list more slowly by pointing to one of the arrows at the end of the scroll bar and either clicking or pressing and holding the left button. All scroll bars work in the same way.

Keyboard Data Entry

In the dialog box shown above, you can enter data in the box labeled File to Save. This can be done entirely from the keyboard as follows:

Press the Tab key to highlight the edit box, then type your data on the keyboard. When you type the first character, the edit box will be erased and the new data will take its place.

If there is more than one edit box, then the Tab key will cycle through all the boxes in one direction, highlighting each box in turn. You can use Shift-Tab to go the other direction.

Another way to enter data uses the mouse and keyboard together. Place the arrow cursor in the edit box and double click. This highlights the edit box just like the Tab key for the entry of new data. To change existing data in the edit box place the arrow cursor in the edit box and click once. The cursor will change to a thin vertical line, but the present edit box contents will remain.You can move the line cursor left and right with the arrow keys. If you type characters on the keyboard, they will be INSERTED in the edit box. The Backspace key deletes the character to the left of the cursor and the Del key deletes the character to the right of the cursor.

Which method to use? It depends. Generally, using the Tab key is faster as it erases the present contents of the edit box. However, if you have a long entry in the edit box and only want to change one or two characters, then using the mouse is best. Using the mouse double click feature can also be helpful if the number of edit boxes is large but only one needs to be changed.

III.

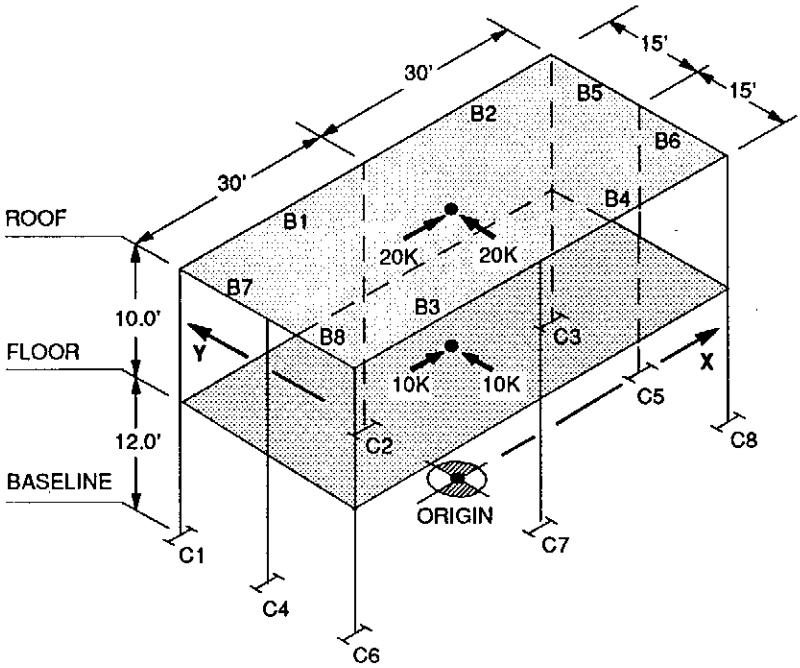
TUTORIAL

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. For a detailed description of all the options of ETABSIN you should refer to Chapter IV of this manual.

The tutorial is composed of two exercises. The first exercise guides you through the initial generation of the structural model and the lateral loads for the example structure shown in Figure III-1. The second exercise shows how to modify the initial model produced in Exercise 1 to add vertical loads to the model.

If you are unfamiliar with the Windows terminology, starting Windows applications or the use of the mouse and the keyboard in association with Windows you should read Chapters I and II of this manual first.

The exercises assume that Microsoft Windows has been installed according to the Windows setup instructions and a path has been made to the directory where the Windows files reside (using the DOS **path** command.) It is also assumed that the ETABSIN disk has been copied onto the harddisk in directory ETABSIN.



- ALL COLUMNS ARE W14 x 120
- ALL BEAMS ARE W33 x 118
- A36 STEEL
- FIXED BASE

EXAMPLE STRUCTURE
Geometry and Lateral Loading
Figure III-1

Exercise One

In order to initially generate an ETABS model of the simple structure that we have chosen you will need to execute the following steps:

a. Start Windows and ETABSIN

b. Interactively generate the structural model and loading

c. Save the ETABS model

d. Quit ETABSIN

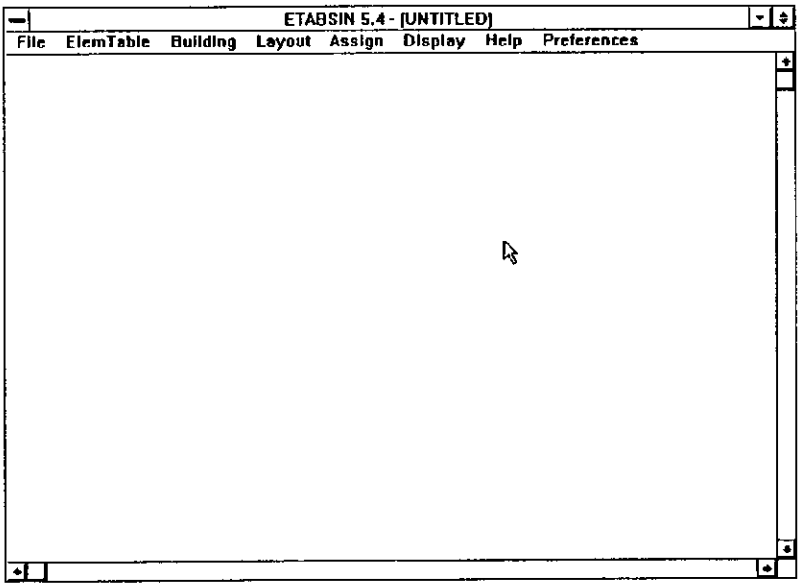
e. Quit Windows

The completion of the steps noted above will produce a text file that contains the input data required to execute ETABS for the analysis of the model generated in ETABSIN.

The following five sections (a through e) of this exercise correspond to the steps specified above. Each section defines in detail the procedures required to implement the associated step.

a. STARTING WINDOWS and ETABSIN

Windows and ETABSIN can be started in several different ways. A few of the methods are discussed earlier in Chapter I of this manual. Using any of these methods will activate the ETABSIN environment. The screen will initially display the ETABSIN window with an hourglass shape as the program initializes the arrays needed for modeling. 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 (pointing to the item using the mouse and pressing the mouse left button.) The pull down menu has a list of commands. A particular command from the pull down menu can be selected by clicking on it.

b. INTERACTIVELY GENERATING THE MODEL

The following are the steps for interactively generating the model:

i. Define story heights and labels

ii. Define architectural grid

iii. Define column line coordinates

iv. Define bay connectivity

v. Define material properties

vi. Define column section properties

vii. Define beam section properties

viii. Assign column properties

ix. Assign beam properties

x. Define lateral loading

xi. Define load cases (combinations)

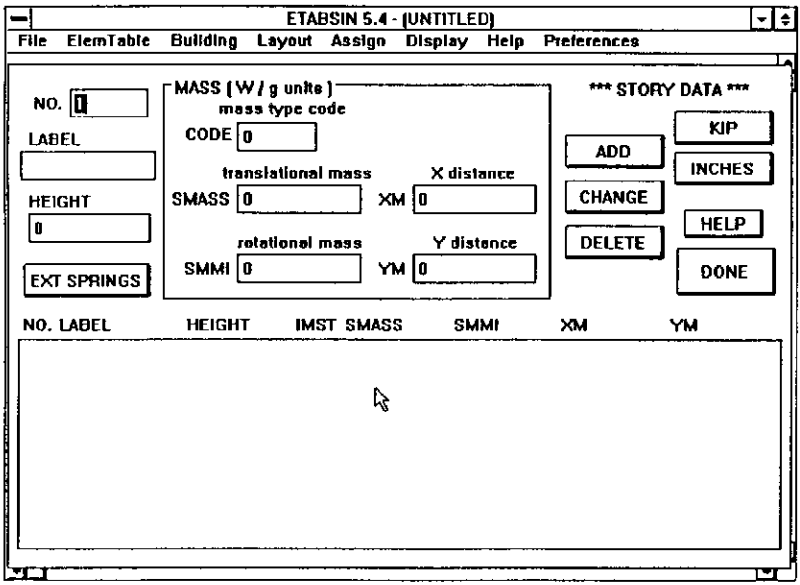
The following subsections (i through xi) of this section correspond to the eleven steps specified above. Each subsection defines in detail the procedures required to implement the associated step.

i. Defining Story Heights and Labels

To define story heights and labels do the following:

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

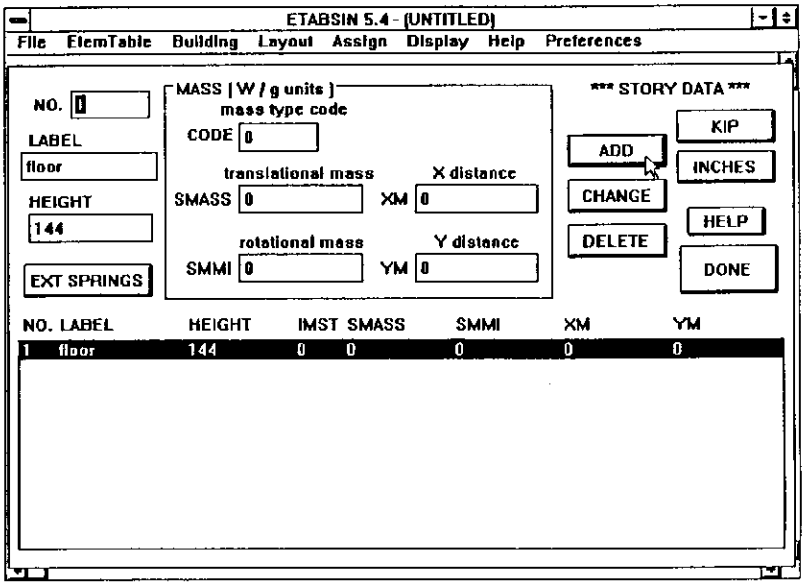
2. Click on **Story** in the **Building** pull down menu. The following STORY DATA dialog box will appear:



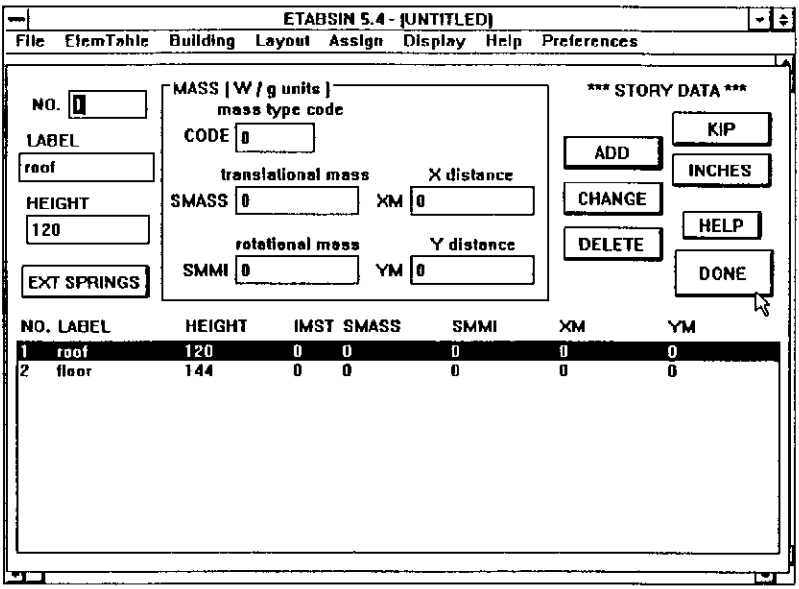
3. Type **floor** in the edit box labeled LABEL as the label associated with the bottommost story level. You can use the TAB key on the keyboard to move to the different edit boxes in the dialog box. When this method is used the highlighted box is where the data will be entered.

4. Type **144** in the edit box labeled HEIGHT as the height associated with the bottommost story.

5. Click on ADD. This will add this story information to the story table and the screen will appear as follows:



6. Repeat steps 3 to 5 above for the roof story level, entering **roof** and **120** for the story label and height, respectively. The screen will appear as follows:



7. Click on DONE. The program will close the STORY DATA dialog box and return with the blank ETABSIN window.

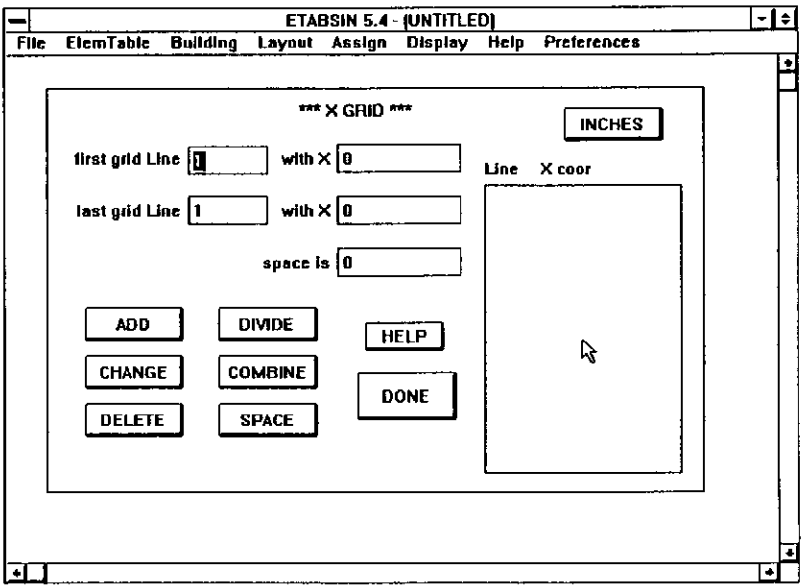
Note that we have entered data from bottom story to top story. This is the most convenient. However, story data can be entered in any order. To insert the data at a particular location in the list, enter the number of the location in the NO. edit box and click on ADD. When this data is completed, the stories listed in the list MUST be from top to bottom.

ii. Defining Architectural Grid

To define the architectural grid do the following:

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

2. Click on **X Grid** in the **Layout** pull down menu. The following X GRID dialog box will appear:



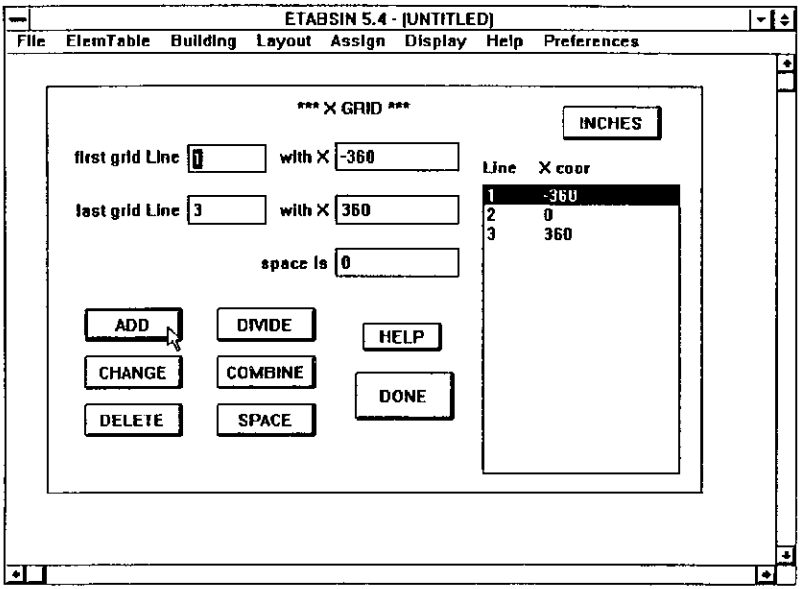
3. Type **1** in the edit box labeled FIRST GRID LINE.

4. Type **3** in the edit box labeled LAST GRID LINE.

5. Type **-360** in the edit box labeled WITH X for the FIRST GRID LINE.

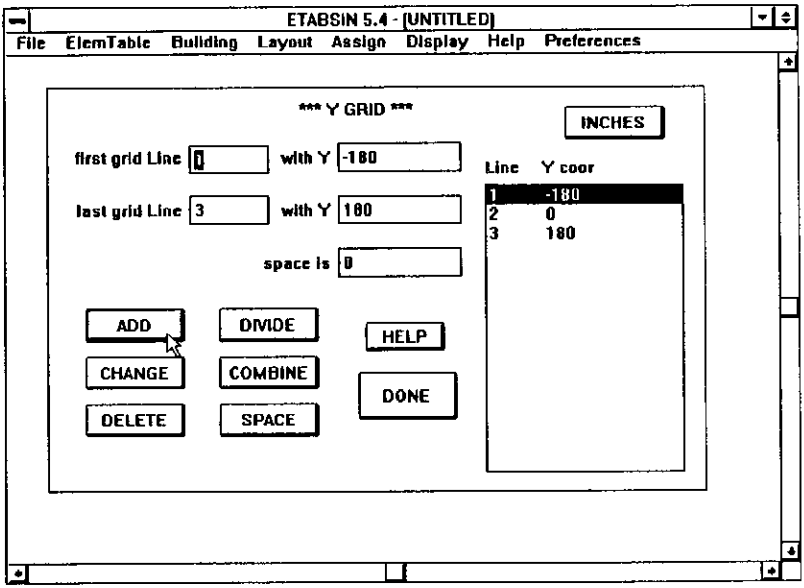
6. Type **360** in the edit box labeled WITH X for the LAST GRID LINE.

7. Click on ADD. This will add grid lines 1,2 and 3 to the table of X grid lines and the screen will appear as follows:

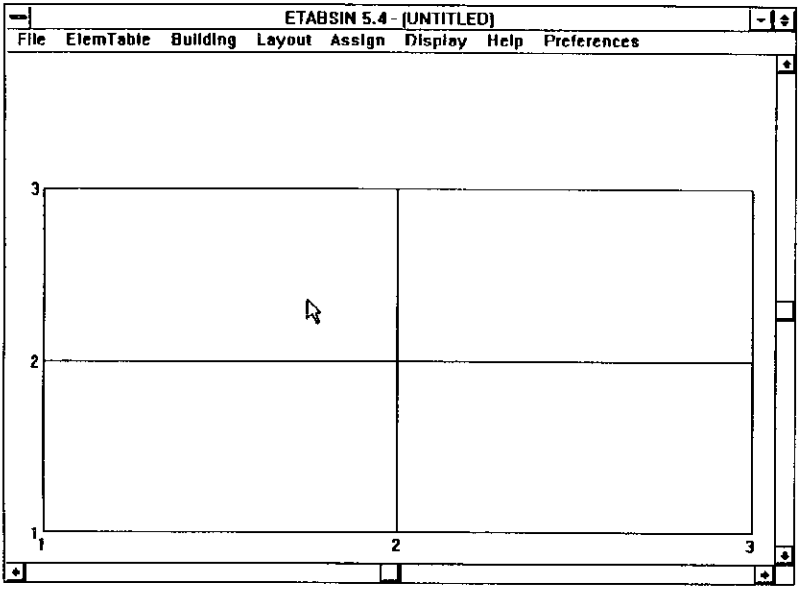


8. Click on DONE. The program will close the X GRID dialog box and return with the blank ETABSIN window.

9. Repeat steps 1 to 7 above, except doing it for the Y grid instead of the X grid and typing **-180** and **180** instead of **-360** and **360**, respectively. The screen will appear as follows:



10. Click on DONE. The program will draw the X and Y grids and the screen will appear as follows:

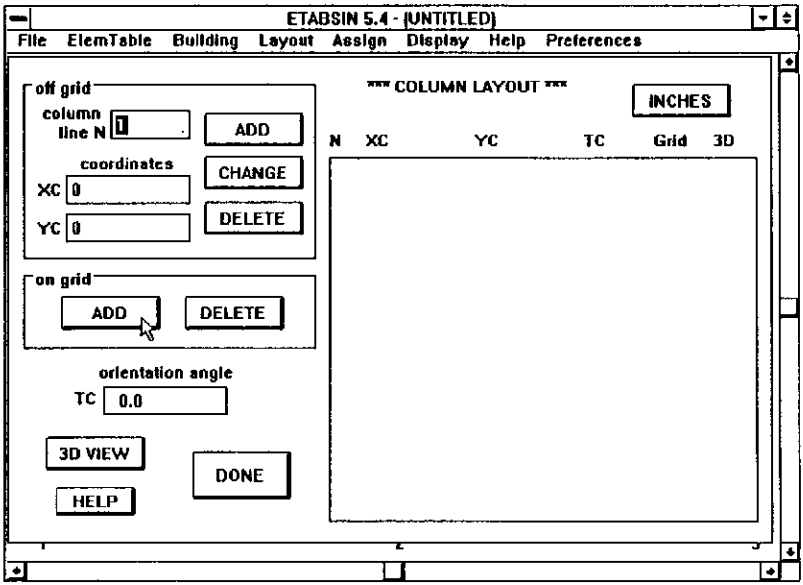


iii. Defining Column Line Coordinates

To define column line coordinates, do the following:

1. Click on **Layout** on the menu bar.

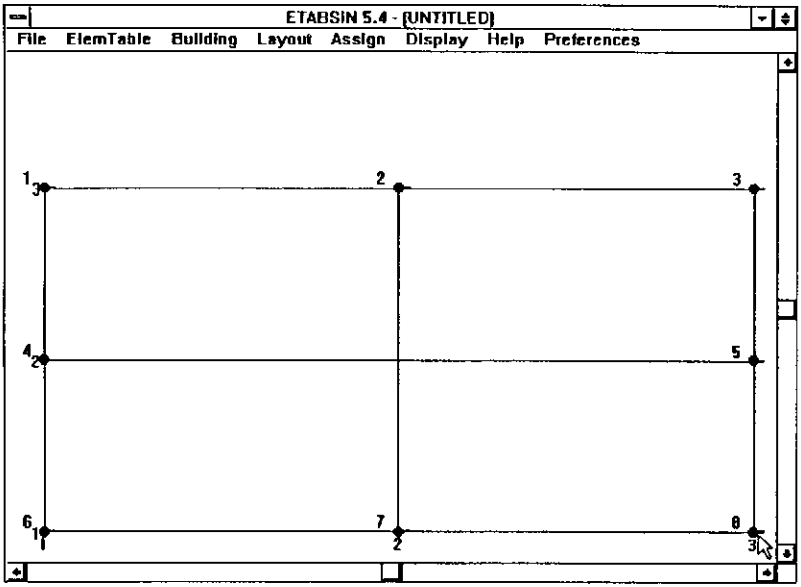
2. Click on **Columns** in the **Layout** pull down menu. The following COLUMN LAYOUT dialog box will appear:



3. Click on ADD in the ON GRID box. The screen will show the X and Y grid lines previously defined.

4. Click on the intersections of X and Y grid lines where the column lines are to be located, one column line at a time. The program will locate a small red dot at the intersections clicked and number these dots consecutively. The red dots show the locations of the column lines and the numbers indicate the order in which they will be referred to in the ETABS input data file to be created. There is also a small red line drawn. It indicates the major axis direction of the

column (for example it is the direction of the web for an I-section.) The screen will appear as follows after all of the eight column lines for this structure are located:

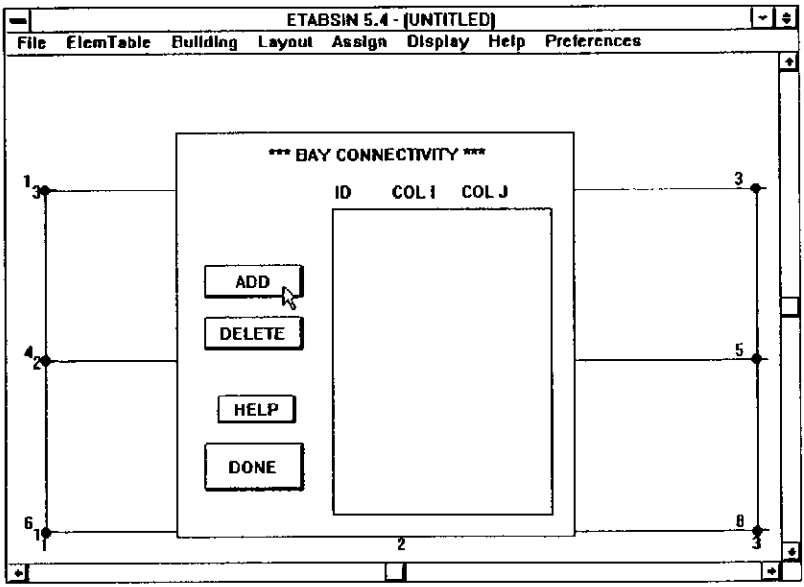


iv. Defining Bay Connectivity

To define bay connectivities do the following:

1. Click on **Layout** on the menu bar.

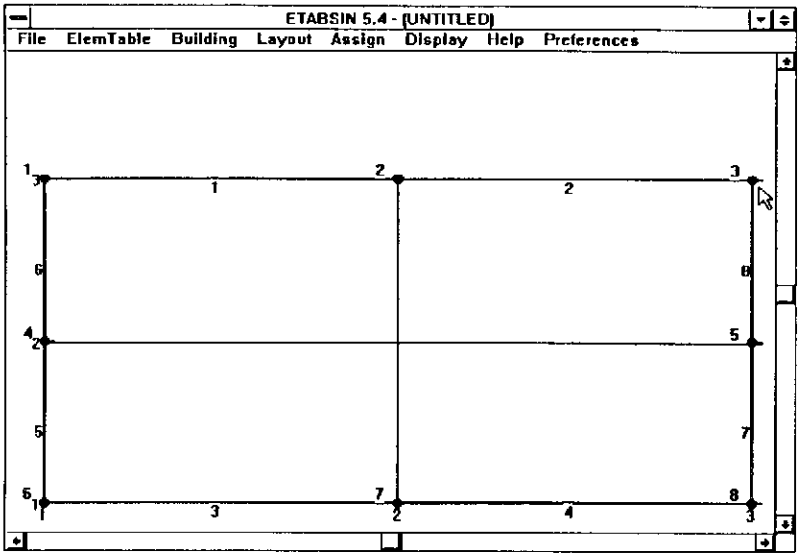
2. Click on **Bays** in the **Layout** pull down menu. The following BAY CONNECTIVITY dialog box will appear:



3. Click on ADD. The screen will show the X and Y grid lines and the column lines previously defined.

4. Add Bay 1 by pointing at column line 1, pressing the mouse left button, moving the pointer to column line 2 while keeping the mouse button pressed and then releasing the mouse button. The program will draw a yellow line to indicate the location of the bay and number it with the bay number. Column line 1 now corresponds to I-end of the bay and column line 2 corresponds to J-end of the bay.

5. Repeat step 4 above for bays 2 to 8. When all bay connectivities have been defined the screen will appear as follows:



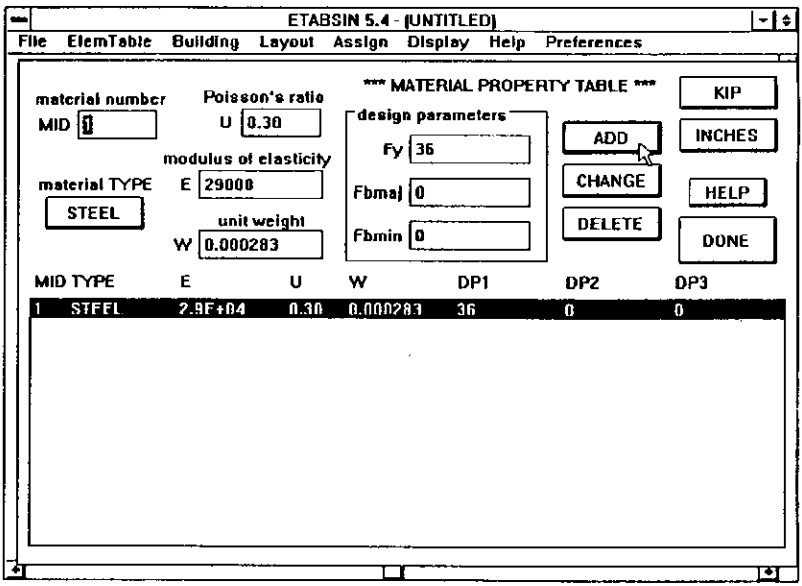
v. Defining Material Properties

To define material properties do the following:

1. Click on **ElemTable** on the menu bar.

2. Click on **Material** in the **ElemTable** pull down menu. The MATERIAL PROPERTY TABLE dialog box will appear. The material type and the material properties we need are already defaulted. If different values had to be used, we would enter them at this time.

3. Click on ADD. This will add the material properties showing in the edit boxes to the material property table. The screen will now appear as follows:



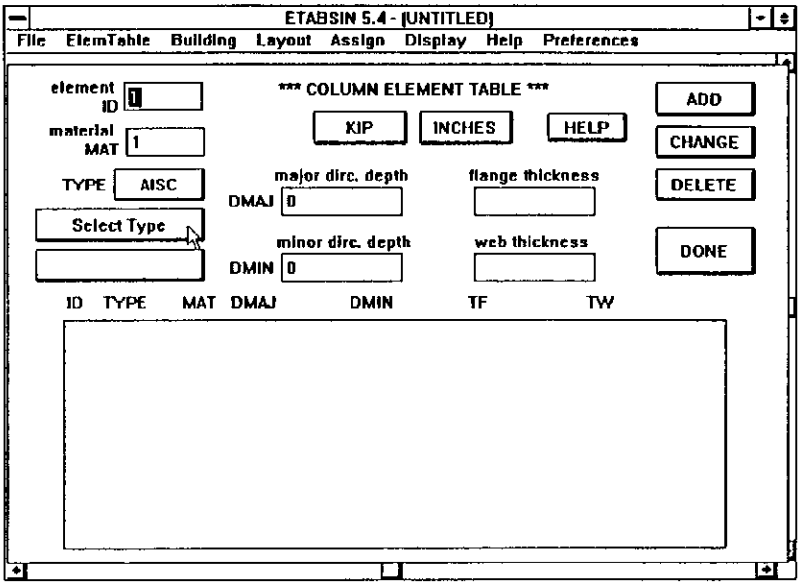
4. Click on DONE. The program will close the MATERIAL PROPERTY TABLE dialog box and return the blank ETABSIN window.

vi. Defining Column Section Properties

To define column section properties do the following:

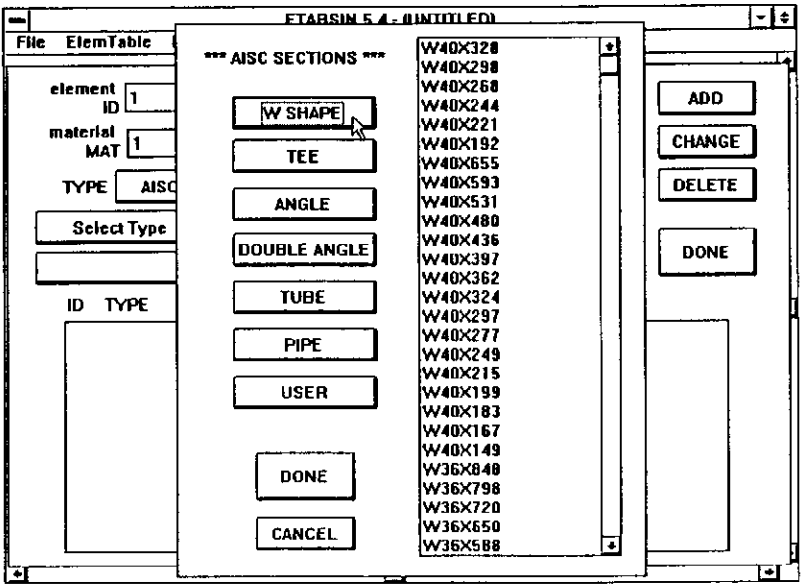
1. Click on **ElemTable** on the menu bar.

2. Click on **Column** in the **ElemTable** pull down menu. The following COLUMN ELEMENT TABLE dialog box will appear:



3. As shown above the TYPE is already defaulted to AISC. Click on SELECT TYPE to select the required AISC shape. The AISC SECTIONS dialog box will appear.

4. Click on W SHAPE to show the available wide flange section labels. The screen will appear as follows:

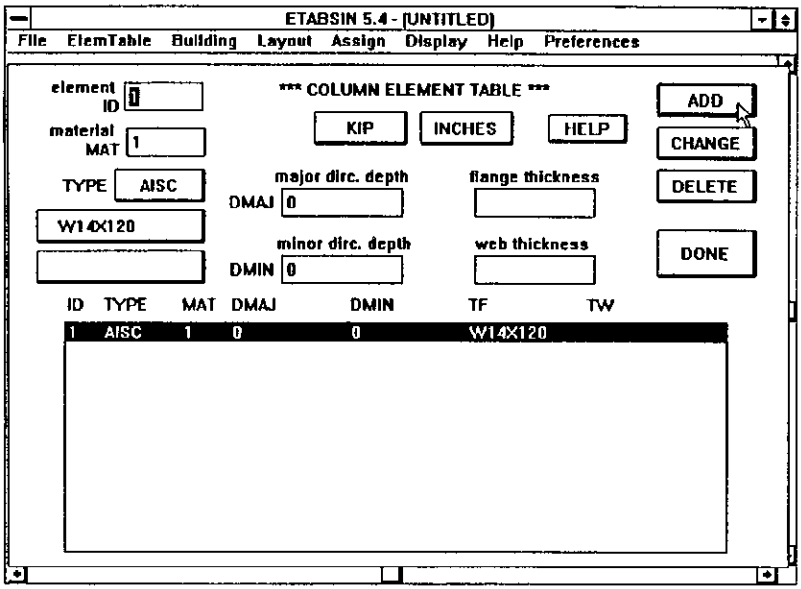


5. Use the scroll bar showing on the right of the list of labels to move up or down the list until W14X120 is in view. The scroll bar can be used by either clicking on the arrows at the end to move the list a little at a time, or by pointing to the white box and pressing the mouse left button and moving the pointer with the mouse button down (the white box will move with it) and releasing the mouse button where we want to locate the white box. The location of the white box along the vertical scroll bar shows the location of the list in view compared to the full list.

6. Click on W14X120 to select it.

7. Click on DONE. The program will close the AISC SECTION dialog box and will return to the COLUMN ELEMENT TABLE dialog box with W14X120 showing in the box which originally said SELECT TYPE.

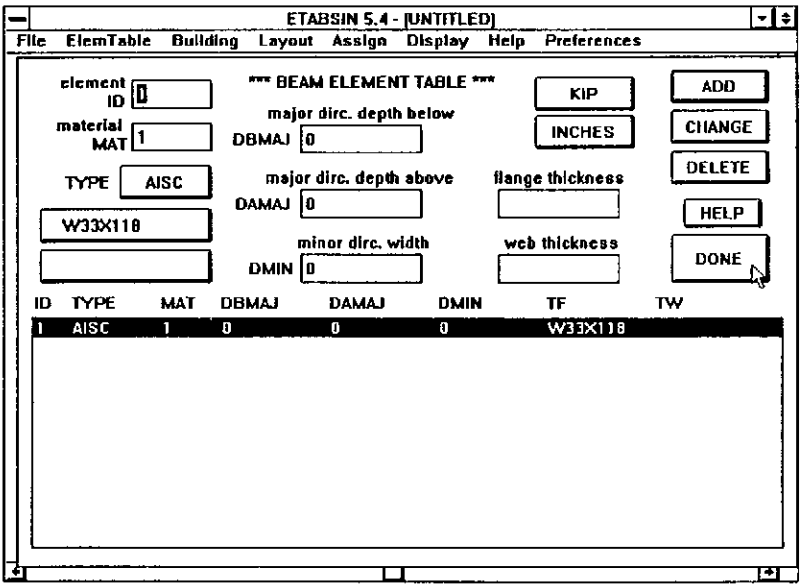
8. Click on ADD. The program will enter the selected section in the column element table and the screen will appear as follows:



9. Click on DONE. The program will close the COLUMN ELEMENT TABLE dialog box and return the blank ETABSIN window.

vii. Defining Beam Section Properties

To define the beam section properties follow the same steps as were made to define the column section properties in Section vi above, except the **Beam** command should be selected from the **ElemTable** pull down menu and the W33X118 section should be selected in the AISC SECTIONS dialog box. Before the DONE button is clicked the screen should appear as follows:

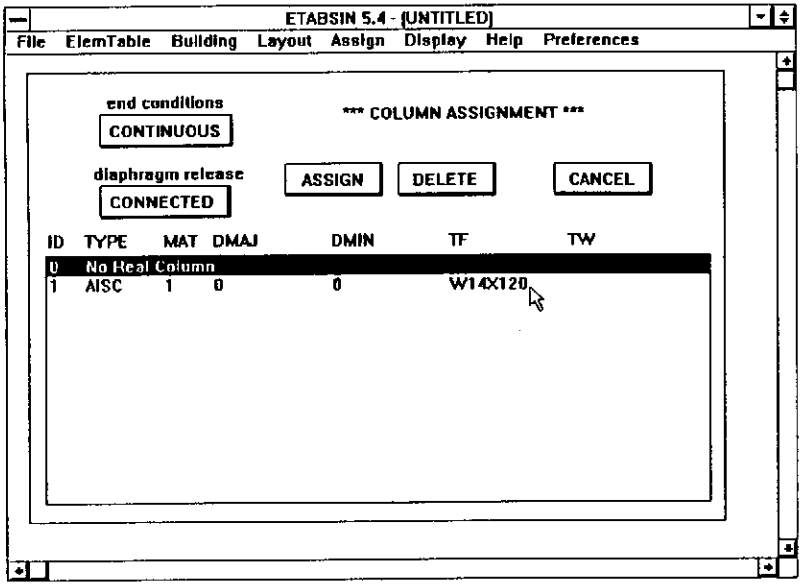


viii. Assigning Column Properties

To assign column properties to the column lines where columns exist do the following:

1. Click on **Assign** on the menu bar.

2. Click on **Columns** in the **Assign** pull down menu. The following COLUMN ASSIGNMENT dialog box will appear:



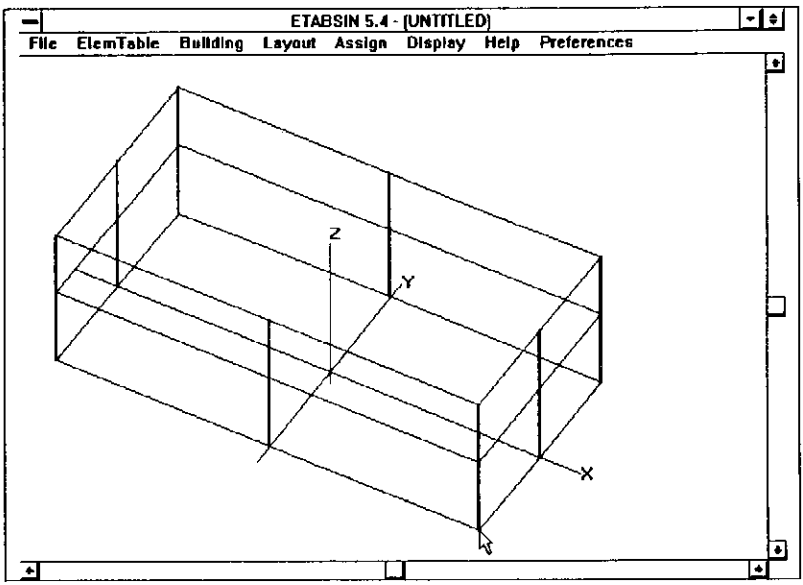
3. Click on the line in the list of available column section properties that represents the W14X120 shape that we want to assign. This line will become highlighted.

4. The default values on the END CONDITIONS and DIAPHRAGM RELEASE is what we want, so click on ASSIGN. The program will close the COLUMN ASSIGNMENT dialog box and will show a three dimensional view of the structure with the column lines and bays showing.

This is the default view. The program is now ready for assignment of column properties.

5. Point to the top of column line 1 and press the mouse left button. Move the pointer while still pressing the mouse button to the bottom of column line 1 and release the mouse button. The program will color column line 1 magenta. This indicates that this column line has now been assigned the section property that was selected from the COLUMN ASSIGNMENT dialog box. It should also be noted that we have assigned this section property to column line 1 for both stories. This could have been done one story at a time also by only moving the pointer one story at a time before releasing it.

6. Repeat step 5 for column lines 2 to 8. The order in which this is done is unimportant. Also the pointer may be moved from bottom to top of a column line. After all column properties are assigned the screen will appear as follows:

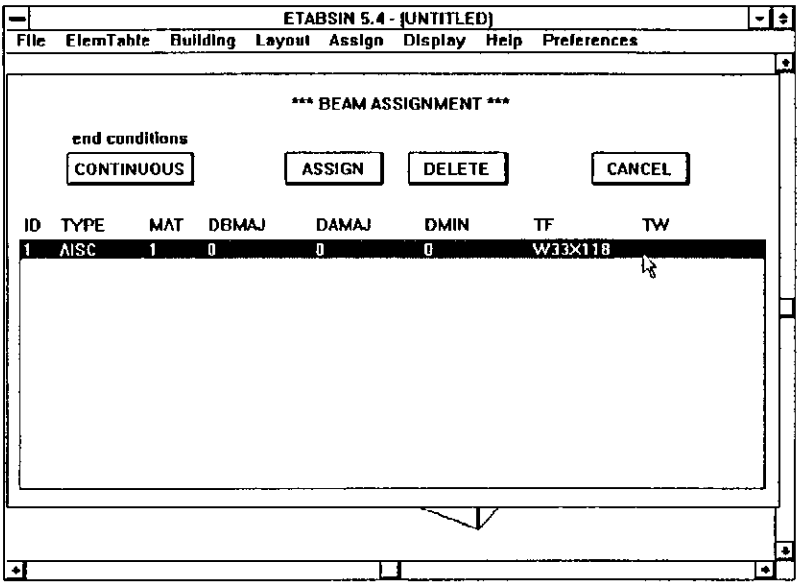


ix. Assigning Beam Properties

To assign beam properties to bays where beams exist do the following:

1. Click on **Assign** on the menu bar.

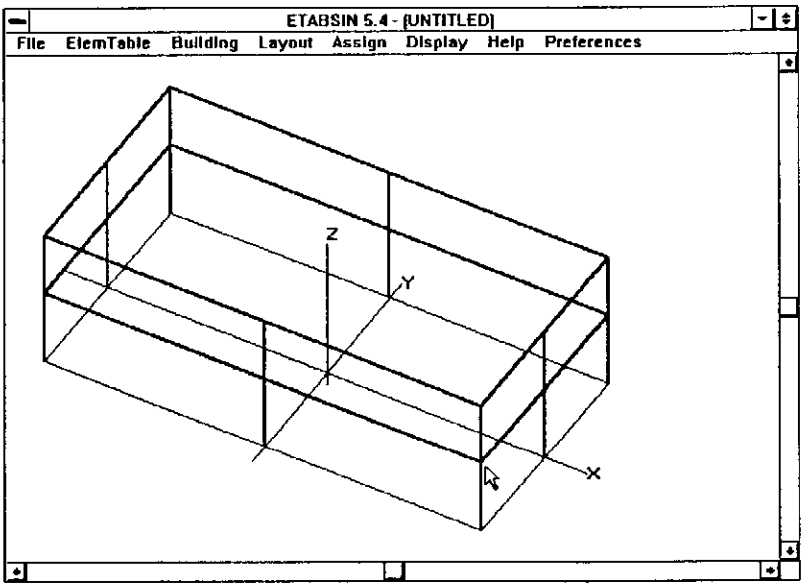
2. Click on **Beams** in the **Assign** pull down menu. The following BEAM ASSIGNMENT dialog box will appear:



3. Click on the W33X118 property line in the list of properties.

4. The default values on the END CONDITIONS is what we want, so click on ASSIGN. The program will close the BEAM ASSIGNMENT dialog box and will show a three dimensional view of the structure with the columns already assigned showing in red. The bays are showing as thin white lines. The program is now ready for assignment of beam properties.

5. The beams are assigned similarly to the columns by pointing to one end of the bay, pressing the mouse left button, moving the pointer to the other end of the bay while keeping the mouse button pressed and then releasing the mouse button. Only one bay can be assigned at a time, although they can be generated vertically. If you move horizontally along the bay, only one bay is assigned the beam properties. If you move diagonally along the bay while going up or down vertically over several stories, the bay will be assigned the beam properties over all stories crossed. As properties are assigned to each bay the program colors it magenta. So assign the beam properties to all the bays. After the assignments are made the screen will appear as follows:

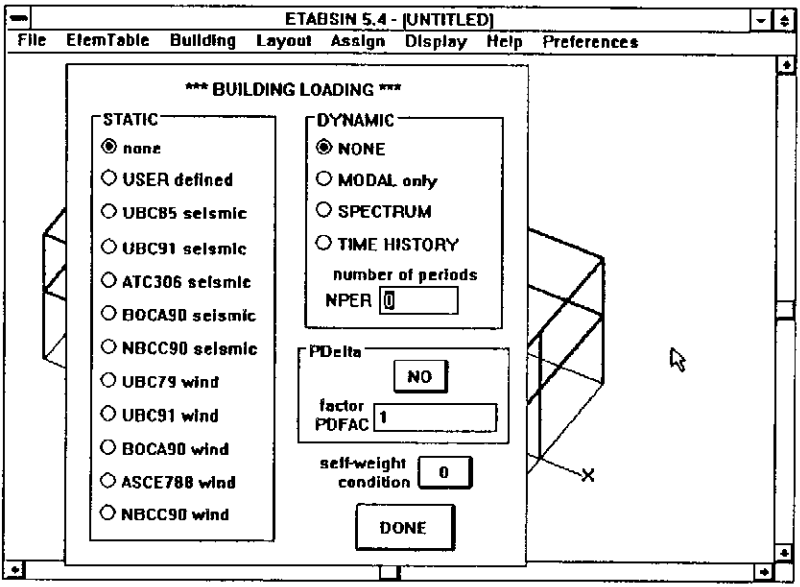


x. Defining Lateral Loading

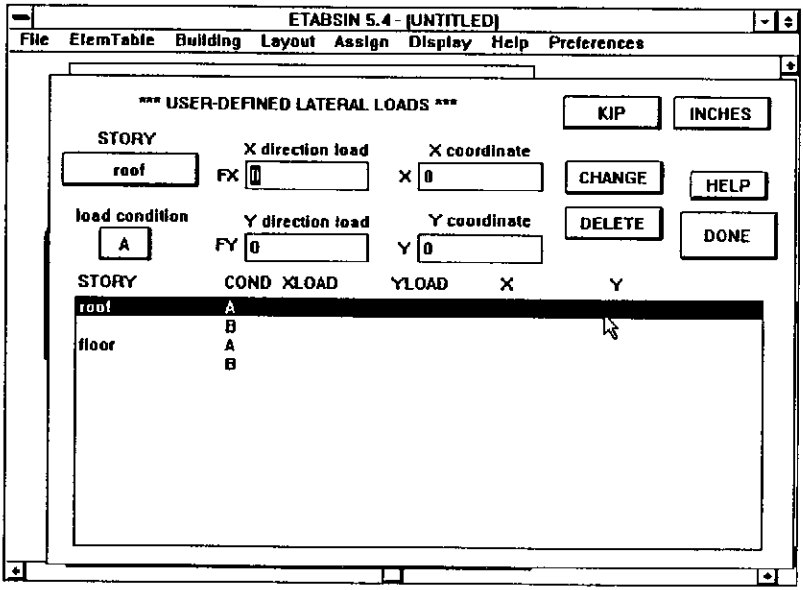
To define the lateral loading on the structure do the following:

1. Click on **Building** on the menu bar.

2. Click on **Loading** in the **Building** pull down menu. The following BUILDING LOADING dialog box will appear:



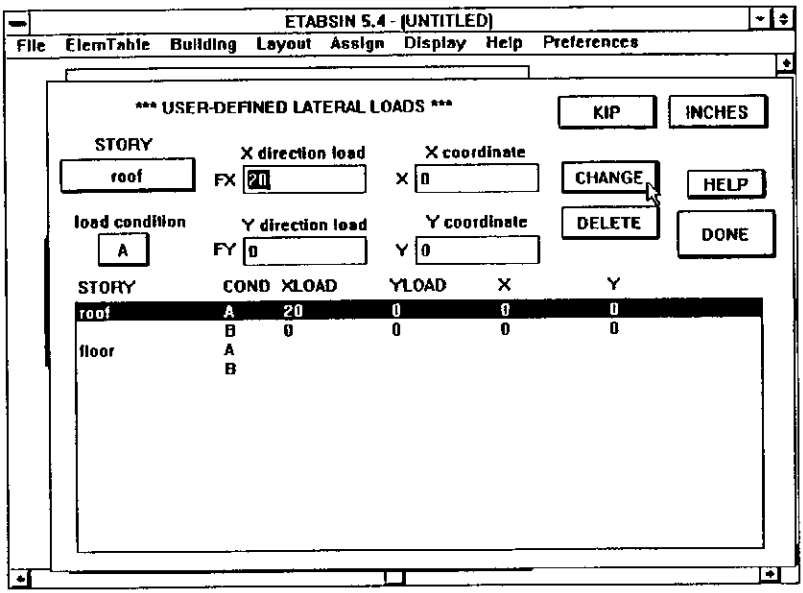
3. Click on USER DEFINED. The following USER-DEFINED LATERAL LOADS dialog box will appear:



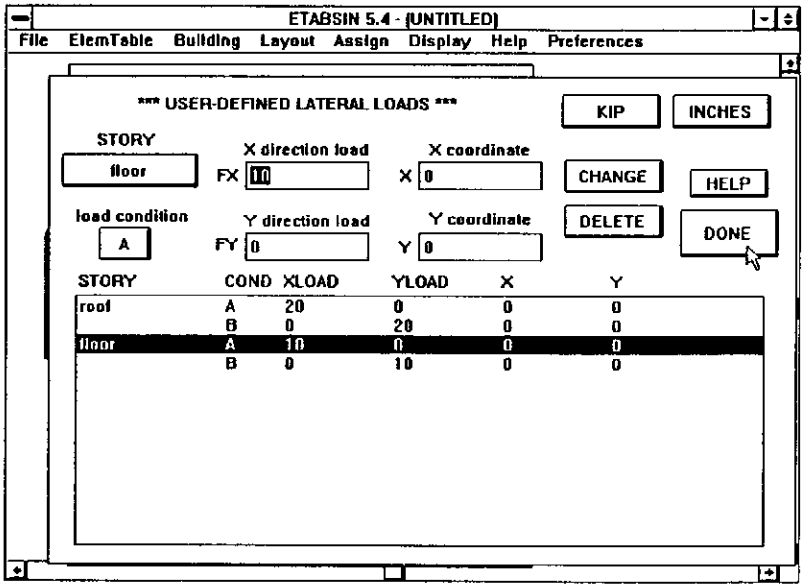
4. As can be seen the lateral loads for all stories for both lateral load conditions A and B are already initialized to zeros (blank values.) Click on the line for STORY roof and COND A. This will be highlighted and its values brought to the edit boxes for modifications.

5. Type **20** in the X LOAD edit box.

6. Click on CHANGE. The program will modify the values in the list of lateral loads to the ones shown in the edit box and the screen will appear as follows:



7. Repeat steps 4 to 6 above for the load condition B for the roof and for load conditions A and B for the floor; except in step 5 type **20** in the Y LOAD edit box for roof load condition B, type **10** in the X LOAD edit box for floor load condition A and type **10** in the Y LOAD edit box for floor load condition B. After these steps are repeated the screen will appear as follows:



8. Click on DONE. The program will close the USER-DEFINED LATERAL LOADS dialog box and return to the BUILDING LOADING dialog box.

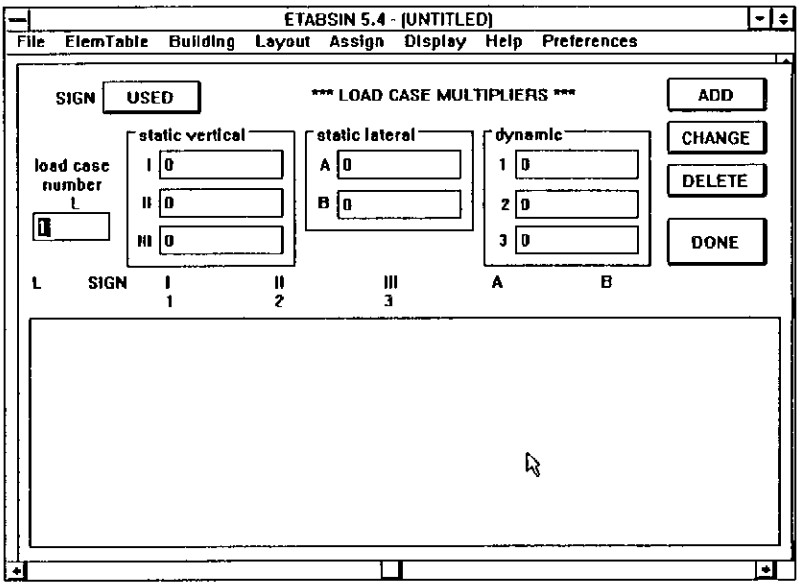
9. Click on DONE. The program will close the BUILDING LOADING dialog box and return the blank ETABSIN window.

xi. Defining Load Cases

We have provided loads to the program for load conditions A and B only. We will request data for these two load conditions separately. To define load cases (combinations) to the program do the following:

1. Click on **Building** on the menu bar.

2. Click on **Load Case** in the **Building** pull down menu. The following LOAD CASE MULTIPLIERS dialog box will appear:



3. Type **1** in the STATIC LATERAL A edit box.

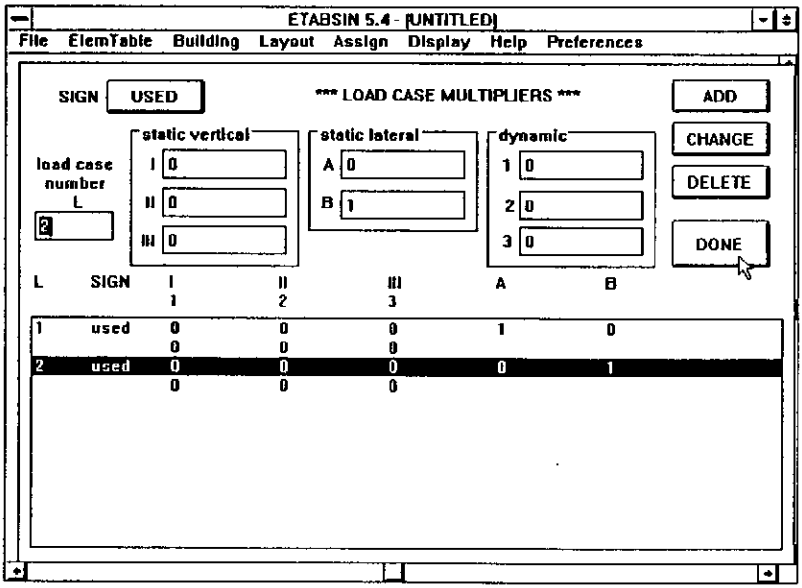
4. Click on ADD. This will add the first loading combination to the list.

5. Type **2** in the Load case number L edit box.

6. Type **0** in the STATIC LATERAL A edit box.

7. Type **1** in the STATIC LATERAL B edit box.

8. Click on ADD. This will add the second load case to the list and the screen will appear as follows:



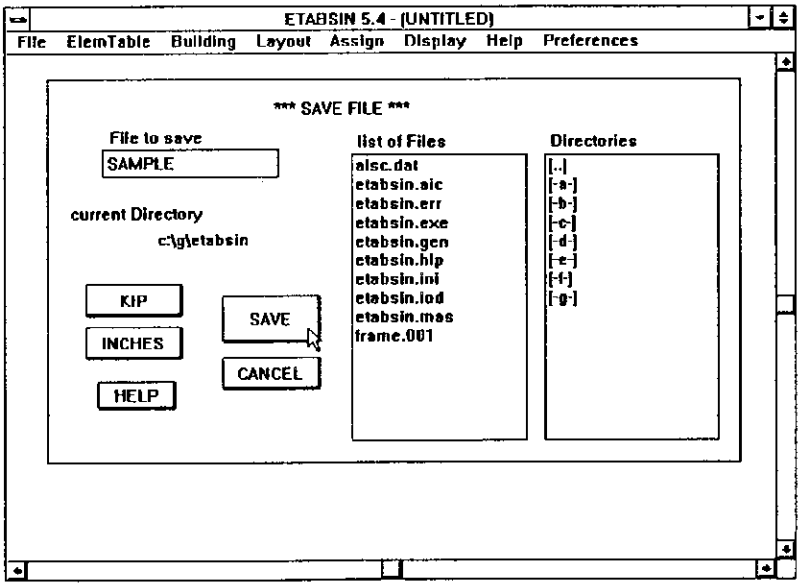
c. SAVING THE MODEL

The modeling of the structure and the lateral load specification is now complete. We can now save the model. It should be pointed out here that intermediate models can also be saved and later brought back into ETABSIN for editing and completion. For any significant size problem one should save quite often while working on a model. To save the model do the following:

1. Click on **File** on the menu bar.

2. Click on **Save as** in the **File** pull down menu. The SAVE FILE dialog box will appear.

3. Type **SAMPLE** in the FILE to SAVE edit box. The screen will appear as follows:



4. Noting that the units are set to KIP and INCHES, click on SAVE. The program will now save the model and the loads in an ASCII file called SAMPLE. The file is in a

format directly readable by ETABS for analysis and by ETABSIN for further editing. The program will then close the SAVE FILE dialog box and return the blank ETABSIN window.

d. QUITTING ETABSIN

To quit ETABSIN double click on the small box on the left upper corner (called the Control-menu box) of the ETABSIN window. Refer to the Windows Users Guide for other methods of quitting applications. The program will close itself and return you to the Desktop, Program Manager or File Manager based on how the program was started.

e. QUITTING WINDOWS

To quit Windows you may first need to close all active application windows. Double click on the small box on the left upper corner (Control-menu box) of these applications to close them. Once the applications are closed, double clicking on the Control-menu box of the Program Manager quits Windows. If the Program Manager is iconized, clicking on its icon will open its Control-menu. In this case click on Close in the Control-menu to quit Windows. Windows will confirm that you want to quit it. After you confirm by clicking on OK, the Windows program will return you to DOS. Refer to the Windows Users Guide for other methods of quitting Windows.

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

```
$ This is file SAMPLE written by ETABSIN on Fri May 29 16:57:56 1992
$ Units are KIP and INCHES


$ Control data
 2 1 1 0 2 0 1 1 1 0 0 1 0 0 0 4 0 0 0 5.40 0 0
$ Miscellaneous parameters
 386.4 0.0001 0 1
$ Mass data
$ Story data
 roof     120 0 0 0 0 0 0 0 0
 floor    144 0 0 0 0 0 0 0 0
$ Frame member Material property data
   1 S 29000 0.000283 0.3 36 0 0
$ Column section property data
   1   1 W14X120          0 0 0 0
$ Beam section property data
   1   1 W33X118          0 0 0 0 0
$$$$$$$$$$ Frame Control Data

 1 2 8 8 0 0 0 0 0 3 3
$ X Grid line data
-360 0 360
$ Y Grid line data
-180 0 180
$ Column line coordinates and orientation
   1   -360.000    180.000       0.000   1   3   3
   2      0.000    180.000       0.000   2   3   3
   3    360.000    180.000       0.000   3   3   3
   4   -360.000      0.000       0.000   1   2   3
   5    360.000      0.000       0.000   3   2   3
   6   -360.000   -180.000       0.000   1   1   3
   7      0.000   -180.000       0.000   2   1   3
   8    360.000   -180.000       0.000   3   1   3
$ Bay connectivity
   1   1   2
   2   2   3
   3   6   7
   4   7   8
   5   6   4
   6   4   1
   7   8   5
   8   5   3
$ Column Location Data
   1 0 roof       1   1   0   0
   2 0 roof       1   1   0   0
   3 0 roof       1   1   0   0
   4 0 roof       1   1   0   0
   5 0 roof       1   1   0   0
   6 0 roof       1   1   0   0
   7 0 roof       1   1   0   0
   8 0 roof       1   1   0   0

$ Beam Location Data
   1 0 roof       1   1   0
   2 0 roof       1   1   0
   3 0 roof       1   1   0
   4 0 roof       1   1   0
   5 0 roof       1   1   0
   6 0 roof       1   1   0
   7 0 roof       1   1   0
   8 0 roof       1   1   0

$$$$$$$$$$ Frame Location data
   1   0       0.000       0.000     0.0
$$$$$$$$$$ Building Loading
$ User Defined Static Lateral Loads
 20 0 0 0 0 20 0 0
 10 0 0 0 0 10 0 0
$$$$$$$$$$ Building Load Cases
   1 0     0.00     0.00     0.00     1.00     0.00     0.00     0.00     0.00
   2 0     0.00     0.00     0.00     0.00     1.00     0.00     0.00     0.00
```

File SAMPLE
Figure III-2

Exercise Two

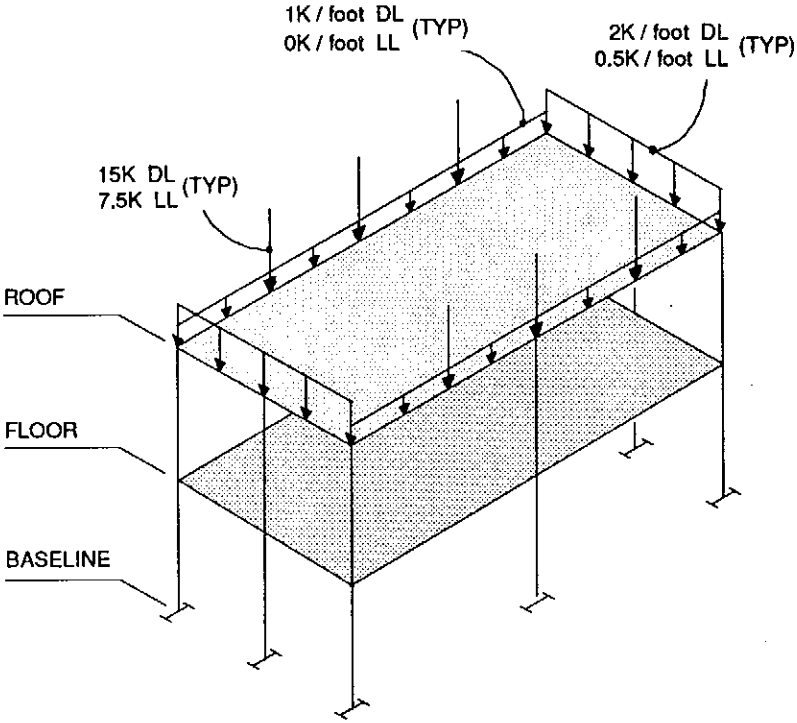
In this exercise we will add gravity loads to the model generated and saved in Exercise One. Heading information will also be added to the model. The gravity loads to be modeled on the structure are shown in Figure III-3. In order to modify the model generated earlier you will need to execute the following steps.

a. Open existing file

b. Modify the model

c. Save the revised model

The following three sections (a through c) of this exercise correspond to the steps specified above. Each section defines in detail the procedures required to implement the associated step.



• FLOOR LOADS are the same as the ROOF

EXAMPLE STRUCTURE
Gravity Loading
Figure III-3

a. OPENING AN EXISTING FILE

To open an existing file in ETABSIN do the following:

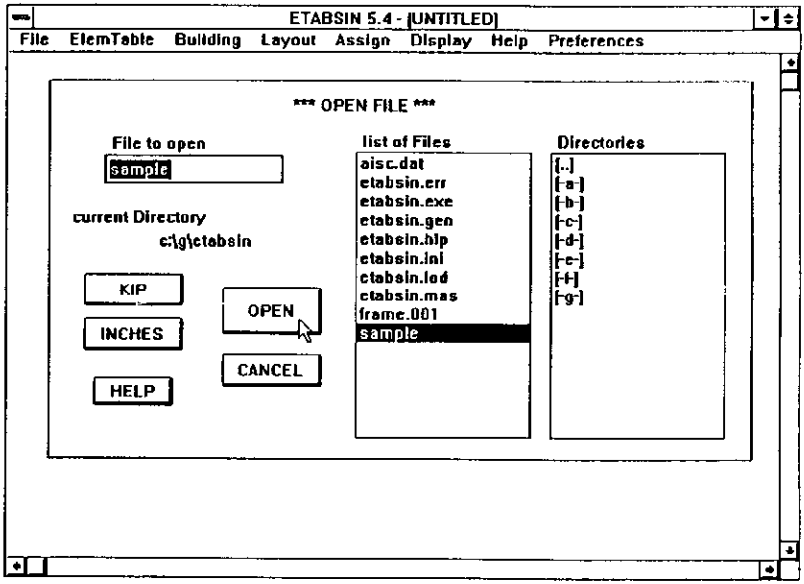
1. Start Windows and ETABSIN the same as Exercise One.

2. Click on **File** on the menu bar.

3. Click on **Open**. The OPEN FILE dialog box will appear.

4. Use the scroll bar to get file SAMPLE in the LIST of FILES list box in view. (If the file SAMPLE is not in the directory shown in the CURRENT DIRECTORY, then first select the correct directory in the DIRECTORIES list box.)

5. Click on SAMPLE. This filename will be highlighted and it will also appear in the FILE to OPEN edit box. The screen will appear as follows:



6. Noting that the file being opened was saved in the same units as currently showing on the screen (KIP and INCHES), click on OPEN. A caution message, warning that Open File will overwrite any current structure, will appear. Click on the OK button. An hourglass shape will appear indicating that the program is initializing. After a few moments the program will close the OPEN FILE edit box, the blank ETABSIN screen will appear and the filename SAMPLE will appear in the title bar with ETABSIN.

b. MODIFYING THE MODEL

The following are the steps to interactively modify the model to introduce the gravity loads and the heading:

i. Define beam span loading patterns

ii. Assign beam loading patterns

iii. Add load cases

iv. Add heading

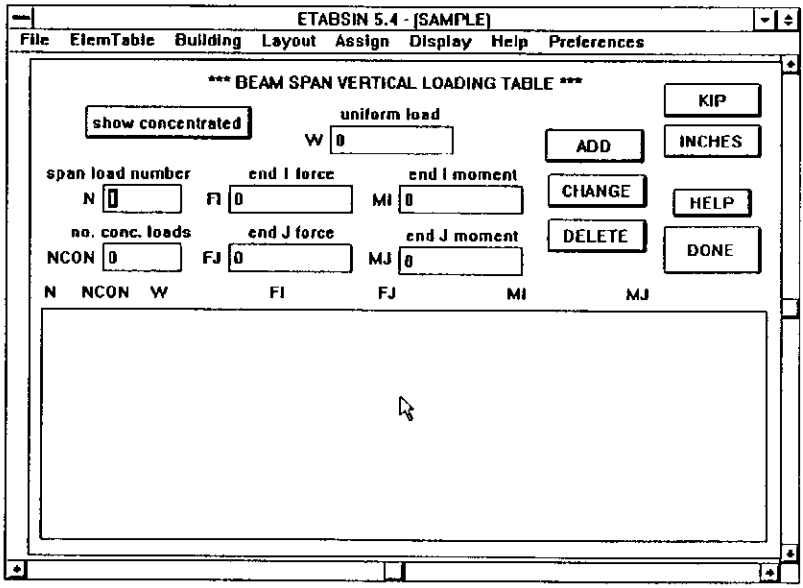
The following subsections (i through iv) of this section correspond to the four steps specified above. Each subsection defines in detail the procedures required to implement the associated steps.

i. Defining Beam Span Loading Patterns

To define beam span loading patterns do the following:

1. Click on **ElemTable** on the menu bar.

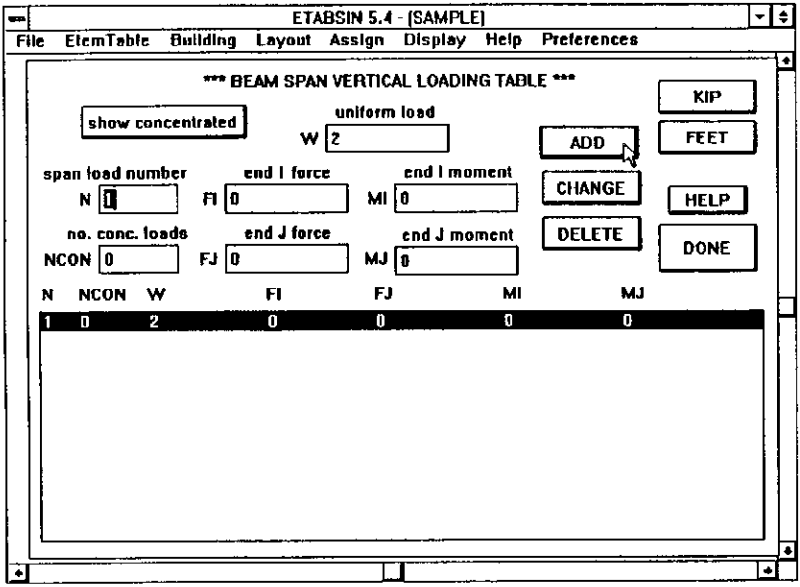
2. Click on **Span Load** in the **ElemTable** pull down menu. The following BEAM SPAN VERTICAL LOADING TABLE dialog box will appear:



3. Click on the button showing INCHES. It will change to FEET. This is the unit we will use to enter the loading patterns.

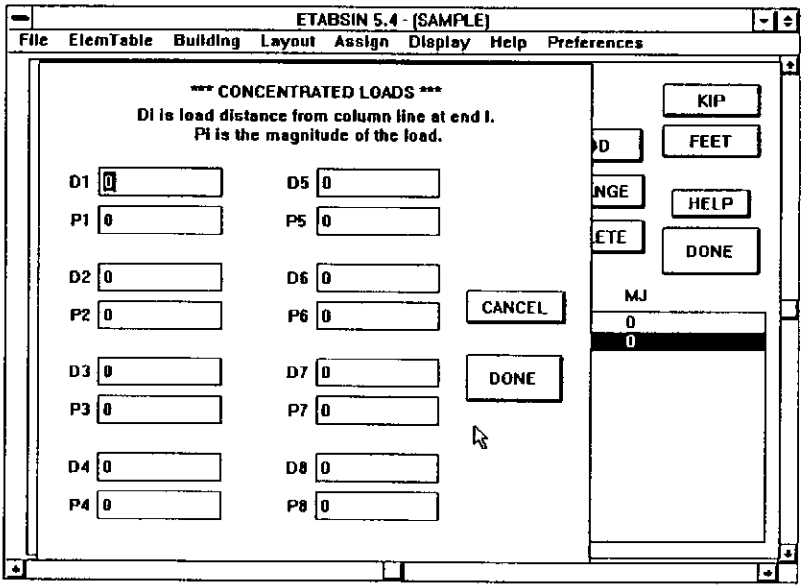
4. Type **2** in the W edit box.

5. Click on ADD. This will add the first loading pattern to the list of patterns and the screen will appear as follows:

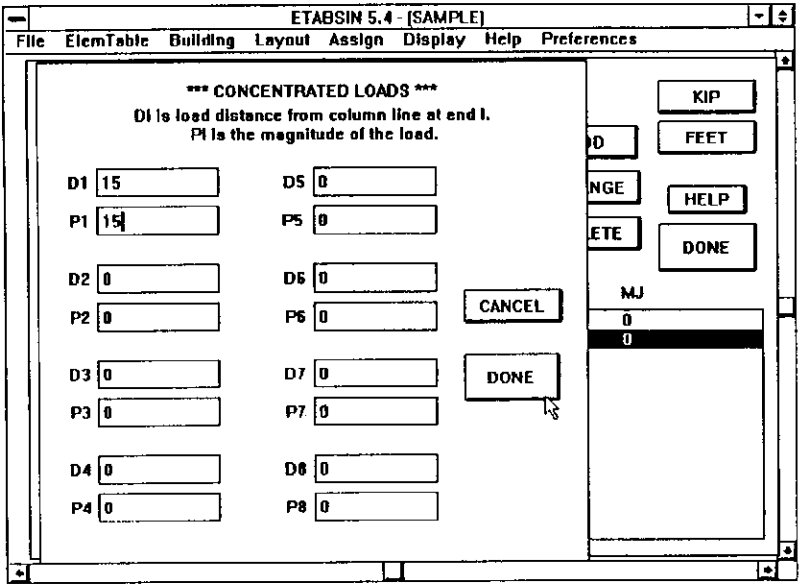


6. There are a total of 6 loading patterns on the example structure. One dead load and one live load pattern on the short bays; and two dead load and two live load patterns on the long bays, one with and one without the point load at the end of the bay. The second loading pattern with only uniform load is added very similar to the first one. Type **2** in the Span load number N edit box, type **0.5** in the W edit box and click on ADD.

7. To add the third loading pattern with concentrated loads and a load at the end the following steps are required. Type **3** in the Span load number N edit box; type **1** in the NCON edit box; type **1** in the W edit box; type **15** in the FJ edit box and then to add the concentrated load on the span, click on SHOW CONC button. The following CONCENTRATED LOADS dialog box will show:

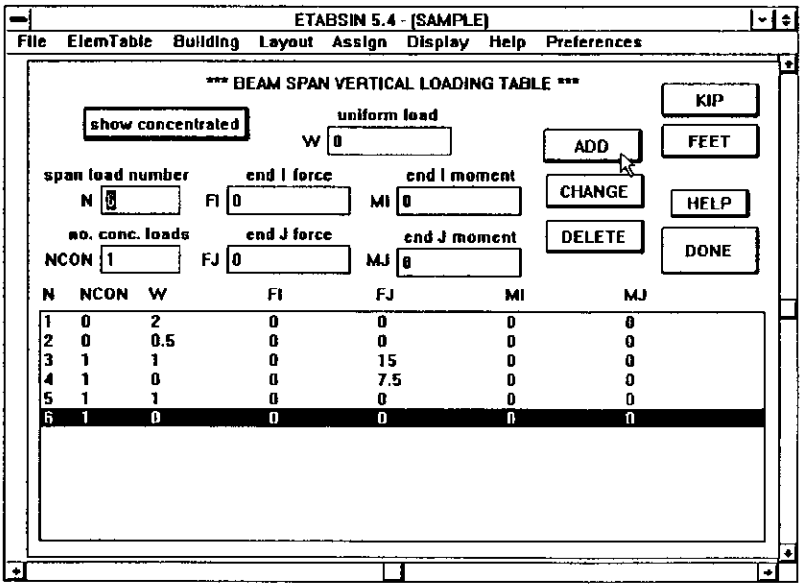


Now type **15** in the D1 edit box and type **15** in the P1 edit box. The screen will look as follows:



Click on DONE.The program will close the CONCENTRATED LOADS dialog box and return to the BEAM SPAN VERTICAL LOADING TABLE dialog box. Click on ADD to add this pattern to the table.

8. Repeat step 7 above to similarly add the remaining three loading patterns. After these loads have been added the screen will appear as follows:



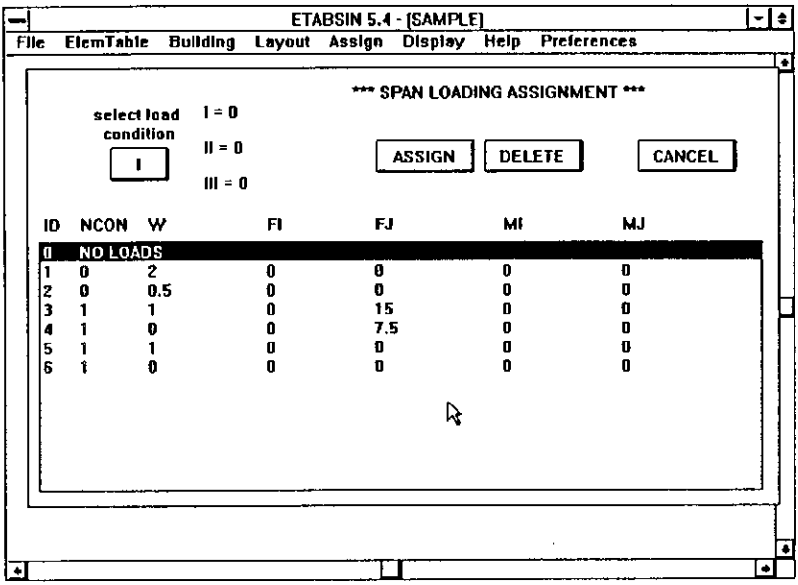
9. Click on DONE. This will close the BEAM SPAN VERTICAL LOADING TABLE dialog box and the program will return with a blank ETABSIN window.

ii. Assigning Beam Loading Patterns

To assign beam span loading patterns do the following:

1. Click on **Assign** on the menu bar.

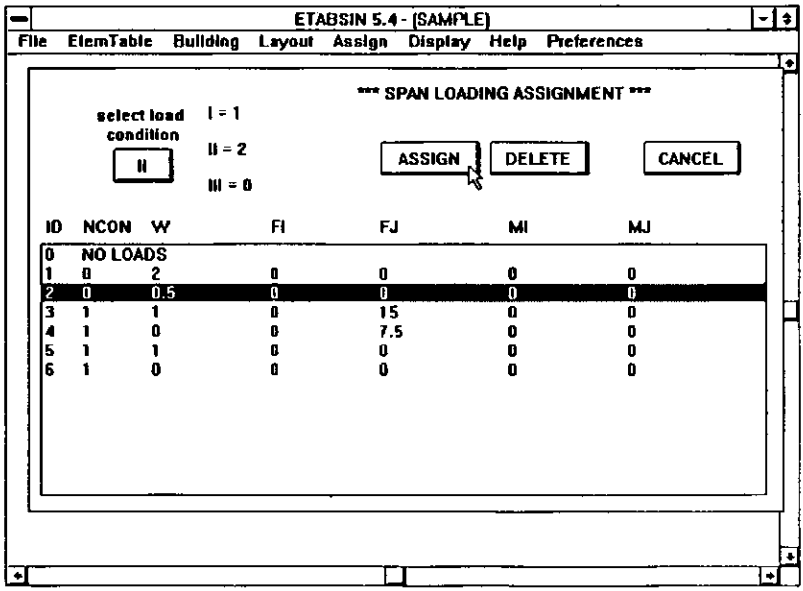
2. Click on **Span Loads** in the **Assign** pull down menu. The following SPAN LOADING ASSIGNMENT dialog box will appear:



3. Click on pattern number 1. This will change the pattern number corresponding to load condition I to 1.

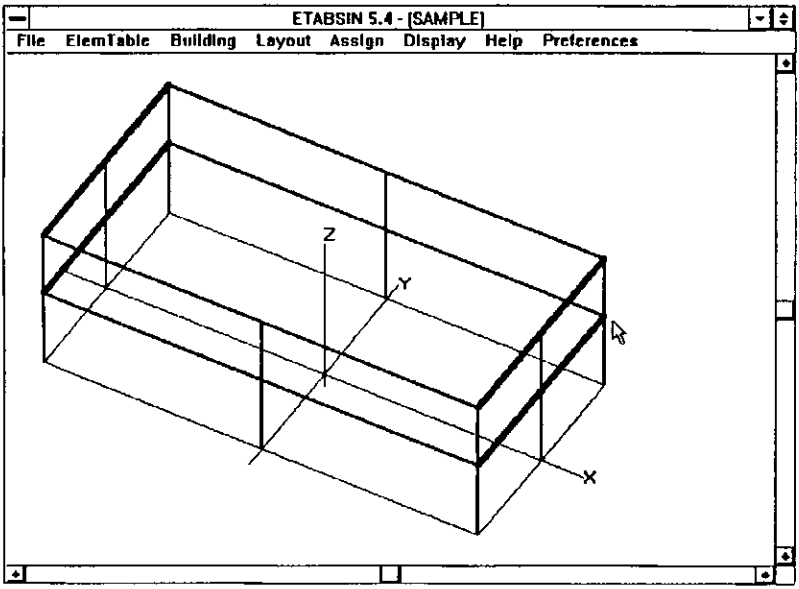
4. Click on the SELECT LOAD CONDITION button. This will change the selection to load condition II.

5. Click on pattern number 2. This will change the pattern number corresponding to load condition II to 2. The screen will appear as follows:

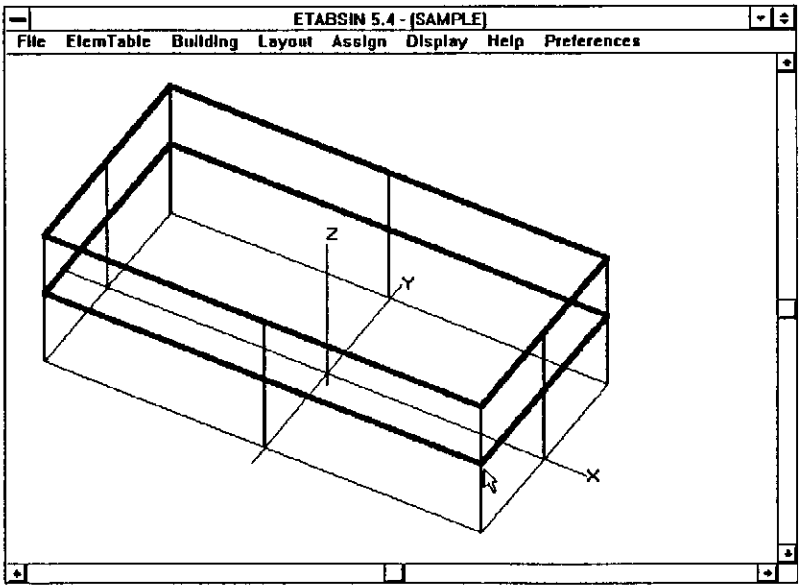


6. Click on ASSIGN. The SPAN LOADING ASSIGNMENT dialog box will close, the program will show a three dimensional view of the structure with the columns in red and the beams in yellow color. The program is now ready to assign beam span load pattern number 1 to load condition I and pattern number 2 to load condition II.

7. The actual assignment of beam span loads graphically on the screen is identical to the assignment of beam section properties to the bays as described in Exercise One. So assign the beam span loads to the short beams. The program will show the beams with span loads assigned with the current assignment with wide magenta lines and the screen will appear as follows:

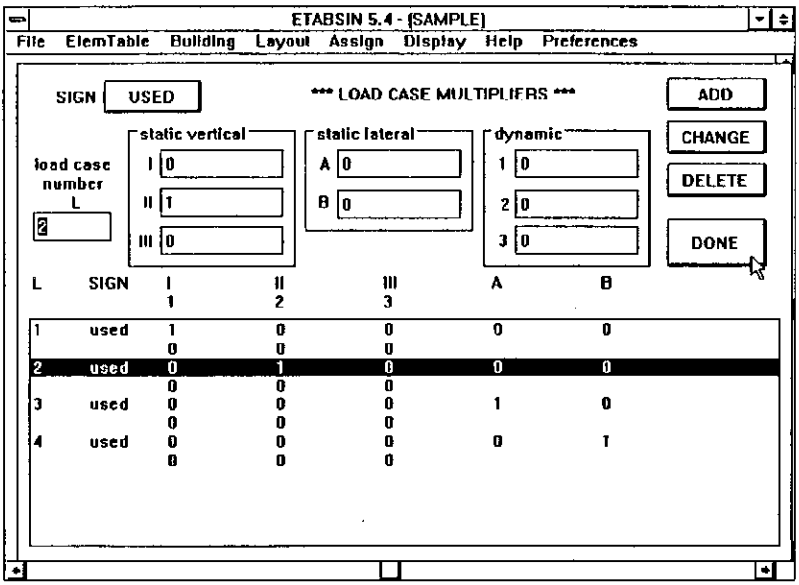


8. Repeat steps 1 to 7 above for the other beams, except choose the appropriate loading pattern numbers. When all beams have been assigned span loads the screen will appear as follows:



iii. Adding Load Cases

Since gravity loads have been added to load conditions I and II, to print their results out separately, two new loading combinations need to be added. Follow the procedure for defining loading combinations described in Exercise One to add these new loading combinations. It should be noted that when new loading combinations are added with Load case numbers L of 1 and 2 the original loading combinations with these numbers are automatically renumbered as 3 and 4. After the new combinations are added the LOAD CASE MULTIPLIERS screen will appear as follows:



Click on DONE to close the LOAD CASE MULTIPLIERS dialog box.

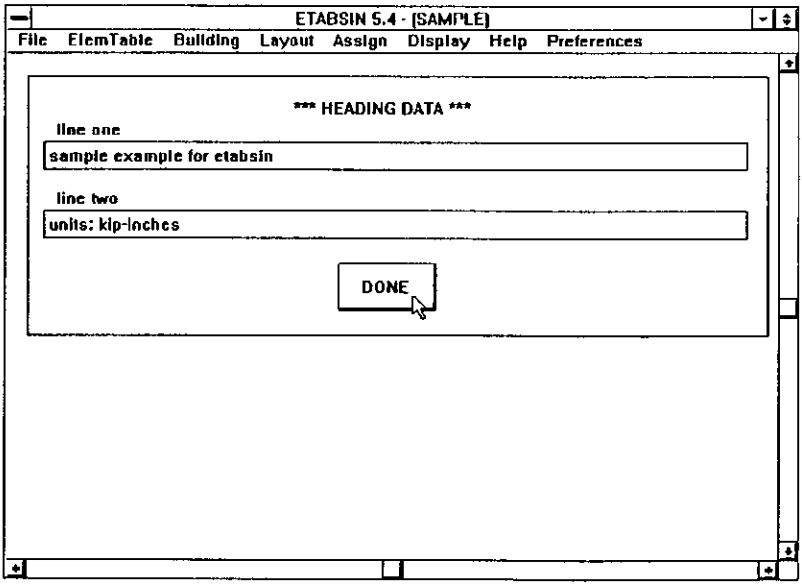
iv. Adding Heading

To add heading information to the ETABS input file to be created do the following:

1. Click on **File** on the menu bar.

2. Click on **Heading** in the **File** pull down menu. The HEADING DATA dialog box will appear.

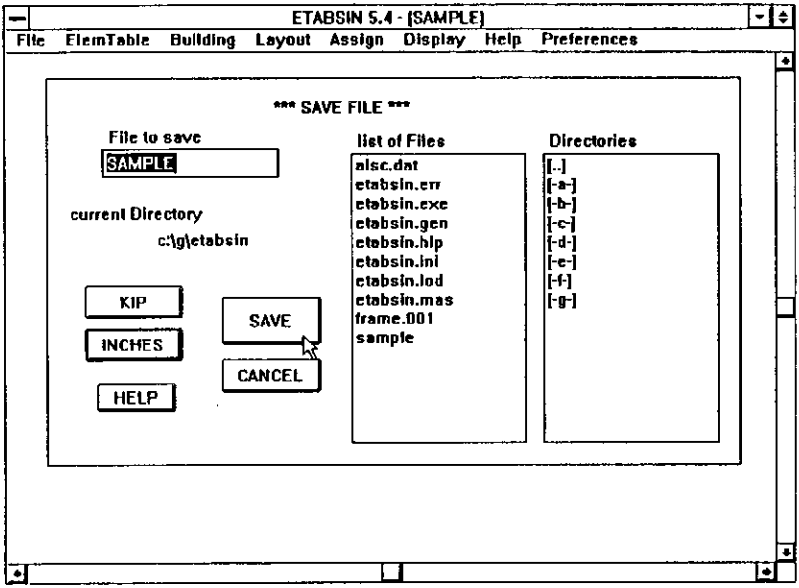
3. Type **sample example for ETABSIN** in the LINE 1 edit box and **units: kip-inches** in the LINE 2 edit box. The screen will appear as follows:



4. Click on DONE. This will close the HEADING DATA dialog box and the program will return the ETABSIN window.

c. SAVING THE REVISED MODEL

To save the new model follow the same steps as for saving the model in Exercise One, except before clicking on SAVE, click on the length unit button several times so it returns to INCHES. Just before saving the screen will appear as follows:



Quit ETABSIN and Windows following the procedure described in Exercise One. The revised SAMPLE file now created will be as shown in Figure III-4. This file can now be used in ETABS for the analysis of the structure we have modeled.

```
$ This is file SAMPLE written by ETABSIN on Fri May 29 17:12:49 1992
$ Units are KIP and INCHES
sample example for etabsin
units: kip-inches
$ Control data
 2 1 1 0 4 0 1 1 1 0 0 1 0 0 0 4 0 0 0 5.40 0 0
$ Miscellaneous parameters
 386.4 0.0001 0 1
$ Mass data
$ Story data
 roof     120 0 0 0 0 0 0 0 0
 floor    144 0 0 0 0 0 0 0 0
$ Frame member Material property data
   1 S 29000 0.000283 0.3 36 0 0
$ Column section property data
   1   1 W14X120           0 0 0 0
$ Beam section property data
   1   1 W33X118           0 0 0 0 0
$$$$$$$$$$ Frame Control Data

 1 2 8 8 0 0 0 6 1 3 3
$ X Grid line data
-360 0 360
$ Y Grid line data
-180 0 180
$ Column line coordinates and orientation
   1   -360.000    180.000      0.000   1   3   3
   2      0.000    180.000      0.000   2   3   3
   3    360.000    180.000      0.000   3   3   3
   4   -360.000      0.000      0.000   1   2   3
   5    360.000      0.000      0.000   3   2   3
   6   -360.000   -180.000      0.000   1   1   3
   7      0.000   -180.000      0.000   2   1   3
   8    360.000   -180.000      0.000   3   1   3
$ Bay connectivity
   1   1   2
   2   2   3
   3   6   7
   4   7   8
   5   6   4
   6   4   1
   7   8   5
   8   5   3
$ Beam span vertical loading patterns
   1   0 0.166667 0 0 0 0
   2   0 0.0416667 0 0 0 0
   3   1 0.0833333 0 15 0 0
 180 15
   4   1 0 0 7.5 0 0
 180 7.5
   5   1 0.0833333 0 0 0 0
 180 15
   6   1 0 0 0 0 0
 180 7.5
$ Column Location Data
   1 0 roof        1   1   0   0
   2 0 roof        1   1   0   0
   3 0 roof        1   1   0   0
   4 0 roof        1   1   0   0
   5 0 roof        1   1   0   0
   6 0 roof        1   1   0   0
   7 0 roof        1   1   0   0
   8 0 roof        1   1   0   0

$ Beam Location Data
   1 0 roof        1   1   0
   2 0 roof        1   1   0
   3 0 roof        1   1   0
   4 0 roof        1   1   0
   5 0 roof        1   1   0
   6 0 roof        1   1   0
   7 0 roof        1   1   0
   8 0 roof        1   1   0
```

Revised File SAMPLE
Figure III-4

```
$ Span Loading Location Data
   1 0 roof        3   4   0   1
   2 0 roof        5   6   0   1
   3 0 roof        3   4   0   1
   4 0 roof        5   6   0   1
   5 0 roof        1   2   0   1
   6 0 roof        1   2   0   1
   7 0 roof        1   2   0   1
   8 0 roof        1   2   0   1

$$$$$$$$$$ Frame Location data
   1    0        0.000        0.000    0.0
$$$$$$$$$$ Building Loading
$ User Defined Static Lateral Loads
 20 0 0 0 0 20 0 0
 10 0 0 0 0 10 0 0
$$$$$$$$$$ Building Load Cases
   1 0      1.00      0.00      0.00      0.00      0.00      0.00      0.00      0.00
   2 0      0.00      1.00      0.00      0.00      0.00      0.00      0.00      0.00
   3 0      0.00      0.00      0.00      1.00      0.00      0.00      0.00      0.00
   4 0      0.00      0.00      0.00      0.00      1.00      0.00      0.00      0.00
```

Revised File SAMPLE (continued)
Figure III-4

IV.

REFERENCE

The reference section provides a detailed explanation for each of the menu items. It is organized in the order in which the menus appear.

Most of the dialog boxes in ETABSIN have HELP push buttons. Click on these to display an abbreviated help message. Many of the dialog boxes also have Units push buttons. In this manual, all examples are shown as KIP and INCHES. Click on these buttons to change units as desired, but be aware that the units are changed in the ENTIRE program.

The **File** menu allows you to define, edit and save the files that ETABSIN uses. ALL files are "ETABS INPUT DATA FILES" as defined in Chapter V of the ETABS Users Manual. These files are in the standard ASCII format and may be edited using any text editor in ASCII mode. DO NOT use a text editor in non-ASCII mode as it will put characters in the file that can not be recognized by either ETABSIN or ETABS. Following is a definition of **File** commands:

File Help

Clicking on **File Help** brings up a dialog box which contains abbreviated information on using the **File** menu.

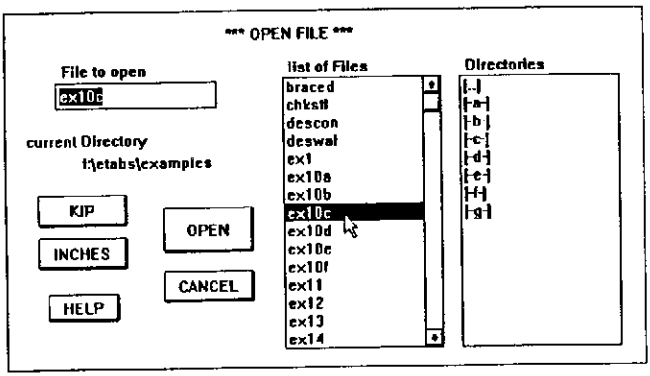
New

Clicking on **New** resets ETABSIN to the state it is in when started. If there is any structure defined, it will all be erased, including all elements and assignments. Further, the default units are set from the file ETABSIN.INI (if there is one).

Open

Open allows you to read in the current structure from an existing file. The file becomes the current file and its name appears at the top center of the screen enclosed in parentheses.

When you click on **Open**, this dialog box appears:



First select the proper Current Directory from the Directories list box. The names in this box are drives or directories. For example, [-A-] is drive A: and [name] is directory name. Clicking on a drive will make the root directory of that drive the current directory. Clicking on a directory will make it the current directory. There is the special case of [..] which means the next higher directory level, just as it does in DOS. Clicking on [..] will make the next higher directory the current directory. If the root directory is the current directory, then there is no higher level directory and [..] will not appear.

With the correct current dircctory selected, you can now select a file by clicking on it in the list of files, or alternatively, move the cursor to the File to Open edit box and enter the desired name. Then click on the OPEN button to open the file. You can shortcut this procedure by just double-clicking on the file name in the list box.

The current structure will be read in from the file, with the units specified, OVERWRITING any existing structure.

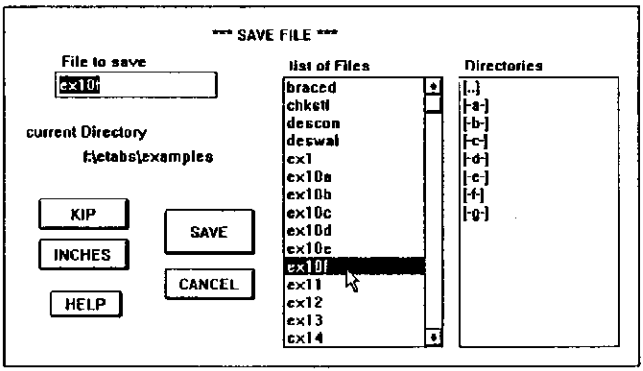
Save

Clicking on **Save** OVERWRITES the current file (name at top of screen) with the current structure. The file will be saved in the units that are currently set. If these units do not match the preference units, you will be allowed to cancel the save if desired.

This is the easy way to periodically save your work, but it is a good idea to change the file name using **Save as** every so often to avoid losing your work.

Save as

Save as allows you to save the current structure in a file whose name you specify. When you click on **Save as**, the following dialog box appears:



First select the proper current directory in exactly the same way as described for **Open**.

Then select a file by clicking on it in the list of files, or if it is not in the list, move the cursor to the File to Save edit box and enter the desired name. Then click on the SAVE button to save the file. Double clicking on the file name in the list of files will also save the file.

The current structure will be written to the file, which becomes the current file. If the file exists, you can choose to OVERWRITE the file with the current structure or cancel the save.

The file will be written in the units specified at the time the file is saved, regardless of the units you may have been previously using. If the units do not match the preference units, you will be allowed to cancel the save if desired.

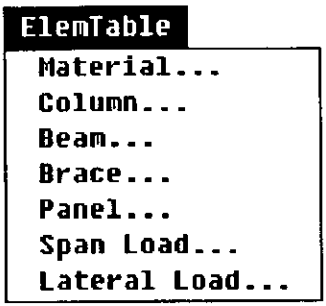
Heading

The first two lines of every ETABS file are **Heading** Data, which may contain any information you desire. Each line is a maximum of 70 characters. Move the cursor to the desired edit box and type in what you want.

When you do **Save** or **Save as**, the current heading information will be written to the file.

About

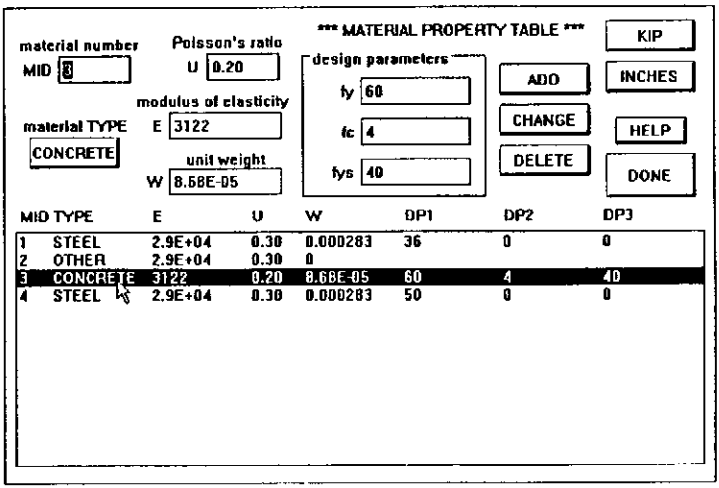
Clicking on **About** displays a dialog box which shows copyright information and the size of available memory.



The **ElemTable** menu allows you to define the materials to be used in the structure and the properties of the various structural members such as beams and panels. You can also define beam span vertical loading patterns and column lateral loading patterns which can be applied to the completed structure. The **ElemTable** commands are as follows:

Material

When you click on **Material**, this dialog box appears:



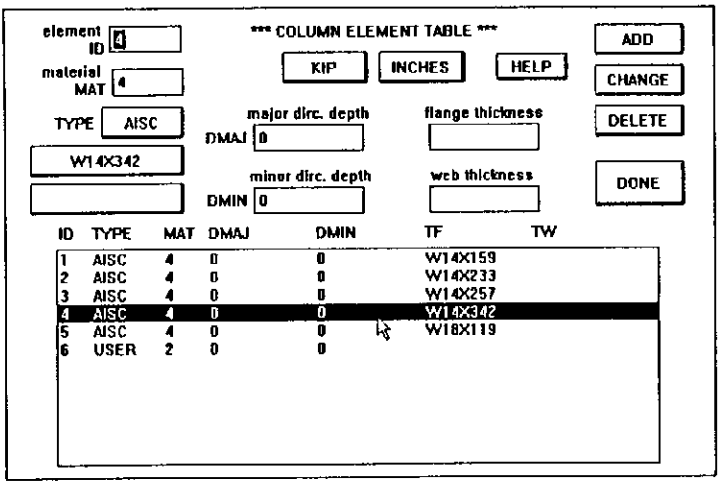
Click on the TYPE button to select the desired material type. Note that the different types have different parameter names but are all shown in the list box as DP1, DP2 and DP3. For the STEEL type, Fy is DP1, Fbmaj is DP2 and Fbmin is DP3.

Enter the parameters desired and then click on ADD to add a new material type.

Clicking on an existing material in the list box will bring its parameters up into the edit boxes. You can then change it by changing the parameters and clicking on CHANGE. Clicking on DELETE will delete the existing material.

Column

Column properties are defined using the **Column** command. Note that you must have already defined all of the materials that you will need to define the column properties. When you click on **Column**, the following dialog box appears:

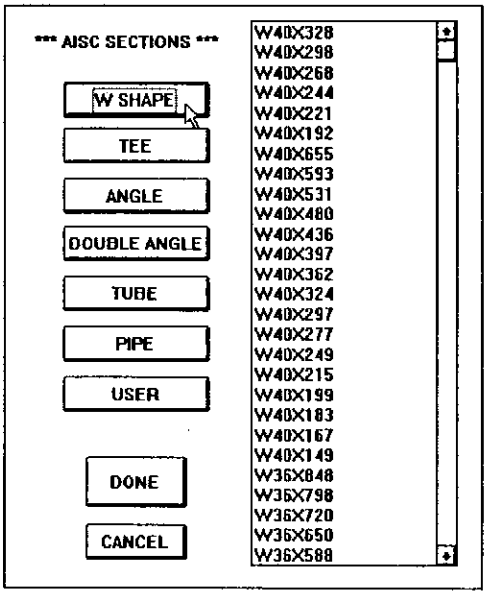


The ID is just a number to aid in identification of the column properties, and is always consecutive from one. The material identification number MAT is the ID number of a previously defined material.

Clicking on TYPE will cycle through a set of available types. There are a number of types, including AISC sections, geometrical shapes and a general USER type.Each of these types has a different set of parameters and some have more than others. When entering parameters, any edit boxes without labels should not be used.

AISC TYPE:

The AISC type requires further selection from the AISC section table. The button immediately below the TYPE button shows the currently selected AISC section, or if none has been selected, will show SELECT TYPE. Click on this button to bring up the following dialog box:



Note that if files with a .DAT extension (other than AISC.DAT) are present in the directory where ETABSIN.EXE resides, the program will bring up a small dialog box asking you to choose, if desired, one of them as the user section property file. Once this is done, the above AISC sections dialog box will appear and the user section properties will be available through the USER button. If no other file with the .DAT extension is found, this intermediate step is omitted. The user file is only valid if it was produced using program PROPER, which is part of the ETABS group of programs.

The first time that the AISC type is selected, there is a delay of a few seconds to generate the AISC table. After that, the delay is negligible.

The AISC sections are divided into groups for ease of selection. Click on the group desired and then select the desired section by clicking on it in the list box. Finally, click on DONE and return to the Column Element Table dialog box. The AISC section that you selected is now shown in the button under the TYPE button.

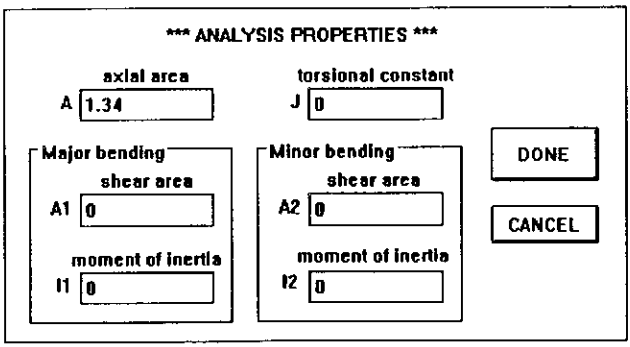
If you want to overwrite the AISC parameters, then you can enter the remaining parameters in the edit boxes, DMAJ and DMIN. Do not use the unlabeled edit boxes.

GEOMETRICAL TYPE

Geometrical types do not require anything other than filling in the parameters in the edit boxes. Again, do not use edit boxes without labels.

USER TYPE:

The User type has two additional sets of parameters, ANALYSIS and DESIGN. Click on the ANALYSIS button and this dialog box will appear:



Enter values as desired and click on OK. Clicking on the DESIGN button produces a similar dialog box with pa-

rameters that can be set as you desire. Finally, enter the remaining parameters, DMAJ and DMIN.

Column ADD, CHANGE and DELETE
With the TYPE selected and other parameters entered, you can then enter an ID and click on ADD to add a new column type.

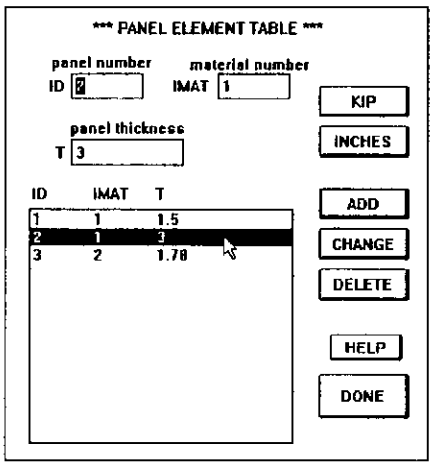
Clicking on an existing column in the list box will bring its parameters up into the edit boxes. You can then change it by changing the parameters and clicking on CHANGE . Clicking on DELETE will delete the existing column.

Beam
Brace

Although some of the parameters are different, **Beam** and **Brace** properties are defined in exactly the same way as column properties. Select a TYPE, enter the desired parameters and use the ADD, CHANGE or DELETE buttons to make up your list of beam or brace properties. See the **Column** section previous to this for details.

Panel

Clicking on **Panel** will show the following dialog box:

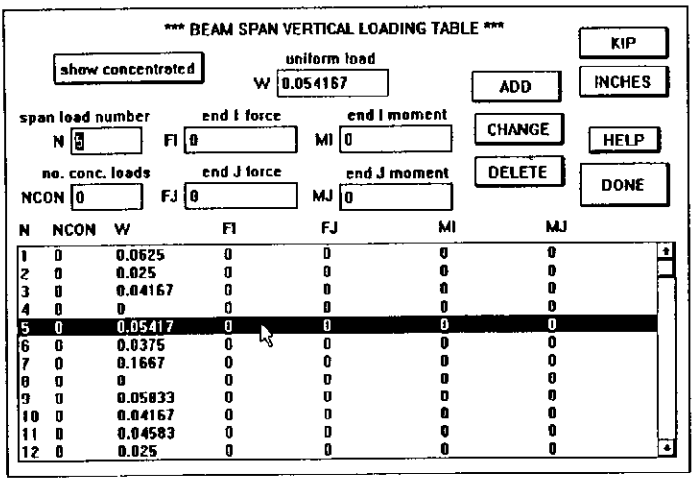


ID and MAT have the same meaning as for columns. There is only one parameter to enter, THICKNESS. Use the ADD, CHANGE and DELETE buttons in the same way as for column properties.

Span Load

Beam Span Vertical Loading Patterns are defined in the same way as structural properties for columns and beams. When the loads are later assigned to beams, they are picked from the list of span loads that you have defined here.

Clicking on **Span Load** brings up this dialog box:



Enter the parameters W, FI, FJ, MI and MJ as desired.

If you have concentrated loads, please refer to the ETABS manual section "Beam Span Vertical Loading Patterns" for a description of how concentrated loads are defined. If distances are positive, then the distances must be increasing from load number 1 to load number NCON. If distances are negative, then they indicate ratios of load distances to beam length, they must be between 0.0 and -1.0, and they must be increasingly negative from load 1 to load NCON. Enter NCON and then click on SHOW CONC which will bring

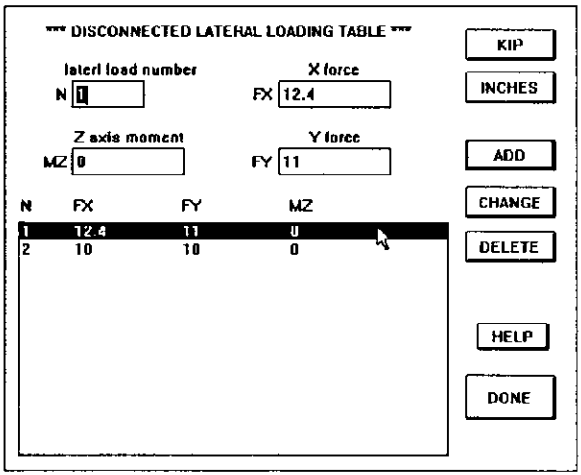
up a dialog box where you can enter D and P for each concentrated load. Any entries past NCON will not be used.

Finally, enter N and use the ADD, CHANGE and DELETE buttons to make the list of Span Loads in the same way as for column properties.

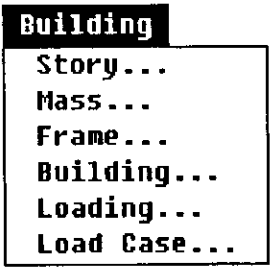
Lateral Load

Disconnected Column Lateral Loads are defined in the same way as structural properties for columns and beams. When the loads are later assigned to columns, they are picked from the list of Lateral Loads that you have defined here.

Clicking on **Lateral Load** brings up this dialog box:



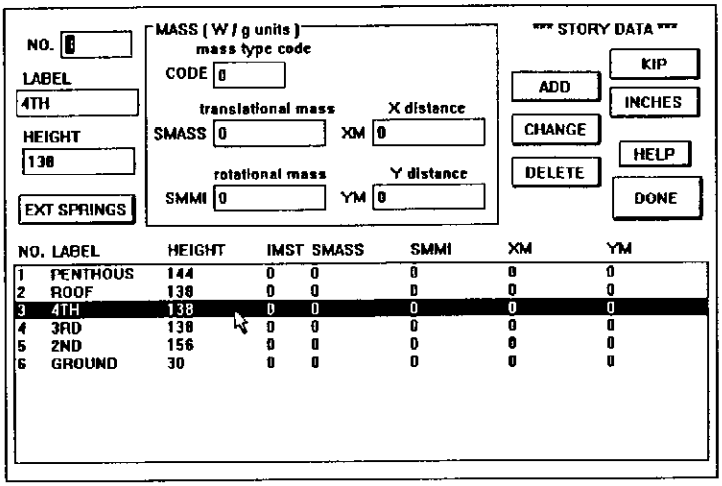
Enter values for FX, FY and MZ as desired; then enter N and use the ADD, CHANGE and DELETE buttons to make the list of Lateral Loads in the same way as for column properties.



The **Building** menu is where you define the stories for the building and the frame(s) that will be used. You also define the story masses, if used, and can set the building static lateral loads using the various building codes or user defined loads. Finally, one or more load cases can be defined. Please keep your ETABS Users Manual handy, particularly for the loading, as descriptions for loading parameters are only given in that manual and not in the ETABSIN manual.

Story

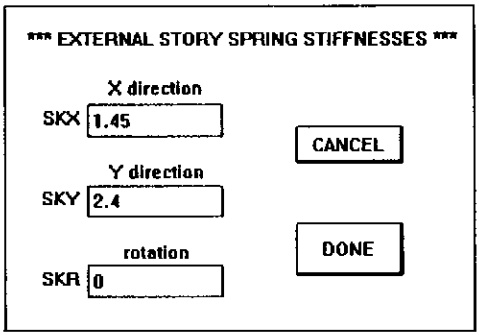
This is where you define the stories for the building. Note that individual frames may have fewer stories, with the topmost story defined in the Frame dialog box. When you click on **Story**, the following dialog box appears:



Be careful here, as the stories MUST be defined with the top story as number 1, and the remaining stories in downward order. Though it is not shown in the list, the bottom

of the structure is always the BASELINE, which you should not define.

Enter the story Number, the Label and the Height. Note that the Label is 8 characters maximum and can NOT contain blanks. Use an _ or a - to simulate a blank. If there are external story springs, click on the EXT SPRINGS button and this dialog box will appear:



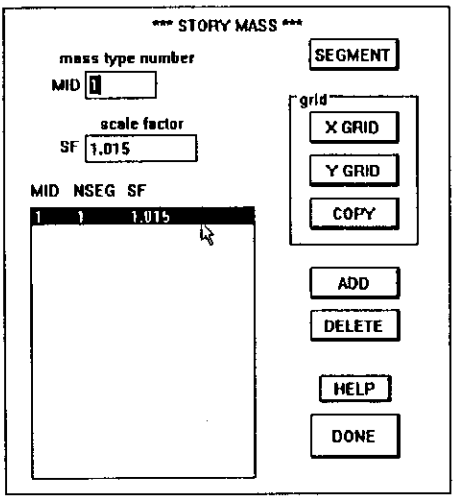
Enter the desired values and click on DONE.

You may specify story masses in two different ways. The first way is to enter a mass CODE of 0 and then specify the values for the translational mass, SMASS, the rotational mass moment of inertia, SMMI, and the location of the mass centroid, XM and YM. The second way is to enter a non-zero CODE which refers to the mass type MID in the Mass dialog box (described later).

Use the ADD, CHANGE and DELETE buttons to make up the list of stories in a similar fashion to the column property definition described earlier. Don't forget that the list must include all stories for your tallest frame.

Mass

Mass allows the graphical definition of story masses for the building. Clicking on **Mass** shows:

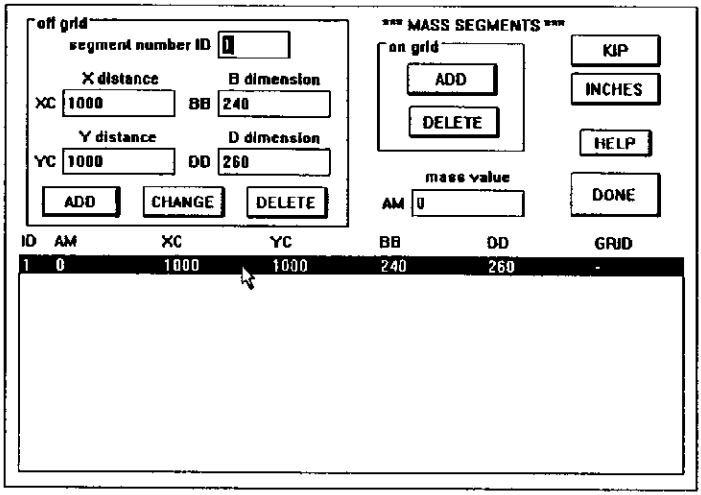


You MUST have a grid to define masses graphically, so the first thing to do is to define a grid. You can do this by clicking on the X GRID button and then the Y GRID button in exactly the same way as grids are defined for a frame. This process is explained later under the Layout Menu. Alternatively, you can copy an existing frame grid. To do this just click on the COPY button. CAUTION, be sure that you have selected the frame whose grid you want to copy (if you have multiple frames). Select the frame using the **Frame** command in the **Building** menu.

When the grid is defined, then you can define different mass types which can be assigned to the various stories. Each mass type is made up of one or more mass segments, and each type has a dimensionless scale factor, SF.

Before defining mass segments, first enter SF and MID and then click on ADD. This is necessary because a type must be selected in the list box BEFORE segment definition.

Click on the mass type in the list box and then click on the SEGMENT button. This dialog box will appear:



Enter a MASS value and then click on ON GRID ADD to add segments graphically. Note that you can make Point, Line or Area masses and that the units of the MASS value should be mass (W/g) units since the scale factor (SF in the Story Mass dialog box) is dimensionless. Clicking on ON GRID DELETE will allow you to delete segments, but only those that are ON GRID.

To ADD Point masses, point to a grid intersection and click the left button. A CYAN circle will be drawn to designate a point mass.

To ADD Line masses, point to a grid intersection, press and hold the left button, move to another intersection on the SAME grid line (X or Y) and release the button. A YELLOW line will be drawn to represent a line mass.

To ADD Area masses, point to a grid intersection, press and hold the left button, move to another intersection not on

EITHER grid line and release the button. A RED rectangle will be drawn to represent the area segment.

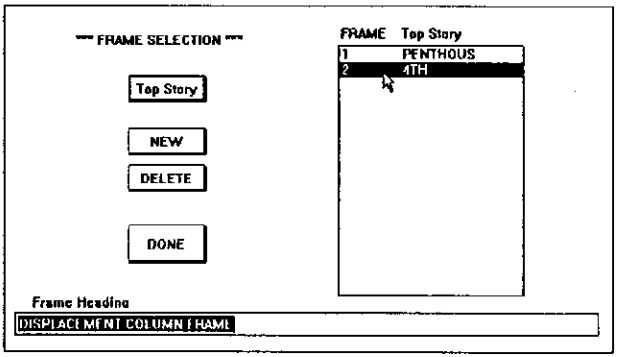
To DELETE masses, follow exactly the same procedure as for adding, but do it over an existing mass.

When the Mass Segment dialog box is showing, you can also ADD mass segments that are not on the grid by entering their values. See the ETABS manual "Automatic calculation of Story Mass" for a definition of the parameters XC, YC, BB and DD. Click on OFF GRID ADD to add the segment to the list. Click on existing segments in the list box to bring their values up into the edit boxes and then use CHANGE or DELETE buttons as desired. This works with both ON and OFF GRID segments.

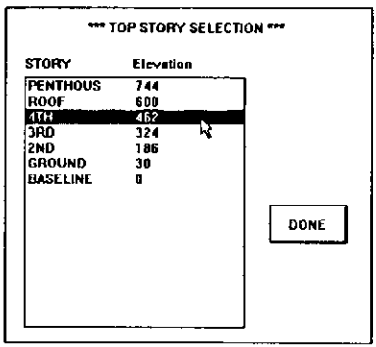
Frame

Frame allows you to define multiple frames for your structure and to select the current frame. Note that the **ElemTable** and **Assign** menus will ONLY show data for the current frame. Data for other frames (if there are more than one) is kept in files on disk and is read in only when that frame is made current.

Clicking on **Frame** shows this dialog box:



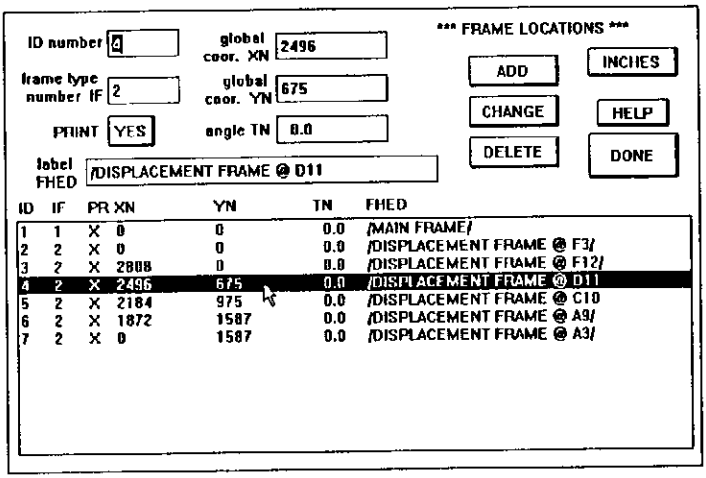
To define a new frame, click on NEW. To select a frame (make it the current frame) click on the frame in the list box highlighting it. To delete a frame, select it in the list box then click on DELETE. You can enter a descriptive heading for each frame and set the top story by clicking on TOP STORY which brings up the following dialog box:



Click on the desired top story in the list box and then click on DONE. Note that there is real advantage to defining the stories FIRST before any additional frames are defined. If you define stories later, then you will have to redo the frames and in some cases assignments can be lost.

Building

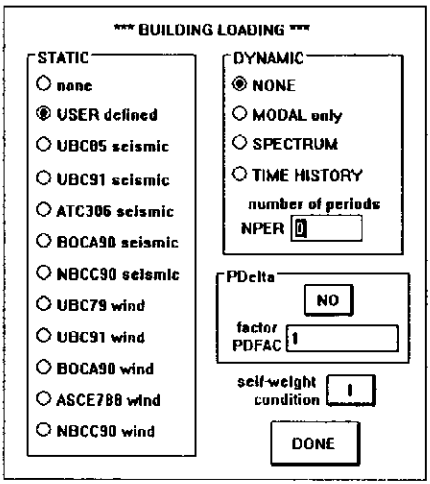
This is where you define the building layout (the location of the frames relative to the global origin) using the frames that have previously been defined. Clicking on **Building** brings up this dialog box:



You can use a frame any number of times, each with its own X,Y location and ANGLE. Note that XN and YN are global coordinates, and the values to be specified are from the global origin to the origin of the frames own local coordinate system. Enter the Frame type number (from the Frame dialog box list) and other parameters and then use the ADD, CHANGE and DELETE buttons to make the list of frame locations in the same manner as for column properties (in the ElemTable menu).

Loading

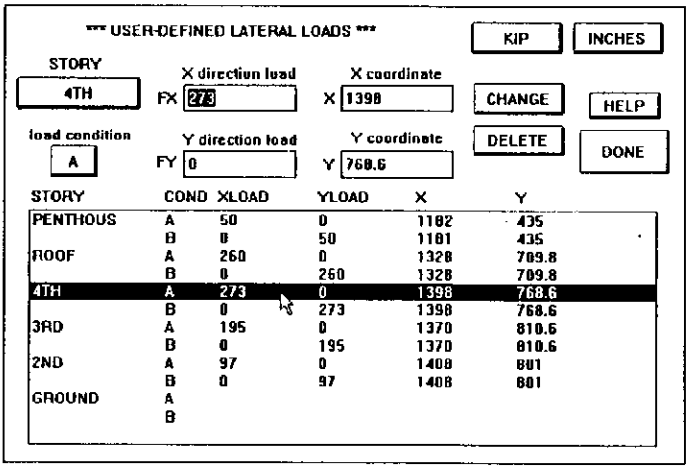
Clicking on **Loading** brings up this dialog box:



Please refer to your ETABS Users Manual "Structural Static Lateral Load Data" for a complete description.

User Defined

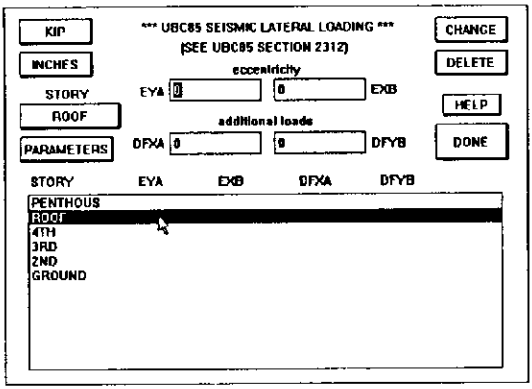
To define a general-purpose static lateral loading, click on USER DEFINED, which will show this dialog box:



If the list box lines for a story are blank, that means that all of the loads for that story are zero. To enter loads for any story, click on that story (either load condition A or B) in the list box, and enter the desired loads and position in the edit boxes. Then click on CHANGE. You can also click on the STORY button and the LOAD CONDITION instead of clicking in the list box. Click on DONE to return to the Building Loading dialog box.

Seismic Loading

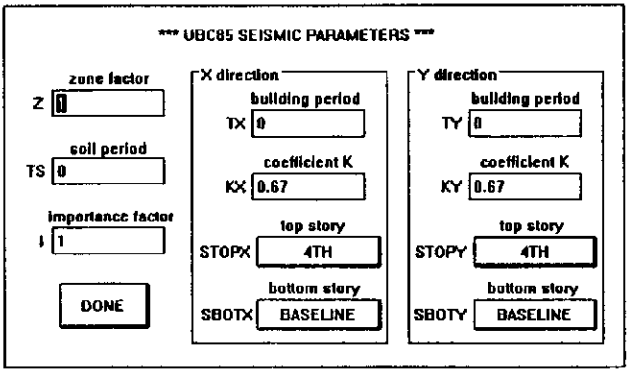
To define seismic loading according to various codes, click on one of the SEISMIC items. For example, clicking on UBC85 SEISMIC brings up this dialog box:



If the data in the list box for a story is blank, that means that all the values are zero. For any story, to enter eccentricities (distances from center of mass to point of application of seismic load) and/or additional loads, click on that story in the list box and then enter the desired loads and position in the edit boxes. Then click on CHANGE. You can also click on the STORY button instead of clicking in the list box.

Clicking on PARAMETERS brings up this dialog box:

You need to refer to the UBC85 code for a clear definition

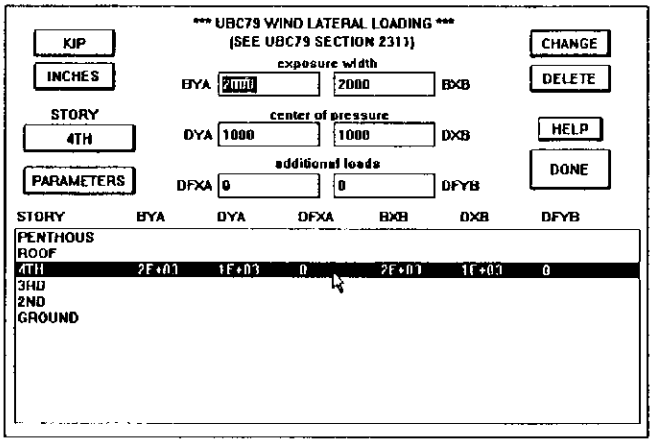


of all these parameters. Enter the desired values in the edit boxes and then set the TOP and BOTTOM STORY by clicking on the butttons.

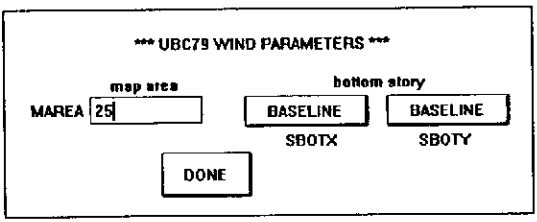
The other seismic loadings, UBC91, ATC306, BOCA90 and NBCC90, have different parameters, but are essentially the same as UBC85.

Wind Loading

To define wind loading according to various codes, click on one of the WIND items. For example, clicking on UBC79 WIND brings up this dialog box:



If the data in the list box for a story is blank, that means that all the values are zero. To enter loads for any story, click on that story in the list box and enter the desired loads and position in the edit boxes. Then click on CHANGE. You can also click on the STORY button to set the story instead of clicking in the list box. Clicking on the PARAMETERS button will bring up this dialog box:



Enter the desired MAP AREA and set the BOTTOM STORY by clicking on the buttons.

The other wind loadings, UBC91, BOCA90, ASCE 7-88 and NBCC90, have different parameters, but are essentially the same as UBC79.

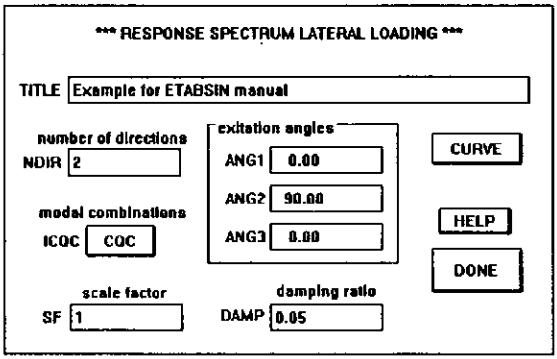
Dynamic Loading

Please refer to the ETABS manual "Data for Response Spectrum Dynamic Loading" and "Data for Time History Dynamic Loading" for a complete description of dynamic loads. Before selecting a dynamic load, be sure to set NPER to a value greater than zero, otherwise the loading will have little meaning.

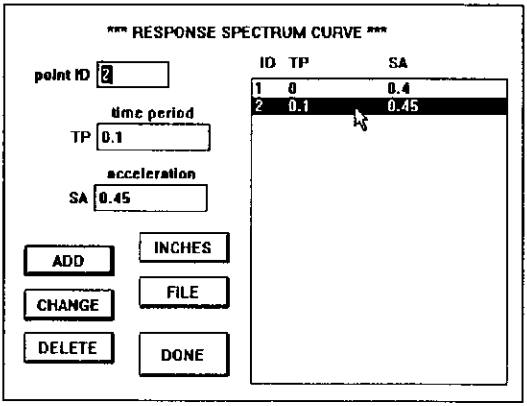
Response Spectrum

The purpose here is to define a ground acceleration spectrum curve to be used in a dynamic response spectrum analysis. The curve data can be entered by hand or from an external file.

Click on SPECTRUM to show this dialog box:



Enter the desired data in the edit boxes and click on the ICQC button to select the Modal Combination. Click on the CURVE button to show this dialog box:



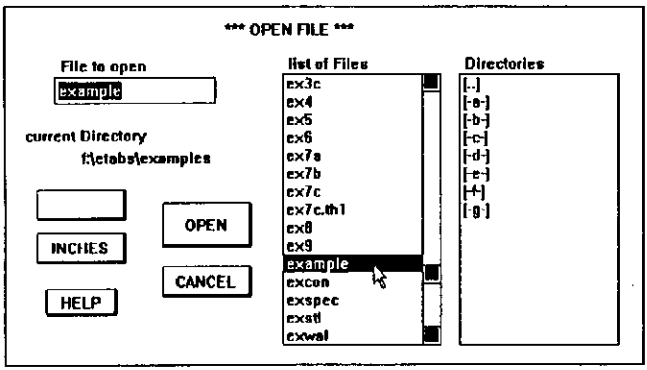
Enter your curve data in the edit boxes and use ADD to add new curve points. Click on an existing curve point in the list box, change the data in the edit boxes and click on CHANGE to change the existing point or click on DELETE to delete the existing point.

Note that the period MUST be increasing, point to point, but the acceleration can be any value.

It is posssible to read in curve data from a file. The data must be in ASCII characters, one point to a line, with period and acceleration separated by spaces. Exponential format is allowed, but no letters are allowed other than "E" or "e". For example:

0.0	0.15
0.15	1.3E-1
1	3.2

When you click on FILE the following box appears:

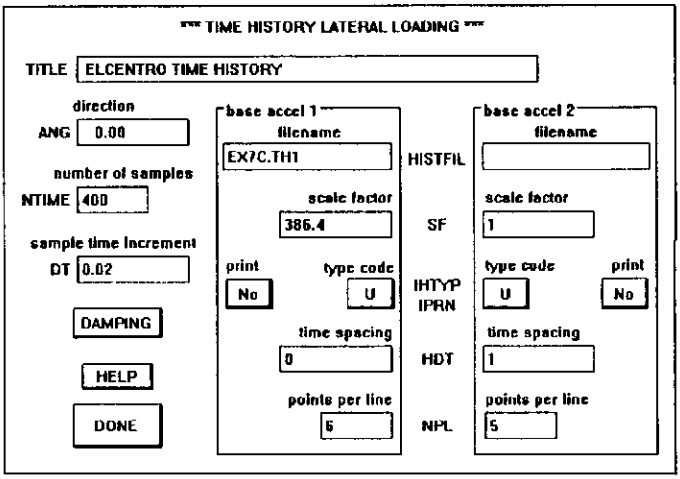


This is the same dialog box that is used in the FILE OPEN menu, described at the beginning of this chapter, and should be used in the same way. If the file is not the proper format, as described above, then it can not be read.

Time History

The purpose here is to define a ground motion accelerogram for a dynamic time history analysis. Damping can be specified for each period and the accelerogram data can be entered by hand or from an external file.

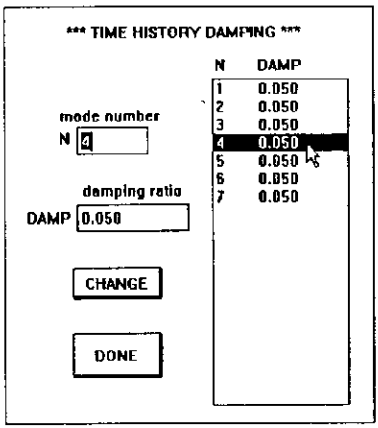
Click on TIME HISTORY to show this dialog box:



Unlike response Spectrum, the acceleration curve data MUST be in a file and can not be seen or edited within ETABSIN. There are two type of accelration data, depending on the selection of IHTYP. If IHTYP = E, then the curve has acceleration values spaced at equal time intervals, so the time need not be specified at each point. If IHTYP = U, then the time must be specified at each point. See the ETABS manual section "Input Data - TIME HISTORY" for a description of the file formats. Note that the file does not have to exist. It will not be checked until the analysis is done.

Enter your data in the edit boxes. Note that if a file name HISTFIL, is blank, then the rest of the data for that file has no meaning.

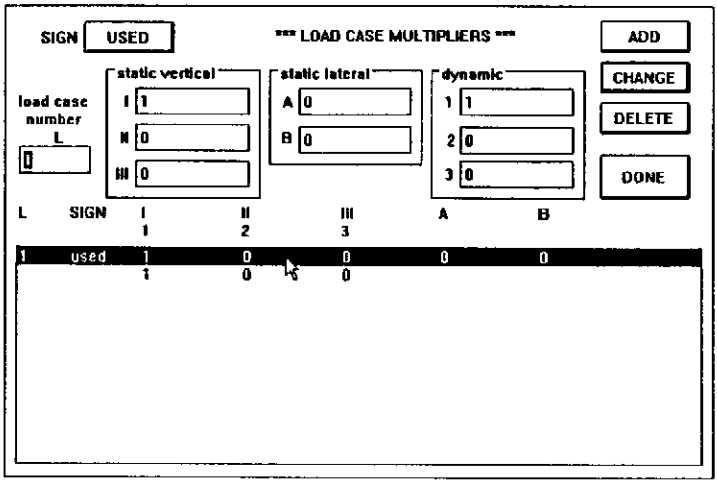
To set the damping. click on DAMPING, which will show this dialog box:



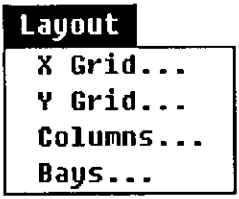
To change a damping value, click on it in the list box, change the value in the edit box and click on CHANGE. Click on DONE to return to the Time History dialog box.

Load Case

Clicking on **Load Case** brings up this dialog box:



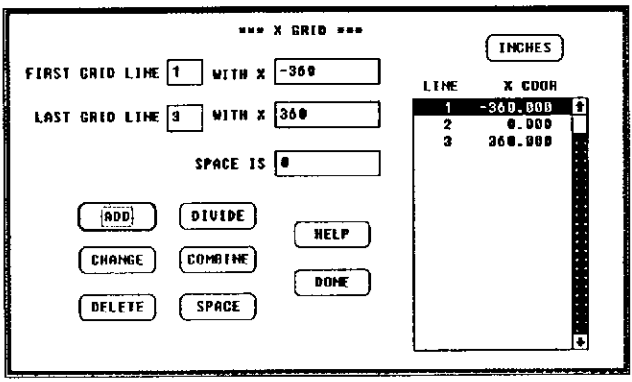
Refer to the ETABS Users Manual "Load Case Definition Data" for a description of the load case multipliers. Enter the multipliers and click on ADD to add a new load case. Click on an existing load case in the list box to bring its values into the edit boxes. Then change values as desired and click on CHANGE to change that case or click on DELETE to delete the case.



The **Layout** menu is where you define the basic frame layout in plan view. Defining a grid allows placement of column lines at grid intersections using the mouse and then definition of bays between two column lines. The column lines and bays are automatically extruded upwards from the BASELINE to the highest story defined. Note that there are NO structural members defined here, but only the framework on which structural members are assigned.

X Grid

Clicking on **X Grid** brings up the following dialog box:



The X axis runs horizontally from left to right on the screen. The X grid lines run vertically and cross the X axis at the X COOR specified for each line. Coordinates can be any value, plus or minus. You can define multiple lines at one time by specifying the coordinate of the FIRST grid line and the coordinate of the LAST grid line in the edit boxes. The grid lines will be generated with equal spaces between them. You can also do the same thing by specifying the coordinate of the FIRST grid line, setting the coordinate of the LAST grid line to 0, and specifying the SPACE between the FIRST and LAST grid lines. In either case, clicking on

ADD will add the grid lines to the list. Note that you can not define two grid lines with the same coordinate. There is an additional restriction in that the grid line coordinates must be increasing with increasing grid line number.

Click on an existing grid line in the list box to bring its coordinates into the edit boxes. You can then click on DELETE to delete the line or change the coordinate and click on CHANGE, keeping in mind the restrictions mentioned above.

To divide the space between two consecutive existing grid lines into multiple lines, click on the lower numbered line in the list box and then enter the FIRST line + the number of spaces desired into the LAST line. For example, to divide the space between lines 3 and 4 into 5 spaces, creating 4 new grid lines, click on line 3 in the list box and enter 8 (3 +5) into the LAST grid line. Then click on DIVIDE. This will make line 4 become line 8 and create new lines 4, 5, 6 and 7. All lines above line 8 will be renumbered.

To combine multiple consecutive grid lines into a single grid line, click on the lowest numbered line in the list box. Then enter the LAST line number to combine. For example, to combine the space between lines 3 and 8 into one space, click on line 3 in the list box, enter 8 into the LAST grid line and click on COMBINE. This will delete lines 4, 5, 6 and 7 and renumber line 8 to line 4. All lines above the new line 4 will be renumbered. COMBINE is the opposite of DIVIDE.

Clicking on SPACE will display the space between the FIRST grid line and the LAST grid line in the SPACE IS edit box. This is useful in laying out lines in some cases.

Y Grid

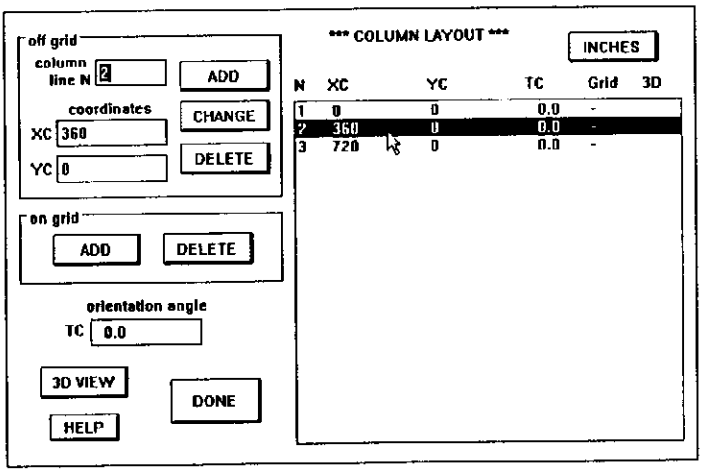
The Y axis runs vertically from bottom to top on the screen. The Y grid lines run horizontally and cross the Y axis at the Y COOR specified for each line. All operations on the Y grid are exactly the same as for X grid which is described earlier.

Note that NO grid will show on the screen unless there are at least two X and two Y grid lines defined.

Columns

Column layout allows you to define the locations where structural columns can later be assigned and their orientation angle. The column layout you define here does NOT contain any structural members. In order to differentiate column layout lines from structural columns, the term "column lines" will always mean column layout lines. They appear as points in plan, but will appear as vertical lines when stories are defined. Structural columns can ONLY be assigned where there is a column line.

When you click on **Columns**, this dialog box appears:



There are two ways to define column lines, ON GRID and

OFF GRID. The ON GRID method is vastly easier, but requires that you have previously defined a X and Y grid which has an intersection at the locations where the column lines are to be placed. Any or all column lines can be placed OFF GRID by simply defining the coordinates and no grid lines are necessary. Probably the best solution is to define a grid for all regularly placed column lines and then place any irregular column lines using OFF GRID coordinates.

When defining column lines, each line has an ANGLE specifying the orientation of the column major direction relative to the X axis (for example, the direction of an I-section web). The ANGLE is measured counter-clockwise from the X axis. Each time you ADD a column line it will have the specified ANGLE, whether placed ON or OFF GRID.

To ADD an OFF GRID column, enter the XC and YC coordinates and the N, and click on ADD. Note that you cannot have two column lines in the same location and that N numbers can not be skipped. Clicking on a defined column line in the list box will bring its values into the edit boxes. You can then change the values and click on CHANGE, or click on DELETE to delete the column line. CHANGE and DELETE work on both ON and OFF GRID column lines.

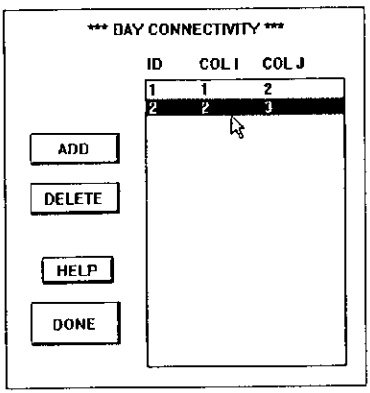
To ADD an ON GRID column line, click on ADD in the ON GRID box. The previously defined grid will appear and you can then click on any grid intersection to place a column line. You can place as many column lines as you wish, but they will all have the same ANGLE unless this parameter is changed. Column lines will be numbered automatically in the order in which they were defined. Clicking on DELETE in the ON GRID box shows the grid and clicking on any existing ON GRID column line will delete it. OFF GRID column lines can NOT be deleted this way.

The 3D VIEW button is used to select or de-select column lines to be included in the simplified 3D view shown in the 3D VIEW SELECT item in the DISPLAY menu. Click on an existing column line in the list box and then click on 3D VIEW to select or de-select the column line.

Bays

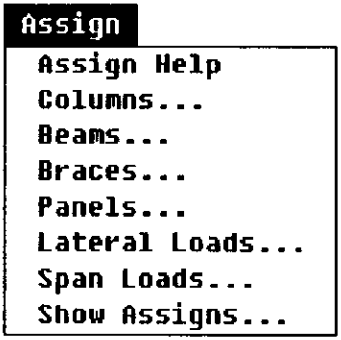
Bays are layout lines like column lines, except they are horizontal and are used to assign structural beams. Beams can ONLY be assigned where there is a bay. Although bays are defined on a plan view, they appear on every defined story except the BASELINE.

When you click on **Bays**, the following dialog box appears:



To add new bays, click on ADD and the existing column lines and any existing bays will appear. Point to a column line, press and hold the left mouse button, move to another column line and release the button. Bays are numbered automatically in the order in which they were defined. You can not have two bays in the same location.

To delete a bay, click on DELETE and repeat the same procedure as for ADD to an existing bay, which will delete it.



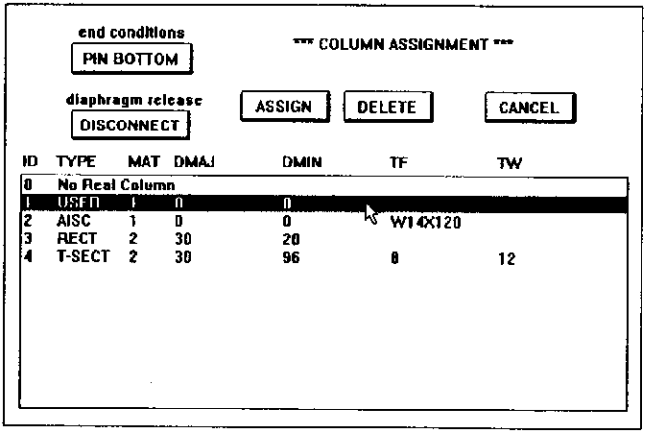
The **Assign** menu allows you to assign structural member properties to the current frame. By the time you start assign, the column lines and the bays should have been defined using the Layout menu, the stories defined in the Building menu and structural element properties defined in the ElemTable menu. It is MUCH better to have all these items completed before assignment, than to change the layout or stories after assignment is started. Changing stories or layout after assignment may cause existing assignments to be deleted and you would have to redo them.

Assign Help

Clicking on **Assign Help** brings up a dialog box containing abbreviated information about using the **Assign** menu items.

Columns

Clicking on **Columns** brings up this dialog box:



The list box shows the column properties that you have previously defined in the ElemTable menu. Click on the one you wish to assign or delete. Also, if assigning, select END CONDITIONS and DIAPHRAGM RELEASE by clicking on the buttons. Dummy columns with disconnected column lines are assigned using property zero.

Now click on ADD or DELETE and the 3D view of the frame appears. If the 3D view is not what you expect, go to the Display menu items 3D View Select and Story Select and make sure these are set correctly. A common problem is forgetting to define stories or not setting the proper top and bottom stories in Story Select.

In the 3D view, the column lines and bays are shown as thin WHITE lines. To assign a column, point to an intersection of a column line and story, press and hold the left button, move to another intersection on the SAME column line and release the button. You can assign columns between one or more stories as desired. The assigned column is shown as a MAGENTA (purple) line. Any previously assigned columns with the same column element type will also show as magenta. Previously assigned columns with a different element type will be shown as RED.

To DELETE columns, follow the same procedure as for ASSIGN, but over an existing column of the same element type (MAGENTA). You can NOT delete an assignment of other element types, without going to the Column assignment menu and selecting the correct element type in the list box.

When assigning or deleting, you can use Zoom from the Display menu to select a smaller portion of the 3D view and then use the scroll bars to pan this smaller view over the total 3D view. If your zoomed view is too small, select 3D Frame from the Display menu to return to the full 3D view.

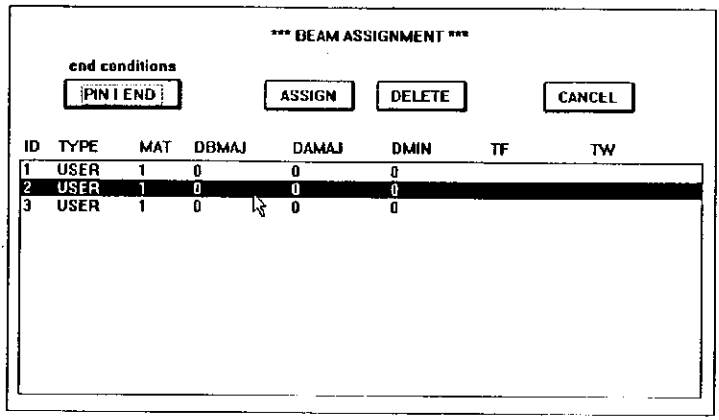
The judicious use of Zoom and Story Select (Display menu) will make assignment easier.

You can also assign columns in PLAN view, that is, the Elevation Angle in 3D View Select is set to EXACTLY 90 degrees. In this case, clicking on a column line will ADD or DELETE column assignments on ALL stories selected in Story Select. Column colors are the same as normal 3D view.

Beams

Read over the column assignment preceding this, as most of it applies to assigning beams. Only the differences will be shown here.

Clicking on **Beams** brings up this dialog box:



Select the beam property by clicking on it in the list box and select the END CONDITIONS, then click on ADD or DELETE. The assignment procedure is exactly the same as for columns except that beams are assigned to bays. This means that the start point is on one column line while the end point is on another and the two column lines must be connected by a bay. You can assign beams to several consecutive stories by having the start and end points on

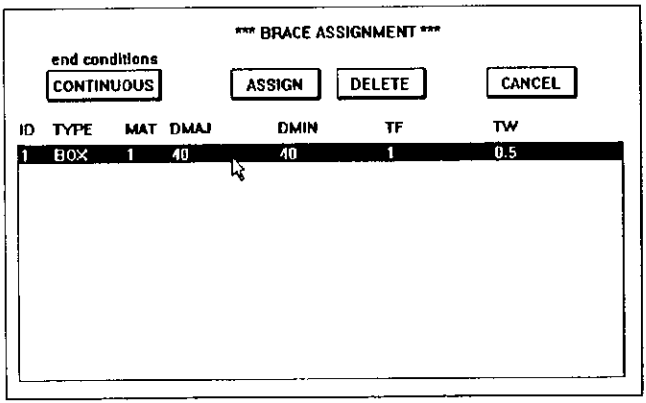
different stories. Beams with the present element type are MAGENTA while other element types are YELLOW.

Deleting is the same as assigning; remember that you can only delete the present property (MAGENTA).

You can also ADD or DELETE Beams in PLAN view (see Column assign) which affects ALL stories selected in Story Select.

Braces

Clicking on **Braces** shows this dialog box:



Braces are assigned and deleted in much the same way as Columns, so read the Column assign section earlier as only differences will be given here.

Select the desired brace property from the list box, the desired END CONDITION, and then click on ADD or DELETE. Braces are assigned between column lines, but do not have any relationship to bays and thus can be assigned between any two column lines. However, Braces must be assigned between two consecutive story levels.

To assign a Brace, start at the intersection of a column line and story and end on the intersection of a column line and story on one story higher or lower. Braces of the present

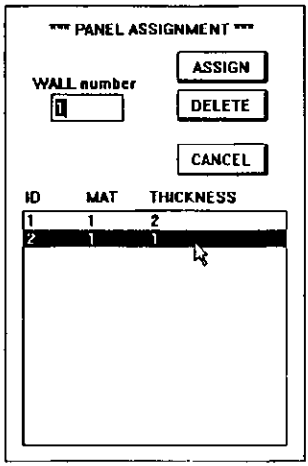
element type are MAGENTA while Braces of other element types are shown in CYAN.

Only Braces of the presently selected property (MAGENTA) can be deleted. Use the same procedure as assignment.

You can NOT ADD or DELETE braces in PLAN view.

Panels

Clicking on **Panels** brings up this dialog box:



Panels are assigned and deleted in much the same way as Columns, so refer to the Column assign description earlier, as only differences will be given here.

Besides selecting the panel property number to be assigned, the number of the wall to which the panel is assigned must also be entered. Refer to the ETABS Users Manual for the special significance of the wall number.

Panels are assigned between column lines, but do not have any relationship to bays, so they can be assigned between any two column lines. A panel is one story high, but panels can be assigned over several consecutive stories at once, if

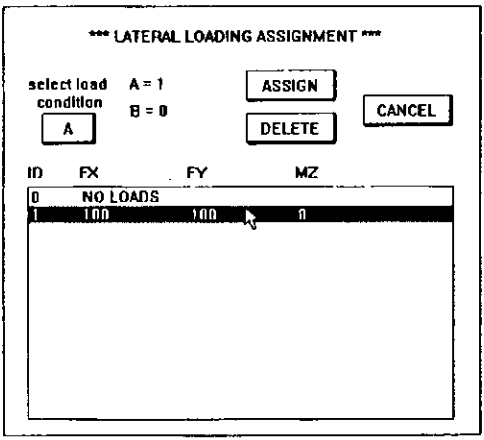
the wall number is the same. To assign panels, start at the intersection of a column line and story and end at the intersection of another column line and story. The start and end points must be at least one story apart , but can be more than one story apart for multiple panels. Panels with the present property are shown as MAGENTA, while those with other properties are shown as BLUE.

Delete panels in the same way as assignment, including multiple panels, keeping in mind that only the present property (MAGENTA) can be deleted.

You can ADD or DELETE panels in PLAN view, which affects ALL stories selected in Story Select.

Lateral Loads

Clicking on **Lateral Loads** brings up this dialog box:



Lateral Loads are assigned and deleted in a similar fashion to Columns, so refer to that section, as only differences will be given here.

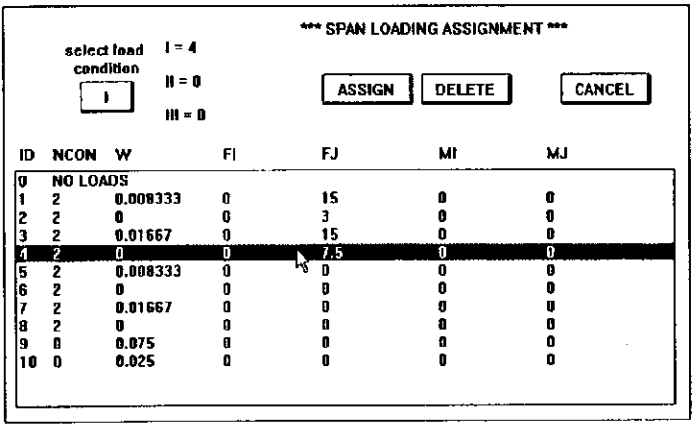
Click on SELECT LOAD CONDITION and then click on a load number in the list box. Do this for both A and B load conditions. When A and B conditions are set, click on

ASSIGN or DELETE. Lateral Loads are assigned to a single column line. Generally, start at the intersection of a column line and story and end at the intersection of the SAME column line and any story. The start and end points are the same for a single load or different for several identical loads at consecutive story levels. The loads are shown as MAGENTA squares at the intersections, when they are the present A condition load number, and in WHITE for other A condition load numbers. The B condition does not affect the color. Deleting is the same as assignment, remembering that only the present A type (MAGENTA) can be deleted.

You can ADD or DELETE Lateral loads in PLAN view, which affects ALL stories selected in Story Select.

Span Loads

Assignment and deletion of Span Loads is exactly the same as for Beam Assign, so please refer to that section for the basic procedures. Clicking on **Span Loads** brings up this dialog box:



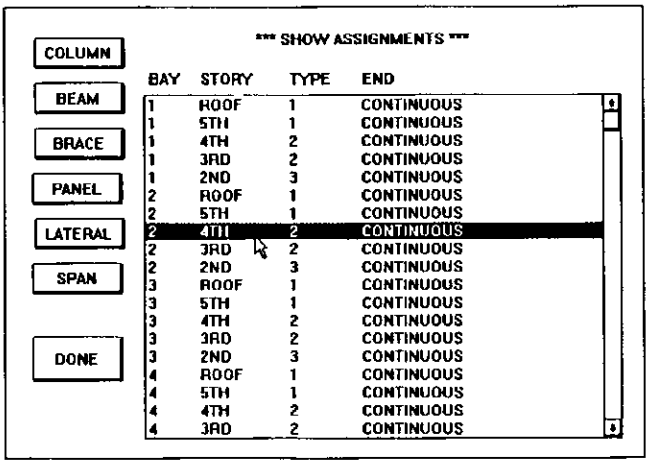
Span Loads can ONLY be assigned where there is already a Beam property assigned. The only difference between Span Loads and Beams is in the selection of the load condition number and the determination of the colors shown on screen.

Click on SELECT LOAD CONDITION to select the condition, I, II or III. Then click on the desired load number in the list box. Do this for all three load conditions. When Span Loads are shown on screen, they change the color of the existing beam. If the load is the present I condition, then the load is shown as MAGENTA. Other loads with different conditions are shown as RED. The II and III conditions have no effect on the screen colors.

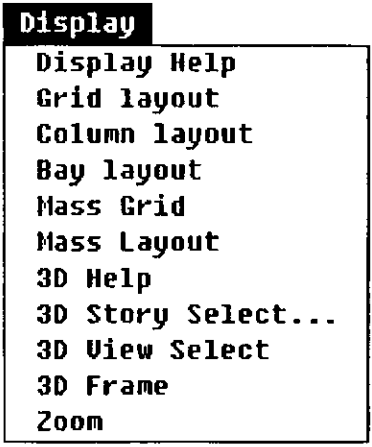
You can ADD or DELETE span loads in PLAN view, which affects ALL stories selected in Story Select.

Show Assigns

Show Assigns allows you to see all the present element assignments in a tabular form. Clicking on **Show Assigns** brings up this dialog box:



Click on the desired type, COLUMN, BEAM, etc. to bring up the list of assignments for that type. Use the list box scroll bar to see assignments in long lists.



The **Display** menu controls what is shown on the screen and how it is shown. The 3D view direction can be set as well as the number of stories shown. All displayed items are for the current frame, which is selected in the **Frame** command of the **Building** menu.

Display Help

Brings up a dialog box which contains abbreviated information about the display menu use.

Grid Layout

Shows the present complete grid layout.

Column Layout

Shows the complete column line layout in plan view including the grid but not the bays.

Bay Layout

Shows the complete layout in plan view including grid, column lines and bays.

Mass Grid

Shows the complete Mass grid.

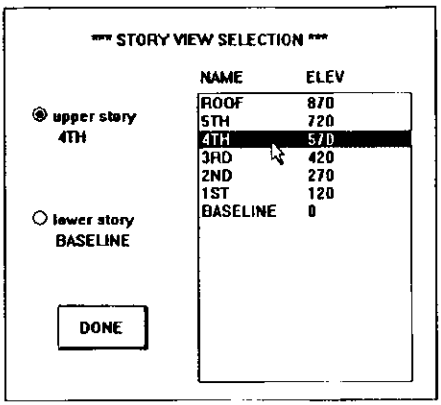
Mass Layout

Shows the complete mass grid and all assigned masses for the presently selected segment. The segment is selected in the **Mass** command in the **Building** menu.

3D Story Select

3D Story Select allows you to show any number of stories in the 3D view. This is particularly helpful in assignment of elements when the frame is complex.

Clicking on **3D Story Select** shows this dialog box:

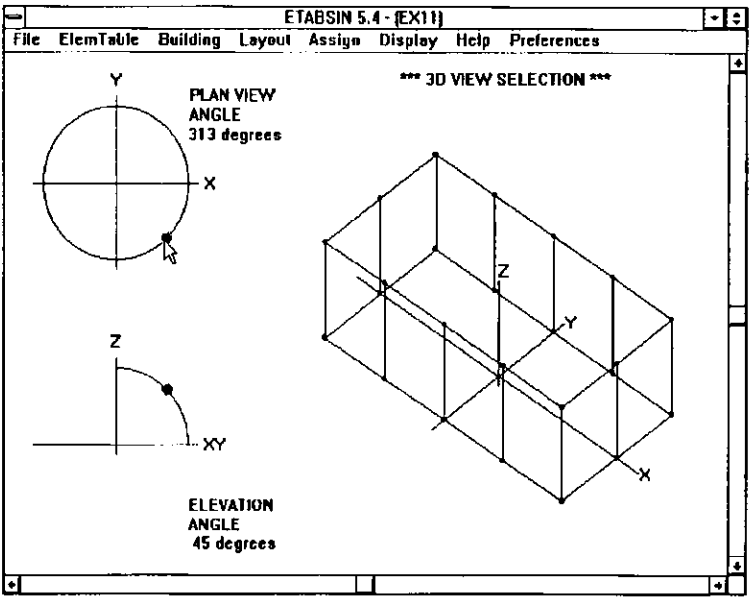


The list box shows the list of stories that have been defined. Click on UPPER STORY and then click on the desired upper story in the list box. Now click on the LOWER STORY and then click on the desired story in the list box.

The 3D view will show everything BETWEEN the UPPER STORY and the LOWER STORY. For Example, to show the entire frame, set UPPER STORY to the top story (ROOF in the example shown) and set the LOWER STORY to BASELINE.

3D View Select

3D View Select allows you to set the angle from which the frame is viewed when the 3D view is shown. Click on **3D View Select** to show the following screen:



To change the PLAN VIEW ANGLE, point to the dot on the plan view circle, press and hold the left button and then move the dot around the circle. While doing so, the angle, shown in degrees, should change. Once the dot has been captured and the mouse button is still held down, you can move the cursor away from the circle which will make setting an exact angle easier. When you have the desired angle, release the button.

Changing the ELEVATION ANGLE is done in the same way as for plan view, but elevation angle is restricted to between 0 and 90 degrees.

If you have defined column lines and bays for the current frame, then a simplified view of the frame will be shown

on the right side of the screen. This REQUIRES that the defined column lines have 3D VIEW set, which is done in the Column Layout menu. Bring up the Column Layout dialog box and check the column lines in the list box. Use the 3D VIEW button to select or de-select the column lines. It is better to select only the minimum number of column lines necessary to define the frame since the simplified view will draw faster.

3D Frame

Shows the complete frame in 3D with only the stories that have been set in 3D Story Select showing. Also cancels any properties that were last set in assignment, so that NO MAGENTA elements are shown.

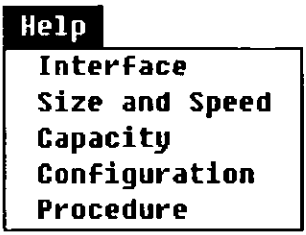
Zoom

Zoom allows you to show an enlargement of a portion of the complete layout, mass layout or 3D frame. Zoom can be used at any time that the grid, column lines, bays, mass grid, mass elements, or any 3D view is on screen. When any of these items are on screen, click on Zoom. Then point to the upper left corner of the desired area to zoom to, press and hold the left button and move the mouse which will show a box. Make the box enclose the area you want and release the button. The area enclosed by the box will then fill the screen. If desired, you can zoom again using the same procedure.

When the desired screen is shown, then return to what you were doing, for example, assigning. The same view will stay during assignment. If you want to return to the complete view, click on one of the complete view items in the display menu, for example, 3D Frame.

While zoomed, you can use the scroll bars at right and bottom of the screen to pan across the complete view while maintaining the size of the zoomed view. Click on the

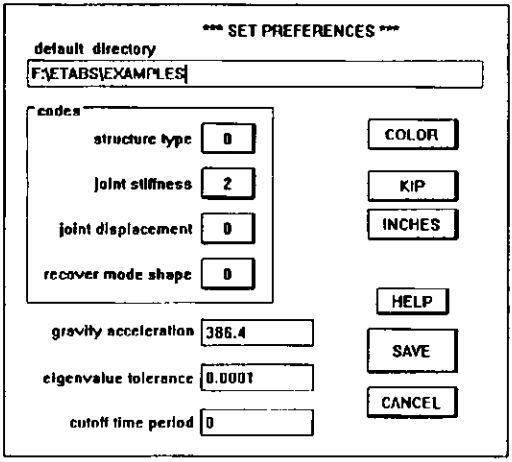
arrows at the end of the scroll bars to make small panning movements. Point to the box in the scroll bar, press and hold the left button, move the box to another position and release the button. Note that using the scroll bars moves the screen in steps. One click on the arrows at the end of the scroll bars moves the view an amount equal to the size of the zoomed view. Therefore, the movement depends on the size of the zoomed view.



The **Help** menu includes several abbreviated help messages which apply to the entire ETABSIN program. Some of this information is specific to the version of ETABSIN that you are using and is not included in this manual. Click on the menu items to bring up the help dialog boxes.

Preferences **Preferences** allow you to set certain items, such as units, that are automatically set each time ETABSIN is started. These items are saved in a file called ETABSIN.INI which, if it exists, is read at startup. If it does not exist, then default settings are used.

Clicking on **Preferences** brings up this dialog box:

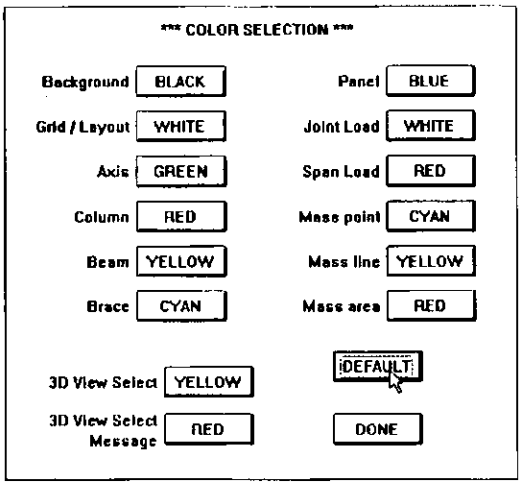


You can enter a default directory, which is the directory that will be displayed when you do an **Open** or **Save as** in the **File** menu. To set the default directory, position the cursor in the edit box and enter the desired directory name, which SHOULD include the drive. If you do not include the drive, then the directory MUST be on the same drive as ETABSIN.EXE. The directory name will be checked to see if it exists when you click on the SAVE button.

Set the CODES by clicking on the buttons. See the ETABS Users Manual "Control Information" for a description of these codes. Enter the edit box item (GRAVITY, etc.) as desired. Click on the units buttons to set the units. Finally, click on SAVE to save the settings in the file ETABSIN.INI.

Please note that the units can be changed in many other dialog boxes within ETABSIN. The settings here only control what is set when the program is started or **New** in the **File** menu is done.

Click on the COLOR button to bring up this dialog box:



This dialog box allows you to set the colors used for the ETABSIN screen when displaying grids, joints and structural elements. Click on the button next to the items that you want to change until the desired color is shown. There is a choice of 7 colors for each item. Be SURE that you do not set an item to be the same color as the Background unless you want the item to be invisible! Note that MAGENTA is always used for presently selected structural elements and is not available otherwise.

Clicking on the DEFAULT button will change the colors to a set of colors defined in ETABSIN. This is convenient if you set colors that are not usable and want to start over again. Also, the colors of the different elements described in the **Assign** menu section are all default colors.

The colors that are set will take effect when you click on DONE and remain in effect until **New** is done.

V.

BIBLIOGRAPHY

1. **Habibullah, A.**

   "ETABS - Three Dimensional Analysis of Building Systems", Computers and Structures, Inc., Berkeley, California, 1992.

2. **Microsoft Corporation**

   "Microsoft Windows Users Guide Version 3.1", Microsoft Corporation, 1992.

Index

C

D

E

F

G

H

I

K

L

M

W

X

Y

Z