ETABSOUT

A Display Post Processor for ETABS®

COPYRIGHT

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

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

<div align="center">

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 U.S.A.
Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com

</div>

DISCLAIMER

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

A Quick Tour of ETABSOUT

Overview

ETABSOUT is a Windows based interactive display post processor for the ETABS program. It is part of the ETABS package and is automatically installed when the ETABS program is installed.

This short manual is organized as a quick tutorial aimed at giving the first time user hands-on experience while describing some of the basic features and capabilities of ETABSOUT. It is assumed that you are familiar with the ETABS concept and terminology. It is recommended that you use the comprehensive on-line Help included in the program. If required, refer to the ETABS User's Manual and other documentation provided in your package.

It is possible that the screens shown in this tutorial may appear slightly different from what you see on your computer screen. This may be due to different screen resolution and/or font settings on your computer.

It is important to recognize that the order in which the menus are used in this tutorial is immaterial. After some practice, you may choose to perform the operations shown in a different order to arrive at the same display.

The ETABSOUT program has options for plotting and printing two-dimensional and three dimensional views of:

- Undeformed structural geometry
- Frame loading diagrams
- Displacements
- Mode shapes
- Member force diagrams
- STEELER stress ratios

In addition, the program has the capability to display and print:

- Response spectrum curves from a time history analysis
- Time history curves of various response quantities
- Load - displacement curves for the non-linear elements
- Output results in tabular form

Exploring the ETABSOUT Screen

The ETABSOUT screen consists of the following key components:
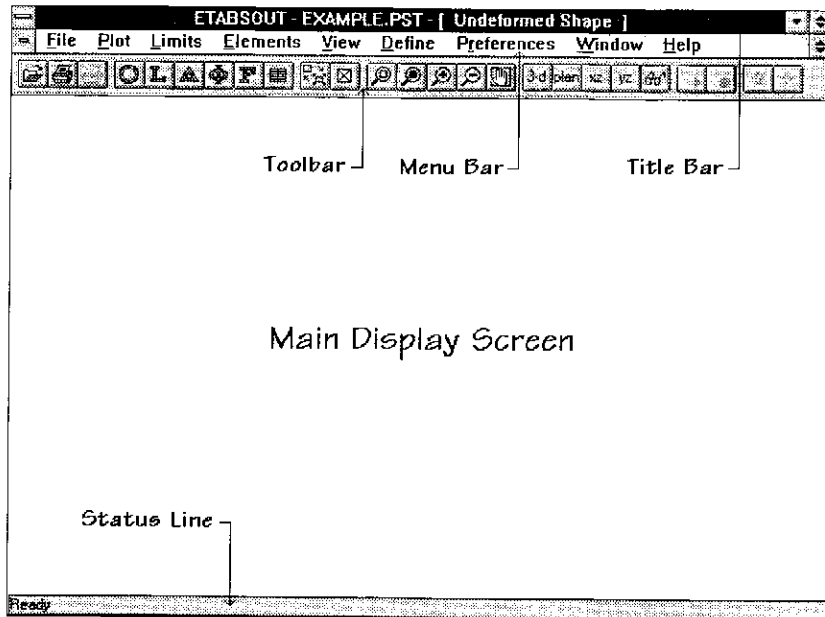
Title Bar

The title bar contains the name of the current ETABS postprocessing file (*.PST) and information about the graphics shown on the screen.

Menu Bar

The Menu Bar contains the names of the ETABSOUT menu options.

Toolbar

The Toolbar provides quick access to commonly used features. Most of the features available on the Toolbar can also be accessed from the Menu Bar. You can move the Toolbar, re-arrange and re-position it on the screen. The buttons on the Toolbar are displayed below:

	Open .PST File
	Print Graphics
	Print Tabulated Output
	Display Undeformed Shape
	Display Loads
	Display Deformed Shape
	Display Mode Shape
	Display Member Force Diagram
	Display Tabulated Output
	Shrink/Unshrink Elements
	Check/Uncheck Elements
	Zoom w/Rubberband
	Zoom to Full View
	Zoom In
	Zoom Out
	Pan

[3-d]	**3d View**
[plan]	**Plan View**
[xz]	**XZ Projection**
[yz]	**YZ Projection**
[60°]	**Perspective Toggle**
[▶]	**Start Animation**
[■]	**Stop Animation**
[⇧]	**Increase Load #/Mode #**
[⇩]	**Decrease Load #/Mode #**

Horizontal Scroll Bar

The horizontal scroll bar is used to control the animation speed of the deformed model. It appears along the bottom edge of the Main Display Screen, above the Status Line, once the animation process is activated.

Status Line

The status line along the bottom of the ETABSOUT window displays information associated with options currently activated.

Description of the Model

The model used in this tutorial is for illustrative purposes only. It intentionally uses several different types of elements and multiple diaphragms.

The model consists of a four story and a two story structure with a combined base as shown in Figure 1. The base has concrete shear walls and steel braced frames. The upper part of the structure has steel moment frames.
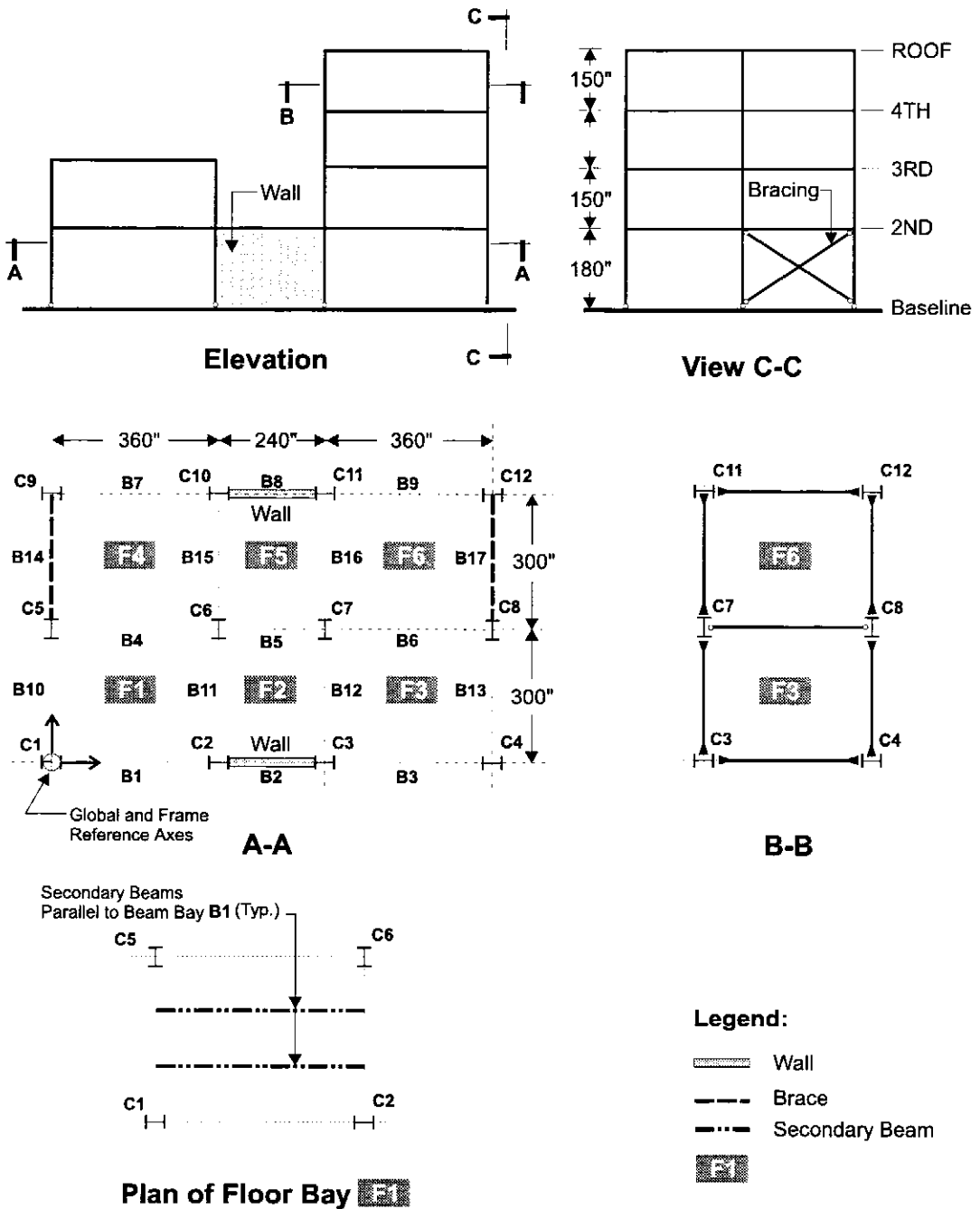
"Floor" elements are used to model gravity loads.

For more detailed description of the model refer to the ETABS Examples Manual.

Five load conditions are considered in the analysis. Load Condition I (Vertical I) is the gravity dead load which includes member self weight. The weight of the floor/roof is applied as part of the floor bay loads. Load Condition II (Vertical II) is the gravity live load which is also applied as floor bay loads. Load Conditions A and B (Lateral A and Lateral B) are user-defined lateral static wind loads applied in the global X- and Y- directions. Load Condition D1 (Dynamic 1) is the linear time history analysis using The Loma Prieta records. For simplicity, the requested five load cases correspond to the five specified load conditions.

The input data file for this model is **EXAMPLE**.

The **EXAMPLE** file and the ETABS postprocessing file **EXAMPLE.PST** are part of the ETABS package and are located in the EXAMPLES subdirectory under the directory and the drive where the program has been installed.

Elevation

View C-C

ROOF
150"
4TH
3RD
150"
Bracing
2ND
180"
Baseline

A-A

B-B

Global and Frame
Reference Axes

Secondary Beams
Parallel to Beam Bay **B1** (Typ.)

Plan of Floor Bay

Legend:
Wall
Brace
Secondary Beam

**Figure 1**

Description of the Model Used in This Tutorial

Starting the Tour

A successful analysis of the "EXAMPLE" model using the ETABS program produces a postprocessing file called EXAMPLE.PST. The postprocessing file contains information pertaining to the building geometry, loading and the analytical results. This file is located in the EXAMPLES subdirectory under the directory and the drive where the program has been installed.

Step 1. Opening ETABS Postprocessing File

1. Double-click on the **ETABSOUT** icon.

2. Click the **Open .PST File,** ⟁ button on the Toolbar. This will open the ETABS Postprocessing File dialog box.

3. In the Open ETABS Postprocessing File dialog box:

    ✓ Select the EXAMPLE.PST file.
    ✓ Click the OK button.

The screen will now display a three-dimensional view of the model in the undeformed state.
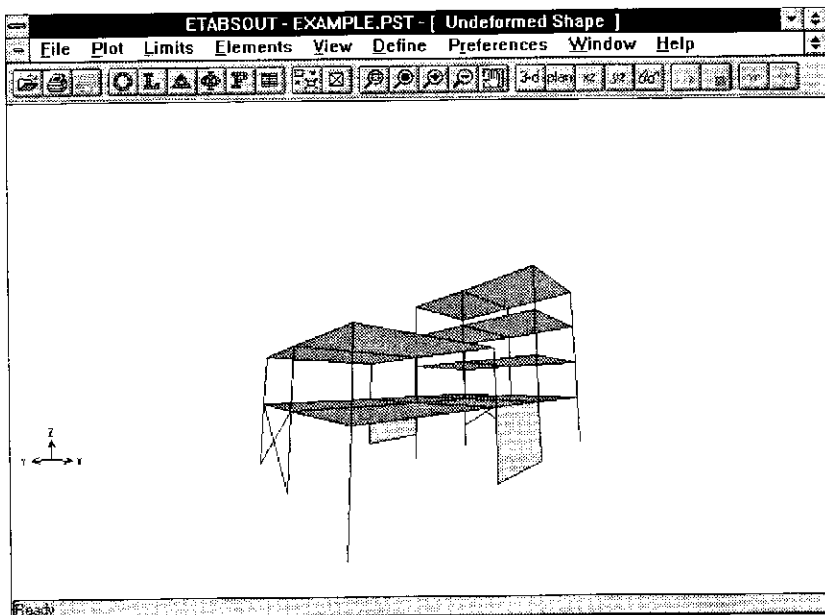
Step 2. Plotting Displacements

Next, we will display the deformed shape of the structure. We will work with the Toolbar to demonstrate the ease of use.

1. Click the **Display Deformed Shape,** ▲ button on the Toolbar. This will open the Deformed Shape dialog box displaying the default settings.

2. In the Deformed Shape dialog box:

   ✓ Accept the default settings by clicking the OK button.

The screen will refresh and display the deformed shape of the structure under Load Condition A, the default load condition.
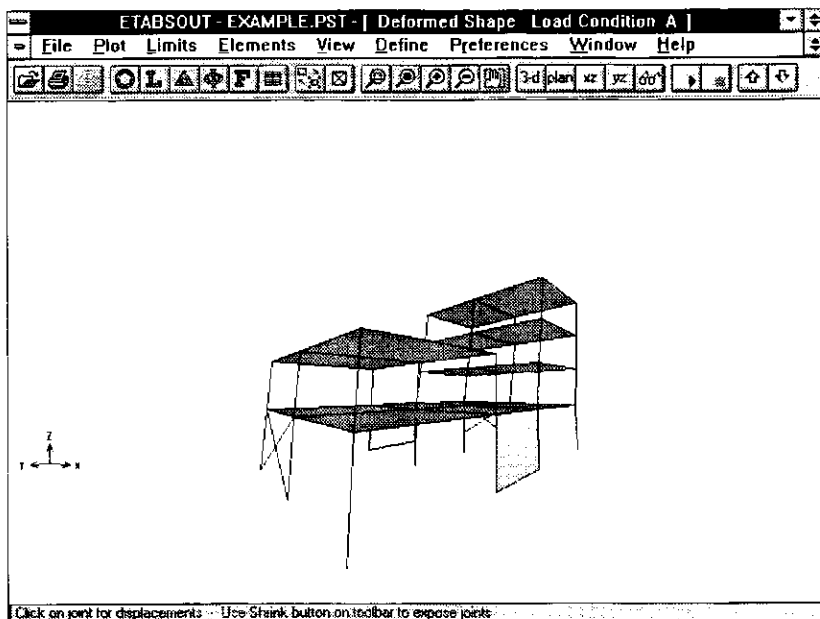
Now we will examine the displacement values of a particular joint for this load condition.

1. Click the **Shrink/Unshrink,** button in the Toolbar. This will shrink the elements and expose the joints.

2. Click on any desired joint.

This will open a floating window in which the values of translations and rotations of the selected joint are displayed. Also, the selected joint will flash on the screen. We will re-position this window so that the model is fully exposed.
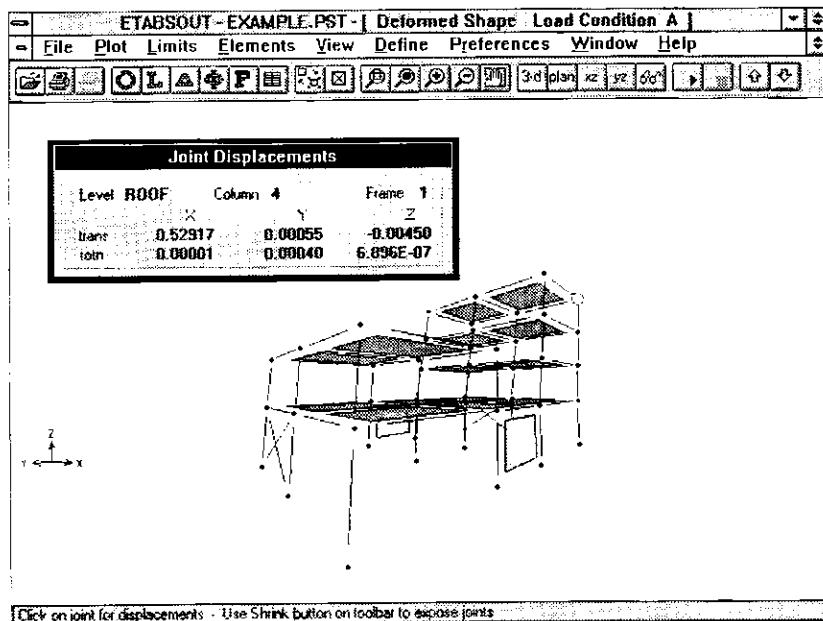
Click on any other joint to see how the information in the floating window is updated.

We will now change the view direction of the display to a plan view and then animate the deformed shape.

1. Click the **Plan View,** plan button on the Toolbar.

The screen will refresh and display the deformed shape of the structure in plan view.

2. Click the **Start Animation,** button on the Toolbar.

The screen will refresh and display the animated deformed shape of the structure.

To change the animation speed use the horizontal scroll bar at the bottom of the screen. Select the left arrow to slow down the animation and the right arrow to increase the animation speed.

Step 3. Plotting Mode Shapes

There are fifteen mode shapes requested in the analysis. We can display and animate any of these mode shapes. More than one window can be opened. Each mode shape is displayed with the corresponding period and the effective mass participation factors.

Displaying Mode Shape 3

1. Click the **Display Mode Shape,** button in the Toolbar. This will open the Mode Shape dialog box with default settings.

2. In the Mode Shape dialog box:

    ✓ Change the mode number to 3 using the spin button up arrow.
    ✓ Check the Cubic Curve box in the Options area.
    ✓ Click the OK button.

The screen will refresh and display the deformed shape in mode number 3.
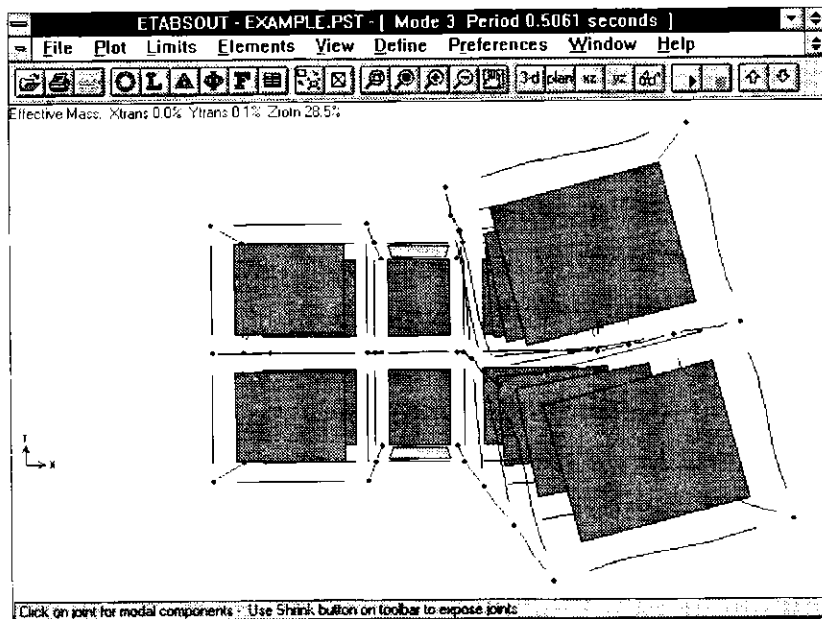
We can now animate the mode shape.

1. Click the **Start Animation,** ▶ button on the Toolbar.

The screen will refresh and display the animated deformed shape of the structure.

2. Click the **Stop Animation,** ■ button to stop the animation.

Using More Than One Window

As an example, we will open a second window to display mode shape number 7. We will also switch off the element shrinkage option in the original window.

1. Click the **Shrink/Unshrink Elements,** button on the Toolbar.
2. Choose '**New' from the Window** menu.

The new window opens with a display of the undeformed model. The viewing direction settings are the same as the display in the original window.

3. Click the **Display Mode Shape,** button on the Toolbar. This will open the Mode Shape dialog box.
4. In the Mode Shape dialog box:

    ✓ Change the mode shape number to 7 using the spin button up arrow.
    ✓ Click the OK button.

This will refresh the new window and display the deformed structure in mode 7.

The display is shown on the following page.
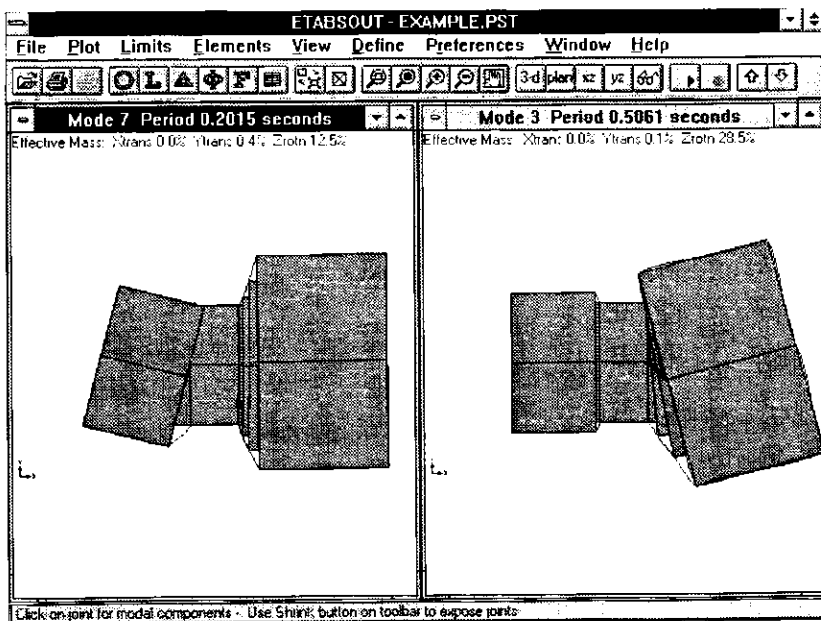
As an exercise you may want to try animating the mode shapes in each window by clicking in the window to activate it and then clicking the **Start Animation,** [▶] button on the Toolbar.

Step 4. Plotting Member Forces

We will now plot the major moment diagram for Load Case number 1. Identical procedures are used to plot other components for various load conditions or load cases.

1. Close the window showing mode shape 7, by double-clicking the Control box of the window.

2. Click the **Maximize** button in remaining window.

3. Click the **3d View,** [3-d] button on the Toolbar to change to three-dimensional view.

4. Click the **Display Member Force Diagram,** [F] button on the Toolbar. This will open the Member Force Diagram dialog box.

5. In the Member Force Diagram dialog box:

    ✓ Click the **Load Case** button in the Load area to select Case 1.
    ✓ Accept other default settings.
    ✓ Click the OK button.
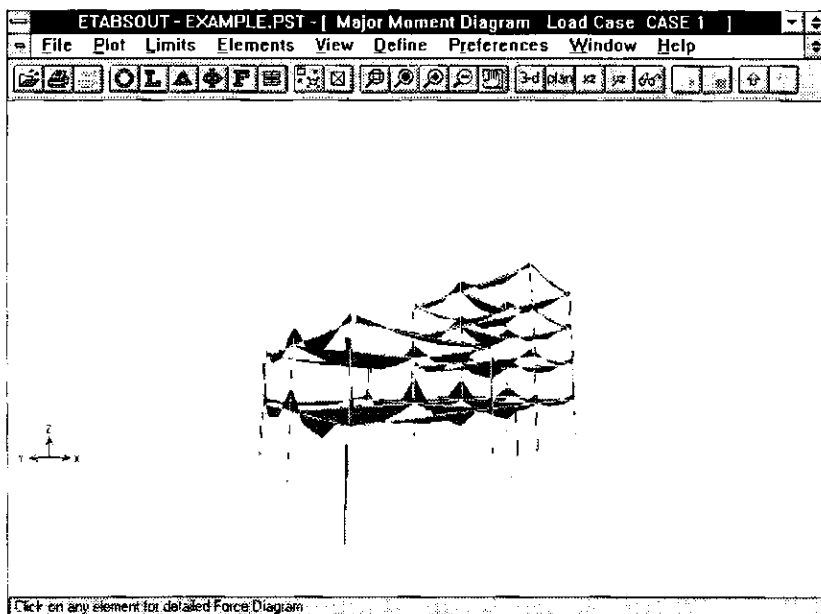
We can now view the major moment diagram on a member by member basis. We will shrink the elements and zoom in on part of the model.

1. Click the **Shrink/Unshrink,** button on the Toolbar.

2. Click the **Zoom w/Rubberband,** button on the Toolbar.

3. Point to a corner of the region to be zoomed, hold down the mouse button and drag to define the rectangular zoom region and release the mouse button.

4. Click on the desired member.
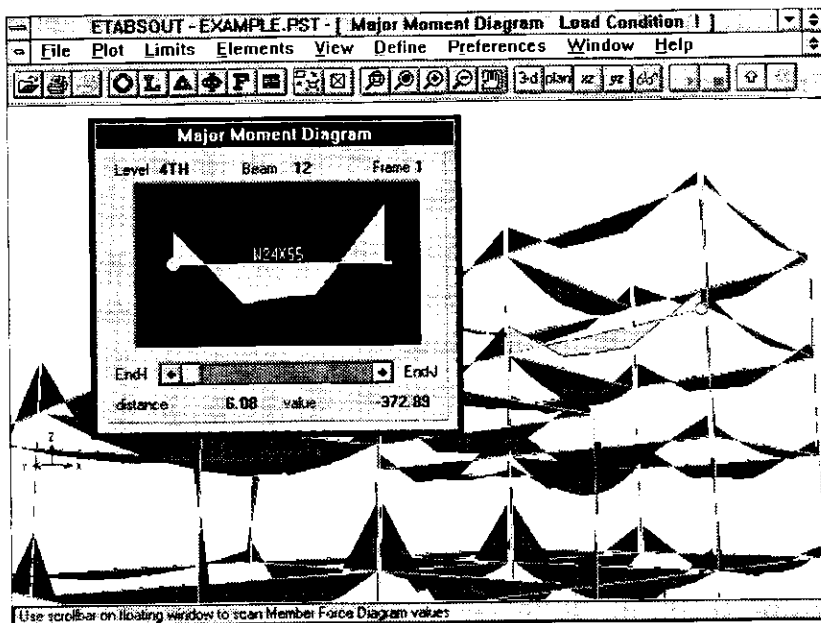
This will open a floating window showing the variation of the major moment over the member length. If more than one element exists within the tolerance of the cursor, a list box of elements is displayed. You can then click on the desired element. The force diagram of the selected member will be highlighted.

Click on any other member to see how the information in the top window is updated.

Use the scroll bar in the top window to display spot values along the length of the member.

Step 5. Displaying Results in Tabulated Form

We will now display member forces, displacements and reactions for different load conditions and load cases in a tabulated form.

1. Click the **Zoom to Full View,** button on the Toolbar to display the complete model.

2. Click the **Display Tabulated Output,** button in the Toolbar. This will open the Loads dialog box.

3. In the Loads dialog box:

    ✓ Check the Vertical I and Vertical II boxes. Notice that these options are nonexclusive; you can choose all the load conditions and the load cases that are requested in the analysis.
    ✓ Uncheck the Load Cases box.
    ✓ Click the OK button.

The screen is updated and the model is displayed with element shrinkage.

Displaying Member Forces

As an example, let us examine the results for a column element.

1. Click once on the desired column.This will open a floating window in which the output of the column forces for all the specified load conditions is displayed in a tabular form. The selected column will be highlighted on the screen. If more than one element exists within the tolerance of the cursor, a list box of elements is displayed. You can then click on the desired element.

    The display is shown on the following page.

Displaying Joint Displacement Values

The joint displacement values are displayed by clicking on any desired joint except the baseline joints.
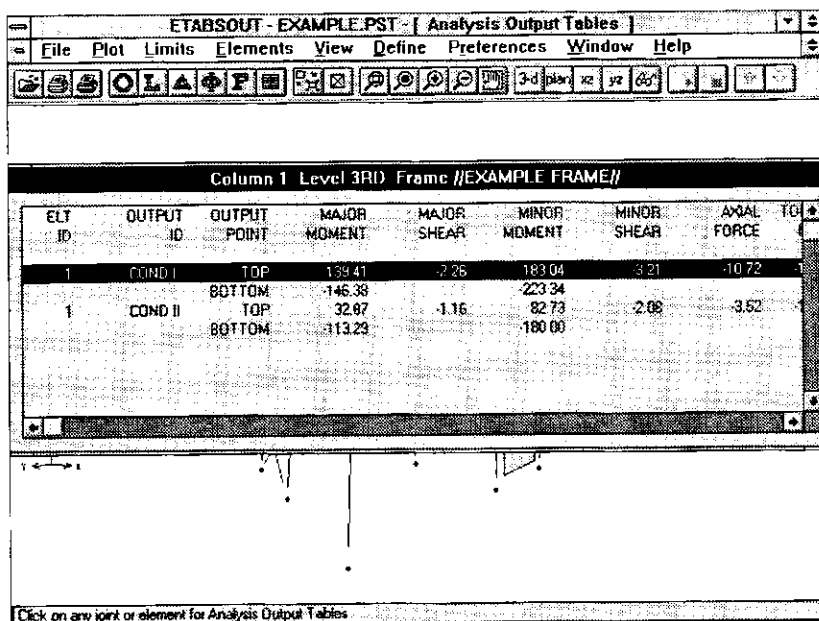
Displaying Reaction Values

The reaction values are displayed by clicking on the baseline joints.

Printing Results

1. Click the **Print Tabulated Output,** 🖨 button in the Toolbar. This will open the Tabulated Output dialog box.

2. In the Tabulated Output dialog box:

   ✓ Accept the default settings by clicking the OK button.

The Print Tabulated Output option will print member force output for all the members displayed for Load Conditions I and II. You can, however, limit the display by activating options on the **Limits** and the **Elements** menus.

Concluding Remarks

This marks the end of the quick tour of ETABSOUT. The intent has been to highlight and demonstrate the basic features and to serve as an introduction to the application. Feel free to experiment and explore other options with different ETABS postprocessing files. Additional information is available within the Help menu.