183 B

Capacity Analysis Program
Version 2.7

Reference Manual

Copyright (c) 1989 - 1994 PMB Engineering, Inc.
Unpublished--rights reserved under the copyright laws of the United States.

Restricted Rights Legend: Use, duplication or disclosure is subject to restrictions stated in Contract No. N47408-90-C-1143 with PMB Engineering, Inc.

Table of Contents

1.0 INTRODUCTION 1-1
2.0 Limitations 2-1
3.0 GUI COMPONENTS 3-1
4.0 STARTING CAP 4-1
5.0 USING CAP 5-1
6.0 FILE MENU 6-1
6.1 Build New Project 6-2
6.2 Import From SeaStar 6-3
6.3 Import From SACS 6-5
6.4 Import From STRUDL 6-7
6.5 Open Project 6-9
6.6 Save As 6-10
6.7 Save 6-11
6.8 Abandon 6-12
6.9 Copy Project 6-13
6.10 Delete Project 6-14
6.11 Delete Model 6-15
6.12 Delete Version 6-16
6.13 Delete Scratch 6-17
6.14 Recover Project 6-18
6.15 Working Directory 6-19
6.16 Recent Modifications 6-20
6.17 About CapView 6-21
6.18 Exit CapView 6-22
7.0 OPTIONS MENU 7-1
7.1 Display Defaults Dialogue Box 7-2
7.2 Display Projection Dialogue Box 7-5
7.3 Pen Attributes Dialogue Box 7-7
7.4 Pen Selection Dialogue Box 7-9
7.5 Inelastic Event Pen Selection Dialogue Box 7-10
7.6 Interaction Ratio Ranges Dialogue Box 7-11
7.7 Print Displayed Structure Dialogue Box 7-12
7.8 Print / View Files Dialogue Box 7-13
7.9 CapHelp Dialogue Box 7-14
8.0 THE MODEL MENU 8-1
8.1 Generate Platform Skeleton... 8-2
8.2 Add Members/Nodes 8-11

8.3 Read Member/Node/Load Files 8-16
8.4 Damping Parameters 8-19
8.5 Fix X-Brace Center Node 8-20
8.6 Soil Profile 8-21
8.7 Water Data 8-25
8.8 Default Analysis Parameters 8-26
8.9 Default Code Check Parameters 8-28
8.10 Node Select/Change 8-29
8.11 Member Select/Change 8-31
8.12 Show Selected Nodes 8-34
8.13 Show Selected Members 8-35
8.14 Node/Member/Load Reports 8-36
8.15 AISC Sections 8-37
8.16 Standard Tube Sections 8-39
9.0 THE LOADS MENU 9-1
9.1 Add Nodal Load 9-2
9.2 Add Nodal Displacement 9-3
9.3 Add Pushover Load 9-4
9.4 Nodal Load Select/Change 9-5
9.5 Nodal Displacement Constraint Select/Change 9-7
9.6 Show Selected Nodal Loads 9-9
9.7 Show Selected Nodal Displavements 9-10
9.8 Wave Load Control 9-11
9.9 Regular Wave Data 9-12
9.10 Irregular Sea Data 9-14
9.11 Current Load 9-18
9.12 Ground Motion Record Dialogue Box 9-19
10.0 THE ANALYSIS MENU 10-1
10.1 Edit Analysis Macros 10-2
10.1.1 Analysis Options 10-3
10.1.2 Header (HEADER) 10-13
10.1.3 End Analysis (NONE) 10-13
10.1.4 Static Analysis (STAT) 10-13
10.1.5 Event-to-Event Dynamic Analysis (DYN1) 10-19
10.1.6 Iterative Dynamic Analysis (DYN2) 10-26
10.1.7 Frequency Analysis (EIGN) 10-31
10.1.8 Save Results for Restart (SAVE) 10-32
10.2 Perform Data Check 10-33
10.3 Run SeaStar Analysis 10-34
10.4 Create SeaStar File 10-35
10.5 Edit Files 10-36
11.0 THE RESULTS MENU 11-1

11.1 Review Input File 11-2
11.2 Review Output File 11-3
11.3 Review Analysis Status File 11-4
11.4 Maximum IR Plot 11-5
11.5 Deflected Shape Plot 11-6
11.6 Animated Deflected Shape Plot 11-7
11.7 Inelastic Events Plot 11-9
11.8 Node Displacement LS Plot 11-10
11.9 Member IR LS Plot 11-11
11.10 Member Force LS Plot 11-12
11.11 Member Force/Deformation LS Plot 11-13
11.12 Beam Column Interaction LS Plot 11-14
11.13 Buckling Beam Inter. Force LS Plot 11-15
11.14 Buckling Beam Inter. Displacement LS Plot 11-16
11.15 Pile Capacity 11-17
11.16 Cut Plane Force LS Plot 11-18
11.17 Force Profile LS Plot 11-20
11.18 Pushover Load LS Plot 11-21
11.19 Pushover Load Ratio LS Plot 11-22
11.20 Unbalance Force LS Plot 11-23
11.21 Mode Shape Plot Dialogue Box 11-24
11.22 Structure Frequencies Dialogue Box 11-25
11.23 Mass Participation Factors Dialogue Box 11-26
11.24 Modal Forces Dialogue Box 11-27
11.25 Envelope Interaction Ratio Plot Dialogue Box 11-28
11.26 Member Envelope Interaction Ratio Report 11-29
11.27 Nodal Response Spectra Plot 11-30
12.0 HOW TO...? 12-1
APPENDIX A: HARDWARE AND SOFTWARE REQUIREMENTS A-1
APPENDIX B: SOFTWARE INSTALLATION B-1
APPENDIX C: CAP FILES C-1
APPENDIX D: INTRODUCTION TO UNIX D-1
APPENDIX E: INTRODUCTION TO X WINDOW SYSTEM E-1

1.0 INTRODUCTION

1.1 What Is CAP

This manual describes how to use CapView Version 2.0, the graphical shell of the Capacity Analysis Program (CAP). CAP is a computer-aided engineering (CAE) tool that incorporates a graphical user interface (GUI), an interactive model builder and generator, an object database system, a nonlinear dynamic structural analysis program, an interactive graphical post-processor with animation capabilities and a set of program-to-program data translation facilities. CAP is an excellent tool for performing almost any kind of analysis on ocean structures, including nonlinear static and dynamic analysis, storm wave analysis, seismic analysis and capacity (pushover) analysis.

1.2 Operating Environment

CapView runs on SPARC compatible workstations (e.g. SUN Microsystems Inc.'s SPARCstations, SPARCservers, sun-4's, and other SPARC compatatible workstations) running the SunOS 4.1 (or later) operating system (a UNIX† operating system) or a compatible operating system (e.g. Solaris 1.0, and SPARC OS 1.1) and the X Window System, version 11 release 4 (X11R4).

1.3 About This Manual

This manual is intended to be a reference manual for CapView. Throughout this manual, the following is assumed to be true:

1- CAP executable and data files are installed properly. (Except in Appendix B, Software Installation).

2- The user has read, write and execute permissions to CAP project directories.

3- The user is familiar with the UNIX operating system and the X Window System (or OpenWindows). This includes initializing a login session, starting the X Window System, using the window manager, running an X application, and terminating the login session. Running an X application may include running the X application on a remote machine while using a local machine, or an X terminal, as an X display server. The user is strongly advised to seek help if the user is not familiar with any of the above. Appendix D provides some information on UNIX and Appendix E provides some information on the X Window System. Additional information can be found elsewhere.

1.4 Assumptions About the User

CAP users are qualified offshore structural engineers who are also familiar with the UNIX operating system and the X Window System. Users must have an understanding of the following:

1- the principles of finite element analysis,

2- modeling offshore structures,

3- modeling soil conditions,

4- modeling wave loads on offshore structures, and

† UNIX is a trademark of Bell Laboratories.

5- the concepts of nonlinear static and dynamic analysis.

1.5 Order In Which You Read this Manual

The first three sections give the reader a general overview of CAP. It is recommended that all users read them.

Section 4 explains the graphical user interface elements which are parts of CapView. The user is referred to this section several times in the first three sections. It may be helpful to read this section first.

Section 5 describes the behavior of CapView. It is very important that users get familiar with different CapView modes, mouse events, etc. This section should be read after the first four sections.

Sections 6 through 11 describe in detail the various menu selections and dialogue boxes of CapView. The text of these sections is intended to be reference material. It provides tips on how to use CapView. You can read these chapters in any order.

Section 12 gives the reader examples on how to perform certain model building operations. This section may be read after the first five sections.

Appendix A provides a list of hardware and software requirements.

Appendix B describes the software installation procedure.

Appendix C describes how to set the users' environment.

Appendix D provides information on UNIX.

Appendix E provides information on the X Window System.

1.6 Electronic Form of the Manual

The entire text of the Reference manual is available online. Using CapView, the user can review any section of the reference manual by selecting from a menu of sections. The user can then either display this section on the screen or print it. For more information on the electronic manual see section 4.2.

1.7 Other Documentation

Other CAP documentation which you may find useful includes:

- A set of CAP primers which illustrate various aspects of CAP in a brief, informal way. It is highly recommended that you flip through these primers before you start to use CAP.
- A Users' Manual which discusses nonlinear analysis, capacity analysis and CAP's element library.
- A Theoretical Manual which provides details on CAP's solution strategies, loading conditions, element behavior, and so forth.

2.0 Limitations

2.1 Stiffness Model Limitations

CapView is capable of building 3-D finite element models made up of line elements. These line elements can have either framed or truss connections. Some of these members, however, allow for force end releases. The line elements can exhibit different types of linear and nonlinear structural behavior. Any line element can be switched from one type of structural behavior to another by changing the element type.

The finite elements are modeled in a cartesian coordinate system. This coordinate system is fixed in space so that the positive Z-axis points vertically up (and the direction of gravity is parallel to the negative Z-axis).

The finite element analysis program that CapView uses (SeaStar) is capable of using other finite element types. These element types include a quadrilateral finite element, cable spring element and a 3-D gap friction element. Also, SeaStar allows the user to orient any (one) axis vertically. However the CapView Interface does not provide access to these features at this time.

2.2 Foundation Model Limitations

CapView limits the user to either specify rigid displacement boundary conditions or pile supports. The soil properties that the user can specify are limited to automatically generated axial, lateral, and tip force-displacement curves (based on API recommendations), or direct input of the soil P-Y and T-Z curves.

2.3 Load Model Limitations

CapView's load modeling capabilities are limited to nodal force and displacement loads, member gravity (self weight) and buoyancy loads, push-over load patterns derived using a combination wave and/or current loads, member hydrodynamic loads due to waves and current, and seismic acceleration records. The pushover loads are generated automatically by CapView.

3.0 GUI COMPONENTS

When the development of CAP started, none of the popular Graphical User Interface (GUI) standards for the X Window System version 11 were available for developers. Therefore, it was decided to use the Athena Widget Set. This widget set was developed at MIT to provide a guideline on how to implement a GUI for X11. During the development of CAP, it was realized that the Athena Widget Set was not rich enough and that new GUI components were needed. These new components were developed as an extension to the Athena Widget Set. This section provides a brief review of the GUI components used in CAP.

3.1 Button

Buttons are used to execute a command. For most buttons, the name of the command is usually the title on the button. Instead of a text title, some buttons have an icon title which represents the operation. Note that buttons are rectangular in shape and that when running on an X server that supports the shape extension, the corners of the rectangle are rounded.

To select a button (i.e. execute the command) you have to move the mouse cursor to the button. The mouse cursor will change to a pointing finger. Then you must click the left mouse button. Make sure that when you click the mouse button you do not move the mouse cursor outside of the button before you release the mouse button. If you press the left mouse button inside the button and move the cursor outside the button then release the mouse button, the command will not be executed.

Menu buttons are a special case of buttons. The purpose of a menu button is to display a pull-down menu. For more information on these buttons see the pull-down menus section below.

3.2 Menus

Menus are used to present the user with a list of selections. The selection list is usually long and all the items that are in one menu are somehow related.

There are two types of menus, pull-down and pop-up. To display a pull-down menu move the mouse cursor to the menu button. Then press the left mouse button. This will cause the menu to appear. If you release the mouse button the menu will disappear. If you drag the mouse cursor inside the menu, you will notice that the individual selections in the menu reverse the foreground and background colors (reverse video) when the mouse is close to them and recover their original colors as the mouse leaves them. If you release the mouse button while any of the selections is in reverse video, this selection is executed and the menu disappears. If none of the selections is in reverse video, then the menu disappears without any further actions.

Some menu selections have a submenu icon to the far right of the menu selection. When the mouse cursor is dragged inside one of these menu selections a submenu pops up. The behavior of the pop-up menus is similar to the behavior of the pull-down menus. Note, however, that a pop-up menu will disappear if the mouse cursor is dragged back into the pull-down selection that pops up the pop-up menu.

3.3 Dialogue Box

A dialogue box is a pop-up window that presents the user with data and/or questions and requests some user input. Most of the dialogue boxes in CAP freeze all activities in CAP (except for the dialogue box itself) until the user satisfies the questions or requests of the dialogue box. However, some dialogue

boxes allow the use of the rest of CAP.

3.4 Error Message

An error message (notifier) is a pop-up window that presents the user with an error message and a button labeled OK. The error message freezes all activities in CAP until the user selects the OK button.

3.5 Label

A label is a piece of text or an icon that identifies another GUI component. Labels are used as titles for dialogue boxes, entry fields, or groups of GUI controls.

3.6 Edit Field

Edit fields allow the user to type input from the keyboard. Edit fields respond to a large list of commands that can be typed on the keyboard or issued by the mouse buttons. Note that an edit field responds to these commands only when the mouse cursor is inside the edit field rectangle.

Following is a list of the control commands. Note that <Meta> in this table refers to the meta key on the keyboard, which is usually identified with a diamond symbol.

KEY	ACTION	DESCRIPTION
<Ctrl>-K	Kill to end of line	This action deletes the entire line to the right of the insert point position, and stuffs the deleted text into the kill buffer.
<Ctrl>-Y	Unkill	This action retrieves the contents of the kill buffer and inserts it at the current insert point location.
<Ctrl>-A <Ctrl>-E	Beginning of line End of line	These actions move to the beginning or end of the current line. If the insert point is already at the end or beginning of the line, then no action is taken.
<Ctrl>-D <Delete> <Backspace>	Delete next character Delete previous character	These actions remove the character immediately before or after the insertion point. If a Carriage Return is removed, then the next line is appended to the end of the current line.
<Ctrl>-F <Ctrl>-B	Forward character Backward character	These actions move the insert point forward or backward one character in the edit field. If the insertion point is at the end or beginning of a line this action will move the insert point to the next (or previous) line.
<Meta>-F <Meta>-B	Forward word Backward word	These actions move the insertion point to the next or previous word boundary. A word boundary is defined as a Space, Tab or Carriage Return.
<Meta>-D <Meta>-H	Delete next word Delete previous word	These actions remove all characters between the insertion point location and the next word boundary. A word boundary is defined as a Space, Tab or Carriage return.

<Meta>-D <Meta>-H	Kill word Backward kill word	These actions act exactly like the delete-next-word and delete-previous-word actions, but they stuff the word that was killed into the kill buffer.
<Left Down>	Select start	This action sets the insertion point to the current pointer location. It will then begin a selection at this location. If many of these selection actions occur quickly in succession, then the selection count mechanism will be invoked.
<Left Motion>	Select adjust	This action allows a selection started with the select-start action to be modified (add or drop characters).
<Left Up>	Select end	This action ends a text selection that began with the select-start action.
<Right Down>	Extend start	This action finds the nearest end of the current selection, and moves it to the current pointer location.
<Right Motion>	Extend adjust	This action allows a selection started with an extend-start action to be modified.
<Right Up>	Extend end	This action ends a text selection that began with the extend-start action.
<Center Down>	Insert selection	This action retrieves the value of the first (left-most) named selection that exists or the cut buffer that is not empty and inserts it into the text at the current insert point location.
<Ctrl>-W	Kill selection	This action deletes the current selection and stuffs the deleted text into the kill buffer.
Any other key	Insert character	This action inserts an equivalent character (sequence) into the text at the current insert point location.

3.7 Radio Button List

A radio button list is a list of exclusive settings (selections). The user can select one and only one selection from the list. Radio buttons are displayed as a text or icon label preceded by square box. If the radio button is selected, then a solid square is displayed inside the radio button's square box. When the mouse cursor is close to the radio button, the cursor turns into a check mark. If you press the left mouse button inside an unselected radio button, then this radio button is selected and the previously selected radio button in the list is unselected.

3.8 Combo Box

The purpose of a combo box is similar to that of a radio button list. However, in the interest of saving screen space, only the title of the selected radio button is normally displayed. In addition, there is a button with a down arrow. When this button is pressed, the full list of selections is displayed. Note that the selections do not resemble radio buttons and that the selected item is displayed in reverse video. The user can either select the down arrow button to hide the list without changing the selection, or click the left mouse button on any of the items to select it and hide the list.

3.9 Check Box

A check box list is a list of nonexclusive settings (selections). The user can either select none of the items or select one or more items from the list. Check boxes are displayed as a text or icon label preceded by diamond box. If the check box is selected, then a solid diamond is displayed inside the check box's diamond box. When the mouse cursor is close to the check box, the cursor turns into a check mark. If you press the left mouse button inside an unselected check box, then this check box is selected. If you press the left mouse button inside a selected check box, then this check box is unselected.

3.10 Reverse Video Check Box List

A reverse video check box list is a check box list with a different shape. A reverse video check box utilizes reverse video to show that it is selected instead of using the diamond box. Reverse video check box lists are used when the number of selections is large, and it is difficult to identify selected items.

3.11 Scroll Bar

A scroll bar is used to present a large object (a list of items, or a large GUI component) in a small region. Scroll bars can be either horizontal or vertical. Each scroll bar has a thumb (a slide bar). The thumb is used to scroll the displayed object and is used to give visual feedback on the percentage of the object that is visible. When the thumb is in the lower (left side) part of the slide area, it indicates that the upper (right side) of the object is visible. In effect, the thumb behaves like an elevator counter weight.

Each mouse button invokes a specific action. The left and the right mouse buttons move the thumb of the scroll bar a preset "jump" amount. The left mouse button, when clicked over the vertical (horizontal) scroll bar, moves the displayed list or component upwards (to the left) while the right mouse button moves the displayed object downwards (to the right).

The center mouse button, when clicked over the vertical (horizontal) scroll bar, moves the object so that the top (left edge) of the thumb moves to the location of the mouse cursor. If you keep the center button pressed and move the mouse

cursor, the thumb tries to follow the mouse cursor, and in effect the object can be scrolled "smoothly". If the object is complicated requiring a long time to redraw, and/or the machine is slow, this scrolling will not be smooth.

3.12 Message Field

A message field is a field in which text is displayed. A message field may have a vertical and/or horizontal scroll bar if the text to be displays does not fit in the provided region.

With every message field there is a search button on the dialogue box (normally below the message field). The Search button, when selected, displays a search dialogue box. The search dialogue box allows the user to search forward and backward for text. The user can select between forward and backward search by toggling between the two provided selections. The user can enter the text to be searched for in the edit field labeled "Search For". To perform the search select the button labeled Search. To close the search dialogue box, select the button labeled Cancel.

3.13 Display Window

The display window is a GUI component that is capable of displaying a drawing. It has vertical and horizontal scroll bars. It also has the ability of taking mouse button commands. For more information on the display window and its capabilities, see Chapter 5 (Using CAP).

4.0 STARTING CAP

To run CAP, make sure that you are running X11 release 4 or a compatible X11 server (e.g. OpenWindows 3.0) and type the following command at the UNIX system prompt.

CapView

Note that you can also run CapView through the window manager by programming a window manager menu selection or using OPEN LOOK's drag and drop capabilities to start CapView from OpenWindows' file manager.

When you first run CapView, two applications (with two windows) are created: CapView and CapHelp (Figure 4-1). The first application, CapView, is the application that allows the user to create models, view them, perform analyses, and postprocess analysis results. CapView, as shipped, starts a portrait window. This window has six menu buttons across its top, a large scrollable draw window in the middle, several buttons across the bottom, a small window displaying orientation axes of the scrollable window, and a status line informing the user that there are no open projects.

The second application, CapHelp, is an on-line documentation browser. It allows the user to browse through this Reference Manual. CapHelp,as shipped, starts in an iconized state. When CapHelp's icon is opened (Figure 4-2), it opens a landscape window. This window has an index organized in a scrollable list of selectable sections. Underneath the help index the CapHelp window has three buttons.

4.1 CapView Menus

CapView has 6 pull-down menu buttons across the top of its window (Figure 4-1). These menu buttons are labeled File, Model, Loads, Analysis, Results, and Options.

The File menu presents the user with selections that will help in creating, importing, opening, saving, abandoning, recovering, and deleting model files. It also helps the user change working directories, and exit CAP (Figure 4-3). For more information see section 6.0.

The Model menu presents the user with selections that will help in editing models by adding new structural elements, nodes, material properties, and cross-section descriptions to the model description (Figure 4-4). For more information see section 8.0.

The Loads menu presents the user with selections that will help in editing loads by adding new nodal loads and boundary conditions, specifying current and wave loads and specifying seismic loads (Figure 4-5). For more information see section 9.0.

The Analysis menu presents the user with selections that will help in defining edit macros, performing data checks, creating and editing a SeaStar input files and running a SeaStar analysis (Figure 4-6). For more information see section 10.0.

The Results menu presents the user with selections that will help in reviewing and processing the results of the analysis, animating the deformed structure, color coding members (based on their interaction ratios and inelastic events) and creating X-Y plots of member forces, IRs, node displacements, etc (Figure 4-7). For more information see section 11.0.

The Options menu presents the user with selections that will help the user in customizing the features of CapView to the user's preferences. It also allows

the user to create geometric plot files and print these and other print files created through the Results menu selections (Figure 4-8). For more information see section 7.0.

4.2 Using CapHelp

The CapHelp window has a scrollable list of exclusive selections (Figure 4-2). These selections represent the sections of this Reference Manual. Notice that one and only one selection is displayed in reverse video (with the colors of the foreground and background reversed when compared with the other selections). This selection is the selected item. To make a different selection, the user must move the mouse cursor to the desired selection. As the mouse cursor enters the region of a selection, it changes to a check mark. Click the left mouse button. Notice that the new selection reverses its foreground and background colors while the old selection returns to normal colors.

Below the scrollable list of selections, there are three buttons labeled: Show, Print, and Done. If the Show button is selected, CapHelp will create a dialogue box labeled Help Section that has a scrollable message field (text window) (Figure 4-9). This window displays a section from the manual (without figures). Below the scrollable window there are two buttons labeled Search and Done. The Done button, when selected, closes the Help Section dialogue box. The Search button, when selected, displays a search dialogue box (Figure 4-10). The search dialogue box allows the user to search forward and backward for text. The user can search forward or backward by toggling between the two provided selections. The user can enter the text to be searched for in the edit field labeled "Search For". To perform the search, select the button labeled Search. To close the search dialogue box, select the button labeled Cancel.

If the Print button of the CapHelp window is selected, CapHelp will issue a UNIX print command to print the selected section on the default printer.

If the Done button of the CapHelp window is selected, CapHelp will display a dialogue box to verify that the user wants to exit CapHelp (Figure 4-11). The dialogue box provides two buttons labeled Yes and No. Selecting the button labeled Yes will terminate CapHelp (without affecting CapView) while selecting the button labeled No will ignore the exit request.

4.3 Exiting CAP

To exit CAP, select Exit CapView from the File Menu (Figure 4-3). This will display a verification dialogue box verifying that the user wants to exit CapView (Figure 4-12). Select the button labeled Yes. Note that when you exit CapView, CapHelp also terminates.

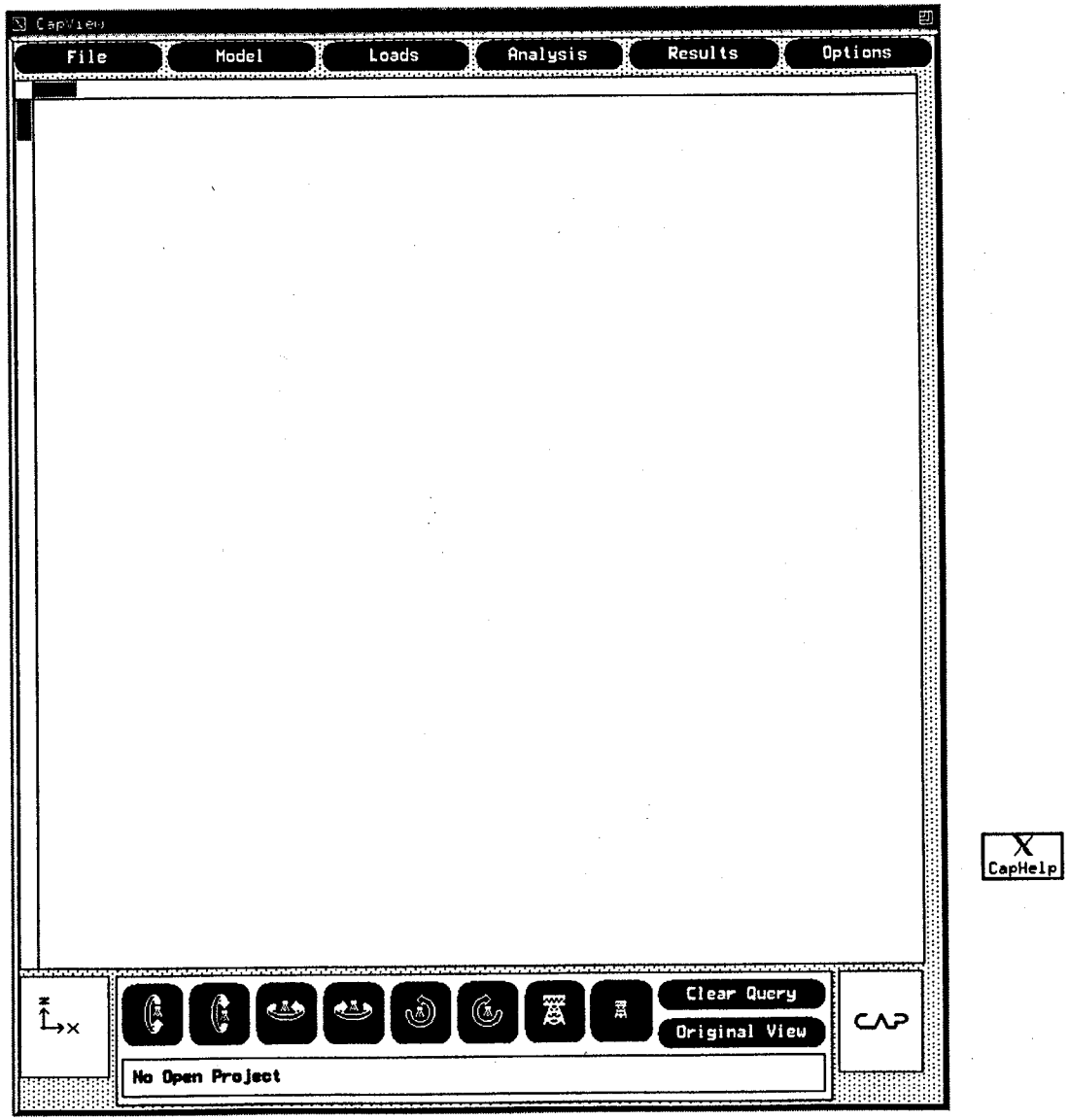

CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
No Open Project
CapHelp

Figure 4- 1

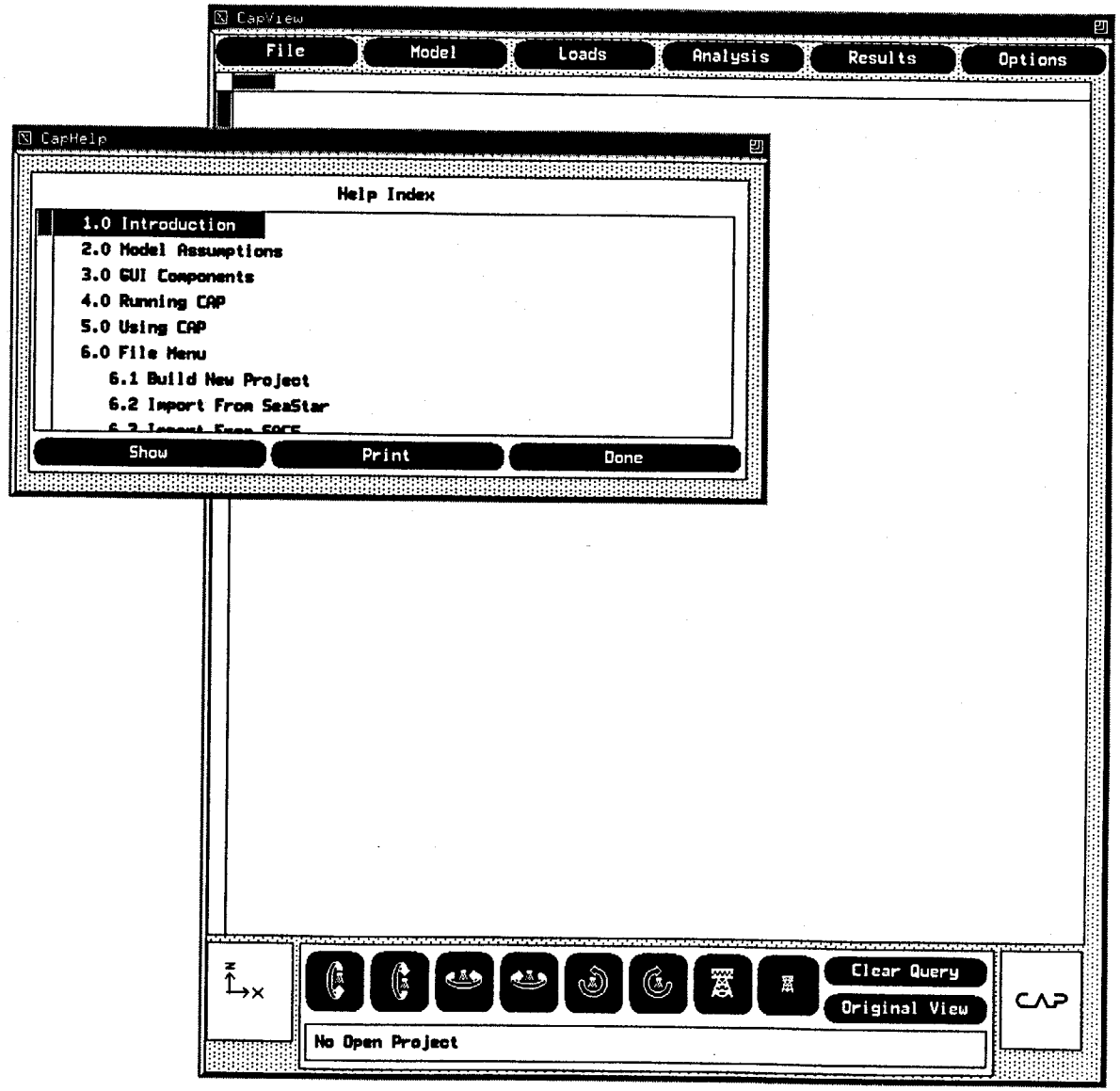
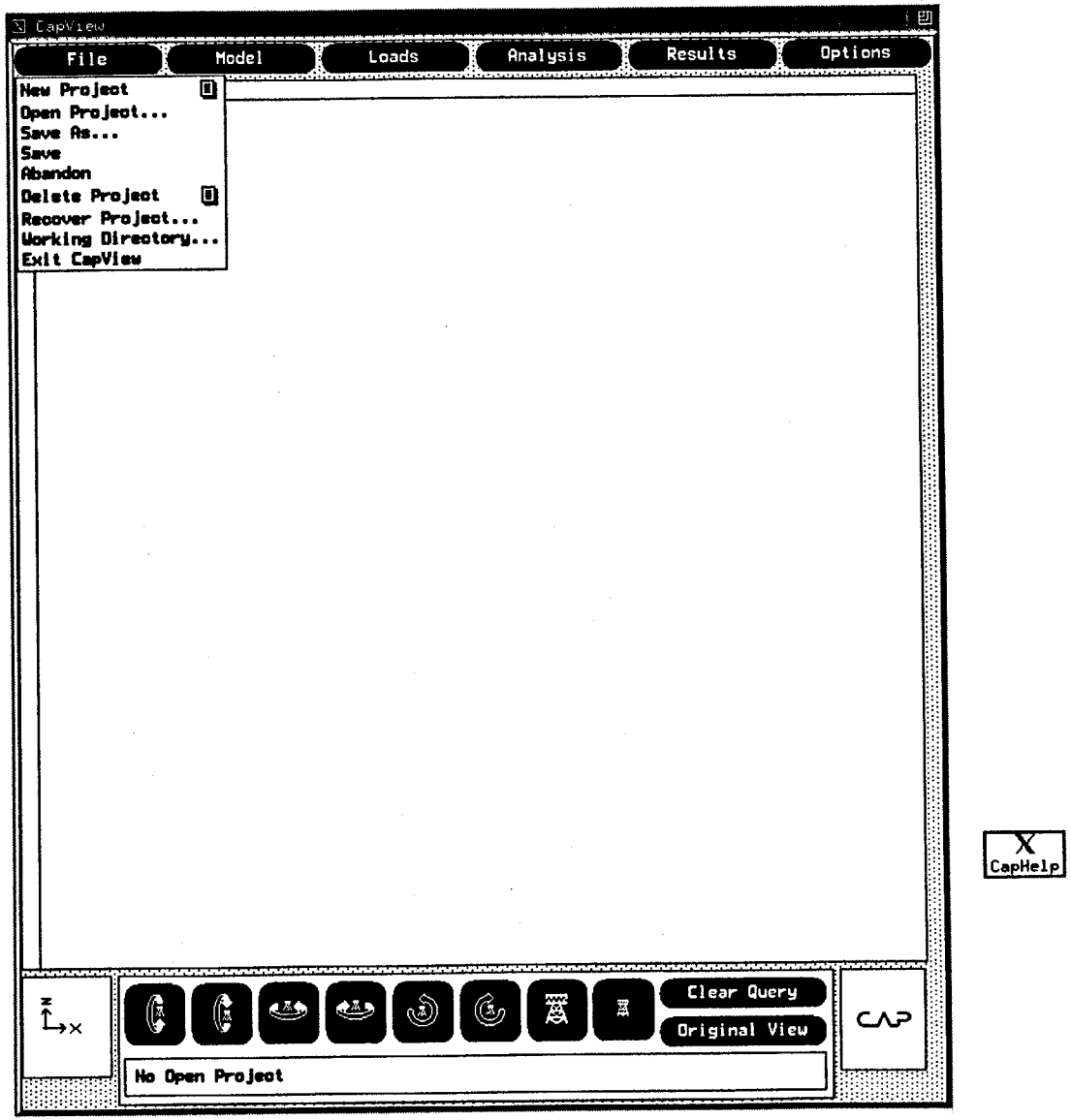

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
Help Index
1.0 Introduction
2.0 Model Assumptions
3.0 GUI Components
4.0 Running CAP
5.0 Using CAP
6.0 File Menu
6.1 Build New Project
6.2 Import From SeaStar
Show
Print
Done
Clear Query
Original View
No Open Project
CAP

Figure 4-2


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
Clear Query
Original View
No Open Project

CapHelp

Figure 4-3

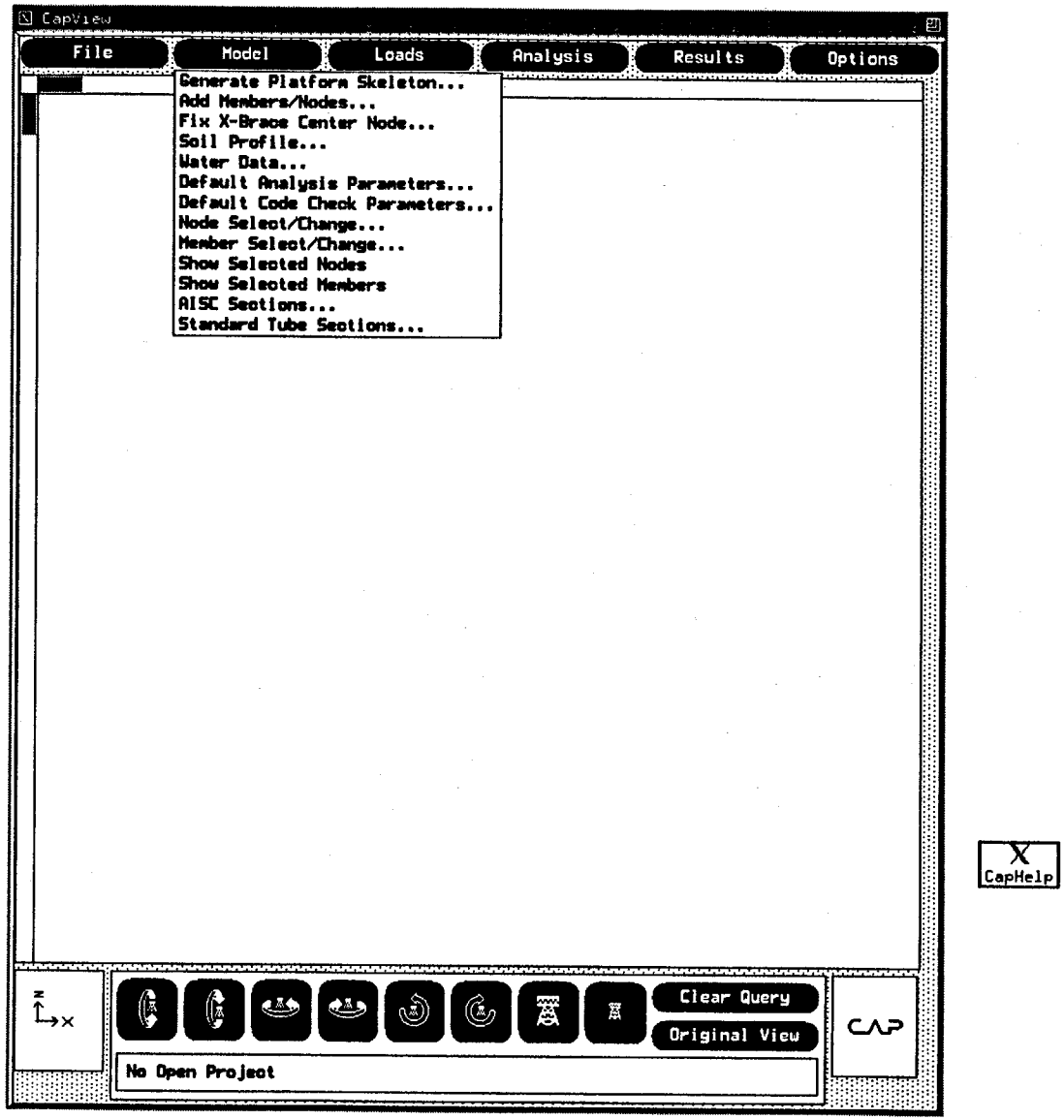

CapView
File
Model
Loads
Analysis
Results
Options
Generate Platform Skeleton...
Add Members/Nodes...
Fix X-Brace Center Node...
Soil Profile...
Water Data...
Default Analysis Parameters...
Default Code Check Parameters...
Node Select/Change...
Member Select/Change...
Show Selected Nodes
Show Selected Members
AISC Sections...
Standard Tube Sections...
Clear Query
Original View
No Open Project
CapHelp

Figure 4-4

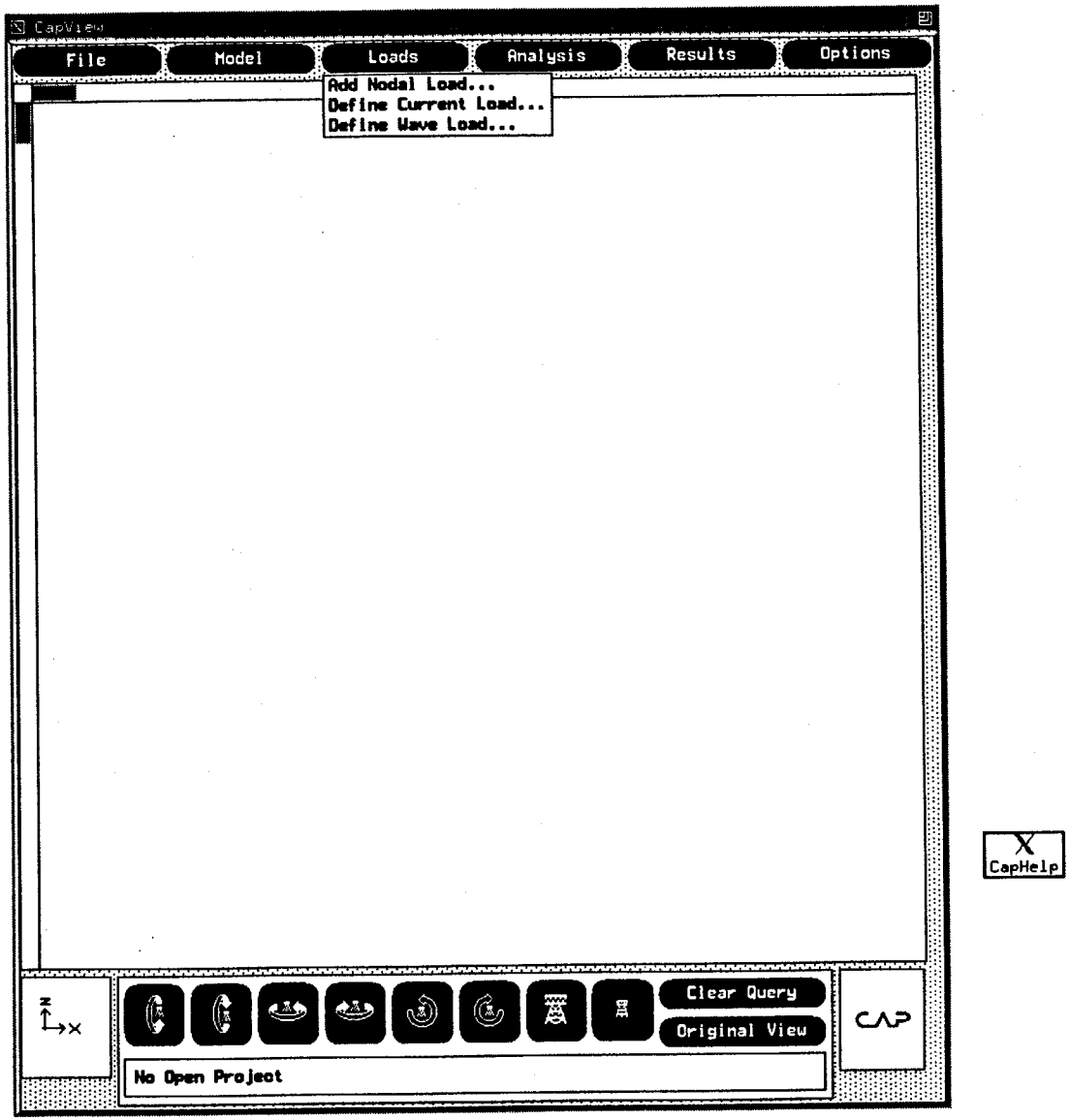

CapView
File
Model
Loads
Analysis
Results
Options
Add Nodal Load...
Define Current Load...
Define Wave Load...
Clear Query
Original View
No Open Project
CapHelp

Figure 4-5

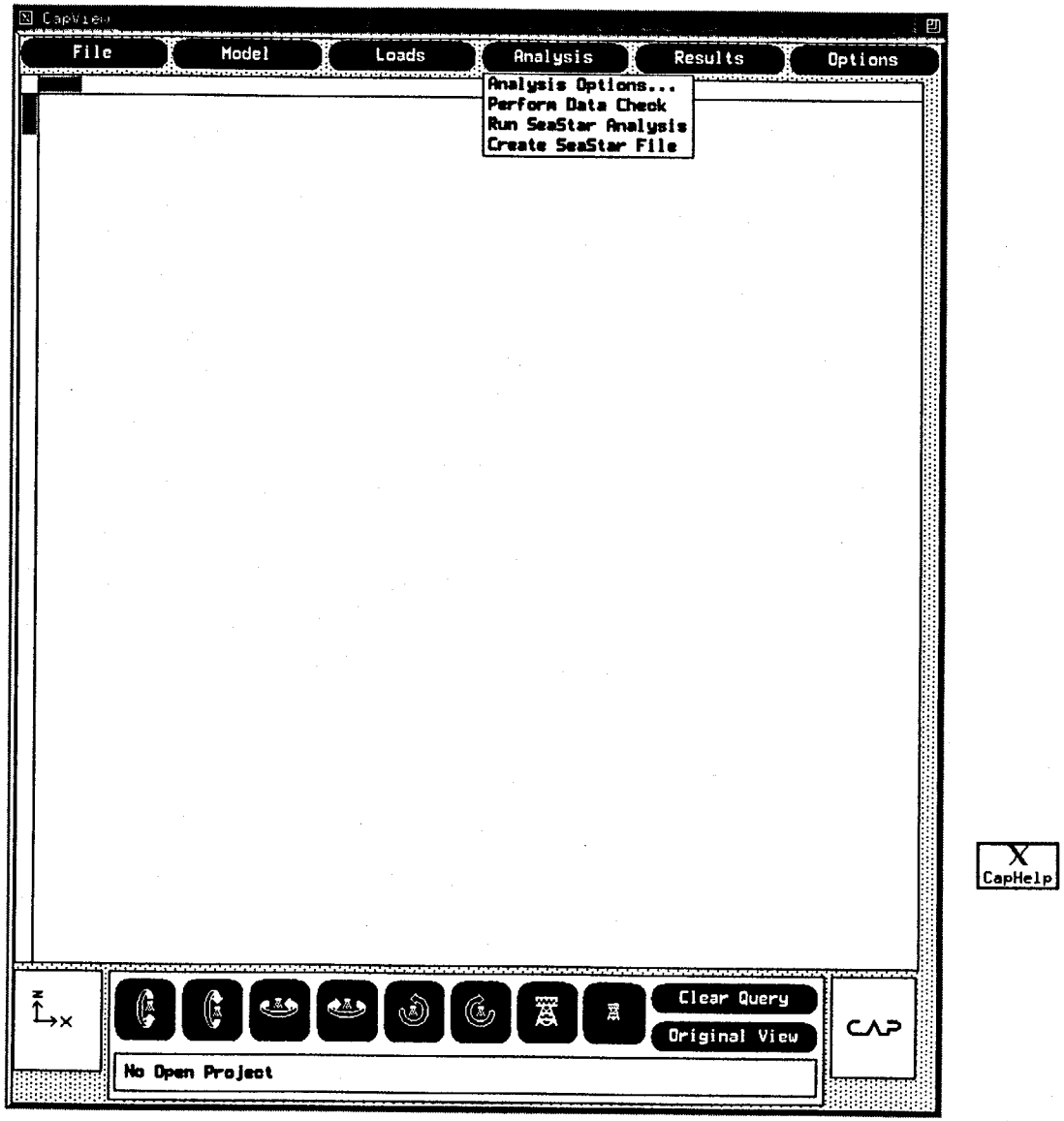

CapView
File
Model
Loads
Analysis
Results
Options
Analysis Options...
Perform Data Check
Run SeaStar Analysis
Create SeaStar File
Clear Query
Original View
No Open Project
CapHelp

Figure 4-6

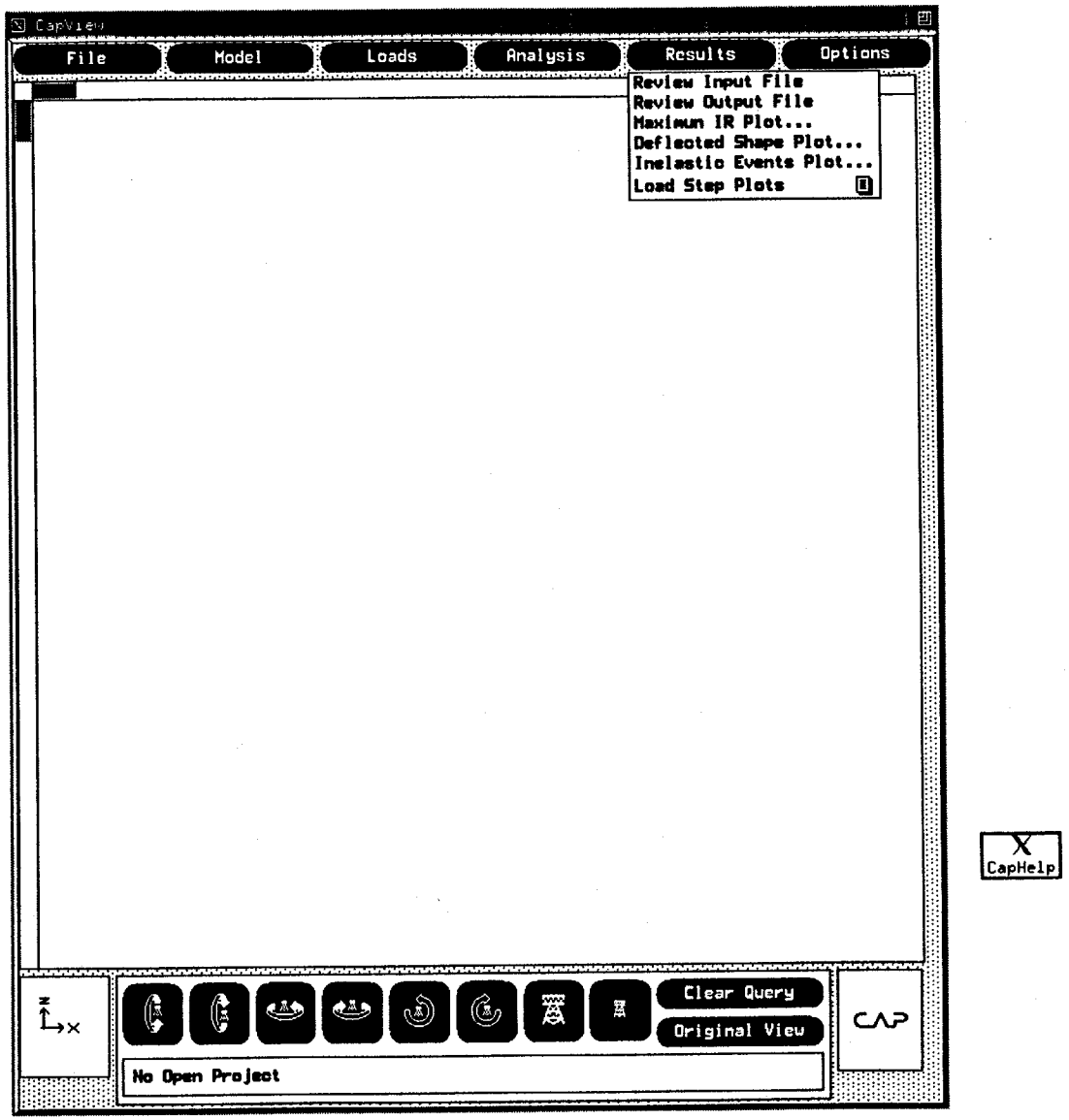

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Maximun IR Plot...
Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Clear Query
Original View
No Open Project
CapHelp

Figure 4-7

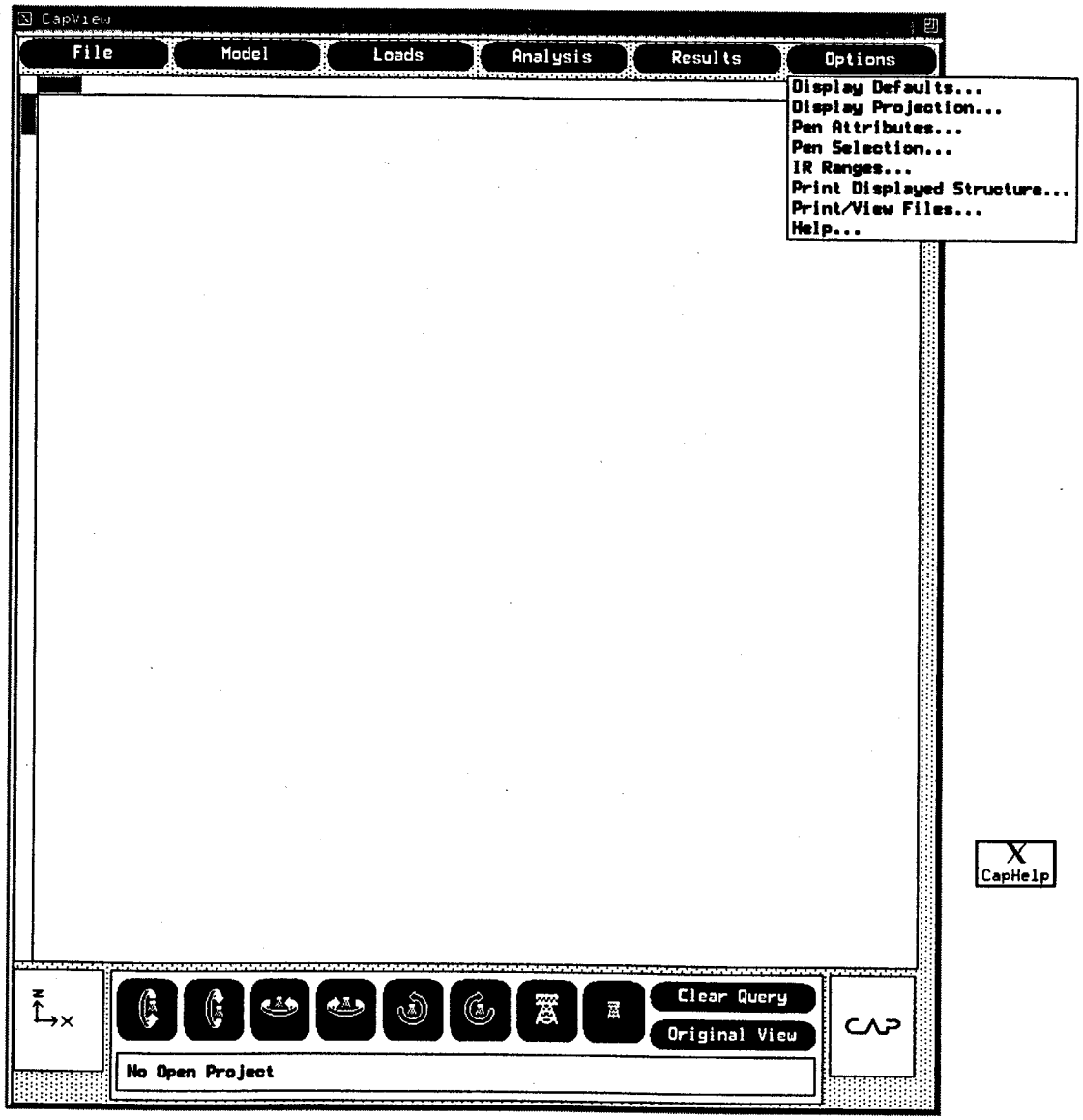

CapView
File
Model
Loads
Analysis
Results
Options
Display Defaults...
Display Projection...
Pen Attributes...
Pen Selection...
IR Ranges...
Print Displayed Structure...
Print/View Files...
Help...
CapHelp
Clear Query
Original View
No Open Project

Figure 4-8

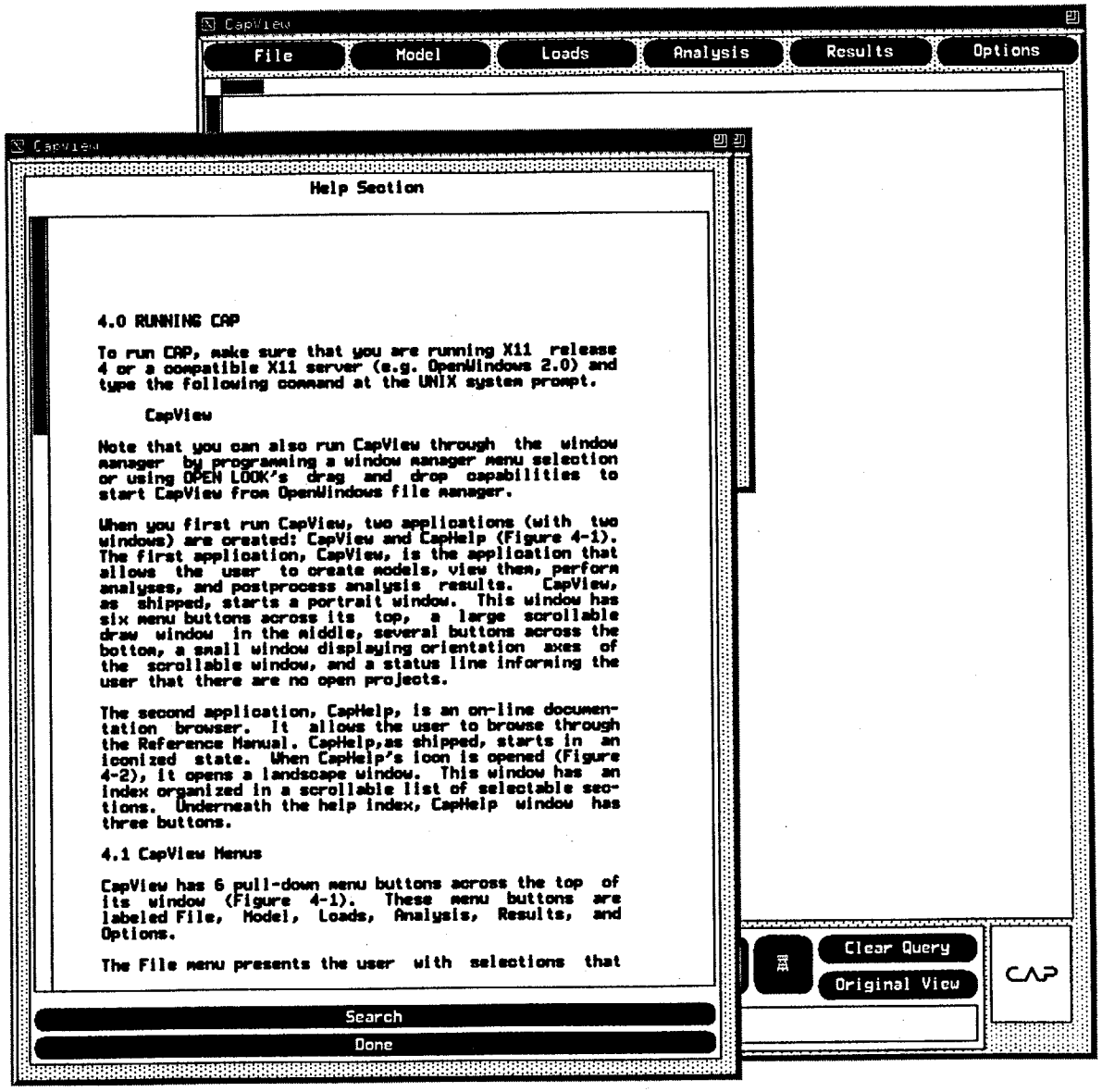

CapView
File
Model
Loads
Analysis
Results
Options
Help Section
4.0 RUNNING CAP
To run CAP, make sure that you are running X11 release 4 or a compatible X11 server (e.g. OpenWindows 2.0) and type the following command at the UNIX system prompt.
CapView
Note that you can also run CapView through the window manager by programming a window manager menu selection or using OPEN LOOK's drag and drop capabilities to start CapView from OpenWindows file manager.
When you first run CapView, two applications (with two windows) are created: CapView and CapHelp (Figure 4-1). The first application, CapView, is the application that allows the user to create models, view them, perform analyses, and postprocess analysis results. CapView, as shipped, starts a portrait window. This window has six menu buttons across its top, a large scrollable draw window in the middle, several buttons across the bottom, a small window displaying orientation axes of the scrollable window, and a status line informing the user that there are no open projects.
The second application, CapHelp, is an on-line documentation browser. It allows the user to browse through the Reference Manual. CapHelp,as shipped, starts in an iconized state. When CapHelp's icon is opened (Figure 4-2), it opens a landscape window. This window has an index organized in a scrollable list of selectable sections. Underneath the help index, CapHelp window has three buttons.
4.1 CapView Menus
CapView has 6 pull-down menu buttons across the top of its window (Figure 4-1). These menu buttons are labeled File, Model, Loads, Analysis, Results, and Options.
The File menu presents the user with selections that
Search
Done
Clear Query
Original View
CAP

Figure 4-9

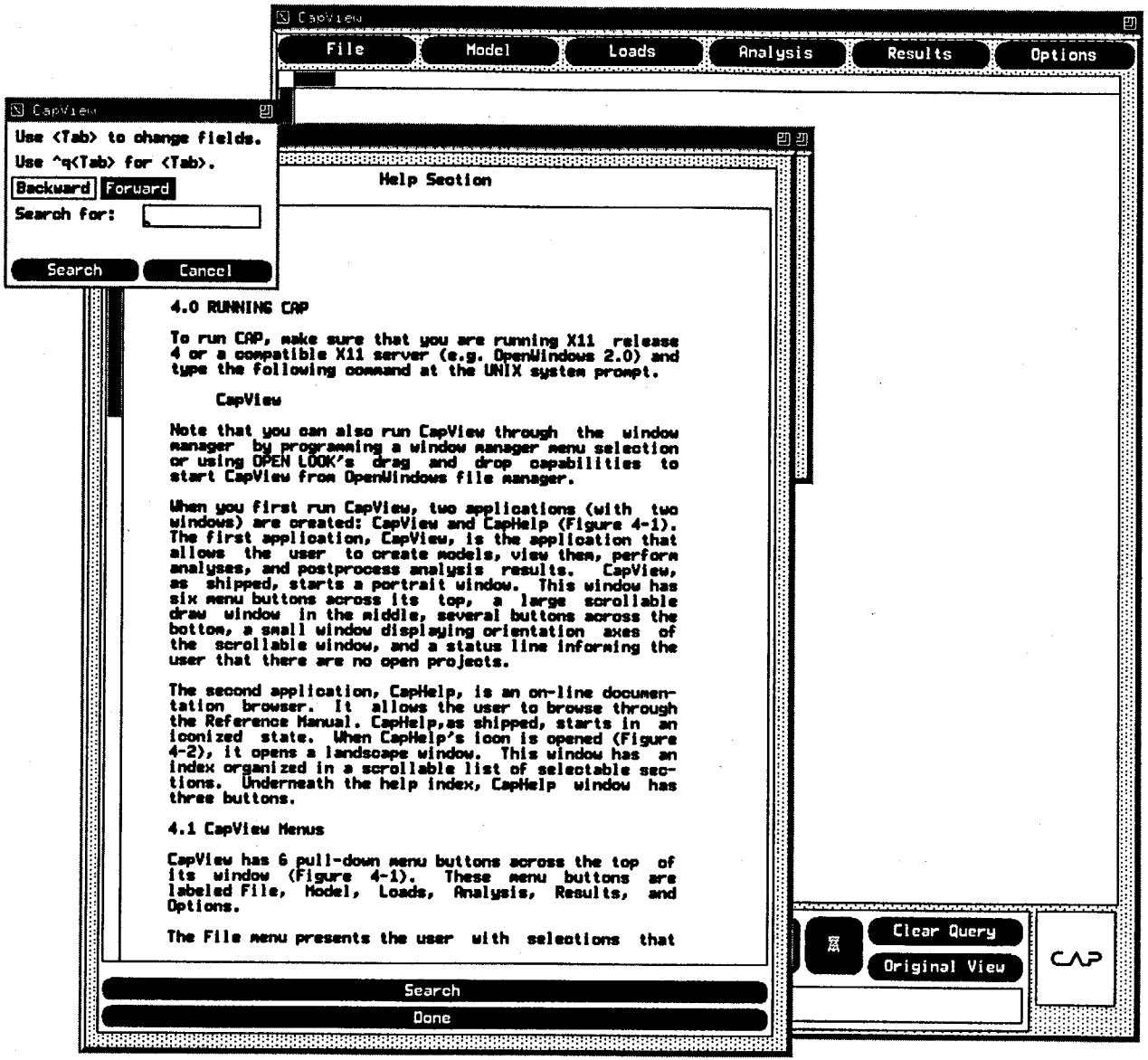

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Use <Tab> to change fields.
Use ^q<Tab> for <Tab>.
Backward
Forward
Search for:
Search
Cancel
Help Section
4.0 RUNNING CAP
To run CAP, make sure that you are running X11 release 4 or a compatible X11 server (e.g. OpenWindows 2.0) and type the following command at the UNIX system prompt.
CapView
Note that you can also run CapView through the window manager by programming a window manager menu selection or using OPEN LOOK's drag and drop capabilities to start CapView from OpenWindows file manager.
When you first run CapView, two applications (with two windows) are created: CapView and CapHelp (Figure 4-1). The first application, CapView, is the application that allows the user to create models, view them, perform analyses, and postprocess analysis results. CapView, as shipped, starts a portrait window. This window has six menu buttons across its top, a large scrollable draw window in the middle, several buttons across the bottom, a small window displaying orientation axes of the scrollable window, and a status line informing the user that there are no open projects.
The second application, CapHelp, is an on-line documentation browser. It allows the user to browse through the Reference Manual. CapHelp,as shipped, starts in an iconized state. When CapHelp's icon is opened (Figure 4-2), it opens a landscape window. This window has an index organized in a scrollable list of selectable sections. Underneath the help index, CapHelp window has three buttons.
4.1 CapView Menus
CapView has 6 pull-down menu buttons across the top of its window (Figure 4-1). These menu buttons are labeled File, Model, Loads, Analysis, Results, and Options.
The File menu presents the user with selections that
Search
Done
Clear Query
Original View
CAP

Figure 4-10

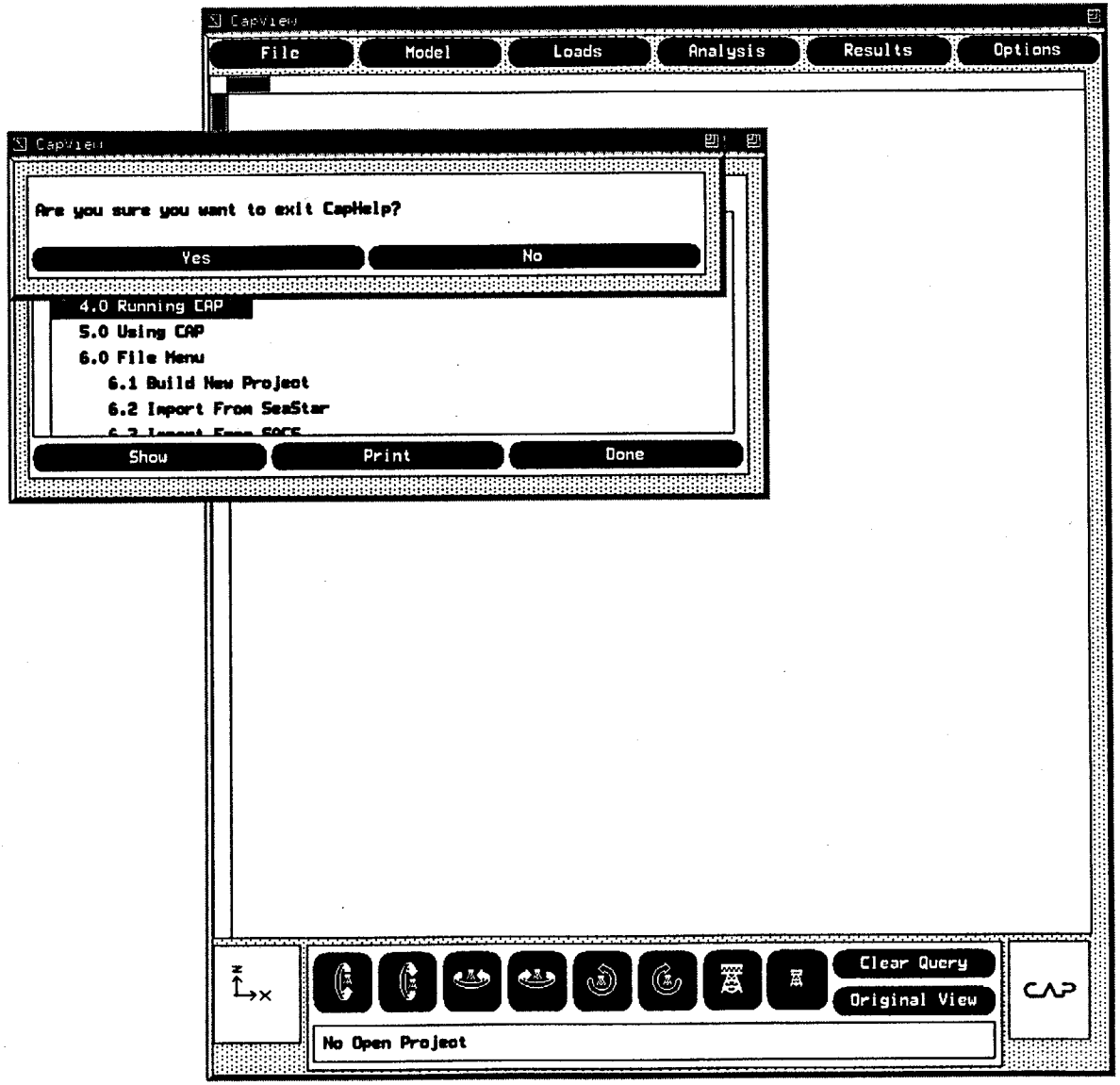

Capview
File
Model
Loads
Analysis
Results
Options
Capview
Are you sure you want to exit CapHelp?
Yes
No
4.0 Running CAP
5.0 Using CAP
6.0 File Menu
6.1 Build New Project
6.2 Import From SeaStar
Show
Print
Done
Clear Query
Original View
No Open Project

Figure 4-11

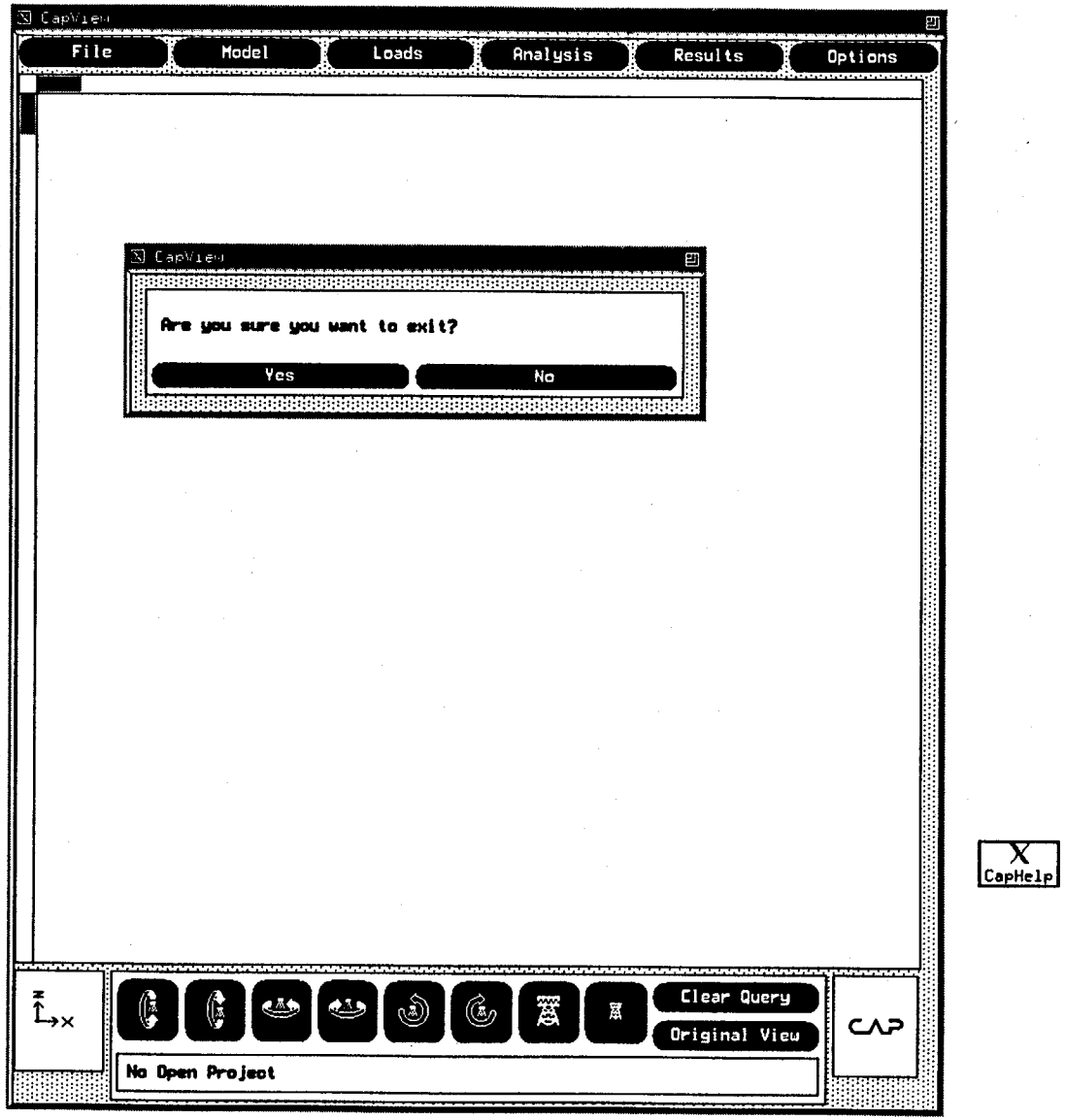

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Are you sure you want to exit?
Yes
No
Clear Query
Original View
No Open Project
CAP
CapHelp

Figure 4-12

5.0 USING CAP

In the previous section the reader was introduced to how to start CapView and how to make use of the CapHelp utility. This section attempts to introduce the user to different mouse actions and CapView modes, representation of different structural objects (nodes, members, soil springs, and loads), basic view controls, and information dialogue boxes of Nodes, Generic Members, Groups, Generic Materials, Nodal Forces, and PSAS Elements.

5.1 Mouse Actions

The display window of CapView reacts to several mouse actions (events). In the following subsections these actions are described.

5.1.1 Structural Objects Mouse Highlight

Mouse highlights are used to identify an object as the operand for a subsequent mouse action. As the mouse cursor moves inside the display window, it passes near the various structural objects of a displayed model. Each time the mouse cursor gets near a structural object, this object gets highlighted (Mouse Highlight) and the previously highlighted object becomes unhighlighted (Figures 5-1 and 5-2). At any point in time, there can be at most one highlighted object. CapView uses a set of pens to draw objects (sections 7.3 and 7.4). Normally an object is drawn using the Default Pen. When a object is highlighted, it is drawn using the Mouse Highlight Pen to set it apart from other objects.

5.1.2 Structural Objects Information (Dialogue Boxes)

When the user clicks the Left Mouse Button inside the display window, CapView looks for the mouse highlighted object. If it finds one, CapView displays a dialogue box with information about that object. Later in this section, the data displayed in the various structural object information dialogue boxes are described.

5.1.3 Rubber-Band Zoom

Rubber-band zoom allows the user to zoom to any portion of the display window. To perform a rubberband zoom, move the mouse cursor to the upper left corner of the rectangle to be zoomed, press and hold the Center Mouse Button (Figure 5-3), drag the mouse towards the lower right hand corner of the rectangle (Figure 5-4), and release the Center Mouse Button. After the button is released, CapView zooms to the region bounded by the rubber-band rectangle (Figure 5-5). While the center mouse button is pressed, CapView will display a rectangle representing the area to be zoomed. This rectangle grows and shrinks as the mouse cursor is moved (like a rubber-band rectangle). The width and height of the rubber-band rectangle stays proportional to the width and height of the display window at all times. Note that if you drag the mouse (while the Center Mouse Button is pressed) to the left of or above the upper left corner of the rectangle, that corner is moved to follow the mouse.

If you press the right mouse button, CapView will display the view that was shown previous to the most recent rubber-band zoom operation. Note that if you press the right mouse button before you perform the first rubber-band zoom you will get an error message since there is no "previous" view.

5.1.4 Mouse (Stack) Select

Mouse highlight allows the user to select one structural object to be an operand for a subsequent mouse action. Certain operations require more than one

operand. Some operations require a specific order in which the operands are specified. For these operations, the mouse select feature is provided. Mouse selections are used for several operations including adding or deleting structural objects (sections 8.2 and 9.1) and selecting members in an X-brace to fix the location of the center node (section 8.3).

In mouse (stack) selection, up to eight different structural objects can be selected and arranged in a stack. The behavior of the stack is similar to that of calculators using Reverse Polish Notation (RPN). When CapView starts, the mouse select stack is empty. The user can add objects to the stack. When the user adds the fifth object to the stack the first object is dropped. Each additional object added to the stack will drop the oldest object in the stack.

To add an object to the stack, move the mouse cursor near the object to highlight it, then click <Shift> Left Mouse Button. Note that during the time the mouse button is pressed, the status line displays a list of the selected objects (Figure 5-6). To review this list, press and hold the Center Mouse Button. In both cases, when you release the mouse button, the status line will display the message it displayed before displaying the selected objects list.

5.1.5 Query Select

For operations that require an unlimited number of operands without any order, the query selection feature is provided. Query selection is used for selecting objects matching certain criteria, and for change operations on these objects (section 8.8 on Select/Change Nodes and section 8.9 on Select/Change Members), showing members that had inelastic events (section 11.5 on Inelastic Events Plot), and defining the member and node operands for Load Step Plots (sections 11.6 through 11.10).

Objects which are query selected are drawn using the Query Select Pen to differentiate them from other objects (sections 7.3 and 7.4). Note that if the object is highlighted as a result of moving the mouse cursor near it, the object will be drawn using the Mouse Select Pen.

To query select an object, move the mouse cursor near the object to highlight it and click <Meta> Left Mouse Button. When the mouse cursor is moved and another object is highlighted, the object which was query highlighted will be drawn using the Query Select Pen (section 7.4). To unselect an object that is query selected, move the mouse cursor near the desired object to highlight it and click <Meta> Right Mouse Button.

Other operations can also query select and unselect objects. For more information see, the sections on Node Select/Change (8.8), Member Select/Change (8.9), and Inelastic Events Plot (11.5).

5.1.6 Add Members, Nodes, and Nodal Loads.

Utilizing the mouse select actions of the display window along with other mouse clicks, CapView implements an interactive point and click model building capability. The Add Members/Nodes (section 8.2) and Add Nodal Load (section 9.1) dialogue boxes provide a way for defining various add operations and switching among them. Using these dialogue boxes the user can add (1) various member configurations, from a single member, to X-Braces, K-Braces, member splits, etc.; (2) various node configurations, from a single node to a grid of nodes; and (3) nodal loads.

All these operations are accomplished by following simple consistent steps. The details and sequence of the steps required to perform these operations are provided to the user in the dialogue boxes. After selecting the appropriate

dialogue box (Add Members/Nodes (Figure 5-7) or Add Nodal Load (Figure 5-8)) follow these steps:

1- Read the instructions in the instructions section of the dialogue box.
2- Make sure all the data edit fields, check boxes, and radio buttons have the appropriate data and selections.
3- Perform the necessary mouse select(s), if any are required to assemble an argument list, and
4- Click the <Shift> Right Mouse button.

Note that the mapping of the <Shift> Right Mouse button click to "add" operations is valid only when an appropriate add dialogue box is displayed and that one and only one add dialogue box can be displayed at any time. Further information is provided in chapters 8 and 9.

5.1.7 Delete Members, Nodes, and Nodal Loads.

The user can also delete members, nodes, and nodal loads by selecting them with the mouse and clicking <Control> Center mouse button. This operation will delete the last selected object and remove it from the mouse select stack. Note that you cannot delete a node that is connected to a member or has a nodal load applied to it.

5.1.8 Undo

Some of CapViews actions are reversible. CapView has an undo feature that undoes certain operations. Every time a model is opened, CapView starts a new list of reversible operations. This list is kept around until the model is closed. Every time a reversible operation is performed, a record of the operation and how it can be reversed is added to the top of the list of reversible operations. These operations can be undone in reverse chronological order (the most recent operation in the list will be the first operation to be reversed).

Reversible operations include all operations performed using the following dialogue boxes from the Model and Loads Menus (except for undo operations): (1) Add Members/Nodes (section 8.2), (2) Add Nodal Load (section 9.1), and (3) Fix X-Brace Center Node (section 8.3). Delete operations performed using the mouse delete and the Node Select/Change and Member Select/Change dialogue boxes are also reversible.

To undo an operation, click <Control> Right Mouse Button.

5.2 CapView Modes

CapView displays structures in different modes. The different modes are governed by three flags. The first flag is the Double Line Drawing flag, the second flag is the IR Mode flag, and the third is the inelastic events flag. These flags can be set and cleared using the Display Defaults dialogue box from the Options menu (section 7.1).

When the Double Line Drawing flag is cleared, CapView displays wire-frame three dimensional images (Figure 5-9). When the flag is set, CapView displays semi-rendered images (Figure 5-10). These images (although not perfect) resemble three dimensional views with all hidden lines removed. Each member is drawn as a rectangle with a width proportional to the diameter of the member (for a member with a non-tubular cross section, the width of the rectangle will be proportional to the display diameter of the member (section 5.5.2)). Also, when the Double Line Drawing flag is set, the mouse highlight feature is disabled and nodes are not drawn.

When the IR Mode flag is set, CapView displays members in different pens (sections 7.3 and 7.4). For each member, the member's Iteraction Ratio (IR) value is matched against several IR ranges and assigned an appropriate IR range number. All members with the same IR range number are assigned an IR range pen. When the Double Line Drawing flag is set, the Fill pen is replaced by the IR range pen. When the Double Line Drawing flag is cleared, the Default Pen, Mouse Highlight and Query Select pens are replaced by the IR range pen. The pen selection and minimum and maximum values for each IR range can be customized using the IR Ranges (section 7.5) and the Pen Attributes (section 7.3) dialogue boxes. Also, when the IR Mode flag is set, the mouse highlight feature is disabled and query selected members are drawn using the appropriate IR range pens.

When the Inelastic Event Mode flag is set, CapView displays members in different pens (sections 7.x and 7.x). This mode is useful when animating the behaviour of a structure during a storm or an earthquake (section x.x). For each time/load step, every member, is assigned an appropriate event code. All members with the same event code are assigned an event code pen. When the Double Line Drawing flag is set, the Fill pen is replaced by the event code pen. When the Double Line Drawing flag is cleared, the Default Pen, Mouse Highlight and Query Select pens are replaced by the event code pen and for Beam Column Yield events, a rectangle is drawn at every joint which yields. The size of the rectangle is proportional to the joints yield surface (up to three sizes are posiible). The pen selection for each event code can be customized using the Inelastic Events Pen Selection (section 7.x) and the Pen Attributes (section 7.3) dialogue boxes. Also, when the IR Mode flag is set, the mouse highlight feature is disabled and query selected members are drawn using the appropriate event code pens.

5.3 Structural Objects Representation

Structural objects are represented by different images. These images change depending on the display mode of CapView.

- If the Double Line Drawing flag is set, nodes are not displayed. Otherwise, nodes are drawn as fat dots on the display window which, when the nodes are highlighted, are replaced by squares (Figure 5-11).
- If the Double Line Drawing flag is set, members are drawn as filled rectangles. Otherwise, they are drawn as lines (Figures 5-10 and 5-11).
- If the Nodal Loads Shown flag is set, nodal loads are displayed as small sticks drawn at an angle (Figure 5-11). Otherwise they are not shown. (Note: the length and direction of the sticks are not proportional to the load magnitude or direction). The Nodal Loads Shown flag is controlled in the Display Defaults dialogue box from the Options menu (section 7.1).
- PSAS elements and shim elements are displayed as small sticks with a kink in them. The tip elements are drawn in a generally vertical direction, the others in a generally horizontal direction.

5.4 View Controls

CapView provides the user the capability of rotating, zooming and panning (scrolling) a displayed structure. CapView has six buttons with rotation icons at the bottom of the main window (Figure 5-11). The first and second buttons rotate the structure around a horizontal axis in the plane of the screen. The third and fourth buttons rotate the structure around a vertical axis in the plane of the screen. The fifth and sixth buttons rotate the structure around an axis

perpendicular to the screen.

When one of these buttons is pressed, the structure is rotated by a fixed angle around the appropriate axis. The magnitude of this angle can be customized using the the Display Projection dialogue box from the Options menu (section 7.2).

Note that as you rotate the structure, the global axes window shows the new orientation of the global axes of the structure. Note also that the rotations are not around the global axes of the structure. They are around the axes of the display window as described above.

For zooming CapView has two buttons, Magnify and Reduce (Figure 5-11). Magnify is the button to the right of the six rotation buttons and has a big platform icon. Reduce is the button to the right of the Magnify button and has a small platform icon. The user can magnify the structure by pressing the Magnify button and reduce the structure by pressing the Reduce button. The magnification scale used to magnify and reduce structures can be customized using the the Display Projection dialogue box from the Options menu (section 7.2). In addition the user can use the Rubber-Band Zoom feature to magnify structures (section 5.1.3).

The vertical and horizontal scroll bars can be used to pan the structure vertically and horizontally, respectively (Figure 5-11). Each scroll bar has a thumb (a slide bar). The thumb is used to scroll the displayed structure and is used to give a visual feedback on the percentage of the structure that is visible. When the thumb is in the lower (left side) part of the slide area, it indicates that the upper (right side) of the structure is visible. In effect, the thumb behaves like an elevator counter weight.

Each mouse button invokes a specific action. The left and the right mouse buttons move the thumb of the scroll bar a preset "jump" amount. The left mouse button, when clicked over the vertical (horizontal) scroll bar, moves the displayed structure upwards (to the left) while the right mouse button moves the displayed structure downwards (to the right). The center mouse button, when clicked over the vertical (horizontal) scroll bar, moves the structure so that the top (left edge) of the thumb moves to the location of the mouse cursor. If you keep the center button pressed and move the mouse cursor, the thumb tries to follow the mouse cursor, and in effect the structure can be scrolled "smoothly". If the structure is large and/or the machine is slow, this scrolling will not be smooth.

There are also two buttons one labeled Original View and the other Center View (Figure 5-11). When the Original View button is pressed, CapView displays the structure using a reduced-to-fit scale with the Z-axis pointing up and the X-axis pointing to the right. When the Center View button is pressed, CapView displays the structure such that the median point of the structure is in the center of the view window.

5.5 Structural Objects Information

CapView provides instantaneous information on all displayed structural objects. To get information on any displayed object, simply move the mouse cursor to that object (to highlight it), then click the Left Mouse button. Different dialogue boxes are displayed for different structural objects. In the following subsections an overview of the Node, Generic Member, Group, Generic Material, Nodal Force, PSAS Element, Clay Material, and Sand Material Information dialogue boxes is presented.

5.5.1 Node Information

The Node Information dialogue box has information about a node (Figure 5-12). The dialogue box has the following:

- An edit field showing the node name (which may be alphanumeric but is usually an integer).
- A button labeled Slaving Data will display a slaving information dialogue box (Figure 5-13) or an appropriate error message if the node is not slaved.
- A list of 6 check boxes representing the fixed DOFs of the node.
- Three edit fields showing the x, y, and z coordinate location.
- Three edit fields showing the mass in x, y, and z directions.
- Three edit fields showing the rotational inertia in x, y, and z directions.

In addition to the above the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first node in the Node table. Nodes in the Node table are in alphabetical order (by node name).
- Last button. Selecting this button will close the dialogue box and open an new one showing the last node in the Node table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next node in the Node table. If the current node is the last node in the table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous node in the Node table. If the current node is the first node in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open a new one showing the node with the given name. If the Node table does not have a record matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the table so that the stored record will reflect any modification the user applied to the dialogue box.
- Add button. Selecting this button will add a new record to the Node table with the properties displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table. Note that if the node connects to members or there is a nodal load applied to it, it will not be possible to delete it and an appropriate message will be displayed.
- Done button. Selecting this button will close the dialogue box.

5.5.2 Generic Member Information

The Generic Member Information dialogue box has information about a member (Figure 5-14). The dialogue box has the following:

- Two edit fields showing the member name and group. Note that a member is uniquely identified by a string made up by concatinating the group string, a hyphen, and the name string (i.e. two members may have the same name as long as they are in different groups and the users does not intend to move them to the same group). Also, note that the member

name string and group string combined can only be 14 characters long and the group string can only be 7 characters long. The group edit field label is actually a button that, when pressed, will bring the Group Information dialogue box for the specified group (Figure 5-15).

- Three edit fields and buttons corresponding to node I (member start node), node J (member end node), and node K (major axis orientation node). When the any of the buttons is pressed, it will bring up the Node Information dialogue box for the specified node.
- Six edit fields showing the eccentricity of the member from the start and end nodes (I and J) in global coordinates.
- A button and an edit field for the member's material. When the button is pressed, it will bring the Generic Material Information dialogue box for the specified material (Figure 5-16).
- A combo box for the member type.
- If the member type is linear beam (Figure 5-14), then a list of 12 check boxes is shown to allow the user to specify end releases at nodes I and J. If the member type is buckling beam (Figure 5-17), then three edit fields showing the out-of-straightness factor and angle, and the number of damage points, along with a button to show the Members' Damage Parameters dialogue box (Figure 5-18). The latter allows a series of 5 damage points that can be holes or dents, their distances from node I, depths, lengths, and angles can be customized using a list of two radio buttons and 4 edit fields per damage point.
- A radio button list to select the section type. Available selections are: tube, AISC rolled section and a general section.
  - If the section is tubular, then two edit fields are provided (for the outside diameter and wall thickness) (Figure 5-14).
  - If the section is an AISC rolled section (Figure 5-19), then an edit field is provided for the AISC section ID along with a button to display the AISC Section Information dialogue box for the selected section (Figure 5-20) and a button, labeled "?", that, when selected, will display a scrollable list of AISC sections in the AISC section table (Figure 5-21). Also, there are two edit fields for equivalent diameters for display and wave calculations. Display diameters are used to determine the width of a member in in Double-Line Drawing mode (section 5.2). Wave diameters are used for calculating the wave loads on the members.
  - If the section is general (Figure 5-22), then six edit field are shown for the member areas and inertias. Also, there are two edit fields for equivalent diameters for display and wave calculations.
- A button labeled Analysis Parameters that, when pressed, displays the Member's Anslysis Parameters dialogue box (Figure 5-23). This dialogue box shows the following:
  - A combo box for member buoyancy type. Available selections are:
    - no buoyancy,
    - variable w/internal fluid,
    - variable w/o internal fluid,
    - invariable w/internal fluid, and
    - invariable w/o internal fluid.

- A check box to request saving analysis results.
- A check box to include geometric stiffness.
- An edit field to specify initial axial force (tension is positive).
- Six edit fields showing: % decrease in strength for simulating damage in struts, marine growth, drag coefficient, tangential drag coefficient, inertia coefficient, and added mass coefficient. And, for the last five edit fields, there are five check boxes that allow the user to toggle between local values and global values (defined in the Default Anslysis Parameters dialogue box selectable from the the Model menu (section 8.6)). If the box is checked, then whatever value is shown on screen is used regardless of any changes made to the corresponding global values. If the box is not checked, then the global values are used.
- If the element type is buckling beam, an edit field will show the minimum number of subelements. Note that if the value is 0, a default value of 7 will be used in the analysis.

- A button labeled Code Check Parameters that, when pressed, displays the Member's Code Check Parameters dialogue box (Figure 5-24). The data in this dialogue box are used during API code checks. This dialogue box shows the following:
  - Two edit fields for the unbraced length in Y and Z member axes.
  - Two edit fields for the buckling factor (K) in Y and Z member axes.
  - Two radio button lists for the Cm factors in Y and Z member axes. The possible choices for a Cm factor are: A, B, and C. If A is selected, then Cm is set to 0.85. If B is selected, the the Cm factor is calculated using the following formula:

    $$Cm = 0.6 - 0.4(M1/M2)$$

    If C is selected, the the Cm factor is calculated using the following formula:

    $$Cm = 1.0 - 0.4(fa/Fpe)$$
  - Edit fields for the load factor, yield stress, and ring spacing.
  - For the buckling factors, Cm factors, load factor, and yield stress, there are four check boxes that allow the user to toggle between local values and global values (defined in the Default Code Check Parameters dialogue box selectable from the the Model menu (section 8.7)). If the box is checked, then whatever value is shown on screen is used regardless of any changes made to the corresponding global values. If the box is not checked, then the global values are used.
  - Six edit fields for the Can/Stub properties. These edit fields describe the outside diameter, wall thickness and length at end cans/stubs at both ends of the member.
  - For elements with buckling beam type, two edit fields will show the can material and two edit fields will show the can joint stiffness. Four buttons display the appropriate information dialogue boxes.

In addition to the above the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first member in the Member table. Members in the Member table are in alphabetical order.

- Last button. Selecting this button will close the dialogue box and open an new one showing the last member in the Member table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next member in the Member table. If the current member is the last member in the table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous member in the Member table. If the current member is the first member in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open an new one showing the member for the given name. If the Member table does not have a record matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the table so that the stored record will reflect any modification the user applied to the dialogue box.
- Add button. Selecting this button will add a new record to the Member table with the properties displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table.
- Done button. Selecting this button will close the dialogue box.

5.5.3 Group Information

The Group Information dialogue box has information about a group (Figure 5-15). The dialogue box has the following:

- An edit field for the group name. Note that group names are sometime shown with a hyphen as the last character. Internally, all group names have a hyphen. However, all group edit fields are intelligent enough to deal with group names with and without the trailing hyphen.
- A combo box for the group element type. Available selections are: non-linear truss, large displacement beam, linear beam, gap friction, linear 4-to-8 quadrilateral, beam wave, Marshall struct, PSAS, 3-d cable spring, shim, and buckling beam. In addition there is is a flag to toggle between generic and non-generic member type. Generic member types of any element type can be changed to any element type in the Generic Member element type list. Generic member types is the norm. Non-generic member types exist only in models imported from a SeaStar input file. The user should not change this flag if there are members in the group.
- Edit fields for the element count, first element number, number of gauss integration points, initial and tangential stiffness damping factor.
- A check box to include the results in post processing (activated automatically by CAP). If you want to review results for elements in this group after an analysis, this box should be checked. However, doing so will use more diskspace.
- A radio button list to select the status of wave induced fixed end moments. Available selections are: not relevant, exists and don't exist.

In addition to the above, the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first group in the Group table. Groups in the Group table are in alphabetical order.
- Last button. Selecting this button will close the dialogue box and open an new one showing the last group in the Group table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next group in the Group table. If the current group is the last group in the table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous group in the Group table. If the current group is the first group in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open an new one showing the group with the given name. If the Group table does not have a record matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the table so that the stored record will reflect any modification the user applied to the dialogue box.
- Add button. Selecting this button will add a new record to the Group table with the properties displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table.
- Done button. Selecting this button will close the dialogue box.

5.5.4 Generic Material Information

The Generic Material Information dialogue box has information about a material (Figure 5-16). The dialogue box has the following:

- Edit fields for the material name, yield stress (in tension if nonlinear truss), Young's modulus, weight density, and Poisson's ratio.
- A check box for beam column facture and an edit field for the fracture strain.
- For nonlinear trusses, two edit fields for Ehard/E strain hardening modulus, and yield stress (in compression or elastic buckling stress).
- For Marshall struts, two edit fields for Esoft1/E and Esoft2/E, first and second strain softening moduli.
- Two edit fields for weight density and pressure of internal fluid,
- For beam columns, a set of radio button to select curvature type.
- For beam columns, twelve edit fields for the stiffness softening moduli of axial, torsional and bending stiffnesses.
- For beam columns, nine edit fields for the yield force/moment softening moduli of axial, torsional and bending forces/moments.

In addition to the above the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first material in the Material table. Materials in the Material table are in alphabetical order.

- Last button. Selecting this button will close the dialogue box and open an new one showing the last material in the Material table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next material in the Material table. If the current material is the last material in the table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous material in the Material table. If the current material is the first material in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open an new one showing the material with the given name. If the Material table does not have a record matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the table so that the stored record will reflect any modification the user applied to the dialogue box.
- Add button. Selecting this button will add a new record to the Material table with the properties displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table.
- Done button. Selecting this button will close the dialogue box.

5.5.5 Nodal Force Information

The Nodal Force Information dialogue box has information about a nodal force (Figure 5-25). The dialogue box has the following:

- Edit fields for the load case, load name, and the loaded node.
- Three edit fields for the components of the applied force in global coordinates.
- Three edit fields for the components of the applied moment in global coordinates.

In addition to the above the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first nodal force in the Nodal Force table. Nodal forces in the Nodal Force table are in alphabetical order.
- Last button. Selecting this button will close the dialogue box and open an new one showing the last nodal force in the Nodal Force table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next nodal force in the Nodal Force table. If the current nodal force is the last nodal force in the table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous nodal force in the Nodal Force table. If the current nodal force is the first nodal force in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open an new one showing the nodal force with the given name. If the Nodal Force table does not have a record matching the given name, an appropriate

message will be displayed.

- Replace button. Selecting this button will modify the table so that the stored record will reflect any modification the user applied to the dialogue box.
- Add button. Selecting this button will add a new record to the Nodal Force table with the properties displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table.
- Done button. Selecting this button will close the dialogue box.

5.5.6 PSAS Element Information

The PSAS Element Information dialogue box has information about a (soil spring) PSAS element (Figure 5-26). The dialogue box has the following:

- An edit field for the element name.
- An edit field for the group name.
- An edit field for free-field node (node I). There is also a button to get information on this node.
- An edit field for pile node (node J). There is also a button to get information on this node.
- An edit field for axial direction node (node K). This can be any node along the pile axis except the I node. There is also a button to get information on this node.
- Two edit fields for the effective pile diameter for lateral and axial behavior.
- An edit field for the tributary (effective) pile length to be used in calculating the total spring force. If zero is entered, a default value will be calculated by SeaStar as half the distance to the element above plus half the distance to the element below. If this element is the top (bottom) element, the default value will be the distance half way to the next (previous) element below (above). This value is ignored if the element is a tip element.
- A check box to toggle between tip element and non-tip elements.
- A check box to request saving analysis results.

In addition to the above, the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first PSAS in the PSAS table. PSASs in the PSAS table are in alphabetical order.
- Last button. Selecting this button will close the dialogue box and open an new one showing the last PSAS in the PSAS table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next PSAS in the PSAS table. If the current PSAS is the last PSAS in the table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous PSAS in the PSAS table. If the current PSAS is the first PSAS in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open an new one showing the PSAS with the given name. If the PSAS table does

not have a record matching the given name, an appropriate message will be displayed.

- Replace button. Selecting this button will modify the table so that the stored record will reflect any modification the user applied to the dialogue box.
- Add button. Selecting this button will add a new record to the PSAS table with the properties displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table.
- Done button. Selecting this button will close the dialogue box.

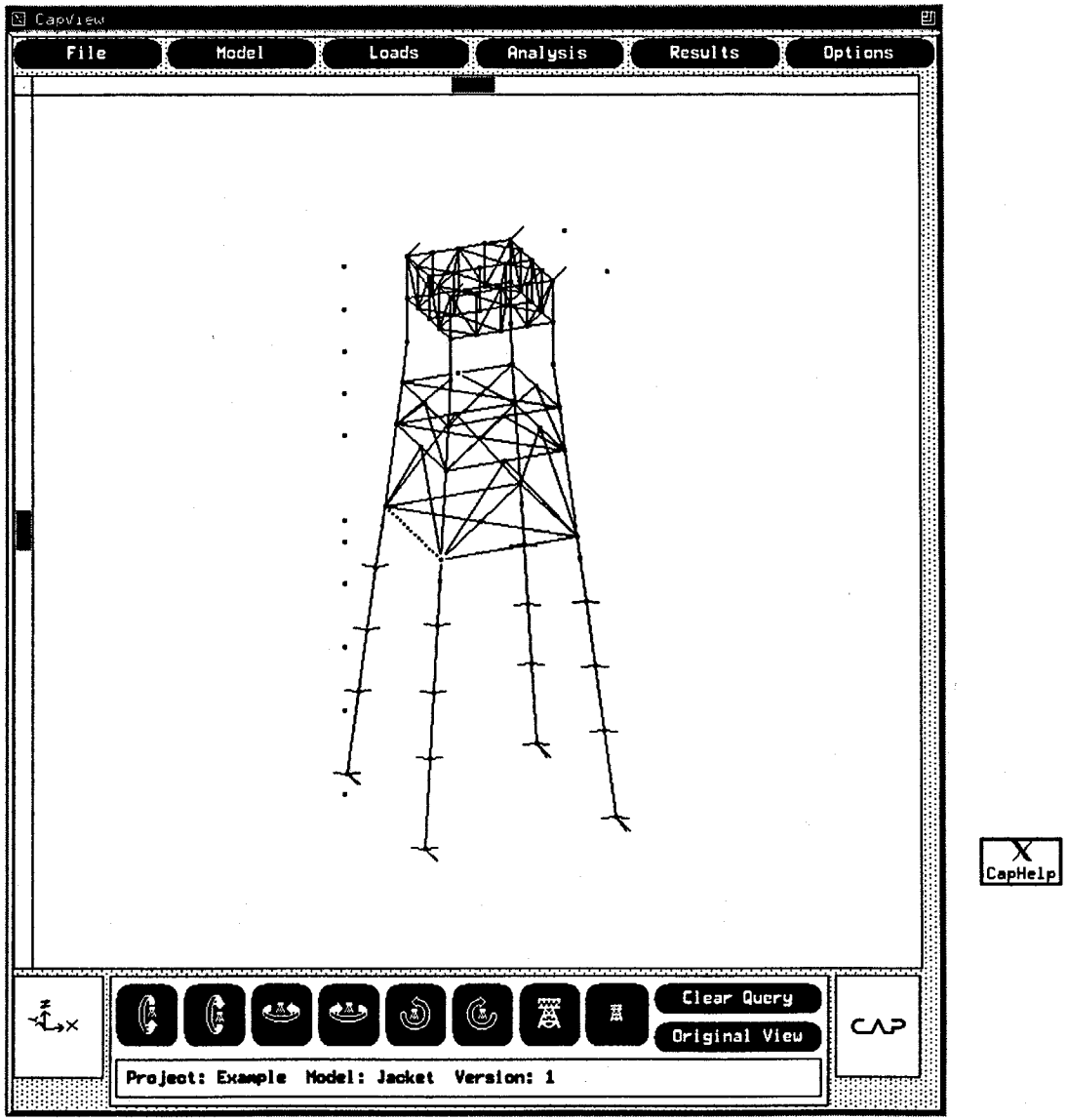

Capview
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CAP
CapHelp

Figure 5-1

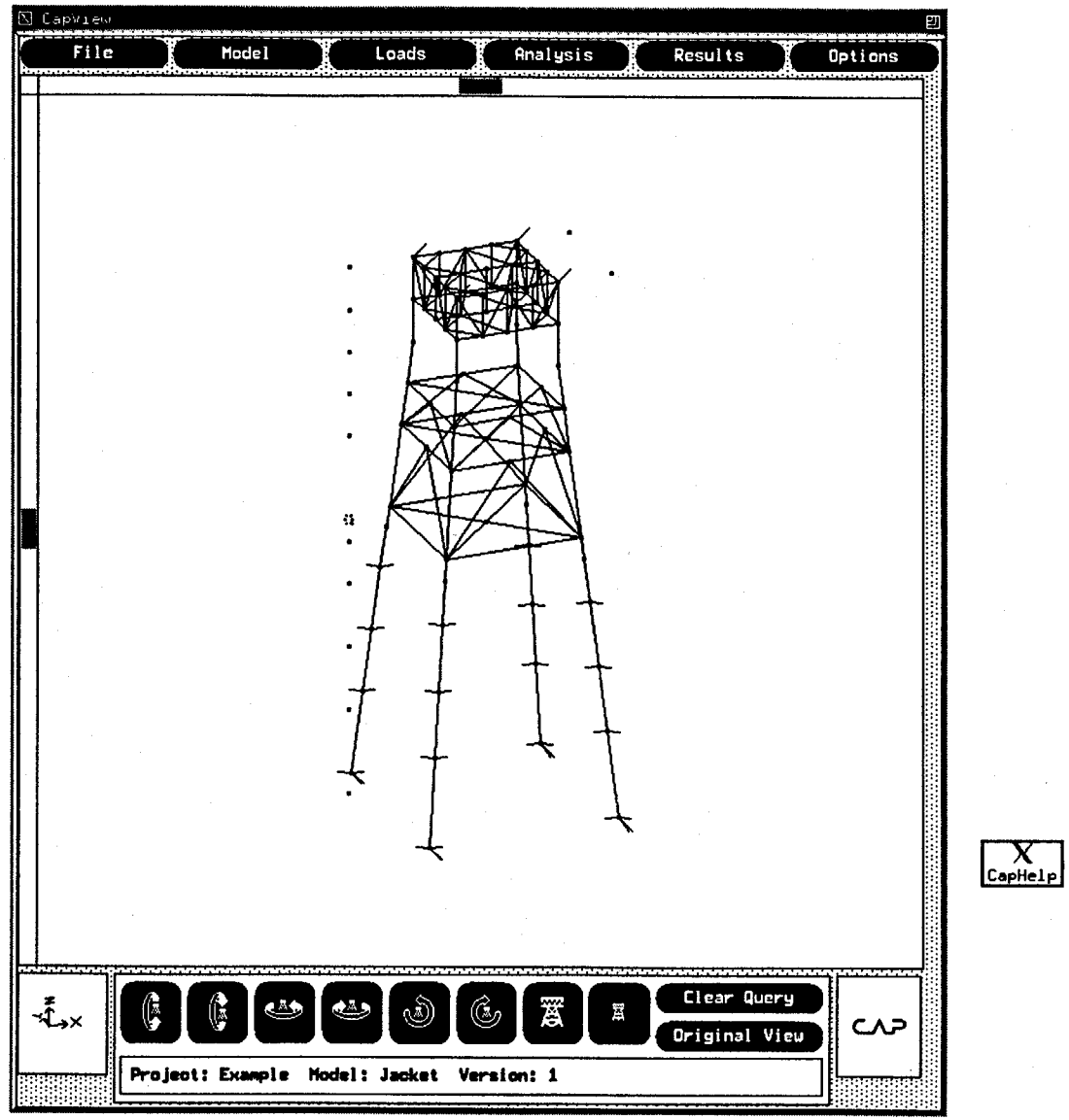

CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
CAP
Project: Example Model: Jacket Version: 1
CapHelp

Figure5-2

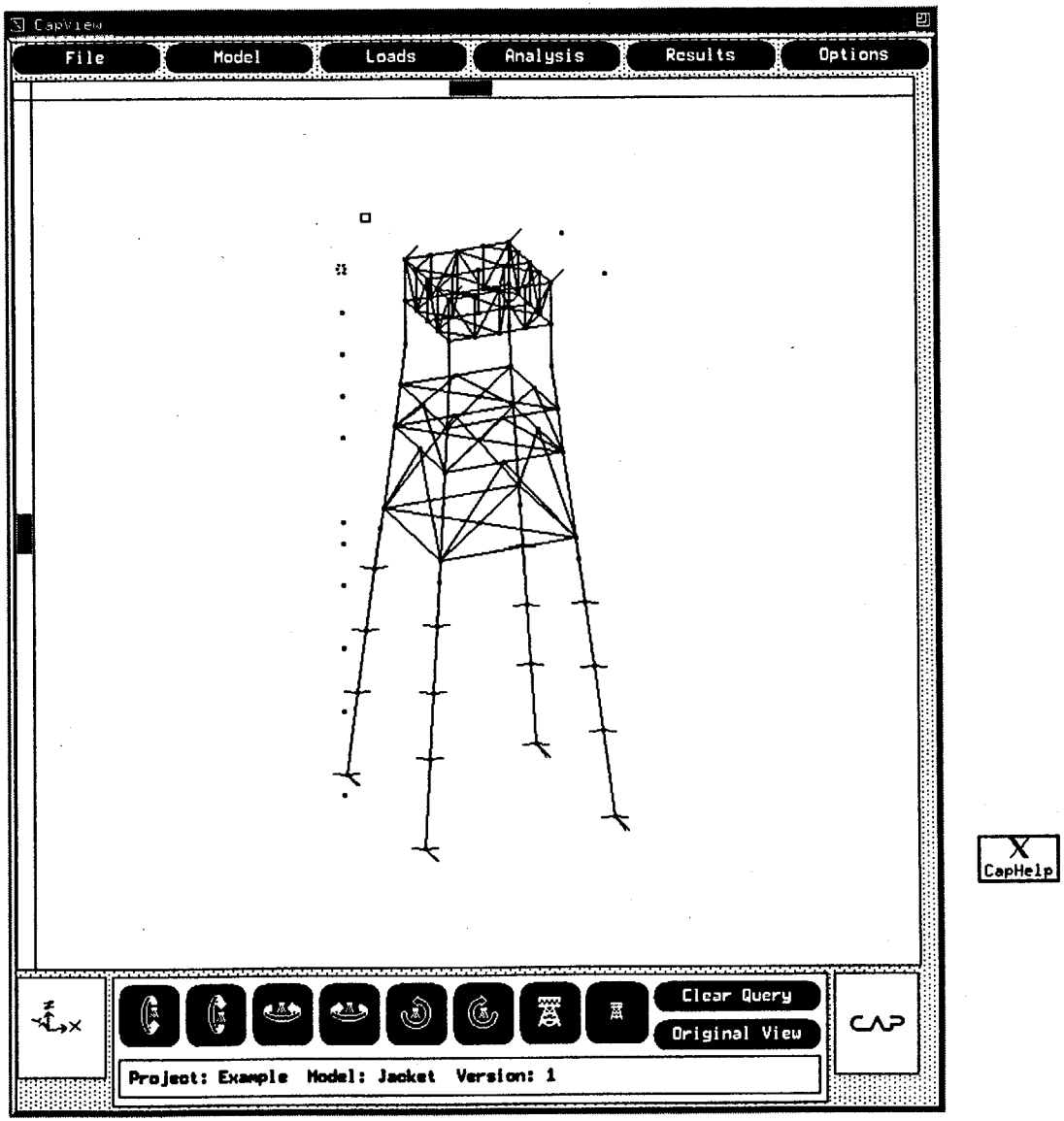

Capview
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-3

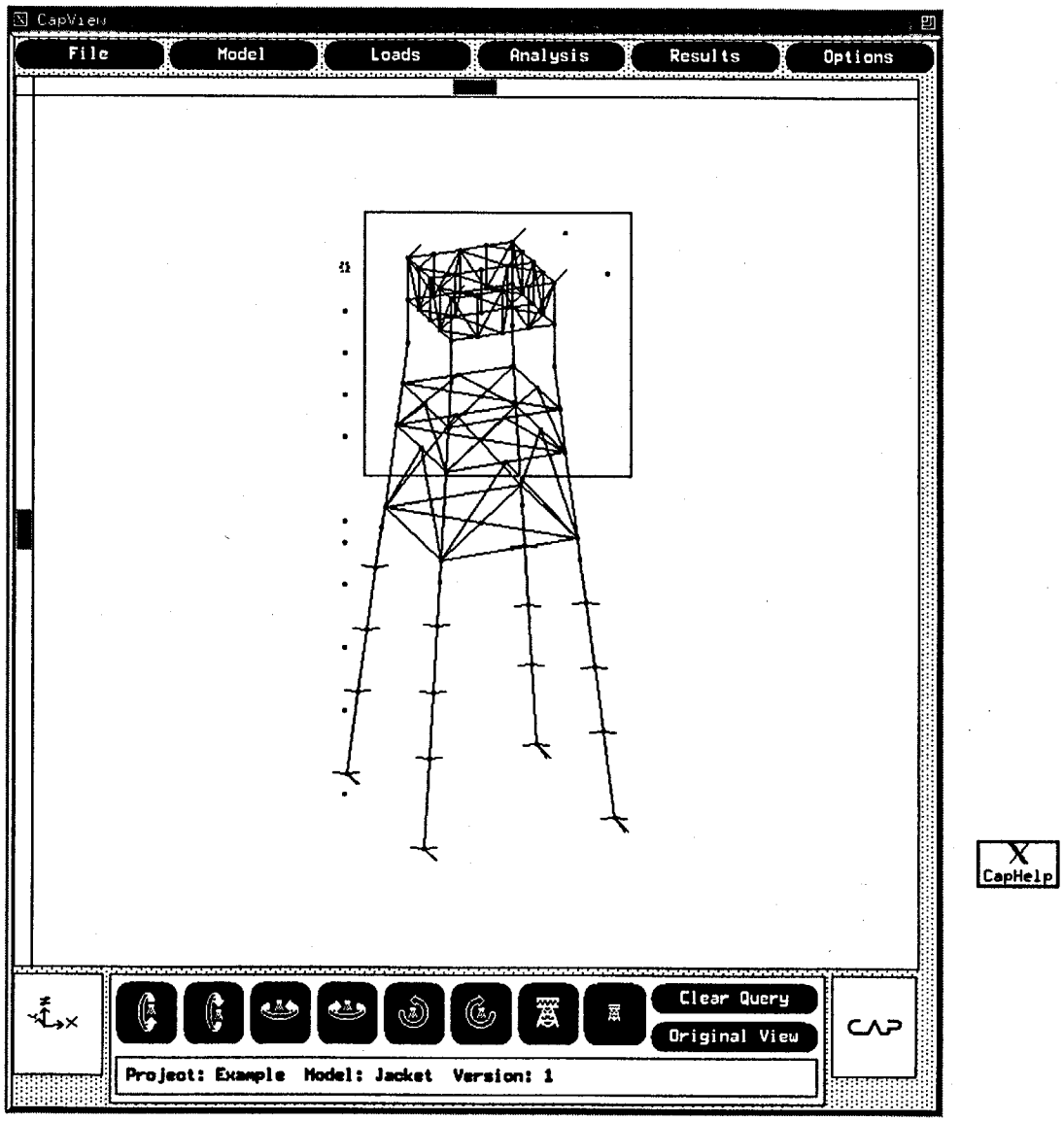

CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-4

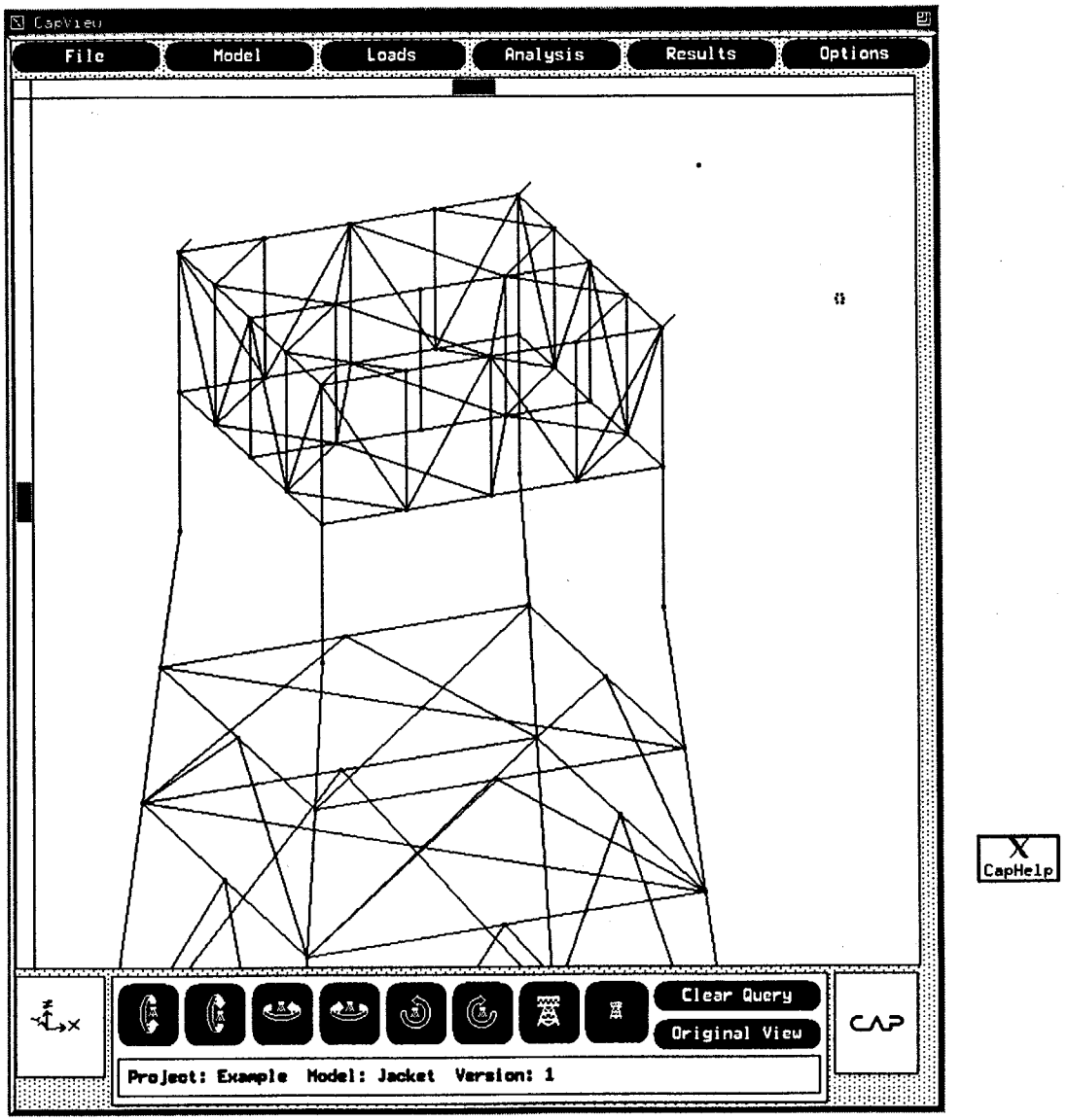

CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-5

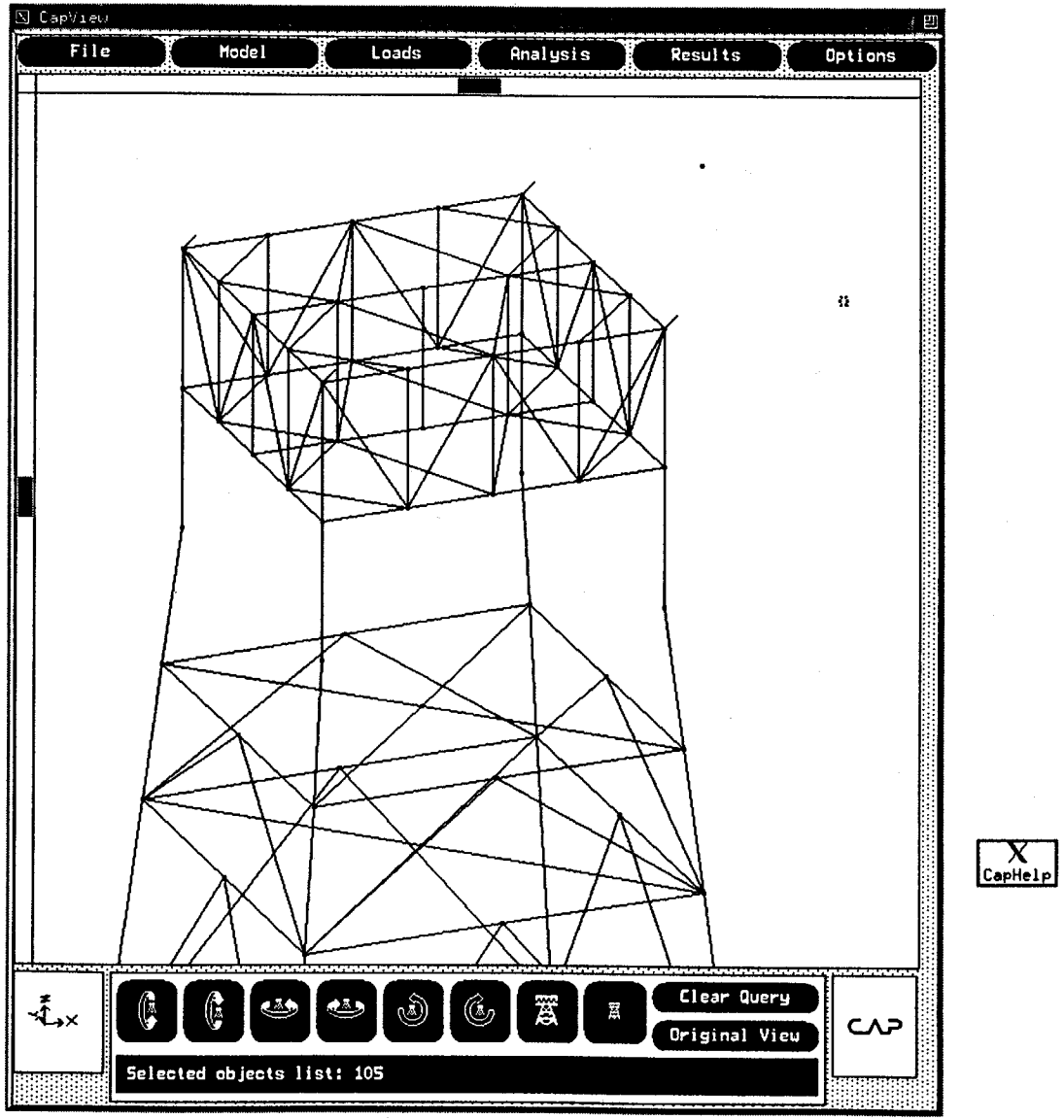

CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Selected objects list: 105
CAP
CapHelp

Figure 5-6

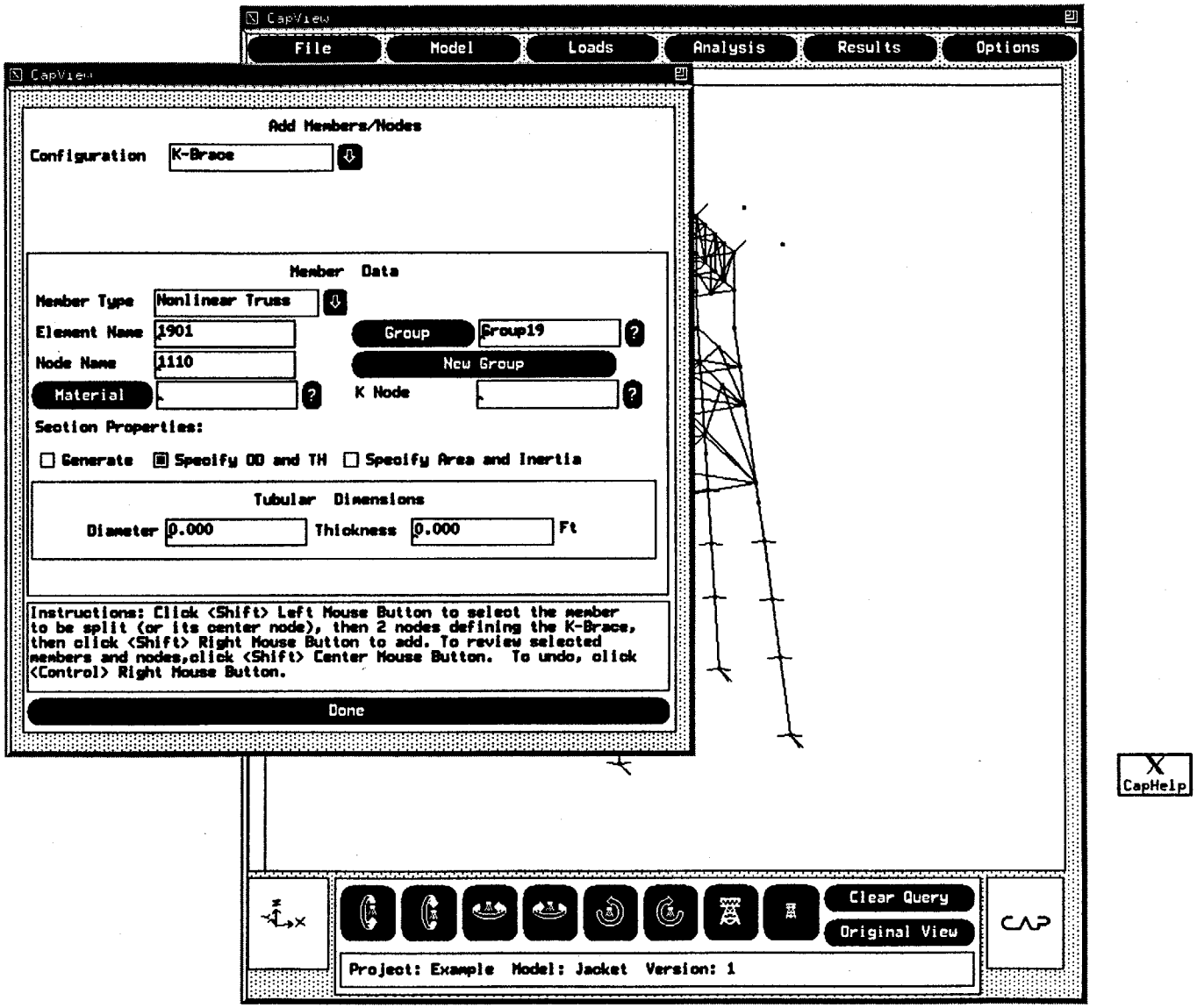

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
K-Brace
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Tubular Dimensions
Diameter
0.000
Thickness
0.000
Ft
Instructions: Click <Shift> Left Mouse Button to select the member to be split (or its center node), then 2 nodes defining the K-Brace, then click <Shift> Right Mouse Button to add. To review selected members and nodes,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-7

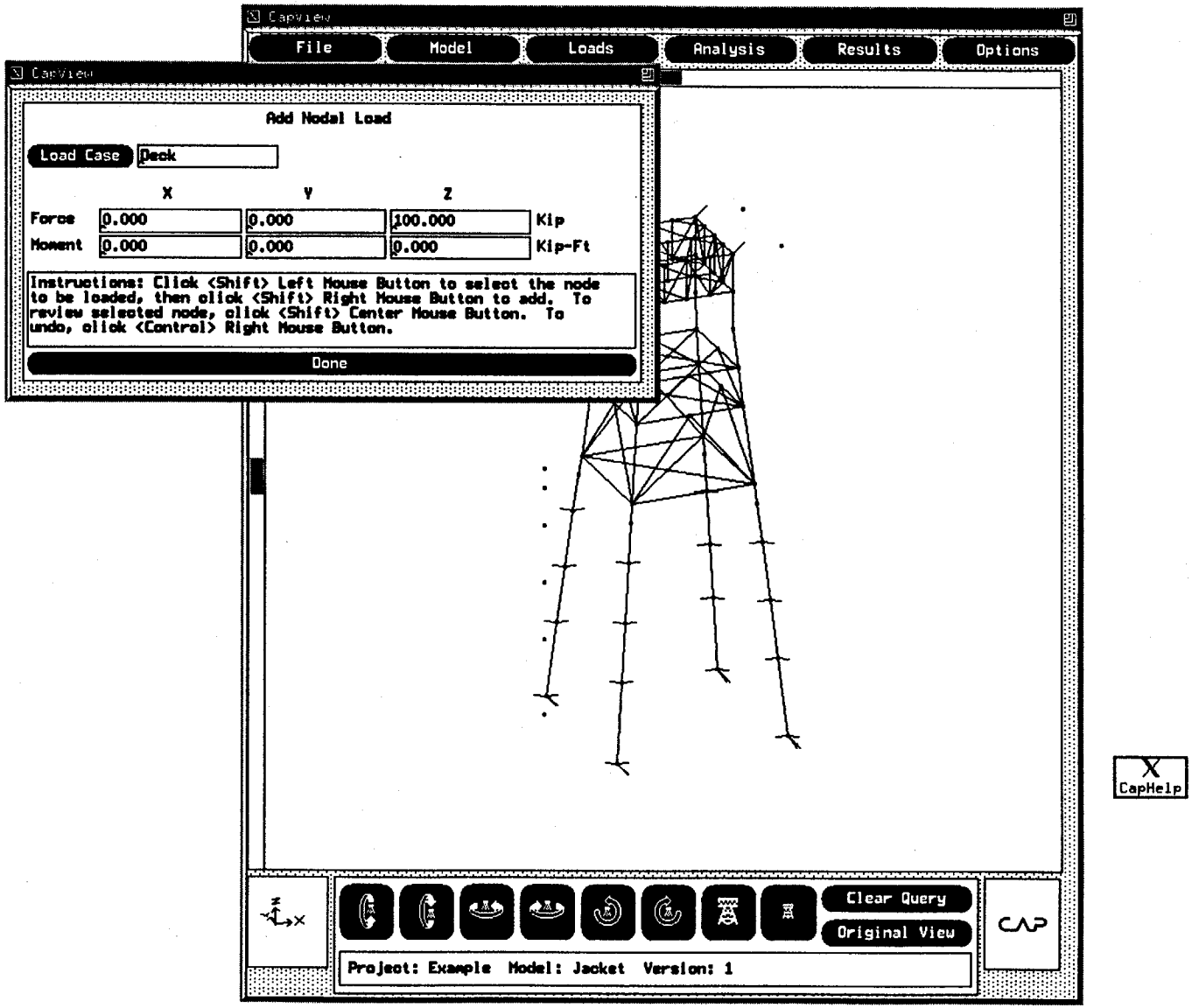

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Nodal Load
Load Case
Deck
X
Y
Z
Force
0.000
0.000
100.000
Kip
Moment
0.000
0.000
0.000
Kip-Ft
Instructions: Click <Shift> Left Mouse Button to select the node to be loaded, then click <Shift> Right Mouse Button to add. To review selected node, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-8

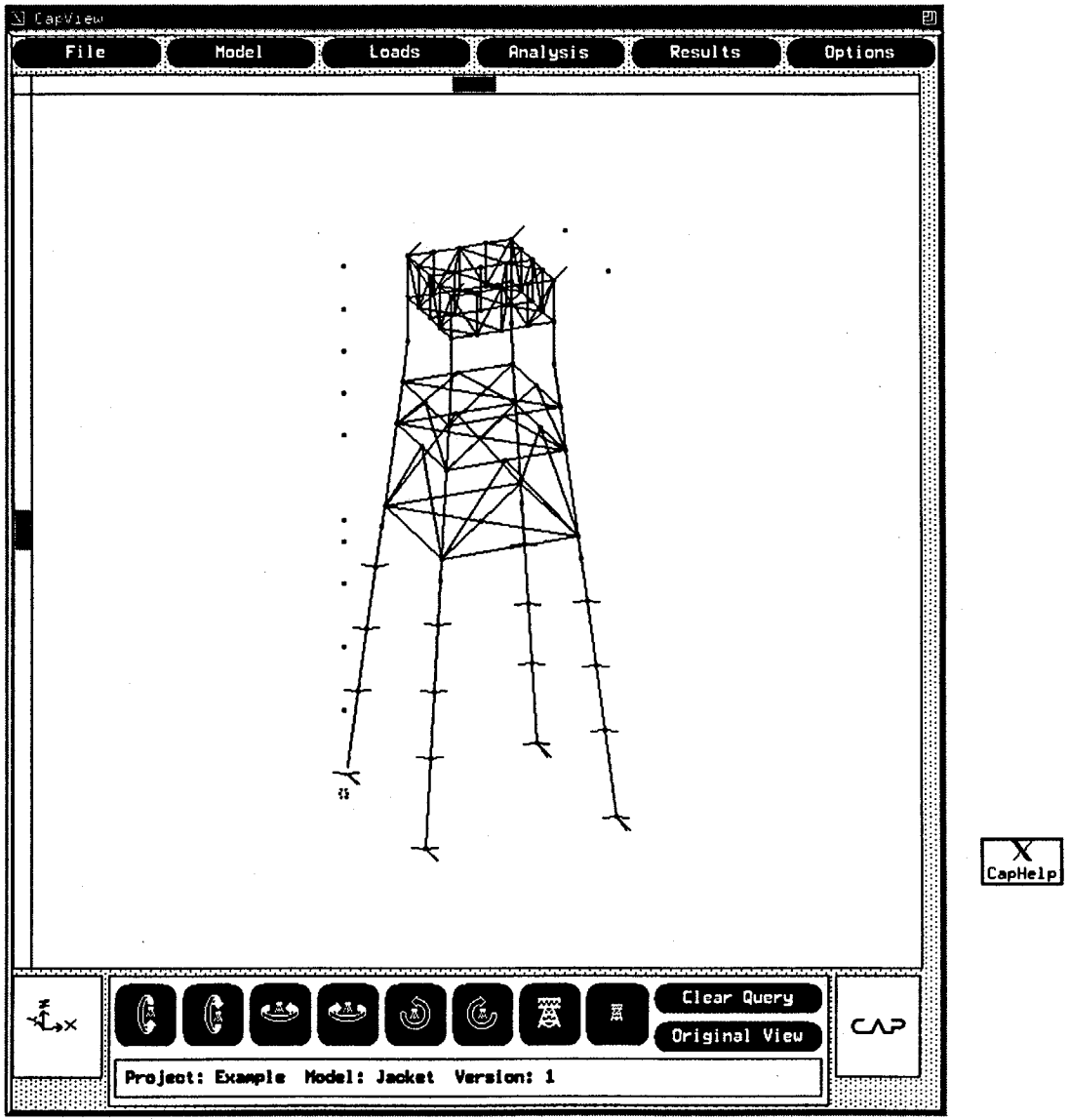

CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-9

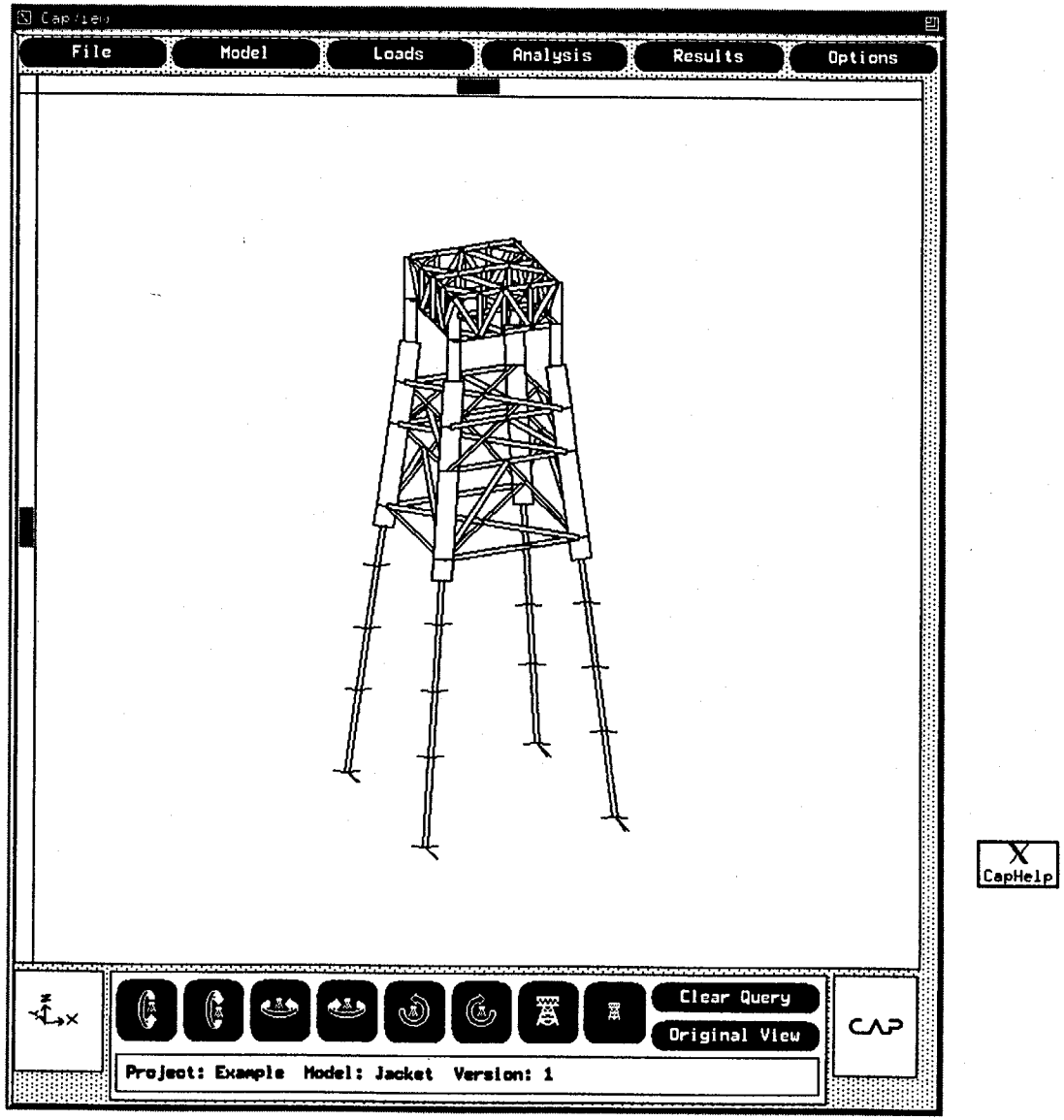

File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
CAP
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-10

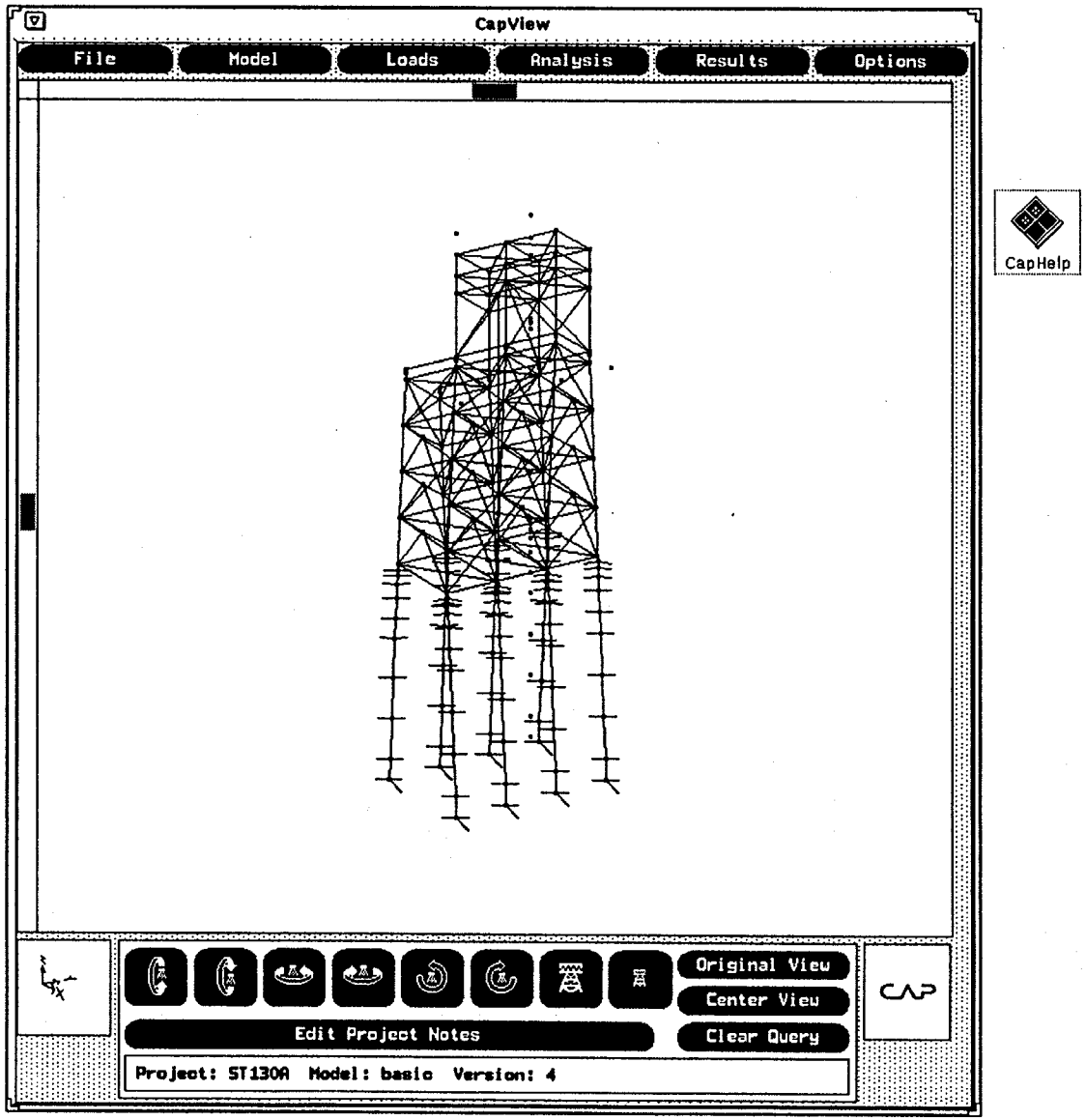

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 5-11

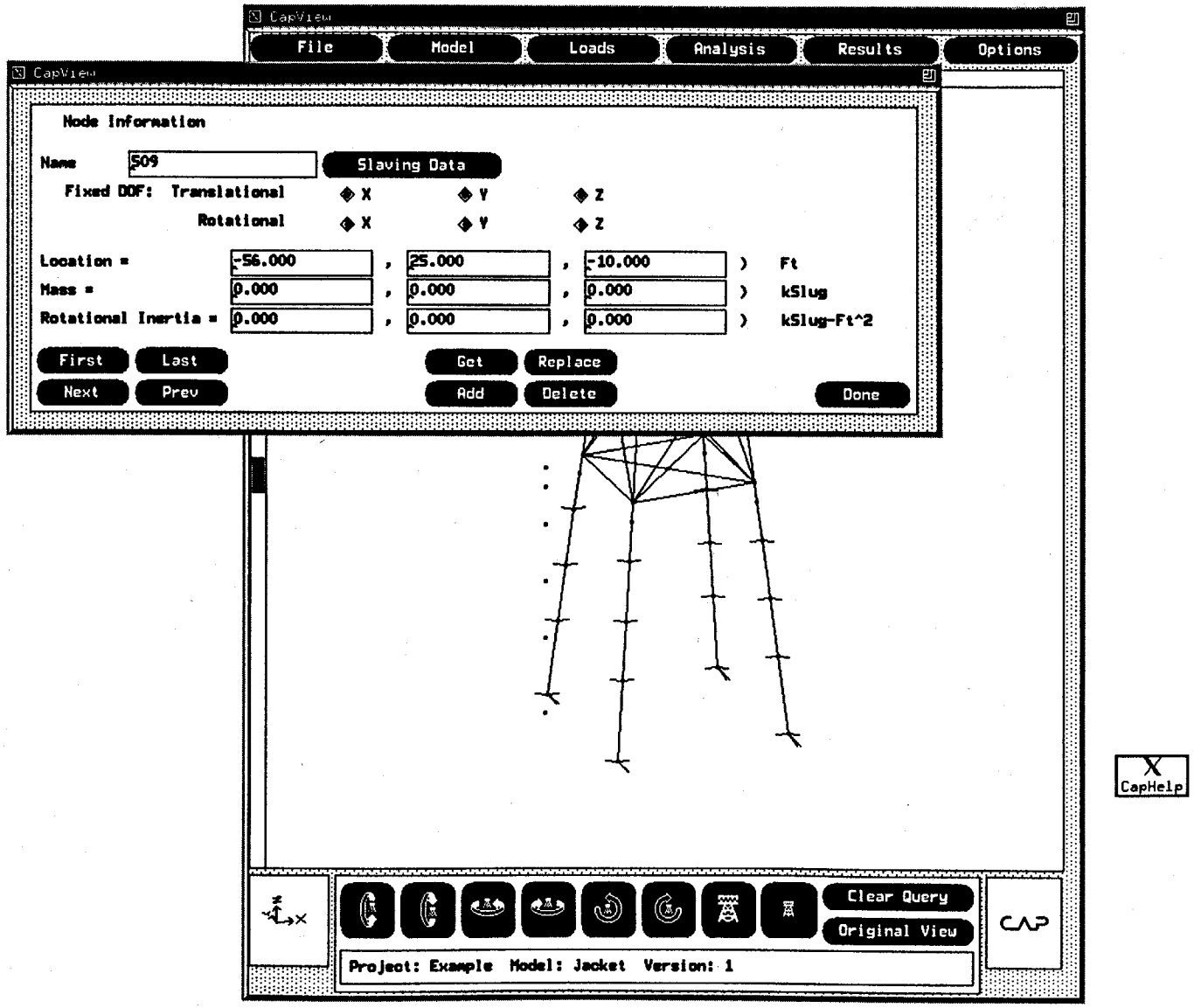
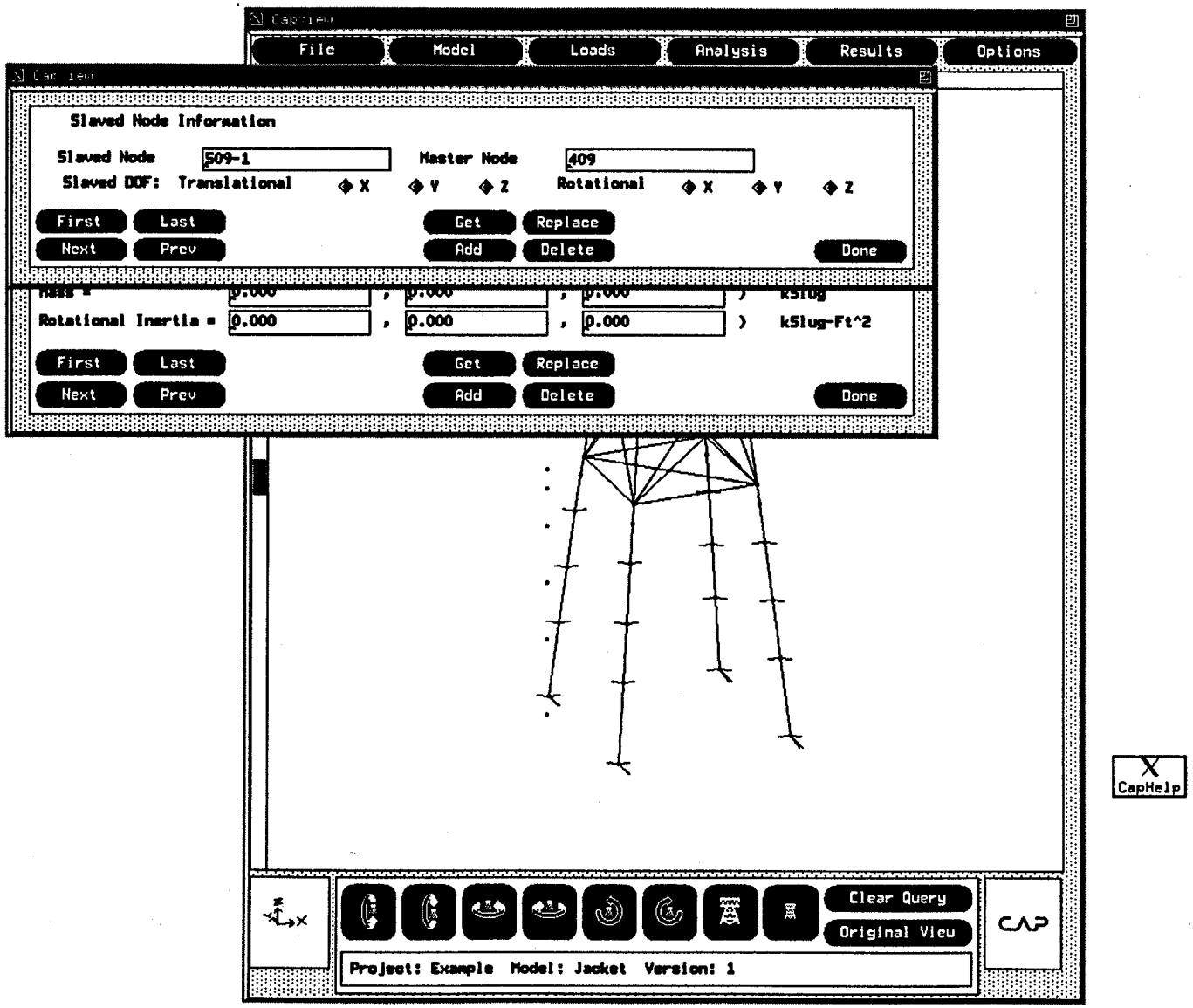

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Node Information
Name
509
Slaving Data
Fixed DOF: Translational
X
Y
Z
Rotational
X
Y
Z
Location =
-56.000
25.000
-10.000
Ft
Mass =
0.000
0.000
0.000
kSlug
Rotational Inertia =
0.000
0.000
0.000
kSlug-Ft^2
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-12



X
CapHelp

Figure 5-13

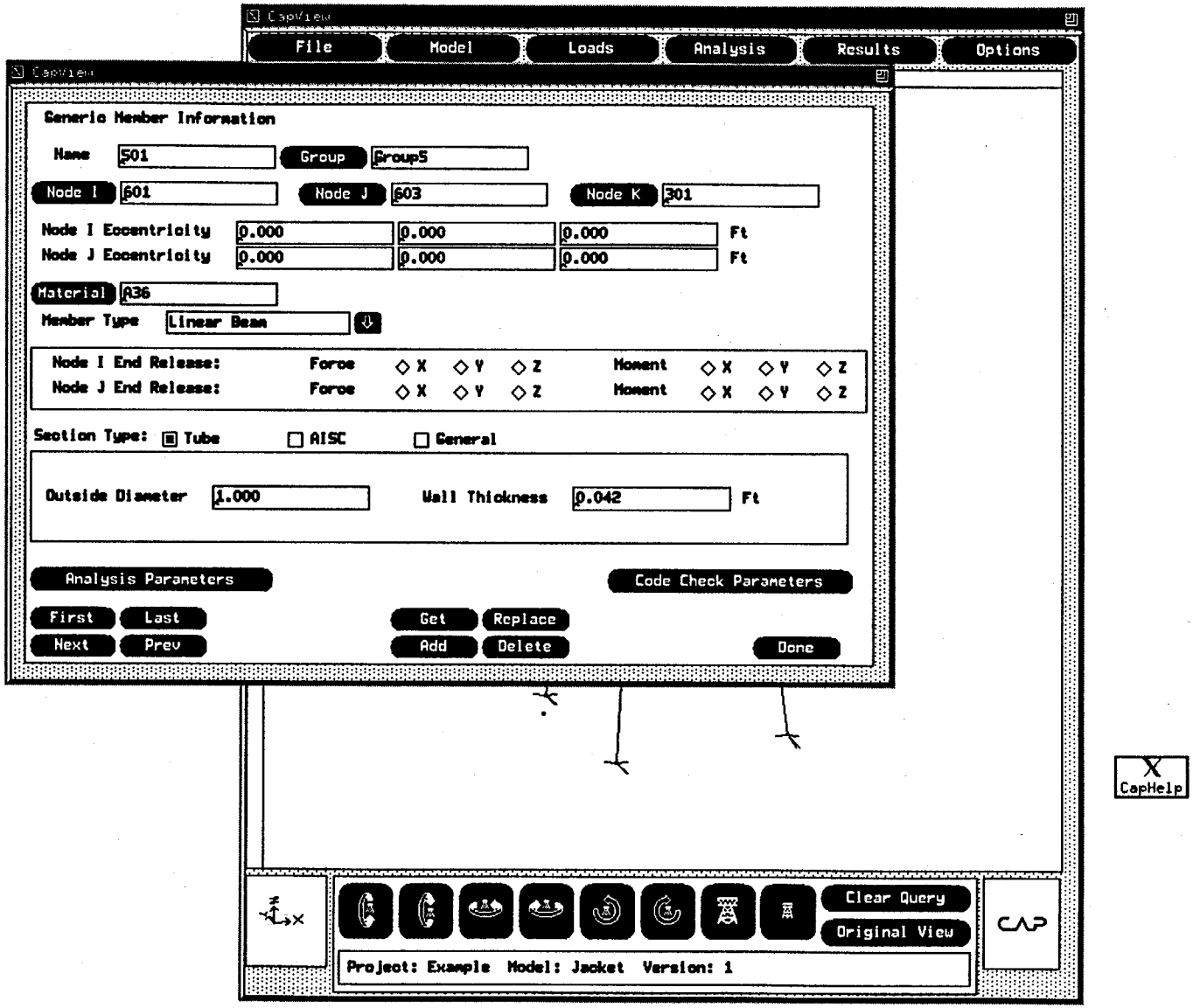
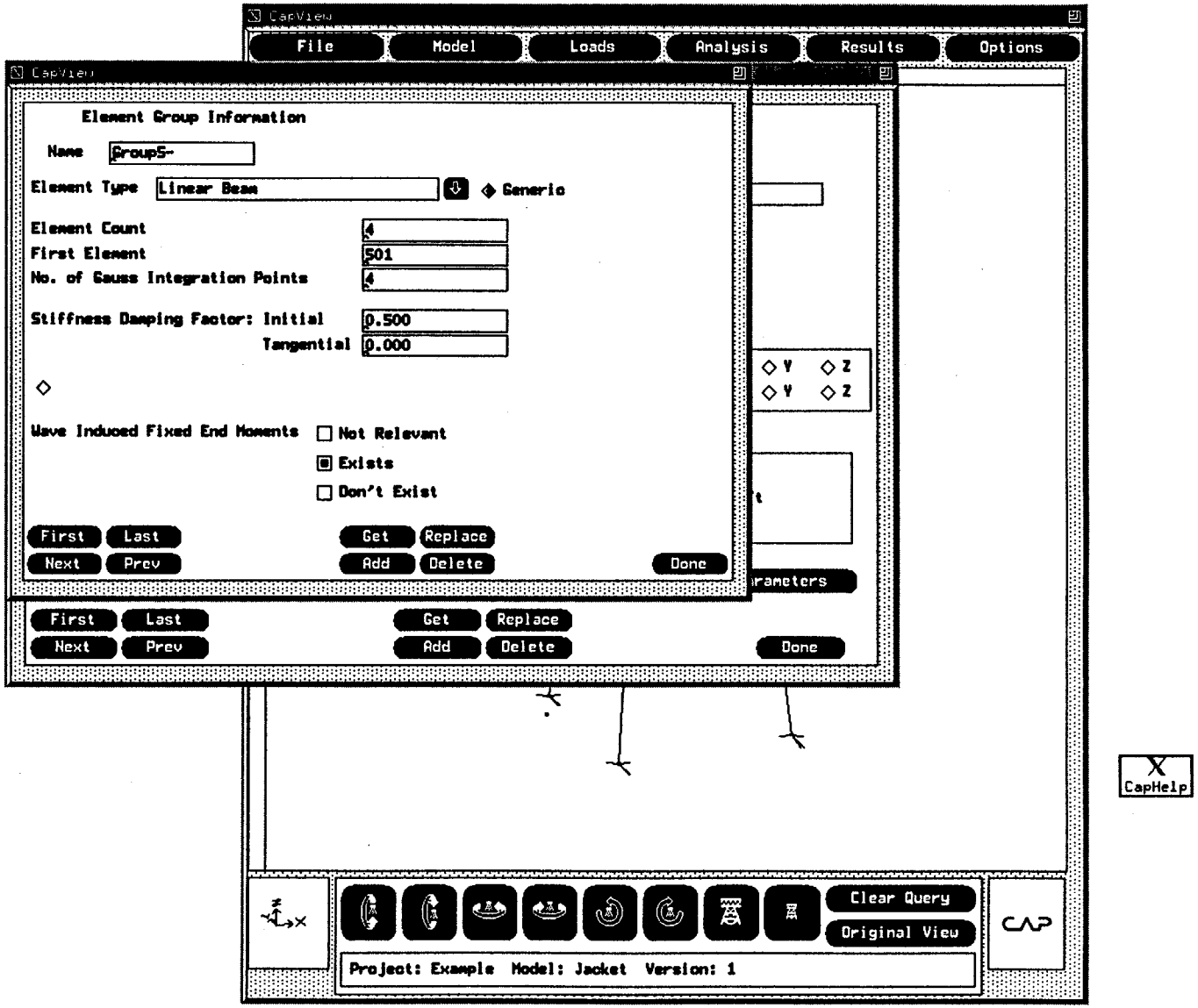

CapView
File
Model
Loads
Analysis
Results
Options
Generic Member Information
Name
501
Group
Group5
Node I
601
Node J
603
Node K
301
Node I Eccentricity
0.000
0.000
0.000
Ft
Node J Eccentricity
0.000
0.000
0.000
Ft
Material
A36
Member Type
Linear Beam
Node I End Release:
Force
X
Y
Z
Moment
X
Y
Z
Node J End Release:
Force
X
Y
Z
Moment
X
Y
Z
Section Type:
Tube
AISC
General
Outside Diameter
1.000
Wall Thickness
0.042
Ft
Analysis Parameters
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-14



CapHelp

Figure 5-15

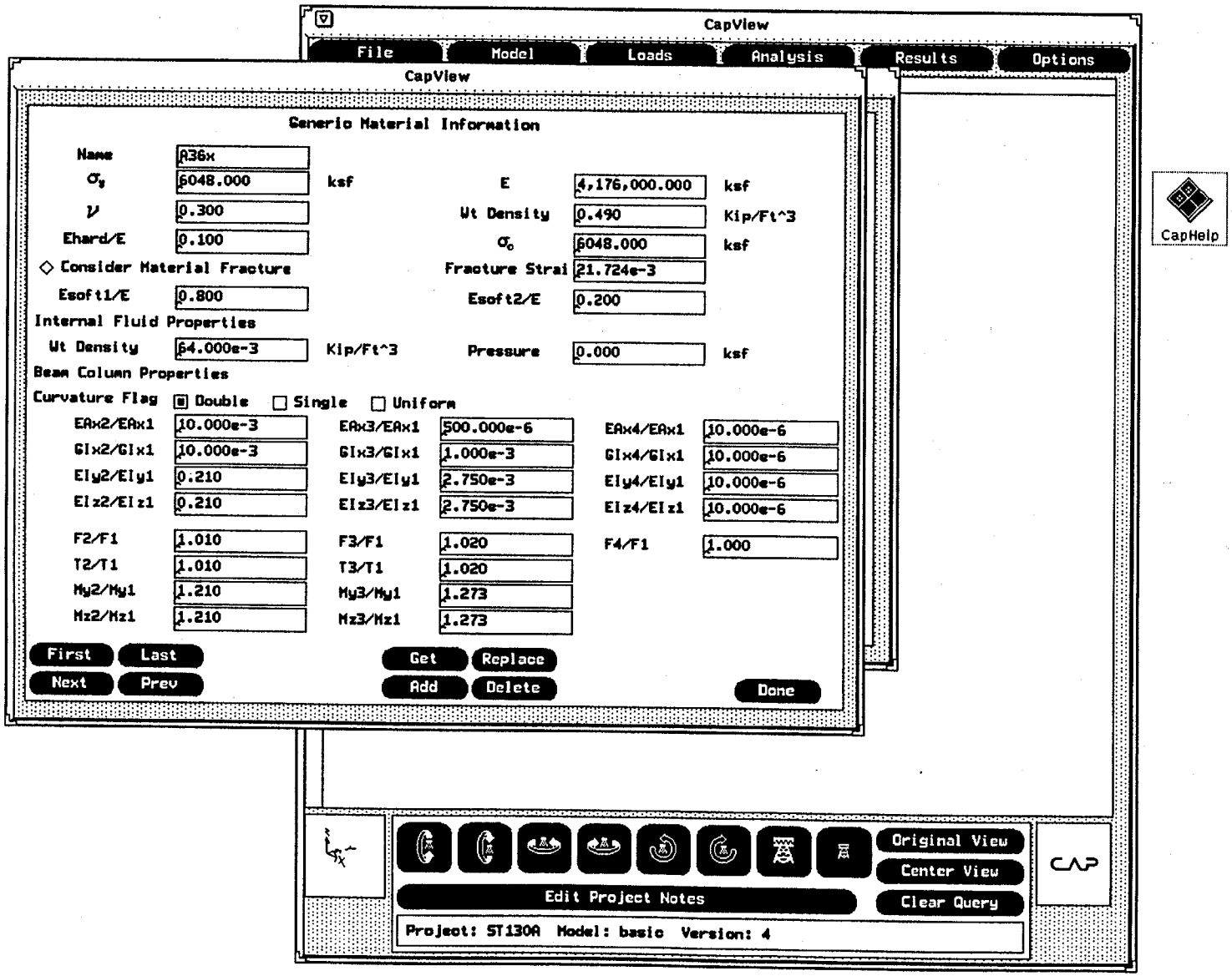

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Generic Material Information
Name
A36x
σy
6048.000
ksf
E
4,176,000.000
ksf
ν
0.300
Wt Density
0.490
Kip/Ft^3
Ehard/E
0.100
σo
6048.000
ksf
Consider Material Fracture
Fracture Strai
21.724e-3
Esoft1/E
0.800
Esoft2/E
0.200
Internal Fluid Properties
Wt Density
64.000e-3
Kip/Ft^3
Pressure
0.000
ksf
Beam Column Properties
Curvature Flag
Double
Single
Uniform
EAx2/EAx1
10.000e-3
EAx3/EAx1
500.000e-6
EAx4/EAx1
10.000e-6
GIx2/GIx1
10.000e-3
GIx3/GIx1
1.000e-3
GIx4/GIx1
10.000e-6
EIy2/EIy1
0.210
EIy3/EIy1
2.750e-3
EIy4/EIy1
10.000e-6
EIz2/EIz1
0.210
EIz3/EIz1
2.750e-3
EIz4/EIz1
10.000e-6
F2/F1
1.010
F3/F1
1.020
F4/F1
1.000
T2/T1
1.010
T3/T1
1.020
My2/My1
1.210
My3/My1
1.273
Mz2/Mz1
1.210
Mz3/Mz1
1.273
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
CapHelp
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 5-16

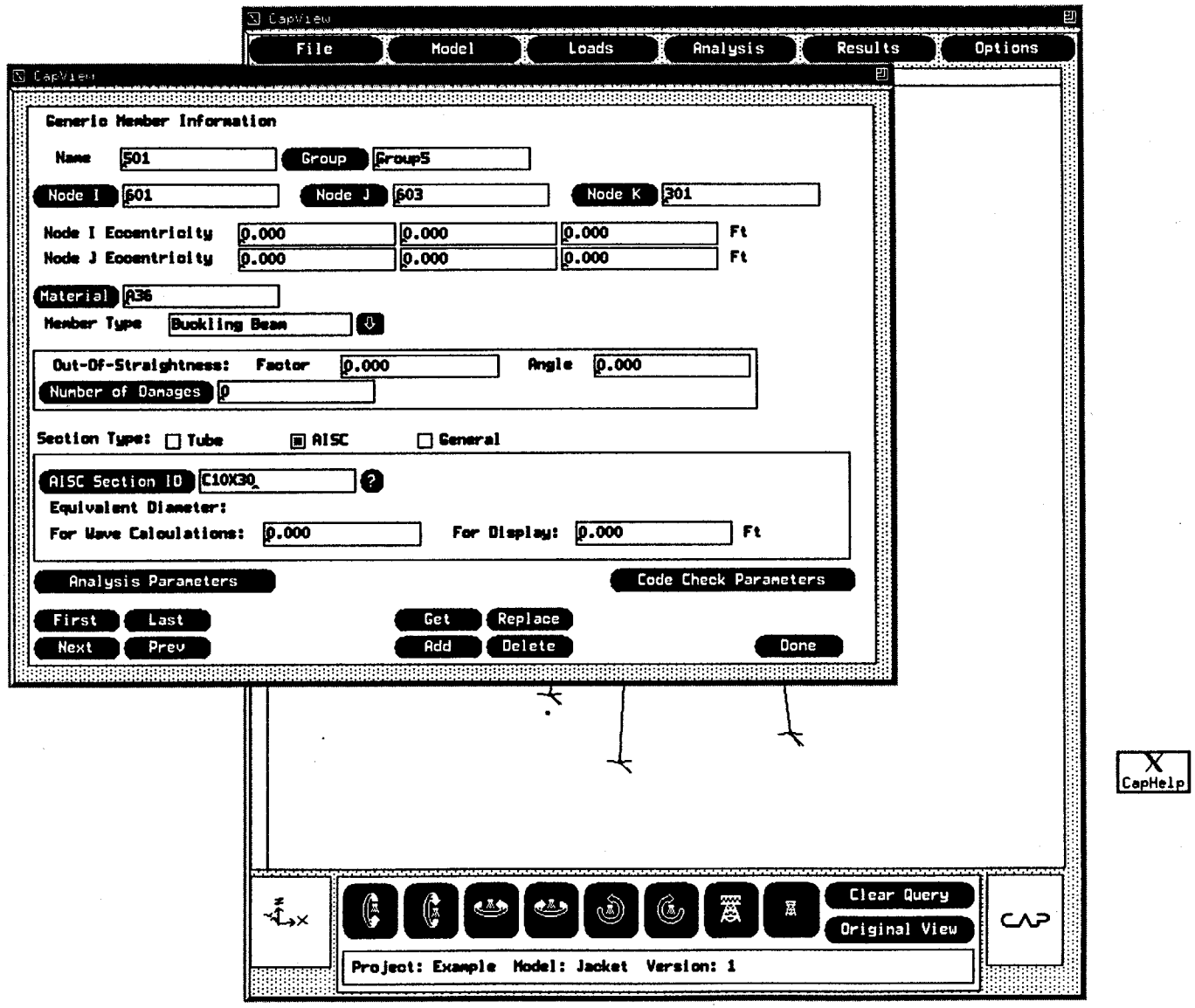

CapView
File
Model
Loads
Analysis
Results
Options
Generic Member Information
Name
501
Group
Group5
Node I
601
Node J
603
Node K
301
Node I Eccentricity
0.000
0.000
0.000
Ft
Node J Eccentricity
0.000
0.000
0.000
Ft
Material
A36
Member Type
Buckling Beam
Out-Of-Straightness:
Factor
0.000
Angle
0.000
Number of Damages
0
Section Type:
Tube
AISC
General
AISC Section ID
C10X30
Equivalent Diameter:
For Wave Calculations:
0.000
For Display:
0.000
Ft
Analysis Parameters
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-17

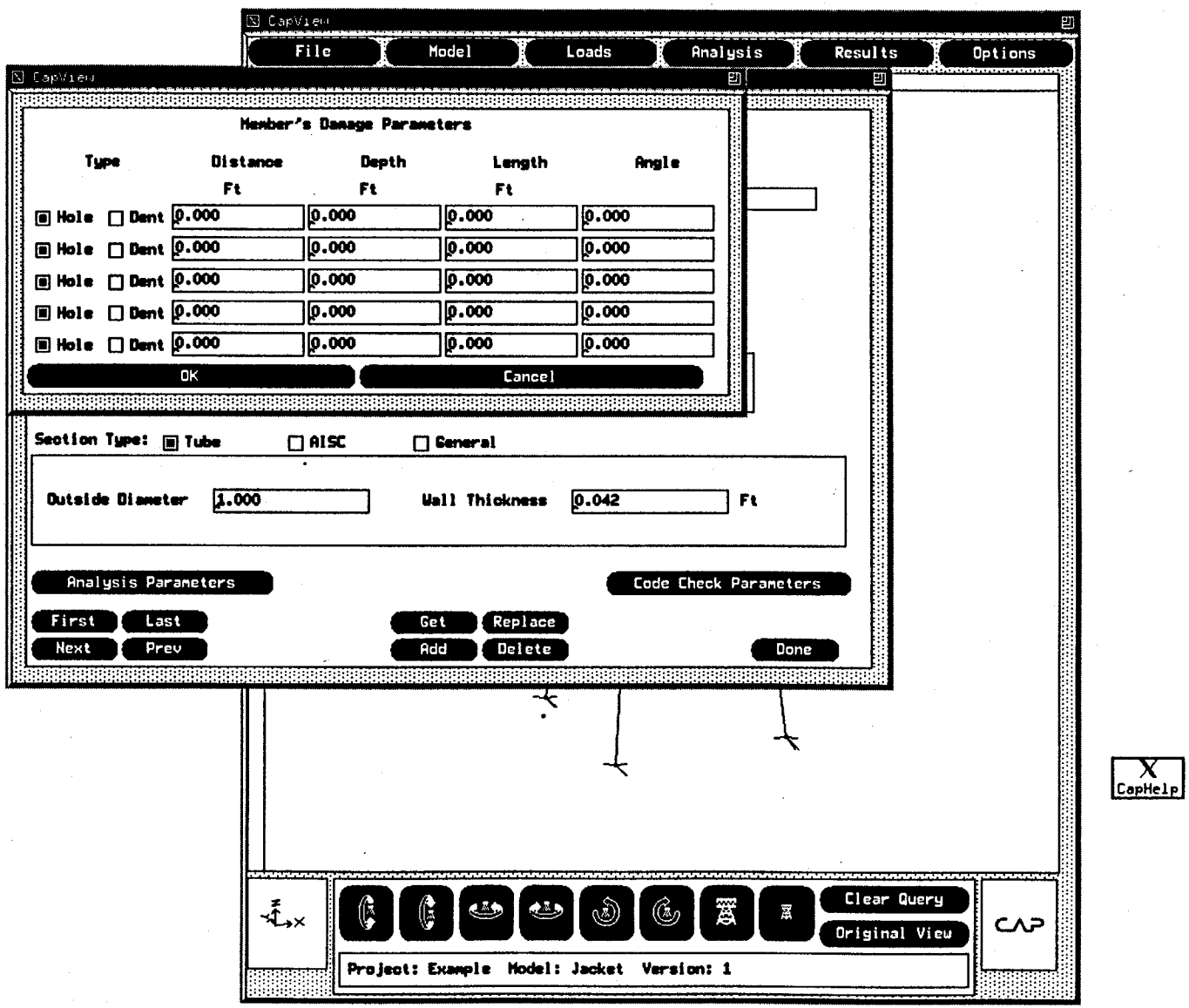

CapView
File
Model
Loads
Analysis
Results
Options
Member's Damage Parameters
Type
Distance
Ft
Depth
Ft
Length
Ft
Angle
Hole
Dent
0.000
OK
Cancel
Section Type:
Tube
AISC
General
Outside Diameter
1.000
Wall Thickness
0.042
Ft
Analysis Parameters
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
CAP
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-18

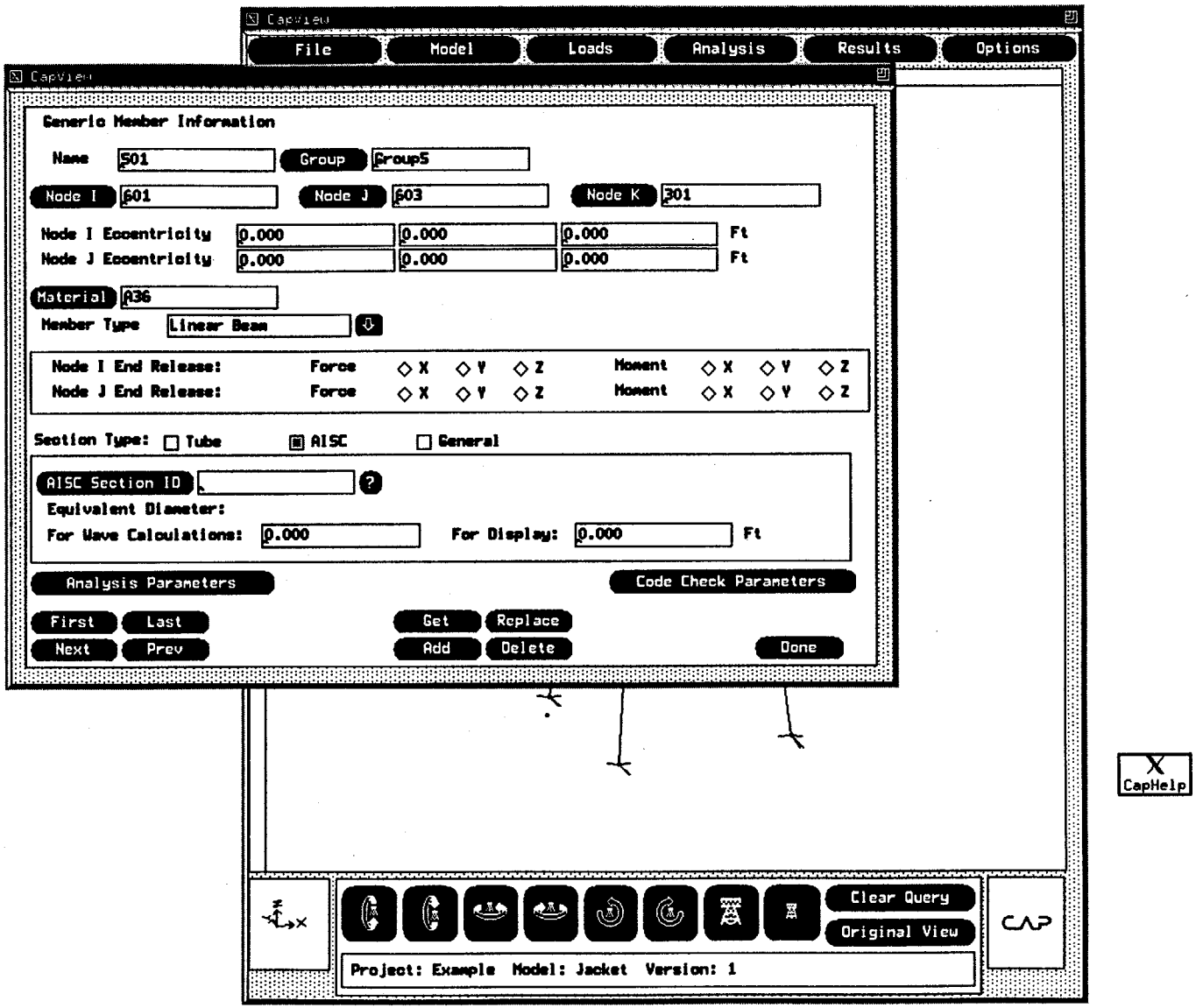

CapView
File
Model
Loads
Analysis
Results
Options
Generic Member Information
Name
501
Group
Group5
Node I
601
Node J
603
Node K
301
Node I Eccentricity
0.000
0.000
0.000
Ft
Node J Eccentricity
0.000
0.000
0.000
Ft
Material
A36
Member Type
Linear Beam
Node I End Release:
Force
X
Y
Z
Moment
X
Y
Z
Node J End Release:
Force
X
Y
Z
Moment
X
Y
Z
Section Type:
Tube
AISC
General
AISC Section ID
Equivalent Diameter:
For Wave Calculations:
0.000
For Display:
0.000
Ft
Analysis Parameters
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-19

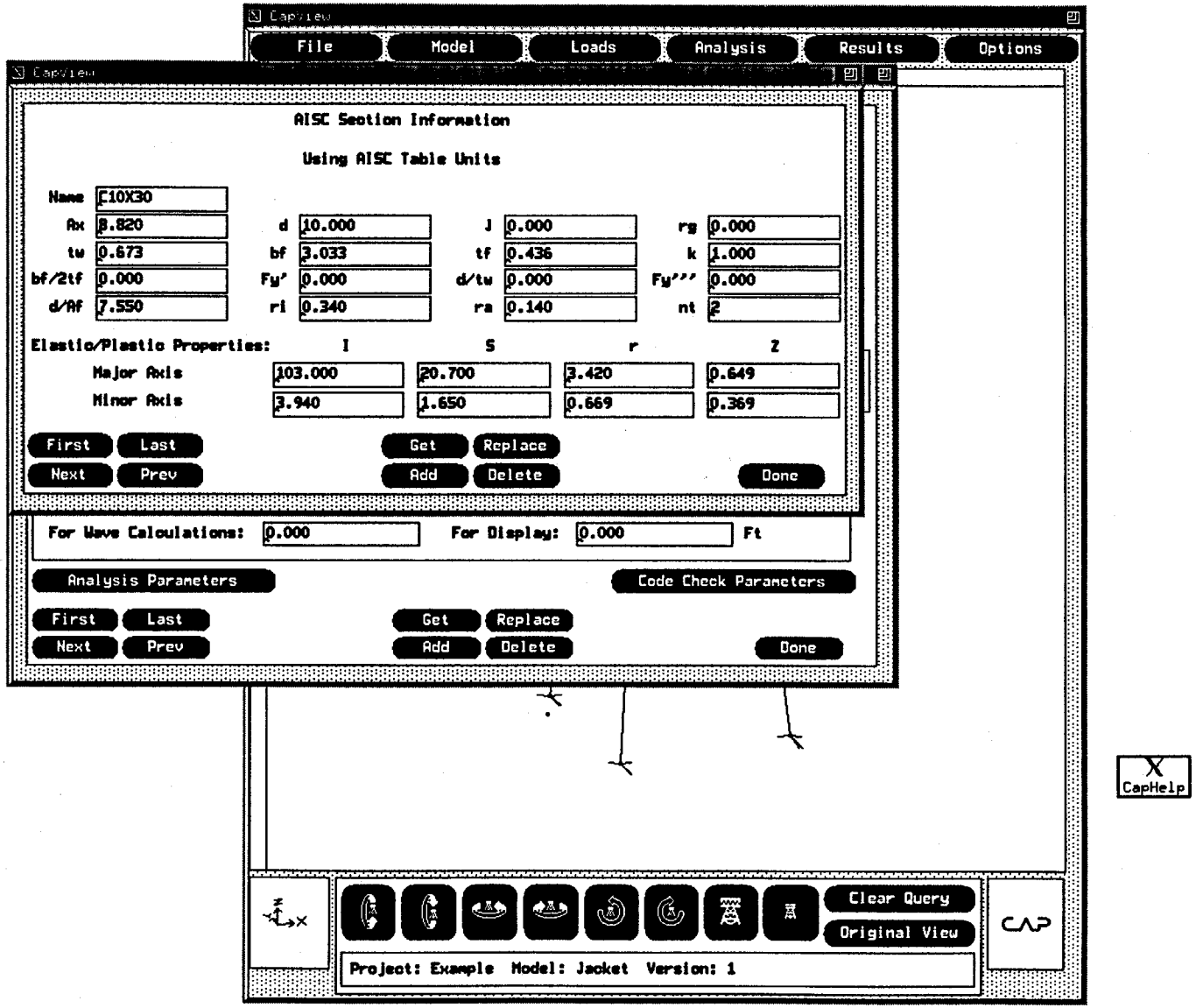

File
Model
Loads
Analysis
Results
Options
AISC Section Information
Using AISC Table Units
Name C10X30
Ax 8.820
d 10.000
J 0.000
rg 0.000
tw 0.673
bf 3.033
tf 0.436
k 1.000
bf/2tf 0.000
Fy' 0.000
d/tw 0.000
Fy''' 0.000
d/Af 7.550
ri 0.340
ra 0.140
nt 2
Elastic/Plastic Properties:
I
S
r
Z
Major Axis 103.000 20.700 3.420 0.649
Minor Axis 3.940 1.650 0.669 0.369
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
For Wave Calculations: 0.000
For Display: 0.000
Ft
Analysis Parameters
Code Check Parameters
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-20

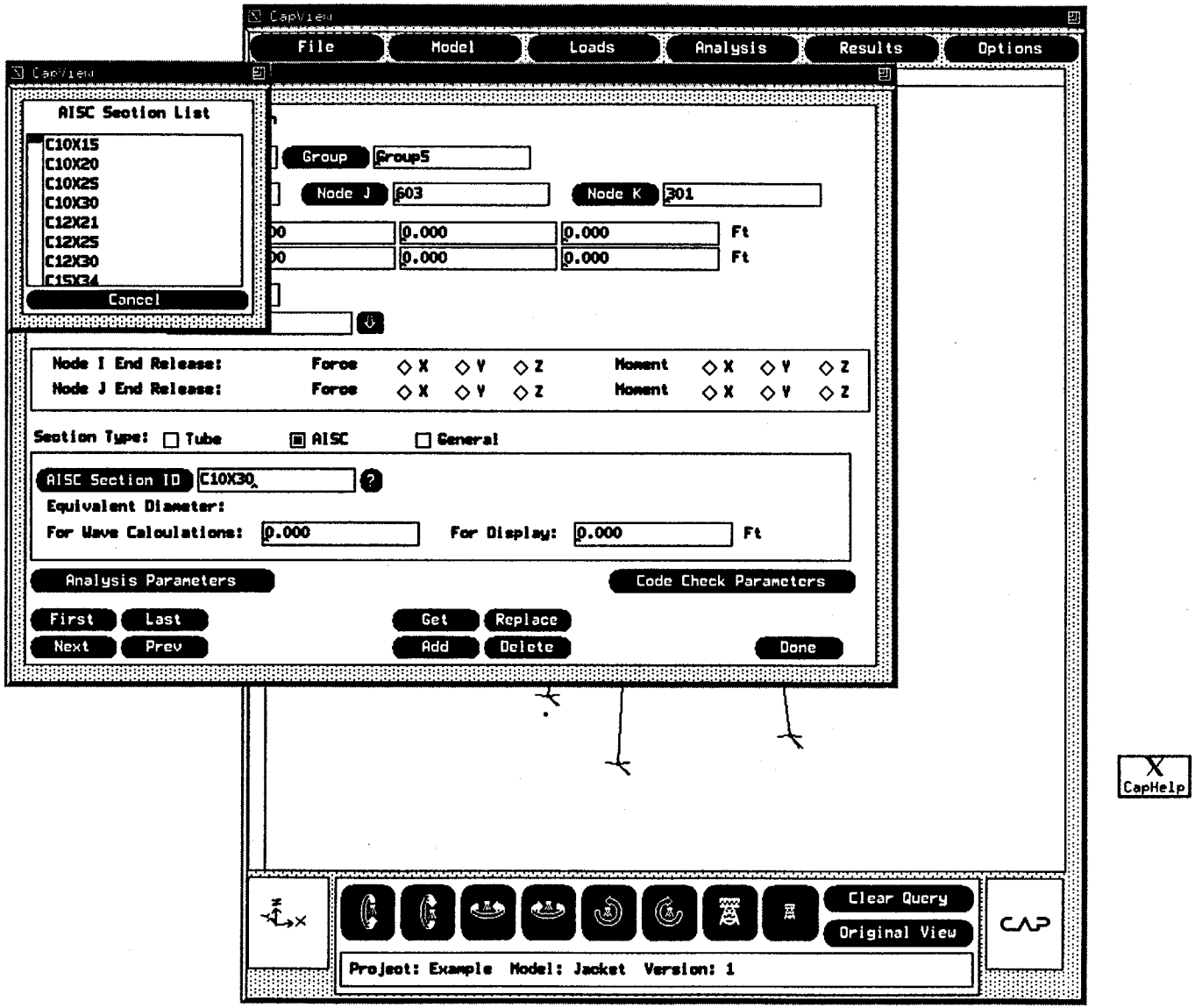

CapView
File
Model
Loads
Analysis
Results
Options
AISC Section List
C10X15
C10X20
C10X25
C10X30
C12X21
C12X25
C12X30
C15X34
Cancel
Group
Group5
Node J
603
Node K
301
0.000
0.000
Ft
0.000
0.000
Ft
Node I End Release:
Node J End Release:
Force
Moment
X
Y
Z
Section Type:
Tube
AISC
General
AISC Section ID
C10X30
Equivalent Diameter:
For Wave Calculations:
0.000
For Display:
0.000
Ft
Analysis Parameters
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-21

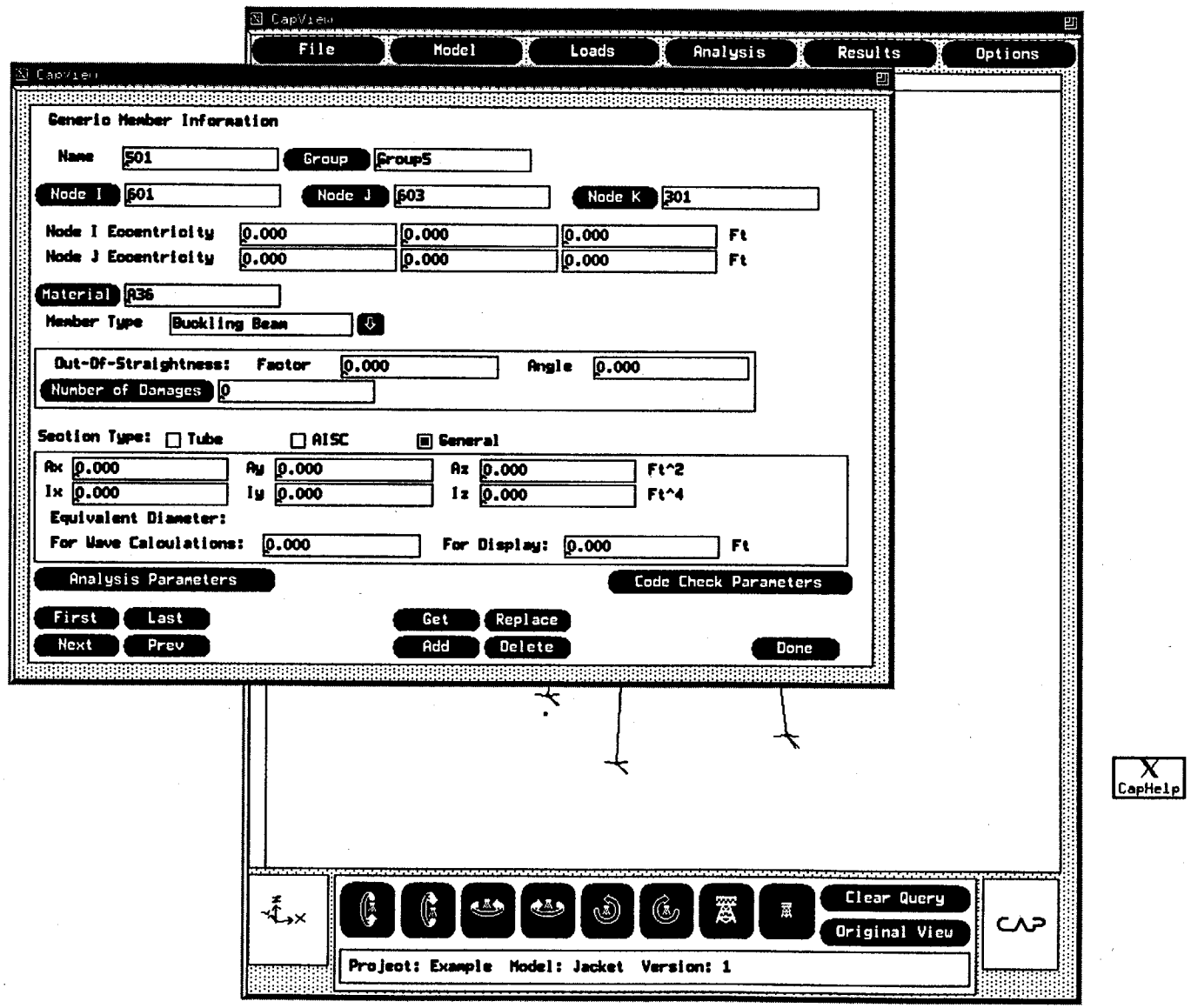

CapView
File
Model
Loads
Analysis
Results
Options
Generic Member Information
Name
501
Group
Group5
Node I
601
Node J
603
Node K
301
Node I Eccentricity
0.000
0.000
0.000
Ft
Node J Eccentricity
0.000
0.000
0.000
Ft
Material
A36
Member Type
Buckling Beam
Out-Of-Straightness:
Factor
0.000
Angle
0.000
Number of Damages
0
Section Type:
Tube
AISC
General
Ax
0.000
Ay
0.000
Az
0.000
Ft^2
Ix
0.000
Iy
0.000
Iz
0.000
Ft^4
Equivalent Diameter:
For Wave Calculations:
0.000
For Display:
0.000
Ft
Analysis Parameters
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-22

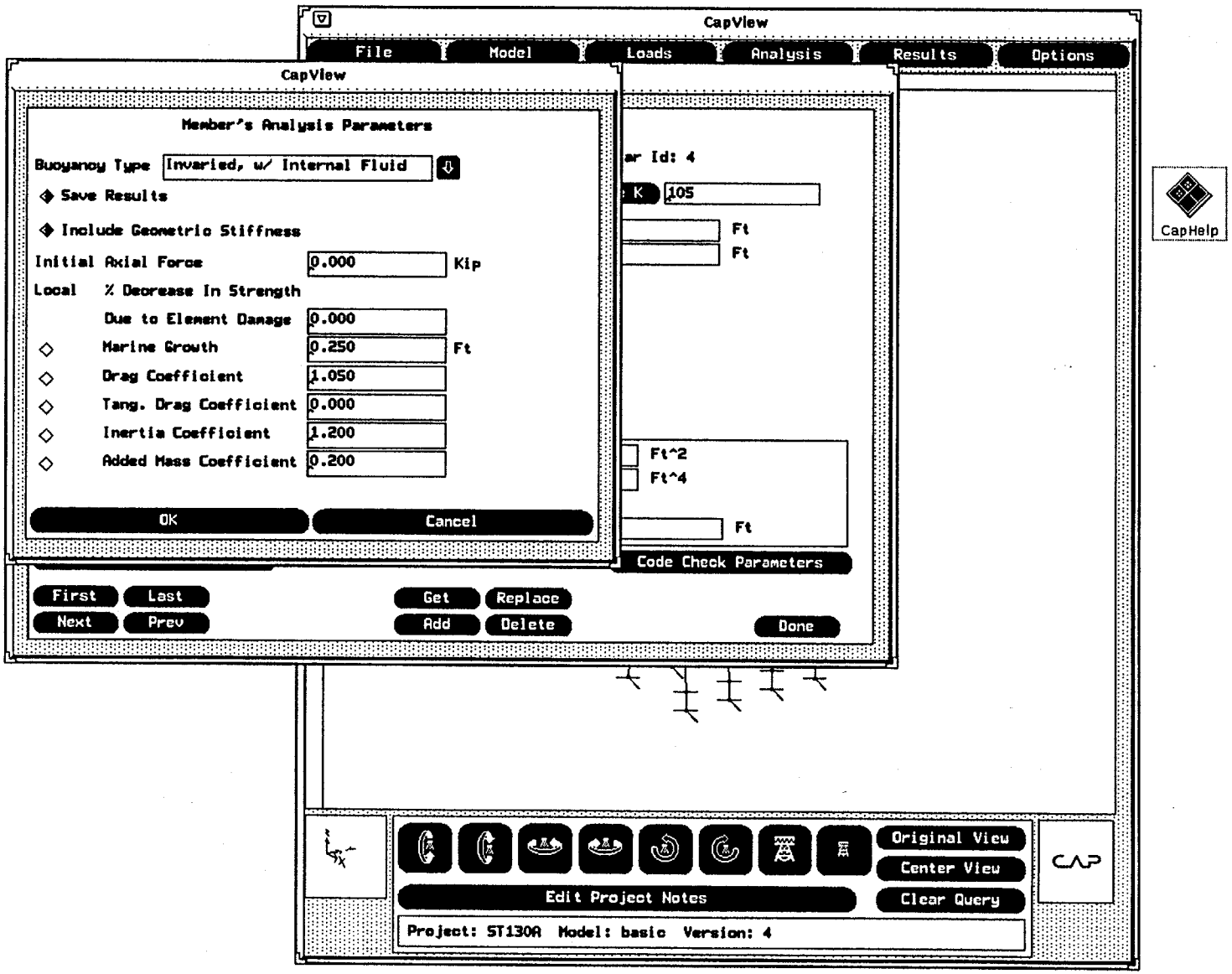

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Member's Analysis Parameters
Buoyancy Type
Invaried, w/ Internal Fluid
Save Results
Include Geometric Stiffness
Initial Axial Force
0.000
Kip
Local % Decrease In Strength
Due to Element Damage
0.000
Marine Growth
0.250
Ft
Drag Coefficient
1.050
Tang. Drag Coefficient
0.000
Inertia Coefficient
1.200
Added Mass Coefficient
0.200
OK
Cancel
ar Id: 4
105
Ft
Ft
Ft^2
Ft^4
Ft
Code Check Parameters
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 5-23

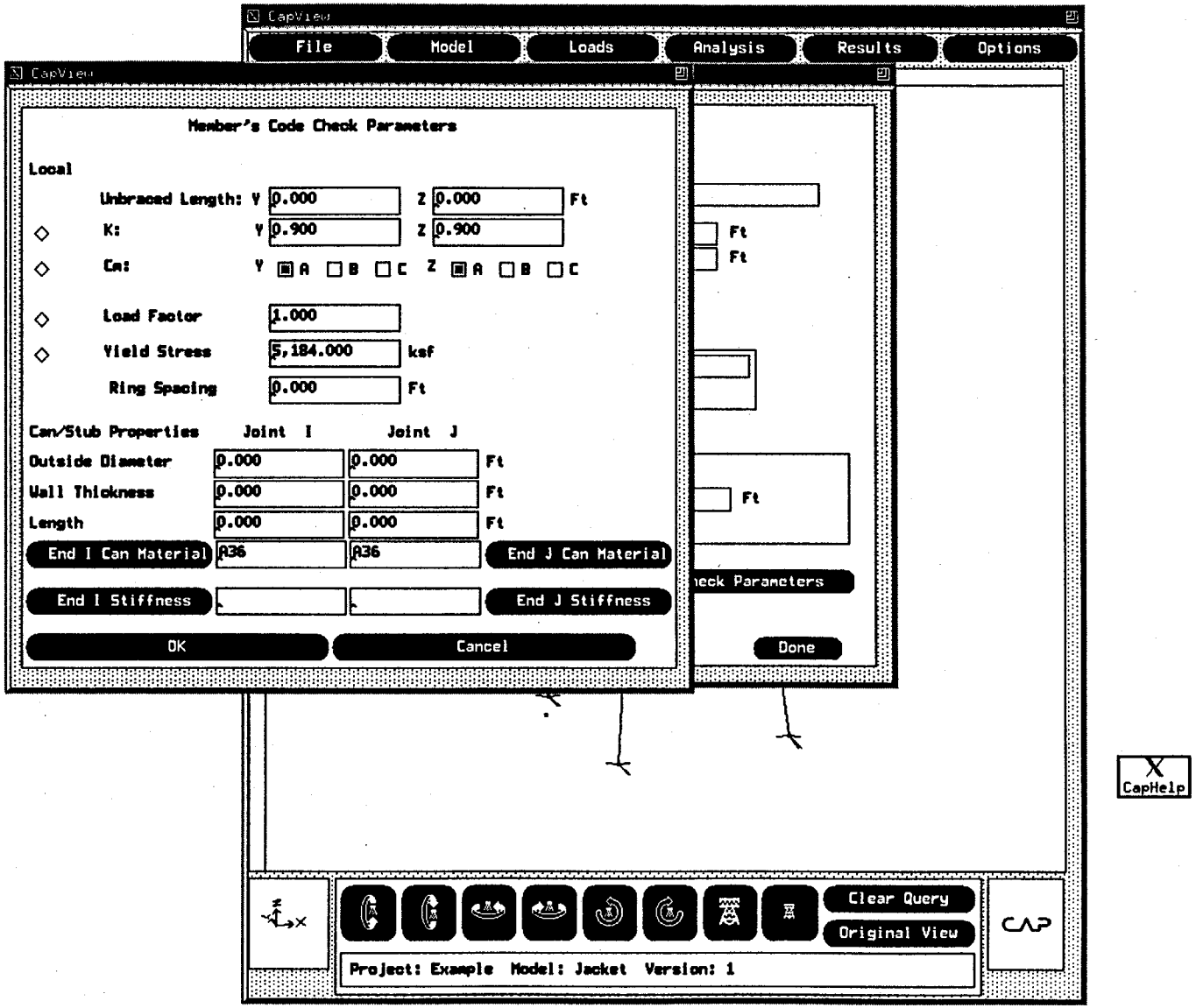
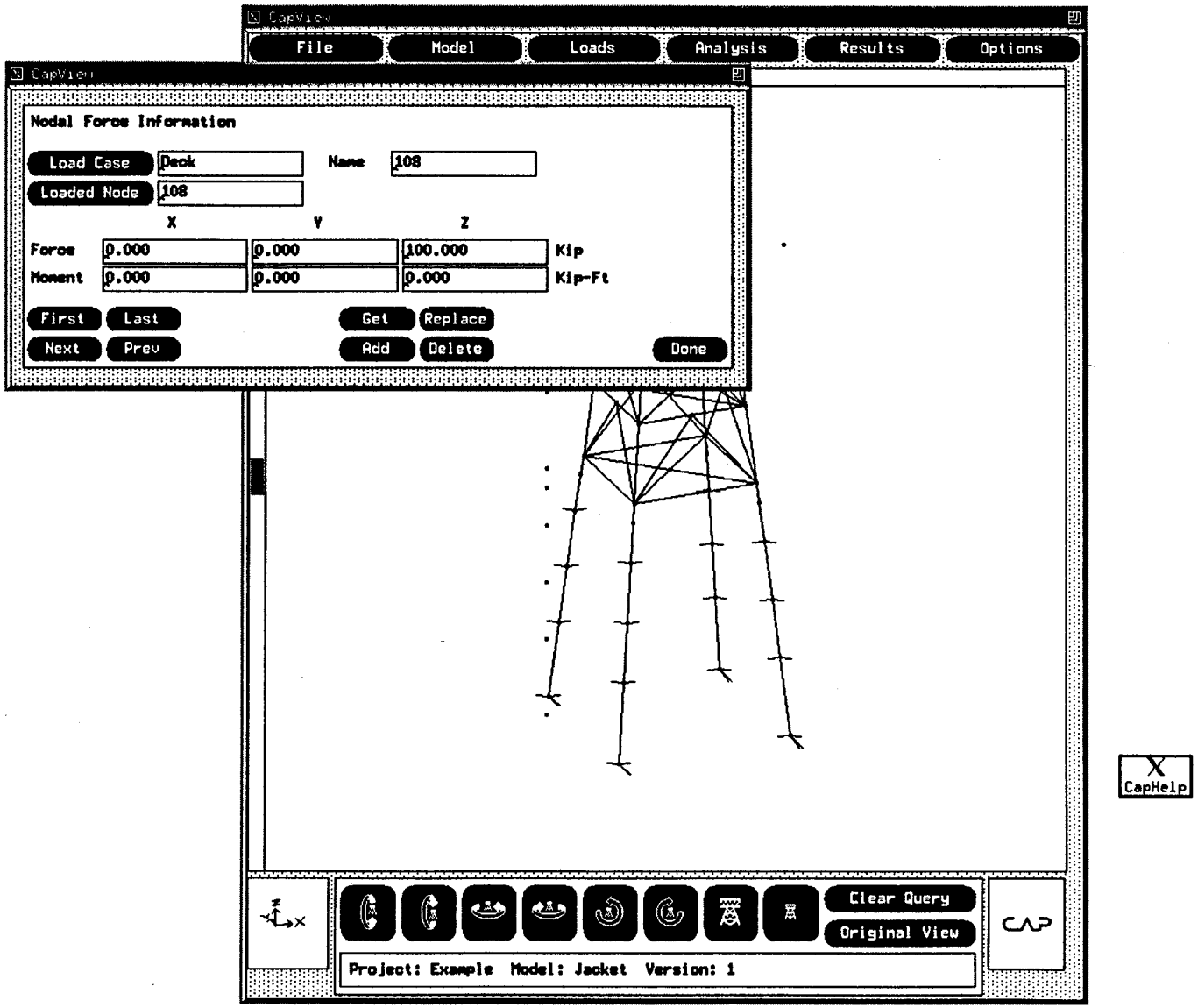
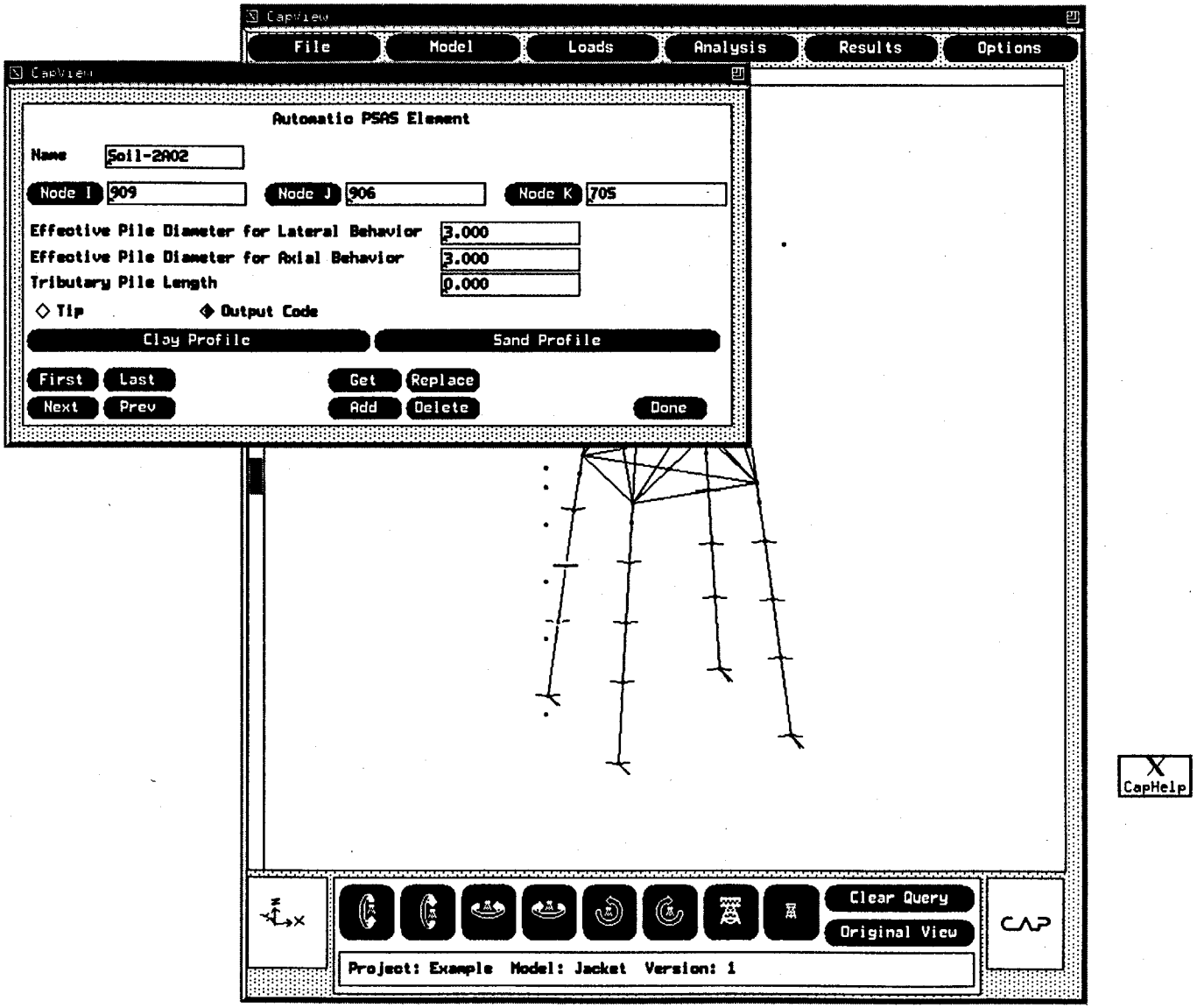

CapView
File
Model
Loads
Analysis
Results
Options
Member's Code Check Parameters
Local
Unbraced Length: Y 0.000 Z 0.000 Ft
K: Y 0.900 Z 0.900
Cm: Y A B C Z A B C
Load Factor 1.000
Yield Stress 5,184.000 ksf
Ring Spacing 0.000 Ft
Can/Stub Properties Joint I Joint J
Outside Diameter 0.000 0.000 Ft
Wall Thickness 0.000 0.000 Ft
Length 0.000 0.000 Ft
End I Can Material A36 A36 End J Can Material
End I Stiffness End J Stiffness
OK
Cancel
Ft
Ft
Ft
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-24


CapView
File
Model
Loads
Analysis
Results
Options
CapView
Nodal Force Information
Load Case
Deck
Name
108
Loaded Node
108
X
Y
Z
Force
0.000
0.000
100.000
Kip
Moment
0.000
0.000
0.000
Kip-Ft
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
Clear Query
Original View
CAP
Project: Example Model: Jacket Version: 1

CapHelp

Figure 5-25


CapView
File
Model
Loads
Analysis
Results
Options
Automatic PSAS Element
Name
Soil-2A02
Node I
909
Node J
906
Node K
705
Effective Pile Diameter for Lateral Behavior
3.000
Effective Pile Diameter for Axial Behavior
3.000
Tributary Pile Length
0.000
Tip
Output Code
Clay Profile
Sand Profile
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1

CapHelp

Figure 5-26

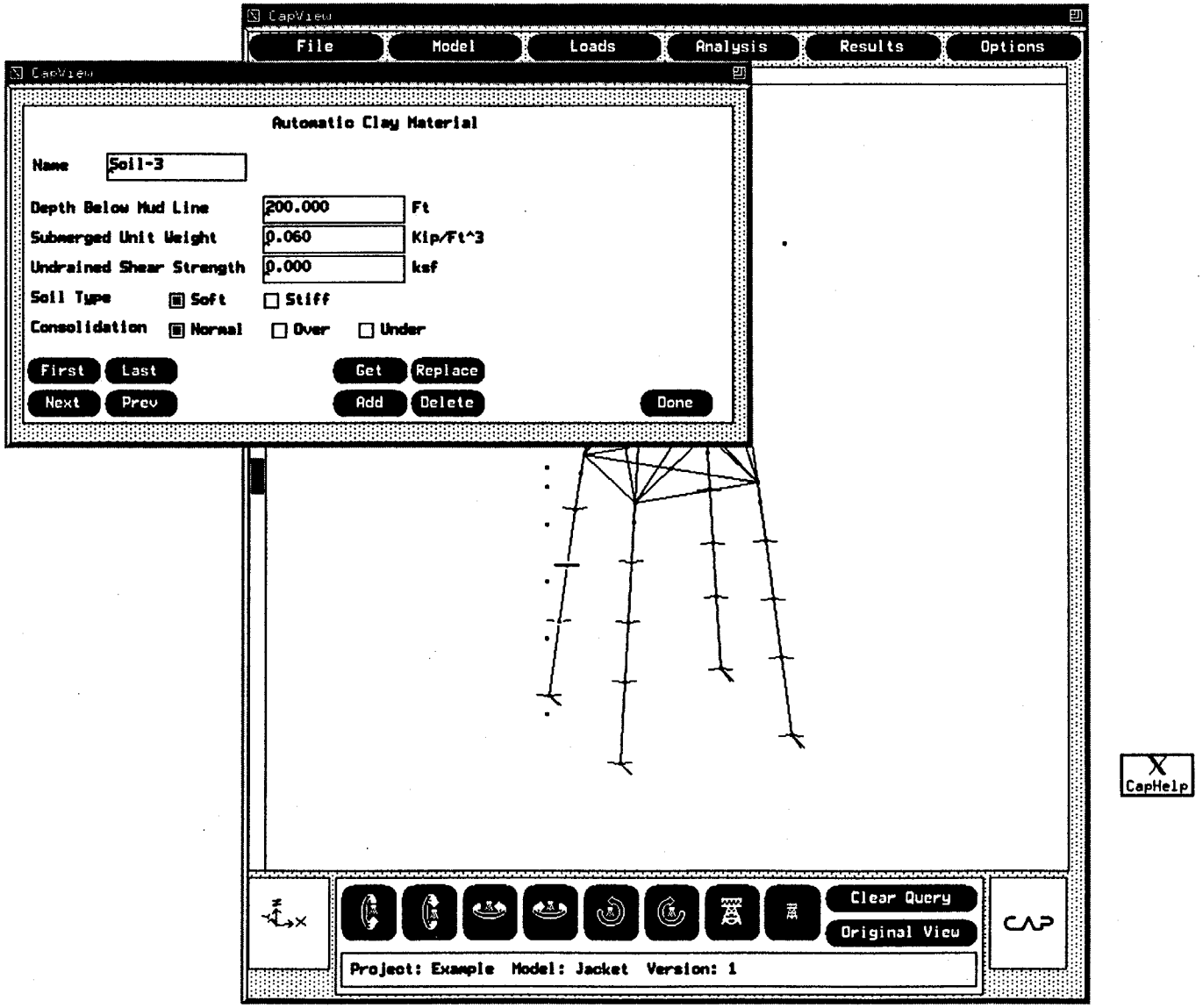

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Automatic Clay Material
Name
Soil-3
Depth Below Mud Line
200.000
Ft
Submerged Unit Weight
0.060
Kip/Ft^3
Undrained Shear Strength
0.000
ksf
Soil Type
Soft
Stiff
Consolidation
Normal
Over
Under
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-27

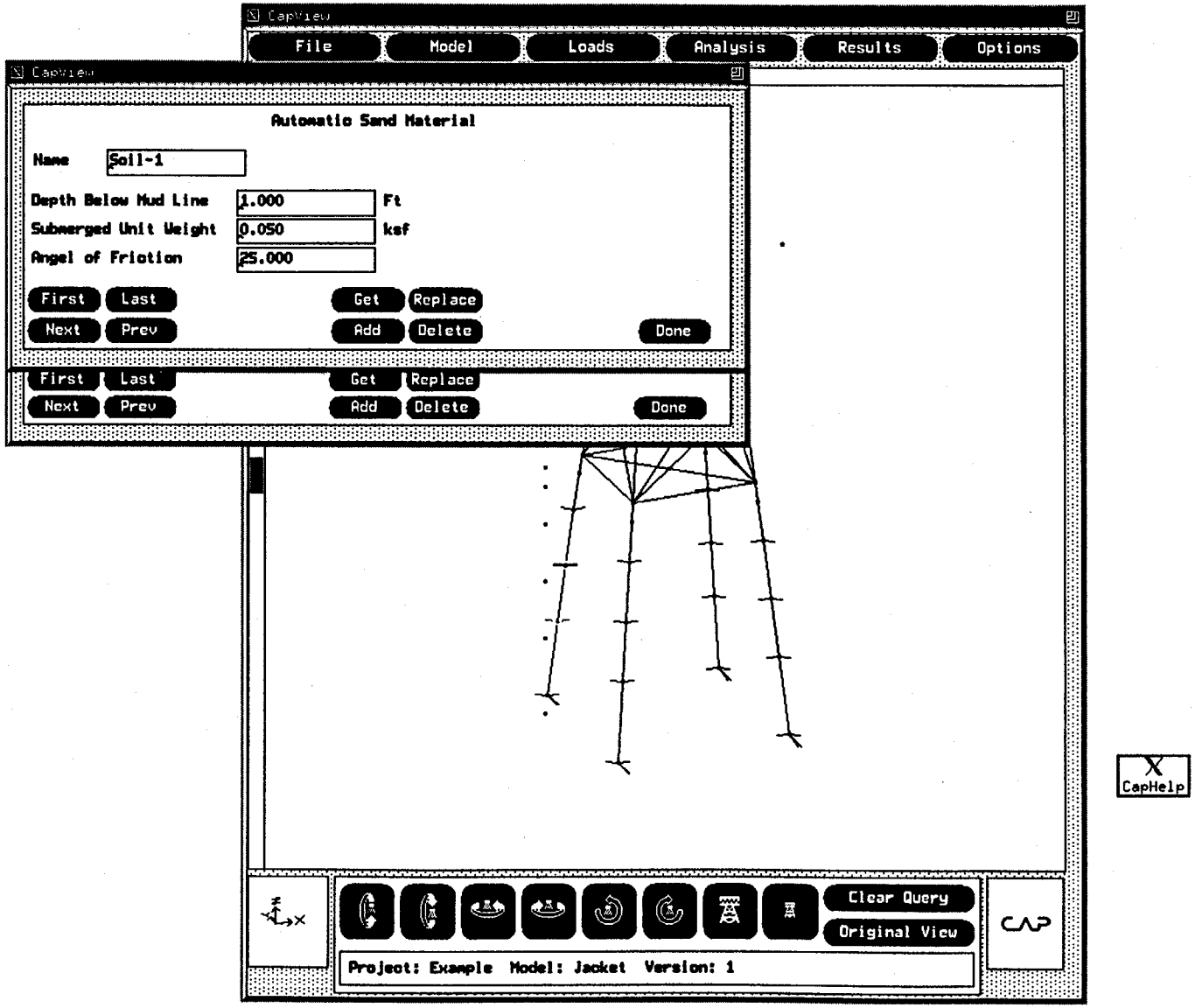

CapView
File
Model
Loads
Analysis
Results
Options
Automatic Sand Material
Name
Soil-1
Depth Below Mud Line
1.000
Ft
Submerged Unit Weight
0.050
ksf
Angel of Friction
25.000
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CapHelp

Figure 5-28

6.0 FILE MENU

The File menu presents the user with selections that are used to create, open, close and delete project files and directories. Each CapView project is given its own directory. Within the project directory there are model subdirectories, and within the model subdirectories there are version subdirectories which contain the actual data files. CapView looks for project directories below whichever directory it is started from. You can change to a new directory without leaving CapView using the "Working Directory" menu selection (see below). When the File menu is selected, the following menu is displayed (Figure 6-1):

New Project ->
Open Project ...
Save As ...
Save
Abandon
Copy Project...
Delete Project ->
Recover Project ...
Working Directory ...
Recent Modifications
About CapView
Exit CapView

When the mouse is at the "New Project" selection, the following sub menu is displayed (Figure 6-2):

Build New Project...
Import From SeaStar...
Import From SACS...
Import From STRUDL...

When the mouse is at the "Delete Project" selection, the following sub menu is displayed (Figure 6-3):

Delete Project...
Delete Model...
Delete Version...
Delete Scratch...

6.1 Build New Project

The first selection of the File menu's New Project submenu will display the Build New Project dialogue box (Figure 6-4). This is used to create a new project, model or version. The dialogue box has the following edit fields:

- Project Name: This is where the Project name is entered. It can be a new Project name or an existing one. If a new project name is given then a new project directory will be created under the current working directory.
- Model Name: This is where the Model name is entered. It can be a new Model name or an existing one. If a new model name is given then a new model directory will be created under the project directory.

It also has the following buttons:

- Accept: Select this button to accept the Project and Model names that have been entered. If there are no projects with these names then a subdirectory under the model directory will be created. It will be named version1. Otherwise a directory will be created with the next available version number.
- Cancel button: Select this button to cancel the request.

6.2 Import From SeaStar

The second selection of the File menu's New Project submenu will display the Import from SeaStar dialogue box (Figure 6-5). This is used to create a new project, model or version based on an existing SeaStar file. The dialogue box has the following edit fields:

- Project Name: This is where the Project name is entered. It can be a new Project name or an existing one. If a new project name is given then a new project directory will be created under the current working directory.
- Model Name: This is where the Model name is entered. It can be a new Model name or an existing one. If a new model name is given then a new model directory will be created under the project directory.
- SeaStar File: This is where the SeaStar run name (name without any extension) to import into CapView is entered. CapView imports the SeaStar input file (*.inp) along with following output files (if they exist): *.acc, *.bid, *.chk, *.cnf, *.ele, *.erg, *.err, *.evt, *.fat, *.gravity, *.ldf, *.load, *.mem, *.mod, *.nct, *.nds, *.ok, *.out, *.ovr, *.psas, *.pst, *.pvt, *.rst, *.sno, *.sts, *.thf, *.thf.ir, *.thf.ir.max, *.unb, and *.wav.
- SeaStar Path: This is where the path to the SeaStar files to import is entered. Note, if the SeaStar file is in the current working directory, you can leave this field blank.

It also has the following buttons:

- Accept button: Select this button to accept the Project and Model names that have been entered. If there are no projects with these names then a subdirectory under the model directory will be created. It will be named version1. Otherwise a directory will be created with the next available version number.
- Cancel button: Select this button to cancel the request.

6.2.1 Notes on Importing From SeaStar

When CapView imports a SeaStar input file the following is assumed:

- All information in section D is imported with the exception of static current. Note that although rotated coordinate system and their assignments are imported, they are not not displayable or editable.
- All Element information (section E) is imported. Each group is assigned a CapView group name made up of the letter 'g' along with a sequential number, followed by the character '-' (e.g. "g11-").
- Linear beam, wave, and shim elements are imported into Generic CapView elements which can be edited extensively. However, the rest of the line type type elements, gab friction elements and quad elements are imported into a rigid format that can be edited in a limited way.
- CapView has a limit of 50 members in Generic CapView groups. If a group of linear beams, wave or shim elements has more than 50 members, this group is split to accommodate this restriction.
- PSAS elements are imported into a generic CapView PSAS element type. Note that CapView can extensively edit the fully automatic and the generalized elements. The Semi automatic elements are imported but their semi-automatic properties are not editable or displayable.
- CapView can edit the soil properties of a single PSAS group (called "Soil-"). When you import a SeaStar file, it is possible that your model has more than one PSAS group. The first PSAS group in the input file is

imported into "Soil-" group and, therefore, is editable. The subsequent PSAS groups will be imported into groups named in the same way non-PSAS groups are named. The soil properties of the latter groups are not editable.

- Static nodal force, imposed nodal displacements, element self weight and buoyancy loads are imported; the rest of the static load conditions of section F are skipped.
- Wave loads (regular and irregular) and ground motion records are imported. The rest of the dynamic load conditions of section G are skipped.
- Output specification (section H) is skipped.
- All analysis macros are imported.

6.3 Import From SACS

The third selection of the File menu's New Project submenu will display the Import from SACS dialogue box (Figure 6-6). This is used to create a new project, model or version based on an existing SACS file. The dialogue box has the following edit fields:

- Project Name: This is where the Project name is entered. It can be a new Project name or an existing one. If a new project name is given then a new project directory will be created under the current working directory.
- Model Name: This is where the Model name is entered. It can be a new Model name or an existing one. If a new model name is given then a new model directory will be created under the project directory.
- SACS File: This is where the name of the SACS file to import into CapView is entered. It is not necessary to add any extension. The complete path must be given if the SACS files are not in the current working directory.

It also has the following buttons:

- Show SACS Defaults button: Select this button to view or edit the SACS conversion defaults in the SACS Conversion Defaults dialogue box (Figure 6-7).
- Accept button: Select this button to accept the Project and Model names that have been entered. If there are no projects with these names then a subdirectory under the model directory will be created. It will be named version1. Otherwise a directory will be created with the next available version number.
- Cancel button: Select this button to cancel the request.

6.3.1 SACS Conversion Defaults dialogue box

This dialogue box is used to set conversions needed to import a SACS file. It has the following:

- Minimum Length Ratio of End Segment edit field: This edit field is used to input the minimum length ratio of a segment of a variable cross section member that is going to saved as a segment. Only segments whose length exceeds this ratio will be modeled separately in CapView.
- Number of Shim (Wishbone) GRUP Names edit field: This is where the number of Shim GRUP names is entered.
- Show Shim GRUP Names button: This button, when selected, displays a list box where the Shim GRUP names are shown (Figure 6-8). There will be as many entries as are entered in the Number of Shim (Wishbone) GRUP Names field. The entries may be edited. When the entries are correct they may be saved by selecting the Apply button or they may be discarded by selecting the Cancel button.
- Accept button: Select this button to accept the SACS conversion defaults.
- Cancel button: Select this button to discard any changes that were made to the SACS conversion defaults.

6.3.2 Notes on Importing From SACS

The following is a list of SACS cards that are supported by the SACS import utility:

Card Set 5.10 (SECT),

Card Set 6.1 (GRUP),

Card Set 7.1 (MEMBER),

Card Set 7.1.1 (MEMBER for Offsets),

Card Set 11.1 (JOINT),

Card Set 14.3 (MGROV),

Card Set 16.1 (LOADCN), and

Card Set 16.3 (LOAD for MEMBER UNIF, MEMBER CONC, MEMB MOMT and JOIN).

Note that member loads are replaced by nodal loads that are equal in magnitude to the applied member loads but are not statically equivalent loads (i.e. no moment is applied to compensate for location and distribution of loads along the span of the member).

In addition to the above:

Mud line elevation is extracted from Card Set 1.10 (LDOPT),

Clean member drag and inertia coefficients from Card Set 13.1 (CDM),

Y-direction drag and inertia coefficients from Card Set 14.1 (GRPOV), and

Y-direction drag and inertia coefficients from Card Set 14.2 (MEMOV).

When you import a SACS file watch out for the following:

- If you get a warning about a member length that is zero, this is most probably a Shim (Wishbone) element that you forgot to define in the SACS Conversion Defaults dialogue box.
- SACS' Shim element has no direction (K) node, so this field is left blank when importing shims. CapView requires a direction node for each Shim element, and so you must explicitly set the K node for each shim element (the Member Select/Change dialogue box can help you do this quickly). This direction node must be along the line of the pile/leg and must be different from the two other nodes (I and J) of the Shim element.
- If there are load cards which are not supported by the CapView import utility among supported load cards, they tend to terminate load import prematurely. Watch out for this one.
- In general, if there are comment lines in the SACS file, they tend to prematurely terminate the section they exist in.

6.4 Import From STRUDL

The fourth selection of the File menu's New Project submenu will display the Import from STRUDL dialogue box (Figure 6-9). This is used to create a new project, model or version based on an existing STRUDL file. The dialogue box has the following edit fields:

- Project Name: This is where the Project name is entered. It can be a new Project name or an existing one. If a new project name is given then a new project directory will be created under the current working directory.
- Model Name: This is where the Model name is entered. It can be a new Model name or an existing one. If a new model name is given then a new model directory will be created under the project directory.
- STRUDL File: This is where the name of the STRUDL file to import into CapView is entered. It is not necessary to add any extension. The complete path must be given if the STRUDL files are not in the current working directory.

It also has the following buttons:

- Show STRUDL Defaults button: Select this button to view or edit the STRUDL conversion defaults in the STRUDL Conversion Defaults dialogue box (Figure 6-10).
- Accept button: Select this button to accept the Project and Model names that have been entered. If there are no projects with these names then a subdirectory under the model directory will be created. It will be named version1. Otherwise a directory will be created with the next available version number.
- Cancel button: Select this button to cancel the request.

6.4.1 STRUDL Conversion Defaults dialogue box

This dialogue box is used to set conversions needed to import a STRUDL file. It has the following:

- Minimum Length Ratio of End Segment edit field: This edit field is used to input the minimum length ratio of a segment of a variable cross section member that is going to saved as a segment. Only segments whose length exceeds this ratio will be modeled separately in CAP.
- Accept button: Select this button to accept the STRUDL conversion defaults.
- Cancel button: Select this button to discard any changes that were made to the STRUDL conversion defaults.

6.4.2 Notes on Importing From STRUDL

CAP can not import a STRUDL input file, it requires a file created by the PRINT DATA command containing the string PROBLEM DATA FROM INTERNAL STORAGE.

The following is a list of STRUDL commands that are supported by the STRUDL import utility:

- JOINT COORDINATES,
- JOINT RELEASES,
- JOINT INERTIAS,
- MEMBER INCIDENCES,

- MEMBER END ECCENTRICITIES,
- MEMBER PROPERTIES for PRISMATIC, TABLE, and VARIABLE cross sections,
- Material properties from the MEMBER CONSTANTS section including: E, G, DENSITY, BETA (K-node rotation), POISSON, and FYLD (yield stress),
- DESIGN DATA yield stress, Ky, Kz, unbraced length y & z,
- LOADING defining different load cases,
- MEMBER AND ELEMENT LOAD FORCE/MOMENT in X, Y, and Z direction, CONCentrated, UNIForm, and LINEar member force. INERTIA WEIGHT LOAD, UNIFORM DISTortion, CONCENTRATED DISTortion, and TEMPERATURE loading are skipped. Note that member loads are replaced by nodal loads that are equal in magnitude to the applied member loads but are not statically equivalent loads (i.e. no moment is applied to compensate for location and distribution of loads along the span of the member).
- JOINT LOAD
- JOINT DISPLACEMENT (skipped)

6.5 Open Project

The second selection of the File menu will display the Open Project dialogue box (Figure 6-11). This dialogue box is used to open an existing project. The dialogue box has three scrollable lists:

- Project List: When the Open Project dialogue box is displayed this list will be filled with the name of all project directories that are under the current working directory. The current working directory can be changed by using the Working Directory selection from the File menu. Click on the project to open.
- Model List: When a project has been selected in the Project List the models that are under that Project will be listed in this box. Click on the model to open.
- Version List: When a model has been selected in the Model List the versions that are under that model will be listed in this box. Click on the version to open.

The dialogue box has a radio button to toggle between secure and insecure operation modes:

- Secure: This selection is used to select secure operation mode. In secure operation mode the project files are copied to a scratch directory and those copies are operated on by CapView. When the project is saved, the original files are replaced by the scratch files. This helps prevent any accidental loss of files or unwanted changes.
- Insecure: This selection is used to select insecure operation mode. In insecure operation mode the project files are operated on by CapView directly. Because you are working directly on the project files and not on a copy, you cannot revert to the original state of this version of the project. This mode is faster to start and to save.

In addition to the above there are three buttons:

- Edit Project Notes button: When a Project, Model and Version have been chosen, select this button to edit or view the notes for this project. A dialog will be displayed where you may enter notes, view notes previously entered or search for a string within the notes. Note that the main window's Edit Project Notes button does the same thing when the project is open.
- Accept button: Select this button to open the project whose project name, model and version have been selected.
- Cancel button: Select this button to return to Files menu. No project will be opened.

6.6 Save As

The third selection of the File menu will display the Save As dialogue box (Figure 6-12). This dialogue box is used to save the currently open project with a new project name, model or version. The project must have been opened in secure mode. The dialogue box has the following edit fields:

- Project Name: This is the edit field for Project name. It can be a new Project name or an existing one. If a new project name is given then a new project directory will be created under the current working directory.
- Model Name: This is the edit field for Model name. It can be a new Model name or an existing one. If a new model name is given then a new model directory will be created under the project directory.
- Version: This is the edit field for Version number. It must be an integer. If an existing version number is given then an error message will be generated.

Also, the dialogue box has following buttons:

- Accept button: Select this button to accept the project, model and version names that have been entered. Directories will be created as needed and the files will be saved. Note that the currently open project will be closed as a result of this operation, so you must use the File-Open menu to open a project and continue working.
- Cancel button: Select this button to cancel the request.

6.7 Save

The fourth selection of the File menu will save the currently open project into the same directory from which it was opened. Note that the currently open project will be closed as a result of this operation, so you must use the File-Open menu to open a project and continue working.

6.8 Abandon

The fifth selection of the File menu will abandon the currently open project. If the project was opened in secure mode then the original files will be intact. If the project was opened in insecure mode then this selection has no effect. When this selection is made, a confirmation dialogue box is displayed (Figure 6-13). The dialogue box has two buttons. The first, labeled Yes, will abandon the project when selected. The other, labeled No, will ignore the request.

6.9 Copy Project

The sixth selection of the File menu will display the Copy Project dialogue box (Figure 6-23). This dialogue box is used to copy an existing project. The dialogue box has three scrollable lists:

- Project List: When the Copy Project dialogue box is displayed this list will be filled with the name of all project directories that are under the current working directory. The current working directory can be changed by using the Working Directory selection from the File menu. Click on the project to open.
- Model List: When a project has been selected in the Project List the models that are under that Project will be listed in this box. Click on the model to open.
- Version List: When a model has been selected in the Model List the versions that are under that model will be listed in this box. Click on the version to open.

The dialogue box has three edit fields and two check boxes:

- Project Name: This is the edit field for the Project name.
- Model Name: This is the edit field for the Model name.
- Version Number: This is the edit field for the Version Number.
- A check box to include printable files with the copied data.
- A check box to include analysis files with the copied data.

In addition to the above there are two buttons:

- Accept button: Select this button to copy the project whose project name, model and version have been selected.
- Cancel button: Select this button to return to Files menu. No project will be copied.

6.10 Delete Project

The first selection of the File menu's Delete Project submenu will display the Delete Project dialogue box (Figure 6-14). This is used to delete an entire project, including all models and versions. The dialogue box has an edit field labeled Project Name where the Project name is entered. It also has the following buttons:

- Accept button: Press this button to delete the specified project.
- Cancel button: Select this button to cancel the request.

6.13 Delete Scratch

Selecting Delete Scratch from the delete project sub-menu displays the Delete Scratch dialogue box (Figure 6-17). This is used to delete a scratch copy. The dialogue box has two edit fields:

- Project Name : This is the edit field for the Project name.
- Scratch : This is the edit field for the Scratch number. It must be an integer.

The dialogue box also has the following buttons:

- Accept button: Press this button to delete the specified scratch.
- Cancel button: Select this button to cancel the request.

6.14 Recover Project

The eighth selection of the File menu will display the Recover Project dialogue box (Figure 6-18). This is used to recover a project that was in use during a system crash. It is necessary to be in the directory that the project was originally run from. After a recover is performed any changes that were made prior to the crash will be lost. The dialogue box has three scrollable lists:

- Project List: This list box contains the names of all of the projects that are under the current working directory. Select the project to be recovered by clicking on its name. The current working directory can be changed by using the Working Directory selection from the File menu.
- Scratch List: When a project has been selected from the Project List box this list box will display a list of all of the scratch directories that have been created. Select the one to be restored.

In addition to the above there are two buttons:

- Accept button: Press this button to restore the specified scratch files. They will replace the files that were left in the version directory at the time of the crash.
- Cancel button: Select this button to return to the files menu. Nothing will be restored.

6.15 Working Directory

The ninth selection of the File menu will display one of two dialogue boxes, depending on the status of the project.

If there is no open project, the dialogue box (Figure 6-19) will have an edit field labeled Working directory and two buttons:

- Accept button: Selecting this button will change the current working directory to the one entered in the edit field and close the dialogue box.
- Cancel button: Selecting this button will ignore the request and close the dialogue box.

If there is an open project, the dialogue box (Figure 6-20) will show the name of the open project, mode, and version, the current working directory, and a message on the operation mode of the open project (secure or insecure). There is also a button labeled Cancel that when selected will close the dialogue box.

6.16 Recent Modifications

The tenth selection of the File menu will display a dialogue box that shows a list of recent modifications incorporated in the current version of CapView (Figure 6-24).

6.17 About CapView

The eleventh selection of the File menu will display a dialogue box that gives the user information about the current version of CapView and the contacts with PMB Engineering in case you have questions (Figure 6-21).

6.18 Exit CapView

The twelfth selection of the File menu will display one of two dialogue boxes, depending on the status of the project.

If there is no open project (Figure 6-22), or there is an open project in insecure operation mode, the dialogue box will have a confirmation message and two buttons:

- Yes button: Selecting this button will exit CapView.
- No button: Select this button to cancel the request.

If there is an open project in secure operation mode (Figure 6-23) the dialogue box will have a confirmation message and three buttons:

- Yes button: Selecting this button will exit CapView. All changes to the files will be saved first.
- No button: Selecting this button will exit CapView. All changes made will be lost.
- Cancel button: Select this button to cancel the request.

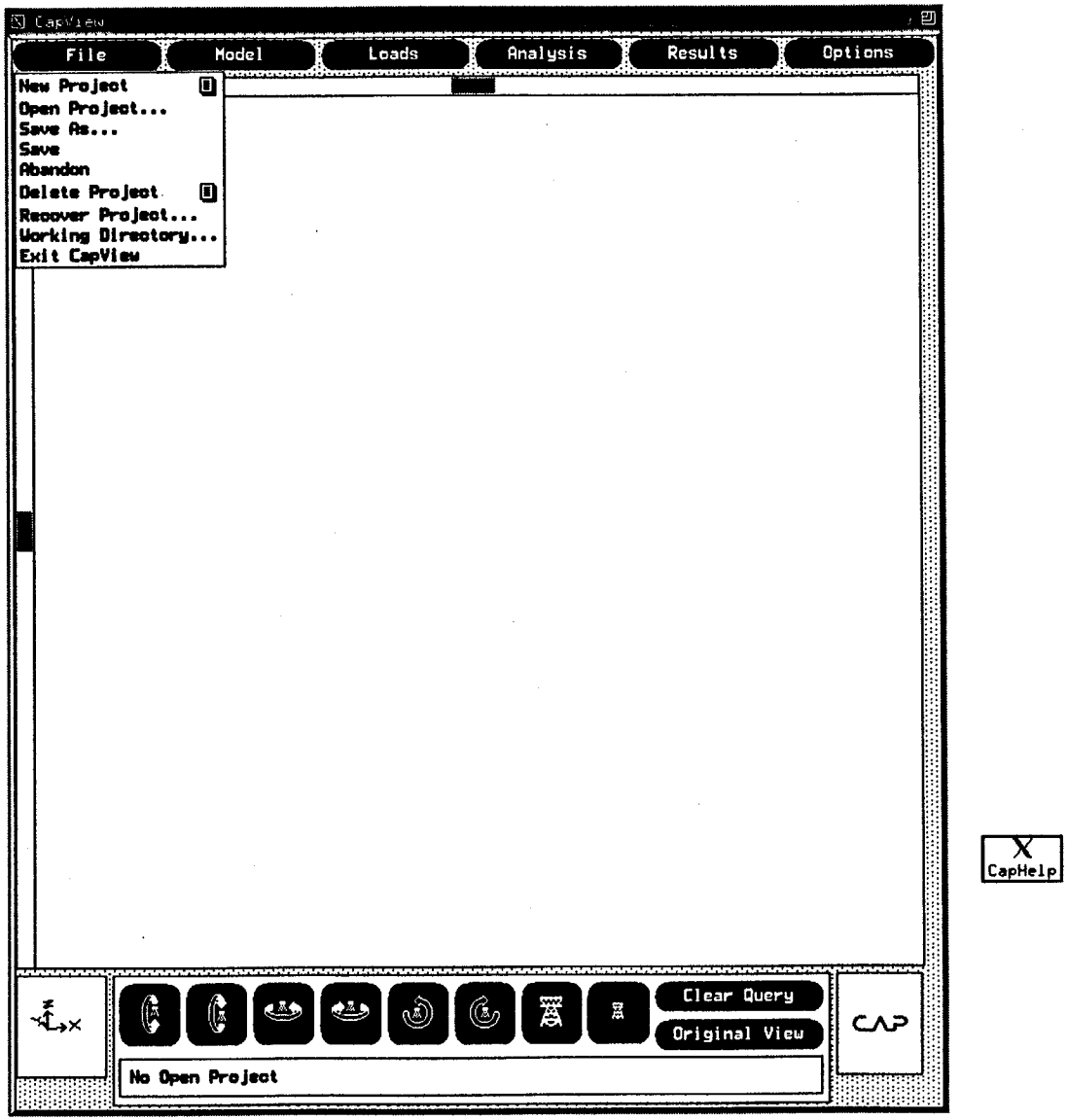

CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
Clear Query
Original View
No Open Project
CapHelp

Figure 6-1

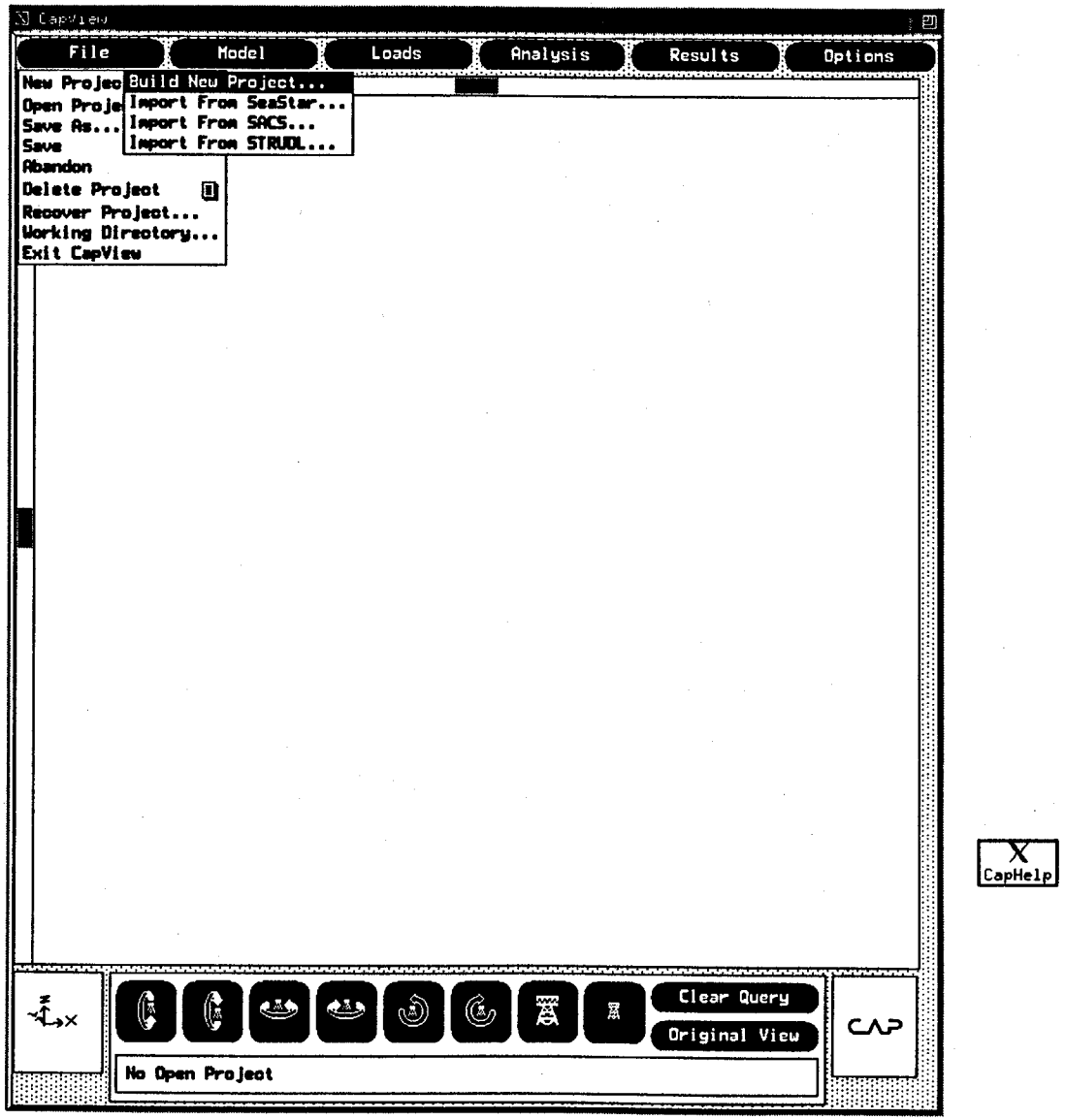
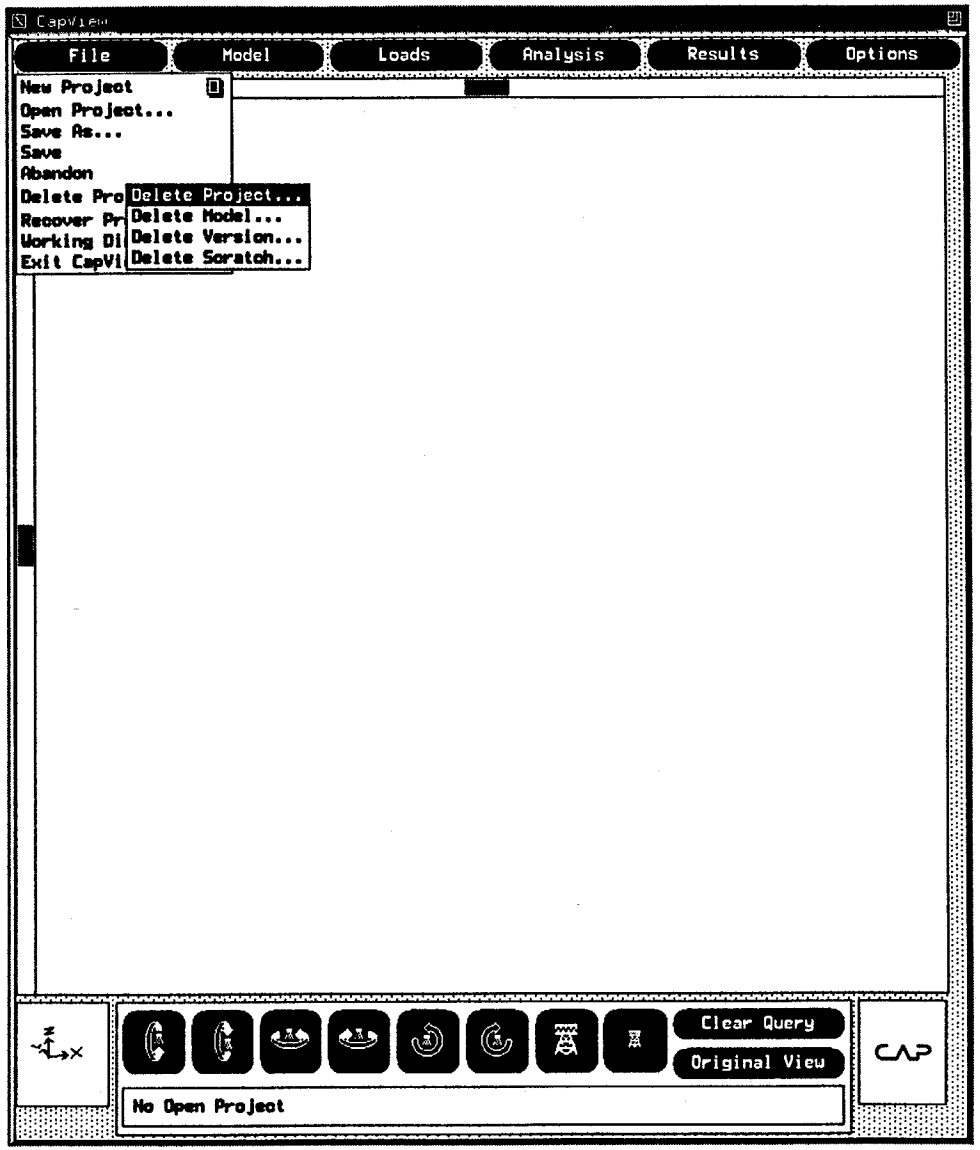
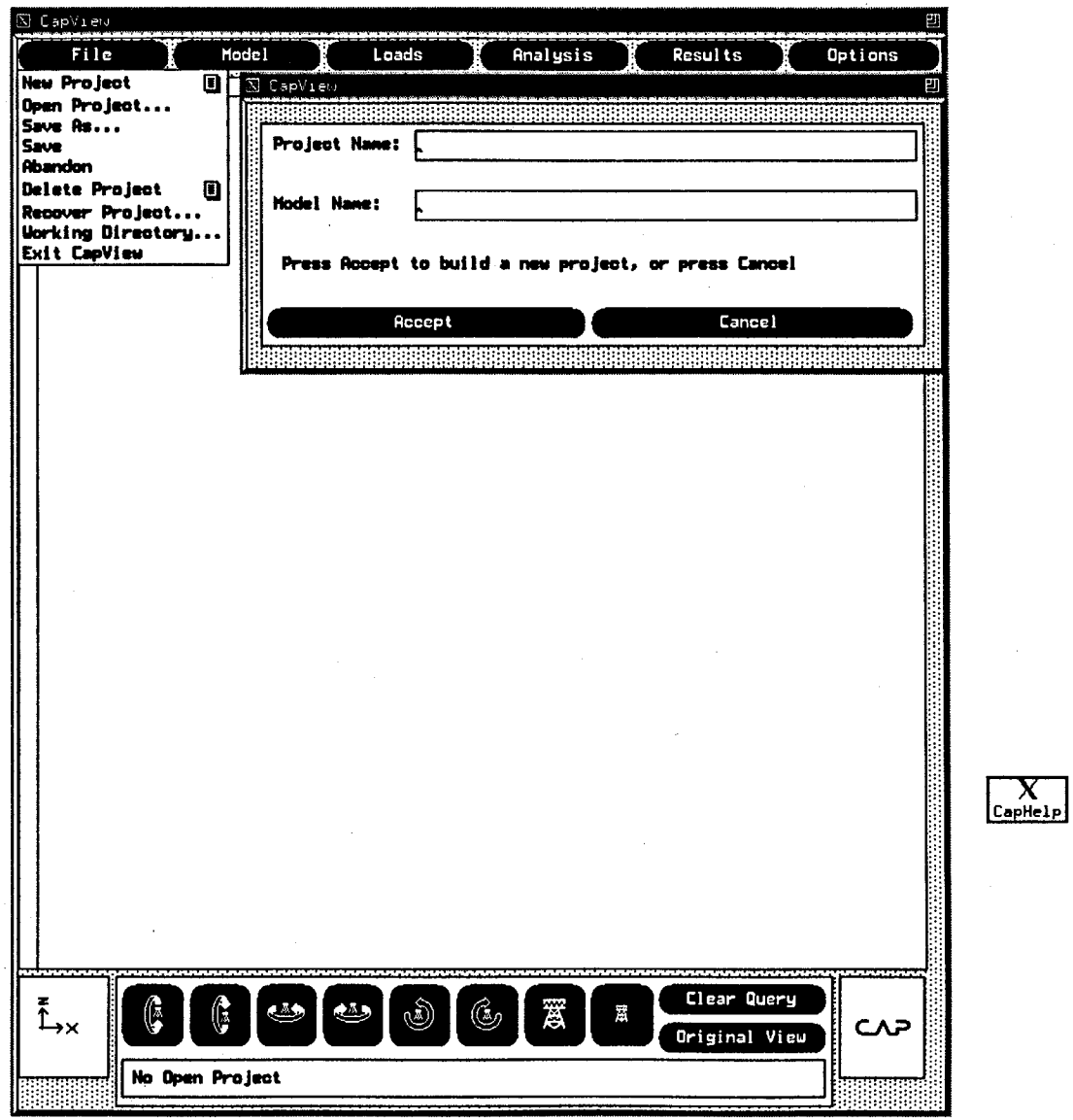

CapView
File
Model
Loads
Analysis
Results
Options
New Projec
Open Proje
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
Build New Project...
Import From SeaStar...
Import From SACS...
Import From STRUDL...
Clear Query
Original View
No Open Project
CapHelp

Figure 6-2


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Pro
Recover Pr
Working Di
Exit CapVi
Delete Project...
Delete Model...
Delete Version...
Delete Scratch...
Clear Query
Original View
No Open Project
CAP

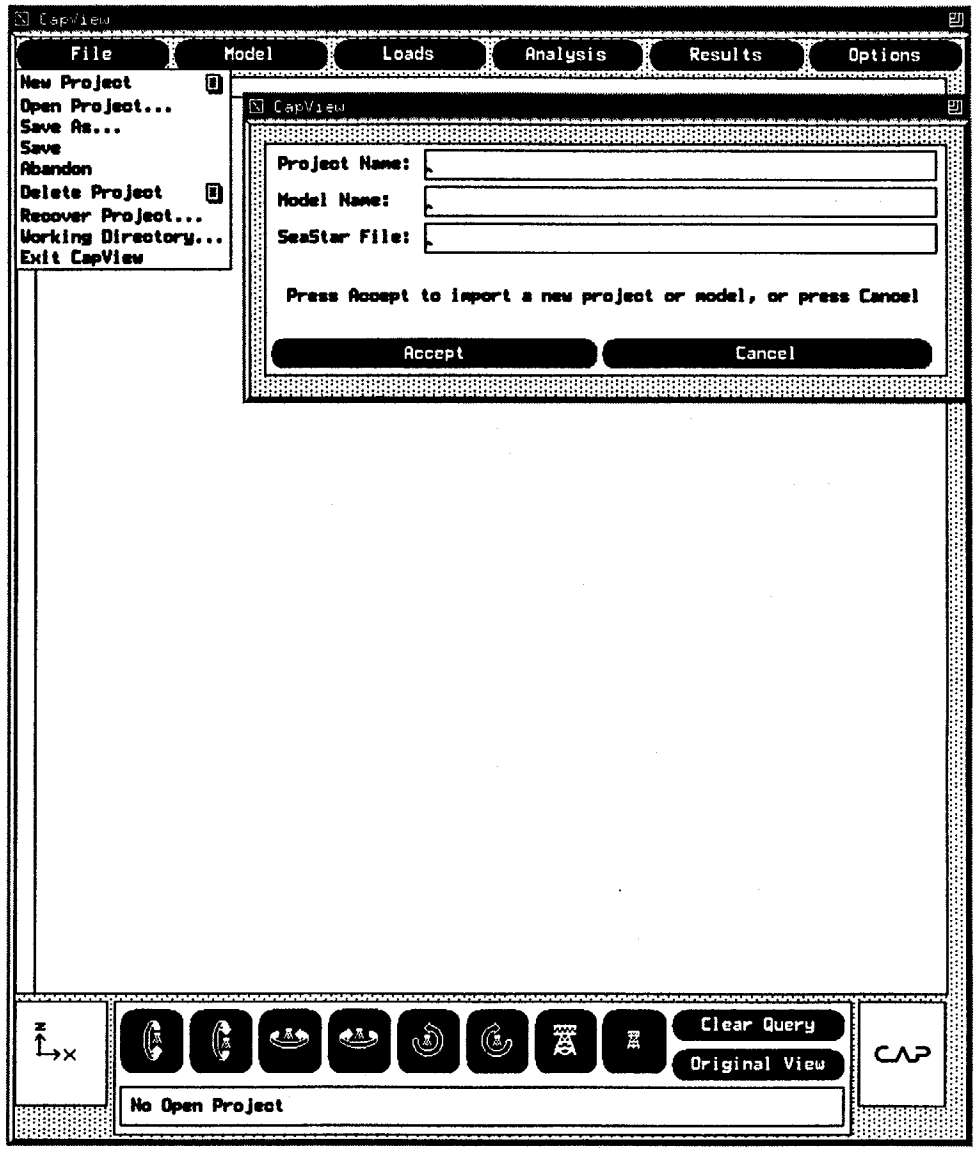
CapHelp

Figure 6-3


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
CapView
Project Name:
Model Name:
Press Accept to build a new project, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP

CapHelp

Figure 6-4


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
CapView
Project Name:
Model Name:
SeaStar File:
Press Accept to import a new project or model, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project

CapHelp

Figure 6-5

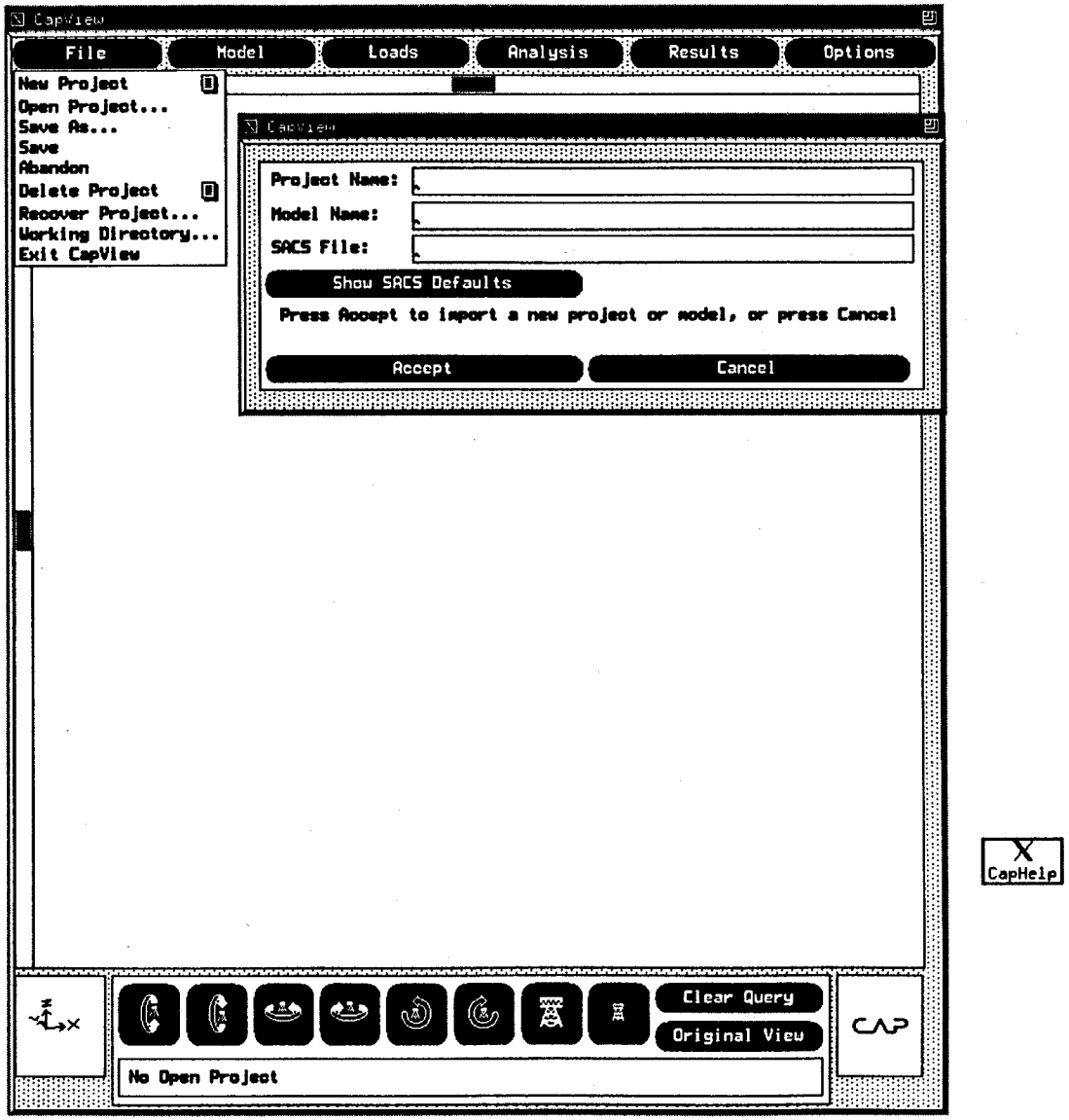

CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
CapView
Project Name:
Model Name:
SACS File:
Show SACS Defaults
Press Accept to import a new project or model, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CapHelp

Figure 6-6

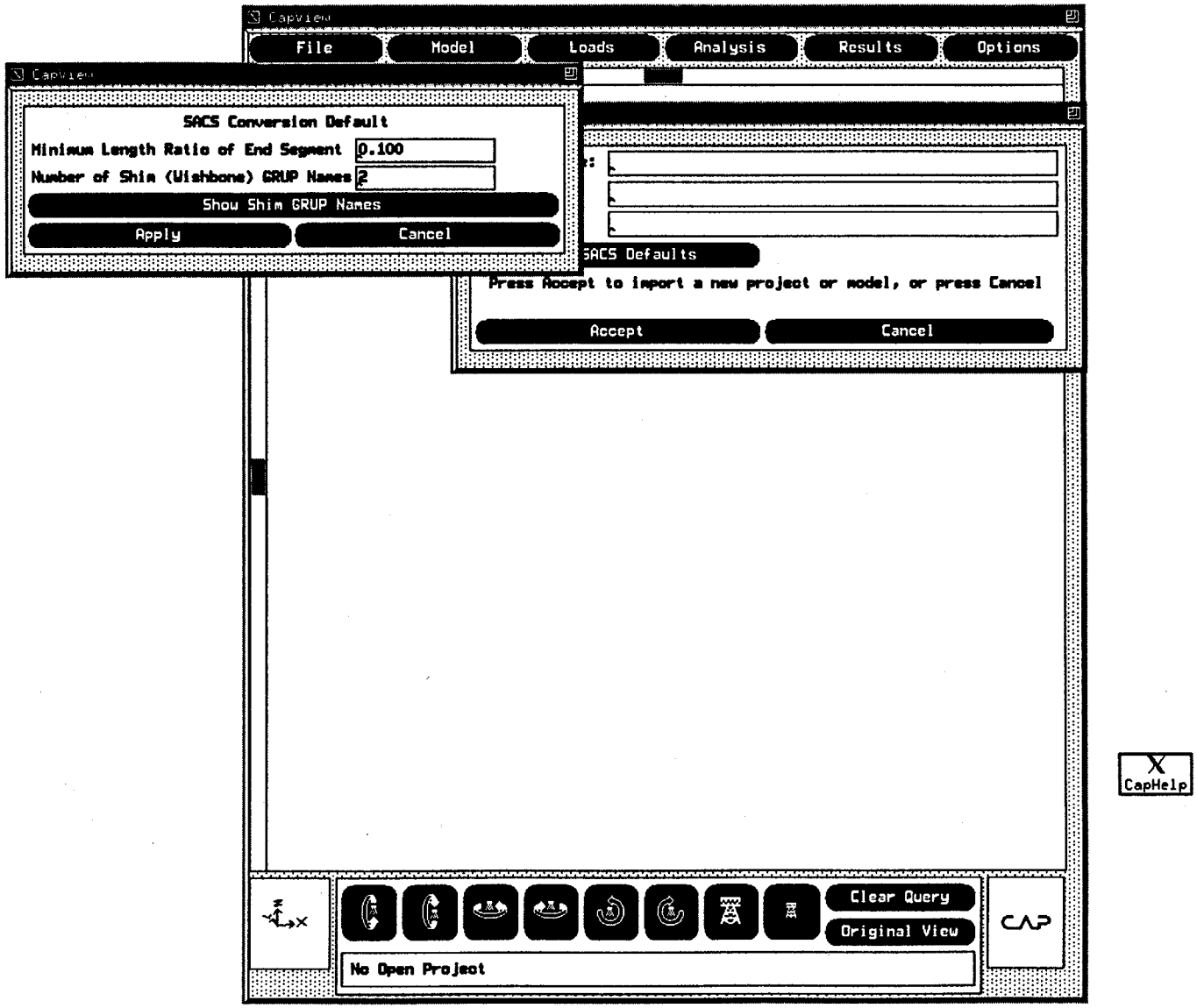

Capview
File
Model
Loads
Analysis
Results
Options
SACS Conversion Default
Minimum Length Ratio of End Segment
0.100
Number of Shim (Wishbone) GRUP Names
2
Show Shim GRUP Names
Apply
Cancel
SACS Defaults
Press Accept to import a new project or model, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CapHelp

Figure 6-7

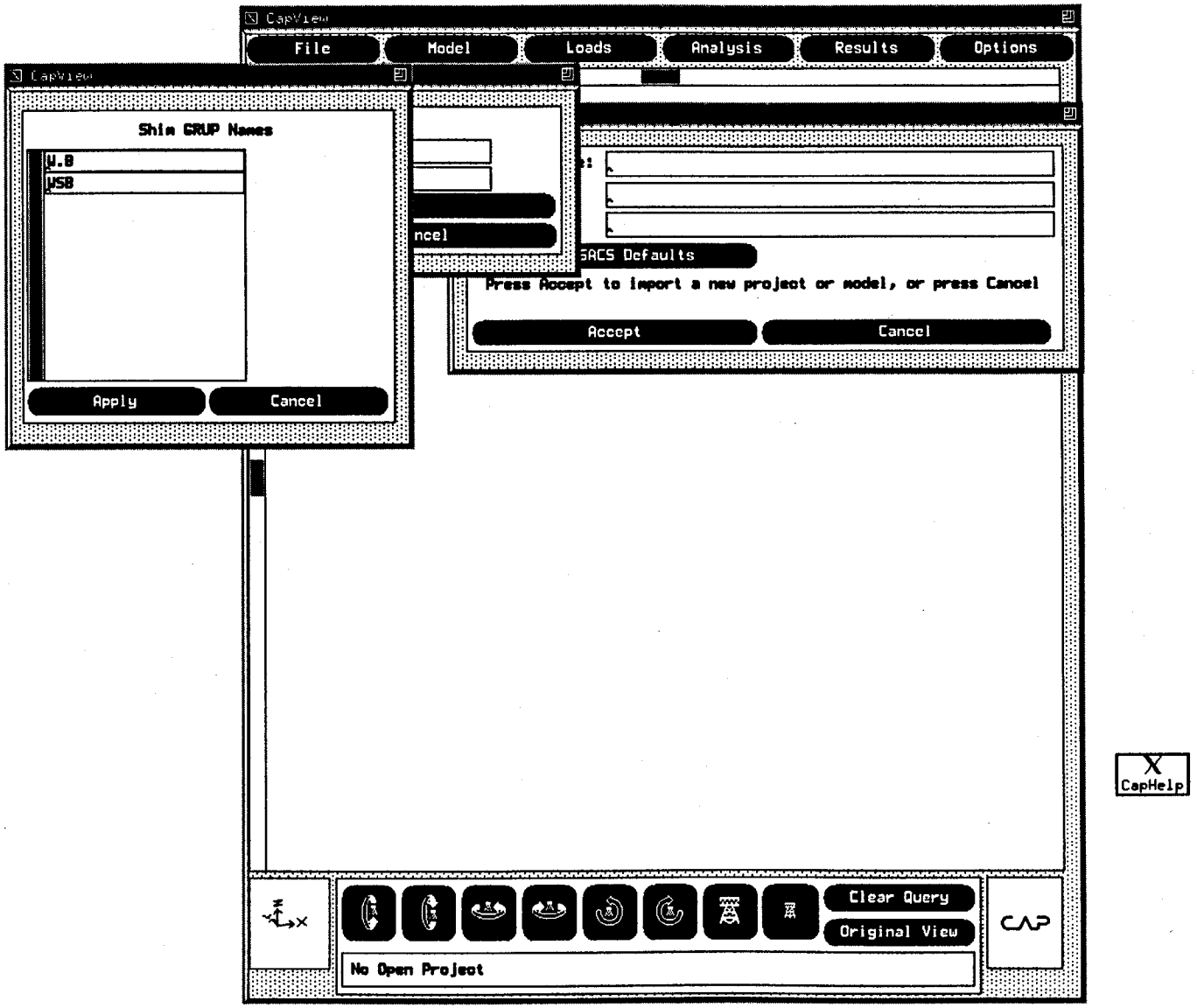
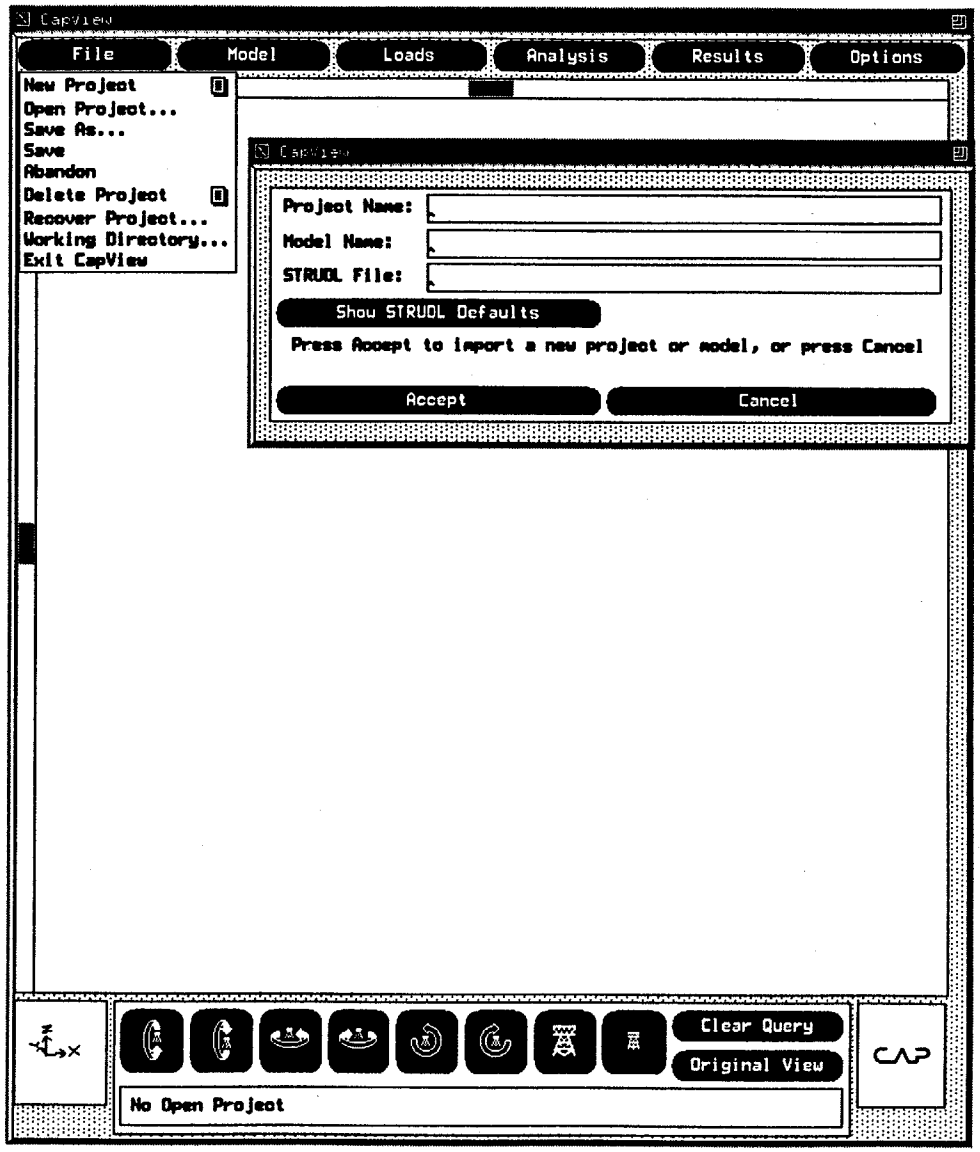

CapView
File
Model
Loads
Analysis
Results
Options
Shim GRUP Names
V.B
VSB
Apply
Cancel
ncel
SACS Defaults
Press Accept to import a new project or model, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP
X
CapHelp

Figure 6-8


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
Project Name:
Model Name:
STRUDL File:
Show STRUDL Defaults
Press Accept to import a new project or model, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project

CapHelp

Figure 6-9

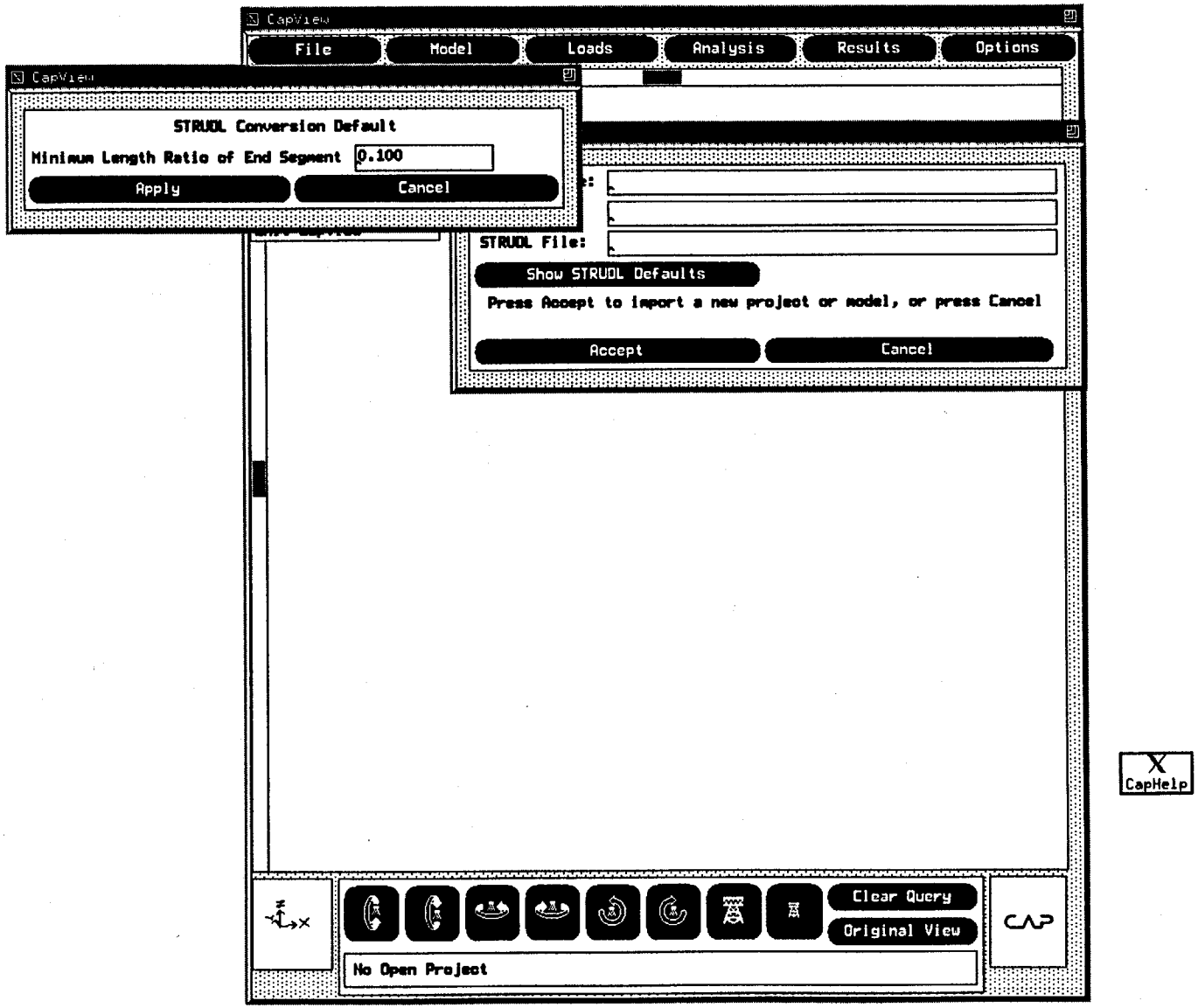

CapView
File
Model
Loads
Analysis
Results
Options
CapView
STRUDL Conversion Default
Minimum Length Ratio of End Segment
0.100
Apply
Cancel
STRUDL File:
Show STRUDL Defaults
Press Accept to import a new project or model, or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CapHelp

Figure 6-10

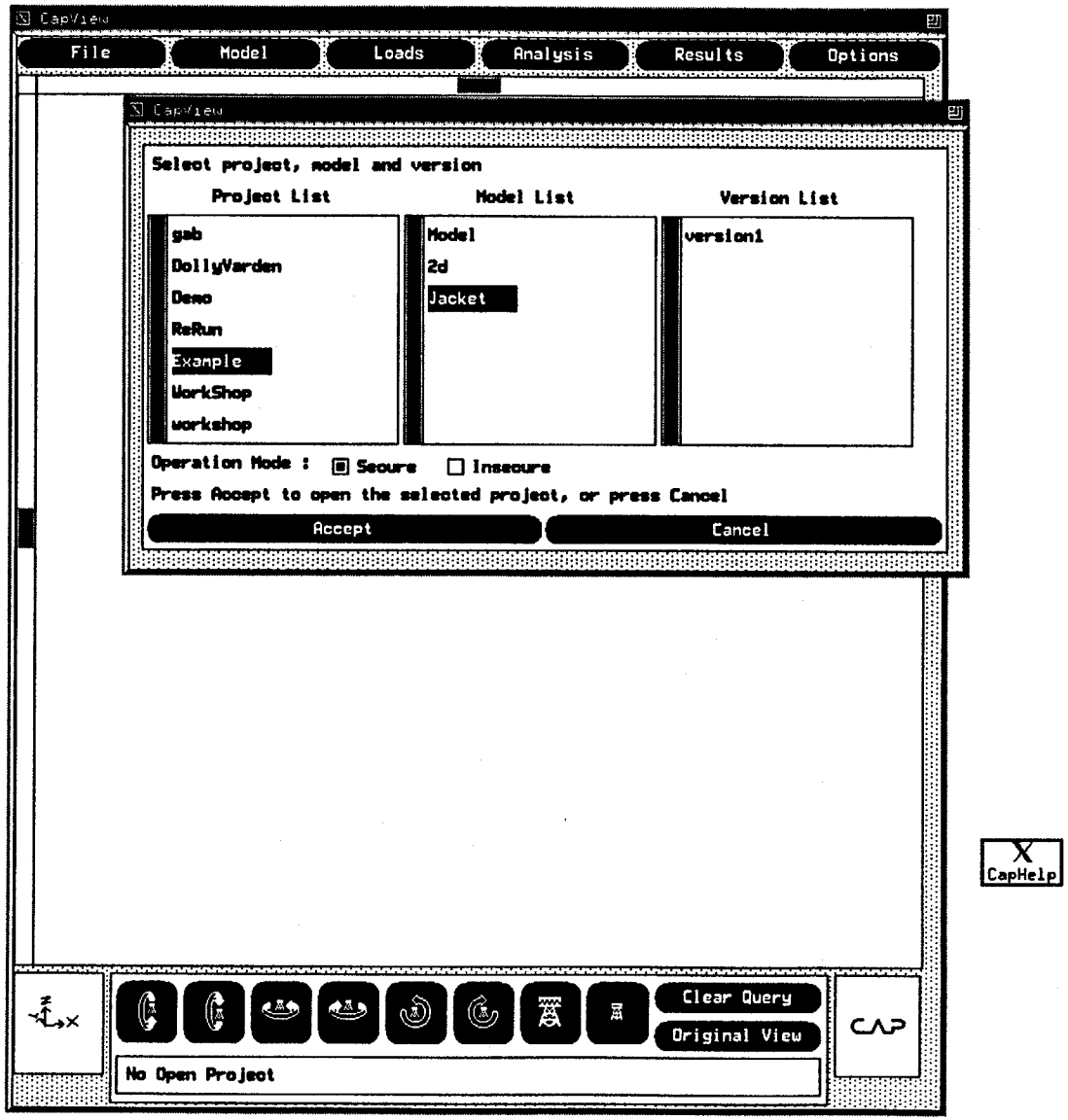

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Select project, model and version
Project List
Model List
Version List
gab
DollyVarden
Demo
ReRun
Example
WorkShop
workshop
Model
2d
Jacket
version1
Operation Mode :
Secure
Insecure
Press Accept to open the selected project, or press Cancel
Accept
Cancel
CapHelp
Clear Query
Original View
No Open Project

Figure 6-11

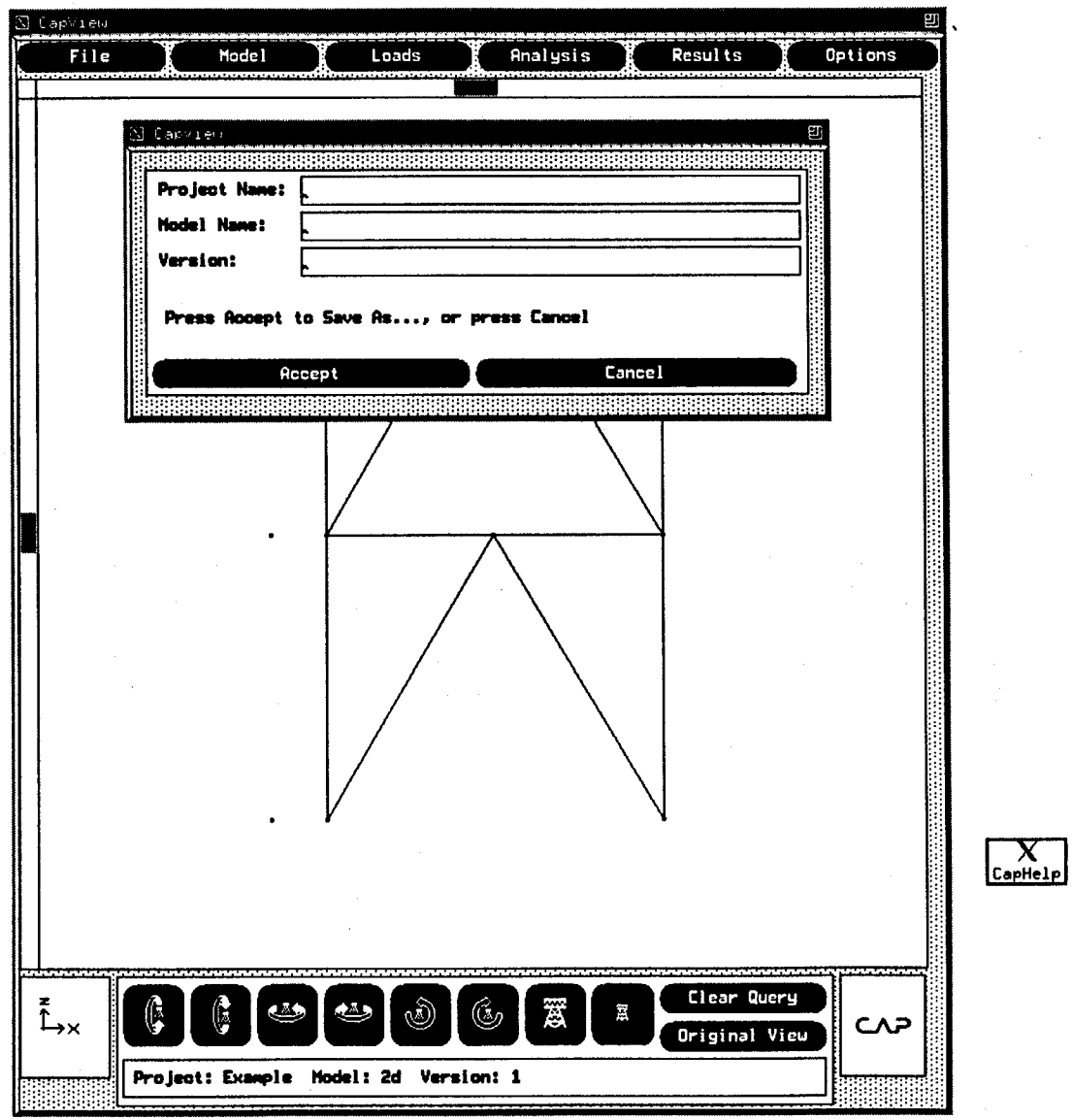

File
Model
Loads
Analysis
Results
Options
Project Name:
Model Name:
Version:
Press Accept to Save As..., or press Cancel
Accept
Cancel
Clear Query
Original View
Project: Example Model: 2d Version: 1
CapHelp

Figure 6 -12

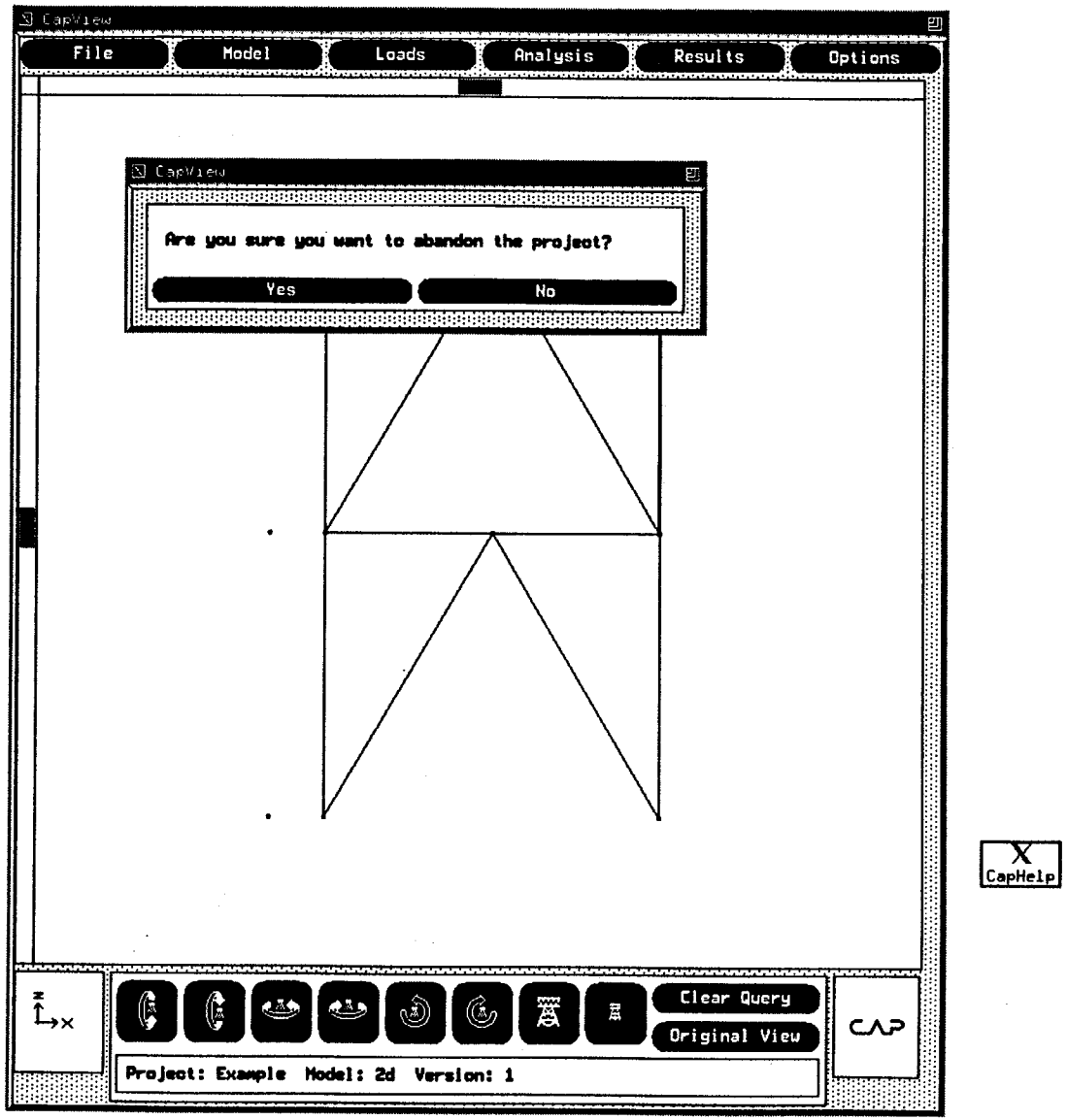
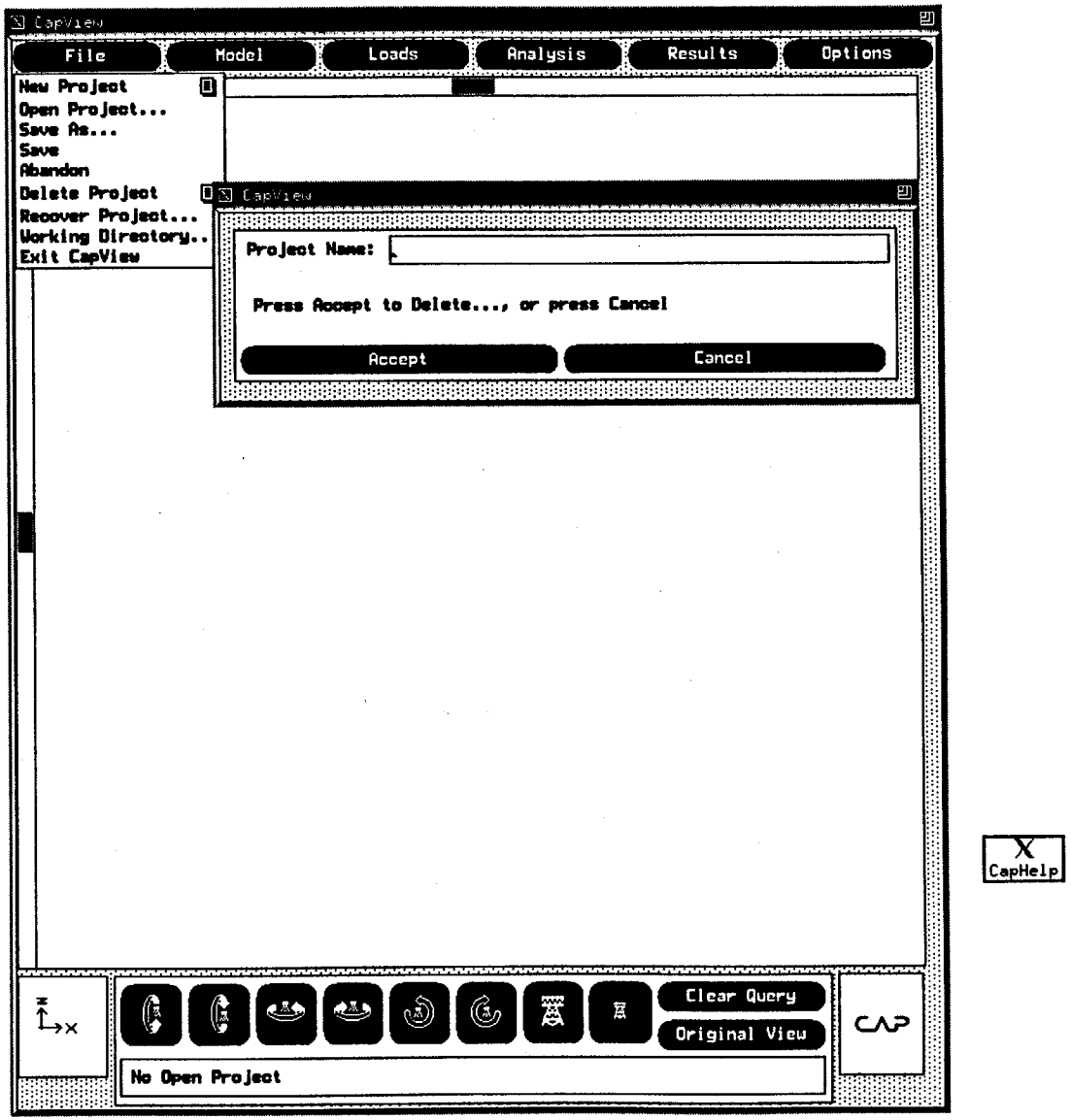

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Are you sure you want to abandon the project?
Yes
No
Clear Query
Original View
CAP
Project: Example Model: 2d Version: 1
X
CapHelp

Figure 6-13


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory..
Exit CapView
CapView
Project Name:
Press Accept to Delete..., or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP

X
CapHelp

Figure 6-14

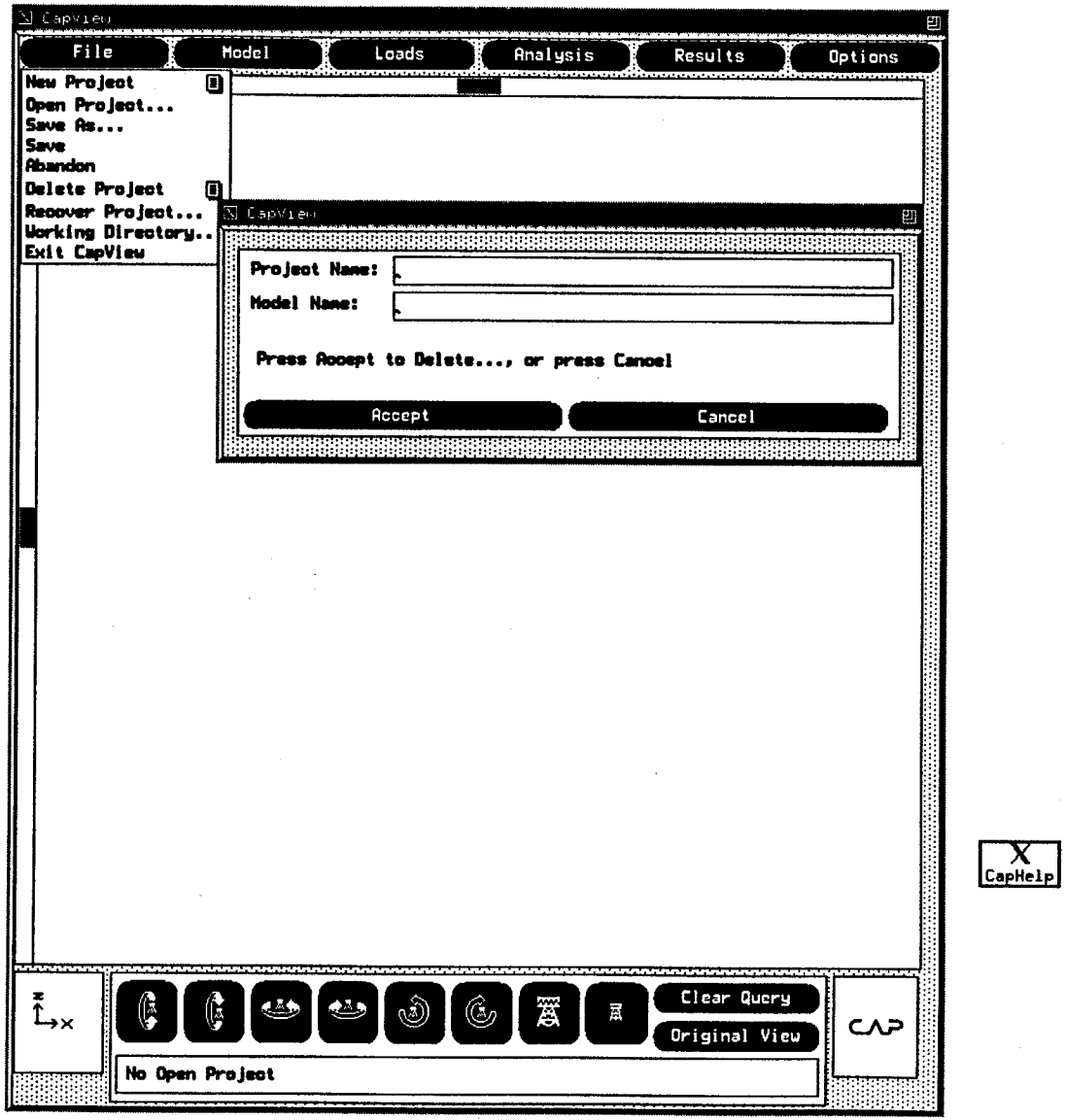
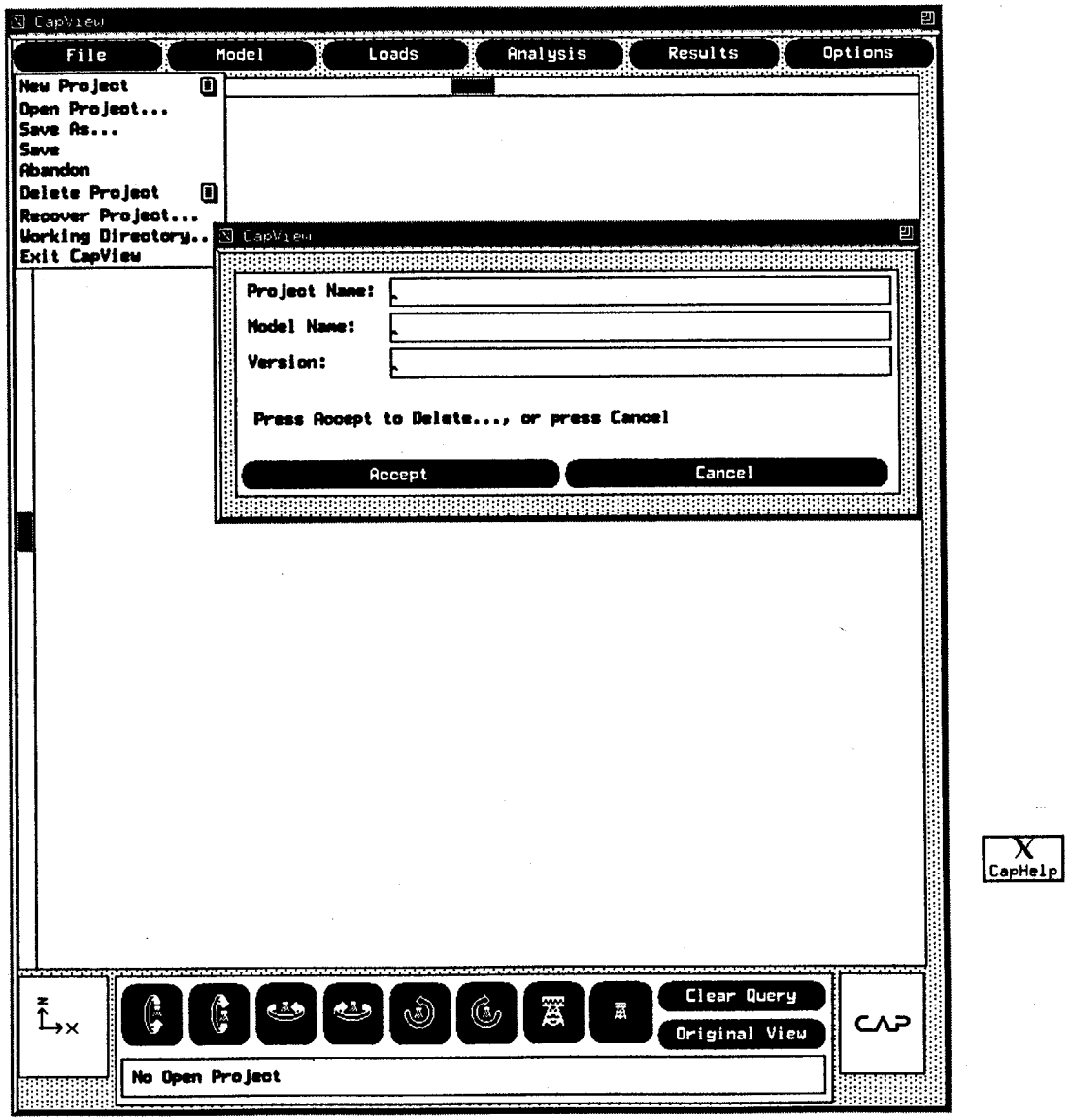
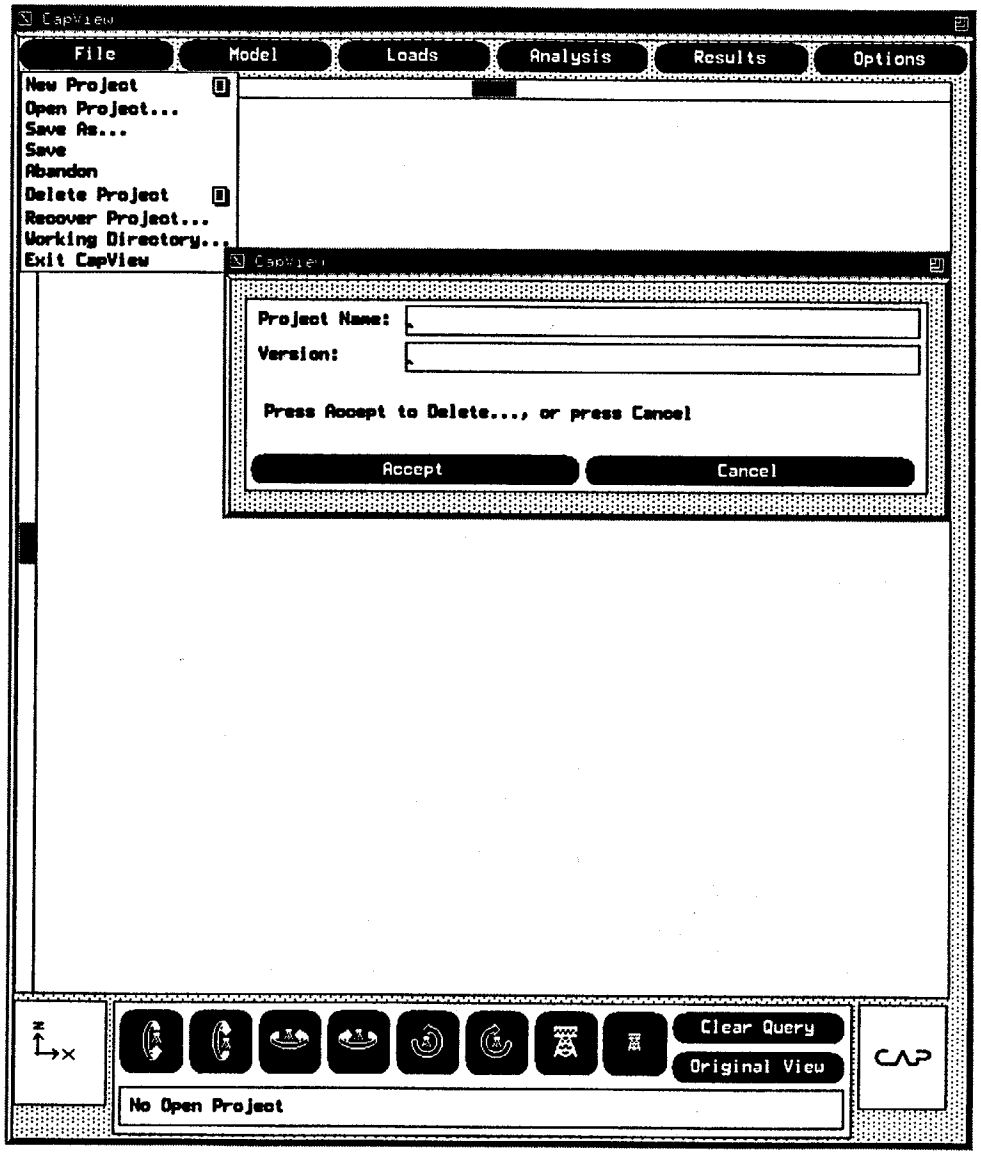

CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory..
Exit CapView
CapView
Project Name:
Model Name:
Press Accept to Delete..., or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP
X
CapHelp

Figure 6-15


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
CapView
Project Name:
Model Name:
Version:
Press Accept to Delete..., or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP

CapHelp

Figure 6-16


CapView
File
Model
Loads
Analysis
Results
Options
New Project
Open Project...
Save As...
Save
Abandon
Delete Project
Recover Project...
Working Directory...
Exit CapView
CapView
Project Name:
Version:
Press Accept to Delete..., or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP

CapHelp

Figure 6-17

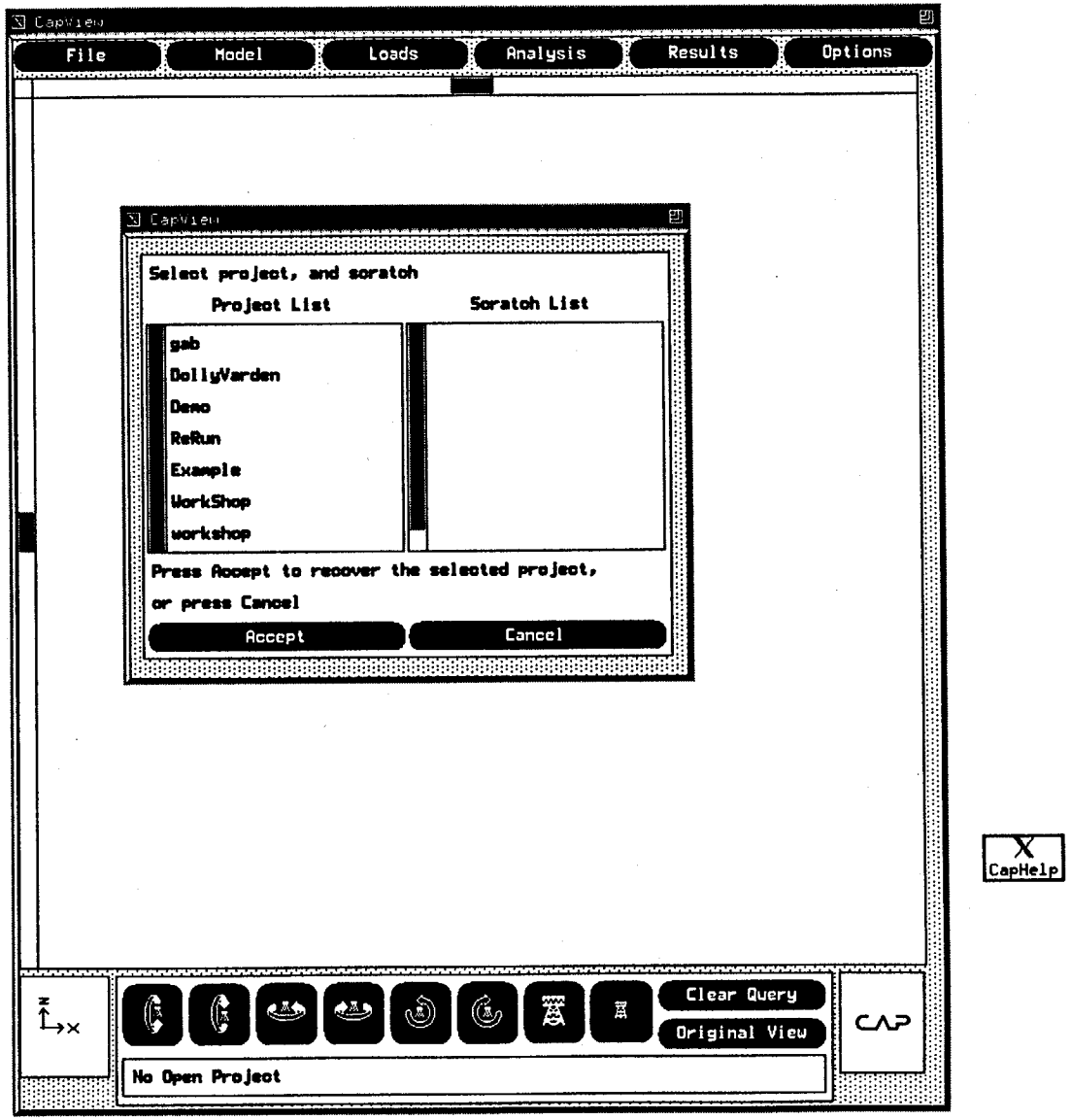

Capview
File
Model
Loads
Analysis
Results
Options
Capview
Select project, and scratch
Project List
Scratch List
gab
DollyVarden
Demo
ReRun
Example
WorkShop
workshop
Press Accept to recover the selected project,
or press Cancel
Accept
Cancel
Clear Query
Original View
No Open Project
CAP
CapHelp

Figure 6-18

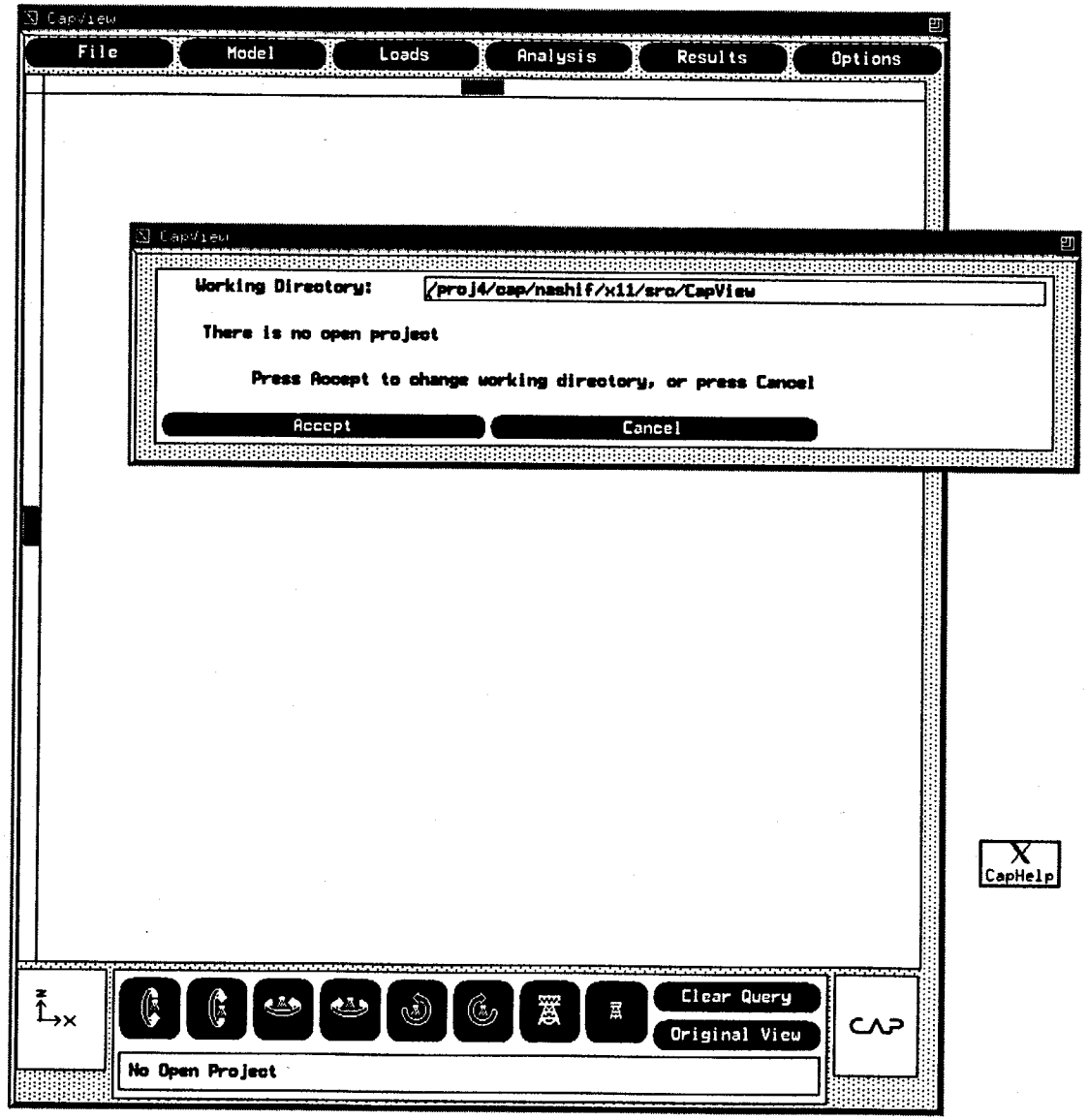

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Working Directory:
/proj4/cap/nashif/x11/src/CapView
There is no open project
Press Accept to change working directory, or press Cancel
Accept
Cancel
CapHelp
Clear Query
Original View
No Open Project

Figure 6-19

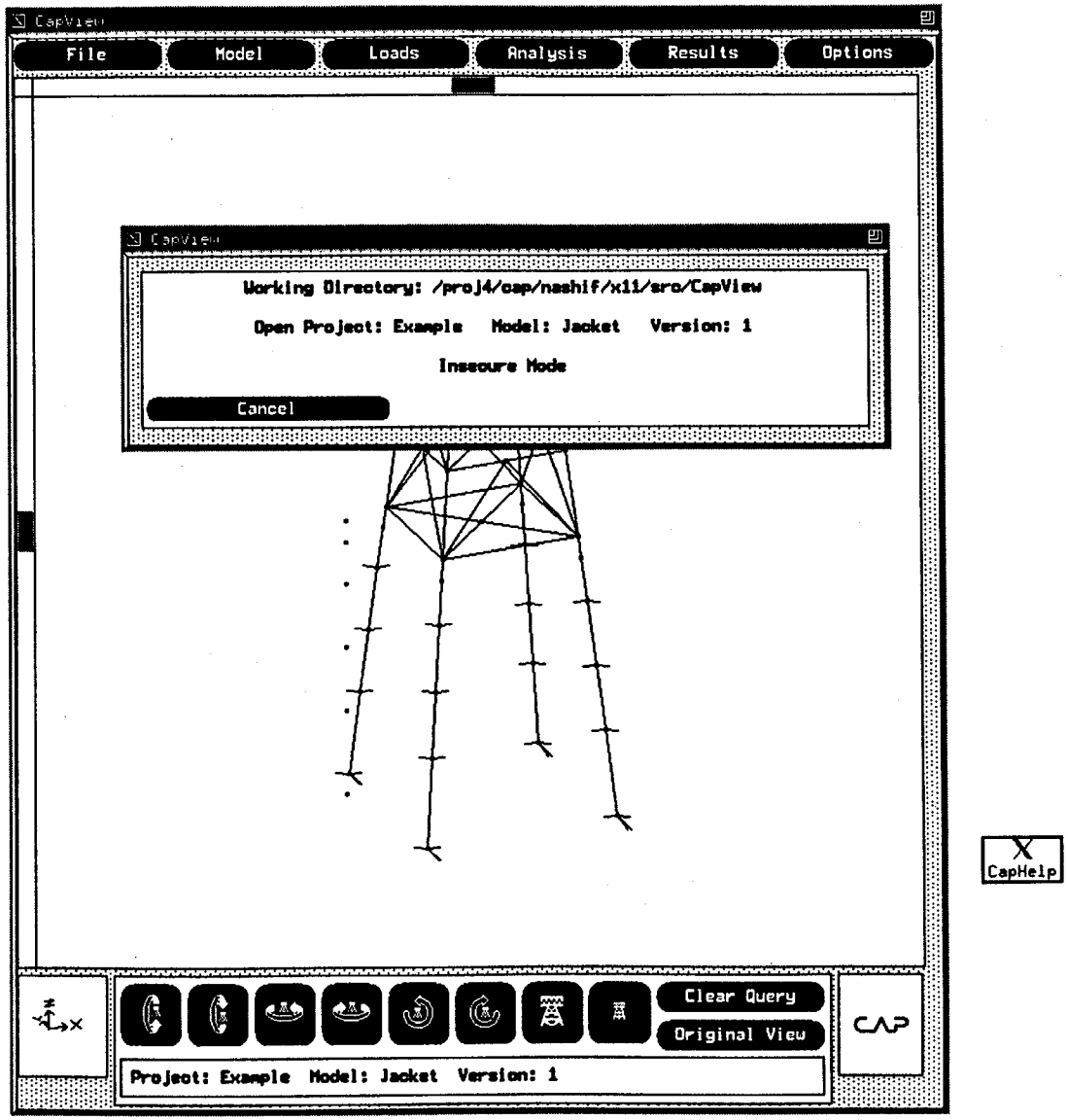

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Working Directory: /proj4/cap/nashif/x11/src/CapView
Open Project: Example Model: Jacket Version: 1
Insecure Mode
Cancel
Clear Query
Original View
Project: Example Model: Jacket Version: 1
CAP
X
CapHelp

Figure 6-20

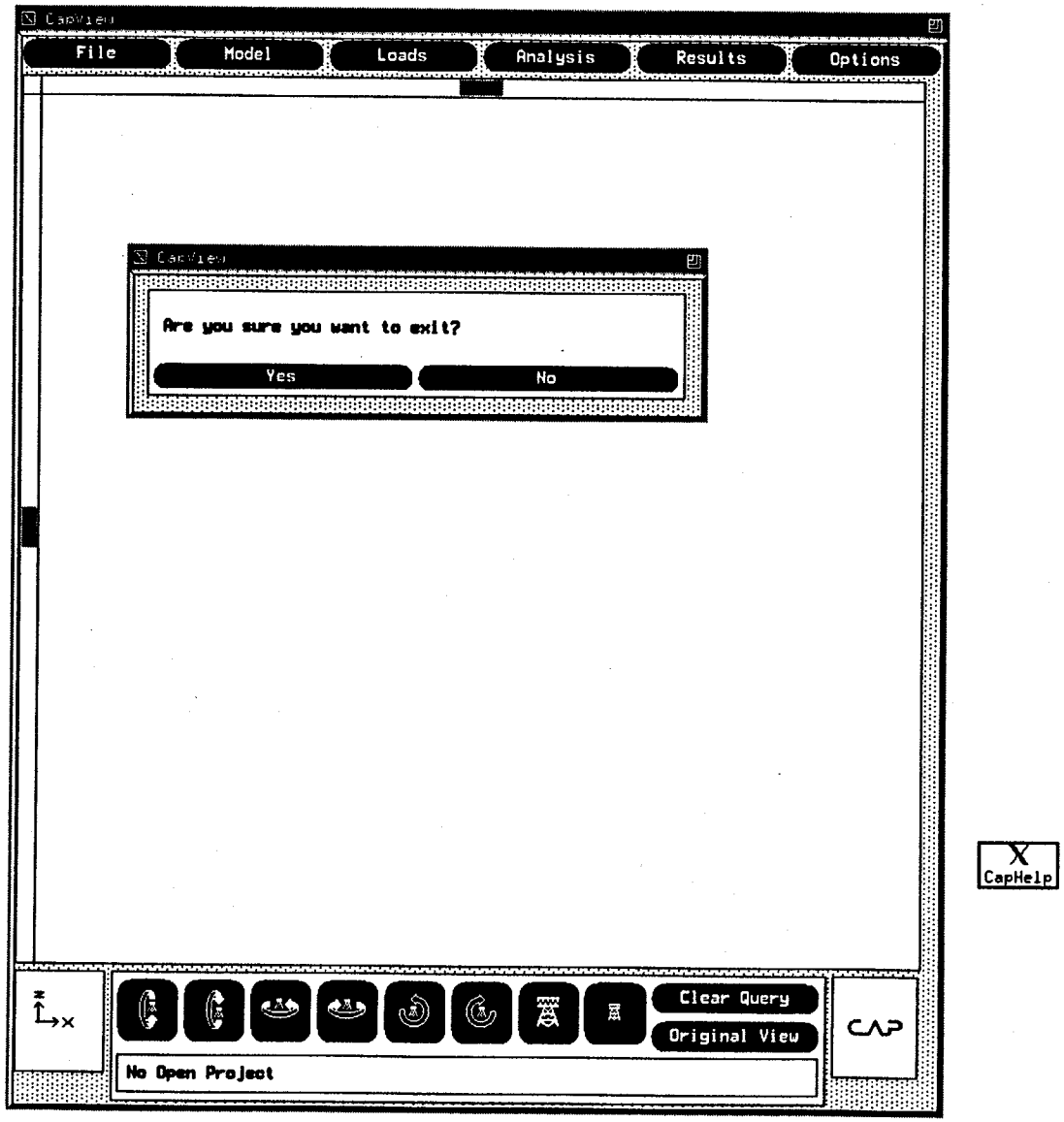

File
Model
Loads
Analysis
Results
Options
Are you sure you want to exit?
Yes
No
Clear Query
Original View
No Open Project
CAP
CapHelp

Figure 6-21

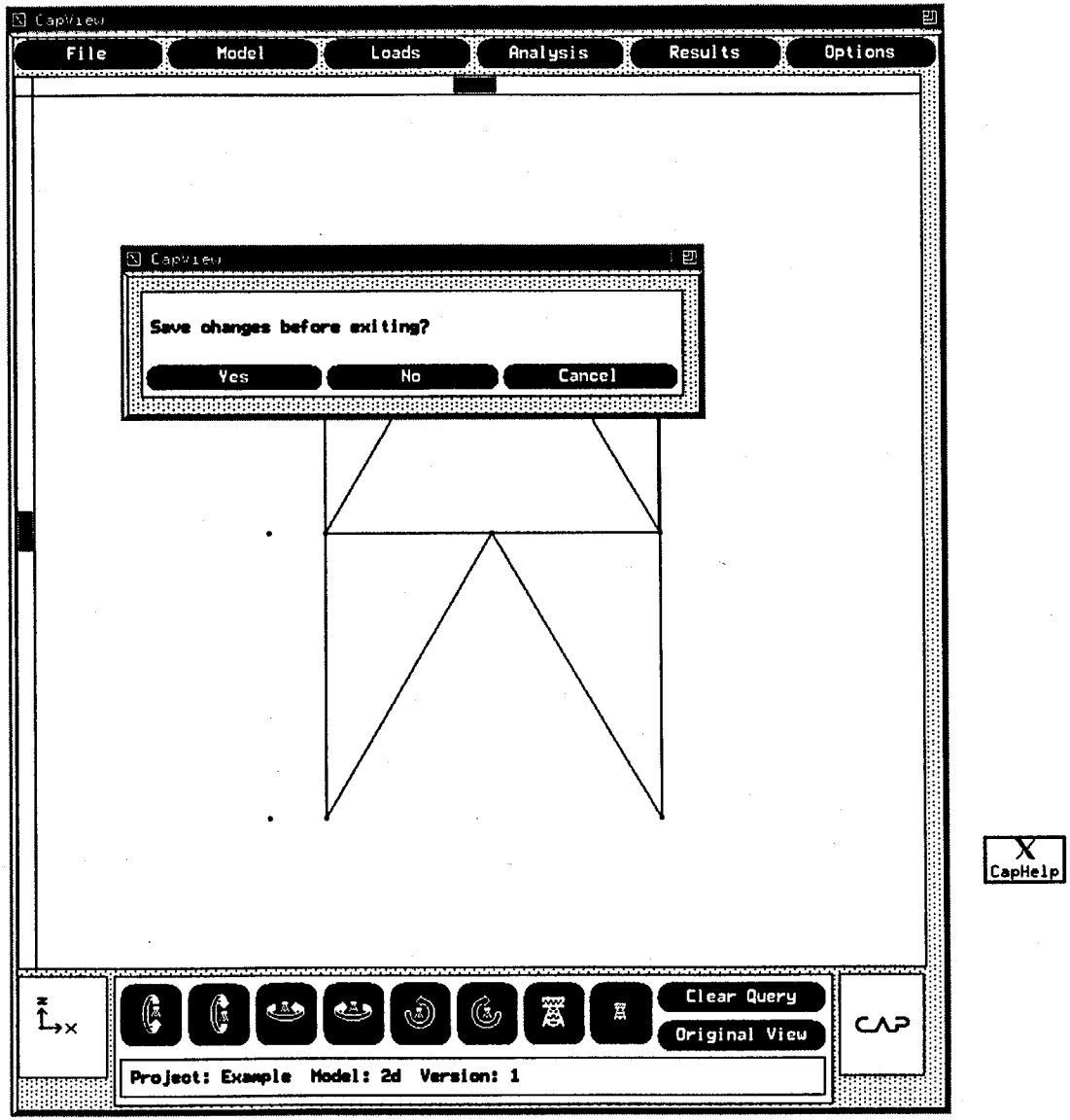

Capview
File
Model
Loads
Analysis
Results
Options
Capview
Save changes before exiting?
Yes
No
Cancel
Clear Query
Original View
Project: Example Model: 2d Version: 1
CapHelp

Figure 6-22

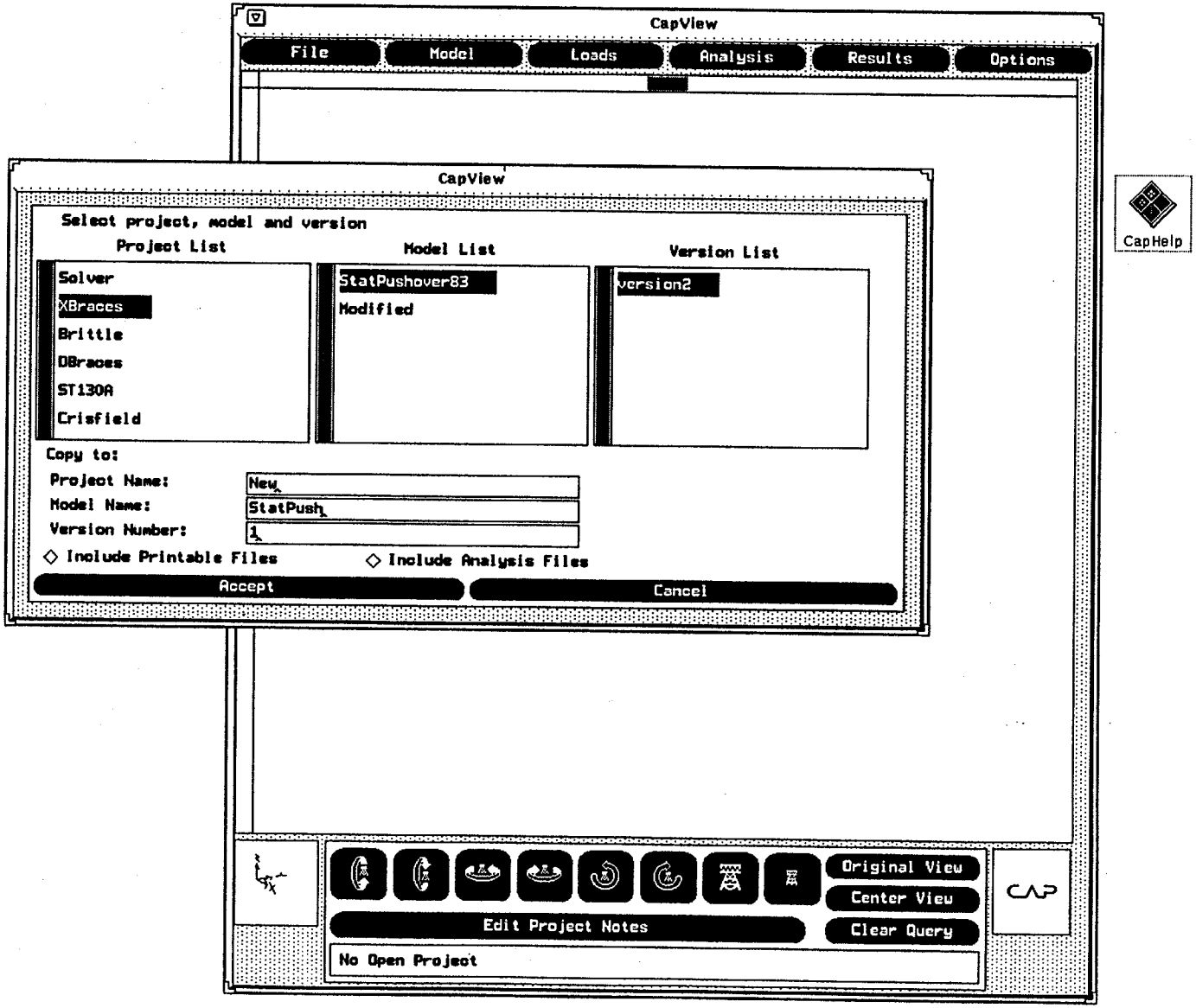

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Select project, model and version
Project List
Model List
Version List
Solver
XBraces
Brittle
DBraces
ST130A
Crisfield
StatPushover83
Modified
Version2
Copy to:
Project Name:
New
Model Name:
StatPush
Version Number:
1
Include Printable Files
Include Analysis Files
Accept
Cancel
Original View
Center View
Edit Project Notes
Clear Query
No Open Project

Figure 6-23

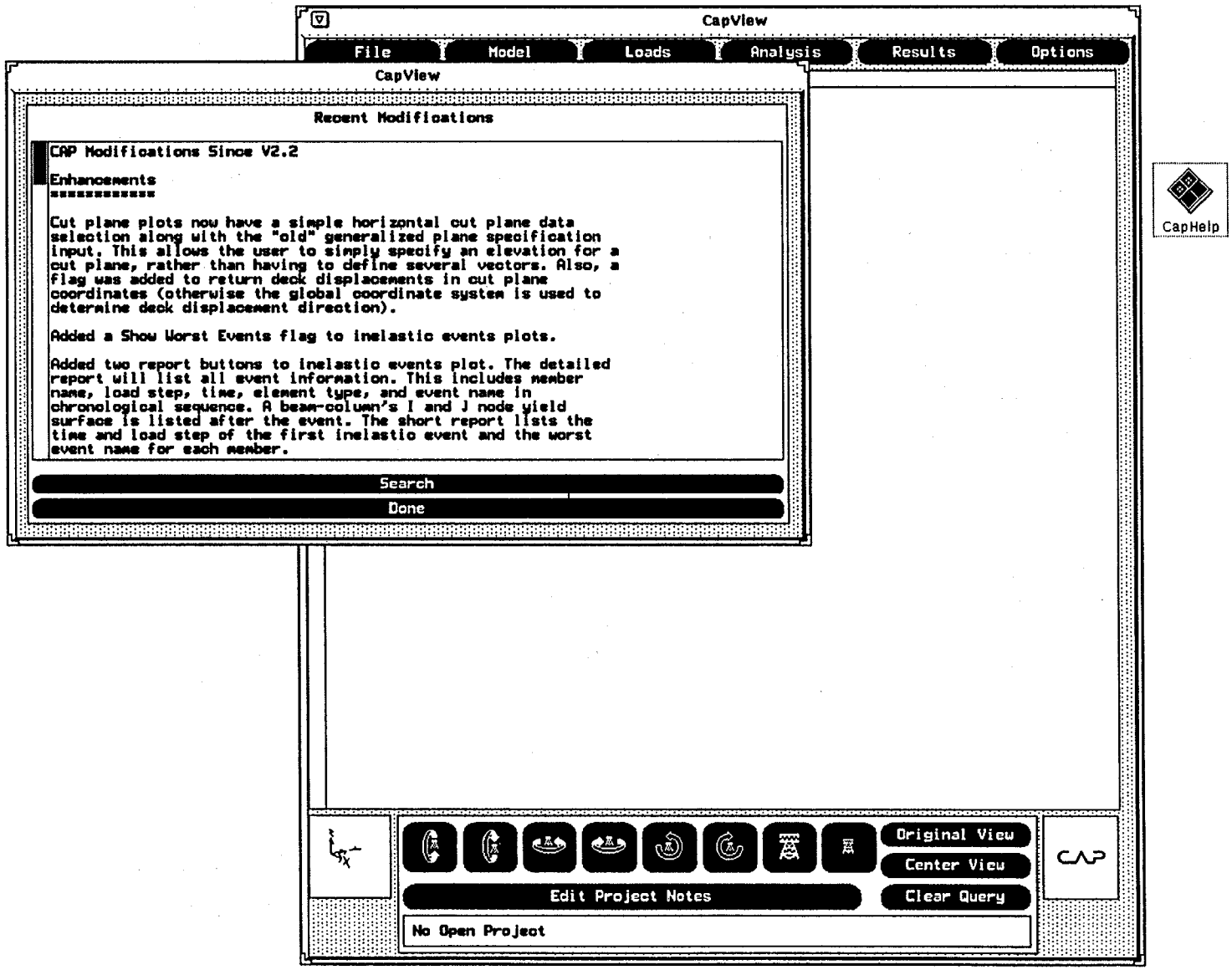

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Recent Modifications
CAP Modifications Since V2.2
Enhancements
==========
Cut plane plots now have a simple horizontal cut plane data selection along with the "old" generalized plane specification input. This allows the user to simply specify an elevation for a cut plane, rather than having to define several vectors. Also, a flag was added to return deck displacements in cut plane coordinates (otherwise the global coordinate system is used to determine deck displacement direction).
Added a Show Worst Events flag to inelastic events plots.
Added two report buttons to inelastic events plot. The detailed report will list all event information. This includes member name, load step, time, element type, and event name in chronological sequence. A beam-column's I and J node yield surface is listed after the event. The short report lists the time and load step of the first inelastic event and the worst event name for each member.
Search
Done
CapHelp
Original View
Center View
Edit Project Notes
Clear Query
No Open Project

Figure 6-24

7.0 OPTIONS MENU

The Options menu contains selections that help the user customize CapView. It also allows the user to create geometric plot files, and to print these and other files created with options on the Results menu. When the Options menu is selected, the following submenu is displayed (Figure 7-1):

Display Defaults...
Display Projection...
Pen Attributes...
Pen Selection...
Inelastic Event Pen Selection...
IR Ranges...
Print Displayed Structure...
Print/View Files...
Help...

7.1 Display Defaults Dialogue Box

The Display Defaults dialogue box is displayed when Display Defaults is chosen from the Options menu (Figure 7-2). The options that may be customized from this dialogue box control the display of objects in CapView. The choices deal with labeling, member display, plot mode, mouse sensitivity, and display units. In this section, default values are shown between angle brackets (<>).

Label Display Defaults

The first set of four check boxes in the Display Defaults dialogue box control labels in the CAP display.

- Shown Id's are SeaStar Id's <off>: This check box toggles between SeaStar and CapView node id's. When checked, the node id's displayed are those from the SeaStar input file, otherwise they are from CapView.
- Node Id's Shown <off>: This check box toggles the displaying of node id's. When checked the node id's will be displayed.
- Member Id's Shown <off>: This check box toggles the displaying of Member id's, when checked the member id's will be displayed.
- Show Member Direction <off>: This check box toggles the displaying of Member direction, when checked an arrow head poiting from the i node to the j node is shown at the middle of the member.
- Nodal Loads Shown <off>: This check box toggles the displaying of nodal load indicators. When checked, an indicator will be displayed on any nodes with a nodal load.

Double Line Mode

The second set of selections in the Display Defaults dialogue box control double line display of Members.

- Double-Line Members <off>: This check box toggles Double Line Member mode. In Double-Line Member mode the thickness of the members displayed on the screen is proportional to their diameter (or to the "Display Diameter" property of non-tubular members). In addition, the members are drawn from back to front to give a semi-rendered appearance to the structure. Note that in Double-Line mode, operations that change the view of the structures take more time than they normally would. For large structures the additional time can be a minute or more.
- Width Scale <1.000>: This edit field is used to enter the ratio of the thickness of the displayed Double-Line members in screen pixels to the actual diameter in feet. The Width Scale is only used for Double-Line member mode. If while in Double-Line mode the members appear too thin or are lines, increase the Width Scale. If the members appear too thick, decrease the Width Scale.

Member Plot Mode

The third set of selections in the Display Defaults dialogue box control the Plot Mode for Members.

- Default Mode <on>: This radio button sets the plot mode to default. The color of the members is determined by the values set in the Pen Selection dialogue box.
- IR-Plot Mode <off>: This radio button sets the plot mode to Interaction Ratio mode. The interaction ratio values are based on the API 16th edition

member strength and hydrostatic collapse and the AISC 8th edition member strength code checks. When this selection is made, the color of the members is determined by the values set in the Interaction Ratio Range dialogue box.

- Inelastic Event Mode <off>: This radio button sets the plot mode to Inelastic Event Mode. When this selection is made, the color of the members is determined by the values set in the Interaction Ratio Range dialogue box.

Mouse Sensitivity

The fourth group of selections in the Display Defaults dialogue box are related to mouse sensitivity. If you have difficulty picking nodes with the mouse, try increasing the node sensitivity.

- Node Sensitivity <2>: This spin button is used to set the sensitivity of the mouse for picking nodes. This value sets the distance in pixels from the cursor where a node is highlighted.
- Member Sensitivity <2>: This spin button is used to set the sensitivity of the mouse for picking members. This value sets the distance in pixels from the cursor where a member is highlighted.

Display Units

The fifth group of selections in the Display Defaults dialogue box controls the display of units. There are three standard selections and a fourth where the user can customize the values. Note that unit conversion works only on members of the "Generic" type and on nodes. The rest of the members work with consistent units. The Units radio button list is used to select which units will be used in the display. Changing the units will cause the values to be converted and new labels applied.

- SI Units <off>: Use International Standard units.
- English Units (Kip, Ft) <on>: Use English units with feet.
- English Units (Kip, Inch) <off>: Use English units with inches.
- Customizable Units <off>: Use the units which the user has set in the Customizable Units dialogue box.

Display Units: When this button is pressed then a display box will show the selected units and their relationship to English Units (Figure 7-3). When Customizable Units is selected then the Customizable Units dialogue box is displayed (Figure 7-4).

The Customizable Units dialogue box is used to name the units and to set the factor from each unit to English Units. For each, there is an edit field for the factor, an edit field for the unit name and the English units name. After the factors and the names have been edited, the Apply button will implement the changes, and the Cancel button will use the values that were in effect. The Default button will reset all of the values to their defaults, which are equivalent to English Units.

Apply, Default and Cancel

The last group of buttons in the Display Defaults dialogue box are the Apply, Default and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Display Defaults dialogue box. This button must be selected to make any changes effective.

- Default: Selecting this button will reset the selections in the Display Defaults dialogue box to their default values. The Dialogue box is not closed. To make the defaults active, choose Apply after this button.
- Cancel: Selecting this button will close the Display Defaults dialogue box and use the settings that were already in effect. Any changes made will be lost.

7.2 Display Projection Dialogue Box

The Display Projection dialogue box is displayed when Display Projection is chosen from the Options menu (Figure 7-5). From this dialogue box the user selects the projection of the structure in the display. In this section, default values are shown between angle brackets (<>).

Projection Type

The first group is a set of radio buttons used to choose the projection type for the display.

- 3-D Parallel Projection <on>: 3D representation using Parallel Projection.
- 3-D Perspective Projection <off>: 3D representation using Perspective Projection.
- 2-D Slice Defined by 3 Nodes <off>: Show a flat slice of the structure using a plane defined by three nodes.
- 2-D Slice Defined by 2 Members <off>: Show a flat slice of the structure using a plane defined by two members.

A second group will be determined by which Projection Type radio button is selected.

When 3-D Parallel Projection is checked (Figure 7-5):

- Rotational Angle Increment <22.500>: This edit field sets the amount of rotation each of the rotational controls at the bottom of the screen will apply to the structure when they are pressed.
- Magnification Scale <1.250>: This edit field sets the amount of magnification or reduction the zoom buttons at the bottom of the screen will apply to the structure when they are pressed.

When 3-D Perspective Projection is checked (Figure 7-6):

- Rotational Angle Increment <22.500>: This edit field sets the amount of rotation each of the rotational controls at the bottom of the screen will apply to the structure when they are pressed.
- Zoom Distance <2400.000>: This edit field sets how much the apparent distance from the center of the structure to the view screen will be increased or decreased when the zoom buttons at the bottom of the screen are used.

When 2-D Slice Defined by 3 Nodes is checked (Figure 7-7):

- Node ID's: There are three edit fields to enter the ID's of the nodes that determine the plane of the slice.
- Get Nodes From List: This button retrieves the three nodes from the selected node list. Nodes are placed on the list by holding the shift key down and clicking the left mouse button when the node is highlighted.
- Slice Thickness <0.010>: This edit field sets the thickness, in feet, of the slice that is displayed. All nodes and members that are within this distance from the plane of the slice will be displayed.
- Magnification Scale <1.250>: This edit field sets the amount of magnification or reduction the zoom buttons at the bottom of the screen will apply to the structure.

When 2-D Slice Defined by 2 Members is checked (Figure 7-8):

- Member ID's: There are four edit fields to enter the Group and ID of the two members that determine the plane of the slice.

- Get Members From List: This button retrieves the three Members from the selected Member list. Members are placed on the list by holding the shift key down and clicking the left mouse button when the member is highlighted.
- Slice Thickness <0.010>: This edit field sets the thickness, in feet, of the slice that is displayed. All nodes and members that are within this distance from the plane of the slice will be displayed.
- Magnification Scale <1.250>: This edit field sets the amount of magnification or reduction the zoom buttons at the bottom of the screen will apply to the structure.

Apply, Default and Cancel

The last group of buttons in the Display Projection dialogue box are the Apply, Default and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Display Projection dialogue box. This button must be selected to make any changes effective.
- Default: Selecting this button will reset the selections in the Display Projection dialogue box to their defaults. The Dialogue box is not closed. To make the defaults active, choose Apply after this button.
- Cancel: Selecting this button will close the Display Projection dialogue box and use the settings that were already in effect. Any changes made will be lost.

7.3 Pen Attributes Dialogue Box

The third selection on the Options Menu will display the Pen Attributes dialogue box. In the Pen Attributes dialogue box the attributes for each of sixteen pens may be set. The pens emulate the pens on a plotter and are used to set the characteristics of anything drawn on the display. In this section, default values are shown between angle brackets (<>).

There are two Pen Attributes dialogue boxes, one for monochrome systems and one for color systems.

Monochrome

For monochrome systems, there are three sets of radio buttons used to set pen attributes (Figure 7-9).

- P1 - P16: These radio buttons are for the sixteen emulated pens. To set the attributes for a pen, check its box and make the changes. Next to each of the pen radio buttons is a box that contains a line drawn in the current attributes of that pen or a box that contains the color of that pen depending on the current display mode.
- Line Width: These four radio buttons are used to set the line width of the pen whose attributes are being set. There are samples of the widths next to each button.
- Line Style: These four radio buttons are used to set the line style of the pen whose attributes are being set. There are samples of the styles next to each button.

Color

For color systems, the attributes of the pens are changed by assigning a color to each pen (Figure 7-10).

- P1 - P16: These radio buttons are for the sixteen emulated pens. To set the attributes for a pen, check its box and make the changes. Next to each of the pen radio buttons is a box that is filled with the current color of that pen.
- Red, Blue, Green: These sliders are used to adjust the color content of the pen whose attributes are being set. Each of these color sliders shows the contribution that color adds to the pen. The resulting color is shown as the sliders are adjusted.
- Color <on>: Leave this radio button on to display color on a color system.
- Monochrome <off>: This radio button is included to force a color system to run in monochrome. If monochrome is selected then the color Pen Attributes dialogue box will have the same choices as the monochrome Pen Attributes dialogue box.

Apply, Default and Cancel

The last group of buttons in the Pen Attributes dialogue box are the Apply, Default and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Pen Attributes dialogue box. This button must be selected to make any changes effective.
- Default: Selecting this button will reset the selections in the Pen Attributes dialogue box to their defaults. The Dialogue box is not closed. To make the defaults active, choose Apply after this button.

- Cancel: Selecting this button will close the Pen Attributes dialogue box and use the settings that were already in effect. Any changes made will be lost.

7.4 Pen Selection Dialogue Box

The fourth selection on the Options Menu will display the Pen Selection dialogue box (Figure 7-11). The Pen Selection dialogue box is used to assign one of the sixteen pens to each of the properties of the screen display. For each of the properties there are sixteen pens (P1-P16), one of which is selected. There is a sample of each of the sixteen pens in two rows across the top of the dialogue box. In this section, default values are shown between angle brackets (<>).

- Canvas <P5>: Pen used for drawing the background canvas. This selection is not used for monochrome display.
- Default Pen <P4>: Pen used for drawing normal Nodes and Members.
- Mouse Highlight <P2>: Pen used for drawing Nodes or Members that have been highlighted by the mouse.
- Query Highlight <P1>: Pen used for drawing Nodes and Members that have been selected by a query.
- Node ID <P1>: Pen used for writing the Node ID labels. This selection is not used for monochrome display.
- Member ID <P1>: Pen used for the writing Member ID labels. This selection is not used for monochrome display.
- Border <P4>: Pen used for drawing the border of Members when Double Line mode is in effect. This selection is not used for monochrome display.
- Fill <P2>: Pen to used for filling the area of Members when Double Line mode is in effect. This selection is not used for monochrome display.

Apply, Default and Cancel

The last group of buttons in the Pen Selection dialogue box are the Apply, Default and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Pen Selection dialogue box. This button must be selected to make any changes effective.
- Default: Selecting this button will reset the selections in the Pen Selection dialogue box to their defaults. The Dialogue box is not closed. To make the defaults active, choose Apply after this button.
- Cancel: Selecting this button will close the Pen Selection dialogue box and use the settings that were already in effect. Any changes made will be lost.

7.5 Inelastic Event Pen Selection Dialogue Box

The fifth selection on the Options Menu will display the Inelastic Events Pen Selection dialogue box (Figure 7-12). The Inelastic Event Pen Selection dialogue box is used to assign one of the sixteen pens to each of the Inelastic Event types on the screen display. For each of the Inelastic Event types there are sisxteen pens (P1-P16), one of which is selected. There is a sample of each of the sixteen pens in two rows across the top of the dialogue box. In this section, default values are shown between angle brackets (<>).

- Elastic <P8>: Pen used to show members with no event (i.e. in the linear elastic state).
- Strut Buckling <P4>: Pen used to show strut members in buckling state.
- Strut Residual <P12>: Pen used to show strut members in residual strength state.
- Strut Reloading <P2>: Pen used to show strut members reloading in tension or compression after buckling events.
- Strut Plastic Tension/Plastic Nonlinear Truss <P9>: Pen used to show strut members in a plastic tension state. Also used to show nonlinear truss members in a plastic tension or compression state.
- Beam Column Initial Yield <P11>: Pen used to show beam column members that are in a yield state, and have not yet reached a fully plastic state.
- Beam Column Fully Plastic <P3>: Pen used to show beam column members which are in a fully plastic state.
- Fracture <P1>: Pen used to show Strut, Nonlinear Truss and Beam Column during and after fracturing.

Apply, Default and Cancel

The last group of buttons in the Inelastic Event Pen Selection dialogue box are the Apply, Default and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Inelastic Event Pen Selection dialogue box. This button must be selected to make any changes effective.
- Default: Selecting this button will reset the selections in the Inelastic Event Pen Selection dialogue box to their defaults. The Dialogue box is not closed. To make the defaults active, choose Apply after this button.
- Cancel: Selecting this button will close the Inelastic Event Pen Selection dialogue box and use the settings that were already in effect. Any changes made will be lost.

7.6 Interaction Ratio Ranges Dialogue Box

Choosing IR Ranges from the Option menu brings up the Interaction Ratio Ranges dialogue box (Figure 7-13). This allows setting ranges used in Interaction Ratio mode. There are six Interaction Ratio ranges (R1-R6). For each Interaction Ratio range, one of the sixteen pens is selected and a range of Interaction Ratio values is assigned. There is a sample of each of the sixteen pens in two rows across the top of the dialogue box. In this section, default values are shown between angle brackets (<>).

Selected Interaction Ratio Range

These radio buttons select the Interaction Ratio range whose pen and range are to be defined.

- R1 <P1> <0.000 - 0.200>: Selecting this radio button will allow interaction Ratio range 1 to be set.
- R2 <P2> <0.200 - 0.400>: Selecting this radio button will allow interaction Ratio range 2 to be set.
- R3 <P3> <0.400 - 0.600>: Selecting this radio button will allow interaction Ratio range 3 to be set.
- R4 <P4> <0.600 - 0.800>: Selecting this radio button will allow interaction Ratio range 4 to be set.
- R5 <P5> <0.800 - 1.000>: Selecting this radio button will allow interaction Ratio range 5 to be set.
- R6 <P6> <1.000 - Infinity>: Selecting this radio button will allow interaction Ratio range 6 to be set.

After one of the Interaction Ratio range radio buttons has been selected its setting may be adjusted.

- P1 - P16: The pen radio check boxes assign the pen to be used for members that fall within the Interaction Ratio range that is being set.
- Min Value: The minimum Interaction Ratio value for the range.
- Max Value: The maximum Interaction Ratio value for the range.

The Interaction Ratio ranges R1 - R6 must be in increasing order. The minimum value for R1 is always zero, the maximum value for R6 is always infinity. When a value for a range is changed, the corresponding value in the adjacent range is automatically changed. In other words, if R2's maximum value is set to 0.300 the minimum value for R3 will be updated to 0.300 automatically.

Apply, Default and Cancel

The last group of buttons in the Interaction Ratio dialogue box are the Apply, Default and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Interaction Ratio dialogue box. This button must be selected to make any changes effective.
- Default: Selecting this button will reset the selections in the Interaction Ratio dialogue box to their defaults. The dialogue box is not closed. To make the defaults active, choose Apply after this button.
- Cancel: Selecting this button will close the Interaction Ratio dialogue box and use the settings that were already in effect. Any changes made will be lost.

7.7 Print Displayed Structure Dialogue Box

The Print Displayed Structure choice on the Options menu will present the user with the Print Displayed Structure dialogue box (Figure 7-14). This dialogue box captures the image of the structure as it is currently shown on the screen. Whatever changes have been made to the structure through the Options menu or elsewhere will be used in the captured image file. In this section, default values are shown between angle brackets (<>).

- Title: This edit field allows the user to add a title to the plot. The project name, model and version are automatically added to the plot, so the title can be used to identify the specific nature of this plot.
- File Name: The plots are saved to a file which is named in this edit box. The extension .prt will be added.
- Overwrite File <on>: This radio button is selected when the user wants the new plot file to replace any existing file with the same name.
- Append To File <off>: This radio button is selected when the user wants the plot file to be appended to any existing file with the same name. This can be used to create a file with containing multiple plots.

Apply and Cancel

The last group of buttons in the Print Displayed Structure dialogue box are the Apply and Cancel buttons.

- Apply: Selecting this button will create a file with the given name and close the Print Displayed Structure dialogue box.
- Cancel: Selecting this button will close the Print Displayed Structure dialogue box without creating a file.

7.8 Print / View Files Dialogue Box

The Print / View Files dialogue box is used to print or view one of the plot files that has already been saved using the Print Displayed Structure dialogue box (Figure 7-15). The user has the choice of viewing the file on the screen, creating a Postscript or HPGL file, or sending one of the PostScript or HPGL files to the appropriate printing device.

- File Names: This scrollable list contains all of the .prt and .tbl files that are in the current version's directory. Files with a .prt extension are plots and files with a .tbl extension are text reports. Next to each file name there is a checkbox which should be checked to select the file. Only checked files will be printed.

The Output File Type radio buttons select the destination of the plot file.

- Screen: This radio button is selected to send the plot file to the screen. This selection does not produce hardcopy.
- Postscript: This radio button is used to produce monochrome Postscript printer files. The CapView monochrome pen attributes will be used. Postscript files may be printed on any Postscript device. Double line mode plots can not be printed in this mode.
- Color Postscript: This radio button is used to produce Color Postscript printer files. The CapView color pen attributes will be used. Color Postscript files may be printed on any Postscript device capable of printing in color. They may also be printed on a monochrome postscript device, in which case the colors will be indicated by different shades.
- HP-GL: This radio button is used to produce HP-GL plotter files. HP-GL files may be printed on any HP-GL compatible device.

There are additional choices to determine how the file shall be printed.

- Create File Only: This checkbox will prevent the file from being sent to a Postscript or a HP-GL device if that is the output file type. When this box is checked, there will be no hardcopy output. This checkbox has no effect with X11R4.
- Delete File After Printing: If this checkbox is checked the .prt file will be deleted after processing. This box should only be checked when the file will not be needed again.
- Output Device Name: This edit field is used to enter the unix name of the device to use. For a printer or a plotter the unix name (e.g. lp) should be entered. For screen the output device name field is hidden.

Delete, Print and Cancel

The last group of buttons in the Print File dialogue box are the Delete, Print and Cancel buttons.

- Delete: Select this button to delete the file as defined above. The Print File dialogue box will not be closed.
- Print: Selecting this button will print the file as defined above and close the Print File dialogue box.
- Cancel: Selecting this button will close the Print File dialogue box without printing a file.

7.9 CapHelp Dialogue Box

The CapHelp window has a scrollable list of exclusive selections (Figure 7-16). These selections represent the sections of this Reference Manual. Notice that one and only one selection is displayed in reverse video (with the colors of the foreground and background reversed when compared with the other selections). This selection is the selected item. To make a different selection, the user must move the mouse cursor to the desired selection. As the mouse cursor enters the region of a selection, it changes to a check mark. Click the left mouse button. Notice that the new selection reverses its foreground and background colors while the old selection returns to normal colors.

Below the scrollable list of selections, there are three buttons labeled: Show, Print, and Done. If the Show button is selected, CapHelp will create a dialogue box labeled Help Section that has a scrollable message field (text window) (Figure 7-17). This window displays a section from the manual (without figures). Below the scrollable window there are two buttons labeled Search and Done. The Done button, when selected, closes the Help Section dialogue box. The Search button, when selected, displays a search dialogue box (Figure 7-18). The search dialogue box allows the user to search forward and backward for text. The user can search forward or backward by toggling between the two provided selections. The user can enter the text to be searched for in the edit field labeled "Search For". To perform the search, select the button labeled Search. To close the search dialogue box, select the button labeled Cancel.

If the Print button of the CapHelp window is selected, CapHelp will issue a UNIX print command to print the selected section on the default printer.

If the Done button of the CapHelp window is selected, CapHelp will display a dialogue box to verify that the user wants to exit CapHelp. The dialogue box provides two buttons labeled Yes and No. Selecting the button labeled Yes will terminate CapHelp (without affecting CapView) while selecting the button labeled No will ignore the exit request.

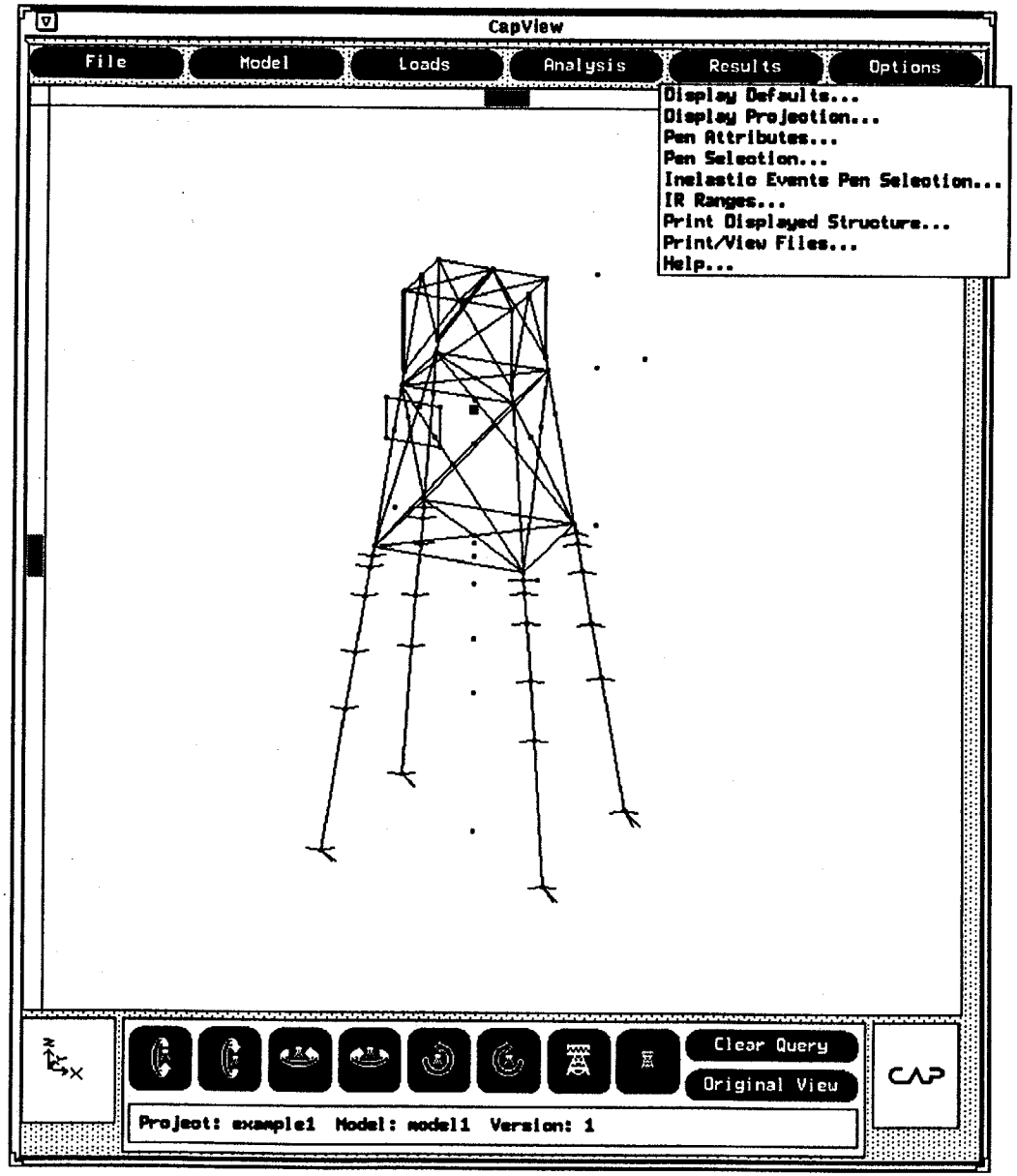

CapView
File
Model
Loads
Analysis
Results
Options
Display Defaults...
Display Projection...
Pen Attributes...
Pen Selection...
Inelastic Events Pen Selection...
IR Ranges...
Print Displayed Structure...
Print/View Files...
Help...
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-1

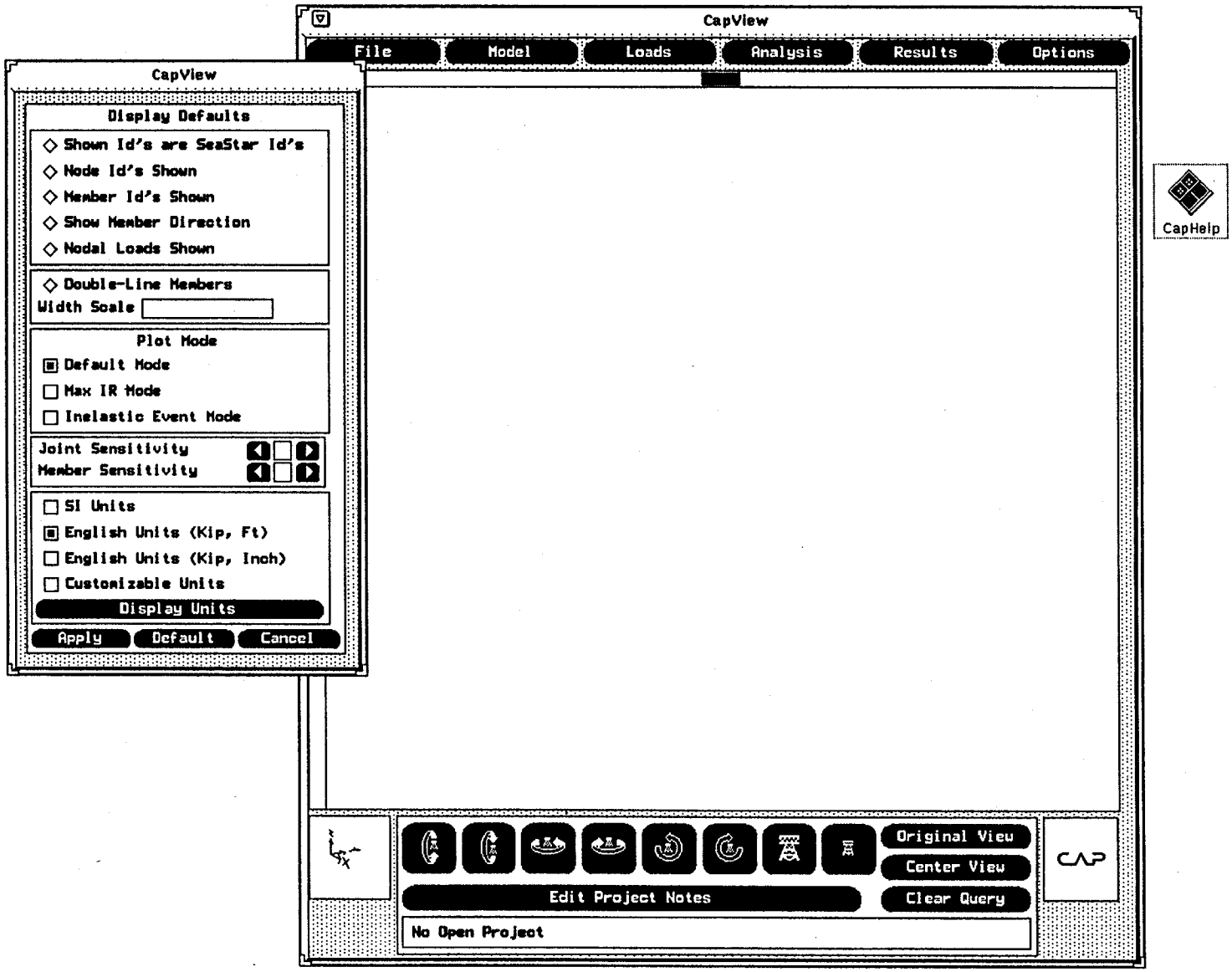

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Display Defaults
Shown Id's are SeaStar Id's
Node Id's Shown
Member Id's Shown
Show Member Direction
Nodal Loads Shown
Double-Line Members
Width Scale
Plot Mode
Default Mode
Max IR Mode
Inelastic Event Mode
Joint Sensitivity
Member Sensitivity
SI Units
English Units (Kip, Ft)
English Units (Kip, Inch)
Customizable Units
Display Units
Apply
Default
Cancel
Original View
Center View
Clear Query
Edit Project Notes
No Open Project

Figure 7-2

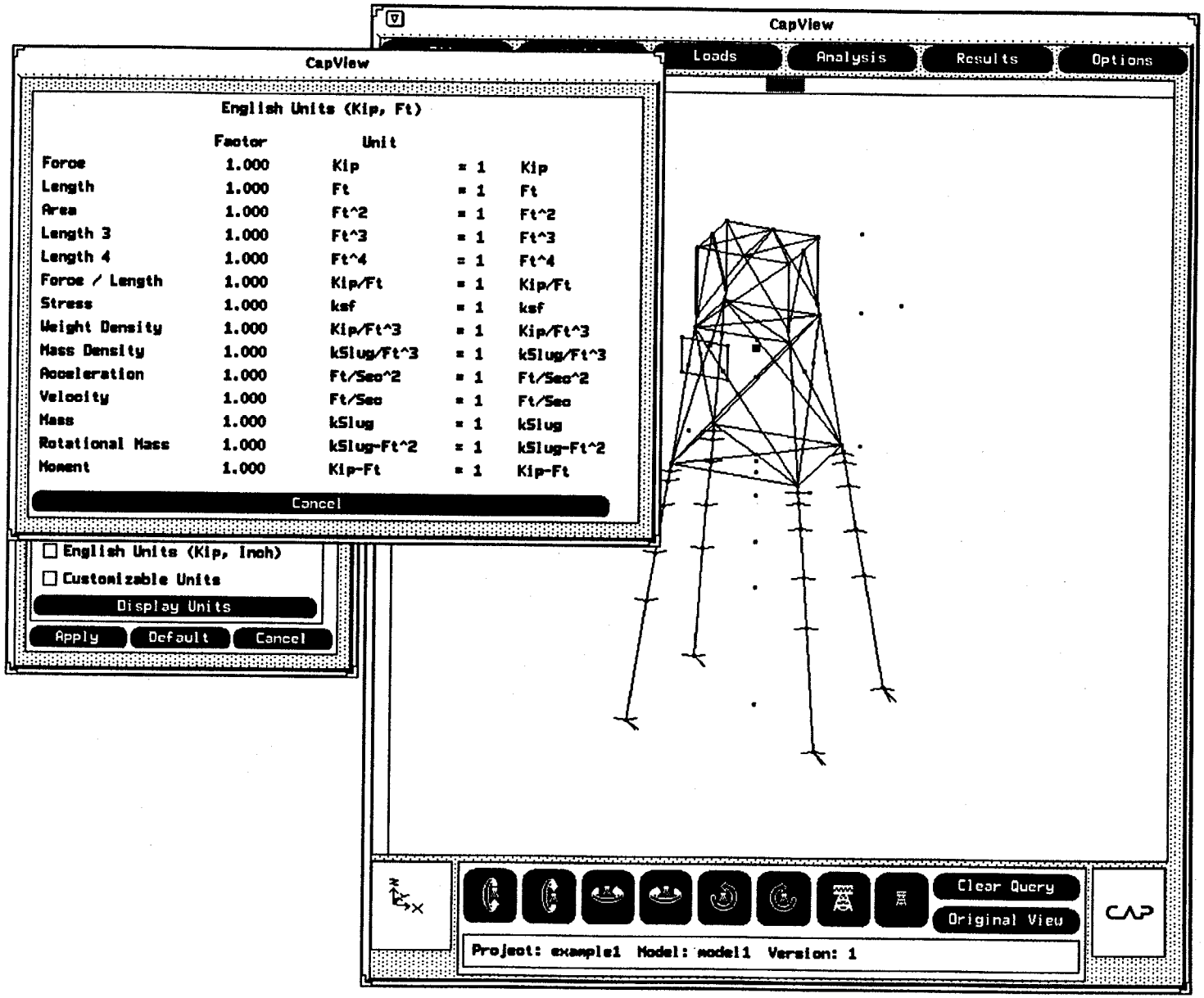

CapView
Loads
Analysis
Results
Options
CapView
English Units (Kip, Ft)
Factor
Unit
Force 1.000 Kip = 1 Kip
Length 1.000 Ft = 1 Ft
Area 1.000 Ft^2 = 1 Ft^2
Length 3 1.000 Ft^3 = 1 Ft^3
Length 4 1.000 Ft^4 = 1 Ft^4
Force / Length 1.000 Kip/Ft = 1 Kip/Ft
Stress 1.000 ksf = 1 ksf
Weight Density 1.000 Kip/Ft^3 = 1 Kip/Ft^3
Mass Density 1.000 kSlug/Ft^3 = 1 kSlug/Ft^3
Acceleration 1.000 Ft/Sec^2 = 1 Ft/Sec^2
Velocity 1.000 Ft/Sec = 1 Ft/Sec
Mass 1.000 kSlug = 1 kSlug
Rotational Mass 1.000 kSlug-Ft^2 = 1 kSlug-Ft^2
Moment 1.000 Kip-Ft = 1 Kip-Ft
Cancel
English Units (Kip, Inch)
Customizable Units
Display Units
Apply
Default
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-3

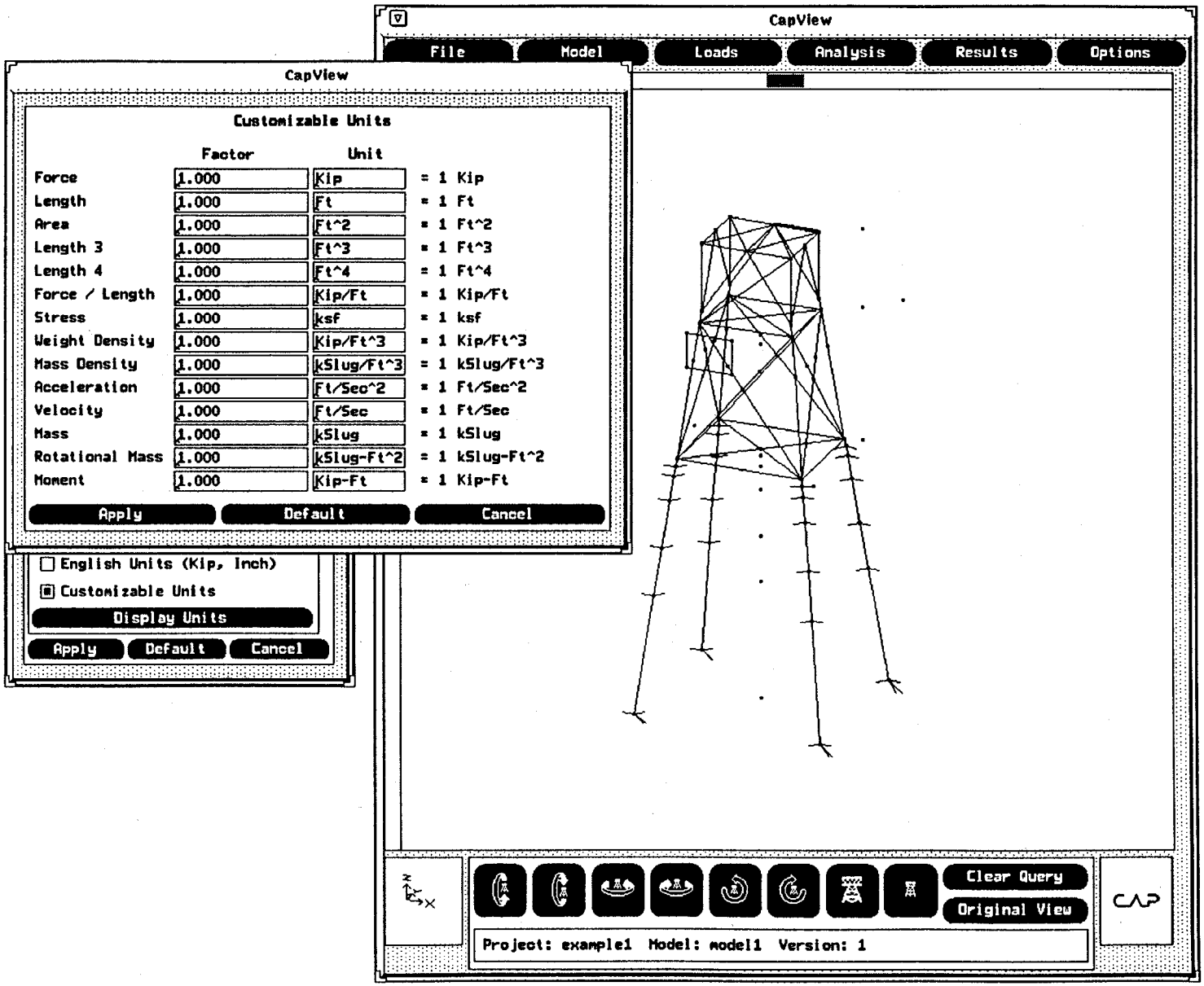

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Customizable Units
Factor
Unit
Force 1.000 Kip = 1 Kip
Length 1.000 Ft = 1 Ft
Area 1.000 Ft^2 = 1 Ft^2
Length 3 1.000 Ft^3 = 1 Ft^3
Length 4 1.000 Ft^4 = 1 Ft^4
Force / Length 1.000 Kip/Ft = 1 Kip/Ft
Stress 1.000 ksf = 1 ksf
Weight Density 1.000 Kip/Ft^3 = 1 Kip/Ft^3
Mass Density 1.000 kSlug/Ft^3 = 1 kSlug/Ft^3
Acceleration 1.000 Ft/Sec^2 = 1 Ft/Sec^2
Velocity 1.000 Ft/Sec = 1 Ft/Sec
Mass 1.000 kSlug = 1 kSlug
Rotational Mass 1.000 kSlug-Ft^2 = 1 kSlug-Ft^2
Moment 1.000 Kip-Ft = 1 Kip-Ft
Apply
Default
Cancel
English Units (Kip, Inch)
Customizable Units
Display Units
Apply
Default
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-4

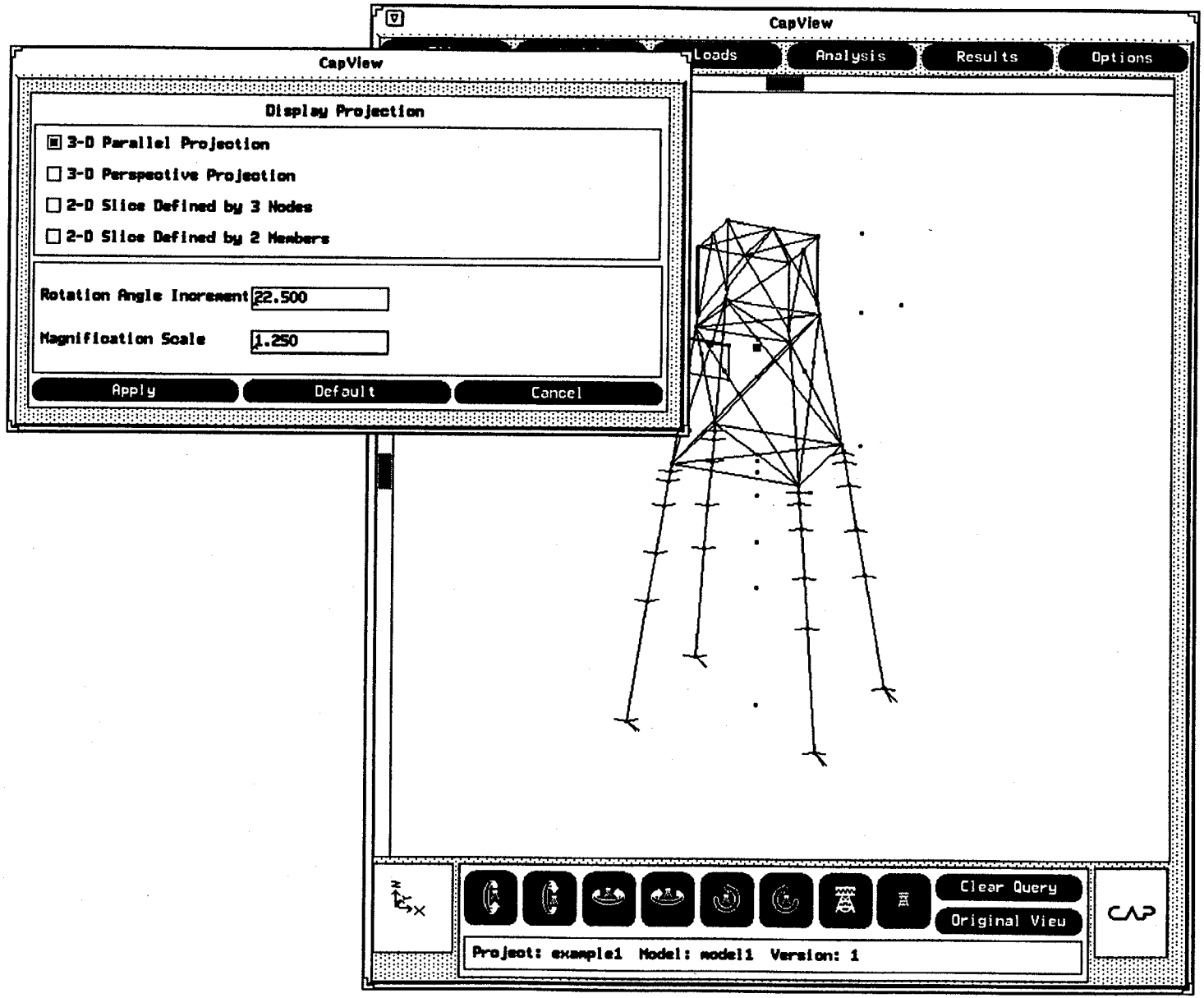

CapView
Loads
Analysis
Results
Options
CapView
Display Projection
3-D Parallel Projection
3-D Perspective Projection
2-D Slice Defined by 3 Nodes
2-D Slice Defined by 2 Members
Rotation Angle Increment
22.500
Magnification Scale
1.250
Apply
Default
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-5

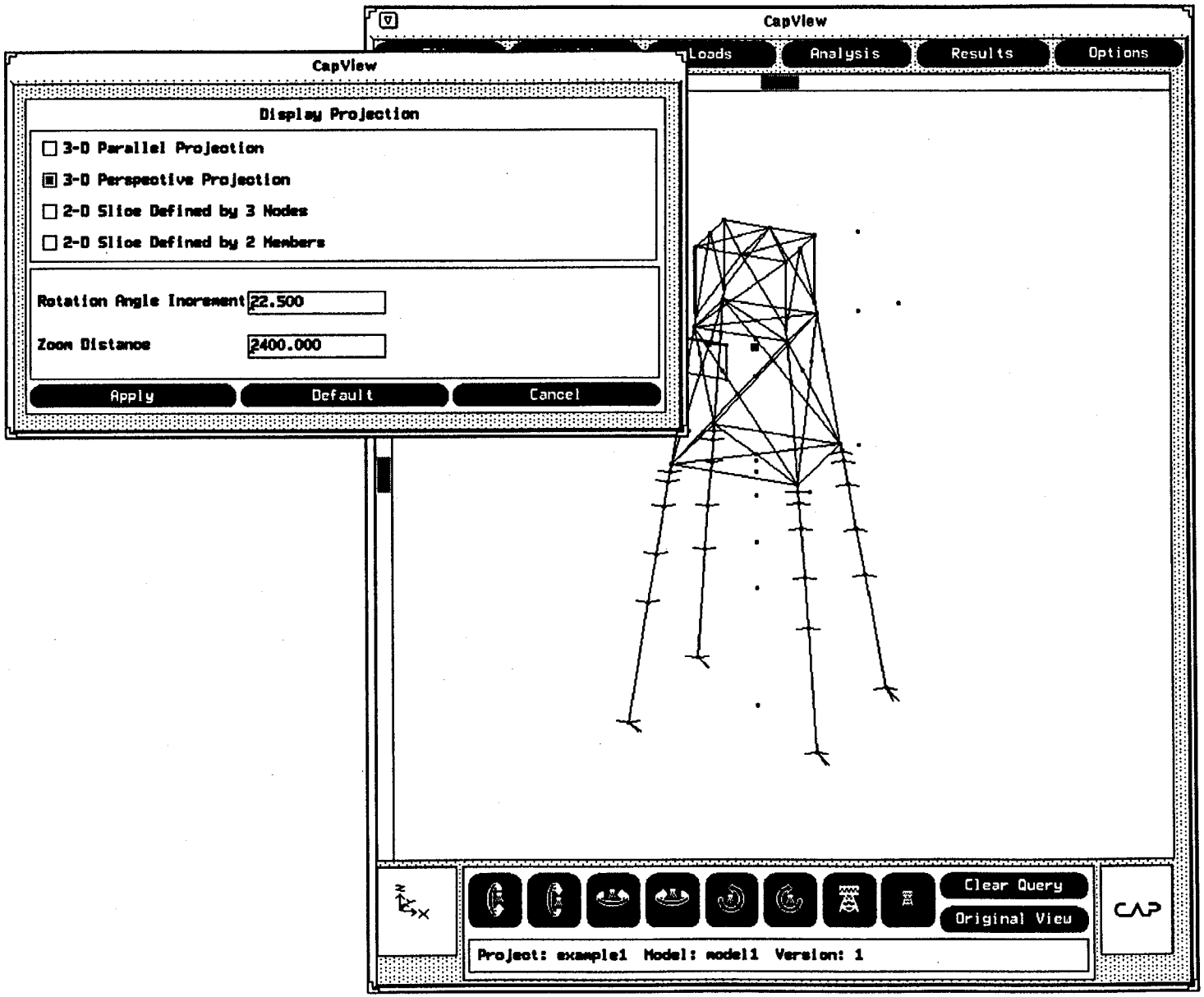

CapView
Loads
Analysis
Results
Options
CapView
Display Projection
3-D Parallel Projection
3-D Perspective Projection
2-D Slice Defined by 3 Nodes
2-D Slice Defined by 2 Members
Rotation Angle Increment
22.500
Zoom Distance
2400.000
Apply
Default
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-6

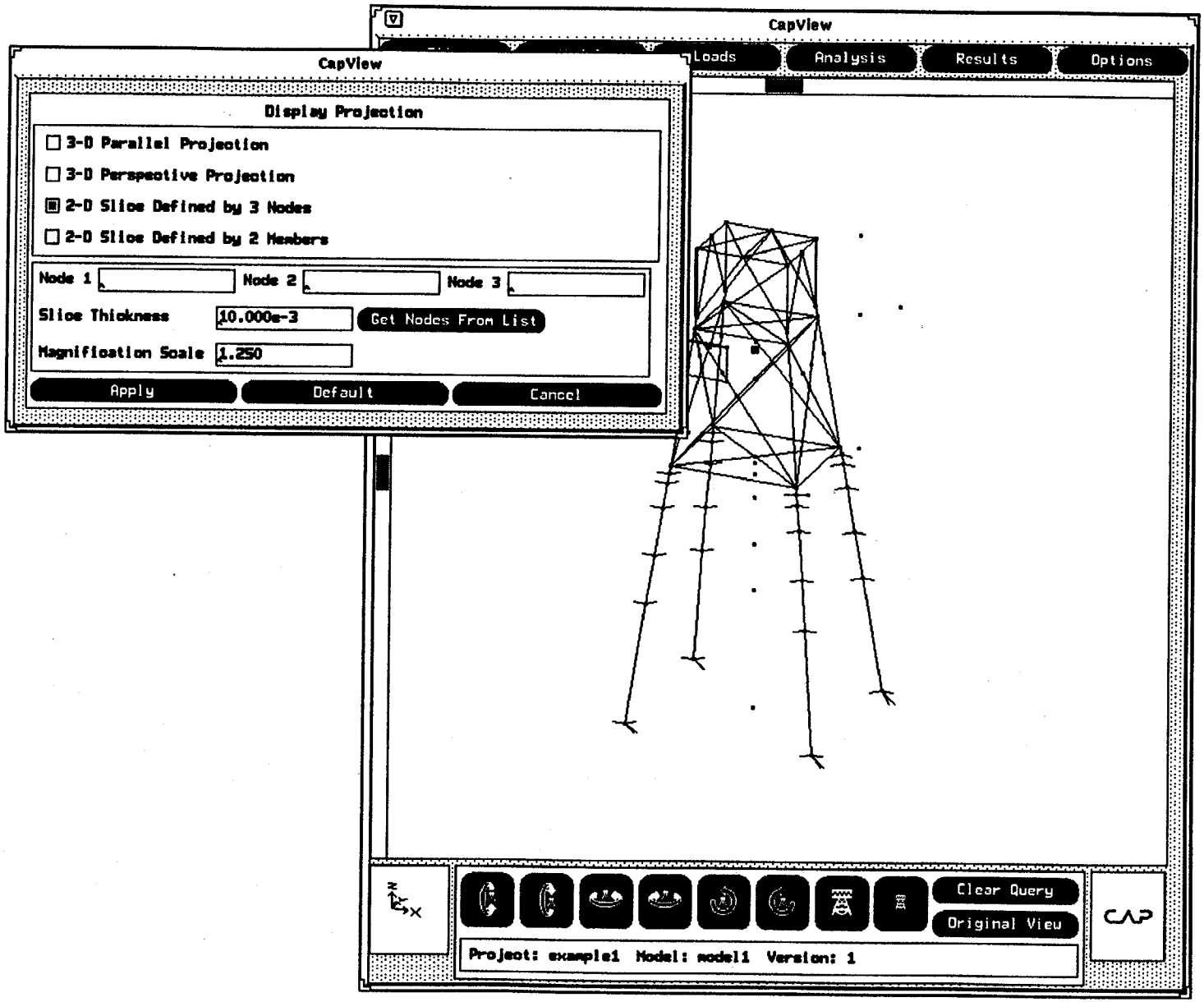

CapView
Loads
Analysis
Results
Options
CapView
Display Projection
3-D Parallel Projection
3-D Perspective Projection
2-D Slice Defined by 3 Nodes
2-D Slice Defined by 2 Members
Node 1
Node 2
Node 3
Slice Thickness
10.000e-3
Get Nodes From List
Magnification Scale
1.250
Apply
Default
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-7

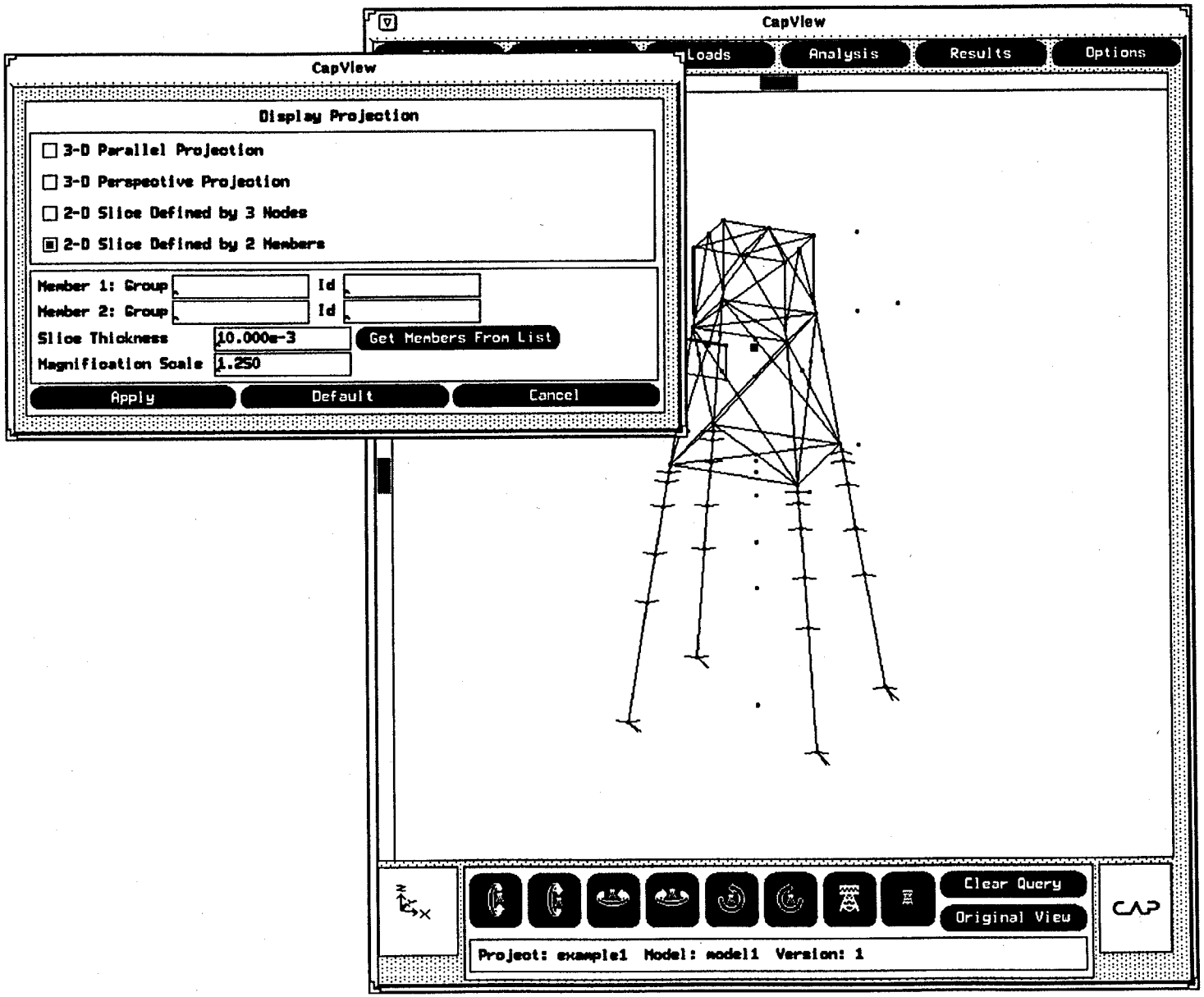

CapView
Loads
Analysis
Results
Options
CapView
Display Projection
3-D Parallel Projection
3-D Perspective Projection
2-D Slice Defined by 3 Nodes
2-D Slice Defined by 2 Members
Member 1: Group
Id
Member 2: Group
Id
Slice Thickness
10.000e-3
Get Members From List
Magnification Scale
1.250
Apply
Default
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-8

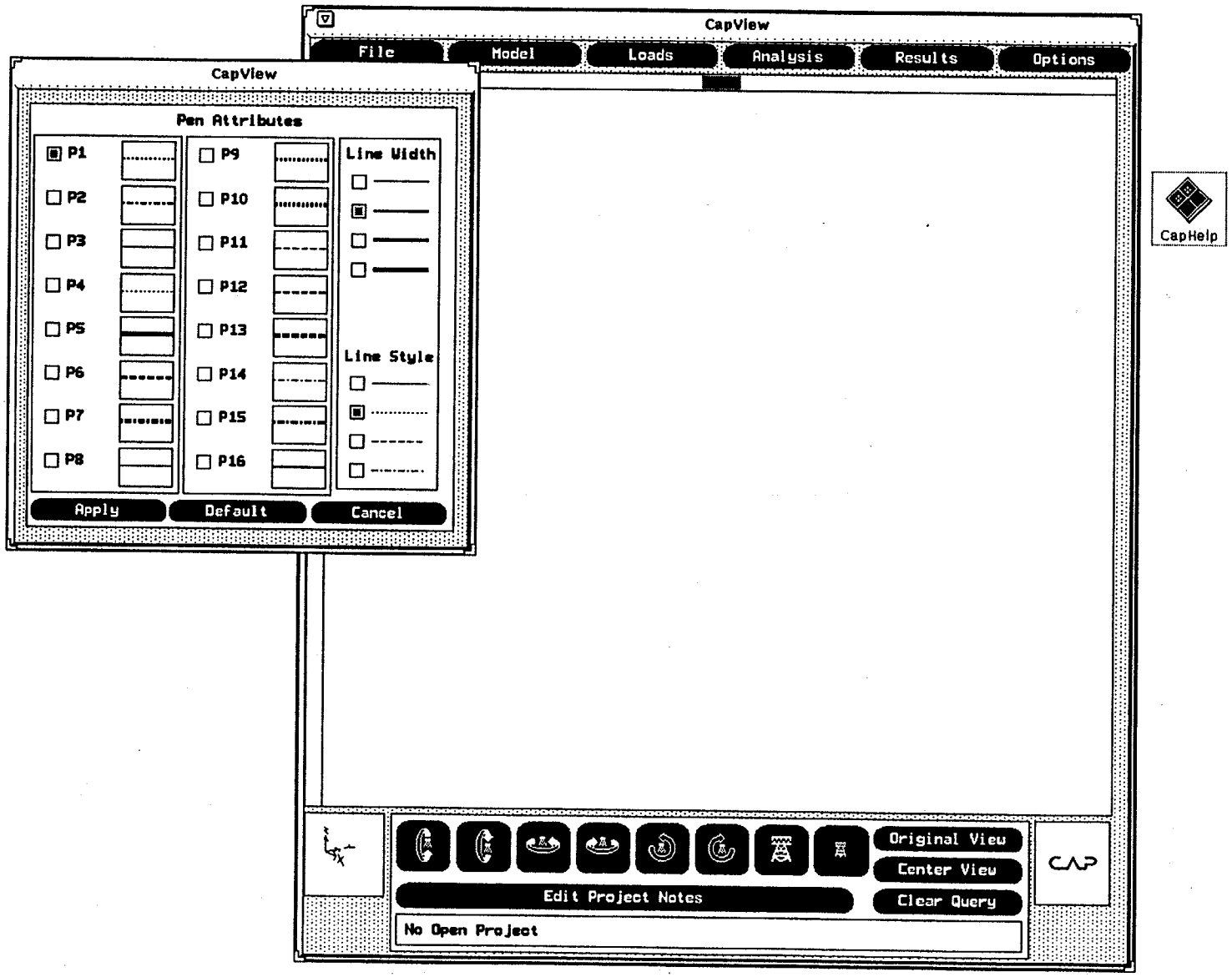

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Pen Attributes
P1
P2
P3
P4
P5
P6
P7
P8
P9
P10
P11
P12
P13
P14
P15
P16
Line Width
Line Style
Apply
Default
Cancel
Original View
Center View
Clear Query
Edit Project Notes
No Open Project

Figure 7-9

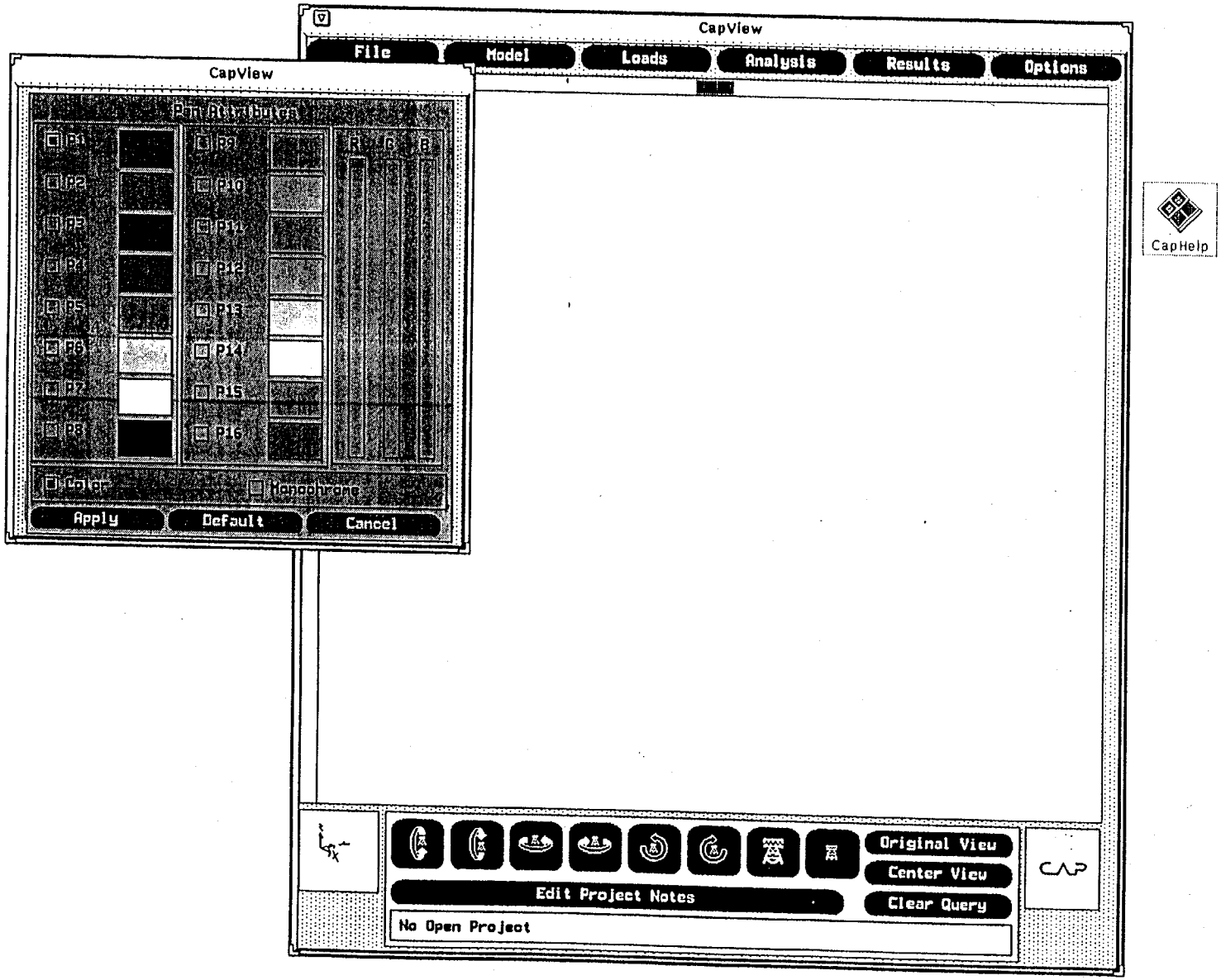

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Color
Monochrome
Apply
Default
Cancel
Original View
Center View
Clear Query
Edit Project Notes
No Open Project
CAP

Figure 7-10

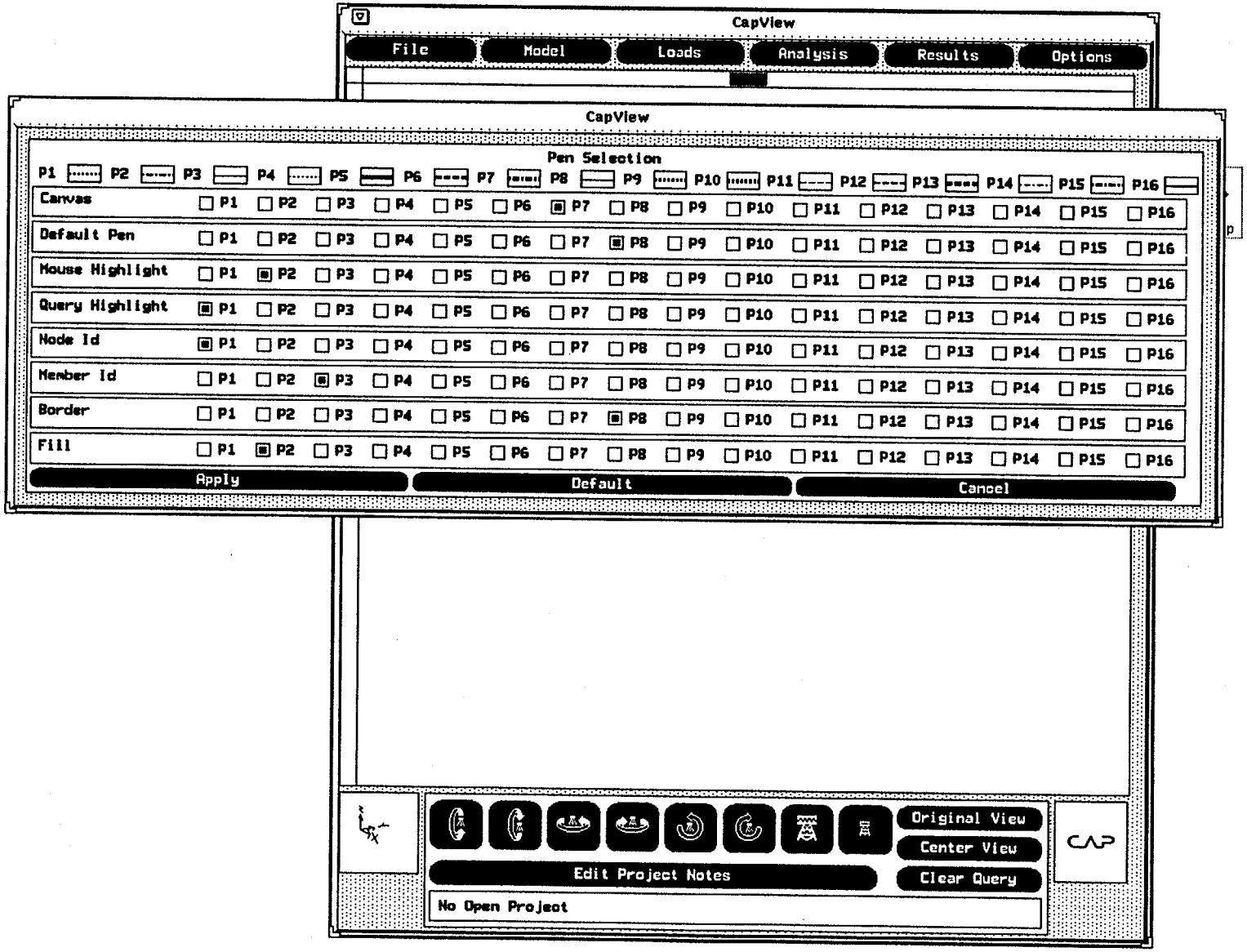

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Pen Selection
P1 P2 P3 P4 P5 P6 P7 P8 P9 P10 P11 P12 P13 P14 P15 P16
Canvas
Default Pen
Mouse Highlight
Query Highlight
Node Id
Member Id
Border
Fill
Apply
Default
Cancel
Original View
Center View
Edit Project Notes
Clear Query
No Open Project

Figure 7-11

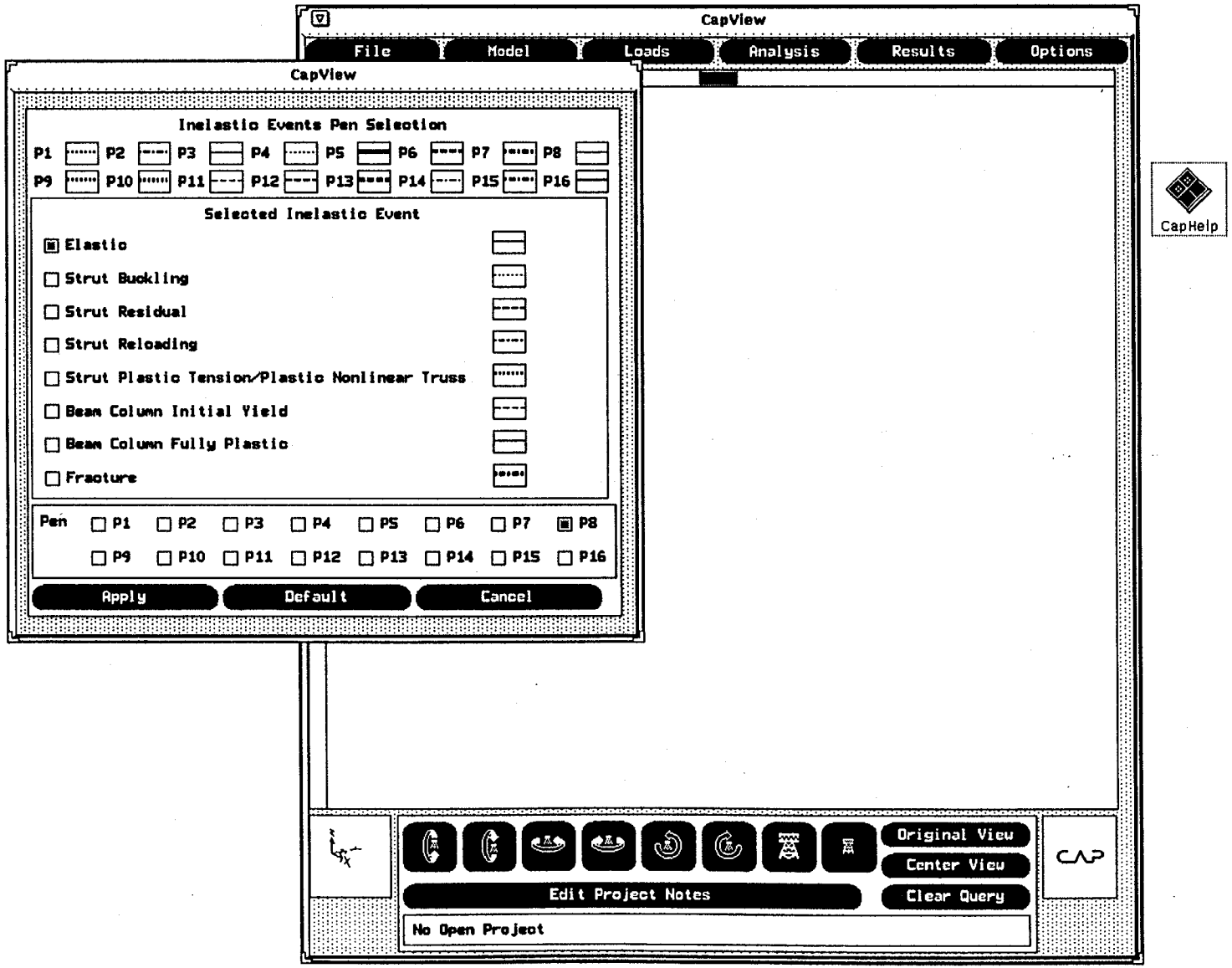

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Inelastic Events Pen Selection
P1 P2 P3 P4 P5 P6 P7 P8
P9 P10 P11 P12 P13 P14 P15 P16
Selected Inelastic Event
Elastic
Strut Buckling
Strut Residual
Strut Reloading
Strut Plastic Tension/Plastic Nonlinear Truss
Beam Column Initial Yield
Beam Column Fully Plastic
Fracture
Pen P1 P2 P3 P4 P5 P6 P7 P8
P9 P10 P11 P12 P13 P14 P15 P16
Apply
Default
Cancel
Original View
Center View
Edit Project Notes
Clear Query
No Open Project

Figure 7-12

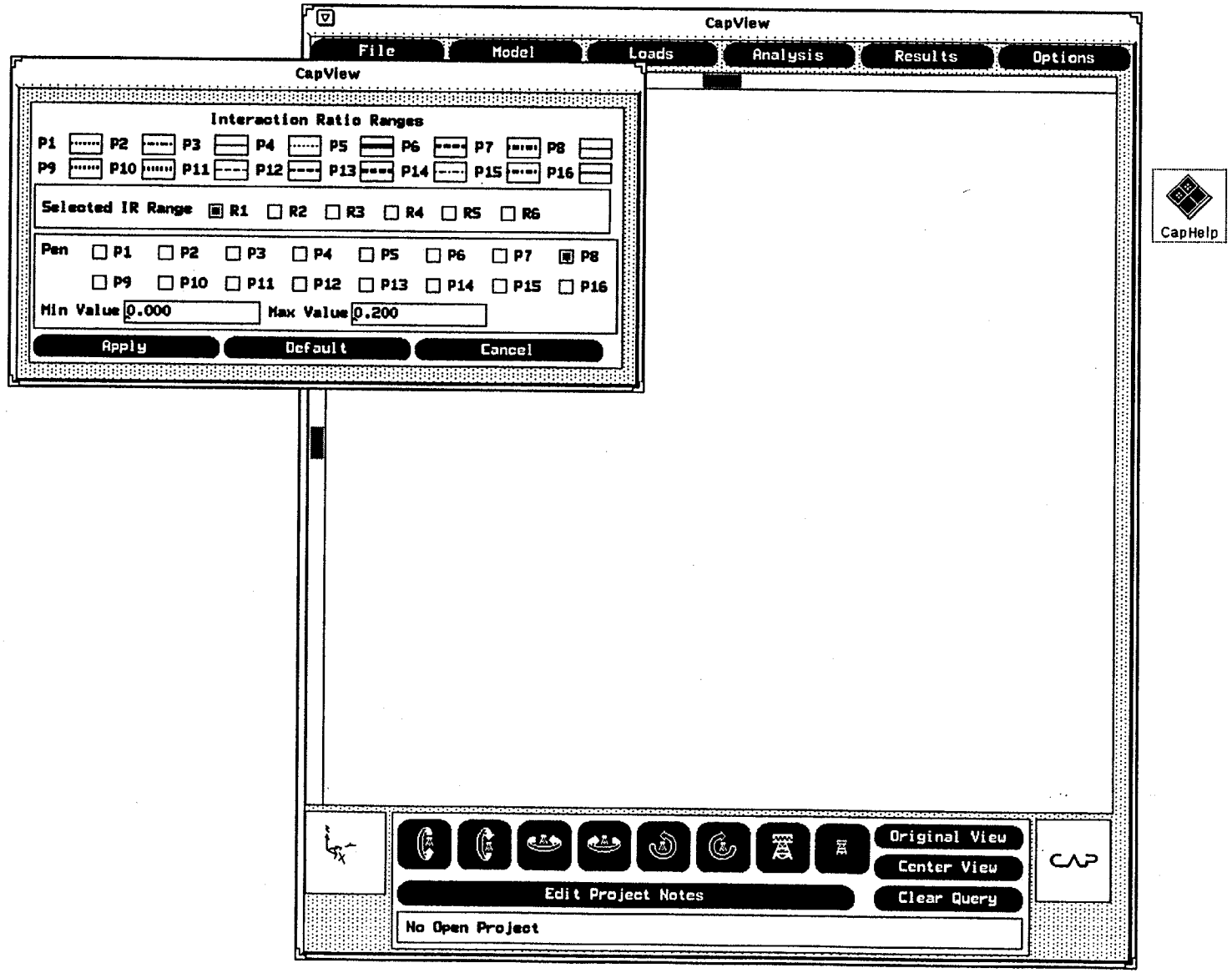

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Interaction Ratio Ranges
P1 P2 P3 P4 P5 P6 P7 P8
P9 P10 P11 P12 P13 P14 P15 P16
Selected IR Range R1 R2 R3 R4 R5 R6
Pen P1 P2 P3 P4 P5 P6 P7 P8
P9 P10 P11 P12 P13 P14 P15 P16
Min Value 0.000
Max Value 0.200
Apply
Default
Cancel
CapHelp
Original View
Center View
Clear Query
Edit Project Notes
No Open Project

Figure 7-13

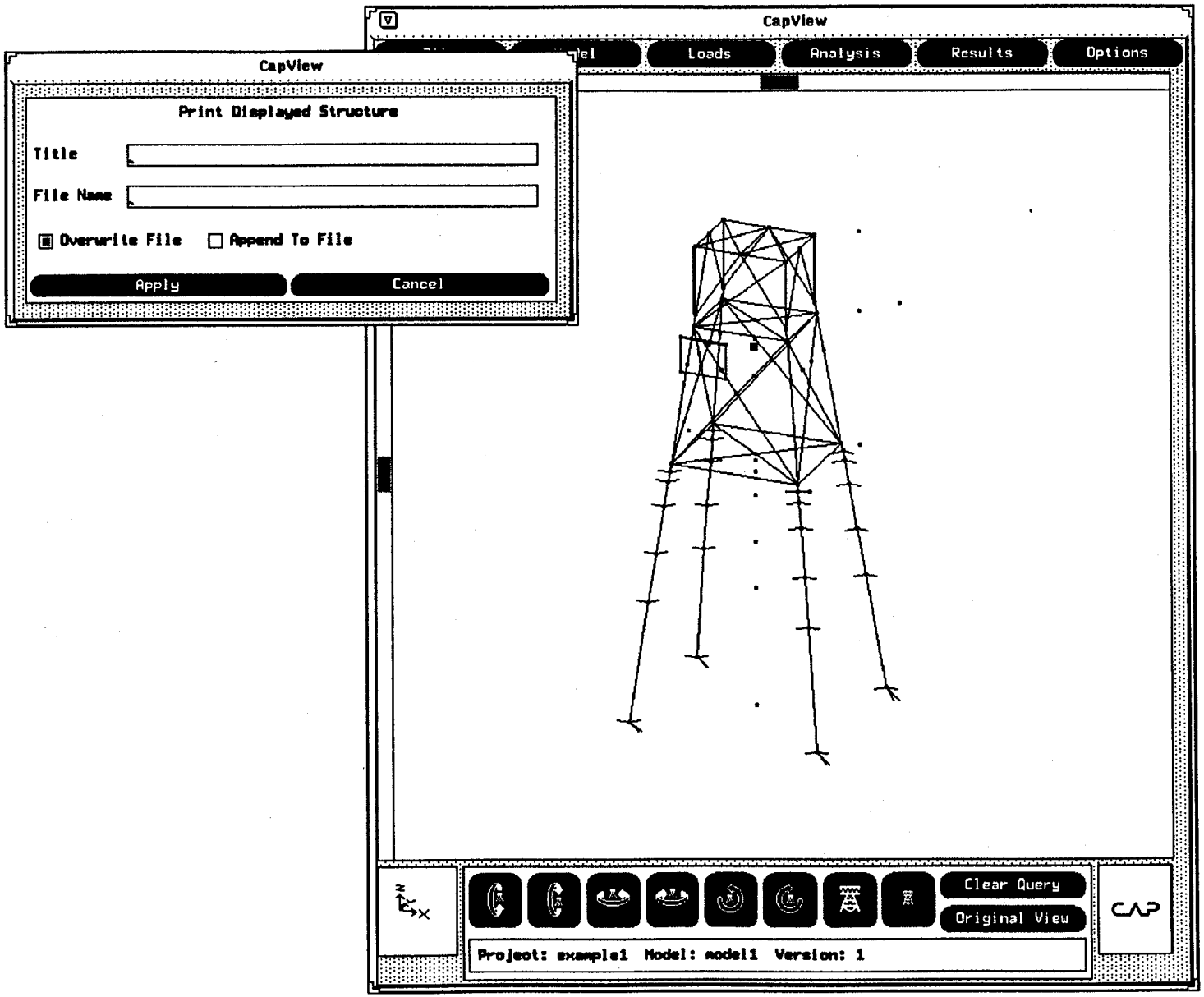

CapView
Loads
Analysis
Results
Options
CapView
Print Displayed Structure
Title
File Name
Overwrite File
Append To File
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-14

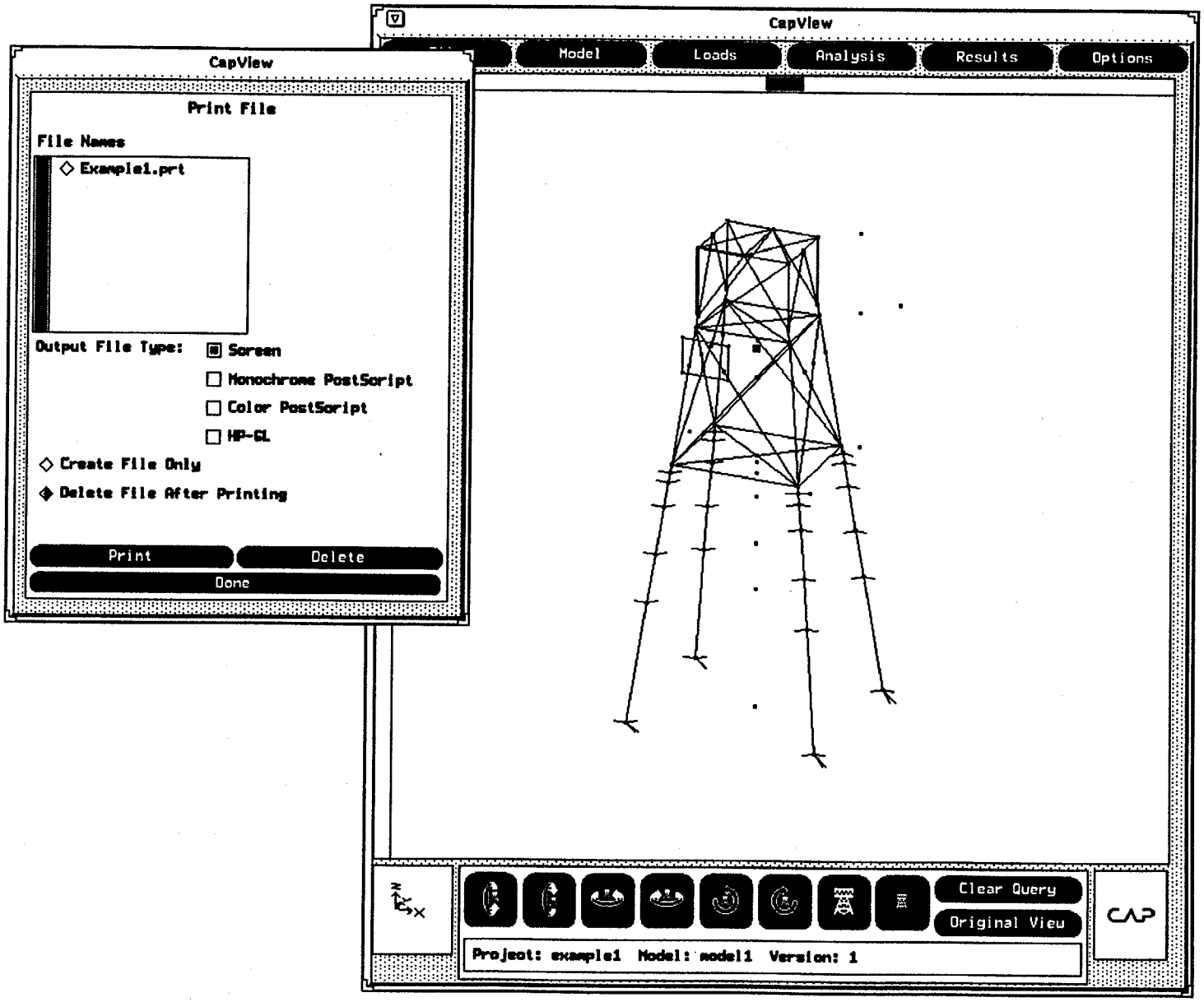

CapView
Model
Loads
Analysis
Results
Options
CapView
Print File
File Names
Example1.prt
Output File Type:
Screen
Monochrome PostScript
Color PostScript
HP-GL
Create File Only
Delete File After Printing
Print
Delete
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-15

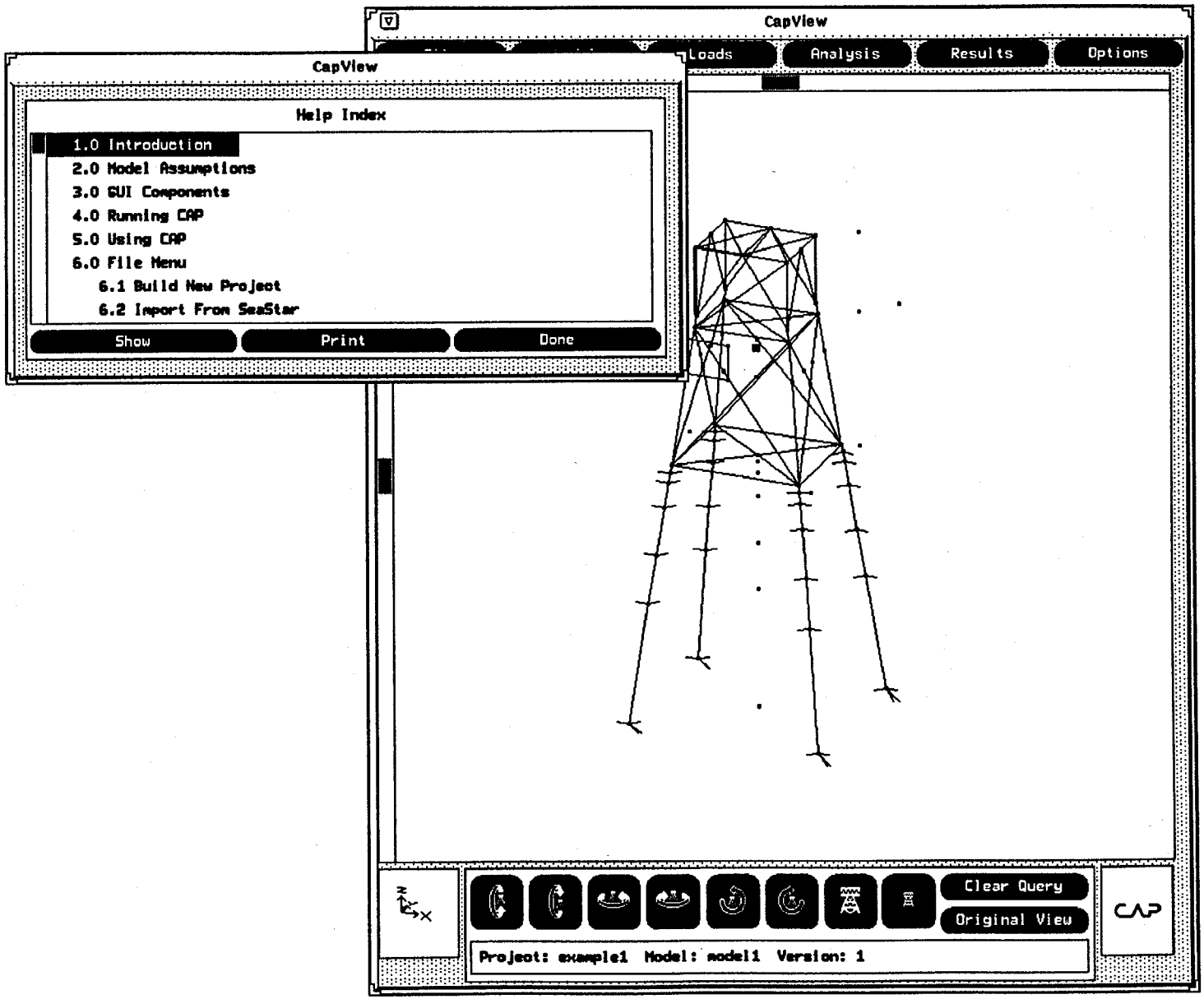

CapView
Loads
Analysis
Results
Options
CapView
Help Index
1.0 Introduction
2.0 Model Assumptions
3.0 GUI Components
4.0 Running CAP
5.0 Using CAP
6.0 File Menu
6.1 Build New Project
6.2 Import From SeaStar
Show
Print
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 7-16

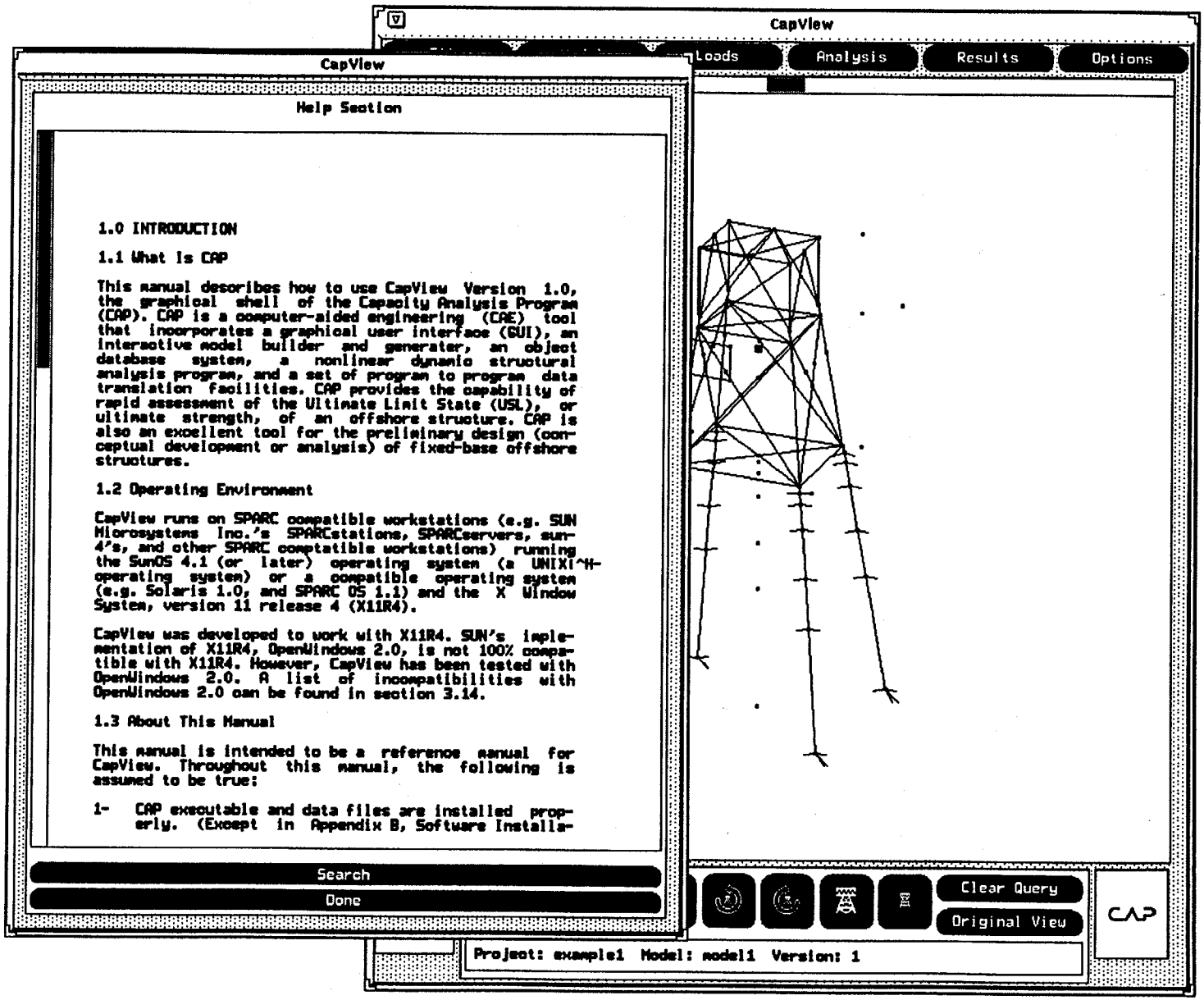

CapView
Loads
Analysis
Results
Options
CapView
Help Section
1.0 INTRODUCTION
1.1 What Is CAP
This manual describes how to use CapView Version 1.0, the graphical shell of the Capacity Analysis Program (CAP). CAP is a computer-aided engineering (CAE) tool that incorporates a graphical user interface (GUI), an interactive model builder and generater, an object database system, a nonlinear dynamic structural analysis program, and a set of program to program data translation facilities. CAP provides the capability of rapid assessment of the Ultimate Limit State (USL), or ultimate strength, of an offshore structure. CAP is also an excellent tool for the preliminary design (conceptual development or analysis) of fixed-base offshore structures.
1.2 Operating Environment
CapView runs on SPARC compatible workstations (e.g. SUN Microsystems Inc.'s SPARCstations, SPARCservers, sun-4's, and other SPARC comptatible workstations) running the SunOS 4.1 (or later) operating system (a UNIX(^H-operating system) or a compatible operating system (e.g. Solaris 1.0, and SPARC OS 1.1) and the X Window System, version 11 release 4 (X11R4).
CapView was developed to work with X11R4. SUN's implementation of X11R4, OpenWindows 2.0, is not 100% compatible with X11R4. However, CapView has been tested with OpenWindows 2.0. A list of incompatibilities with OpenWindows 2.0 can be found in section 3.14.
1.3 About This Manual
This manual is intended to be a reference manual for CapView. Throughout this manual, the following is assumed to be true:
1- CAP executable and data files are installed properly. (Except in Appendix B, Software Installa-
Search
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 7-17

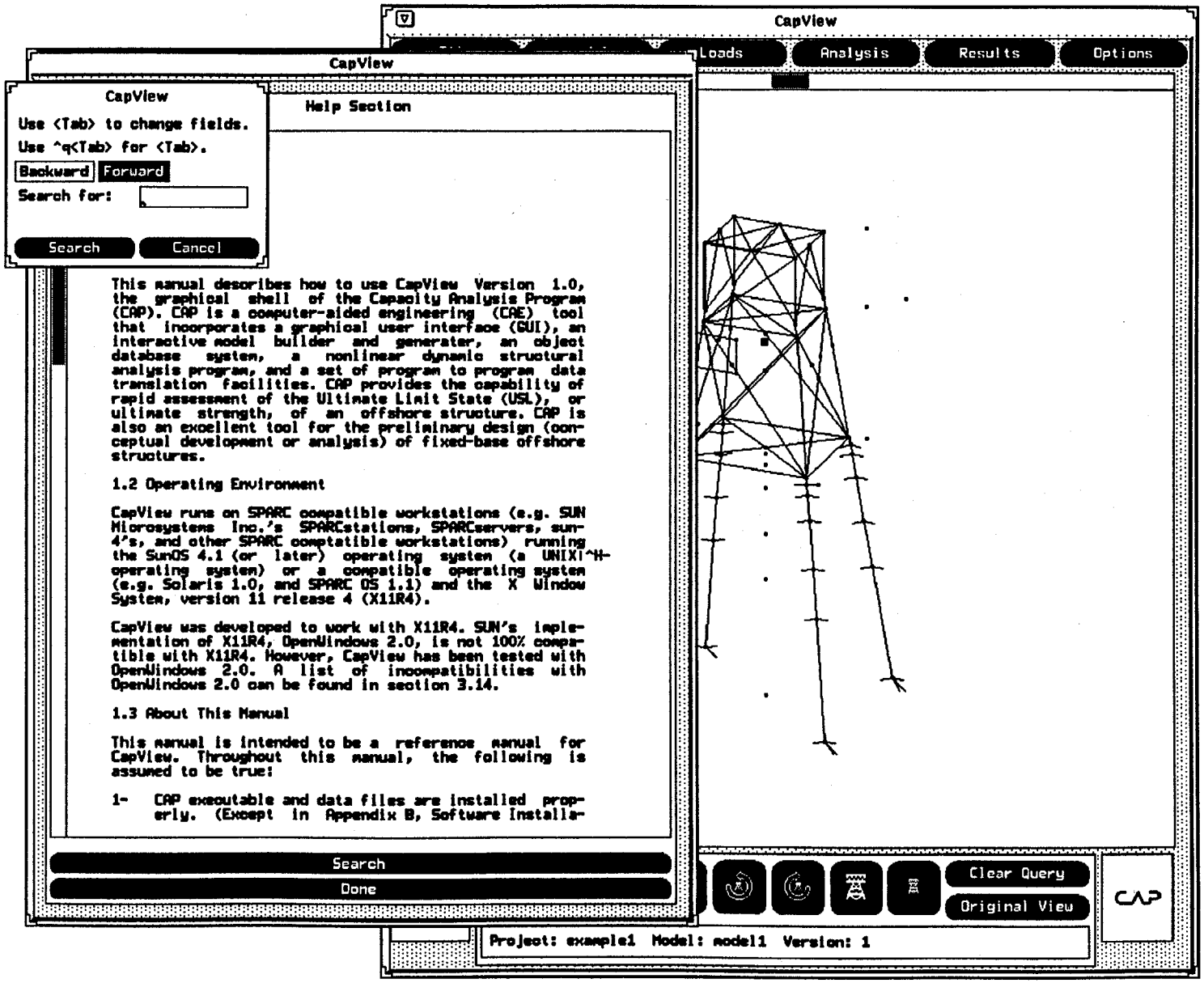

CapView
Loads
Analysis
Results
Options
CapView
Help Section
CapView
Use <Tab> to change fields.
Use ^q<Tab> for <Tab>.
Backward
Forward
Search for:
Search
Cancel
This manual describes how to use CapView Version 1.0, the graphical shell of the Capacity Analysis Program (CAP). CAP is a computer-aided engineering (CAE) tool that incorporates a graphical user interface (GUI), an interactive model builder and generater, an object database system, a nonlinear dynamic structural analysis program, and a set of program to program data translation facilities. CAP provides the capability of rapid assessment of the Ultimate Limit State (USL), or ultimate strength, of an offshore structure. CAP is also an excellent tool for the preliminary design (conceptual development or analysis) of fixed-base offshore structures.
1.2 Operating Environment
CapView runs on SPARC compatible workstations (e.g. SUN Microsystems Inc.'s SPARCstations, SPARCservers, sun-4's, and other SPARC comptatible workstations) running the SunOS 4.1 (or later) operating system (a UNIX(^H-operating system) or a compatible operating system (e.g. Solaris 1.0, and SPARC OS 1.1) and the X Window System, version 11 release 4 (X11R4).
CapView was developed to work with X11R4. SUN's implementation of X11R4, OpenWindows 2.0, is not 100% compatible with X11R4. However, CapView has been tested with OpenWindows 2.0. A list of incompatibilities with OpenWindows 2.0 can be found in section 3.14.
1.3 About This Manual
This manual is intended to be a reference manual for CapView. Throughout this manual, the following is assumed to be true:
1- CAP executable and data files are installed properly. (Except in Appendix B, Software Installa-
Search
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 7-18

8.0 THE MODEL MENU

The Model menu presents selections that will help in editing models by adding new structural elements, nodes, material properties, and cross-section descriptions. When the Model menu is selected, the following menu is displayed (Figure 8-1):

Generate Platform Skeleton...
Add Members/Nodes...
Read Member/Node/Load Files...
Damping Parameters...
Fix X-Brace Center Node...
Soil Data...
Water Data...
Default Analysis Parameters...
Default Code Check Parameters...
Node Select/Change...
Member Select/Change...
Show Selected Nodes
Show Selected Members
Node/Member/Load Reports...
AISC Sections...
Standard Tube Sections...

8.1 Generate Platform Skeleton...

The first selection of the Model menu will display the Platform Skeleton Data dialogue box (Figure 8-2). This dialogue box allows the user to describe and rapidly generate a platform skeleton. The skeleton of a platform is made up of leg and pile elements, horizontal elements, and soil profile descriptions (i.e. all elements except diagonal type braces). Note that platform skeleton descriptions consist of four major parts; Geometry, Elevations, Framing, and Soil Profile. The elements of each major part are bounded by boxes in the Platform Skeleton Data dialogue box.

8.1.1 Platform Geometry

In the Geometry part of a platform skeleton description, the user must select between a tripod platform and a rectangular platform. The Geometry box has two radio buttons to that toggle between tripod and rectangular geometry.

The Geometry box also has a button labeled "Edit Coordinates" that, when pressed, it will display the Platform Skeleton Geometry dialogue box. When the selected geometry is Rectangular, the Geometry box shows two edit field so that the user can specify the number of X grid lines and Y grid lines (Figure 8-3). X (Y) grid lines are imaginary lines parallel to the x-axes (y-axes). The intersection points of the X and Y grid lines represents the locations of the legs in a typical plan.

So if the user selects 2 X grid lines and 2 Y grid lines, the described platform will have 4 legs. If the user selects 1 X grid line and 1 Y grid line, the described platform will have one leg (stick model). A 2-D platform in the XZ plane can be described by specifying 1 X grid line and as many Y grid lines as there are legs in the platform. A 2-D platform in the YZ plane can be described by specifying 1 Y grid line and as many X grid lines as there are legs in the platform.

The Platform Skeleton Geometry dialogue box has two parts. The upper part is used to enter the coordinates of the platform plan at the working point elevation. The lower part is used to enter the batters on each face of the platform.

For a rectangular platform (Figure 8-4), the user must specify the Y coordinates of the X grid lines and the X coordinates of the Y grid lines. The user must also specify the batter of 4 faces (in case of a 3-D model), or 2 faces (in case of a 2-D model). If the model is to be a stick model, then no batters can be specified (the stick will be vertical).

For a tripod (Figure 8-5), the user must specify the distance between all the legs of the tripod. The user must also specify the batter of each leg independently by either specifying the batter of each leg in a plane perpendicular to the opposite face of the leg (Figure 8-5) or the batter of each leg in planes parallel to the XZ and YZ planes (global batter) (Figure 8-6). The Platform Skeleton Geometry dialogue box has two radio buttons that toggle between global batter and batter in a plane perpendicular to opposite face.

Apply and Cancel

The last group of buttons in the Platform Skeleton Geometry dialogue box are the Apply, and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Platform Skeleton Geometry dialogue box. This button must be selected to make any changes effective.
- Cancel: Selecting this button will close the Platform Skeleton Geometry dialogue box and use the settings that were already in effect. Any

changes made will be lost.

8.1.2 Platform Elevations

In the Elevation part of a platform skeleton description (Figure 8-2), the user specifies the elevations at which CapView is to generate nodes. For a structure with piles these include the elevations at which to generate soil elements, as well as the major structure framing elevations.

The Elevation box has an edit field for the user to specify the total number of elevations above the bottom of the legs that the model generator will create horizontal members, and below the bottom of the legs where soil springs (PSAS) elements are to be generated. This number will control a scrollable list of edit fields where the user specifies the elevations. The Elevation box also has a button labeled "Edit Elevations" that, when pressed, will display the Platform Skeleton Elevations dialogue box.

The Platform Skeleton Elevations dialogue box (Figure 8-7) has two parts. The upper part is used to enter the elevations of the working point, water surface, mud line, bottom of the legs, and bottom of the piles. Each of these has its appropriately labeled edit field. The lower part has a scrollable list of edit fields labeled major elevations. The user must enter the elevations at which the model generator will create horizontal members and soil springs (PSAS) elements.

When using this dialogue box keep in mind the following rules:

1- The major elevations can be entered in any order.

2- Do not duplicate any of the major elevations.

3- If the elevation of the working point or bottom of the legs is duplicated in the major elevation list, the model generator will create horizontal members at the elevations, otherwise it won't. (See the Platform Framing section for more details.)

4- If the bottom of the piles elevation is duplicated it will be ignored.

5- Pile elements will be extended below the bottom of the legs only if the bottom of the piles is less that the bottom of the legs. Otherwise, pile elements (if generated) will be cut off at the bottom of the leg regardless of the bottom of the piles specified by the user. (See the Platform Framing section for more details.)

6- The pile elements extend all the way up to the working point.

7- The working point serves as the point where the face batters start. Above the working point the legs are vertical, and below it they are battered.

Apply and Cancel

The last group of buttons in the Platform Skeleton Elevations dialogue box are the Apply, and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Platform Skeleton Elevations dialogue box. This button must be selected to make any changes effective.
- Cancel: Selecting this button will close the Platform Skeleton Elevations dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.1.3 Platform Framing

In the Framing part of a platform skeleton description (Figure 8-2), the user specifies the element properties of the platform skeleton. The Framing box has two buttons labeled "Edit Legs" and "Edit Horizontals". When the first button is pressed, it will display the Platform Skeleton Legs dialogue box. When the second button is pressed, it will display the Platform Skeleton Horizontal Braces dialogue box.

8.1.3.1 Platform Skeleton Legs

The Platform Skeleton Legs dialogue box (Figure 8-8) has 3 major areas. The first area displays a typical plan of the platform. The second area displays the properties of a particular leg. The third area is a group of buttons.

Typical Plan Area

The typical plan of the platform is only intended to help the user visualize the location of each leg with respect to the other legs and is not drawn to scale. It shows the grid lines and the location of each leg. Note that one of the legs is surrounded by a square. This square identifies the leg for which the properties are displayed in the second area of the dialogue box. If you move the mouse cursor to another leg location and press the left mouse button, the surrounding square will be moved to the new leg and the properties area changes to reflect the properties of the selected leg.

Leg Properties Area

The leg properties area shows the following:

- Leg ID.
- Material name modeling the material properties of the leg. Note that there is a button labeled "Material" and next to it is an edit field. If the material button is pressed, it will display a Generic Material Information dialogue box corresponding to the material name entered in the edit field. If there is no Generic Material with that name an appropriate error message is displayed.
- A check box that will enable or disable the generation of elements modeling the leg.
- A combo box to select the element type. The user can select between the following: nonlinear truss, large displacement beam, linear beam, beam column, strut, and buckling beam.
- A radio button list to select the leg type. There are three possible selections: leg only, leg with grouted pile, and leg with ungrouted pile. If leg only is selected, the model generator will create elements with cross sections representing that of the leg. If leg with grouted pile is selected, the model generator will create elements with a combined leg and pile section. If leg with ungrouted pile is selected, the model generator will create separate elements for the legs and the piles along with shim elements to guide the displacements of the pile elements inside the leg elements.
- A radio button list to specify section properties along with a box describing the section. The available selections are: generate, specify outside diameter (OD) and wall thickness (TH), and specify area and inertia.
    - If generate is selected the section properties box will display four edit fields to specify minimum and maximum values for the KL/R ratio and and D/t ratio of the section (Figure 8-8). The model generator

will use these two ranges to select appropriate sections from a standard tubular sections table.

  - If specify OD and TH is selected the section properties box will display four edit fields to specify the outside diameter and thickness of the leg cross section and the pile cross section (Figure 8-9). Note that pile cross section edit fields are not shown when the leg type is leg only.
  - If specify area and inertia is selected the section properties box will display sixteen edit fields to specify the area and inertia of the leg cross section and the pile cross section along with display diameter and a wave diameter (Figure 8-10). Note that pile cross section edit fields are not shown when the leg type is leg only.

- Two entry fields along with a check box to allow the user to specify local buckling factors. If the check box is selected, the specified values for the buckling factor about the Y and Z axes will be used for each element in the leg, otherwise the global buckling factors will be used.
- Two radio button lists along with a check box allow the user to specify local Cm factors. If the check box is selected, the specified values for the Cm factor about the Y and Z axes will be used for each element in the leg, otherwise the global Cm factors will be used. The possible choices for a Cm factor are: A, B, and C. If A is selected, then Cm is set to 0.85. If B is selected, the the Cm factor is calculated using the following formula:

  $$Cm = 0.6 - 0.4(M1/M2)$$

  If C is selected, the the Cm factor is calculated using the following formula:

  $$Cm = 1.0 - 0.4(fa/Fpe)$$

Button Area

The last area in the Platform Skeleton Legs dialogue box has the Propagate Values to Other Legs, Apply, and Cancel buttons.

- Propagate Values to Other Legs: Selecting this button will force all legs to have properties similar to those of the selected leg.
- Apply: Selecting this button will apply the selections currently displayed and close the Platform Skeleton Legs dialogue box. This button must be selected to make any changes effective.
- Cancel: Selecting this button will close the Platform Skeleton Legs dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.1.3.2 Platform Skeleton Horizontal Braces

The Platform Skeleton Horizontal Braces dialogue box has 2 major areas (Figure 8-11). The first area displays the properties of a horizontal brace. The second area is a group of buttons.

Horizontal Braces Properties Area

The horizontal braces properties area shows the following:

- Material name modeling the material properties of the horizontal brace. Note that there is a button labeled "Material" and next to it is an edit field. If the material button is pressed, it will display a Generic Material Information dialogue box corresponding to the material name entered in the

edit field. If there is no Generic Material with that name an appropriate error message is displayed.

- Two check boxs that enable or disable the generation of elements modeling the internal and perimeter elements.
- A combo box to select the element type. The user can select between the following: nonlinear truss, large displacement beam, linear beam, beam column, strut, and buckling beam.
- A radio button list to specify section properties along with a box describing the section. The available selections are: generate, specify OD and TH, and specify area and inertia.
  - If generate is selected, the section properties box will display four edit fields to specify minimum and maximum values for the KL/R ratio and and D/t ratio of the section (Figure 8-11). The model generator will use these two ranges to select appropriate sections from a standard tubular sections table.
  - If specify OD and TH is selected, the section properties box will display two edit fields to specify the outside diameter (OD) and wall thickness (TH) of the horizontal brace cross section (Figure 8-12).
  - If specify area and inertia is selected, the section properties box will display eight edit fields to specify the area and inertia of the horizontal brace cross section along with display diameter and a wave diameter (Figure 8-13).
- Two entry fields along with a check box to allow the user to specify local buckling factors. If the check box is selected, the specified values for the buckling factor about the Y and Z axes will be used for each horizontal brace, otherwise the global buckling factors will be used.
- Two radio button lists along with a check box to allow the user to specify local Cm factors. If the check box is selected, the specified values for the Cm factor about the Y and Z axes will be used for each horizontal brace, otherwise the global Cm factors will be used. The possible choices for a Cm factor are: A, B, and C. If A is selected, then Cm is set to 0.85. If B is selected, the the Cm factor is calculated using the following formula:

  $$Cm = 0.6 - 0.4(M1/M2)$$

  If C is selected, the the Cm factor is calculated using the following formula:

  $$Cm = 1.0 - 0.4(fa/Fpe)$$

Button Area

The last area in the Platform Skeleton Horizontal Braces dialogue box has the Apply, and Cancel buttons.

- Apply: Selecting this button will apply the selections currently displayed and close the Platform Skeleton Horizontal Braces dialogue box. This button must be selected to make any changes effective.
- Cancel: Selecting this button will close the Platform Skeleton Horizontal Braces dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.1.4 Soil Profile

In the Soil Profile part of a platform skeleton description (Figure 8-2), the user specifies whether soil profile and soil spring (PSAS) elements are going to be generated. The generation of soil profiles is a must if the model is to include nonrigid supports. The Soil Profile box has a check box that will enable or disable the generation of soil data. The Geometry box also has a button labeled "Edit Soil Profile" that, when pressed, will display the Soil Control Information dialogue box.

The Soil Control Information dialogue box (Figure 8-14) has the following:

- A list of radio buttons to toggle between virgin and degraded soil response <Virgin>.
- An edit field for the effective lateral pile diameter.
- An edit field for the effective axial pile diameter.
- An edit field for the effective tip pile diameter.
- A list of radio buttons to toggle between soil profile or force-displacement table soil models <Soil Profile>. Soil profile means that you will enter the basic properties of the soil (e.g. clay or sand, unit weight, shear strength, etc) and let CapView generate the P-Y and T-Z curves. Force displacement table means that you will enter the actual P-Y and T-Z curves.
- If the soil model is specified as soil profile: an edit field for the number of soil profile elevations. The user must specify at least two soil profile elevations.
- If the soil model is specified as soil profile: a button labeled "Edit Soil Profile", which when pressed, will display the Soil Profile Properties" dialogue box.
- If the soil model is specified as soil profile: a button labeled "Show Soil Profile", which when pressed, will display a Soil Profile Report summurizing all the soil profile properties. This report can be printed but can not be edited (Figure 8-77).
- If the soil model is specified as force-displacement table (Figure 8-15): a button labeled Soil "Material" followed by an edit field. This field is used by CapView to identify the force displacement table. Next to the edit field there is a button labeled "?" that, when pressed, shows a scrollable list of force displacement table names already in use. The user can use any of these names or specify a new force displacement table name. When the button is pressed, CapView will get information about the force displacement table with the name typed in the edit field.
- If the soil model is specified as force-displacement table: a button labeled "New Soil Material" that when pressed will display the force-displacement table dialogue box, with a new table, using the name in the edit field if present.
- Apply button. Selecting this button will apply the selections currently displayed and close the Soil Control Information dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Soil Control Information dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.1.4.1 Soil Profile Properties

There are one or more Soil Profile Properties dialogue boxes (Figure 8-16). The number of these dialogue boxes is determined in the Soil Control Information dialogue box. The title of each of these dialogue boxes has a sequential number of the dialogue box and the total number of Soil Profile Properties dialogue boxes. Each Soil Profile Properties dialogue box has two areas. The first area has the soil properties at a given elevation. The second area has a group of control buttons.

Soil Properties

The soil properties area has the following:

- An edit field to specify the elevation. The elevation is entered as the vertical distance from the mud line (a positive number).
- An edit field for the submerged unit weight of the soil.
- A radio button list that toggles the soil type between Clay (cohesive) soil and Sand (cohesionless) soil. When the Clay type is selected (Figure 8-16), a box below the radio buttons displays the following Clay properties:
  - A radio button list that toggles the type of the clay between soft and stiff.
  - An edit field for the undrained shear strength.

  When the Sand type is selected (Figure 8-17), the box below the soil type radio buttons displays an edit field for the sand angle of friction.

Control Buttons

The control buttons in the Soil Profile Properties dialogue box are labeled Next, Prev, Apply, Done, and Cancel.

- Next: Selecting this button will advance the soil profile one level. If at the last level, selecting this button will roll back to the first level.
- Prev: Selecting this button will set the soil profile one level backward. If at the first level, selecting this button will roll back to the last level.
- Apply: Selecting this button will apply the selections currently displayed for the elevation. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover old selections for the elevation since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

8.1.4.2 Generic Soil Material

The Generic Soil Material dialogue (Figure 8-18) is used to define a force-displacement table for the soil model. The P-Y, T-Z and Q-Z multilinear relationships refer to discrete nonlinear springs for lateral, axial and tip behavior, respectively. The P-Y, T-Z and Q-Z multilinear relationships must be defined point by point.

Edit Fields

At the top of the Generic Soil Material dialogue there are three edit fields that apply to the entire force-displacement table.

- The Name is used to identify the soil force-displacement table.
- The Submerged Unit Weight <0.0> is used to calculate soil mass density for radiation damping values. Since it is a submerged weight, the soil/water mass density is approximated by the air weight of soil divided by acceleration of gravity. This neglects the water mass in the solid voids.
- The Shear Modulus for Radiation Damping <0.0> is used to determine the radiation damping coefficients for dynamic behavior.

In the middle of the Generic Soil Material dialogue there are two columns of edit fields. The values on the left are used for lateral movement, the values on the right are used for axial movement.

- The Rate Effect Parameter <1.0> scales the ordinates of the static P-Y or T-Z/Q-Z load deflection curves to account for strain rate effects, and obtains dynamic P-Y or T-Z/Q-Z ordinates.
- The Degradation Rate Parameter <0.0> characterizes the cyclic behavior of the soil. It is applied to the ultimate strength of the P-Y or T-Z/Q-Z curves, at the completion of each degradation cycle. A cycle occurs when deformation during loading exceeds the significant deformation value in both a positive loading direction and in a negative loading direction.
- The Significant Deformation for Cyclic Degradation <0.0> is the displacement which has to be exceeded to cause cyclic degradation in the P-Y or T-Z/Q-Z load deflection curves. Degradation information for a particular direction of loading, say the positive direction, is accumulated only while response is moving up a loading curve. Deformation in a gap is ignored. If the accumulated deformation exceeds the significant deformation prior to load reversal, then one half of a degradation cycle has occurred. If the accumulated deformation has not exceeded the significant deformation prior to load reversal, the accumulated positive deformation is reset to zero and will start fresh at the next positive loading cycle. A complete cycle of degradation has occurred when both the positive and negative deformation directions have satisfied the criteria. When a complete cycle has occurred, the appropriate load-deflection curve will be degraded during the step when load reversal occurs. The total deformation could always be positive and yet degradation could occur.
- The Number of Points is used to set the number of points that will be used to define the lateral and axial force-displacement curves.

The last edit fields in the Generic Soil Material dialogue define the force-displacement curves. This is where the lateral and axial values for the force-displacement curves are entered. Up to nine points are allowed to describe the multilinear virgin and degraded P-Y or T-Z/Q-Z load-deflection curves. In this table, the normalized virgin ordinate and the normalized degraded ordinate, along with the corresponding normalized abscissa value, are defined for the points along the P-Y curve and the T-Z/Q-Z curves. The line segments connect the origin to point 1, point 1 to point 2, etc. Note that the origin is assumed to be at the coordinates (0, 0), so you should not enter all zeroes for point 1.

Control Buttons

In addition to the above, the dialogue box also has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first table defined. Tables are in alphabetical order.

- Last button. Selecting this button will close the dialogue box and open an new one showing the last table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next table. If the current table is the last table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous table. If the current table is the first table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open a new one showing the table with the given name. If there is no table matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the named table so that it will reflect any modifications the user applied to the dialogue box.
- Add button. Selecting this button will add a new table with properties similar to those displayed in the dialogue box. However, if there is a table with a matching name, the new table will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed table.
- Done button. Selecting this button will close the dialogue box.

8.1.5 Platform Skeleton Data Control Buttons

In addition to the above four major parts, the Platform Skeleton Data dialogue box has the following:

- An edit field to assign a name to a set of Platform Skeleton Data, so that it can be stored on disk to be used again. Note that next to the entry field there is button with an arrow. When selected, this button will open a radio button list for the user to select from the available Platform Skeleton Data on the disk. Selecting any of the displayed names will close the radio button list and copy the name to the edit field. The user can also use the edit field to enter any name.
- A button labeled "Read" to read Platform Skeleton Data which has a name matching that in the name edit field.
- A button labeled "Write" to write Platform Skeleton Data.
- A button labeled "Generate Skeleton" to generate the skeleton described. When this button is selected, the model generation takes control of CapView and starts creating the necessary records representing the platform skeleton elevation by elevation, then leg by leg. The user can see the nodes and members as they are created and the CapView status line displays an appropriate message.
- A button labeled "Cancel Skeleton" to cancel the generated skeleton. When this button is selected, a confirmation dialogue box is displayed (Figure 8-18). If the user selects the Yes button, all the elements, element groups, nodes, loads and soil data are deleted from the current open project and the confirmation dialogue box disappears. If the user selects No, the confirmation dialogue box disappears without any further action.
- A button labeled "Done" to exit the Platform Skeleton Data dialogue box.

8.2 Add Members/Nodes

The second selection of the Model menu will display the Add Members/Nodes dialogue box (Figure 8-19). This dialogue box allows the user to add various configurations of members and nodes.

There are three major sections in this dialogue box. The first section is used to select the configuration of members or nodes to be added. The second section is used to specify the properties of the members or nodes. The Third section provides the user with instructions on how to add the selected configuration and how to undo it. In addition to these three sections, there is a button at the bottom of the dialogue box labeled "Done", that if pressed, will close the Add Members/Nodes dialogue box.

Note that when this dialogue box is active the rest of CapView is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

8.2.1 Configuration Selection

The configuration section of the Add Members/Nodes dialogue box has two combo boxes.

The first combo box is labeled configuration. The available selections are:

- Single Member (Figure 8-20)
- X-Brace (Figure 8-21)
- K-Brace (Figure 8-22)
- Diamond Brace (Figure 8-23)
- Triangular Brace (Figure 8-24)
- T-Brace (Figure 8-25)
- Split (Figure 8-26)
- Connect (Figure 8-35)
- Join (Colinear) (Figure 8-36)
- Duplicate Frame (Figure 8-37)
- Catenary (Figure 8-38)
- PSAS Element (Figure 8-44)
- Single Node (Figure 8-45)
- Line of Nodes (Figure 8-46)
- Grid of Nodes (Figure 8-47)

The first eleven selections deal with member operations: adding, splitting, and joining in various configurations. The twelfth selection is for adding soil spring (PSAS) elements, and the last three selections are for adding nodes.

The second combo box is for the Split option and is shown only if Split is the selected configuration (Figure 8-26). Available selections are: Ratio, Length, Equal Parts, YZ Plane, XZ Plane, and XY Plane. If Split is the selected configuration a box underneath the Split Type combo box is shown. This box has additional Split-related edit fields. The edit fields are labeled differently for different split types.

When the split type is Ratio, the edit field is used to edit the ratio of length at which members will be split (Figure 8-26). When the split type is Length, the edit field is used to edit the length at which members will be split (Figure 8-27). When the split type is Equal Parts, the edit field is used to edit the number of equal parts which members will be split into (Figure 8-28). When

the split type is YZ Plane, the edit field is used to edit the X coordinate at which members will be split (Figure 8-29). When the split type is XZ Plane, the edit field is used to edit the Y coordinate at which members will be split (Figure 8-30). When the split type is XY Plane, the edit field is used to edit the Z coordinate at which members will be split (Figure 8-31).

8.2.2 Properties Section

The properties section of the Add Members/Nodes dialogue box shows different data depending on the selected configuration. When any of the first eleven configurations is selected, the properties section shows member data, and the section is labeled "Member Data" (Figure 8-31). When the twelfth configuration is selected, the properties section shows soil spring (PSAS) element data, and the section is labeled "PSAS Data". When any of the last three configurations is selected, the properties section shows node group data, and the section is labeled "Node Data".

8.2.2.1 The member data includes the following:

- Member type combo box. This combo box is used to select the type of the element to be added. Available selections are: Nonlinear Truss, Large Displacement Beam, Linear Beam, Beam Column, Wave Load, Strut, Buckling Beam and Shim.
- Element name edit field. This field is used by CapView to name the next member to be added. CapView expects to get an integer number because it automatically increment this number for use when the next element is to be added. When adding a configuration of elements that requires adding more than one element, CapView increments this number each time it adds a new element.
- A button labeled Group followed by an edit field. This field is used by CapView to determine the group of elements to which the new element will be added. Next to the edit field there is a button labeled "?" that, when pressed, shows a scrollable list of group names already in use. The user can use any of those groups or specify a new group name. When the button is pressed, CapView will try to get information about a group that has a name similar to that in typed in the edit field (Figure 5-15). Note: there is a limit of 50 elements per group, except for PSAS elements which have no limit.
- Node name edit field. This field is used by CapView to name the next node to be added. CapView expects to get an integer number because it will automatically increment this number for use when the next node is to be added. When adding a configuration of elements that requires adding more than one node (e.g. a K-Brace, X-Brace, diamond brace, etc), CapView increments this number each time it adds a new node.
- A button labeled "New Group" that when pressed will generate a unique group name using the current group name. CapView expects to get a group name made up of two parts. The first part is a string from the following "Group", "grp", or "g". The second part is an integer number. If you want ot use a more descriptive group name, such as "Frame A", "Plan 1", etc, simply type its name in the group edit field described above.
- A button labeled Material followed by an edit field. The string in this field is used by CapView to determine the material properties of the new elements that will be added. Next to the edit field there is a button labeled "?" that when pressed will show a scrollable list of material names already in use. The user can use any of these material names or specify a new

material name (as long as the user adds a new material record). When the button is pressed CapView will try to get information about a material that has a name similar to that typed in the edit field (Figure 5-16).

- K node edit field. The string in this field is used by CapView to determine the K node of the new elements that will be added. Next to the edit field there is a button labeled "?" that when pressed will show a scrollable list of node names already in use. The user can use any of these node names or specify a new node name (as long as the user adds a new node record).
- A radio button list to specify section properties along with a box describing the section. The available selections are: generate, specify outside diameter (OD) and wall thickness (TH), and specify area and inertia.
  - If generate is selected, the section properties box will display four edit fields to specify minimum and maximum values for the KL/R ratio and and D/t ratio of the section. CapView will use these two ranges to select appropriate sections from a standard tubular sections table (Figure 8-31).
  - If specify OD and TH is selected, the section properties box will display two edit fields to specify the OD and TH of the cross section (Figure 8-32).
  - If specify area and inertia is selected, the section properties box will display eight edit fields to specify the area and inertia of the cross section along with display diameter and a wave diameter (Figure 8-33).
  - If the section is an AISC rolled section (Figure 8-33), then an edit field is provided for the AISC section ID along with a button to display the AISC Section Information dialogue box for the selected section (Figure 5-20) and a button, labeled "?", that, when selected, will display a scrollable list of AISC sections in the AISC section table (Figure 5-21). Also, there are two edit fields for equivalent diameters for display and wave calculations. Display diameters are used to determine the width of a member in in Double-Line Drawing mode (section 5.2). Wave diameters are used for calculating the wave loads on the members.

When adding a Catenary, additional information is required to define the Catenary.

- The Generation Type combo box is used to choose the method that will be used to generate the catenary. The choices are: Tension at Top Node, Horizontal Load, Catenary Length, Angle w/Horizon at Bottom Node, Angle w/Horizon at Top Node and Mooring w/Bottom Interaction.
- The next edit field will accept a value that corresponds with the generation type chosen above.
- A button labeled Catenary Name followed by an edit field. This field is used to name the catenary that will be generated. When the button is pressed, a Catenary Information dialogue box will be displayed, if an existing catenary name is in the edit field. (see below)
- A radio button list to select a Sagging shape or a Straight shape catenary.
- The Number of Nodes (Excluding End Nodes) edit field to specify the number of nodes to be generated.
- The Weight in Water per Unit Length edit field to specify the water weight.

Catenary Information

The Catenary Information dialogue box gives details about the catenaries.

- The Name edit field is used to identify the catenary that is currently displayed.
- The SeaStar Id displays the SeaStar Id number of the catenary.
- The Bottom Node button and edit field are used to identify and get details about the bottom end node.
- The Top Node button and edit field are used to identify and get details about the top end node.
- The Generation Type combo box is used to choose the method that will be used to generate the catenary. The choices are: Tension at Top Node (Figure 8-38), Horizontal Load (Figure 8-39), Catenary Length (Figure 8-40), Angle w/Horizon at Bottom Node (Figure 8-41), Angle w/Horizon at Top Node (Figure 8-42), and Mooring w/Bottom Interaction (Figure 8-43).
- The next edit field will accept a value that corresponds with the generation type chosen above.
- A radio button list to select a Sagging shape or a Straight shape catenary.
- The Node Number Difference edit field is used to enter the amount each node's Id will be incremented.
- The Number of Nodes (Excluding End Nodes) edit field to specify the number of nodes to be generated.
- The Apparent Weight in Water / Unit Length edit field to specify the water weight.

Control Buttons

In addition to the above the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first catenary defined. Catenarys are in alphabetical order.
- Last button. Selecting this button will close the dialogue box and open an new one showing the last catenary.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next catenary. If the current catenary is the last catenary, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous catenary. If the current catenary is the first catenary, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open a new one showing the catenary with the given name. If there is no catenary matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the named catenary so that it will reflect any modifications the user applied to the dialogue box.
- Add button. Selecting this button will add a new catenary with properties similar to those displayed in the dialogue box. However, if there is a catenary with a matching name, the new catenary will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed catenary.

- Done button. Selecting this button will close the Catenary Information dialogue box.

8.2.2.2 The PSAS data includes the following (Figure 8-44):

- Pile name edit field. This field is used by CapView to name the next soil spring (PSAS) element to be added. CapView expects to get an integer number because it automatically increment the number for use when the next PSAS element is to be added.
- An edit field for the pile diameter.
- A check box labeled tip to toggle the type of the spring between tip and non-tip.

Note that when adding PSAS elements you have to specify 2 special nodes. The first node that you have to specify is the free-field node. This node should be a fixed node and is usually the same for all soil elements. It simply provides a reference point for all soil deformation. The second node is a node along the axis of the pile, is vertically above the pile node to which the soil element is attached and which is different from the free-field node. This node is usually the same for all the soil elements along a particular pile.

When adding PSAS elements, you should not add a tip PSAS element except at the lower tip of the pile and that at the same level you must add a non-tip PSAS Element.

8.2.2.3 The Node data includes the following (Figure 8-45):

- Node name edit field. This field is used by CapView to name the next node to be added. CapView expects to get an integer number because it automatically increment this number for use when the next node is to be added. When adding more than one node, CapView increments this number each time it adds a new node.
- Up to three sets of node location coordinates.
  - When the selected configuration is Single Node (Figure 8-45), the location coordinates of one node are shown. This set is labeled 1.
  - When the selected configuration is Line of Nodes (Figure 8-46), the location coordinates of two nodes are shown. The first node is labeled 1, and the second is labeled 2.
  - When the selected configuration is Grid of Nodes (Figure 8-47), the location coordinates of three nodes are shown. The first node is labeled 1, the second is labeled 2, and the third is labeled 3.
- When the configuration generates more than one node, up to two edit fields are provided:
  - When the selected configuration is Line of Nodes, an edit field labeled "Number of Nodes Along Line 1-2" is shown (Figure 8-46).
  - When the selected configuration is Grid of Nodes, an edit field labeled "Number of Nodes Along Line 1-2" and an other edit field labeled "Number of Nodes Along Line 1-3" are shown (Figure 8-47).

Instruction Section

The instructions section of the Add Members/Nodes dialogue box is a box that displays different instructions on keys to press to add the requested configuration and to undo it.

8.3 Read Member/Node/Load Files

The third selection of the Model menu will display the Read Member/Node/Load Files dialogue box (Figure 8-72). This dialogue box allows the user to add members and nodes to a new or existing model by reading the data from ASCII files.

The user must first prepare the files to be read following the format described bellow and then read the files one at a time. For each file, select the type of the file and enter the file name and the path to the file, then select the Parse or Read buttons.

The file types available for reading are:

- Node Coordinates
- Node Mass
- Node Fixity
- Member Connectivity
- Member Parameters
- Nodal Loads.

Input Fields

- File Name. Enter the name of the file here.
- Path. Enter the path to the file here. Note that it the file is in the directory that you started CapView, then you do not have to enter anything in this field.

Control Buttons

- Parse button. Selecting this button will read the specified file and check for errors (such as missing nodes).
- Read button. Selecting this button will read the specified file and add the nodes, members, or nodal loads described in the file to the model.
- Done button. Selecting this button will close the dialogue box.

The files are free format ASCII files. The data must be in consistent English units (Kips and Ft). Note that you can include comment lines by inserting '#' in column 1 and that you shouldn't have empty lines in the files. Following are the file formats:

8.3.1 Node Coordinates

NodeNumber X-Coordinate Y-Coordinate Z-Coordinate

Example

```
1  0.0 0.0 0.0
2  -20.0 0.0 0.0
```

Note that the added nodes will have six free and unslaved degrees of freedom and zero mass and that if you specify a node that already exists in the CAP model the data line will be ignored and flaged as an error.

8.3.2 Node Mass

NodeNumber Xm Ym Zm Xr Yr Zr

where Xm is mass in X direction and

Zr is rotational inertia about X

Example
1 3.0 3.0 3.0 0.0 0.0 0.0
2 3.0 3.0 3.0 0.0 0.0 0.0
Note that if you specify a node that does not exist in the CAP model the data line will be ignored and flaged as an error.

8.3.3 **Node Fixity**

NodeNumber X Y Z XX YY ZZ

where X is translation in X direction and
ZZ is rotation about Z direction

Enter 1 for a fixed dof or 0 for a free dof.

Example
1 1 1 1 0 0 0
2 1 1 1 1 1 1
Note that if you specify a node that does not exist in the CAP model the data line will be ignored and flaged as an error.

8.3.4 **Member Connectivity**

GrpName MemNumber ElemType NodeI NodeJ NodeK SecProperties

where ElemType can be one of the following:
1 Non-Linear Truss
2 Large Deformation Beam
3 Linear Beam
4 Beam Column
7 Wave Load Element
8 Strut
11 Shim
12 Buckling Element

and SecProperties can be one of the following
TUBE OD WallTh
AISC AiscID
GENERAL Ax Ay Az Ix Iy Iz
Note that you enter the string TUBE followed by the values

Example
NewBC 15 4 1 2 12 GENERAL 0.256 0.205 0.205 0.246 0.1229 0.1229
NewStrut 16 8 2 4 12 TUBE 2.0 0.0416667
NewLB 16 3 2 4 12 AISC W12X120
Note that if you specify an element that already exists in the CAP model the data line will be ignored and flaged as an error.

8.3.5 **Member Parameters**

GrpName MemNumber Property

where Property can be one of the following
Cd DragCoefficientValue
Cm InertiaCoefficientValue
Ctd TangentialDragCoefficientValue
Ckm AddedMassCoefficientValue
Buoyancy BuoyancyValue

where BuoyancyValue can be one of the following
-1 No Buoyancy
0 Varied Buoyancy w/o Fluid (Buoyant)
1 Varied Buoyancy w/ Fluid (Flooded)
10 Invaried Buoyancy w/o Fluid (Buoyant)
11 Invaried Buoyancy w/ Fluid (Flooded)

Note that you enter the string Cd followed by the value

Example
New 15 Cd 0.6
New 15 Cm 0.5
New 15 Buoyancy 1
Note that if you specify an element that does not exist in the CAP model the data line will be ignored and flaged as an error.

8.3.6 **Nodal Loads**

LoadCase NodeNumber FX FY FZ MX MY MZ

Example
Wind 1 1.0 0.0 0.0 0.0 0.0 0.0
Gravity 1 0.0 0.0 2.0 0.0 0.0 0.0
Note that if you specify a node that does not exist in the CAP model the data line will be ignored and flaged as an error. Also, if you provide two data lines that specify the same node and load case, then the forces on the two lines will be added together.

8.4 Damping Parameters

The fourth selection of the Model menu will display the Damping Parameters dialogue box labeled Show Structure Damping (Figure 8-48). This dialogue box allows the user to calculate, view and set the values of mass proportional and stiffness proportional (Rayleigh) damping factors. Advanced users who are not using the Preconfigured Macros feature must use this dialogue box to set up the damping parameters for dynamic earthquake and wave analysis.

The first three edit fields are used to set values that CapView will use to calculate the damping factors.

- Target Damping Ratio
- Fundamental period
- Period of Highest Mode of Interest

Note that the fundamental period of the structure can be found by running an eigen analysis first. Next there is a button "Compute Damping Factors" followed by two edit fields. Pressing this button will cause the damping factors to be computed from the values in the previous three edit fields with the results appearing in the two edit fields following. The values in these edit fields may also be entered directly by the user.

- Mass Proportional Damping Factor
- Stiffness Proportional Damping Factor

Now there is a button labeled "Plot Damping" preceded by two edit fields and a radio button list. This button will produce a plot of the damping factors using the current values for Mass Proportional Damping Factor and Stiffness Proportional Damping Factor. A dialogue box will be displayed showing the damping factors curve (Figure 8-49).

- The Plot File Name edit field is used to name a file which will be saved and can be printed or viewed later.
- The User Title edit field is used to enter a title for the damping factors plot.
- The Overwrite File, Append to File radio buttons are used to decide whether to overwrite or append an existing file.

Control Buttons

- Apply button. Selecting this button will change the values of mass proportional and stiffness proportional damping in the model. This button must be selected to make any changes effective.
- Done button. Selecting this button will close the dialogue box without changing the values of mass proportional and stiffness proprotional damping in the model. Any changes made will be lost unless Apply has been selected before selecting Done.

8.5 Fix X-Brace Center Node

The fifth selection of the Model menu will display the Fix X-Brace Center Node dialogue box (Figure 8-50). When an X-Brace is added (see section 8.2) it calculates the coordinate of the center node at the intersection point of the two straight lines from the diagonally opposite nodes that define the X. If you later define eccentricities for one or more of the X-Brace members, the coordinate of the center node is no longer precisely accurate and should be adjusted. This dialogue box allows the user to adjust the location of the center node of an X-Brace taking into account the end eccentricities of the four members of the X-Brace.

The dialogue box has an instructions box. This box displays instructions on keys to press to fix the location of an X-Brace center node and to undo it. There is also a button at the bottom of the dialogue box labeled "Done", that if pressed, will close the Fix X-Brace Center Node dialogue box.

Note that when this dialogue box is active the rest of CAP is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

8.6 Soil Profile

The sixth selection of the Model menu will display the Soil Control Information dialogue box (Figure 8-51). This dialogue box allows the user to generate, review, change, and/or delete soil profile data. Note that there is another dialogue box titled Soil Control Information that has similar functionality as this dialogue box integrated within the Generate Platform Skeleton selection (section 8.1). Use this menu option to edit the profile after you have generated a skeleton, or to create one if you didn't do so before.

The Soil Control Information dialogue box has the following:

- A list of radio buttons to toggle between virgin and degraded soil response <Virgin>.
- An edit field for the effective lateral pile diameter.
- An edit field for the effective axial pile diameter.
- An edit field for the effective tip pile diameter.
- A list of radio buttons to toggle between soil profile or force-displacement table soil models <Soil Profile>. Soil profile means that you will enter the basic properties of the soil (e.g. clay or sand, unit weight, shear strength, etc) and let CapView generate the P-Y and T-Z curves. Force displacement table means that you will enter the actual P-Y and T-Z curves.
- If the soil model is specified as soil profile: an edit field for the number of soil profile elevations. The user must specify at least two soil profile elevations.
- If the soil model is specified as soil profile: a button labeled "Show Soil Profile", which when pressed, will display the Soil Profile Properties" dialogue box.
- If the soil model is specified as force-displacement table: a button labeled Soil "Material" followed by an edit field. This field is used by CapView to identify the force displacement table. Next to the edit field there is a button labeled "?" that, when pressed, shows a scrollable list of force displacement table names already in use. The user can use any of these names or specify a new force displacement table name. When the button is pressed, CapView will get information about the force displacement table with the name typed in the edit field.
- If the soil model is specified as force-displacement table: a button labeled "New Soil Material" that when pressed will display the force-displacement table dialogue box, with a new table, using the name in the edit field if present. If no name is present in the edit field, an error is reported.
- Apply button. Selecting this button will apply the selections currently displayed and close the Soil Control Information dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Soil Control Information dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.5.1 Soil Profile Properties

There are one or more Soil Profile Properties dialogue boxes (Figure 8-16). The number of these dialogue boxes is determined in the Soil Control Information dialogue box. The title of each of these dialogue boxes has a sequential number of the dialogue box and the total number of Soil Profile Properties dialogue boxes. Each Soil Profile Properties dialogue box has two areas. The first area

has the soil properties at a given elevation. The second area has a group of control buttons.

Soil Properties

The soil properties area has the following:

- An edit field to specify the elevation. The elevation is entered as the vertical distance from the mud line (a positive number).
- An edit field for the submerged unit weight of the soil.
- A radio button list that toggles the soil type between Clay (cohesive) soil and Sand (cohesionless) soil. When the Clay type is selected (Figure 8-16), a box below the radio buttons displays the following Clay properties:
  - A radio button list that toggles the type of the clay between soft and stiff.
  - An edit field for the undrained shear strength.

  When the Sand type is selected (Figure 8-17), the box below the soil type radio buttons displays an edit field for the sand angle of friction.

Control Buttons

The control buttons in the Soil Profile Properties dialogue box are labeled Next, Prev, Apply, Done, and Cancel.

- Next: Selecting this button will advance the soil profile one level. If at the last level, selecting this button will roll back to the first level.
- Prev: Selecting this button will set the soil profile one level backward. If at the first level, selecting this button will roll back to the last level.
- Apply: Selecting this button will apply the selections currently displayed for the elevation. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover old selections for the elevation since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

8.5.2 Generic Soil Material

The Generic Soil Material dialogue (Figure 8-18) is used to define a force-displacement table for the soil model. The P-Y, T-Z and Q-Z multilinear relationships refer to discrete nonlinear springs for lateral, axial and tip behavior, respectively. The P-Y, T-Z and Q-Z multilinear relationships must be defined point by point.

Edit Fields

At the top of the Generic Soil Material dialogue there are three edit fields that apply to the entire force-displacement table.

- The Name is used to identify the soil force-displacement table.
- The Submerged Unit Weight <0.0> is used to calculate soil mass density for radiation damping values. Since it is a submerged weight, the soil/water mass density is approximated by the air weight of soil divided by acceleration of gravity. This neglects the water mass in the solid voids.

- The Shear Modulus for Radiation Damping <0.0> is used to determine the radiation damping coefficients for dynamic behavior.

In the middle of the Generic Soil Material dialogue there are two columns of edit fields. The values on the left are used for lateral movement, the values on the right are used for axial movement.

- The Rate Effect Parameter <1.0> scales the ordinates of the static P-Y or T-Z/Q-Z load deflection curves to account for strain rate effects, and obtains dynamic P-Y or T-Z/Q-Z ordinates.
- The Degradation Rate Parameter <0.0> characterizes the cyclic behavior of the soil. It is applied to the ultimate strength of the P-Y or T-Z/Q-Z curves, at the completion of each degradation cycle. A cycle occurs when deformation during loading exceeds the significant deformation value in both a positive loading direction and in a negative loading direction.
- The Significant Deformation for Cyclic Degradation <0.0> is the displacement which has to be exceeded to cause cyclic degradation in the P-Y or T-Z/Q-Z load deflection curves. Degradation information for a particular direction of loading, say the positive direction, is accumulated only while response is moving up a loading curve. Deformation in a gap is ignored. If the accumulated deformation exceeds the significant deformation prior to load reversal, then one half of a degradation cycle has occurred. If the accumulated deformation has not exceeded the significant deformation prior to load reversal, the accumulated positive deformation is reset to zero and will start fresh at the next positive loading cycle. A complete cycle of degradation has occurred when both the positive and negative deformation directions have satisfied the criteria. When a complete cycle has occurred, the appropriate load-deflection curve will be degraded during the step when load reversal occurs. The total deformation could always be positive and yet degradation could occur.
- The Number of Points is used to set the number of points that will be used to define the lateral and axial force-displacement curves.

The last edit fields in the Generic Soil Material dialogue define the force-displacement curves. This is where the lateral and axial values for the force-displacement curves are entered. Up to nine points are allowed to describe the multilinear virgin and degraded P-Y or T-Z/Q-Z load-deflection curves. In this table, the normalized virgin ordinate and the normalized degraded ordinate, along with the corresponding normalized abscissa value, are defined for the points along the P-Y curve and the T-Z/Q-Z curves. The line segments connect the origin to point 1, point 1 to point 2, etc. Note that the origin is assumed to be at the coordinates (0, 0), so you should not enter all zeroes for point 1.

Control Buttons

In addition to the above, the dialogue box also has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new one showing the first table defined. Tables are in alphabetical order.
- Last button. Selecting this button will close the dialogue box and open an new one showing the last table.
- Next button. Selecting this button will close the dialogue box and open an new one showing the next table. If the current table is the last table, an appropriate message will be displayed.
- Prev button. Selecting this button will close the dialogue box and open an new one showing the previous table. If the current table is the first table,

an appropriate message will be displayed.

- Get button. Selecting this button will close the dialogue box and open a new one showing the table with the given name. If there is no table matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the named table so that it will reflect any modifications the user applied to the dialogue box.
- Add button. Selecting this button will add a new table with properties similar to those displayed in the dialogue box. However, if there is a table with a matching name, the new table will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed table.
- Done button. Selecting this button will close the dialogue box.

8.7 Water Data

The seventh selection of the Model menu will display the Water Data dialogue box (Figure 8-52). This dialogue box allows the user to review, and/or change certain water and environment data.

The Water Data dialogue box has the following:

- An edit field for the water surface elevation.
- An edit field for the mud line elevation.
- An edit field for the water weight density.
- An edit field for the acceleration of gravity.
- Apply button. Selecting this button will apply the selections currently displayed and close the Water Data dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Water Data dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.8 Default Analysis Parameters

The eighth selection of the Model menu will display the Default Analysis Parameters dialogue box (Figure 8-53). This dialogue box allows the user to review, and/or change certain global analysis parameters. Generic elements use the default (global) value of a parameter whenever the "Local" flag of such a parameter is not selected.

The Default Analysis Parameters dialogue box has the following:

- An edit field for the number of entries describing the marine growth profile.
- A button labeled "Show Marine Growth Profile" that when pressed will display a Marine Growth Profile dialogue box.
- A pair of radio buttons to select between constant drag and inertia coefficients, and coefficients as a function of member diameter.
- An edit field for the drag coefficient and another for the inertia coefficient. These edit fields are shown when the constant coefficient selection is made.
- An edit field for the number of entries describing the drag and inertia coefficients as a function of diameter and a button labeled "Show Drag/Inertia Coefficients" that when pressed will dispaly a Drag/Inertia Coefficients dialogue box. These are shown when the coefficients as a function of member diameter selection is made.
- An edit field for the tangential drag coefficient.
- An edit field for the added mass coefficient.
- Apply button. Selecting this button will apply the values currently displayed and close the Default Analysis Parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Default Analysis Parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.7.1 Marine Growth Profile

The marine growth profile is a set of edit field pairs defining the marine growth in terms of increase in diameter at various Z-elevations. For a given depth, the increase in diameter is calculated by linear interpolation. The value of the increase in diameter changes linearly between each consecutive Z-elevations and stays constant above and below the maximum and minimum Z-elevations. Thus if you want no marine growth above water you should include one point just above the water surface (e.g. at 0.001) specifying 0 as the marine growth value, in addition to whatever points are required to define marine growth below water. The increase in diameter for a member is defined to be the increase in diameter at the center of the member (that is the average of the Z-elevations of the end nodes).

The Marine Growth Profile dialogue box has a scrollable list of edit field pairs (Figure 8-54). The first edit field of every pair is for the Z-elevation, the second edit field is for the growth (increase in diameter).

In addition the dialogue box has two buttons, Apply and Cancel.

- Apply button. Selecting this button will apply the selections currently displayed and close the Marine Growth Profile dialogue box. This button must be selected to make any changes effective.

- Cancel button. Selecting this button will close the Marine Growth Profile dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.7.2 Drag/Inertia Coefficient Table

The Drag/Inertia Coefficient Table is a set of edit field triples defining the drag and inertia coefficients at various member diameters. For a given diameter, the drag and inertia coefficients are calculated by linear interpolation. The values of the coefficients change linearly between each consecutive diamters and stays constant above and below the maximum and minimum diamters. For Nontubular members the equivalent diameter for wave calculations is used to select the drag and inertia coefficients.

The Drag/Inertia Coefficient Table dialogue box has a scrollable list of edit field tripls (Figure 8-74). The first edit field of every triple is for the diameter, the second edit field is for the drag coefficient and the third edit field is for the inertia coefficient.

In addition the dialogue box has two buttons, Apply and Cancel.

- Apply button. Selecting this button will apply the selections currently displayed and close the Drag/Inertia Coefficient Table dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Drag/Inertia Coefficient Table dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.9 Default Code Check Parameters

The ninth selection of the Model menu will display the Default Code Check Parameters dialogue box (Figure 8-55). This dialogue box allows the user to review, and/or change certain global code check parameters. Generic elements use the default (global) value of a parameter whenever the "Local" flag of such a parameter is not selected.

The Default Code Check Parameters dialogue box has the following:

- Two entry fields labeled K (buckling factor) about the Y and Z axes.
- Two radio button lists labeled Cm about the Y and Z axes. The possible choices for a Cm factor are: A, B, and C. If A is selected, then Cm is set to 0.85. If B is selected, the the Cm factor is calculated using the following formula:

  $$Cm = 0.6 - 0.4(M1/M2)$$

  If C is selected, the the Cm factor is calculated using the following formula:

  $$Cm = 1.0 - 0.4(fa/Fpe)$$

- An edit field for the load factor.
- An edit field for the yield stress.
- Apply button. Selecting this button will apply the selections currently displayed and close the Default Code Check Parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Default Code Check Parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

8.10 Node Select/Change

The tenth selection of the Model menu will display the Node Select/Change dialogue box (Figure 8-56). This dialogue box allows the user to select specific nodes, change node attributes, or delete nodes. The user can specify a single query or group of queries that can be logically combined for complex searches.

The dialogue box has two criteria sections. The upper section of the box contains criteria that can be searched for or changed. The second section controls select-only criteria - criteria that can be the basis for a search but that cannot be changed. Below the criteria sections is the values section. In this part of the dialogue box, the user specifies the values used for the search, or the values that will be assigned to the nodes in a change operation. The two check boxes below the values section control logical functions used in selecting and changing.

In addition to these sections, there is a button at the bottom of the dialogue box labeled "Done", that if pressed, will close the Node Select/Change dialogue box.

Note that when this dialogue box is active the rest of CAP is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

8.9.1 Logical Operations

A logical "OR" function is achieved by running a second (or further) query with the "Clear Previous Selection" check box not selected. CAP will add those nodes to the ones found in a previous search.

The "Exclude Matching Node / Apply Changes to Not Selected Nodes" flag implements a logical "NOT" function.

CAP implements a logical "AND" function by allowing the user to select nodes meeting several criteria in a single query. To use this feature, choose several search criteria for one search (i.e. location and slaving) and CAP will select only those nodes that match all of the criteria.

8.9.2 Selecting Nodes

A node can be selected based on its:

- Fixed degree of freedom (DOF) pattern
- Slaving
- Coordinates
- Translational Mass
- Rotational Mass
- Visibility (i.e. whether it is shown on screen or is hidden)
- Location
- SeaStar Id

Once a user chooses the criteria for the search, he can then choose values for the criteria.

Fixed DOF patterns include translational and rotational DOFs in X, Y, and Z.

Slaving criteria can be general (any slaved node) or specific (nodes slaved to a particular node).

Coordinates can be specified for any combination of X, Y and Z. The coordinate change may be absolute or relative.

Translational Mass may be specified for any combination of X, Y and Z values. The Trans Mass change may be absolute or relative.

Rotational Mass may be specified for any combination of X, Y and Z values. The Rot Mass change may be absolute or relative.

Visibility options are hidden or shown.

Location criteria can be specified as points, planes, or values above (or below) a supplied maximum (minimum). X, Y, and Z parameters are ANDed for searches, while the "<", ">", and "=" operators are ORed. SeaStar Ids may be used.

Once the criteria values are chosen, clicking on the search button will perform the search, and nodes which satisfy the selection criteria will be highlighted (drawn in a different pen). Note that the user can select (unselect) a node by clicking the <Meta> left (right) mouse button next to the node. For more information on this see the Using CapView section.

8.9.3 Changing Nodes

When CAP finishes selecting the nodes desired, changing them is accomplished by changing criteria values and clicking the change button. First, choose a criteria to change. The criteria changed does not have to be the same as that used in the select operation. Second, change the values for this criteria in the values section. Third, click on the change button. The selected nodes now have the new values.

8.9.4 Deleting Nodes

To delete selected nodes, click on the Delete button. The CAP undo command (<Control> Right Mouse Button) works for undoing delete operations.

8.9.5 Select/Change Example

Change nodes which are slaved to node "101" to be unslaved with fixed DOFs.

1- Set the select criteria to Slaving.
2- Choose Slaving as the shown criteria.
3- Set the "Slaved To" check box and enter 101 into the edit field.
4- Click the select button.
5- Select Fixed DOF criteria (along with Slaving).
6- Clear the "Slaved To" check box.
7- Choose Fixed DOF as the shown criteria.
8- Set all the DOF check boxes.
9- Click the change button.

8.11 Member Select/Change

The eleventh selection of the Model menu will display the Member Select/Change dialogue box (Figure 8-57). This dialogue box allows the user to select specific members, change member attributes, or delete members. The user can specify a single query or group of queries that can be logically combined for complex searches.

The dialogue box has two criteria sections. The upper section of the box contains criteria that can be searched for or changed. The second section controls select-only criteria - criteria that can be the basis for a search but cannot be changed. Below the criteria sections is the values section. In this part of the dialogue box, the user specifies the values used for the search or the values that will be assigned to the members in a change operation. The two flags below the values section control logical functions used in selecting and changing.

In addition to these sections, there is a button at the bottom of the dialogue box labeled "Done", that if pressed, will close the Member Select/Change dialogue box.

Note that when this dialogue box is active the rest of CAP is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

8.10.1 Logical Operations

A logical "OR" function is achieved by running a second (or further) query with the "Clear Previous Selection" flag not selected. CAP will add those members selected in the current search to the ones found in a previous search.

The "Exclude Matching Member / Apply Changes to Not Selected Members" flag implements a logical "NOT" function.

CAP implements a logical "AND" function by allowing the user to select members meeting several criteria in a single query. To use this feature, choose several search criteria for one search (i.e. a location, and an IR range) and CAP will select only those members that match all of the criteria.

8.10.2 Selecting Members

A member can be selected based on its:

- Group Name
- Element Type
- Buoyancy Type
- Material Name
- Section Type
- Save Results
- K - Node
- End Releases
- IR Range
- Member Visibility
- Marine Growth
- Drag Coefficient
- Tangential Drag Coefficient
- Added Mass Coefficient

- Inertia Coefficient
- Buckling Factor
- Cm Equation
- Strength Reduction
- Eccentricities
- CC Yield Stress
- CC Load Factor
- Equivalent Diameter
- Can Size
- Location
- Member Connectivity
- SeaStar Id
- Member Connectivity

All but the last four (location, member connectivity, SeaStar Id, Member Connectivity) can also be change criteria. Once a user chooses the criteria for the search, he can then choose values for the criteria.

8.10.3 Specifying Criteria Values

Group Name, Material Name, K - Node, Member Connectivity, and Strength Reduction require only a single, user-entered value for search.

The Element Type criteria may be any of the 12 elements from the combo box.

Buoyancy Type criteria may be any of the 5 buoyancy types from the combo box.

Section Types can be one of three types: Tube, AISC, or General. The user must also specify outside diameter and wall thickness for the tubular sections, AISC section ID, or the areas and inertias describing a general section.

Save Results requires a yes or no response as does the Member Visibility criteria (hidden or shown) and Member Fixity (All DOFs fixed or Some DOFs fixed).

The End Release criteria includes both force and moment releases in X, Y, and Z for both the I and J ends.

The Interaction Ratio Range criteria allows selection of one of the six ranges R1 - R6.

Marine Growth, Drag Coefficient, Tangential Drag Coefficient, Added Mass Coefficient, Inertia Coefficient, CC yield Values and CC Load Factors each require a single value and provide a check box to use local factors for these criteria. If the check box is selected, the specified values for the criteria will be used for each element, otherwise the global defaults will be used.

Two values and a check box control specification of the Buckling Factor. If the check box is selected, the specified values for the criteria will be used for each element, otherwise the global defaults will be used.

Two radio button lists along with a check box control specification of Cm Factors. If the check box is selected, the specified values for the Cm factor about the Y and Z axes will be used for each element, otherwise the global Cm factors will be used. The possible choices for a Cm factor are: A, B, and C. If A is selected, then Cm is set to 0.85. If B is selected, the the Cm factor is calculated using the following formula:

$$Cm = 0.6 - 0.4(M1/M2)$$

If C is selected, the the Cm factor is calculated using the following formula:

$$Cm = 1.0 - 0.4(fa/Fpe)$$

Member Eccentricities can be specified for End I or J and given specific X, Y, and Z values.

Equivalent Diameters may be specified for wave calculation and/or display. These changes may be relative or absolute.

Can Size may be specified by any combination of End Can I outside diameter, End Can I wall thickness, End Can J outside diameter or End Can J wall thickness. These changes may be relative or absolute.

Location criteria can be specified as points, planes, or values above (or below) a supplied minimum (maximum). X, Y, and Z parameters are ANDed for searches, while the "<", ">", and "=" operators are ORed.

SeaStar Ids may be specified.

Once the criteria values are chosen, clicking on the search button will perform the search, and members which satisfy the selection criteria will be highlighted (drawn in a different pen). Note that the user can select (unselect) a member by clicking the <Meta> left (right) mouse button next to the member. For more information on this see the Using CAP section (5.0).

8.10.4 Changing Members

When CAP finishes selecting members, changing them is accomplished by changing criteria values and clicking the change button. First, choose a criteria to change. The criteria changed does not have to be the same as that used in the select operation. Second, change the values for this criteria in the values section. Third, click on the change button. The selected members now have the new values.

8.10.5 Deleting Members

To delete selected members, click on the Delete button. The CAP undo command (<Control> Right Mouse Button) works for undoing delete operations.

8.10.6 Select/Change Example

Change members with Interaction Ratio Range 3 which are located in the X >= 500, Y >= 300 quarter space to have a strength reduction of 10%.

1- Set the select criteria to Location and IR Range.
2- Choose Location as the shown criteria.
3- Click the ">" and "=" buttons for X and Y and set 500 and 300 as the respective values.
4- Choose IR Range as the shown criteria.
5- Choose R3 as the preferred range.
6- Click the select button.
7- Unselect Location and IR Range as criteria and select Strength Reduction.
8- Choose Strength Reduction as the shown criteria.
9- Set 10 as the strength reduction value in the edit field.
11- Click the change button.

8.12 Show Selected Nodes

The twelfth selection of the Model menu will display the print file browser dialogue box labeled Show Selected Nodes (Figure 8-58). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file with a list of selected nodes. If there are no selected nodes an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

8.13 Show Selected Members

The thirteenth selection of the Model menu will display the print file browser dialogue box labeled Show Selected Members (Figure 8-58). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file with a list of selected members. If there are no selected members an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

8.14 Node/Member/Load Reports

The fourteenth selection of the Model menu displays the Node/Member/Load Reports dialogue box labeled Generate Node/Member/Load Reports (Figure 8-60). This dialogue box allows the user to generate a report about the selected nodes, members, or nodal loads.

The reports available for nodes are:

- Node Coordinates (Figure 8-61)
- Node Inertia (Figure 8-62)
- Node Slaving (Figure 8-63)
- Selected Node Loads (Figure 8-73)

The report available for loads is:

- Selected Loads (Figure 8-75)

The reports available for members are:

- Member Connectivity (Figure 8-64)
- Member Structural (Figure 8-65)
- Member Hydrostatic Analysis Parameters (Figure 8-66)
- Member Other Analysis Parameters (Figure 8-76)
- Member Data Check (Figure 8-67)
- Member Can/Stub (Figure 8-68)
- Member Eccentricity (Figure 8-69)

Control Buttons

- Apply button. Selecting this button will generate the selected report. A dialogue box will be displayed with the text of the report. There is a Search button to search through the text, and a Done button to close the view dialogue. A file with the extension .tbl is also created which may be viewed or printed later.
- Done button. Selecting this button will close the dialogue box.

8.14 AISC Sections

The fifteenth selection of the Model menu will display an AISC Section Information dialogue box (Figure 8-70). This is an electronic form of the AISC standard section table. The dialogue box allows the user to review the contents of the AISC section table, and add new sections to the table.

The data that the edit fields show have the same units as the data in the standard AISC section. Following is a description of the edit fields:

Name	Record name based on AISC section designation.
Ax	Area. (inch^2)
d	Nominal depth. (inch)
J	Torsional constant. (inch^4)
rg	The radius of gyration of a section comprising the compression flange plus 1/3 of the compression web area, taken about an axis in the plane of the web. (inch)
tw	Thickness of web. (inch)
bf	Width of flange. (inch)
tf	Thickness of flange. (inch)
k	Distance from outer face of flange to web toe fillet of rolled shape. (inch)
bf/2tf	The ratio of the flange width to twice the flange thickness.
Fy'	The theoretical maximum yield stress based on the width-thickness ratio of 1/2 the unstiffened compression flange, beyond which a particular shape is not compact. (ksi) (Specified as 0 if greater than 65 ksi).
d/tw	The ratio of the depth to the web thickness.
Fy'''	The theoretical maximum yield stress based on the depth-thickness ratio of the web below which a particular shape may be considered compact for any condition of combined bending and axial stresses. (ksi) (Specified as 0 if greater than 65 ksi).
d/Af	The ratio of the depth to the compression flange area. (1/inch)
ri	Minimum fillet radius, design. (inch)
ra	Minimum fillet radius, detailing value (inch) or flange toe value for M, S, C, MC shapes.
nt	Tensile group number per ASTM A6.
I	Moment of inertia about major or minor axis. (inch^4)
S	Elastic section modulus about major or minor axis. (inch^3)
r	Radius of gyration about major or minor axis. (inch)
Z	Plastic modulus about major or minor axis.

In addition to the edit fields the dialogue box has the following buttons:

- First button. Selecting this button will close the dialogue box and open an new AISC dialogue box showing the first section in the AISC table. Sections in the AISC table are in alphabetical order.
- Last button. Selecting this button will close the dialogue box and open an new AISC dialogue box showing the last section in the AISC table.
- Next button. Selecting this button will close the dialogue box and open an new AISC dialogue box showing the next section in the AISC table. If the current section is the last section in the table, an appropriate message will

be displayed.

- Prev button. Selecting this button will close the dialogue box and open an new AISC dialogue box showing the previous section in the AISC table. If the current section is the first section in the table, an appropriate message will be displayed.
- Get button. Selecting this button will close the dialogue box and open an new AISC dialogue box showing the AISC section for the given name. If the AISC table does not have a record matching the given name, an appropriate message will be displayed.
- Replace button. Selecting this button will modify the table so that the shown record will reflect any modification the user applied to the dialogue box. For the current version of CAP this button is disabled.
- Add button. Selecting this button will add a new record to the AISC table with properties similar to those displayed in the dialogue box. However, if the table has a record with a matching name, the new record will not be added and an appropriate error message will be displayed.
- Delete button. Selecting this button will delete the displayed record from the table. For the current version of CAP this button is disabled.
- Done button. Selecting this button will close the dialogue box.

8.16 Standard Tube Sections

The sixteenth selection of the Model menu will display the Standard Tube Sections dialogue box (Figure 8-71). This dialogue box is used to view and edit the standard tubular sections that are available to CapView when you choose to generate section sizes rather than specify them explicitly when generating a platform skeleton. The Standard Tube Sections dialogue box has the following:

- Tubular Sections List: This scrollable list box displays the available tubular sections. The sections will be displayed in sorted order. Each entry has a pair of edit fields, one for the outside diameter (OD) and the other for the wall thickness (TH). The values may be edited by moving the cursor to the edit field and changing it. To delete an entry change either of its values to zero.
- Add New Entry button: Select this button to add a new entry to the table of values. The new entry will be added to the end of the list. The values of OD and TH will both be 0.000 and can be changed to whatever values are desired.
- Apply button: If this button is selected the changes that have been made to the list will be applied and the dialogue box will be closed.
- Cancel button: If this button is selected then the changes will be discarded and the dialogue box will be closed.

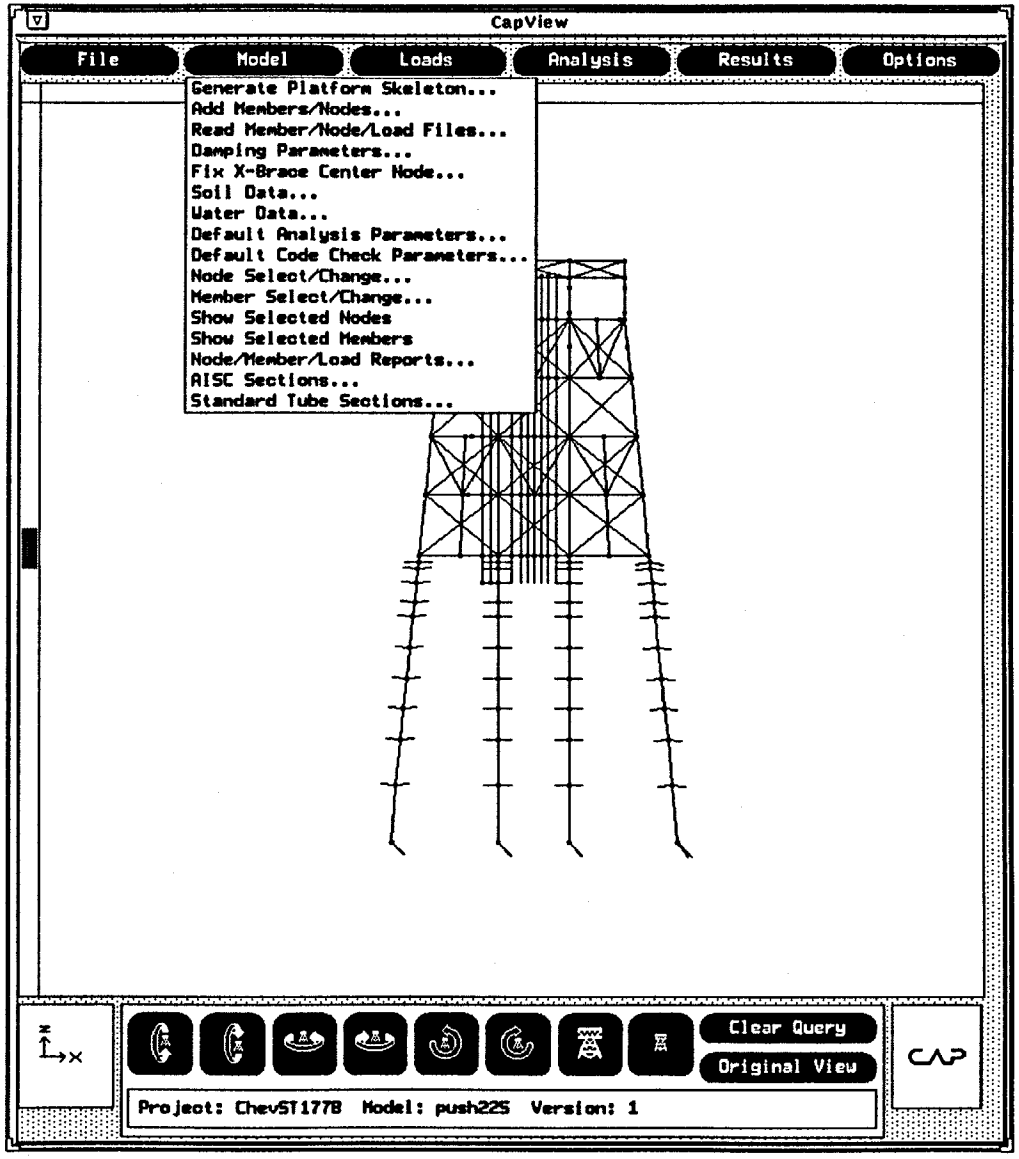

CapView
File
Model
Loads
Analysis
Results
Options
Generate Platform Skeleton...
Add Members/Nodes...
Read Member/Node/Load Files...
Damping Parameters...
Fix X-Brace Center Node...
Soil Data...
Water Data...
Default Analysis Parameters...
Default Code Check Parameters...
Node Select/Change...
Member Select/Change...
Show Selected Nodes
Show Selected Members
Node/Member/Load Reports...
AISC Sections...
Standard Tube Sections...
Clear Query
Original View
Project: ChevST177B Model: push225 Version: 1

Figure 8-1

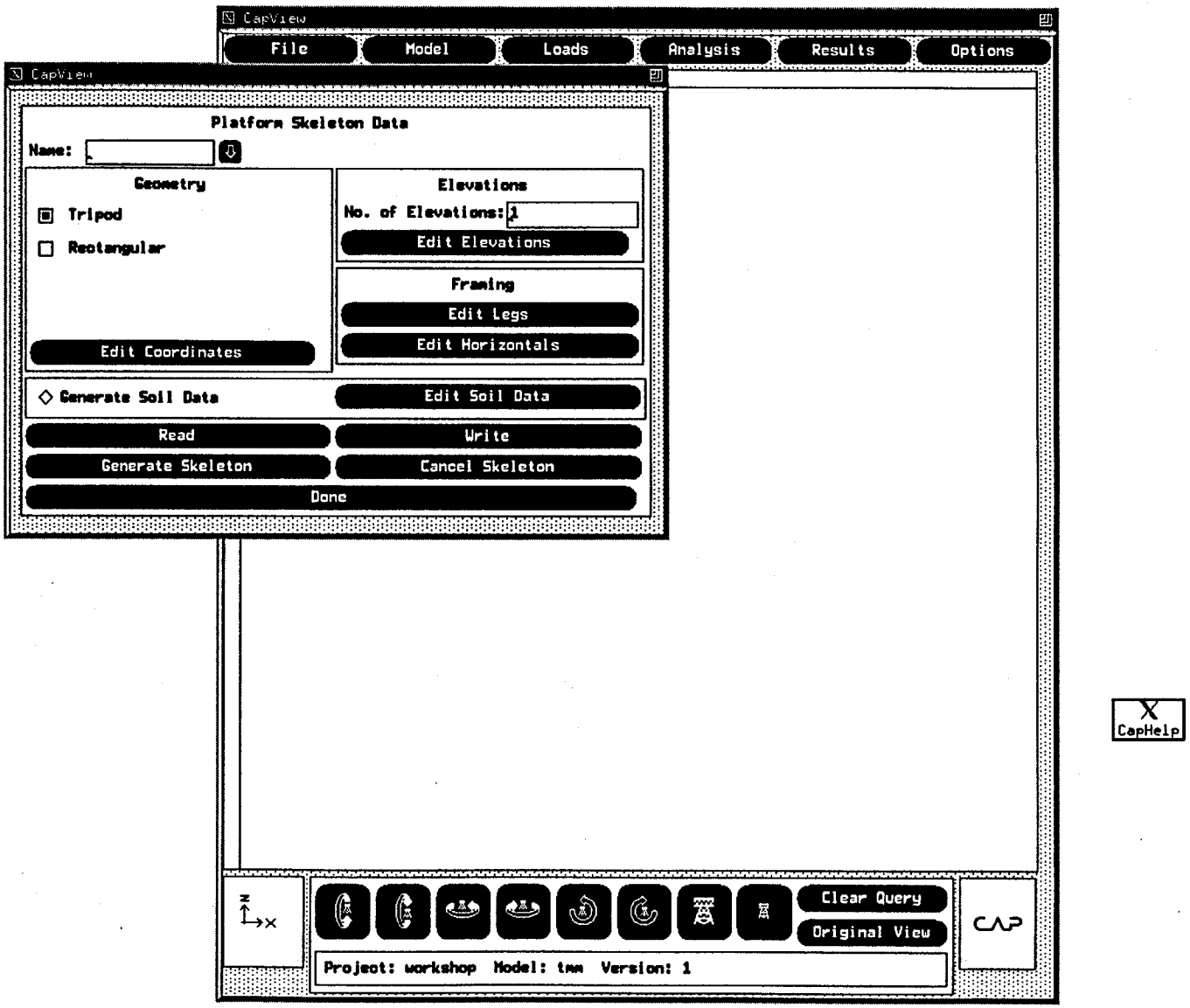
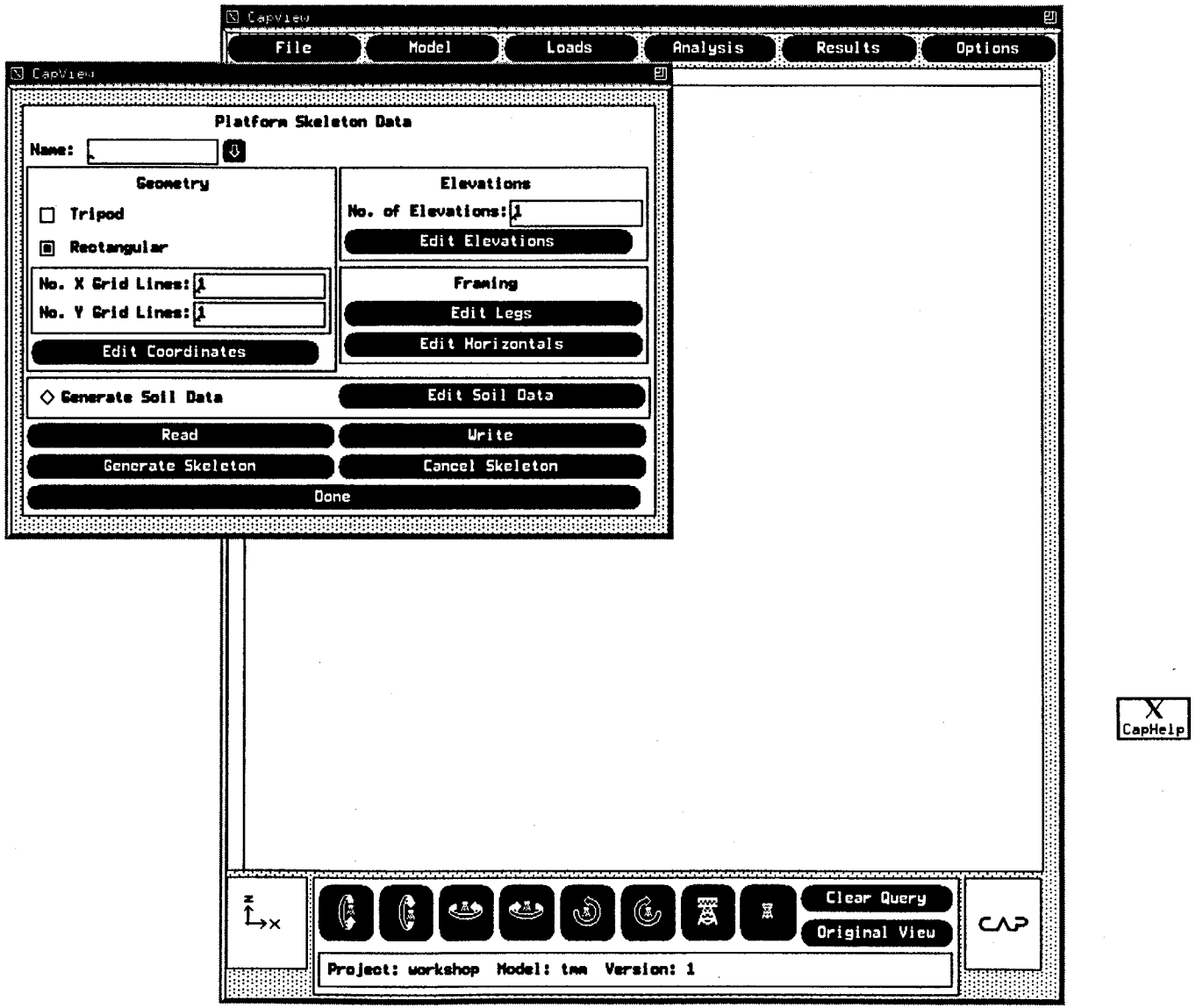

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Data
Name:
Geometry
Tripod
Rectangular
Edit Coordinates
Elevations
No. of Elevations: 1
Edit Elevations
Framing
Edit Legs
Edit Horizontals
Generate Soil Data
Edit Soil Data
Read
Write
Generate Skeleton
Cancel Skeleton
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CAP
X
CapHelp

Figure 8-2


CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Data
Name:
Geometry
Tripod
Rectangular
No. X Grid Lines: 1
No. Y Grid Lines: 1
Edit Coordinates
Elevations
No. of Elevations: 1
Edit Elevations
Framing
Edit Legs
Edit Horizontals
Generate Soil Data
Edit Soil Data
Read
Write
Generate Skeleton
Cancel Skeleton
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1

CapHelp

Figure 8-3

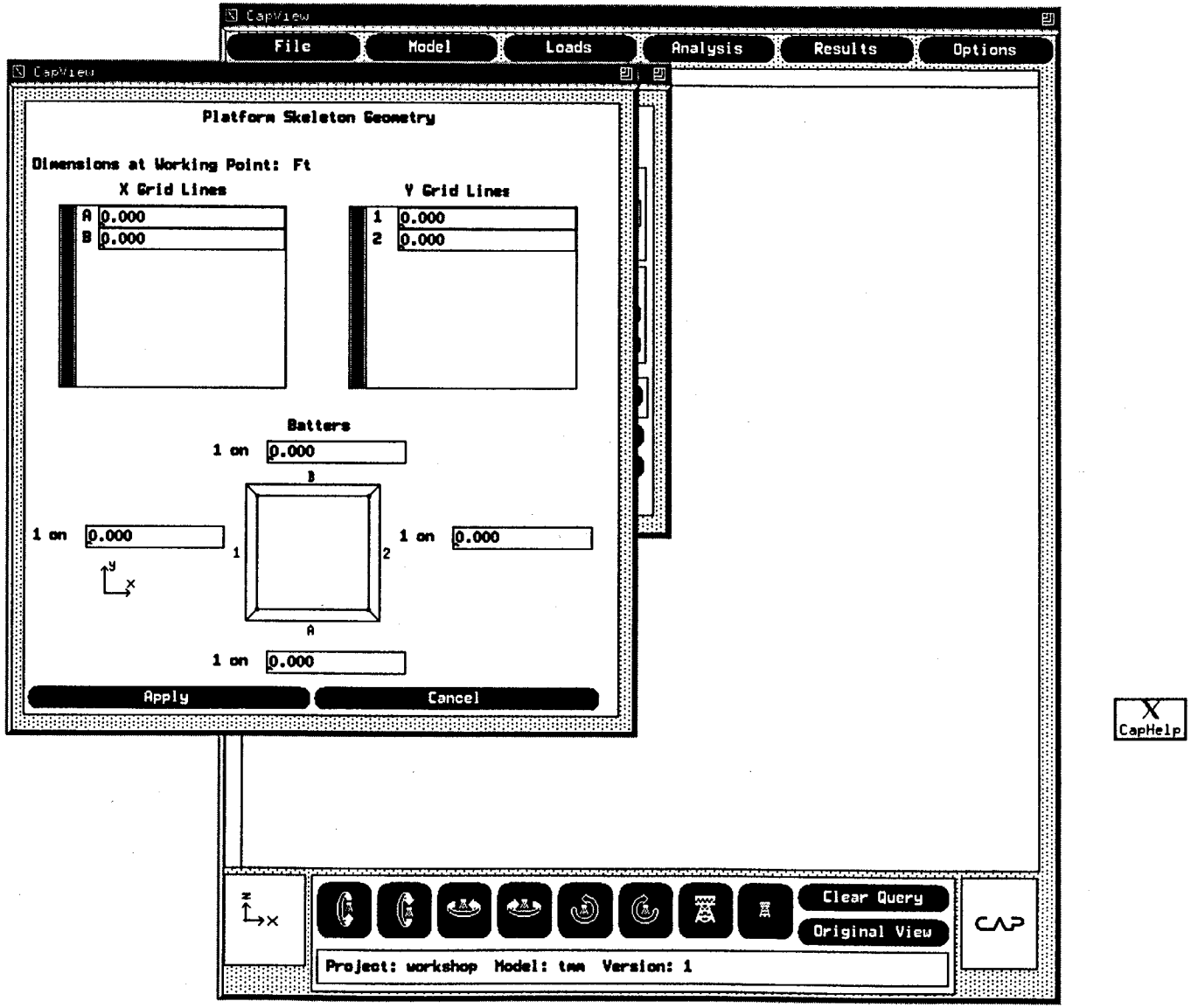

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Geometry
Dimensions at Working Point: Ft
X Grid Lines
A 0.000
B 0.000
Y Grid Lines
1 0.000
2 0.000
Batters
1 on 0.000
B
1 on 0.000
1
2
1 on 0.000
A
1 on 0.000
Apply
Cancel
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-4

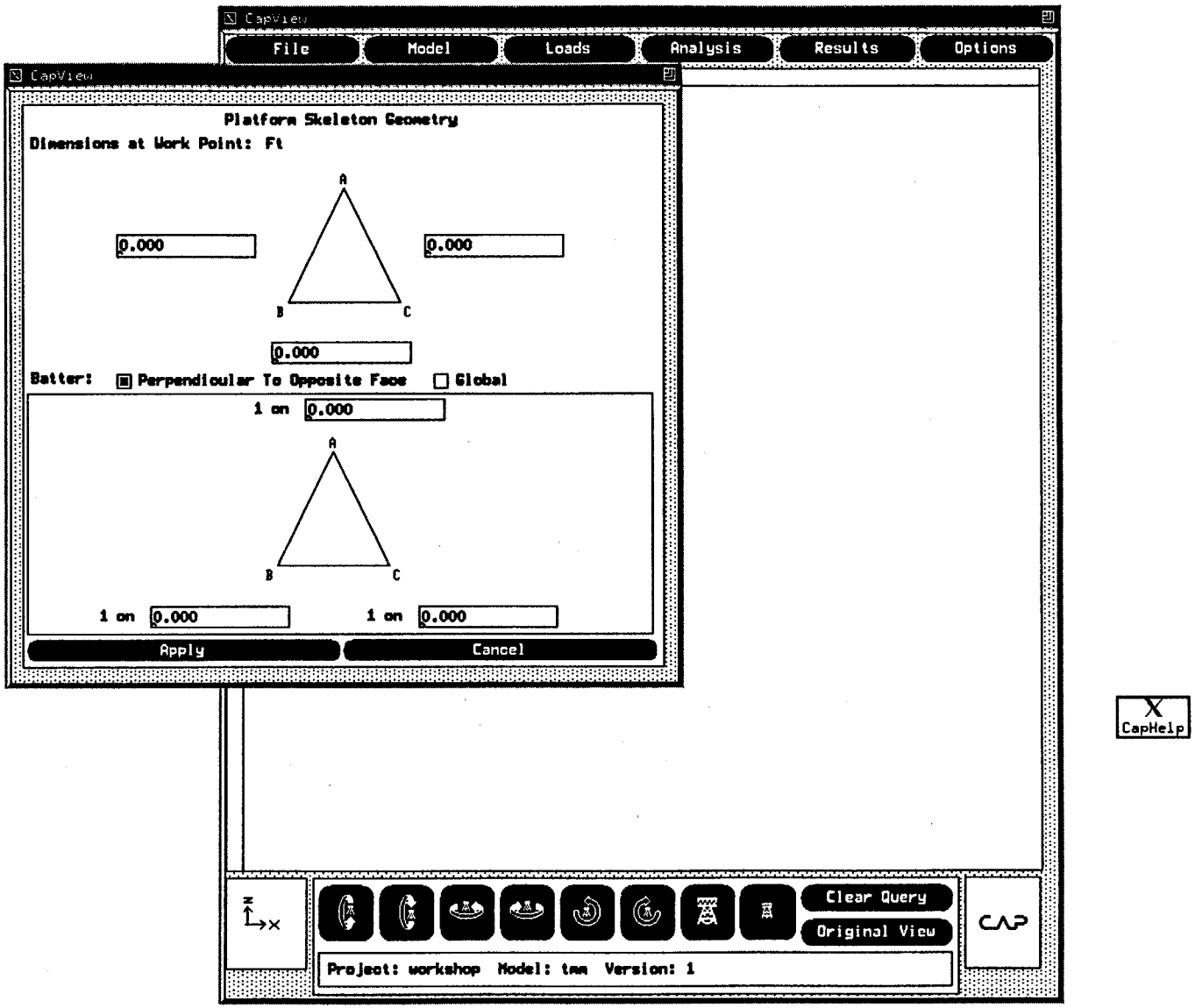

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Geometry
Dimensions at Work Point: Ft
A
0.000
0.000
B
C
0.000
Batter:
Perpendicular To Opposite Face
Global
1 on
0.000
A
B
C
1 on
0.000
1 on
0.000
Apply
Cancel
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-5

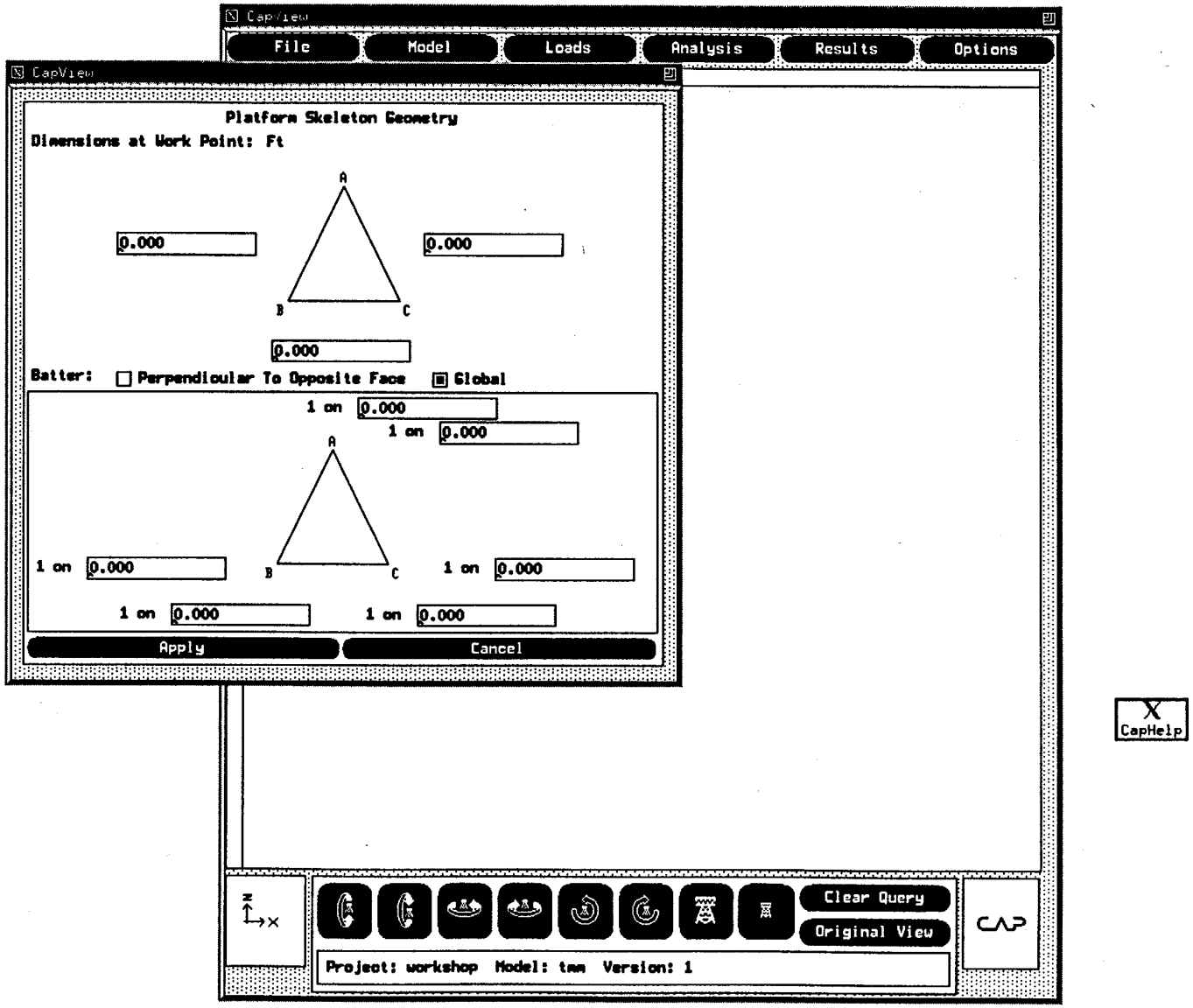

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Geometry
Dimensions at Work Point: Ft
A
B
C
0.000
0.000
0.000
Batter:
Perpendicular To Opposite Face
Global
1 on 0.000
1 on 0.000
A
B
C
1 on 0.000
1 on 0.000
1 on 0.000
1 on 0.000
Apply
Cancel
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-6

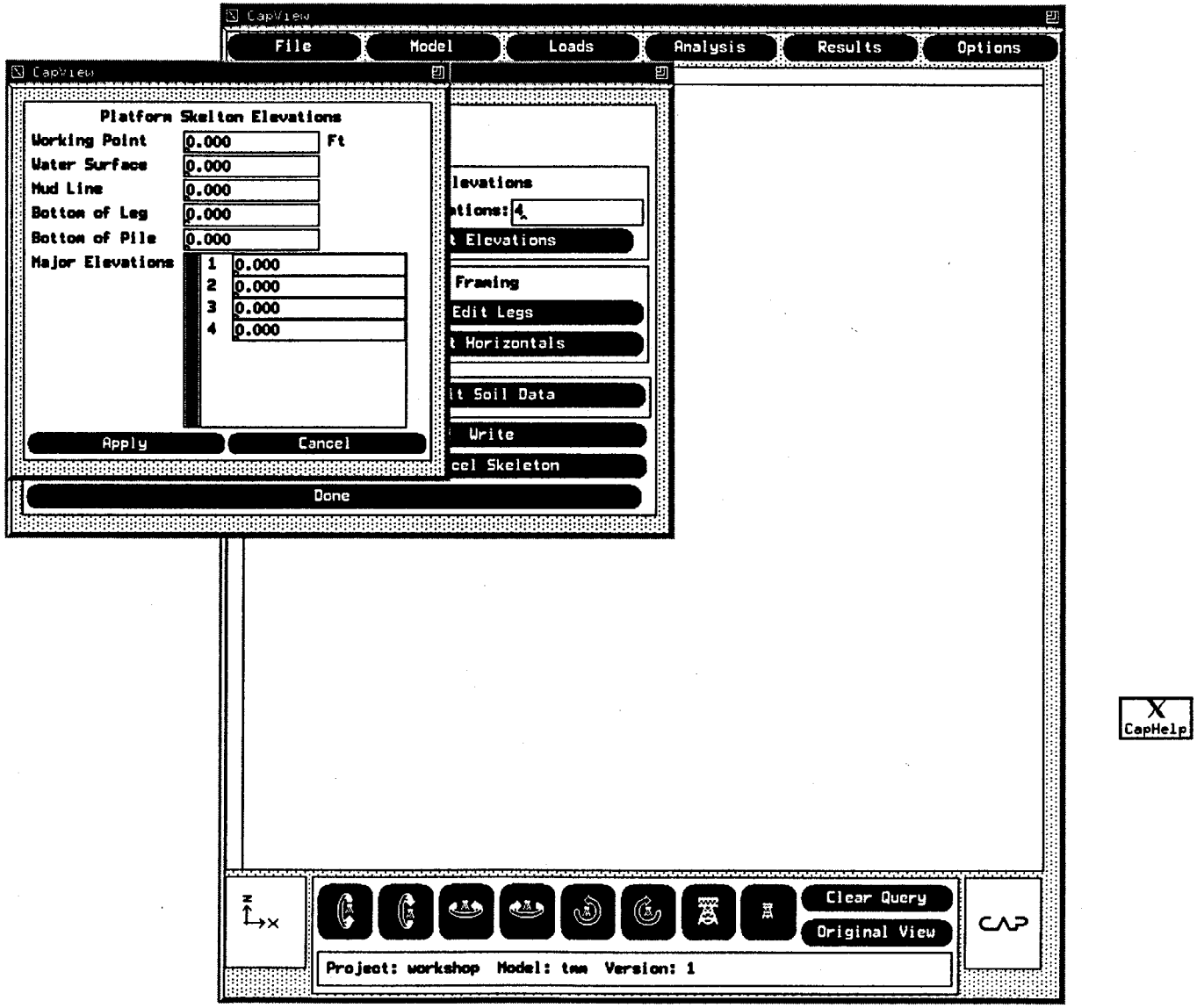

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skelton Elevations
Working Point
0.000
Ft
Water Surface
0.000
Mud Line
0.000
Bottom of Leg
0.000
Bottom of Pile
0.000
Major Elevations
1
0.000
2
0.000
3
0.000
4
0.000
Apply
Cancel
levations
tions:
4
t Elevations
Framing
Edit Legs
t Horizontals
it Soil Data
Write
cel Skeleton
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-7

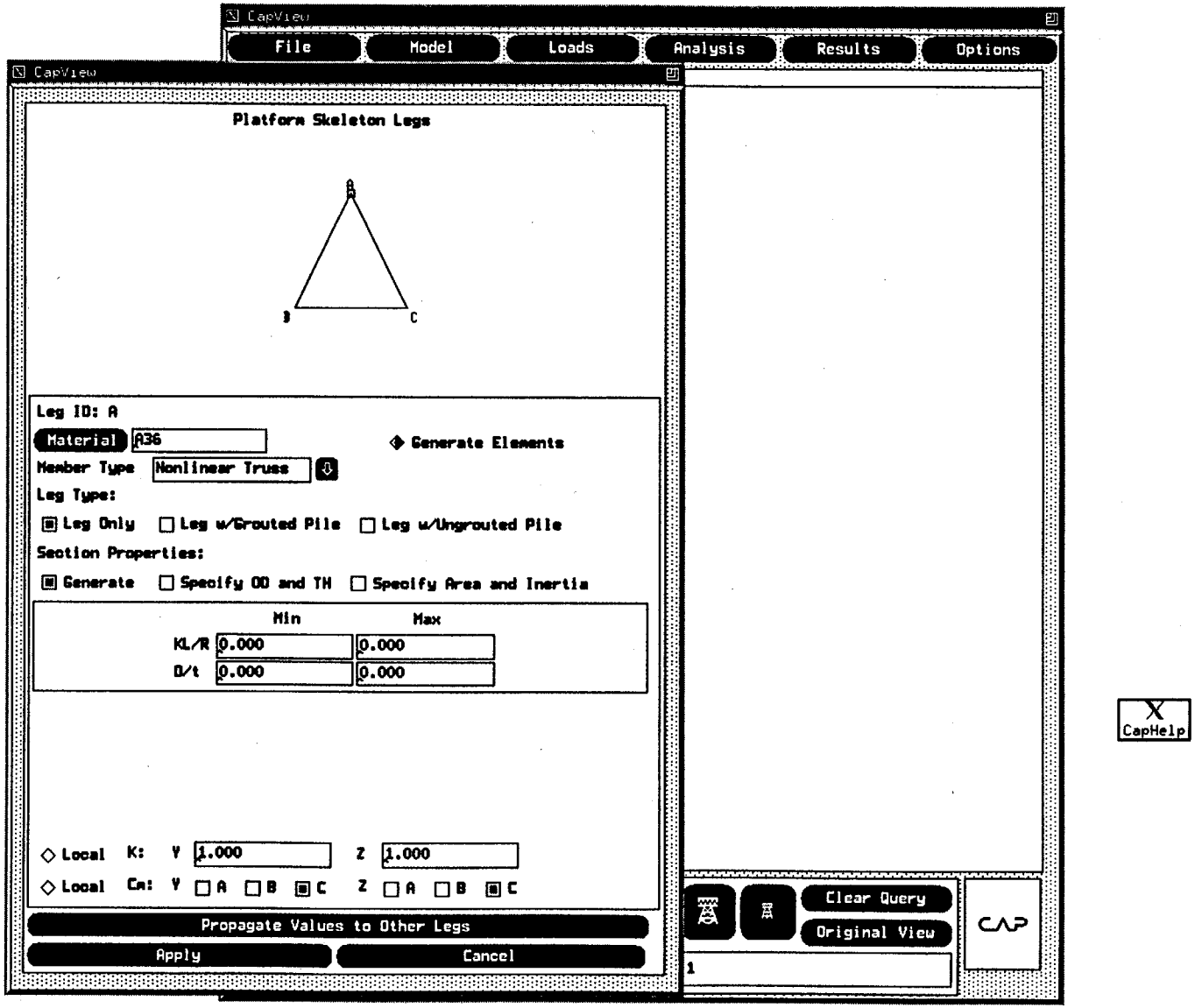
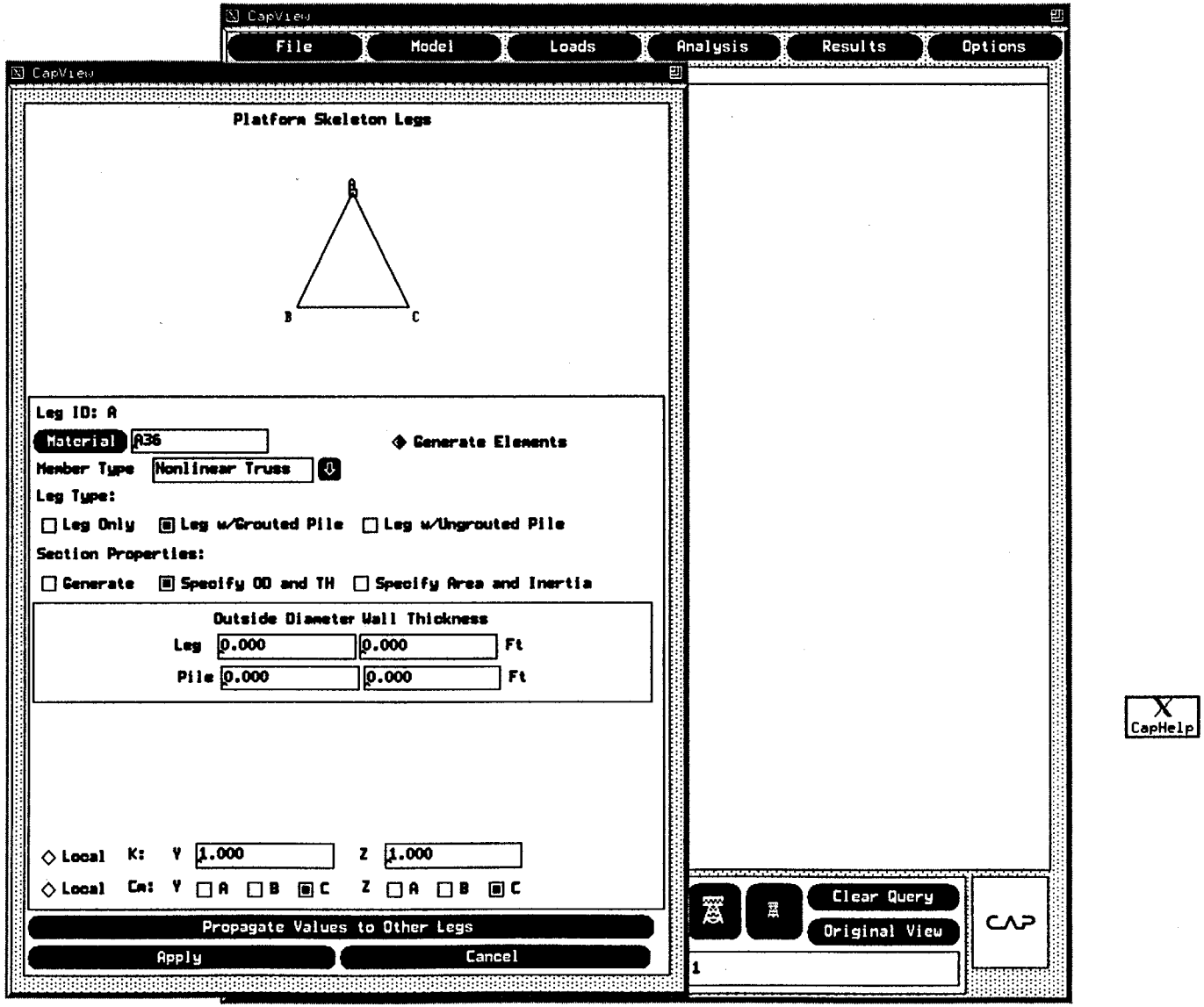

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Legs
A
B
C
Leg ID: A
Material
A36
Generate Elements
Member Type
Nonlinear Truss
Leg Type:
Leg Only
Leg w/Grouted Pile
Leg w/Ungrouted Pile
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Min
Max
KL/R
0.000
0.000
D/t
0.000
0.000
Local K: Y 1.000 Z 1.000
Local Cm: Y A B C Z A B C
Propagate Values to Other Legs
Apply
Cancel
Clear Query
Original View
CAP
1
CapHelp

Figure8-8


CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Legs
A
B
C
Leg ID: A
Material
A36
Generate Elements
Member Type
Nonlinear Truss
Leg Type:
Leg Only
Leg w/Grouted Pile
Leg w/Ungrouted Pile
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Outside Diameter Wall Thickness
Leg
0.000
0.000
Ft
Pile
0.000
0.000
Ft
Local K: Y 1.000 Z 1.000
Local Cm: Y A B C Z A B C
Propagate Values to Other Legs
Apply
Cancel
Clear Query
Original View
CAP
1

CapHelp

Figure 8-9

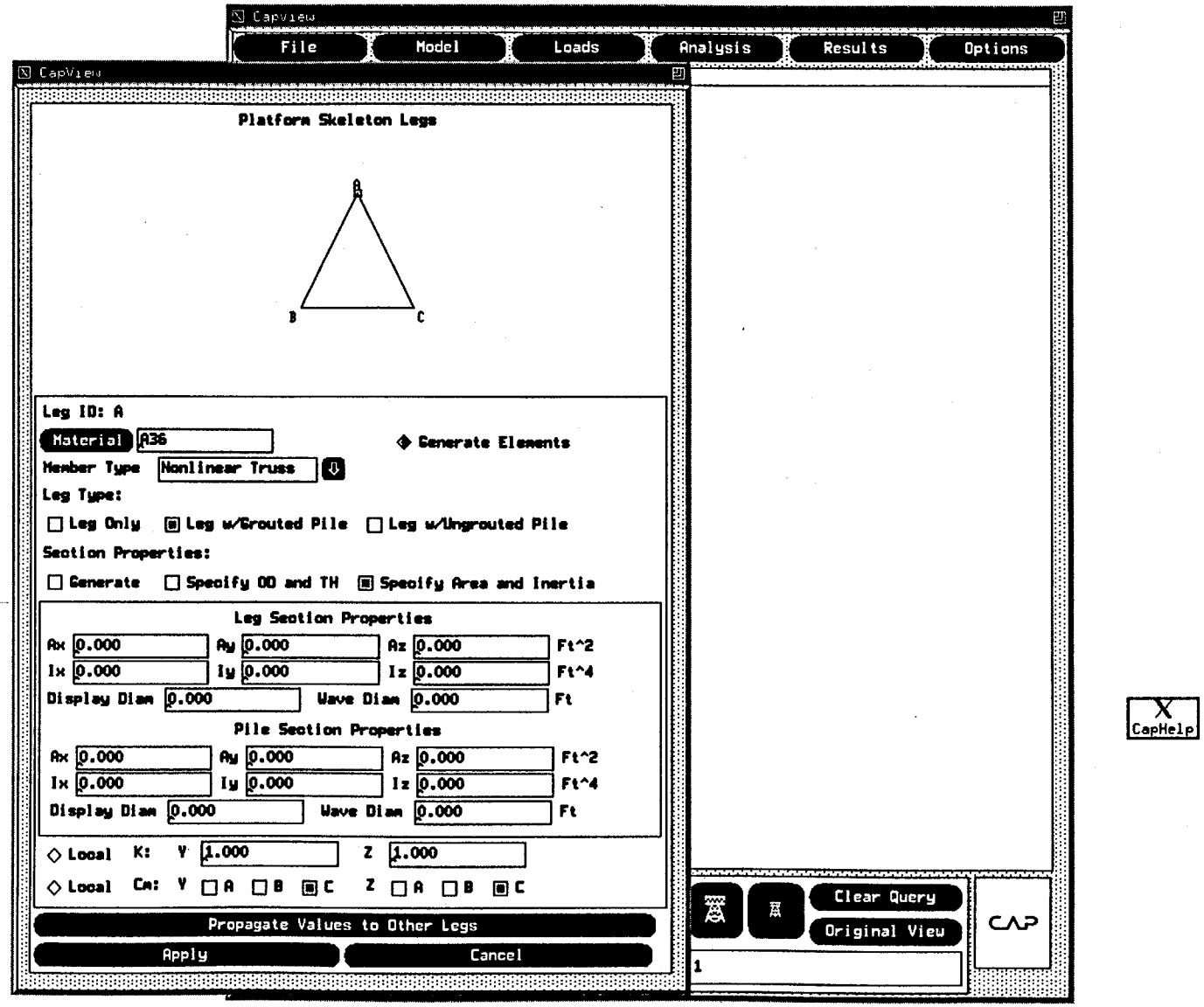

Capview
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Legs
A
B
C
Leg ID: A
Material
A36
Generate Elements
Member Type
Nonlinear Truss
Leg Type:
Leg Only
Leg w/Grouted Pile
Leg w/Ungrouted Pile
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Leg Section Properties
Ax 0.000
Ay 0.000
Az 0.000
Ft^2
Ix 0.000
Iy 0.000
Iz 0.000
Ft^4
Display Diam 0.000
Wave Diam 0.000
Ft
Pile Section Properties
Ax 0.000
Ay 0.000
Az 0.000
Ft^2
Ix 0.000
Iy 0.000
Iz 0.000
Ft^4
Display Diam 0.000
Wave Diam 0.000
Ft
Local K: Y 1.000 Z 1.000
Local Cm: Y A B C Z A B C
Propagate Values to Other Legs
Apply
Cancel
Clear Query
Original View
CAP
1
CapHelp

Figure 8-10

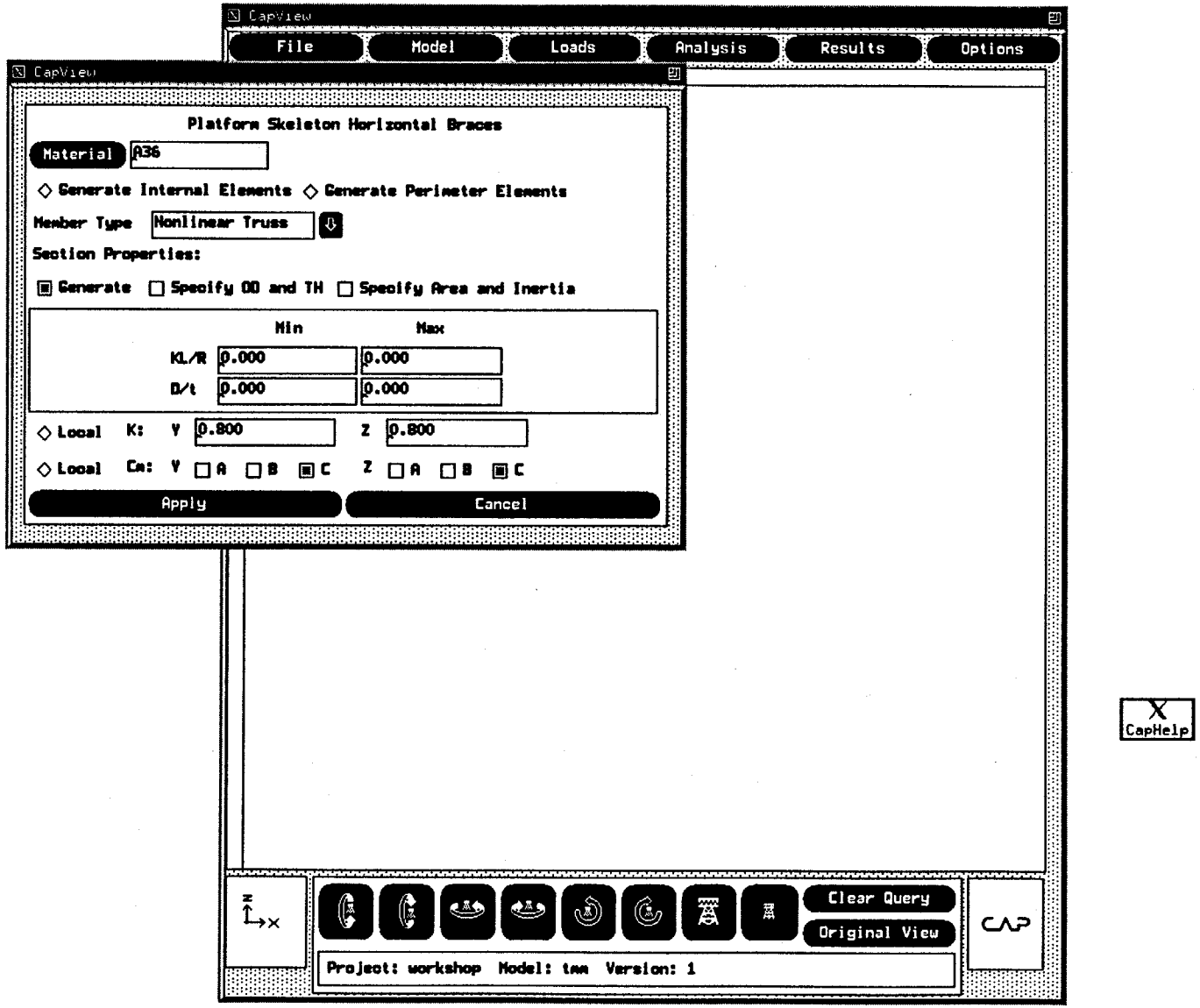

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Horizontal Braces
Material
A36
Generate Internal Elements
Generate Perimeter Elements
Member Type
Nonlinear Truss
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Min
Max
KL/R
0.000
0.000
D/t
0.000
0.000
Local
K:
Y
0.800
Z
0.800
Local
Cm:
Y
A
B
C
Z
A
B
C
Apply
Cancel
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-11

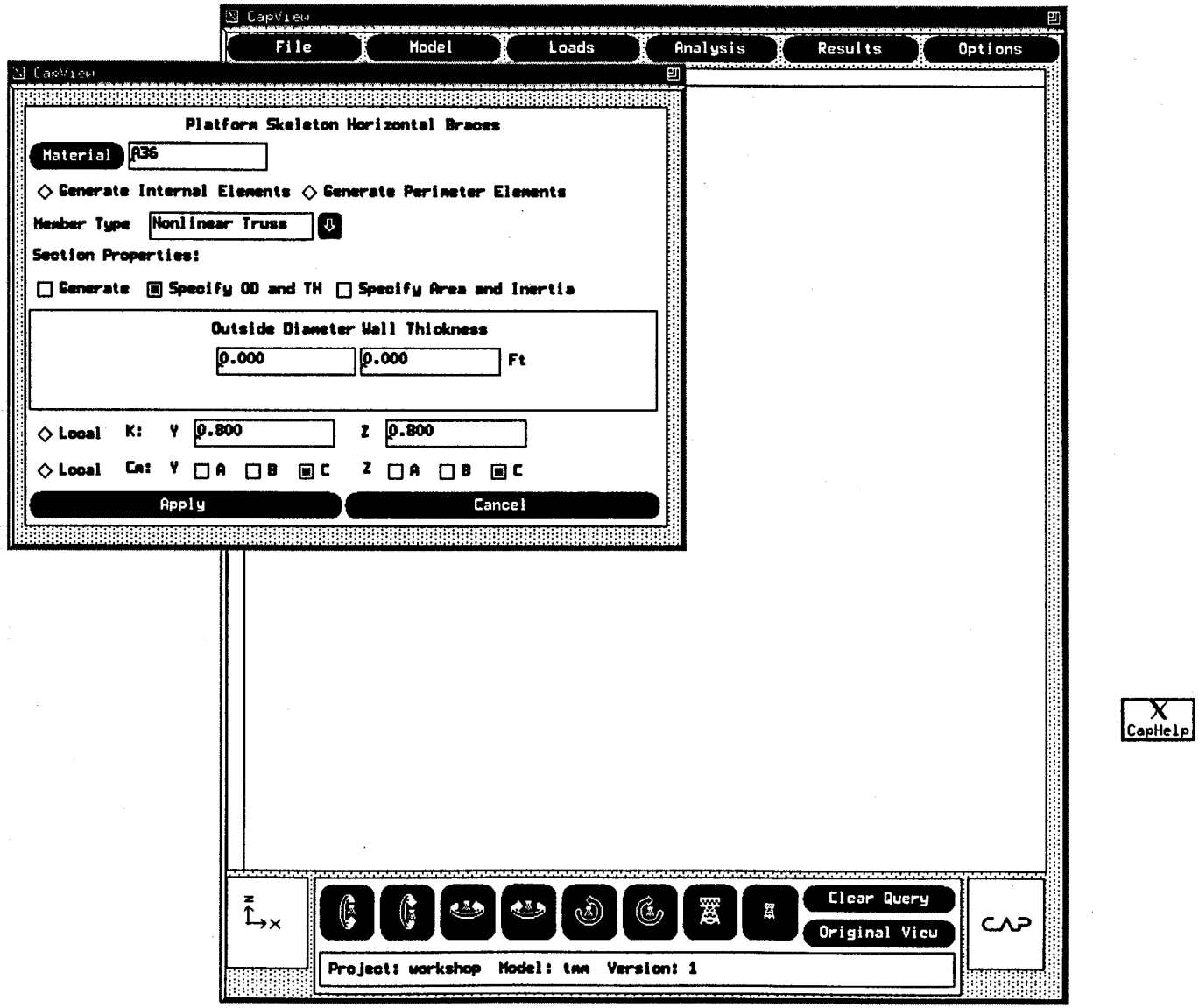

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Platform Skeleton Horizontal Braces
Material
A36
Generate Internal Elements
Generate Perimeter Elements
Member Type
Nonlinear Truss
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Outside Diameter Wall Thickness
0.000
0.000
Ft
Local K: Y 0.800 Z 0.800
Local Cm: Y A B C Z A B C
Apply
Cancel
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-12

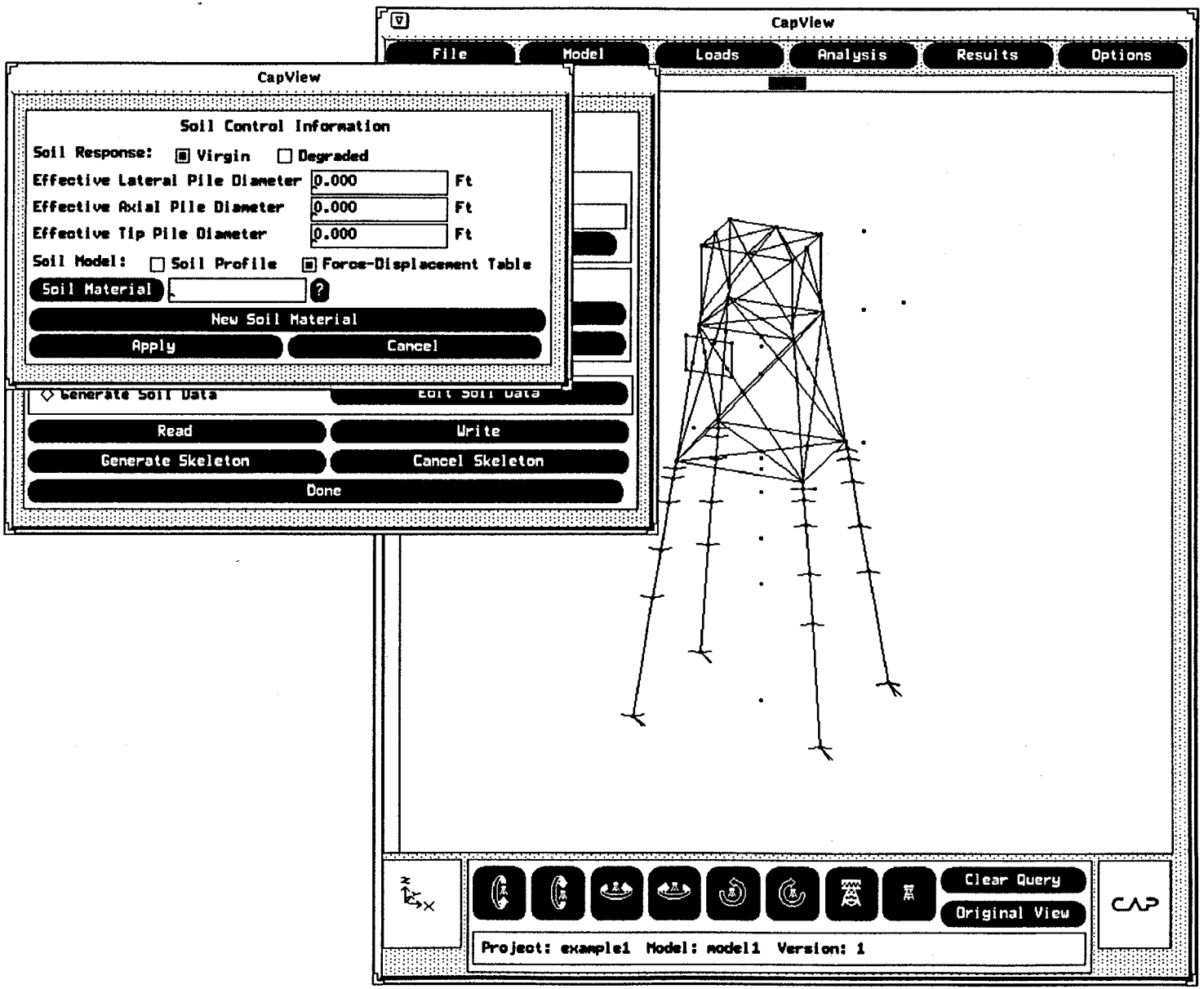

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Soil Control Information
Soil Response:
Virgin
Degraded
Effective Lateral Pile Diameter
0.000
Ft
Effective Axial Pile Diameter
0.000
Ft
Effective Tip Pile Diameter
0.000
Ft
Soil Model:
Soil Profile
Force-Displacement Table
Soil Material
New Soil Material
Apply
Cancel
Generate Soil Data
Read
Write
Generate Skeleton
Cancel Skeleton
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-15

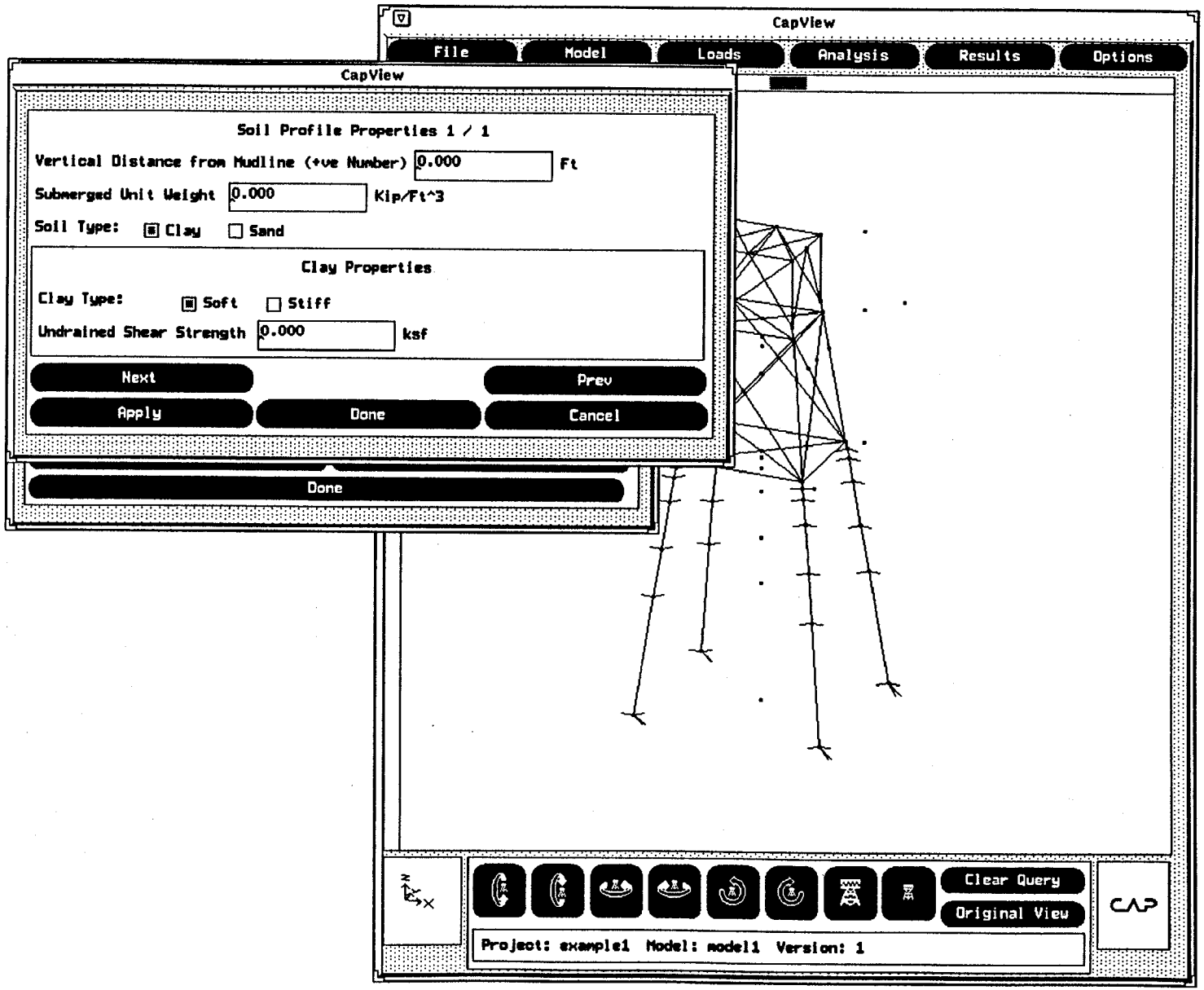

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Soil Profile Properties 1 / 1
Vertical Distance from Mudline (+ve Number) 0.000 Ft
Submerged Unit Weight 0.000 Kip/Ft^3
Soil Type: Clay Sand
Clay Properties
Clay Type: Soft Stiff
Undrained Shear Strength 0.000 ksf
Next
Prev
Apply
Done
Cancel
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-16

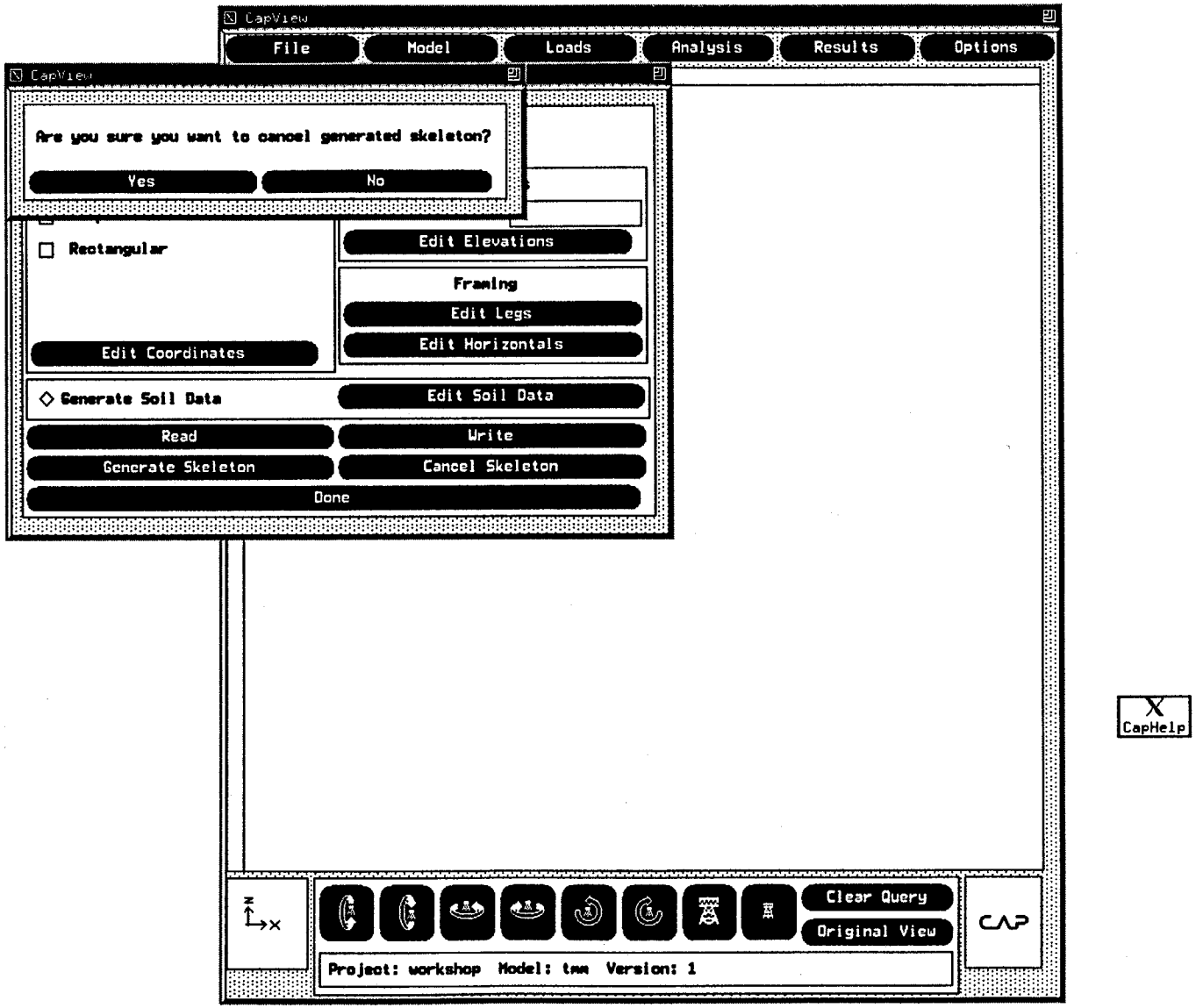

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Are you sure you want to cancel generated skeleton?
Yes
No
Rectangular
Edit Elevations
Framing
Edit Legs
Edit Horizontals
Edit Coordinates
Generate Soil Data
Edit Soil Data
Read
Write
Generate Skeleton
Cancel Skeleton
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-19

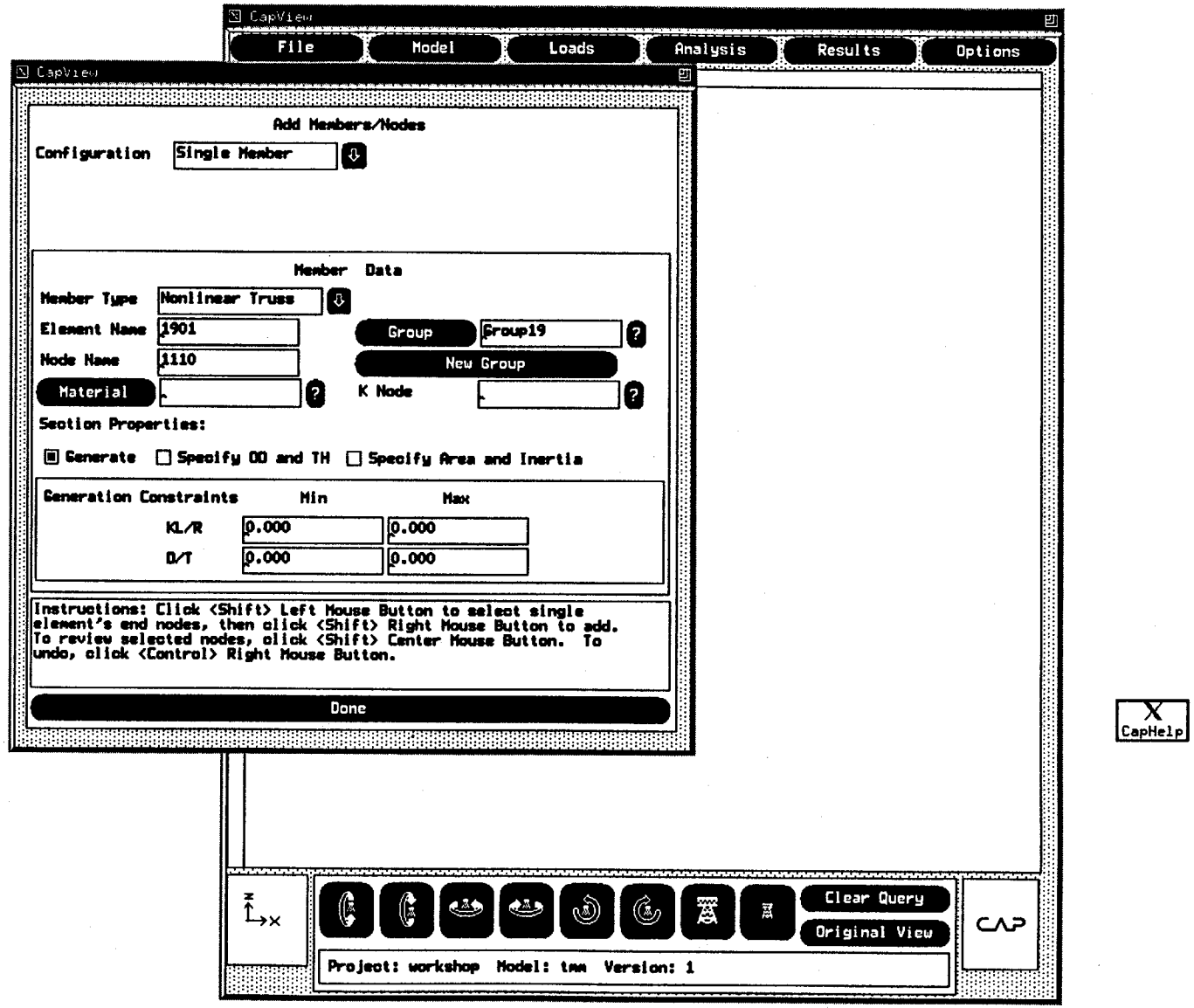

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Single Member
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select single element's end nodes, then click <Shift> Right Mouse Button to add. To review selected nodes, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-20

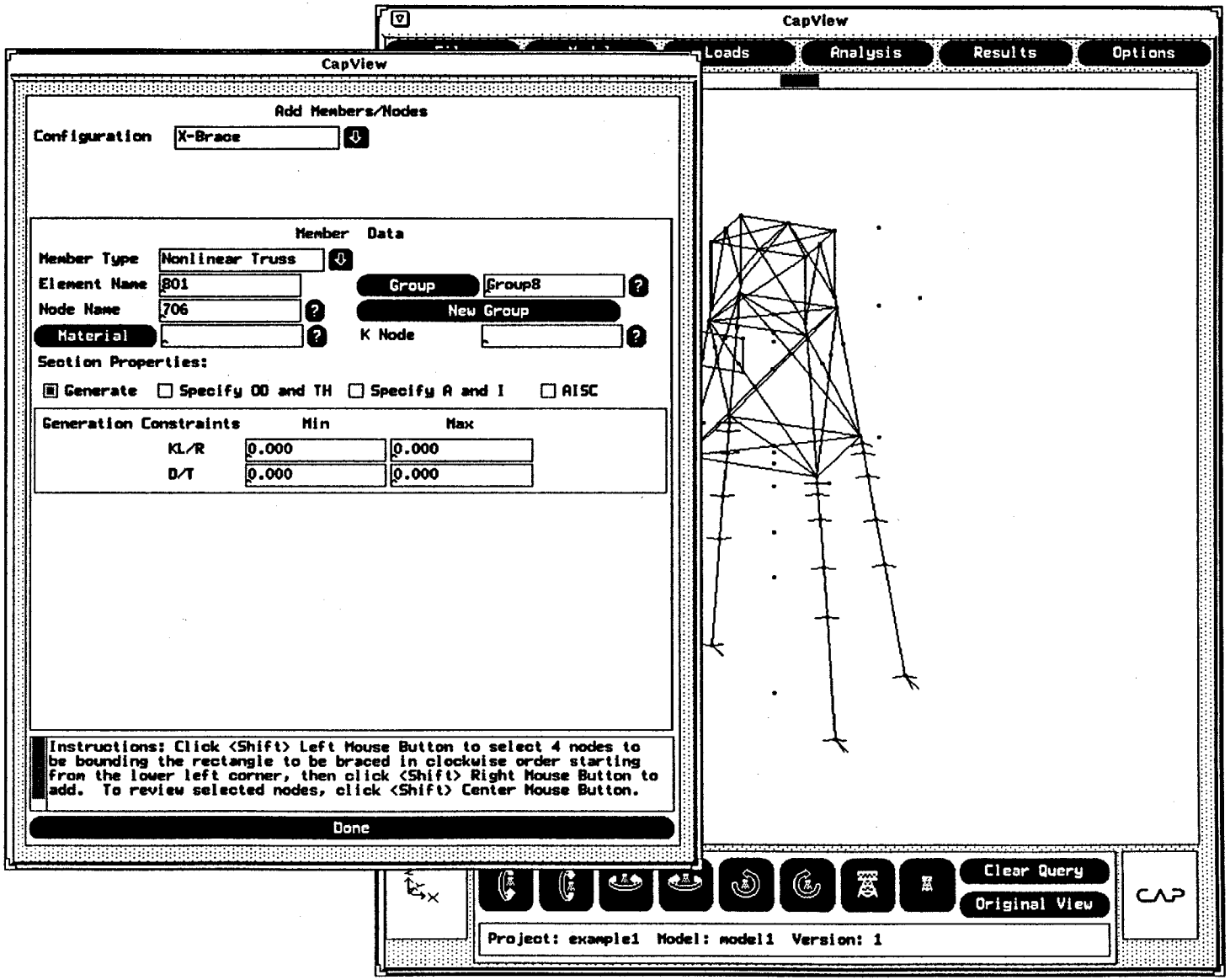

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
X-Brace
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select 4 nodes to be bounding the rectangle to be braced in clockwise order starting from the lower left corner, then click <Shift> Right Mouse Button to add. To review selected nodes, click <Shift> Center Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-21

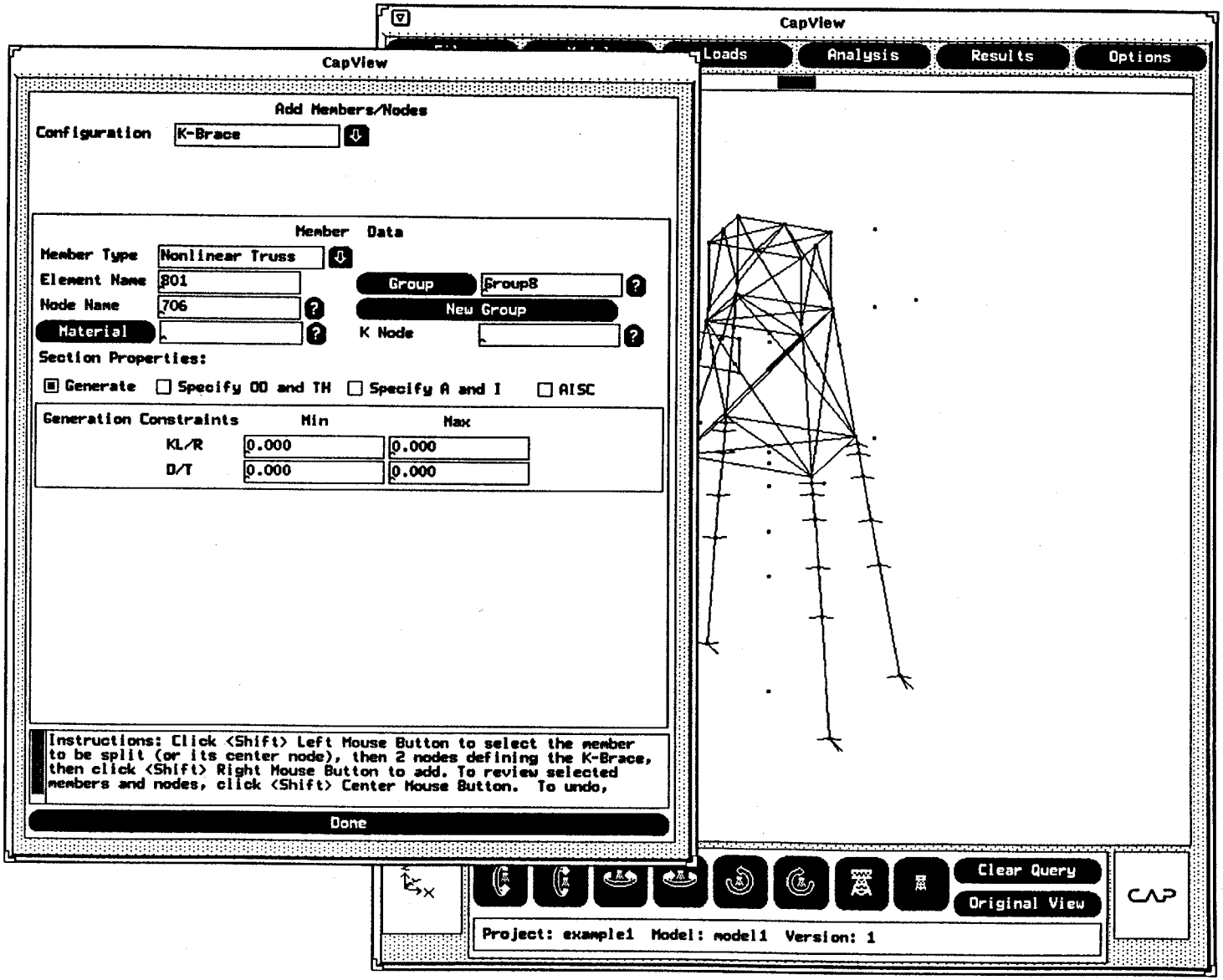

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
K-Brace
Member Data
Member Type
Nonlinear Truss
Element Name
801
Node Name
706
Material
Group
Group8
New Group
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select the member to be split (or its center node), then 2 nodes defining the K-Brace, then click <Shift> Right Mouse Button to add. To review selected members and nodes, click <Shift> Center Mouse Button. To undo,
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-22

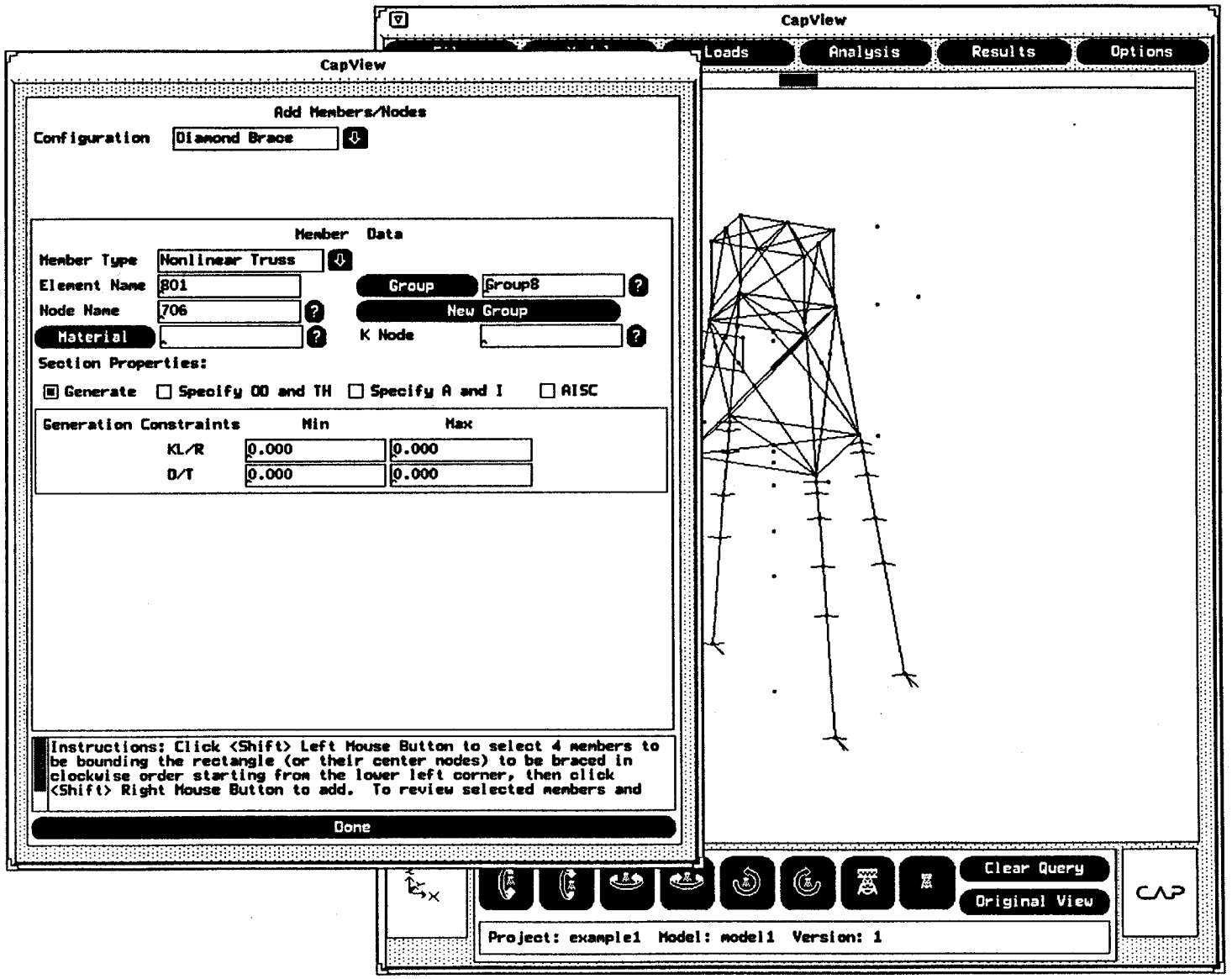

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Diamond Brace
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select 4 members to be bounding the rectangle (or their center nodes) to be braced in clockwise order starting from the lower left corner, then click <Shift> Right Mouse Button to add. To review selected members and
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-23

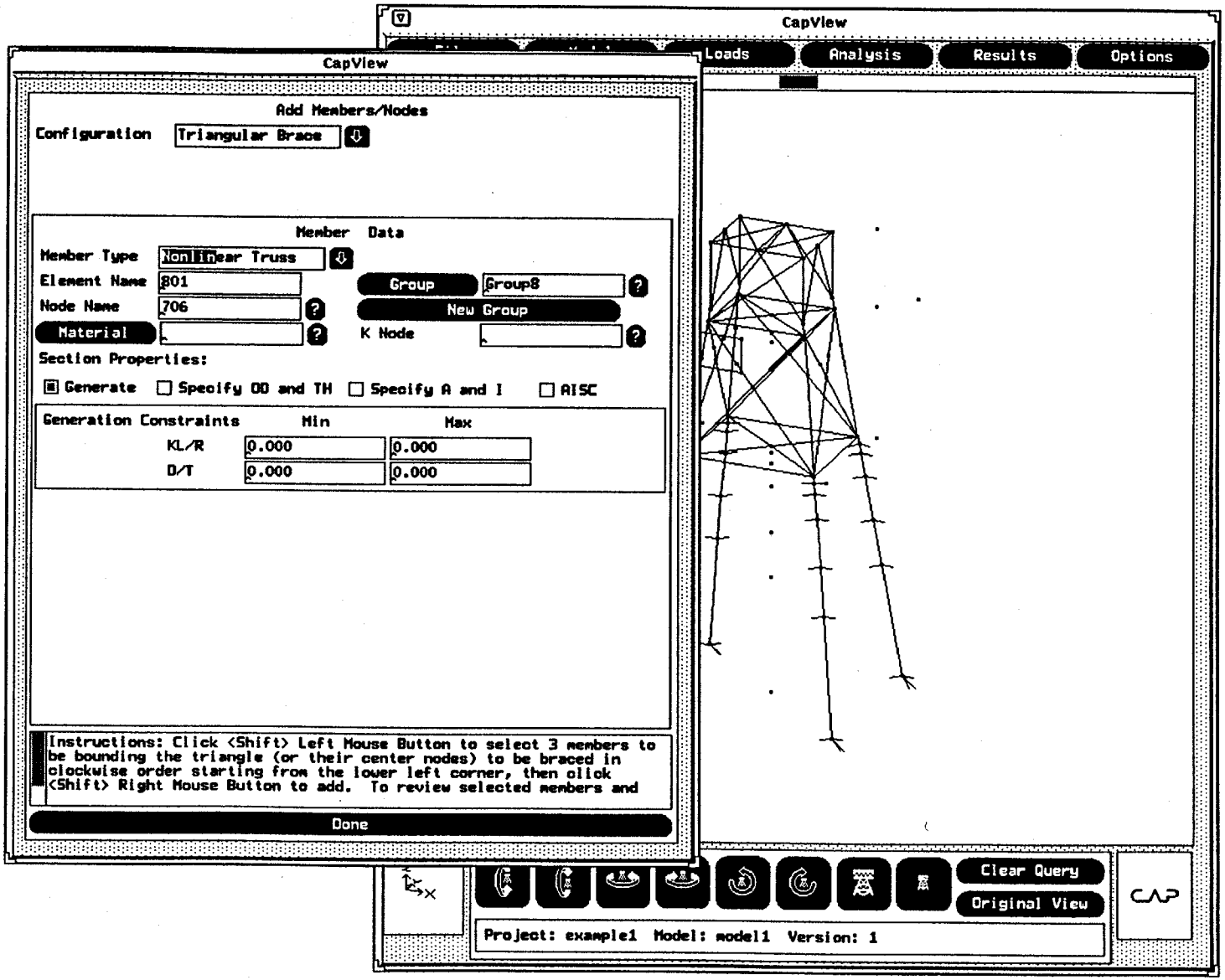

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Triangular Brace
Member Data
Member Type
Nonlinear Truss
Element Name
801
Node Name
706
Material
Group
Group8
New Group
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select 3 members to be bounding the triangle (or their center nodes) to be braced in clockwise order starting from the lower left corner, then click <Shift> Right Mouse Button to add. To review selected members and
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-24

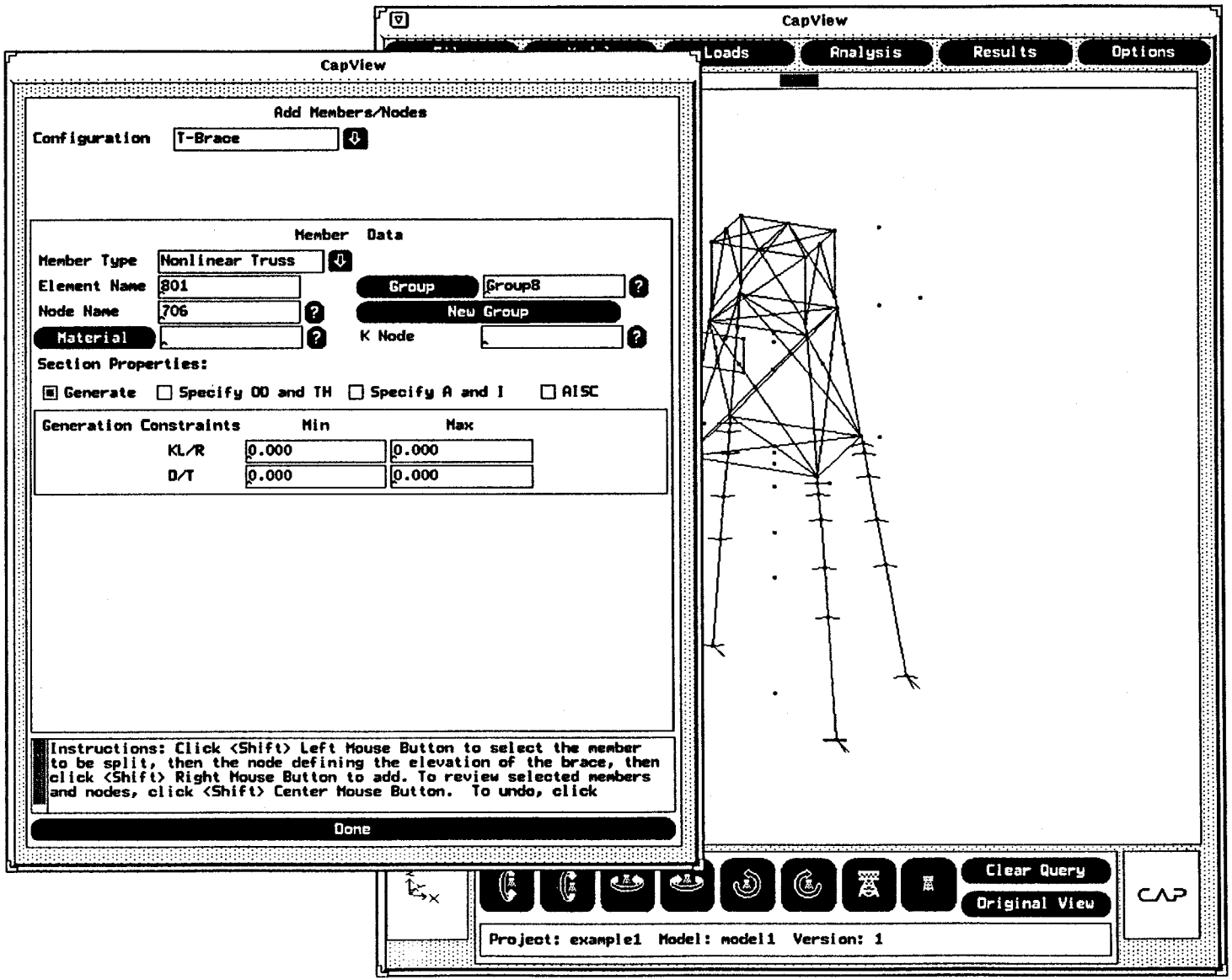

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
T-Brace
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select the member to be split, then the node defining the elevation of the brace, then click <Shift> Right Mouse Button to add. To review selected members and nodes, click <Shift> Center Mouse Button. To undo, click
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-25

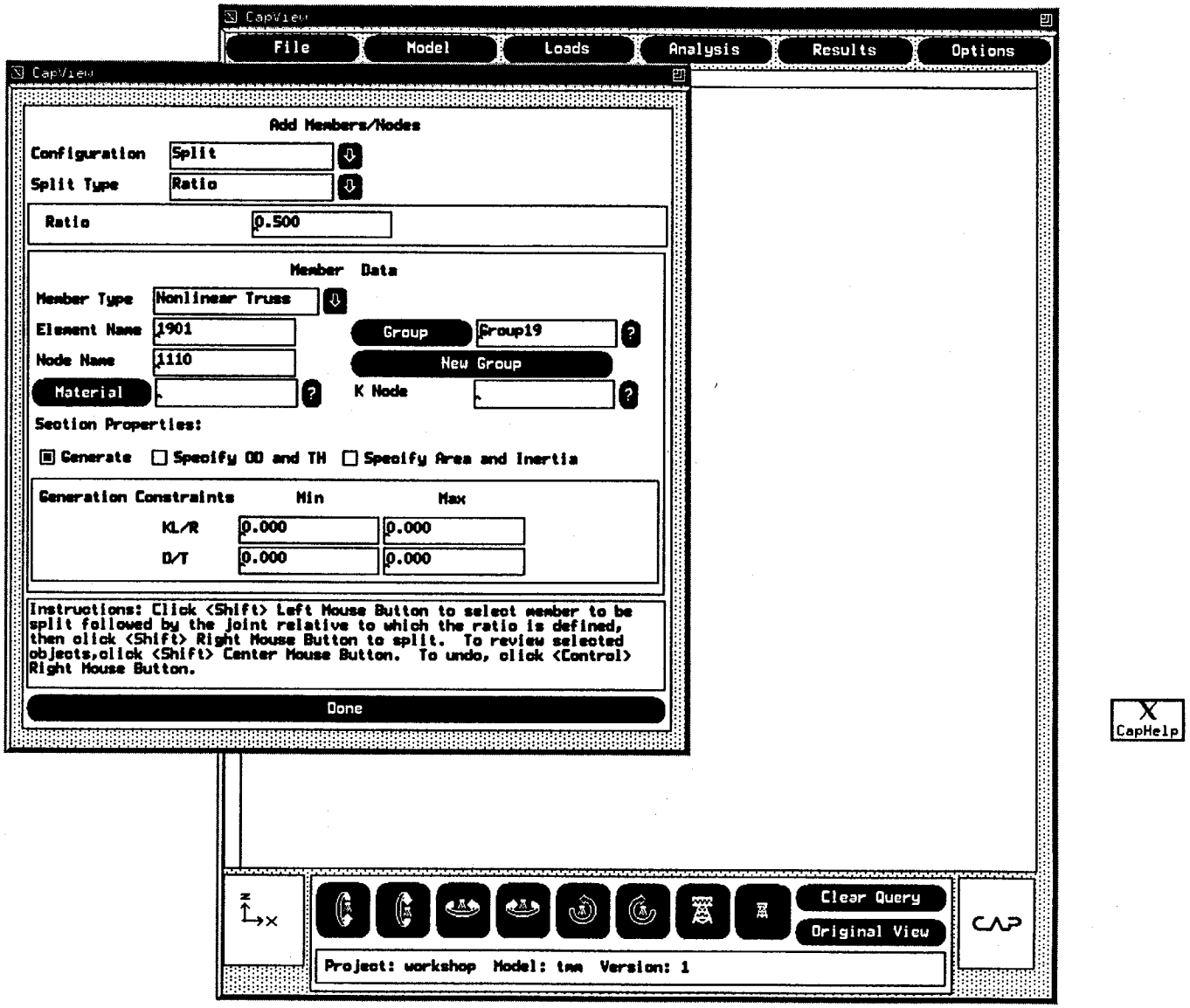

CapView
File
Model
Loads
Analysis
Results
Options
Add Members/Nodes
Configuration
Split
Split Type
Ratio
Ratio
0.500
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be split followed by the joint relative to which the ratio is defined, then click <Shift> Right Mouse Button to split. To review selected objects,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-26

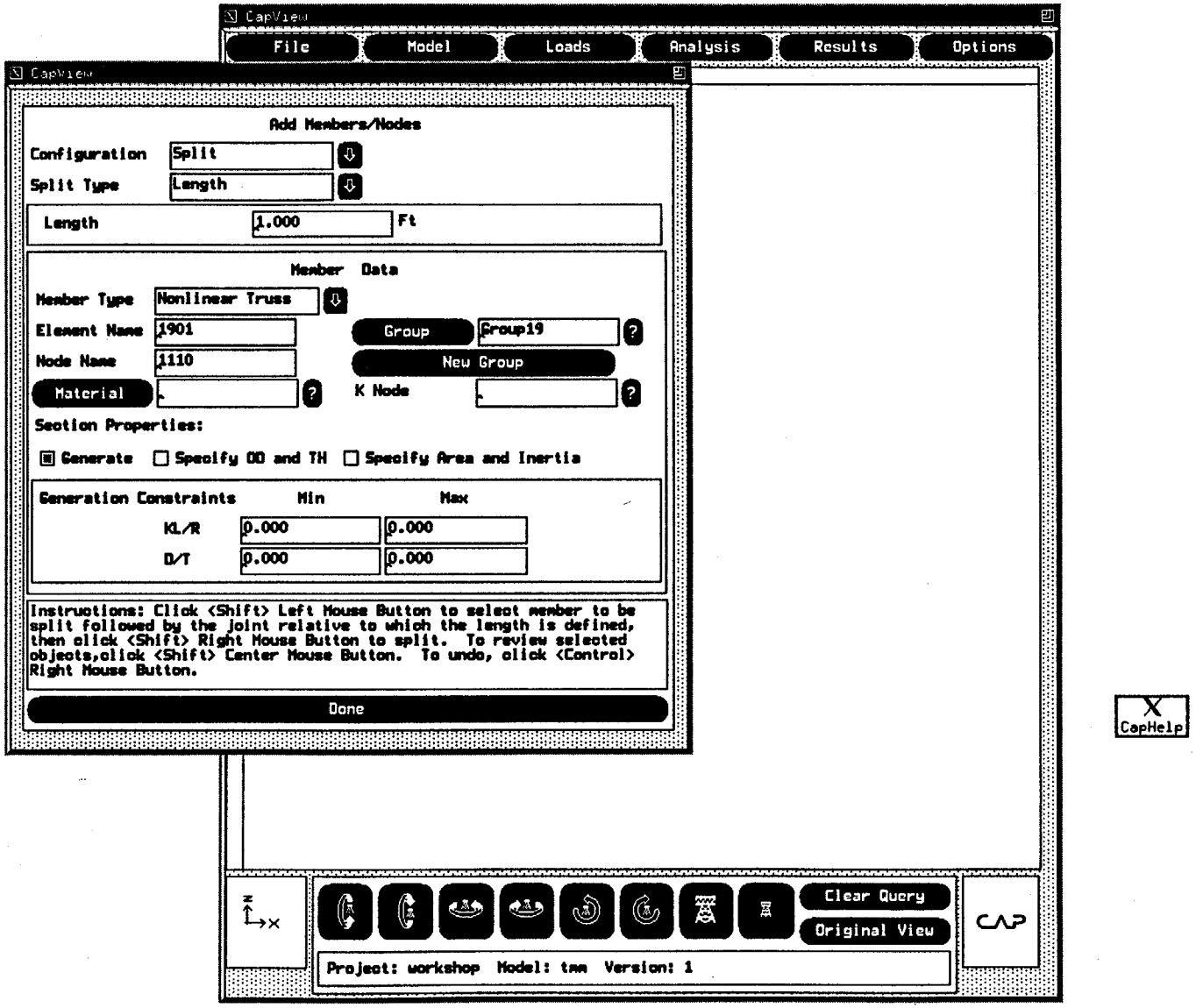

CapView
File
Model
Loads
Analysis
Results
Options
Add Members/Nodes
Configuration
Split
Split Type
Length
Length
1.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be split followed by the joint relative to which the length is defined, then click <Shift> Right Mouse Button to split. To review selected objects,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-27

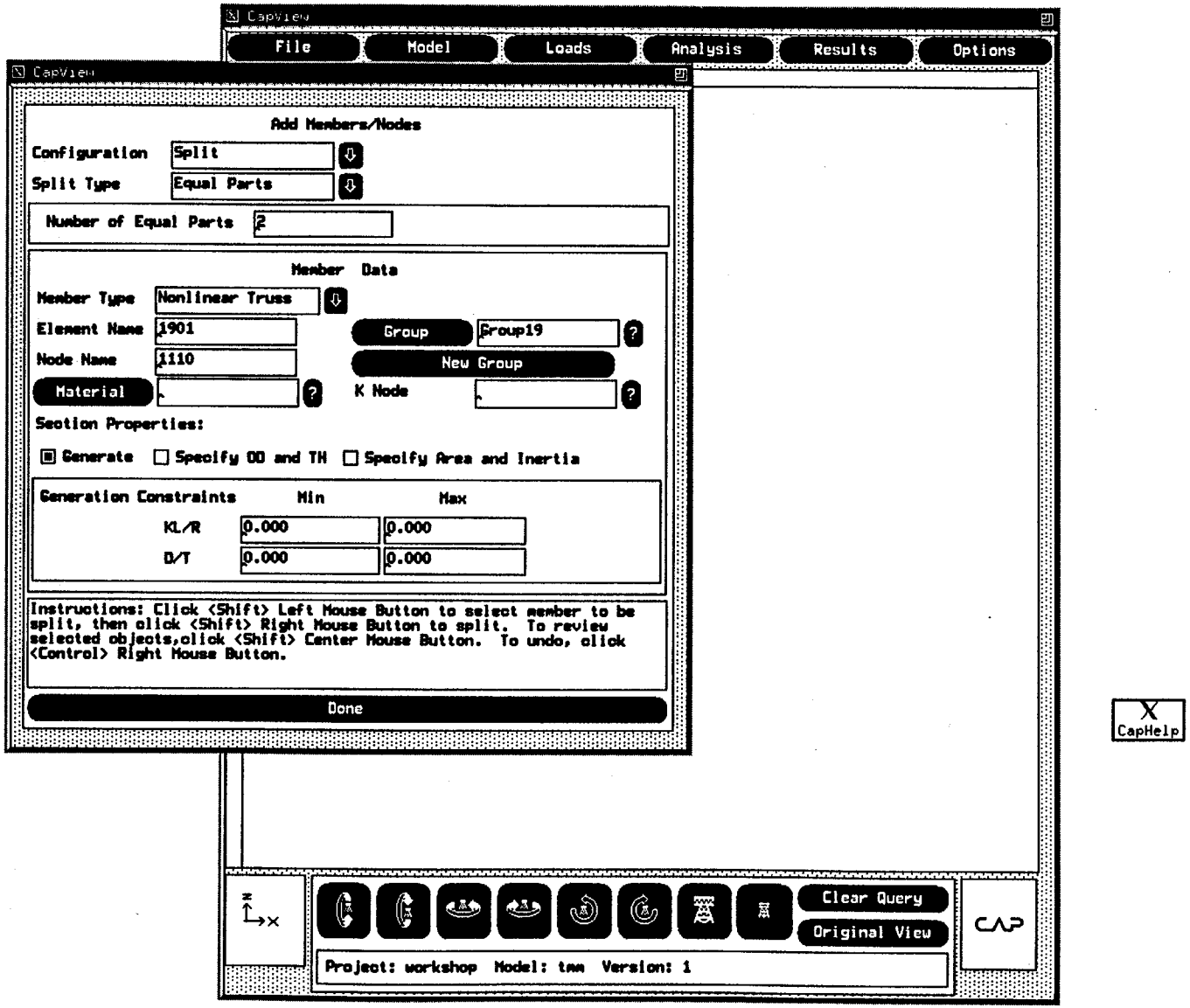

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Split
Split Type
Equal Parts
Number of Equal Parts
2
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-28

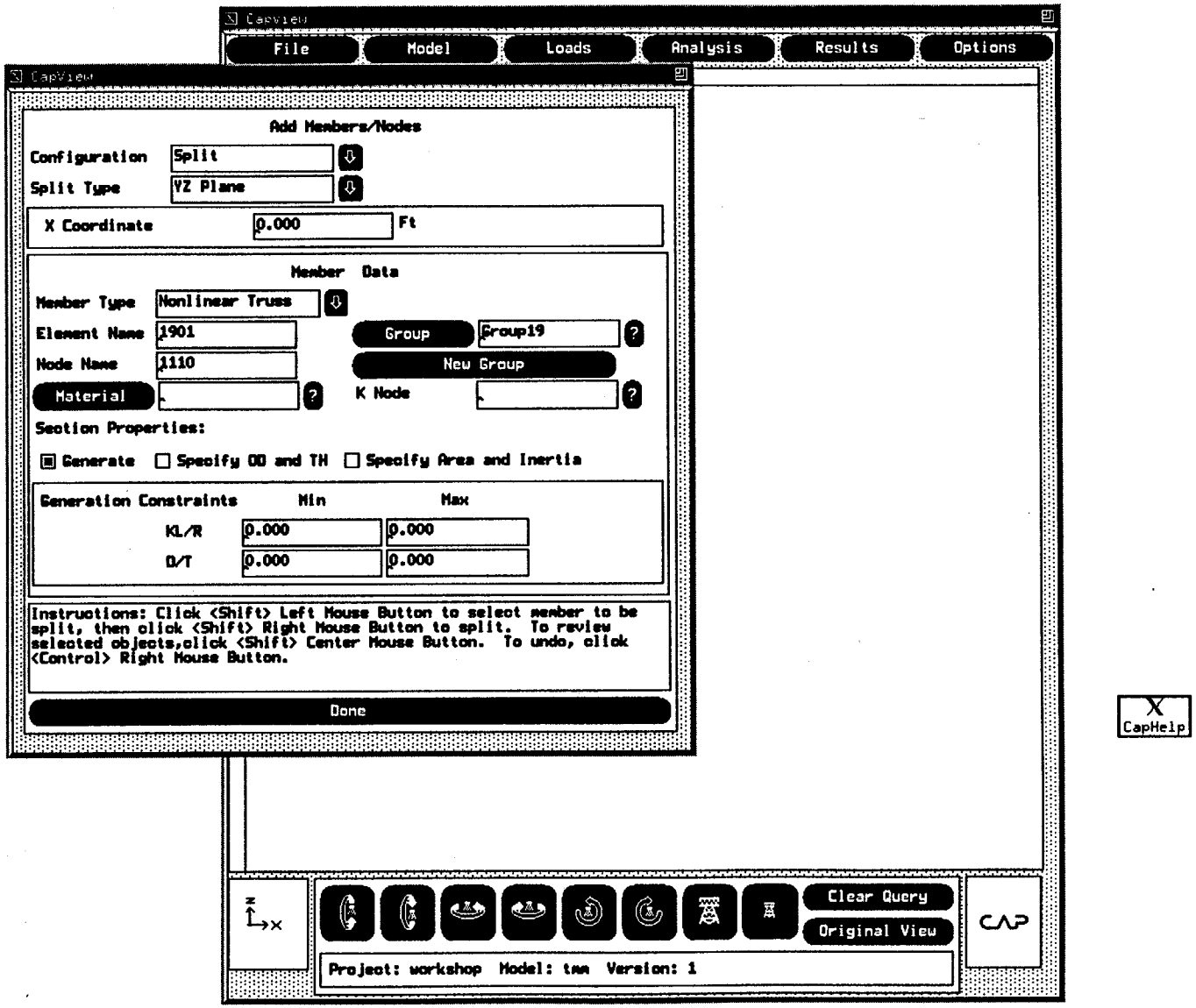

CapView
File
Model
Loads
Analysis
Results
Options
Add Members/Nodes
Configuration
Split
Split Type
YZ Plane
X Coordinate
0.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-29

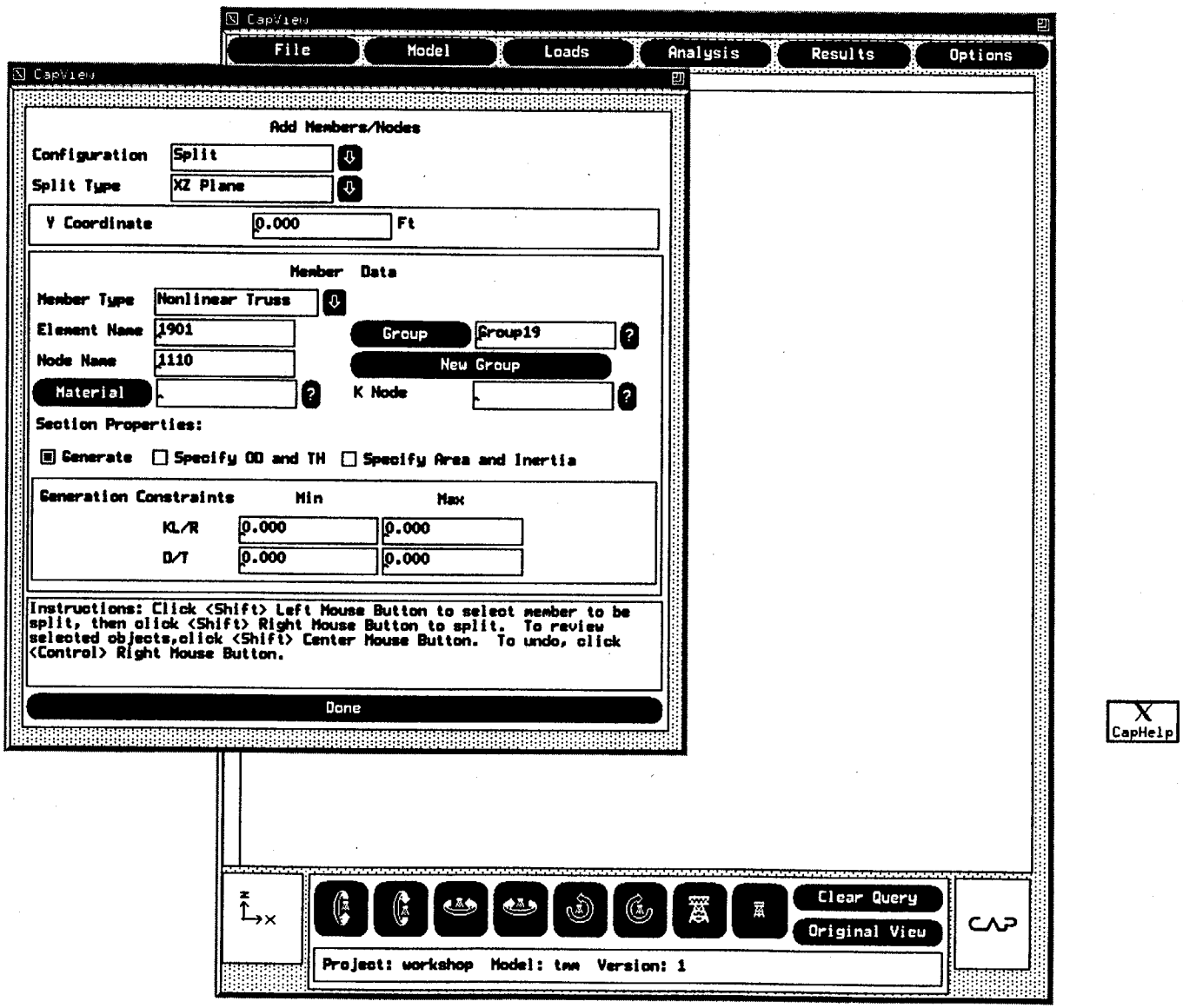

CapView
File
Model
Loads
Analysis
Results
Options
Add Members/Nodes
Configuration
Split
Split Type
XZ Plane
Y Coordinate
0.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
1901
Group
Group19
Node Name
1110
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify Area and Inertia
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-30

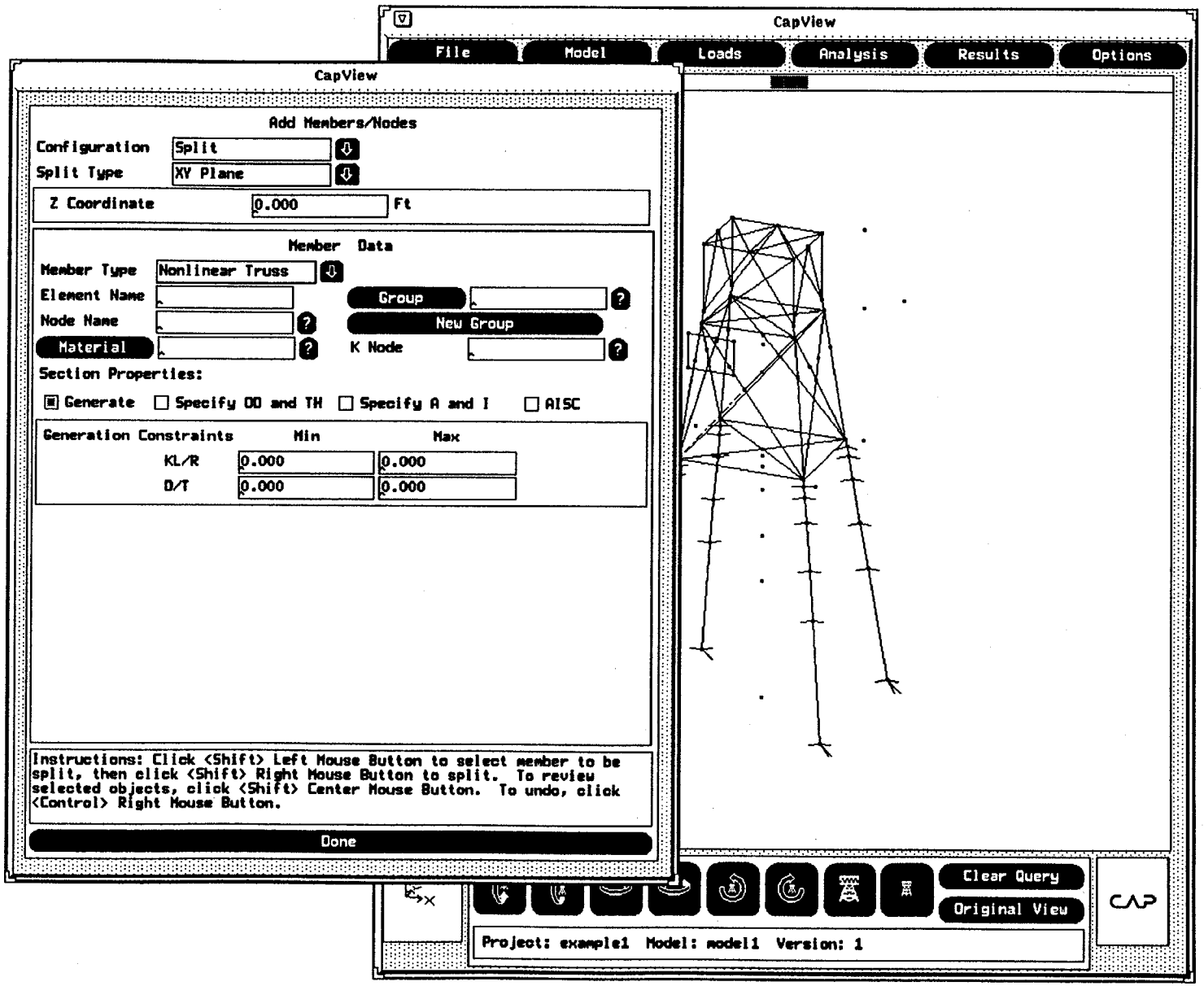

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Split
Split Type
XY Plane
Z Coordinate
0.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
Group
Node Name
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-31

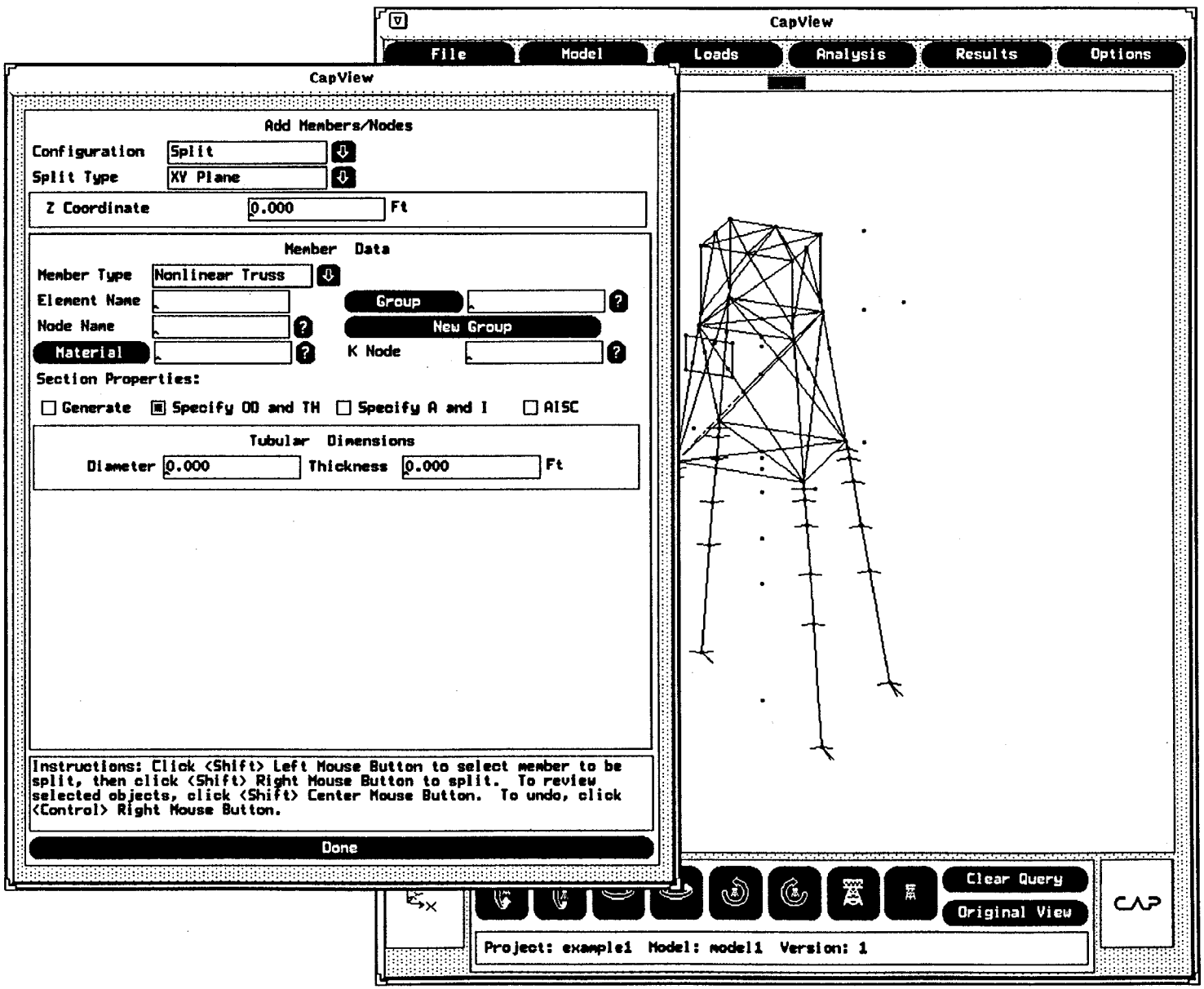

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Split
Split Type
XY Plane
Z Coordinate
0.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
Group
Node Name
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Tubular Dimensions
Diameter
0.000
Thickness
0.000
Ft
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-32

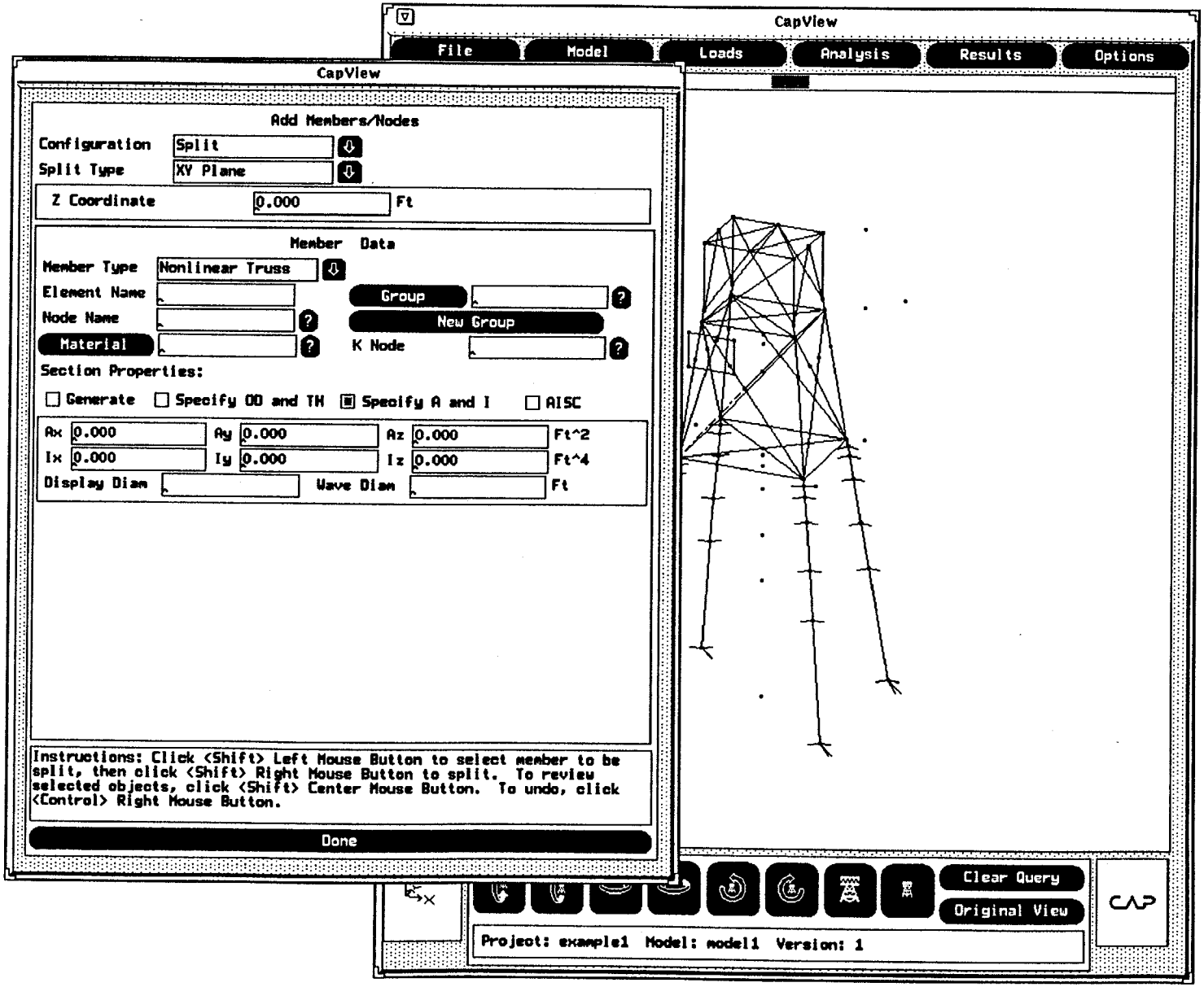

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Split
Split Type
XY Plane
Z Coordinate
0.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
Group
Node Name
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Ax
0.000
Ay
0.000
Az
0.000
Ft^2
Ix
0.000
Iy
0.000
Iz
0.000
Ft^4
Display Diam
Wave Diam
Ft
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-33

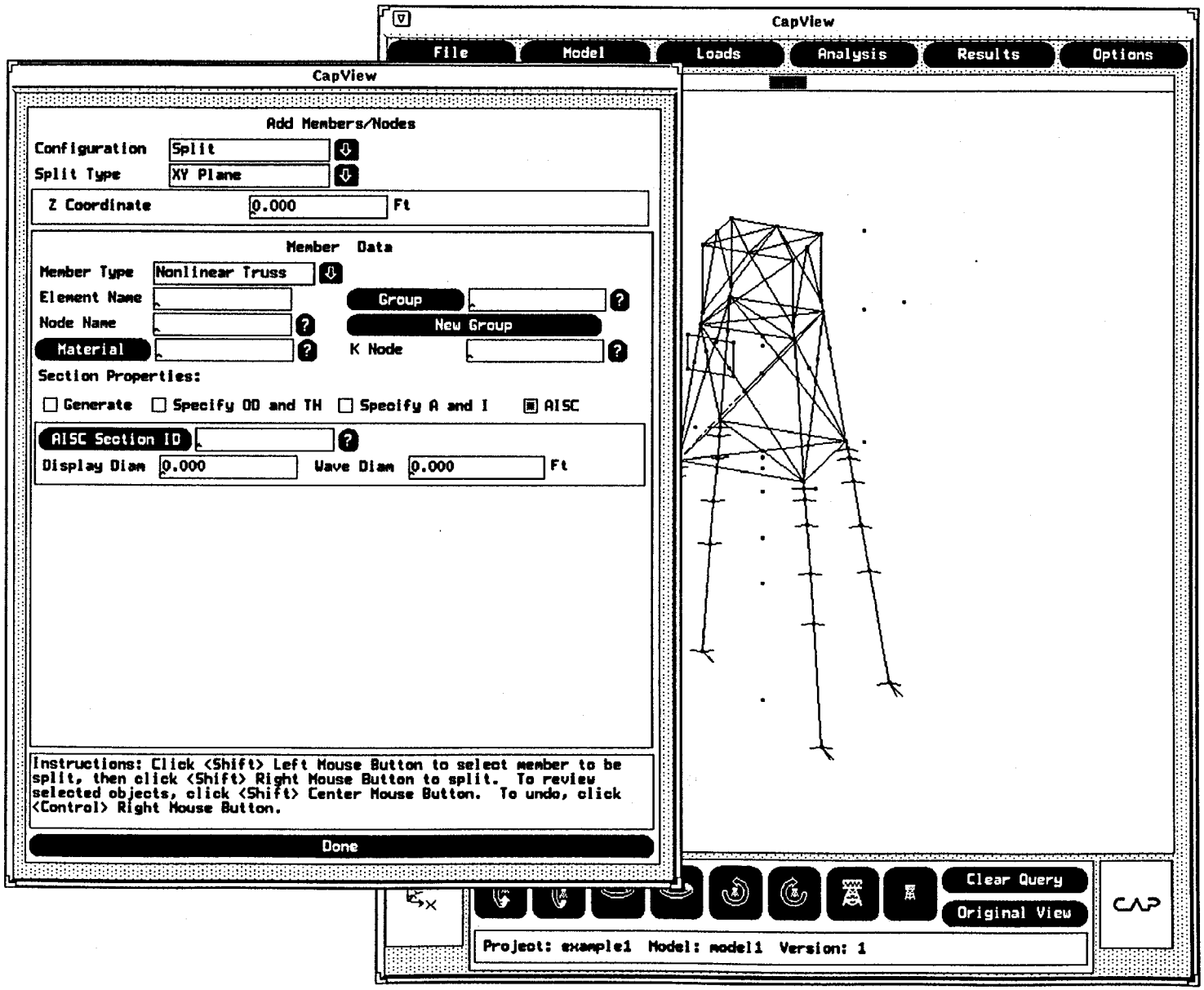

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Split
Split Type
XY Plane
Z Coordinate
0.000
Ft
Member Data
Member Type
Nonlinear Truss
Element Name
Group
Node Name
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
AISC Section ID
Display Diam
0.000
Wave Diam
0.000
Ft
Instructions: Click <Shift> Left Mouse Button to select member to be split, then click <Shift> Right Mouse Button to split. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-34

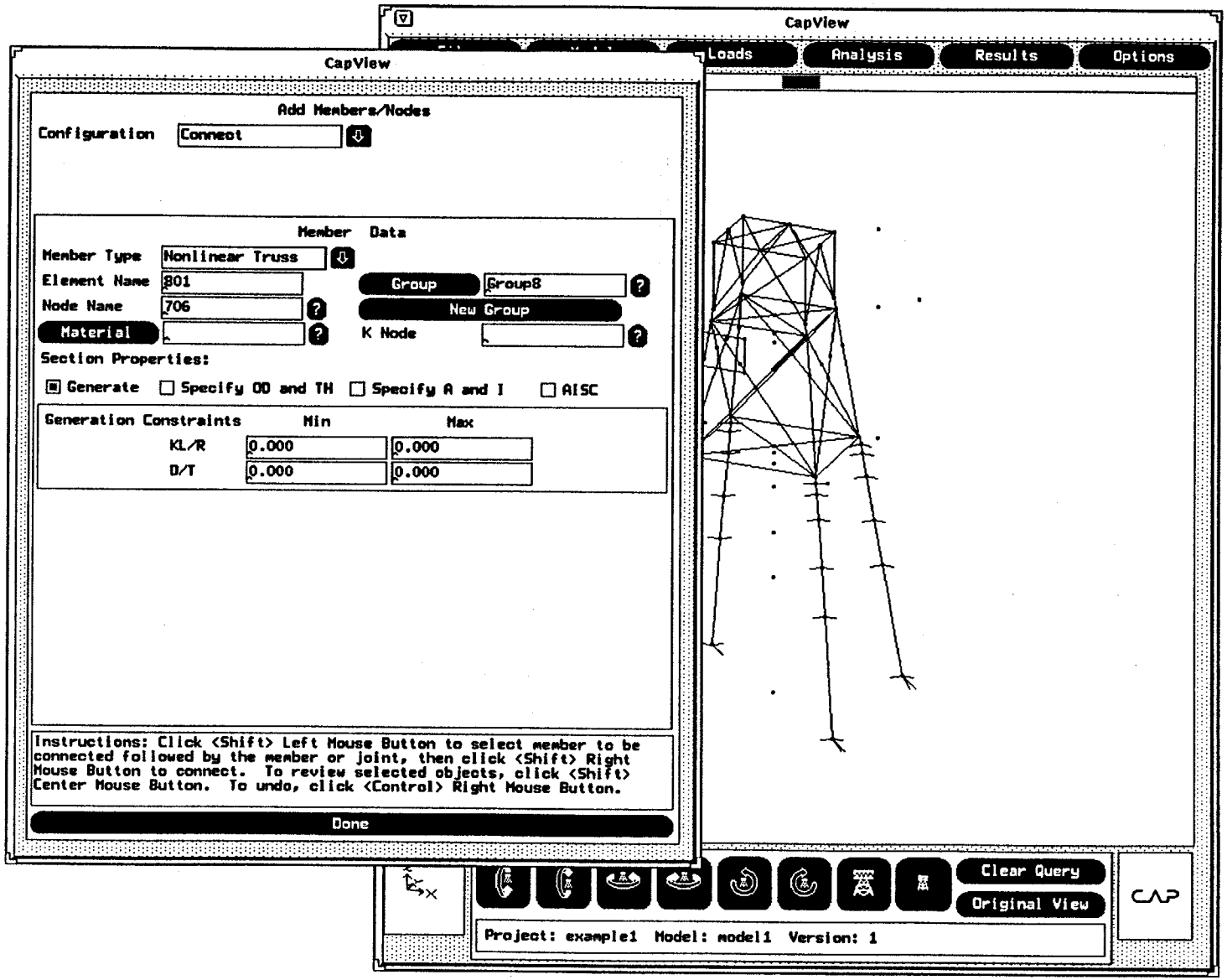

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Connect
Member Data
Member Type
Nonlinear Truss
Element Name
801
Node Name
706
Material
Group
Group8
New Group
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select member to be connected followed by the member or joint, then click <Shift> Right Mouse Button to connect. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-35

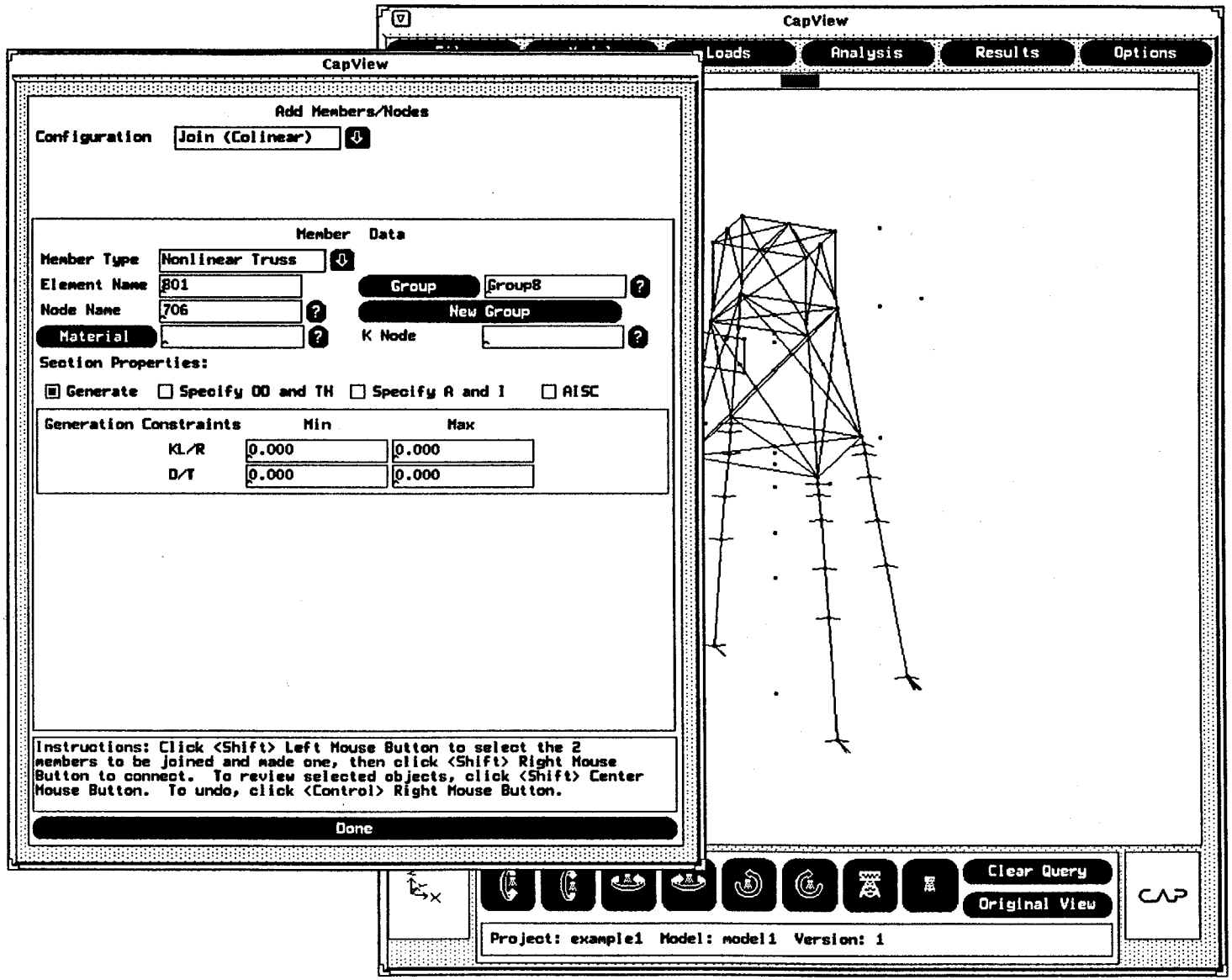

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Join (Colinear)
Member Data
Member Type
Nonlinear Truss
Element Name
801
Node Name
706
Material
Group
Group8
New Group
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Click <Shift> Left Mouse Button to select the 2 members to be joined and made one, then click <Shift> Right Mouse Button to connect. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-36

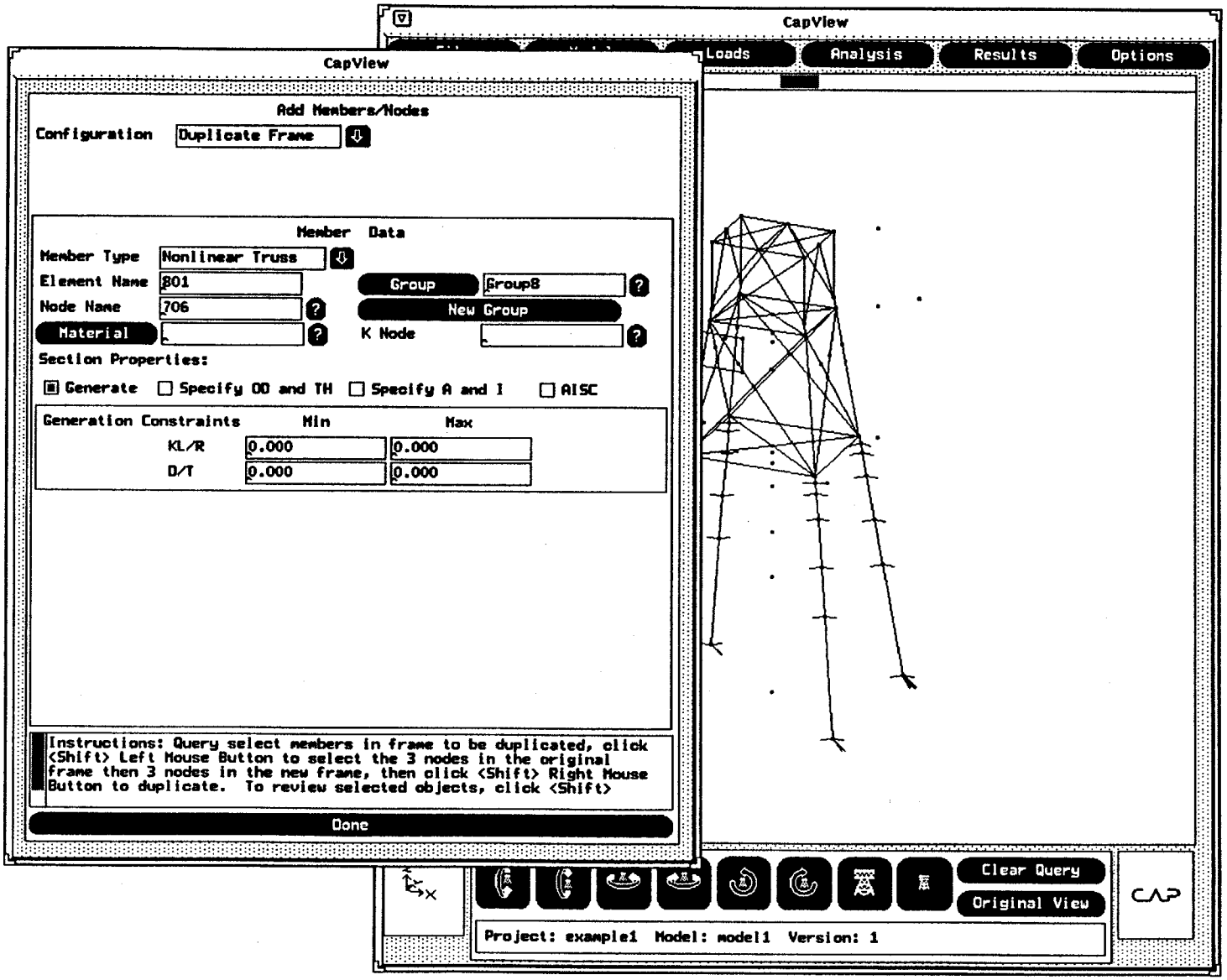

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Duplicate Frame
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Instructions: Query select members in frame to be duplicated, click <Shift> Left Mouse Button to select the 3 nodes in the original frame then 3 nodes in the new frame, then click <Shift> Right Mouse Button to duplicate. To review selected objects, click <Shift>
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-37

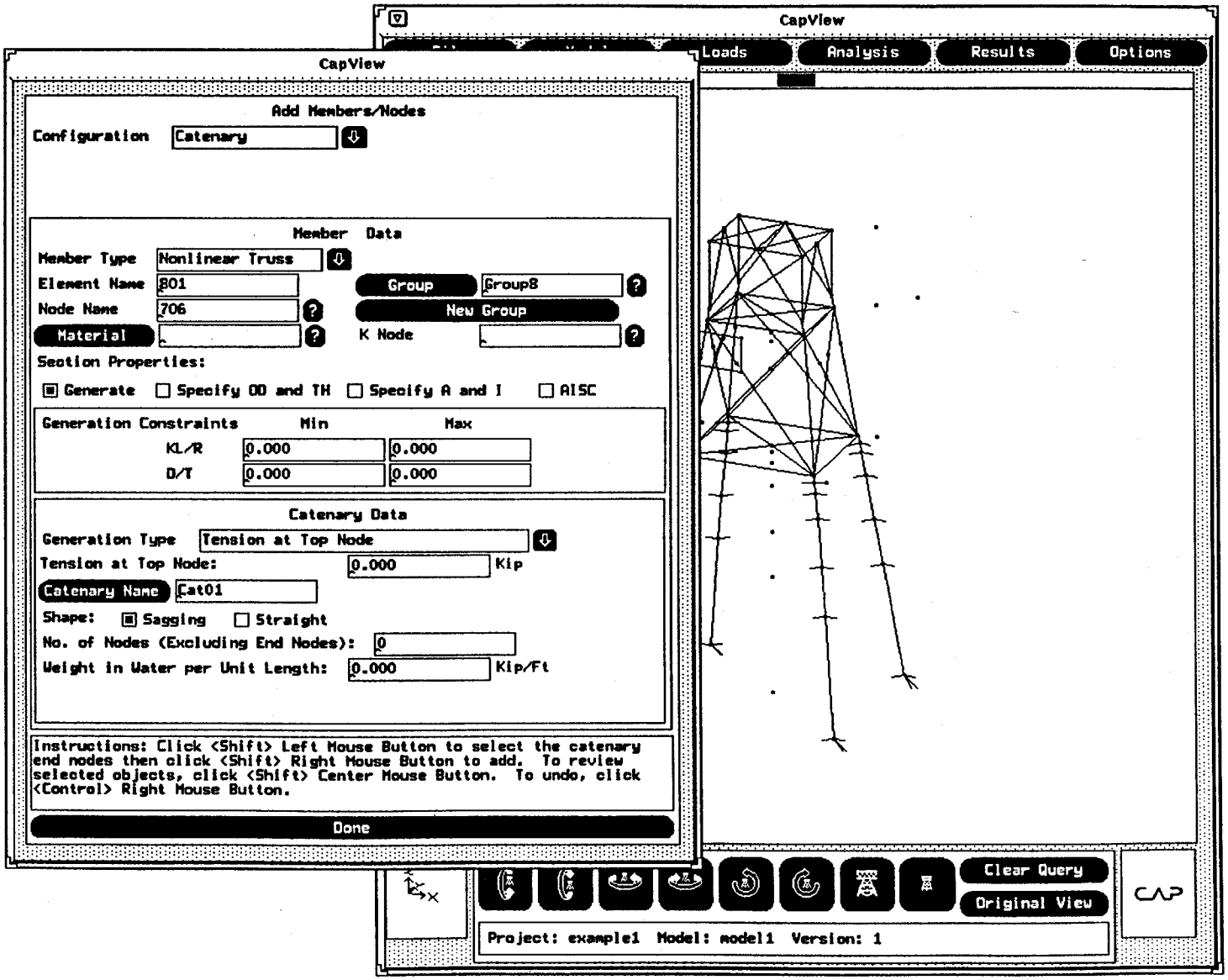

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Catenary
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Catenary Data
Generation Type
Tension at Top Node
Tension at Top Node:
0.000
Kip
Catenary Name
Cat01
Shape:
Sagging
Straight
No. of Nodes (Excluding End Nodes):
0
Weight in Water per Unit Length:
0.000
Kip/Ft
Instructions: Click <Shift> Left Mouse Button to select the catenary end nodes then click <Shift> Right Mouse Button to add. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-38

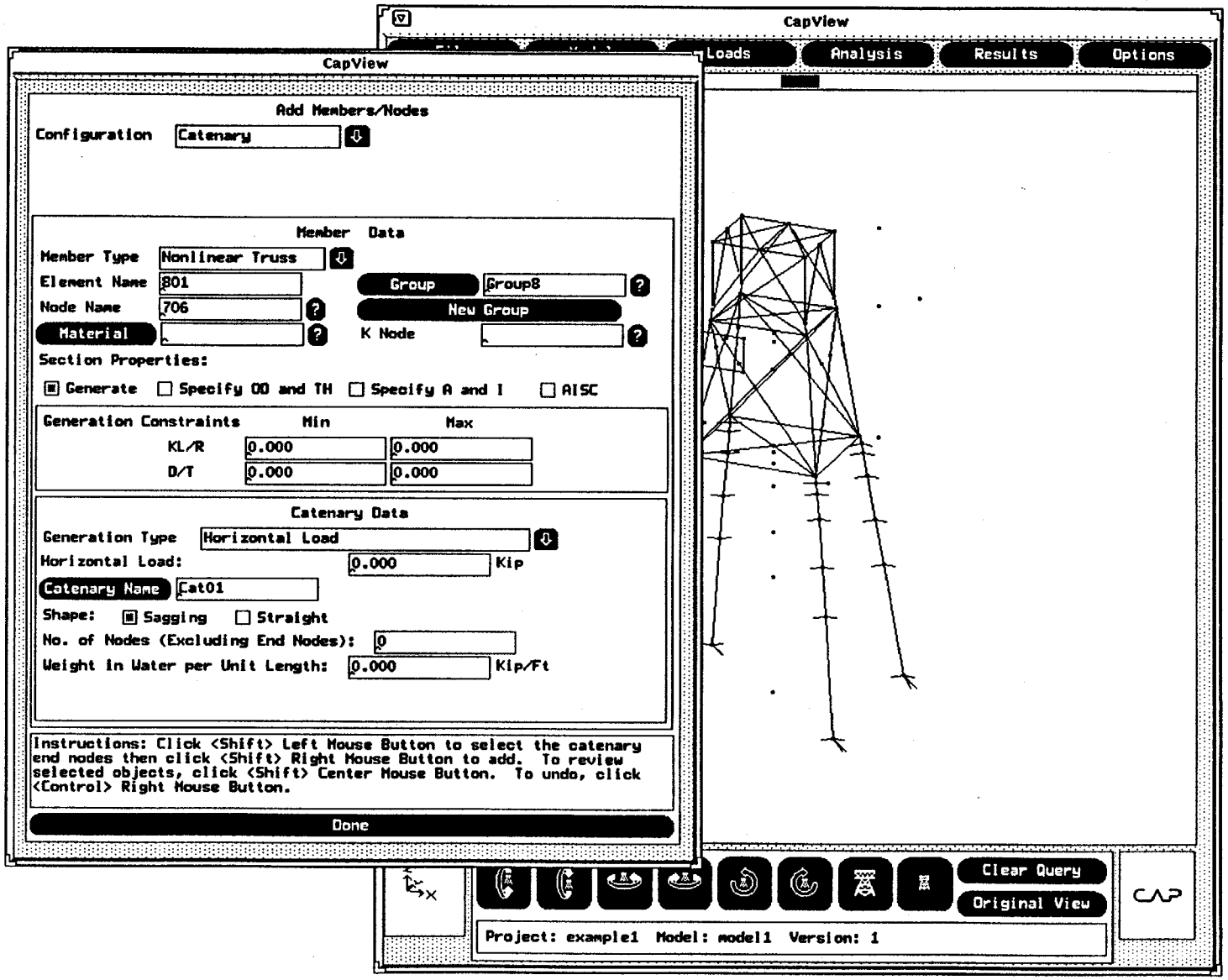

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Catenary
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Catenary Data
Generation Type
Horizontal Load
Horizontal Load:
0.000
Kip
Catenary Name
Cat01
Shape:
Sagging
Straight
No. of Nodes (Excluding End Nodes):
0
Weight in Water per Unit Length:
0.000
Kip/Ft
Instructions: Click <Shift> Left Mouse Button to select the catenary end nodes then click <Shift> Right Mouse Button to add. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-39

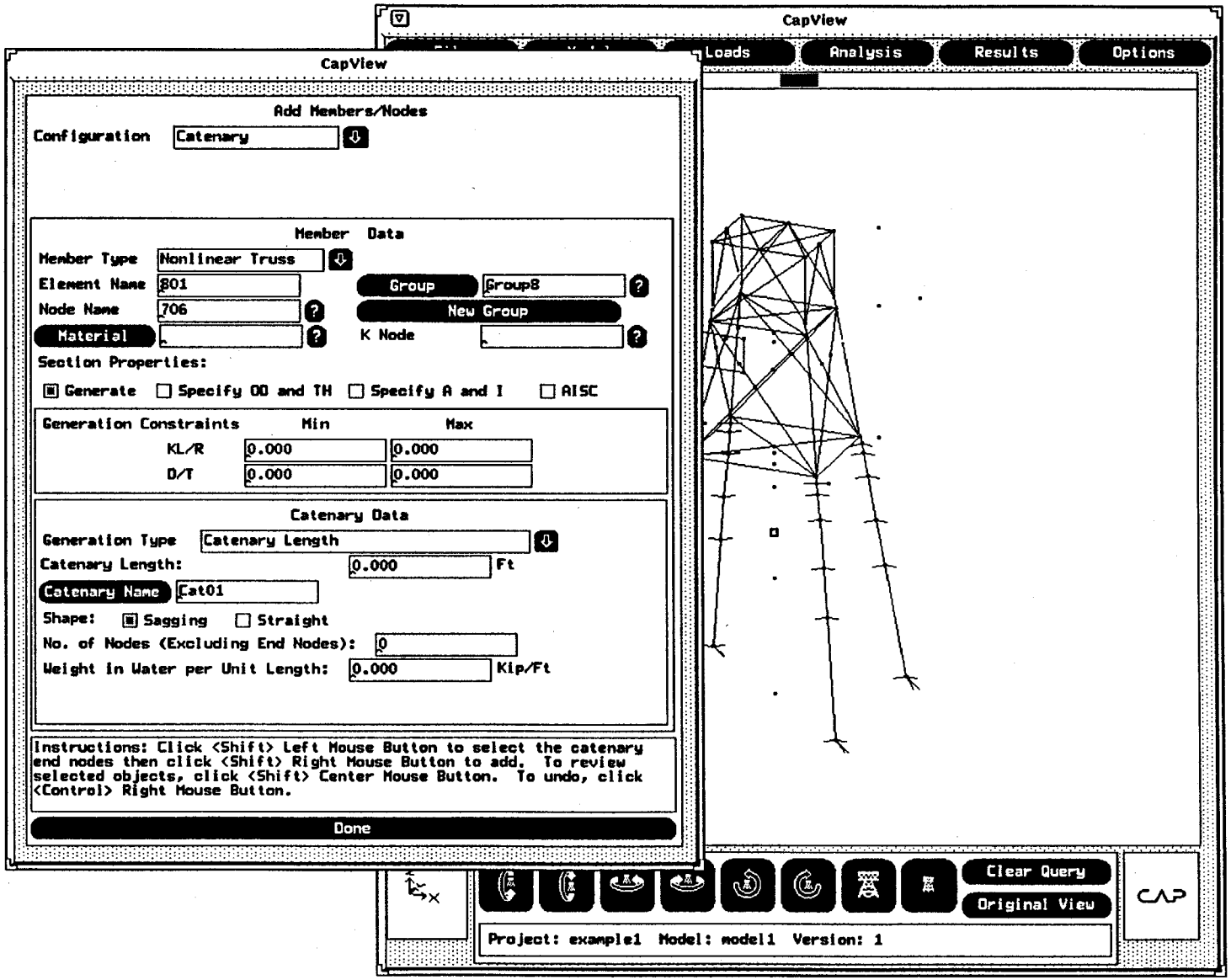

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Catenary
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Catenary Data
Generation Type
Catenary Length
Catenary Length:
0.000
Ft
Catenary Name
Cat01
Shape:
Sagging
Straight
No. of Nodes (Excluding End Nodes):
0
Weight in Water per Unit Length:
0.000
Kip/Ft
Instructions: Click <Shift> Left Mouse Button to select the catenary end nodes then click <Shift> Right Mouse Button to add. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-40

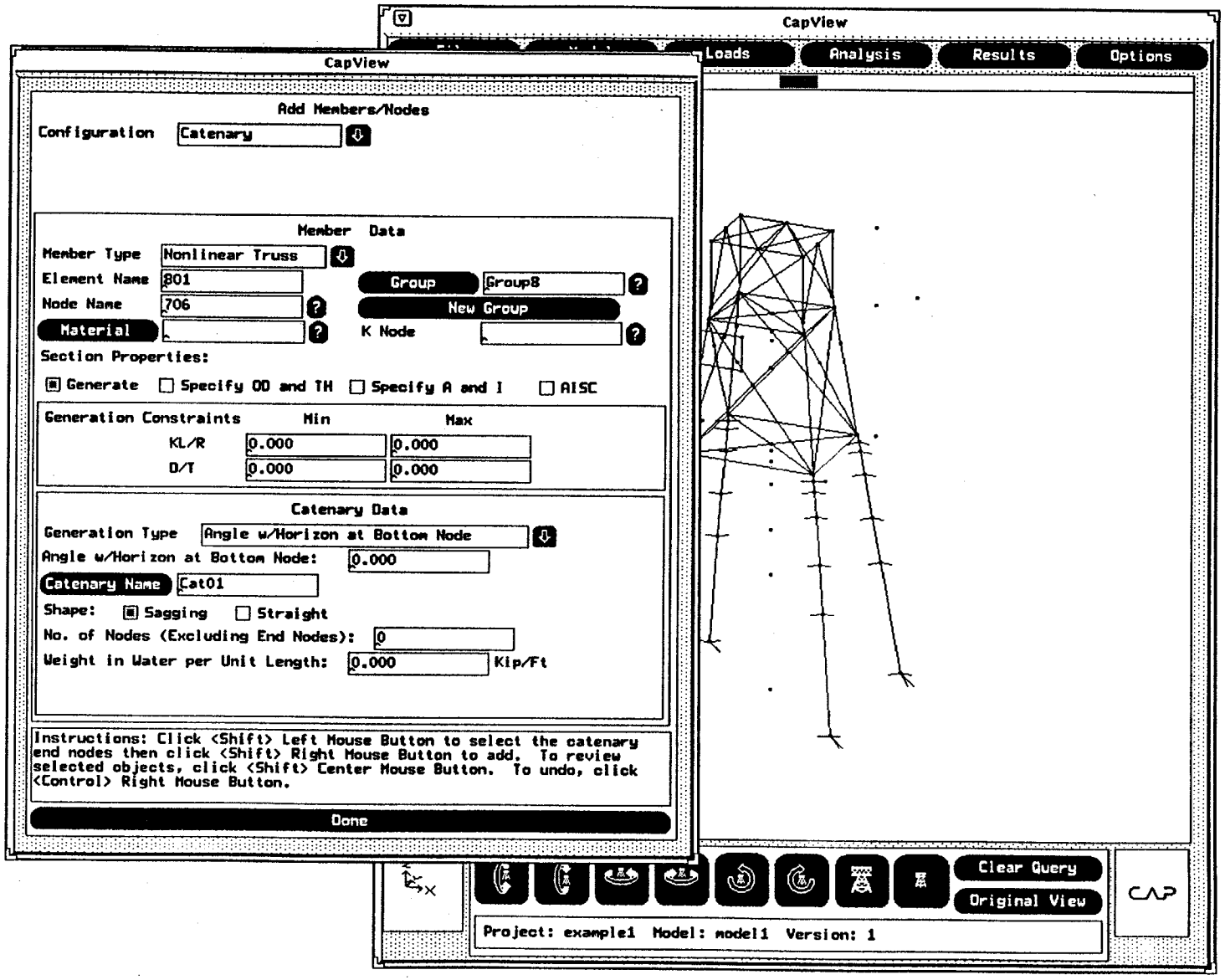

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Catenary
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Catenary Data
Generation Type
Angle w/Horizon at Bottom Node
Angle w/Horizon at Bottom Node:
0.000
Catenary Name
Cat01
Shape:
Sagging
Straight
No. of Nodes (Excluding End Nodes):
0
Weight in Water per Unit Length:
0.000
Kip/Ft
Instructions: Click <Shift> Left Mouse Button to select the catenary end nodes then click <Shift> Right Mouse Button to add. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-41

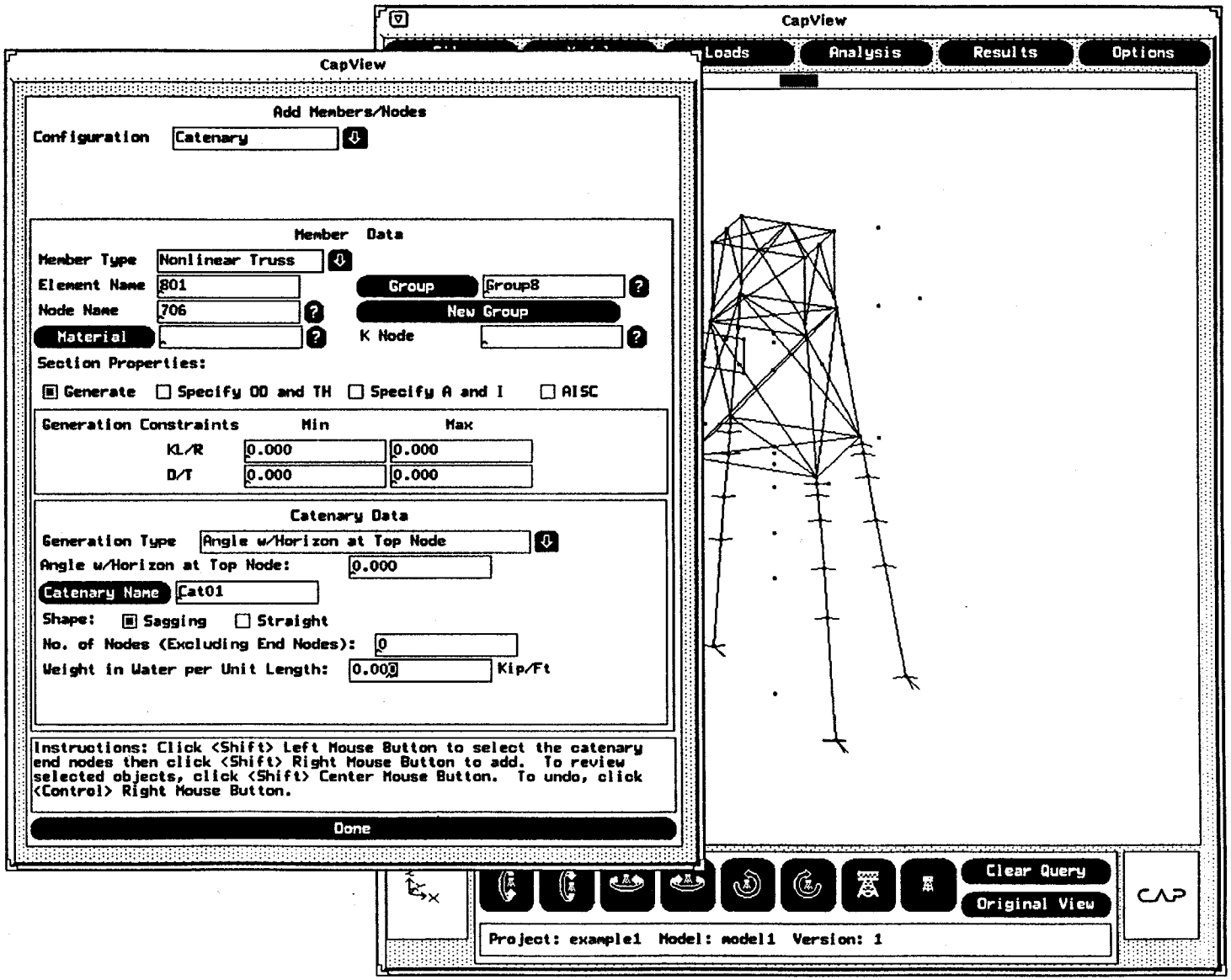

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Catenary
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Catenary Data
Generation Type
Angle w/Horizon at Top Node
Angle w/Horizon at Top Node:
0.000
Catenary Name
Cat01
Shape:
Sagging
Straight
No. of Nodes (Excluding End Nodes):
0
Weight in Water per Unit Length:
0.000
Kip/Ft
Instructions: Click <Shift> Left Mouse Button to select the catenary end nodes then click <Shift> Right Mouse Button to add. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-42

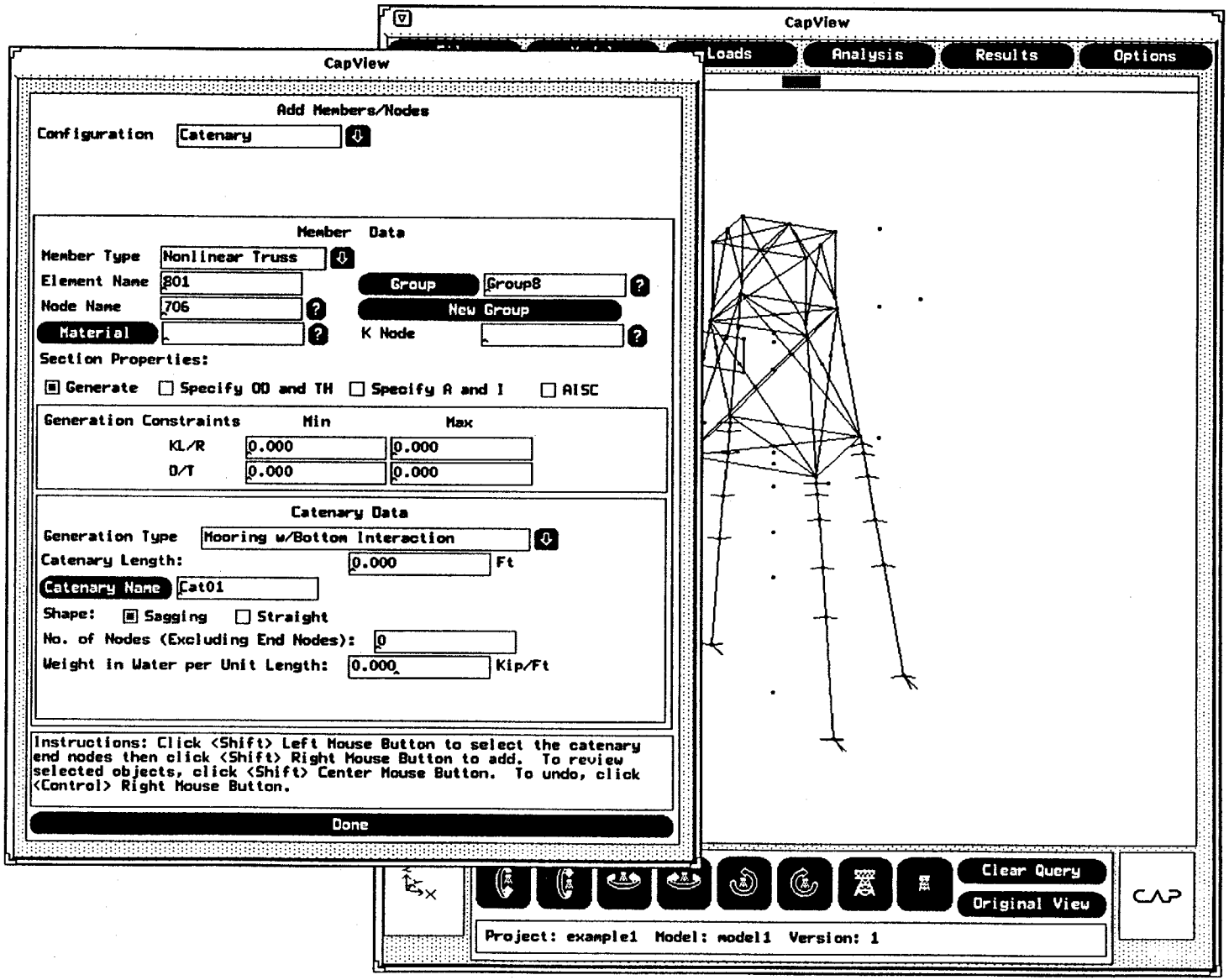

CapView
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Catenary
Member Data
Member Type
Nonlinear Truss
Element Name
801
Group
Group8
Node Name
706
New Group
Material
K Node
Section Properties:
Generate
Specify OD and TH
Specify A and I
AISC
Generation Constraints
Min
Max
KL/R
0.000
0.000
D/T
0.000
0.000
Catenary Data
Generation Type
Mooring w/Bottom Interaction
Catenary Length:
0.000
Ft
Catenary Name
Cat01
Shape:
Sagging
Straight
No. of Nodes (Excluding End Nodes):
0
Weight in Water per Unit Length:
0.000
Kip/Ft
Instructions: Click <Shift> Left Mouse Button to select the catenary end nodes then click <Shift> Right Mouse Button to add. To review selected objects, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-43

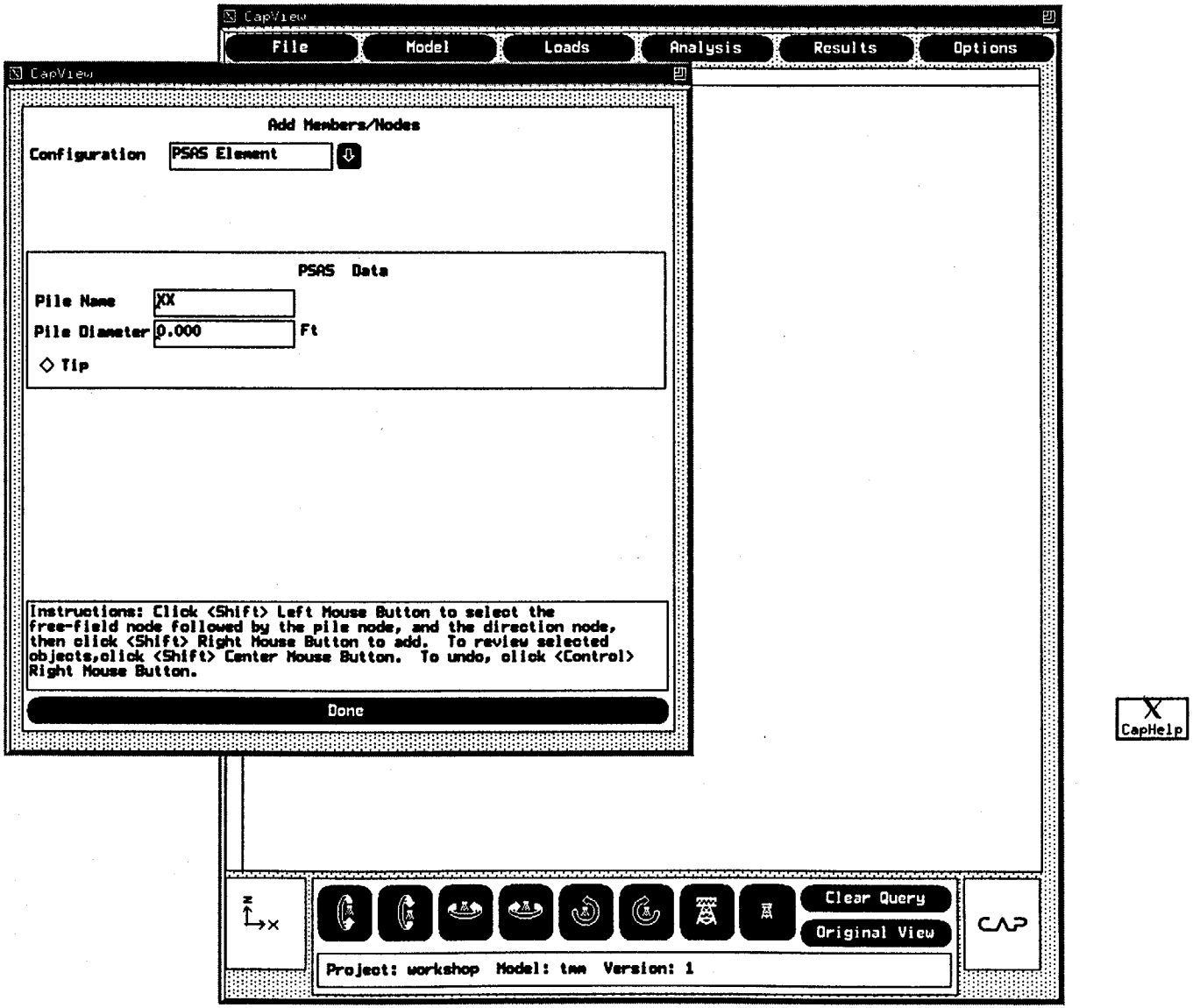

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
PSAS Element
PSAS Data
Pile Name
XX
Pile Diameter
0.000
Ft
Tip
Instructions: Click <Shift> Left Mouse Button to select the free-field node followed by the pile node, and the direction node, then click <Shift> Right Mouse Button to add. To review selected objects,click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-44

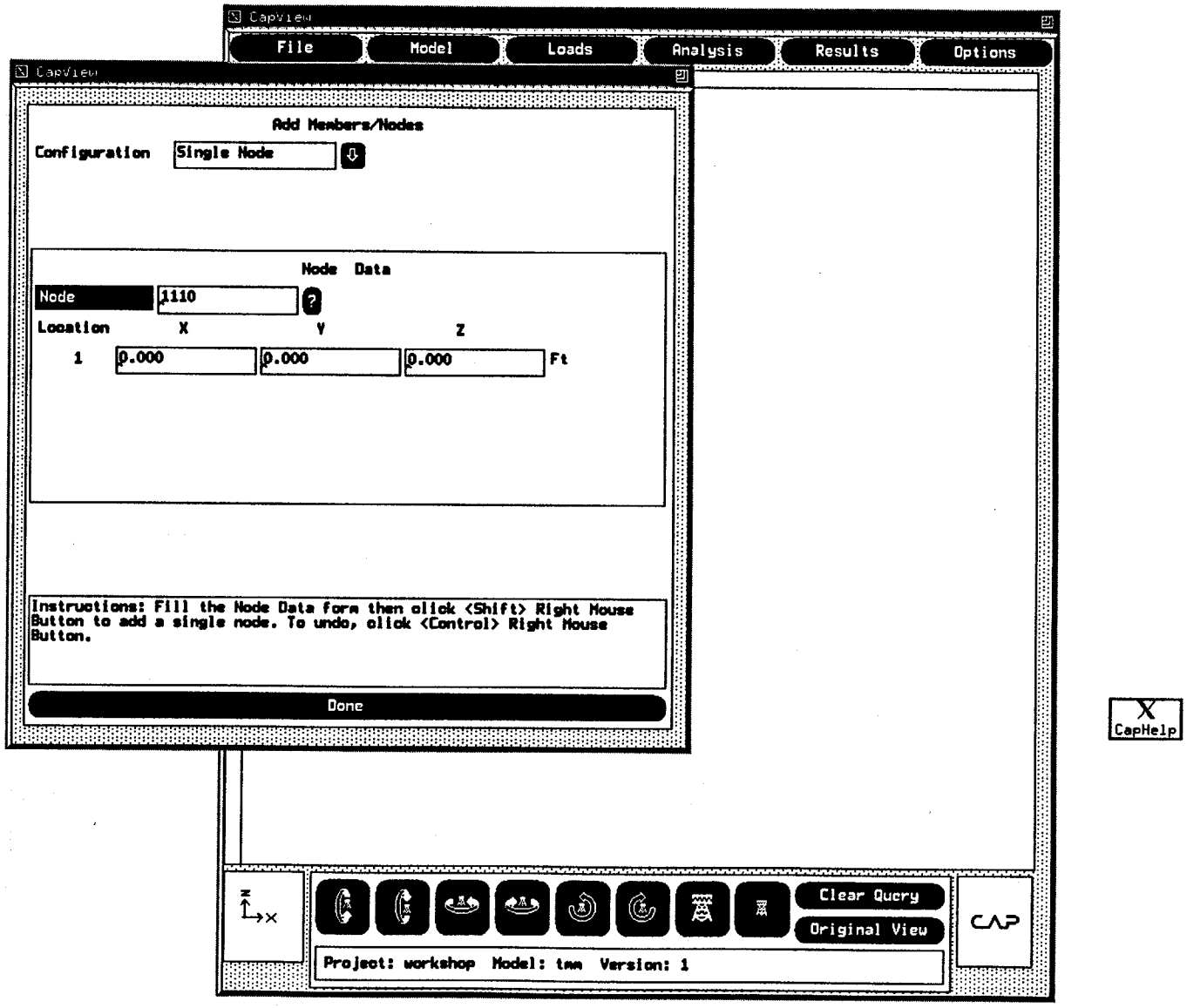

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Single Node
Node Data
Node
1110
Location
X
Y
Z
1
0.000
0.000
0.000
Ft
Instructions: Fill the Node Data form then click <Shift> Right Mouse Button to add a single node. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-45

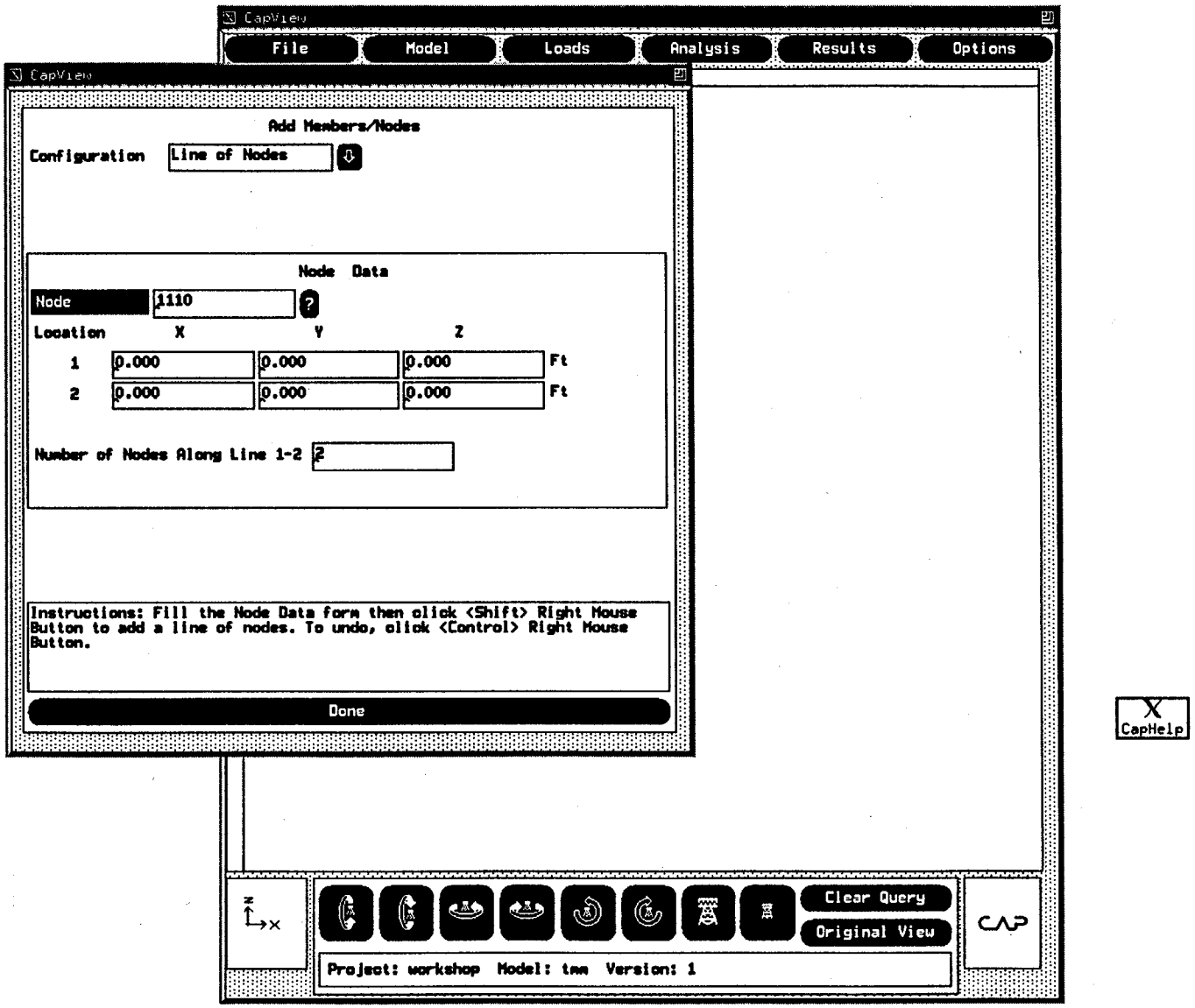

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Line of Nodes
Node Data
Node
1110
?
Location
X
Y
Z
1
0.000
0.000
0.000
Ft
2
0.000
0.000
0.000
Ft
Number of Nodes Along Line 1-2
2
Instructions: Fill the Node Data form then click <Shift> Right Mouse Button to add a line of nodes. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-46

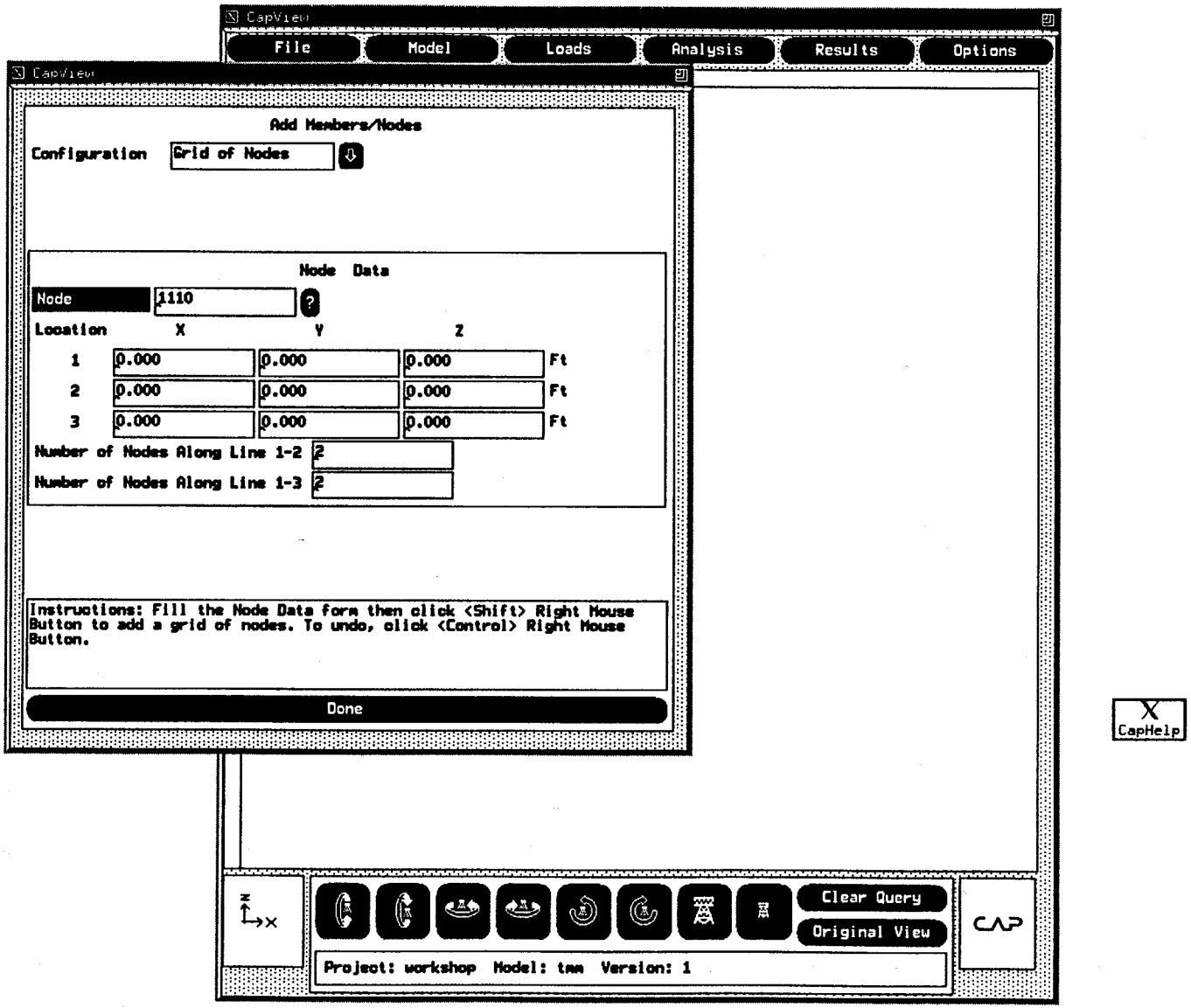

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Members/Nodes
Configuration
Grid of Nodes
Node Data
Node
1110
Location
X
Y
Z
1
0.000
0.000
0.000
Ft
2
0.000
0.000
0.000
Ft
3
0.000
0.000
0.000
Ft
Number of Nodes Along Line 1-2
2
Number of Nodes Along Line 1-3
2
Instructions: Fill the Node Data form then click <Shift> Right Mouse Button to add a grid of nodes. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CapHelp

Figure 8-47

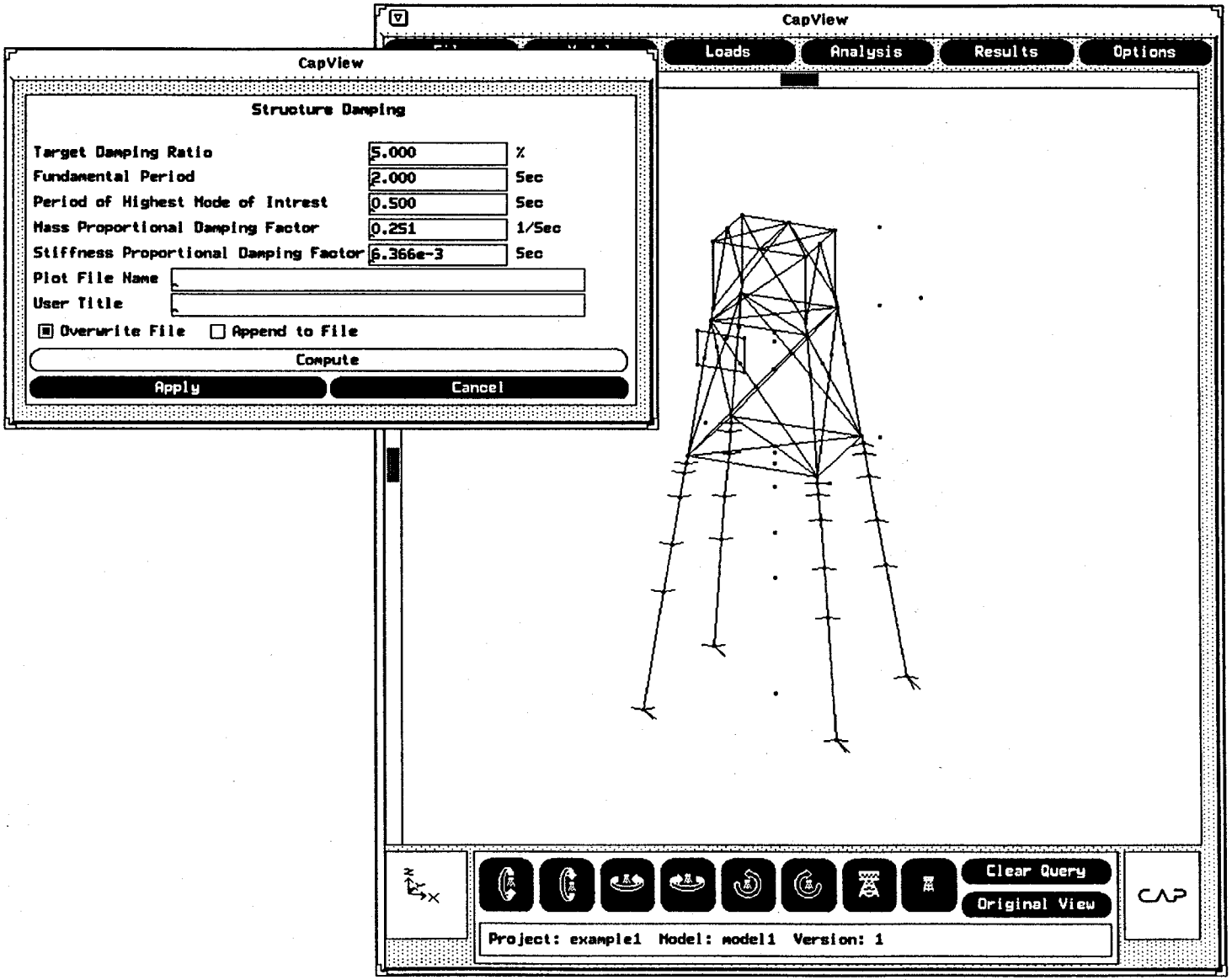

CapView
Loads
Analysis
Results
Options
CapView
Structure Damping
Target Damping Ratio 5.000 %
Fundamental Period 2.000 Sec
Period of Highest Mode of Intrest 0.500 Sec
Mass Proportional Damping Factor 0.251 1/Sec
Stiffness Proportional Damping Factor 6.366e-3 Sec
Plot File Name
User Title
Overwrite File
Append to File
Compute
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-48

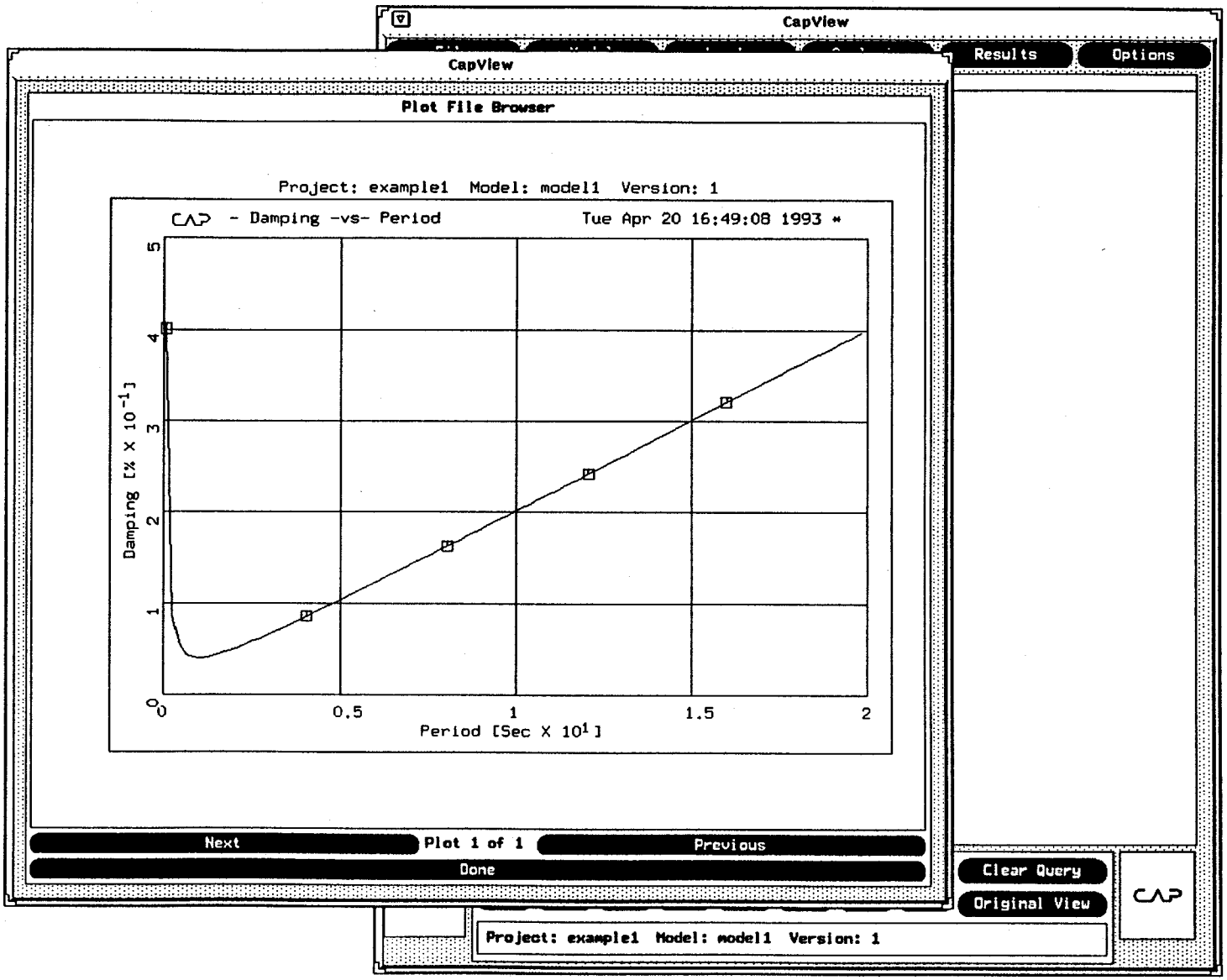

CapView
Results
Options
CapView
Plot File Browser
Project: example1 Model: model1 Version: 1
CAP - Damping -vs- Period
Tue Apr 20 16:49:08 1993
Damping [% X 10^-1]
0
1
2
3
4
5
0
0.5
1
1.5
2
Period [Sec X 10^1]
Next
Plot 1 of 1
Previous
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-49

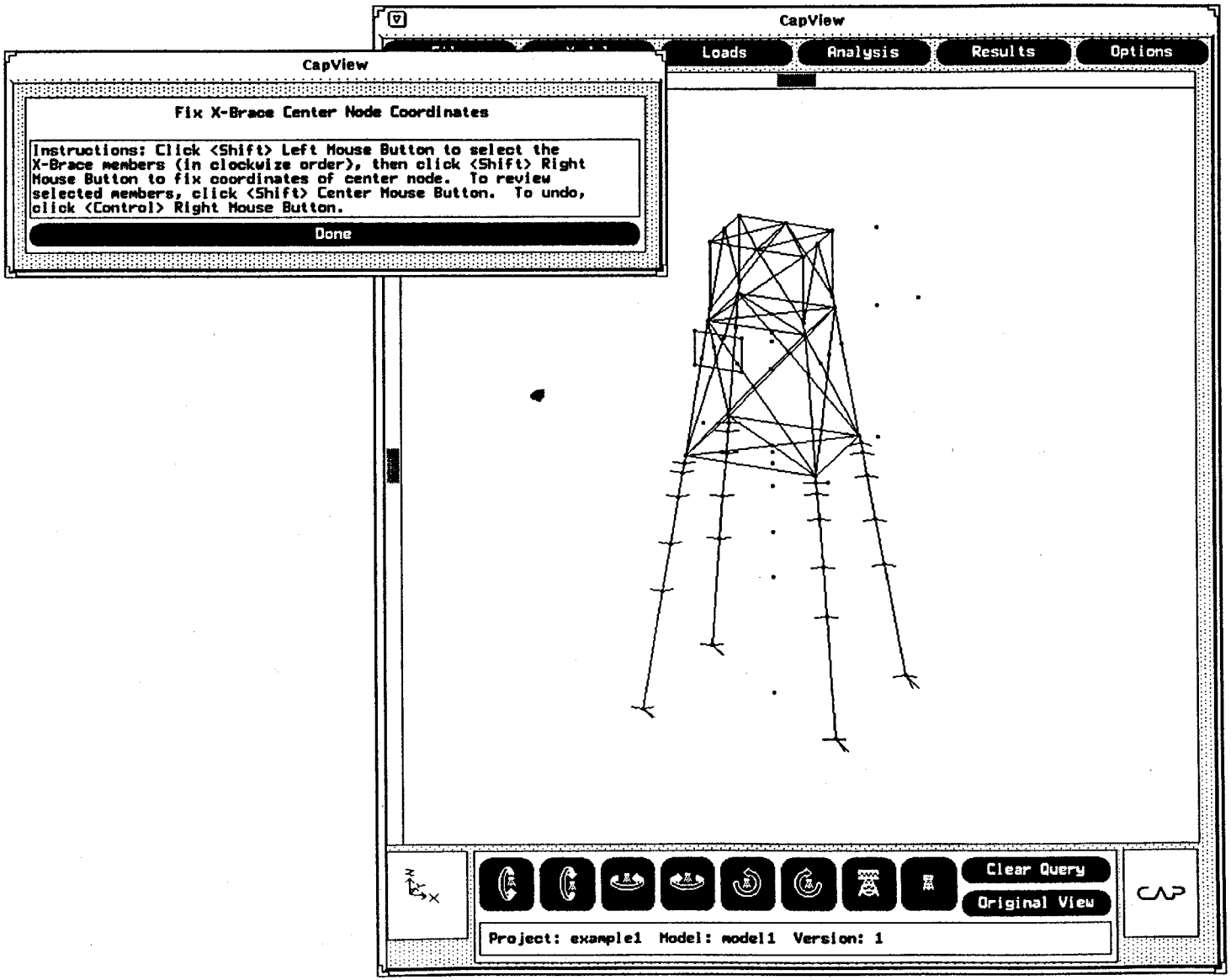

CapView
Loads
Analysis
Results
Options
CapView
Fix X-Brace Center Node Coordinates
Instructions: Click <Shift> Left Mouse Button to select the X-Brace members (in clockwize order), then click <Shift> Right Mouse Button to fix coordinates of center node. To review selected members, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-50

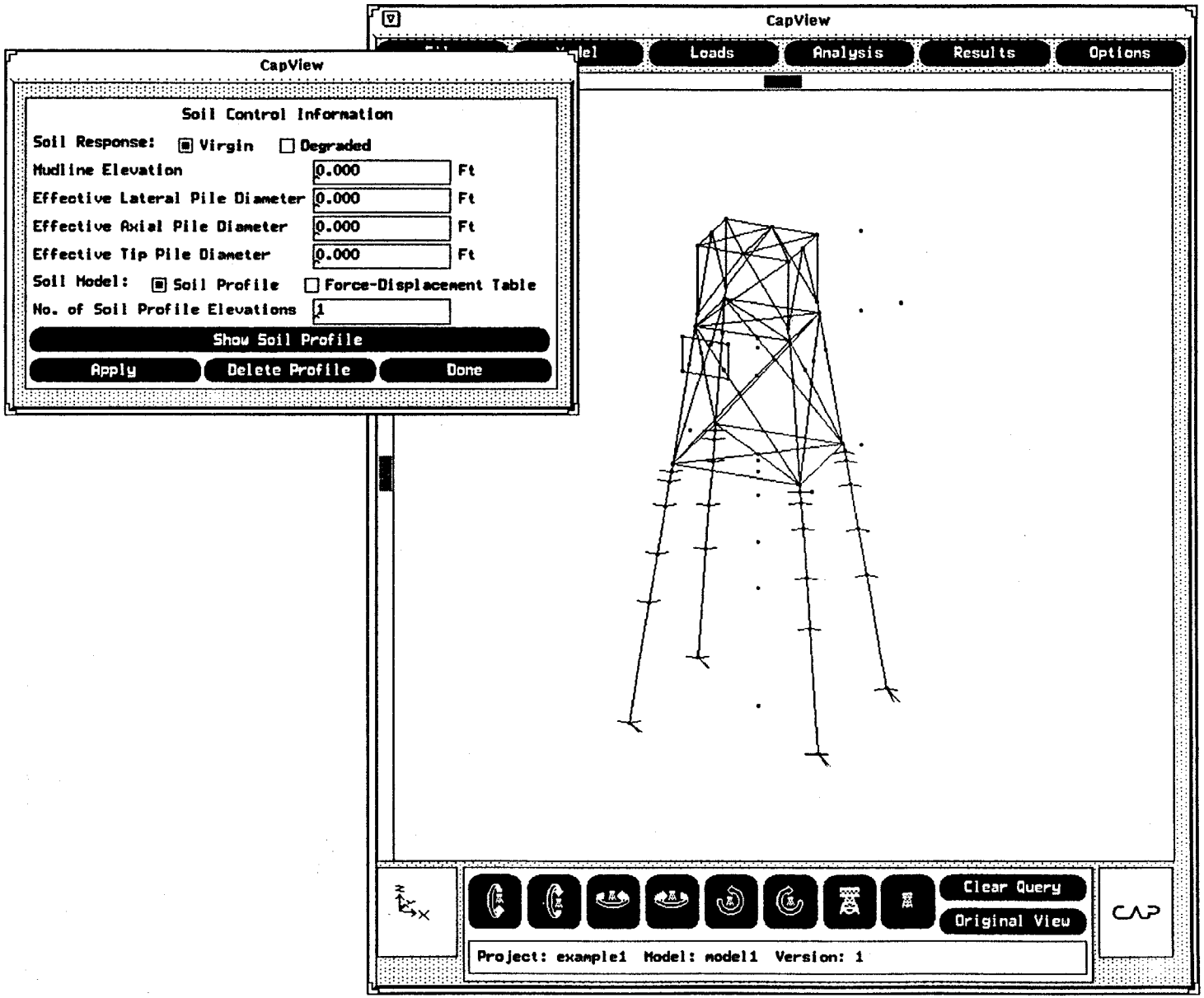

CapView
Loads
Analysis
Results
Options
CapView
Soil Control Information
Soil Response:
Virgin
Degraded
Mudline Elevation
0.000
Ft
Effective Lateral Pile Diameter
0.000
Ft
Effective Axial Pile Diameter
0.000
Ft
Effective Tip Pile Diameter
0.000
Ft
Soil Model:
Soil Profile
Force-Displacement Table
No. of Soil Profile Elevations
1
Show Soil Profile
Apply
Delete Profile
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-51

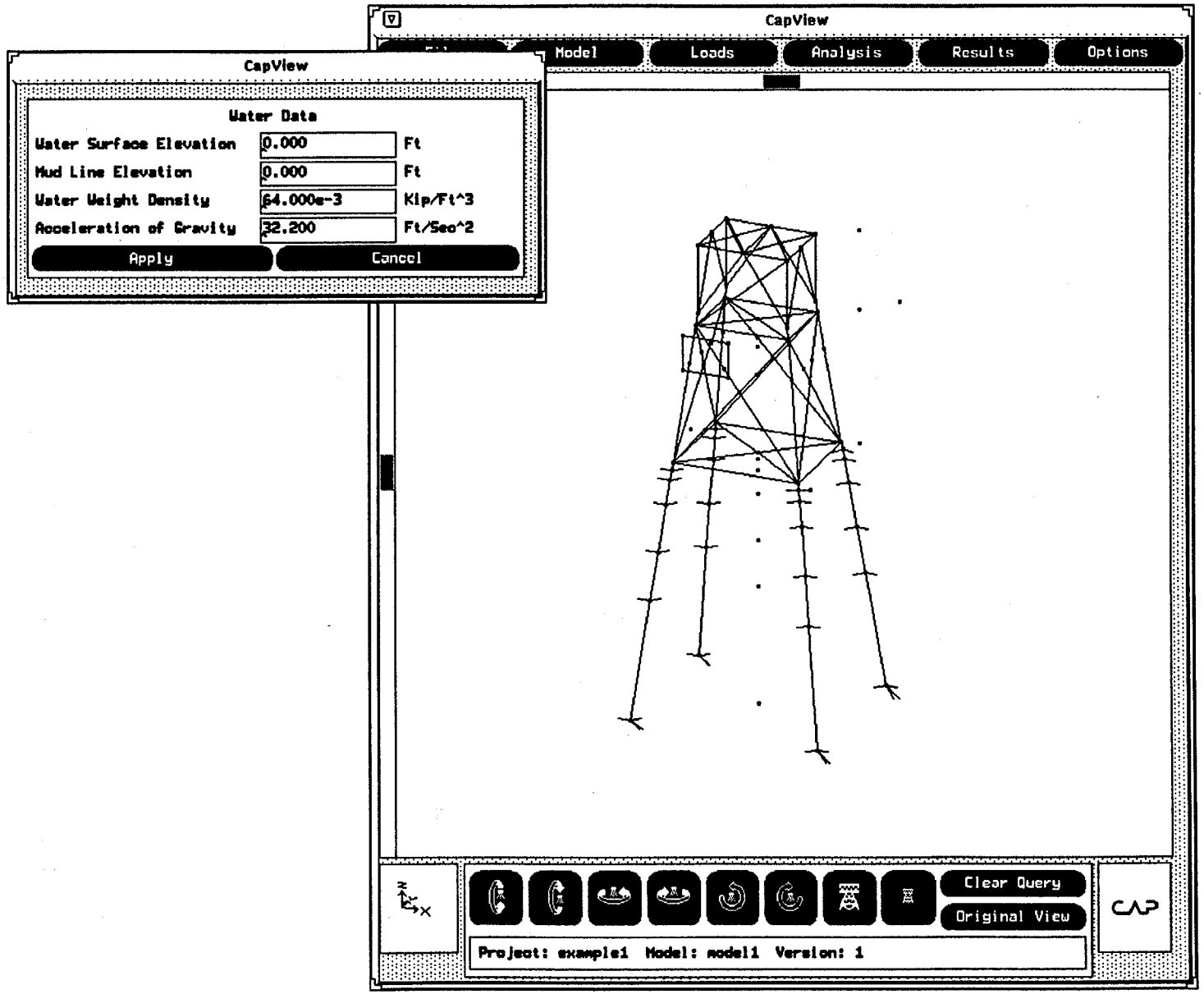

CapView
Model
Loads
Analysis
Results
Options
CapView
Water Data
Water Surface Elevation
0.000
Ft
Mud Line Elevation
0.000
Ft
Water Weight Density
64.000e-3
Kip/Ft^3
Acceleration of Gravity
32.200
Ft/Sec^2
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-52

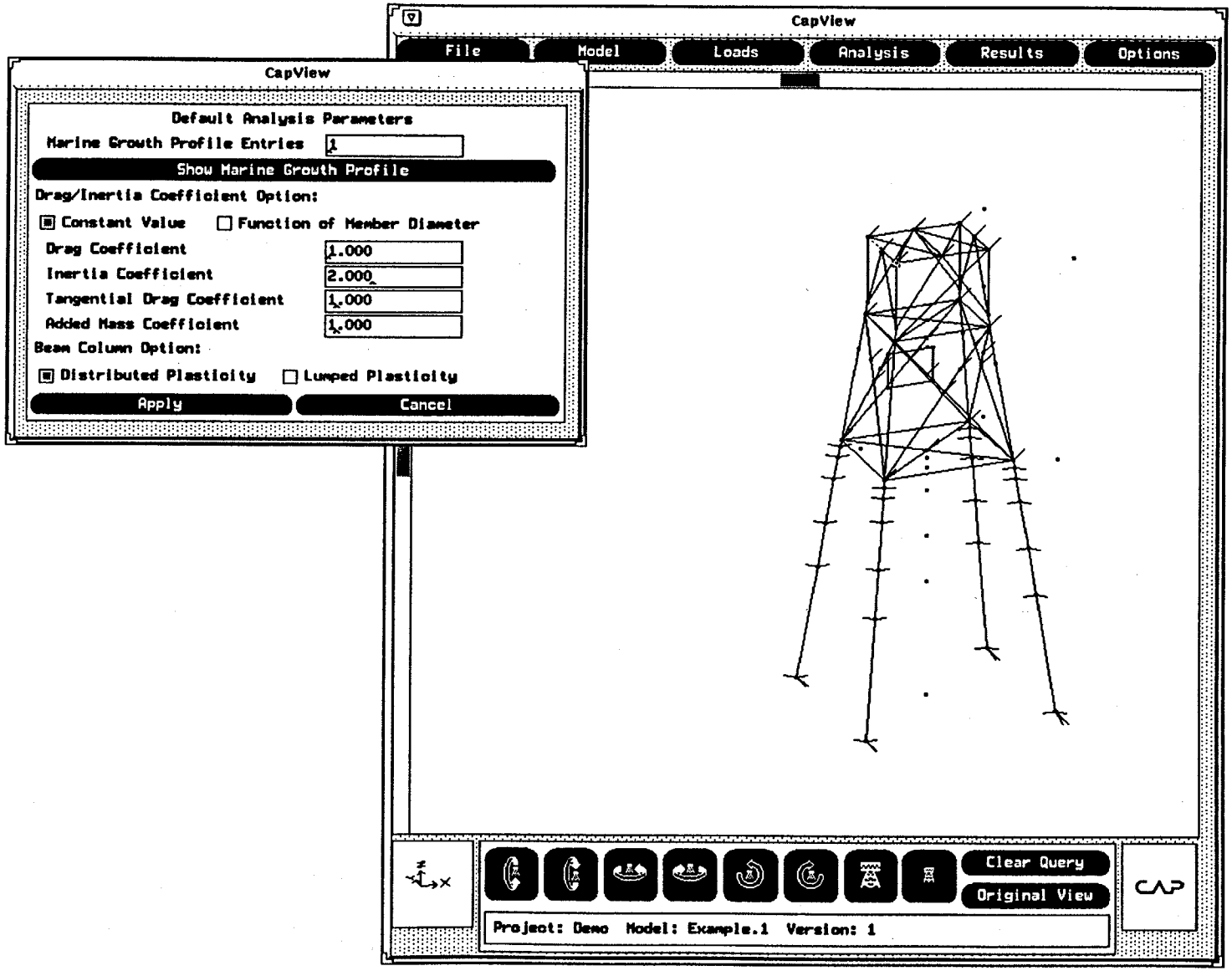

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Default Analysis Parameters
Marine Growth Profile Entries
1
Show Marine Growth Profile
Drag/Inertia Coefficient Option:
Constant Value
Function of Member Diameter
Drag Coefficient
1.000
Inertia Coefficient
2.000
Tangential Drag Coefficient
1.000
Added Mass Coefficient
1.000
Beam Column Option:
Distributed Plasticity
Lumped Plasticity
Apply
Cancel
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 8-53

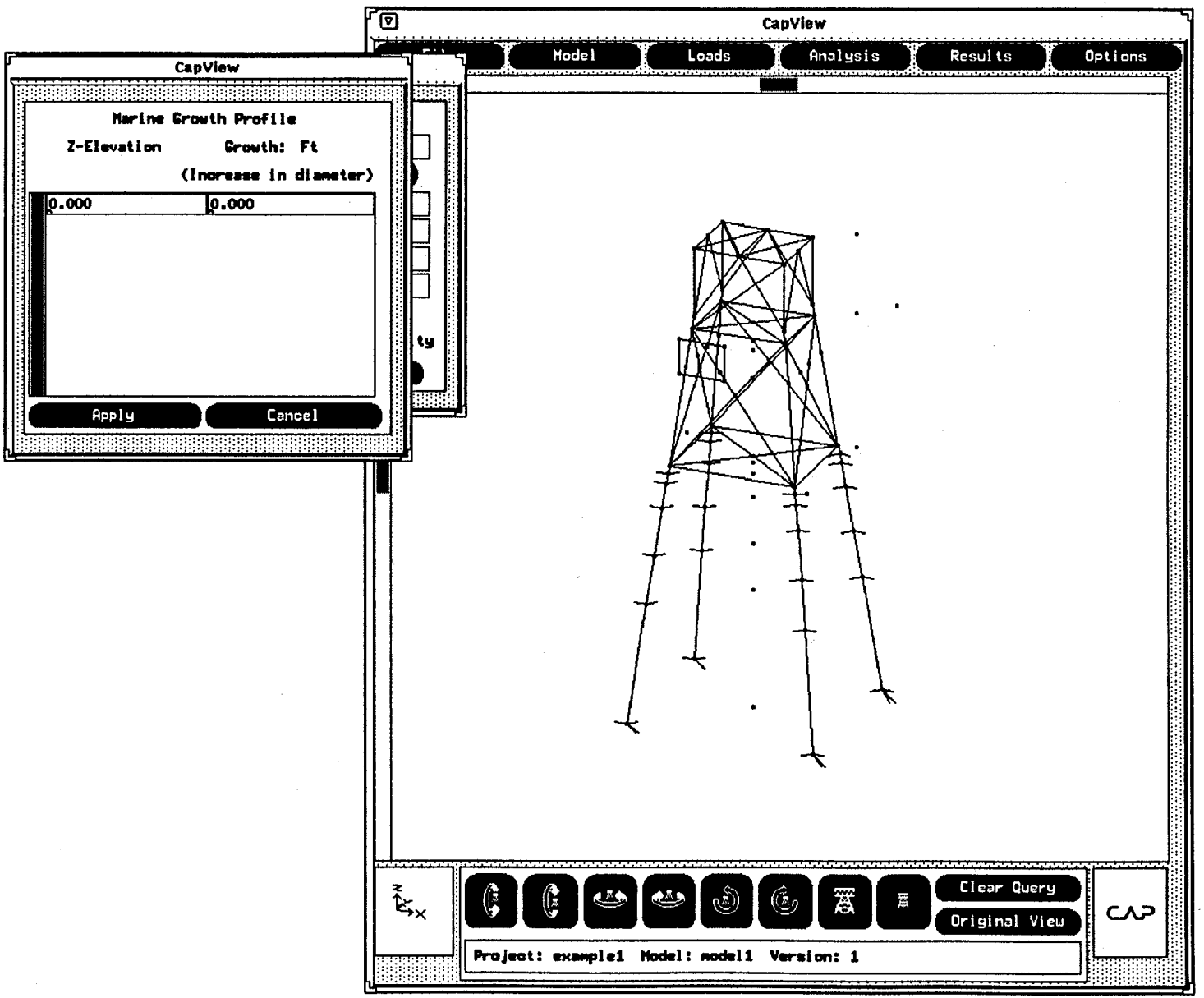

CapView
Model
Loads
Analysis
Results
Options
CapView
Marine Growth Profile
Z-Elevation
Growth: Ft
(Increase in diameter)
0.000
0.000
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-54

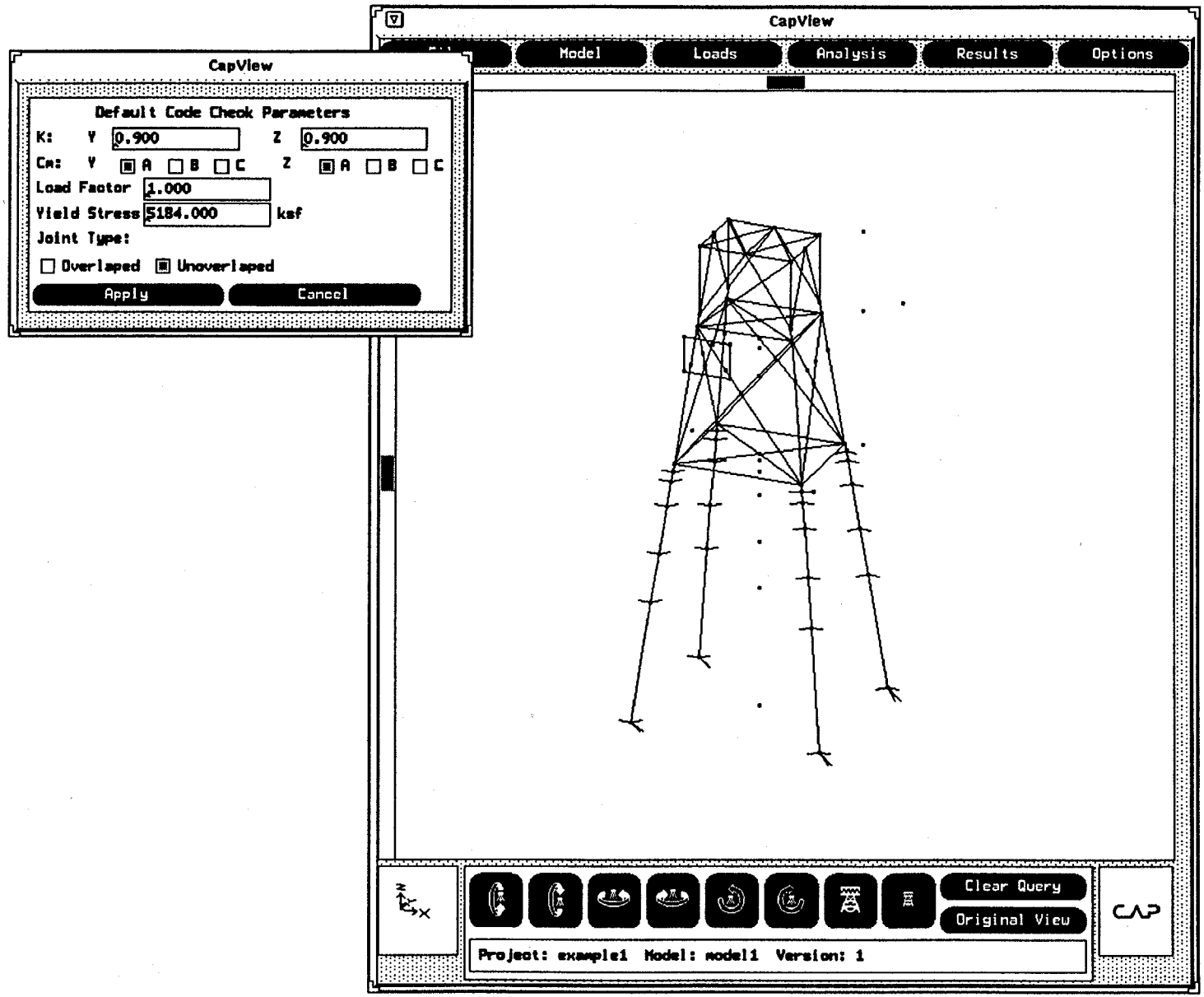

CapView
Model
Loads
Analysis
Results
Options
CapView
Default Code Check Parameters
K: Y 0.900 Z 0.900
Cm: Y A B C Z A B C
Load Factor 1.000
Yield Stress 5184.000 ksf
Joint Type:
Overlaped Unoverlaped
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-55

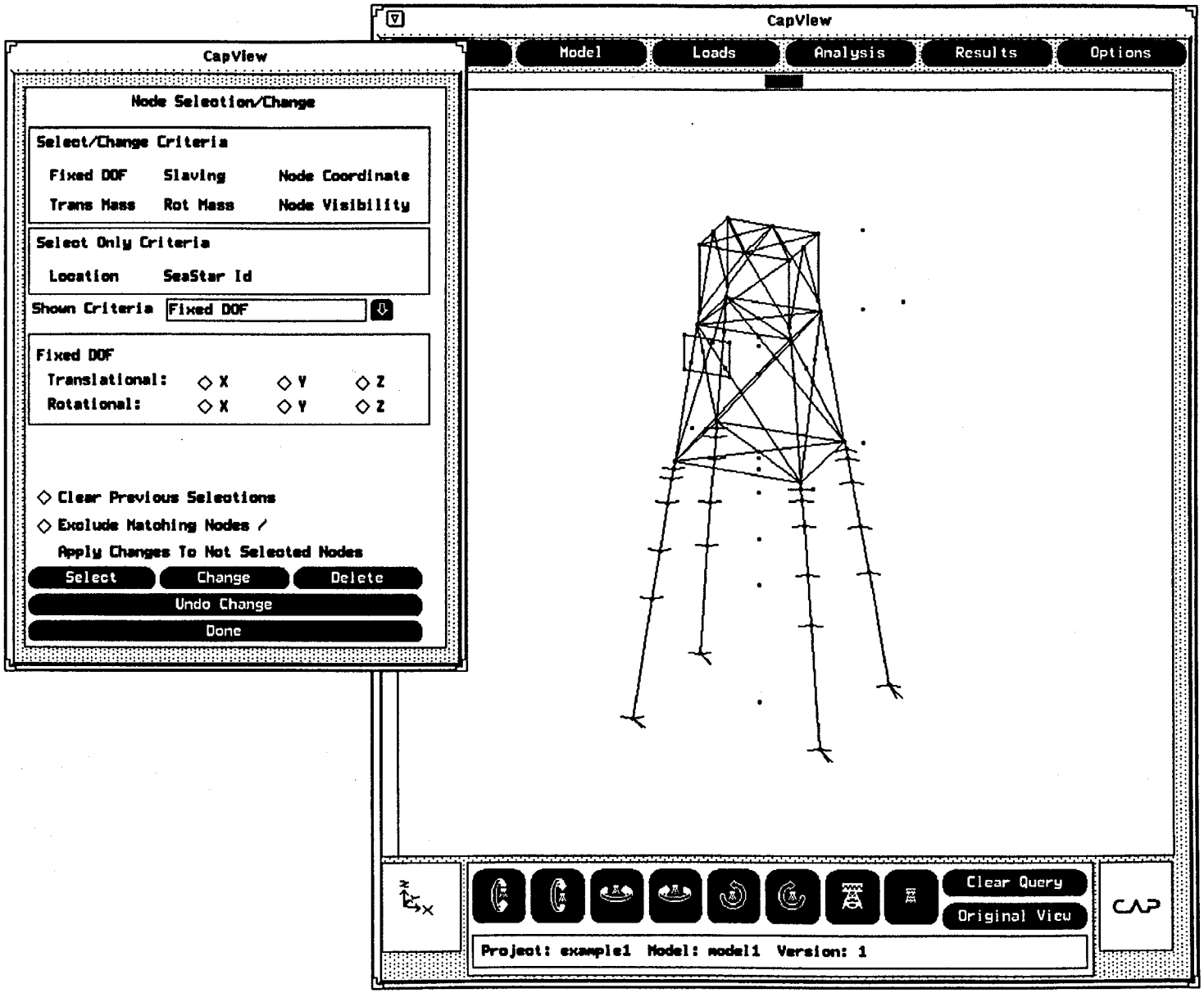

CapView
Model
Loads
Analysis
Results
Options
CapView
Node Selection/Change
Select/Change Criteria
Fixed DOF
Slaving
Node Coordinate
Trans Mass
Rot Mass
Node Visibility
Select Only Criteria
Location
SeaStar Id
Shown Criteria
Fixed DOF
Fixed DOF
Translational:
X
Y
Z
Rotational:
X
Y
Z
Clear Previous Selections
Exclude Matching Nodes /
Apply Changes To Not Selected Nodes
Select
Change
Delete
Undo Change
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-56

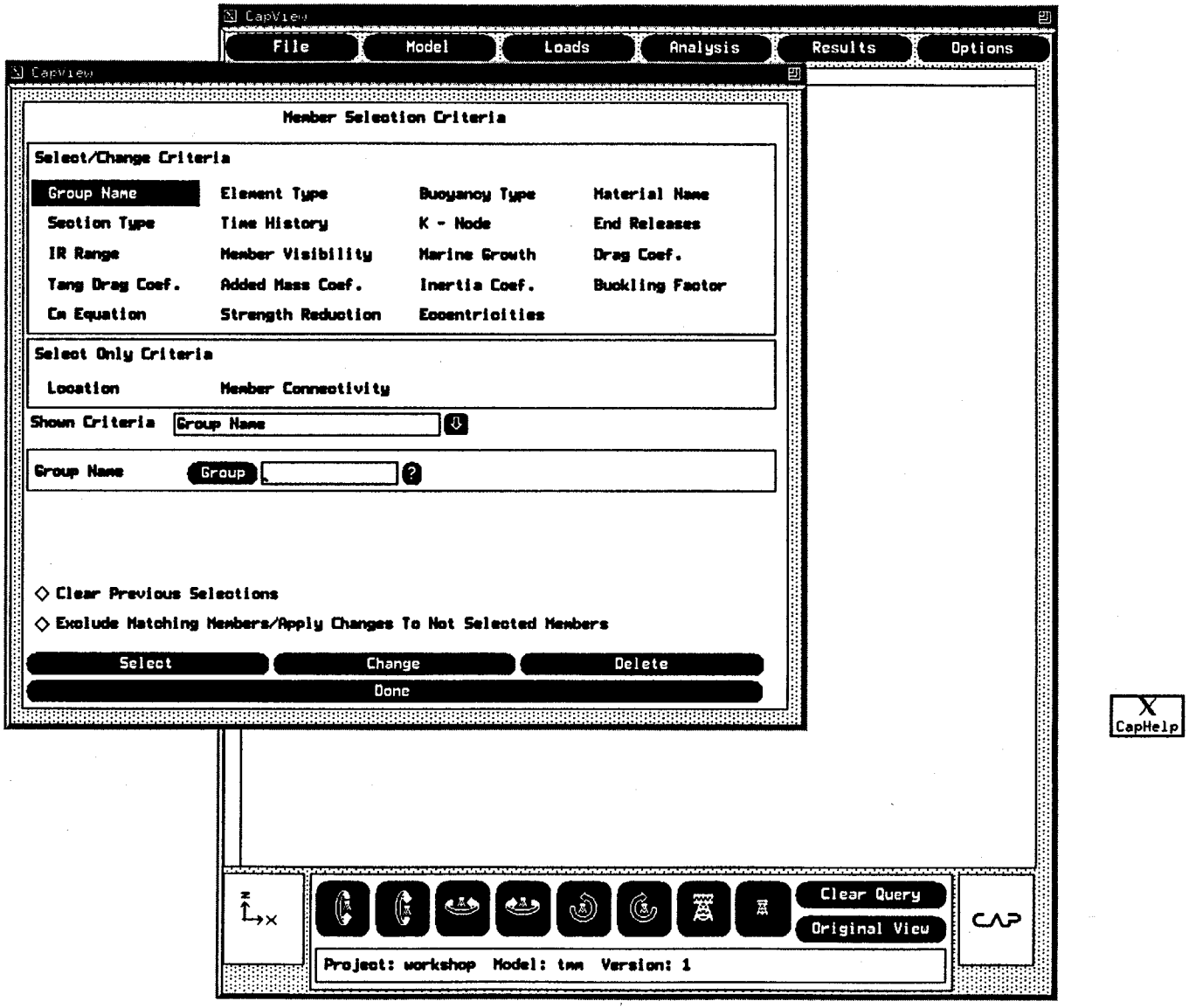

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Member Selection Criteria
Select/Change Criteria
Group Name
Element Type
Buoyancy Type
Material Name
Section Type
Time History
K - Node
End Releases
IR Range
Member Visibility
Marine Growth
Drag Coef.
Tang Drag Coef.
Added Mass Coef.
Inertia Coef.
Buckling Factor
Cm Equation
Strength Reduction
Eccentricities
Select Only Criteria
Location
Member Connectivity
Shown Criteria
Group Name
Group Name
Group
Clear Previous Selections
Exclude Matching Members/Apply Changes To Not Selected Members
Select
Change
Delete
Done
Clear Query
Original View
Project: workshop Model: tmm Version: 1
CAP
CapHelp

Figure 8-57

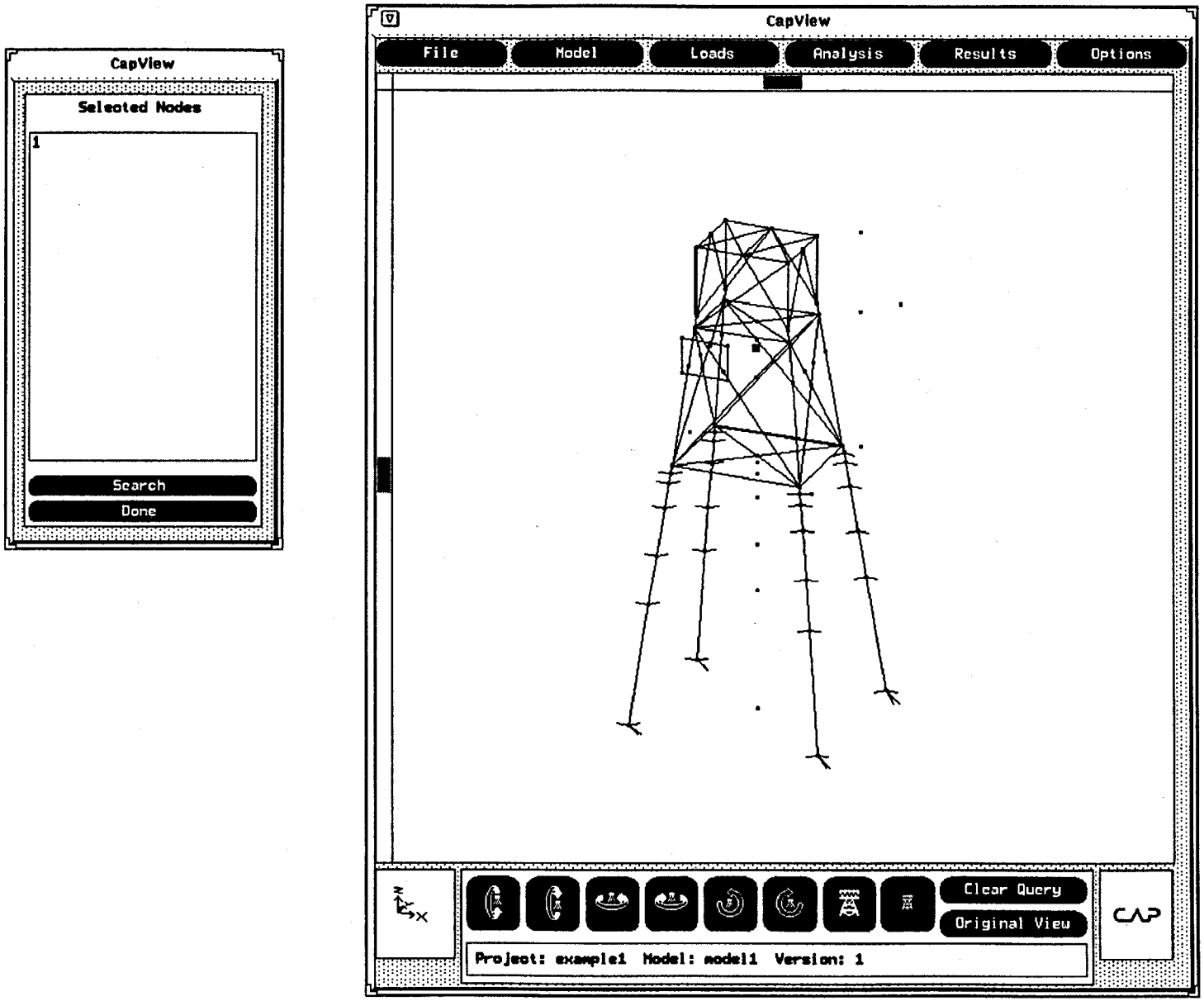

CapView
Selected Nodes
1
Search
Done
CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-58

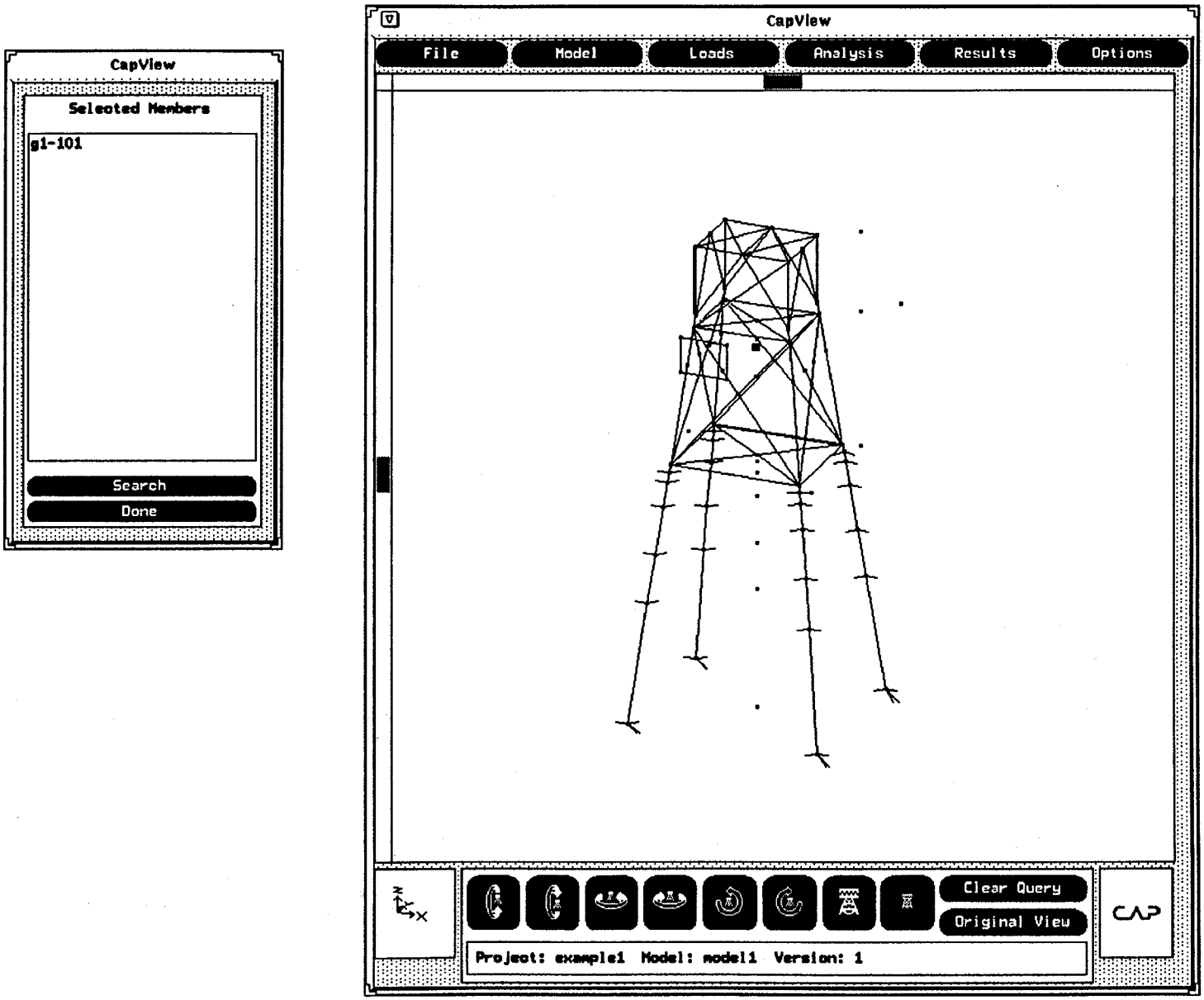

CapView
Selected Members
g1-101
Search
Done
CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-59

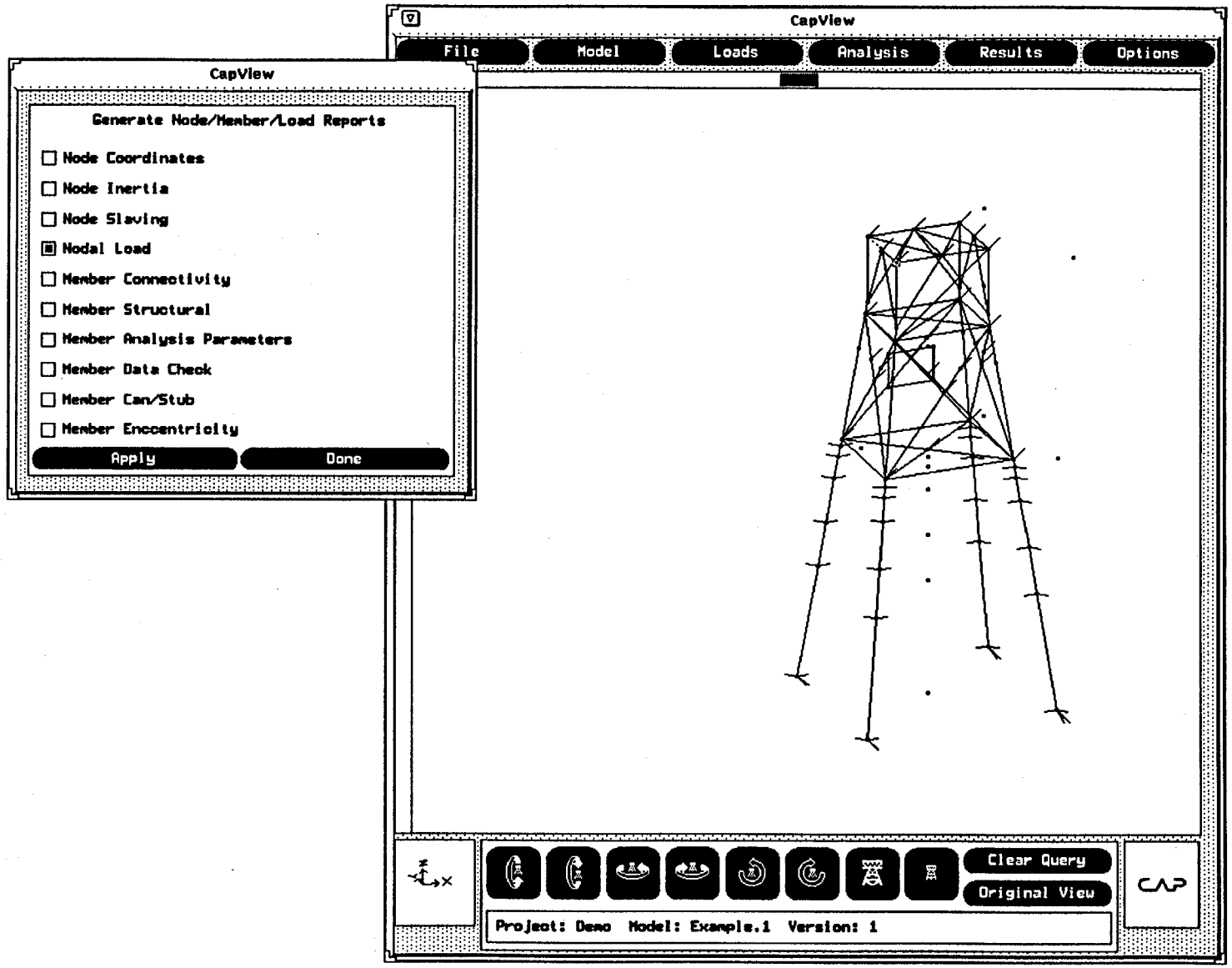

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Generate Node/Member/Load Reports
Node Coordinates
Node Inertia
Node Slaving
Nodal Load
Member Connectivity
Member Structural
Member Analysis Parameters
Member Data Check
Member Can/Stub
Member Eccentricity
Apply
Done
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 8-60

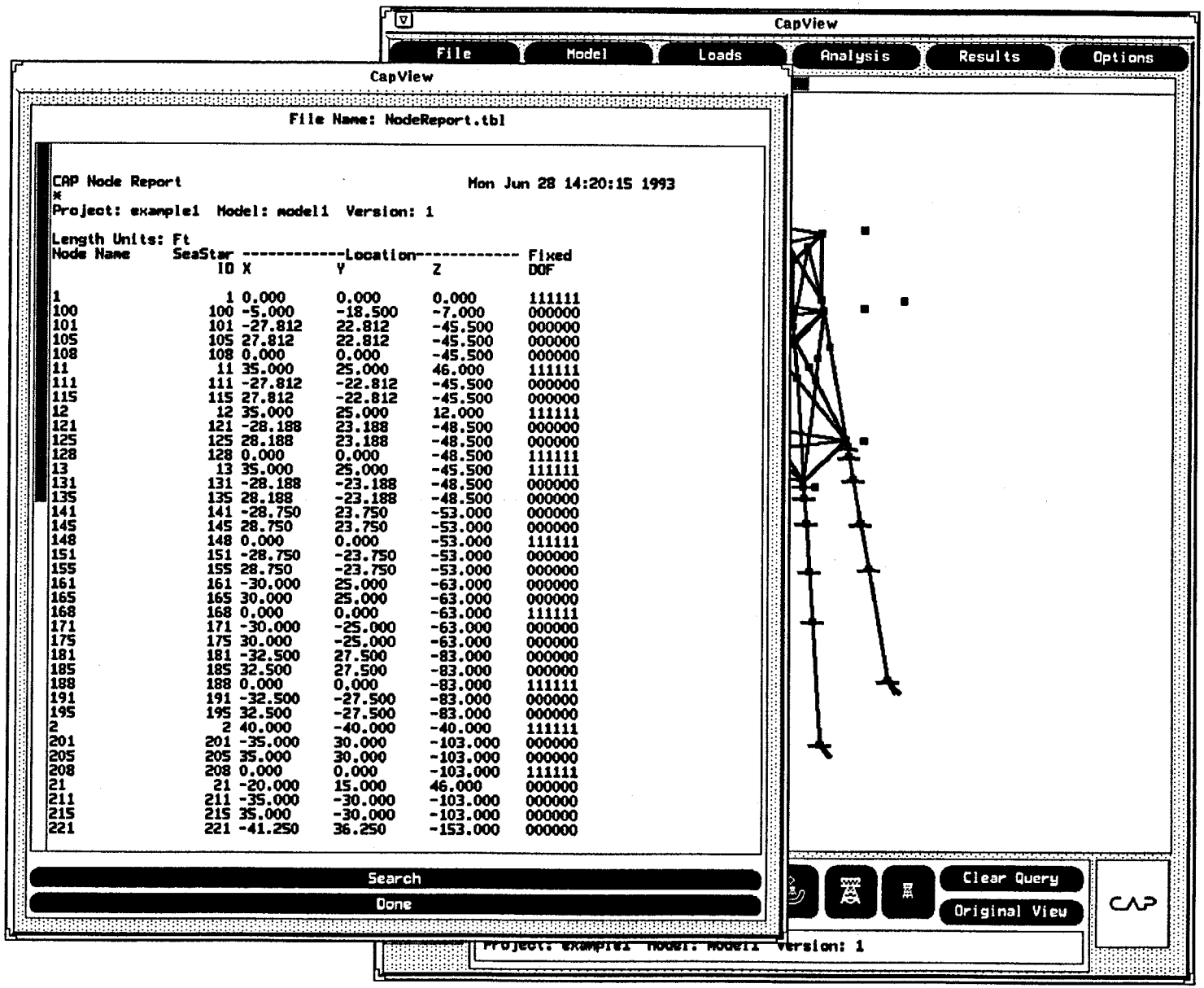

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: NodeReport.tbl
CAP Node Report
Mon Jun 28 14:20:15 1993
*
Project: example1 Model: model1 Version: 1
Length Units: Ft
Node Name SeaStar ID
Location X Y Z
Fixed DOF
1 1 0.000 0.000 0.000 111111
100 100 -5.000 -18.500 -7.000 000000
101 101 -27.812 22.812 -45.500 000000
105 105 27.812 22.812 -45.500 000000
108 108 0.000 0.000 -45.500 000000
11 11 35.000 25.000 46.000 111111
111 111 -27.812 -22.812 -45.500 000000
115 115 27.812 -22.812 -45.500 000000
12 12 35.000 25.000 12.000 111111
121 121 -28.188 23.188 -48.500 000000
125 125 28.188 23.188 -48.500 000000
128 128 0.000 0.000 -48.500 111111
13 13 35.000 25.000 -45.500 111111
131 131 -28.188 -23.188 -48.500 000000
135 135 28.188 -23.188 -48.500 000000
141 141 -28.750 23.750 -53.000 000000
145 145 28.750 23.750 -53.000 000000
148 148 0.000 0.000 -53.000 111111
151 151 -28.750 -23.750 -53.000 000000
155 155 28.750 -23.750 -53.000 000000
161 161 -30.000 25.000 -63.000 000000
165 165 30.000 25.000 -63.000 000000
168 168 0.000 0.000 -63.000 111111
171 171 -30.000 -25.000 -63.000 000000
175 175 30.000 -25.000 -63.000 000000
181 181 -32.500 27.500 -83.000 000000
185 185 32.500 27.500 -83.000 000000
188 188 0.000 0.000 -83.000 111111
191 191 -32.500 -27.500 -83.000 000000
195 195 32.500 -27.500 -83.000 000000
2 2 40.000 -40.000 -40.000 111111
201 201 -35.000 30.000 -103.000 000000
205 205 35.000 30.000 -103.000 000000
208 208 0.000 0.000 -103.000 111111
21 21 -20.000 15.000 46.000 000000
211 211 -35.000 -30.000 -103.000 000000
215 215 35.000 -30.000 -103.000 000000
221 221 -41.250 36.250 -153.000 000000
Search
Done
Clear Query
Original View
CAP

Figure 8-61

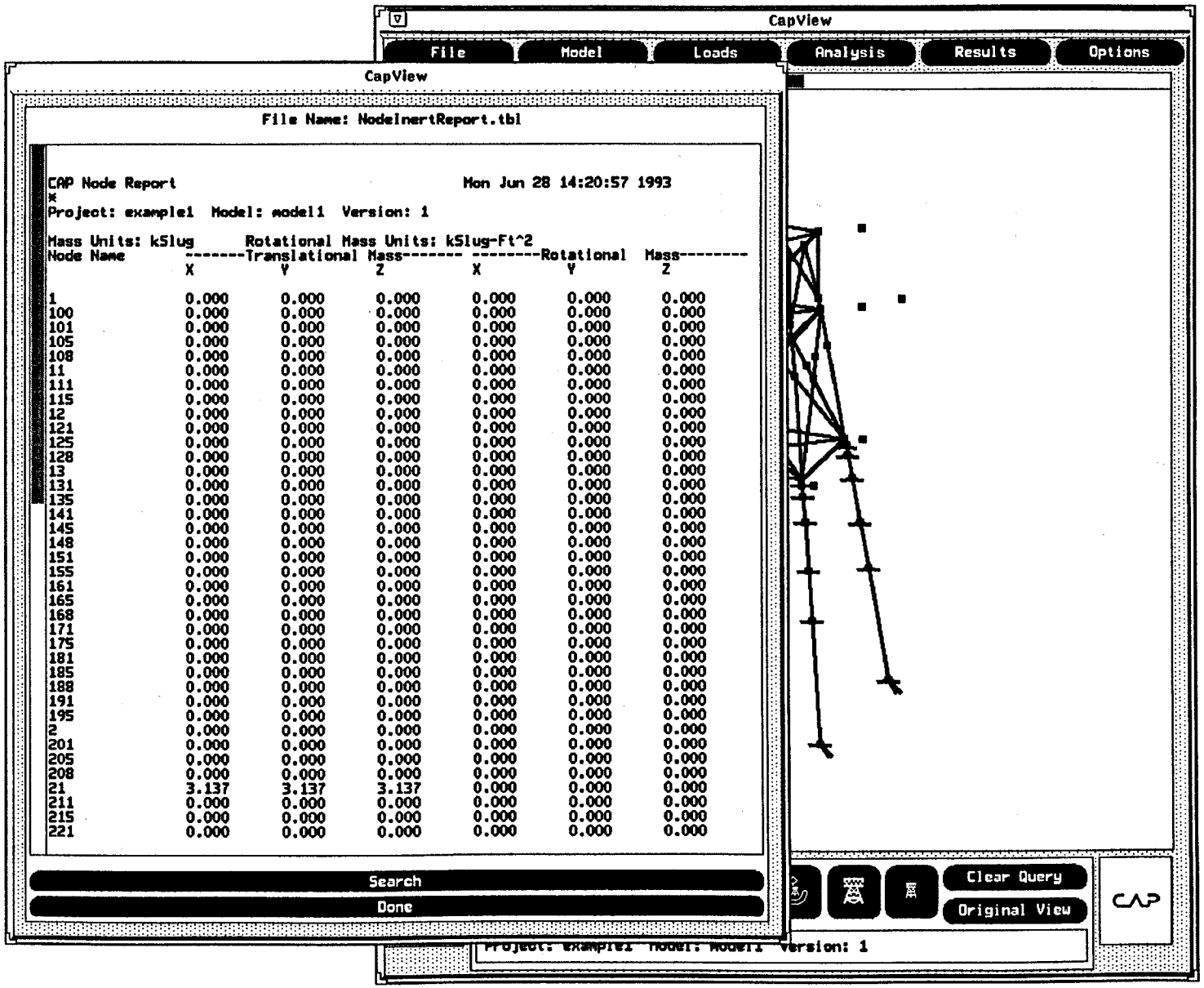

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: NodeInertReport.tbl
CAP Node Report
Mon Jun 28 14:20:57 1993
*
Project: example1 Model: model1 Version: 1
Mass Units: kSlug
Rotational Mass Units: kSlug-Ft^2
Node Name
-------Translational Mass-------
--------Rotational Mass--------
X Y Z X Y Z
1 0.000 0.000 0.000 0.000 0.000 0.000
100 0.000 0.000 0.000 0.000 0.000 0.000
101 0.000 0.000 0.000 0.000 0.000 0.000
105 0.000 0.000 0.000 0.000 0.000 0.000
108 0.000 0.000 0.000 0.000 0.000 0.000
11 0.000 0.000 0.000 0.000 0.000 0.000
111 0.000 0.000 0.000 0.000 0.000 0.000
115 0.000 0.000 0.000 0.000 0.000 0.000
12 0.000 0.000 0.000 0.000 0.000 0.000
121 0.000 0.000 0.000 0.000 0.000 0.000
125 0.000 0.000 0.000 0.000 0.000 0.000
128 0.000 0.000 0.000 0.000 0.000 0.000
13 0.000 0.000 0.000 0.000 0.000 0.000
131 0.000 0.000 0.000 0.000 0.000 0.000
135 0.000 0.000 0.000 0.000 0.000 0.000
141 0.000 0.000 0.000 0.000 0.000 0.000
145 0.000 0.000 0.000 0.000 0.000 0.000
148 0.000 0.000 0.000 0.000 0.000 0.000
151 0.000 0.000 0.000 0.000 0.000 0.000
155 0.000 0.000 0.000 0.000 0.000 0.000
161 0.000 0.000 0.000 0.000 0.000 0.000
165 0.000 0.000 0.000 0.000 0.000 0.000
168 0.000 0.000 0.000 0.000 0.000 0.000
171 0.000 0.000 0.000 0.000 0.000 0.000
175 0.000 0.000 0.000 0.000 0.000 0.000
181 0.000 0.000 0.000 0.000 0.000 0.000
185 0.000 0.000 0.000 0.000 0.000 0.000
188 0.000 0.000 0.000 0.000 0.000 0.000
191 0.000 0.000 0.000 0.000 0.000 0.000
195 0.000 0.000 0.000 0.000 0.000 0.000
2 0.000 0.000 0.000 0.000 0.000 0.000
201 0.000 0.000 0.000 0.000 0.000 0.000
205 0.000 0.000 0.000 0.000 0.000 0.000
208 0.000 0.000 0.000 0.000 0.000 0.000
21 3.137 3.137 3.137 0.000 0.000 0.000
211 0.000 0.000 0.000 0.000 0.000 0.000
215 0.000 0.000 0.000 0.000 0.000 0.000
221 0.000 0.000 0.000 0.000 0.000 0.000
Search
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-62

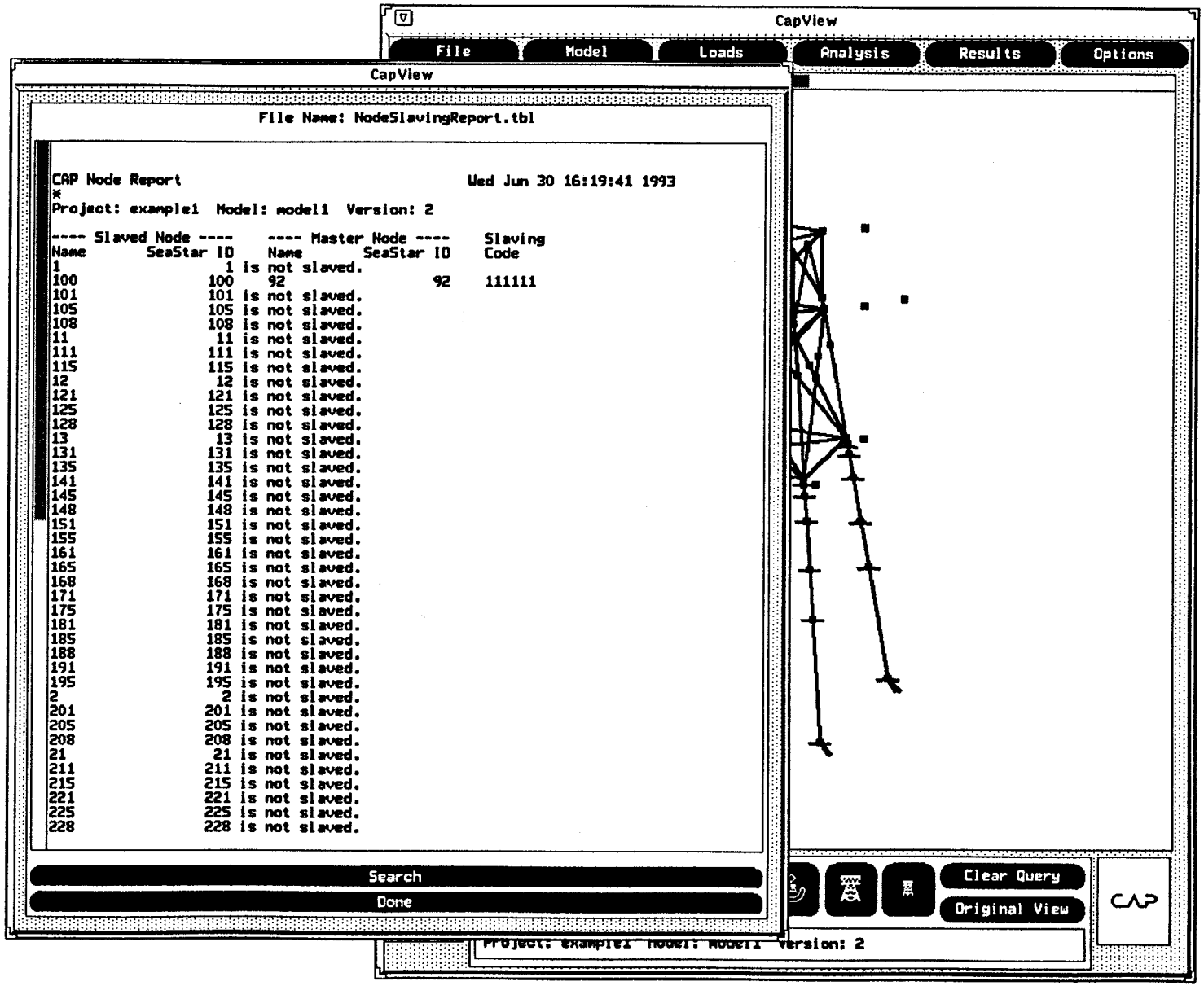

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: NodeSlavingReport.tbl
CAP Node Report
Wed Jun 30 16:19:41 1993
*
Project: example1 Model: model1 Version: 2
---- Slaved Node ---- ---- Master Node ---- Slaving
Name SeaStar ID Name SeaStar ID Code
1 1 is not slaved.
100 100 92 92 111111
101 101 is not slaved.
105 105 is not slaved.
108 108 is not slaved.
11 11 is not slaved.
111 111 is not slaved.
115 115 is not slaved.
12 12 is not slaved.
121 121 is not slaved.
125 125 is not slaved.
128 128 is not slaved.
13 13 is not slaved.
131 131 is not slaved.
135 135 is not slaved.
141 141 is not slaved.
145 145 is not slaved.
148 148 is not slaved.
151 151 is not slaved.
155 155 is not slaved.
161 161 is not slaved.
165 165 is not slaved.
168 168 is not slaved.
171 171 is not slaved.
175 175 is not slaved.
181 181 is not slaved.
185 185 is not slaved.
188 188 is not slaved.
191 191 is not slaved.
195 195 is not slaved.
2 2 is not slaved.
201 201 is not slaved.
205 205 is not slaved.
208 208 is not slaved.
21 21 is not slaved.
211 211 is not slaved.
215 215 is not slaved.
221 221 is not slaved.
225 225 is not slaved.
228 228 is not slaved.
Search
Done
Clear Query
Original View
CAP

Figure 8-63

CapView

File | Model | Loads | Analysis | Results | Options

CapView

File Name: ConnectivityReport.tbl

CAP Member Report Mon Jun 28 14:21:37 1993

*

Project: example1 Model: model1 Version: 1

Member Name	SeaStar ID	Group ID	Element Type	Node I Name	Node I ID	Node J Name	Node J ID	Node K Name	Node K ID
g1-101	101	1	Linear Beam	31	31	51	51	35	35
g1-102	102	1	Linear Beam	51	51	71	71	55	55
g1-103	103	1	Linear Beam	71	71	91	91	55	55
g1-104	104	1	Linear Beam	91	91	111	111	55	55
g1-105	105	1	Linear Beam	111	111	131	131	55	55
g1-106	106	1	Linear Beam	131	131	151	151	55	55
g1-107	107	1	Linear Beam	151	151	171	171	55	55
g1-108	108	1	Linear Beam	171	171	191	191	55	55
g1-109	109	1	Linear Beam	191	191	211	211	55	55
g1-110	110	1	Linear Beam	211	211	231	231	55	55
g10-1001	1001	10	Linear Beam	21	21	23	23	11	11
g10-1002	1002	10	Linear Beam	23	23	25	25	11	11
g10-1003	1003	10	Linear Beam	31	31	33	33	11	11
g10-1004	1004	10	Linear Beam	33	33	35	35	11	11
g10-1005	1005	10	Linear Beam	21	21	27	27	11	11
g10-1006	1006	10	Linear Beam	27	27	31	31	11	11
g10-1007	1007	10	Linear Beam	23	23	33	33	11	11
g10-1008	1008	10	Linear Beam	25	25	29	29	11	11
g10-1009	1009	10	Linear Beam	29	29	35	35	11	11
g10-1010	1010	10	Linear Beam	21	21	33	33	11	11
g10-1011	1011	10	Linear Beam	23	23	31	31	11	11
g10-1012	1012	10	Linear Beam	23	23	35	35	11	11
g10-1013	1013	10	Linear Beam	25	25	33	33	11	11
g11-1101	1101	11	Linear Beam	97	97	98	98	99	99
g11-1102	1102	11	Linear Beam	99	99	100	100	98	98
g11-1103	1103	11	Linear Beam	97	97	99	99	98	98
g11-1104	1104	11	Linear Beam	98	98	100	100	99	99
g11-1105	1105	11	Linear Beam	71	71	35	35	75	75
g2-201	201	2	Linear Beam	35	35	55	55	31	31
g2-202	202	2	Linear Beam	55	55	75	75	51	51
g2-203	203	2	Linear Beam	75	75	95	95	51	51
g2-204	204	2	Linear Beam	95	95	115	115	51	51
g2-205	205	2	Linear Beam	115	115	135	135	51	51
g2-206	206	2	Linear Beam	135	135	155	155	51	51
g2-207	207	2	Linear Beam	155	155	175	175	51	51
g2-208	208	2	Linear Beam	175	175	195	195	51	51
g2-209	209	2	Linear Beam	195	195	215	215	51	51
g2-210	210	2	Linear Beam	215	215	235	235	51	51
g3-301	301	3	Linear Beam	21	21	41	41	25	25

Search

Done

Clear Query

Original View

CAP

Project: example1 Model: model1 Version: 1

Figure 8-64

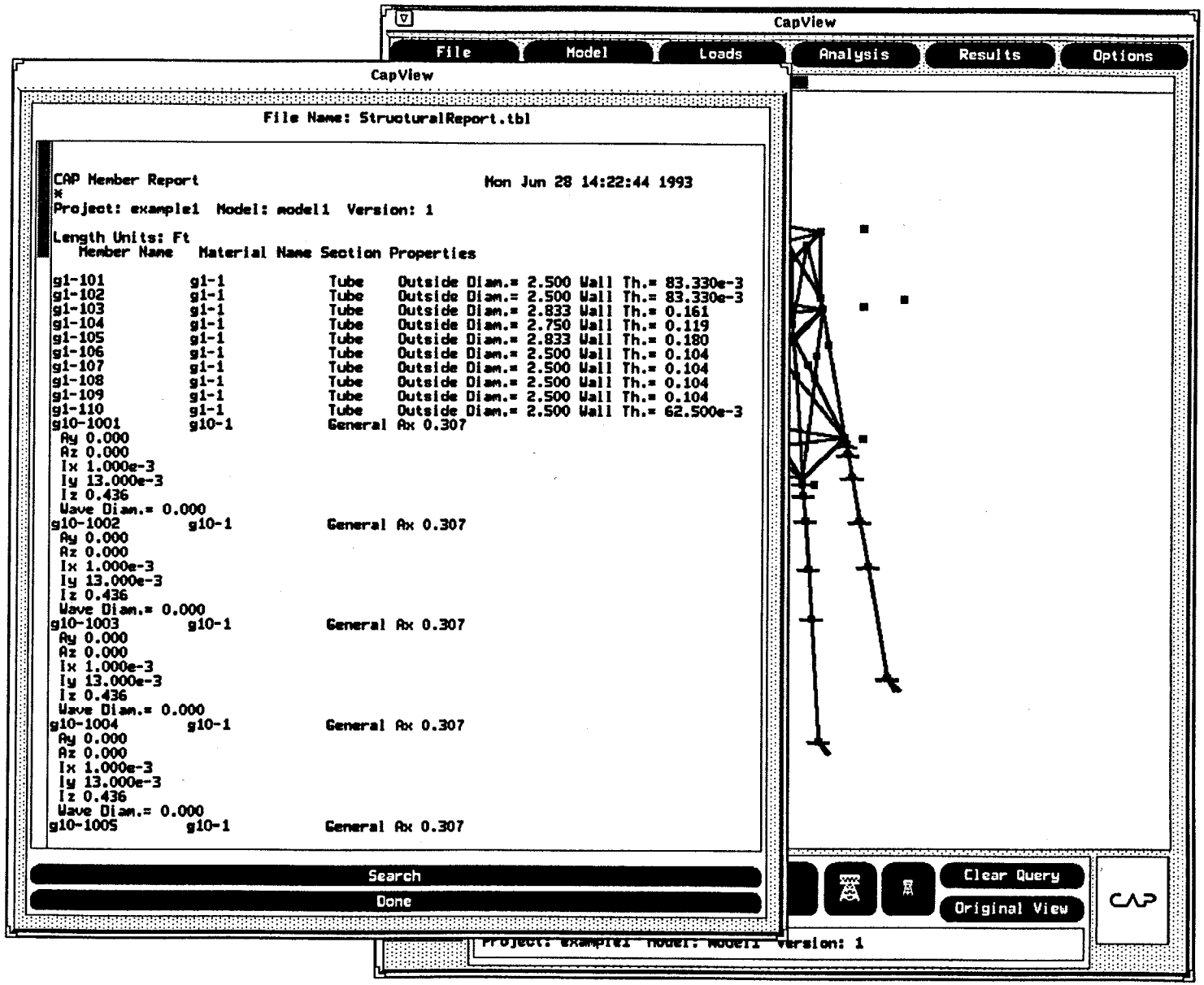

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: StructuralReport.tbl
CAP Member Report
Mon Jun 28 14:22:44 1993
*
Project: example1 Model: model1 Version: 1
Length Units: Ft
Member Name Material Name Section Properties
g1-101 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 83.330e-3
g1-102 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 83.330e-3
g1-103 g1-1 Tube Outside Diam.= 2.833 Wall Th.= 0.161
g1-104 g1-1 Tube Outside Diam.= 2.750 Wall Th.= 0.119
g1-105 g1-1 Tube Outside Diam.= 2.833 Wall Th.= 0.180
g1-106 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 0.104
g1-107 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 0.104
g1-108 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 0.104
g1-109 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 0.104
g1-110 g1-1 Tube Outside Diam.= 2.500 Wall Th.= 62.500e-3
g10-1001 g10-1 General Ax 0.307
Ay 0.000
Az 0.000
Ix 1.000e-3
Iy 13.000e-3
Iz 0.436
Wave Diam.= 0.000
g10-1002 g10-1 General Ax 0.307
Ay 0.000
Az 0.000
Ix 1.000e-3
Iy 13.000e-3
Iz 0.436
Wave Diam.= 0.000
g10-1003 g10-1 General Ax 0.307
Ay 0.000
Az 0.000
Ix 1.000e-3
Iy 13.000e-3
Iz 0.436
Wave Diam.= 0.000
g10-1004 g10-1 General Ax 0.307
Ay 0.000
Az 0.000
Ix 1.000e-3
Iy 13.000e-3
Iz 0.436
Wave Diam.= 0.000
g10-1005 g10-1 General Ax 0.307
Search
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-65

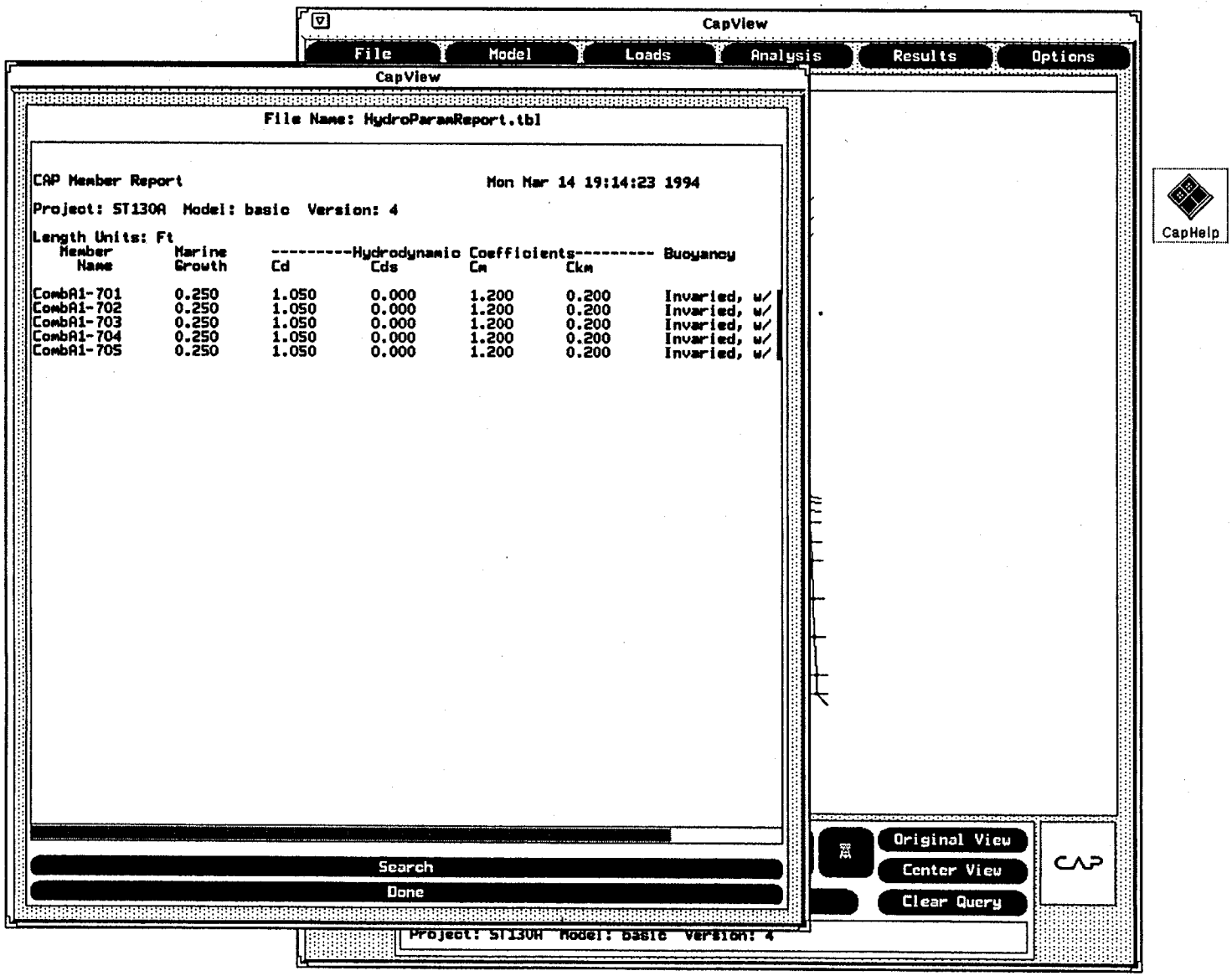

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: HydroParamReport.tbl
CAP Member Report
Mon Mar 14 19:14:23 1994
Project: ST130A Model: basic Version: 4
Length Units: Ft
Member Name | Marine Growth | Hydrodynamic Coefficients Cd | Cds | Cm | Ckm | Buoyancy
CombA1-701 0.250 1.050 0.000 1.200 0.200 Invaried, w/
CombA1-702 0.250 1.050 0.000 1.200 0.200 Invaried, w/
CombA1-703 0.250 1.050 0.000 1.200 0.200 Invaried, w/
CombA1-704 0.250 1.050 0.000 1.200 0.200 Invaried, w/
CombA1-705 0.250 1.050 0.000 1.200 0.200 Invaried, w/
Search
Done
Original View
Center View
Clear Query
CapHelp
Project: ST130A Model: basic Version: 4

Figure 8-66

CapView

File | Model | Loads | Analysis | Results | Options

CapView

File Name: CanStubParamReport.tbl

CAP Member Report Mon Jun 28 14:27:49 1993

*

Project: example1 Model: model1 Version: 1

Length Units: Ft Stress Units: ksf

Member Name	------End I Eccentricities------			------End J Eccentricities------			End Releases	
	Ix	Iy	Iz	Jx	Jy	Jz	End I	EndJ
g1-101	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-102	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-103	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-104	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-105	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-106	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-107	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-108	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-109	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g1-110	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1001	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1002	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1003	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1004	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1005	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1006	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1007	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1008	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1009	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1010	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1011	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1012	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g10-1013	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g11-1101	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g11-1102	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g11-1103	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g11-1104	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g11-1105	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-201	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-202	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-203	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-204	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-205	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-206	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-207	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000
g2-208	0.000	0.000	0.000	0.000	0.000	0.000	000000	000000

Search

Done

Clear Query

Original View

CAP

Project: example1 Model: model1 Version: 1

Figure 8-69

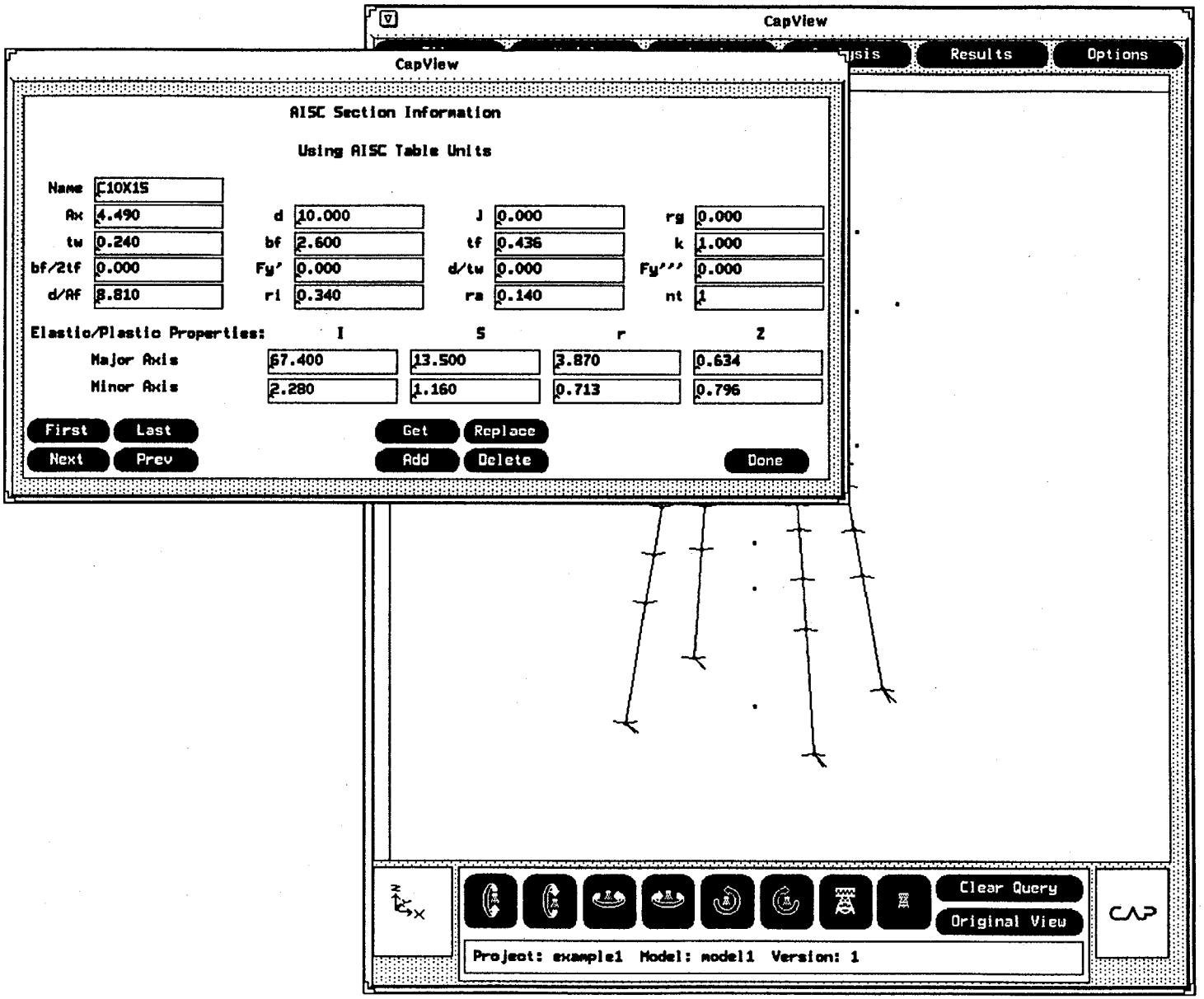

CapView
Results
Options
CapView
AISC Section Information
Using AISC Table Units
Name C10X15
Ax 4.490
d 10.000
J 0.000
rg 0.000
tw 0.240
bf 2.600
tf 0.436
k 1.000
bf/2tf 0.000
Fy' 0.000
d/tw 0.000
Fy''' 0.000
d/Af 8.810
ri 0.340
ra 0.140
nt 1
Elastic/Plastic Properties:
I
S
r
Z
Major Axis 67.400 13.500 3.870 0.634
Minor Axis 2.280 1.160 0.713 0.796
First
Last
Next
Prev
Get
Replace
Add
Delete
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 8-70

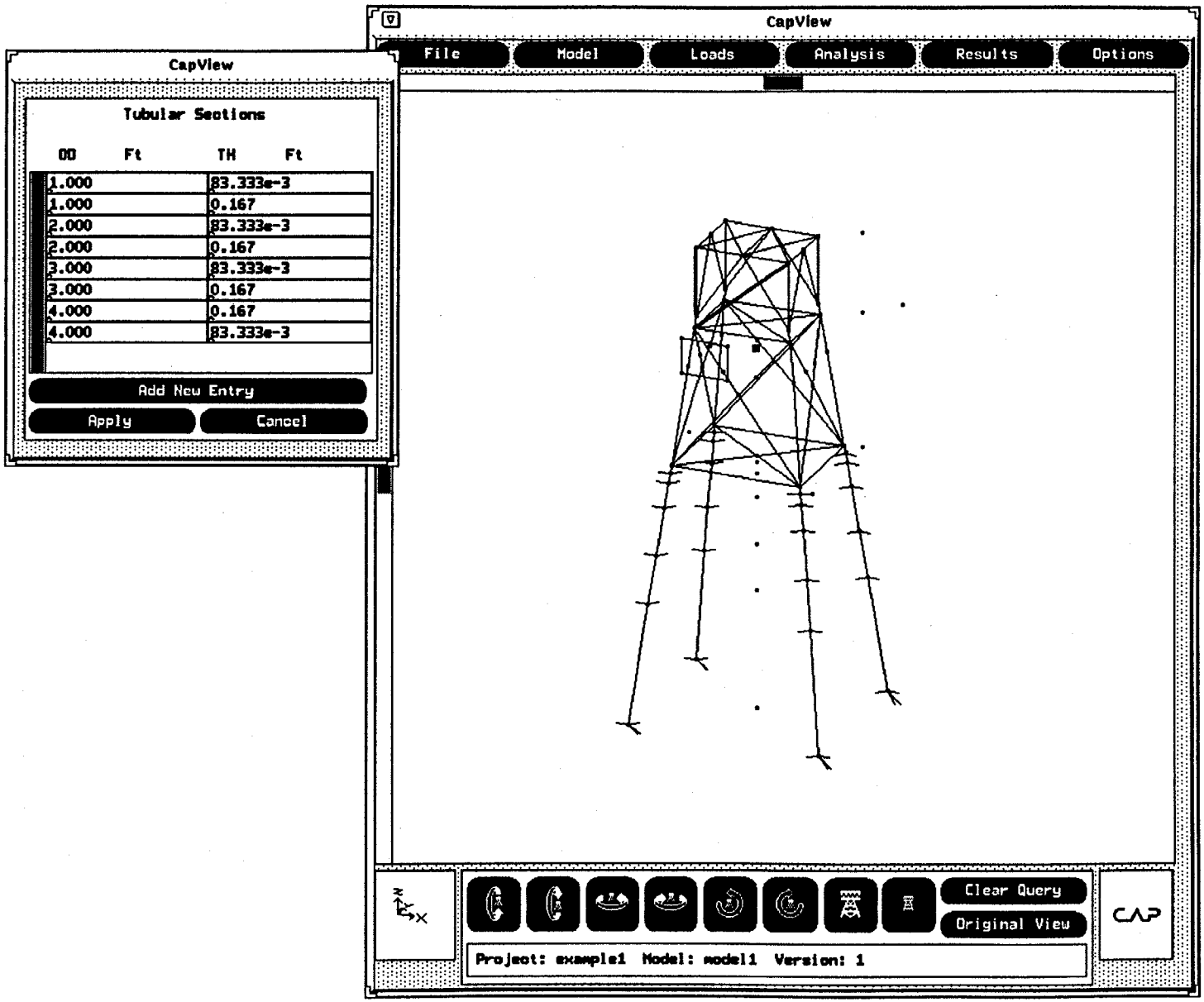

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Tubular Sections
OD Ft TH Ft
1.000 83.333e-3
1.000 0.167
2.000 83.333e-3
2.000 0.167
3.000 83.333e-3
3.000 0.167
4.000 0.167
4.000 83.333e-3
Add New Entry
Apply
Cancel
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 8-71

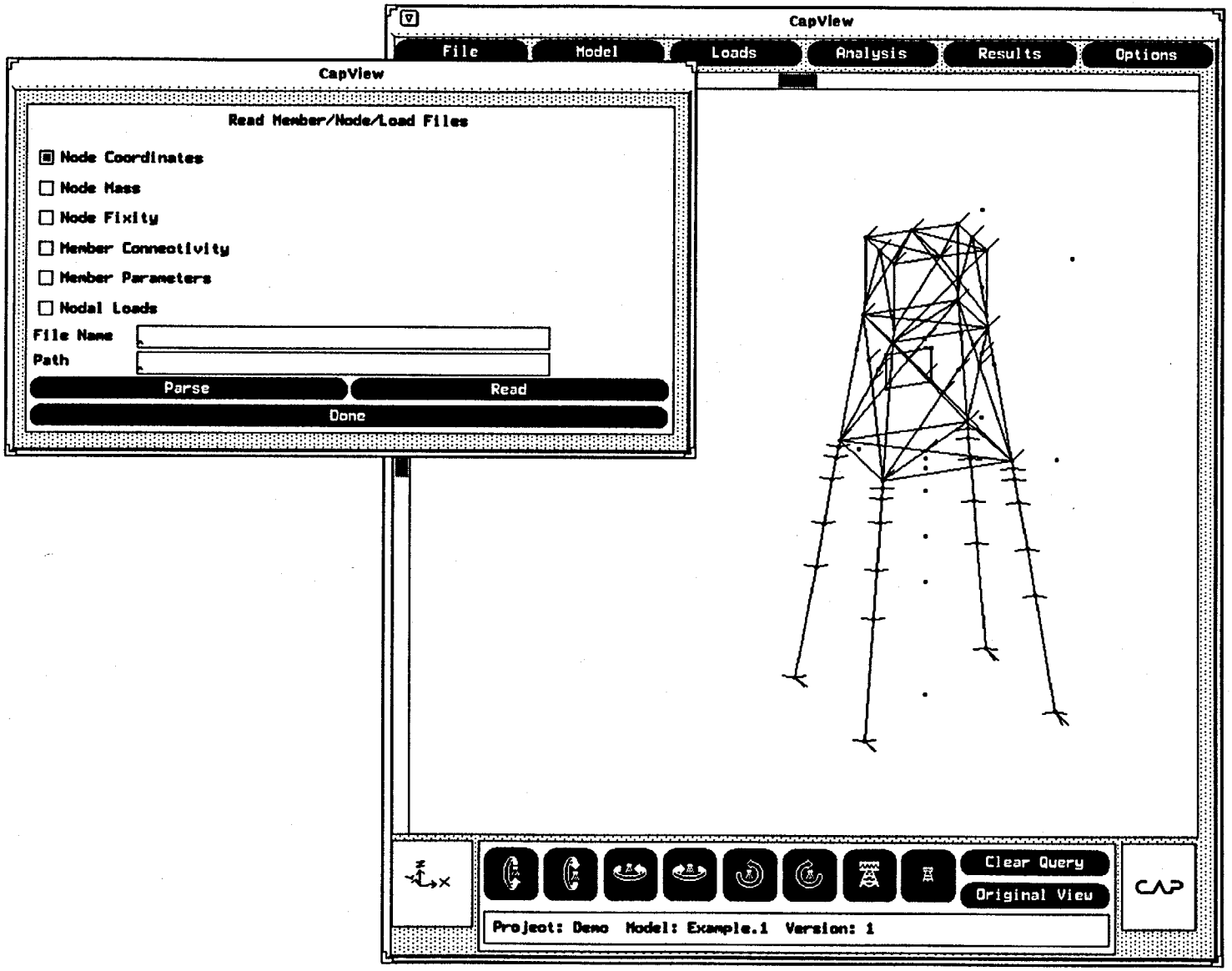

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Read Member/Node/Load Files
Node Coordinates
Node Mass
Node Fixity
Member Connectivity
Member Parameters
Nodal Loads
File Name
Path
Parse
Read
Done
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 8-72

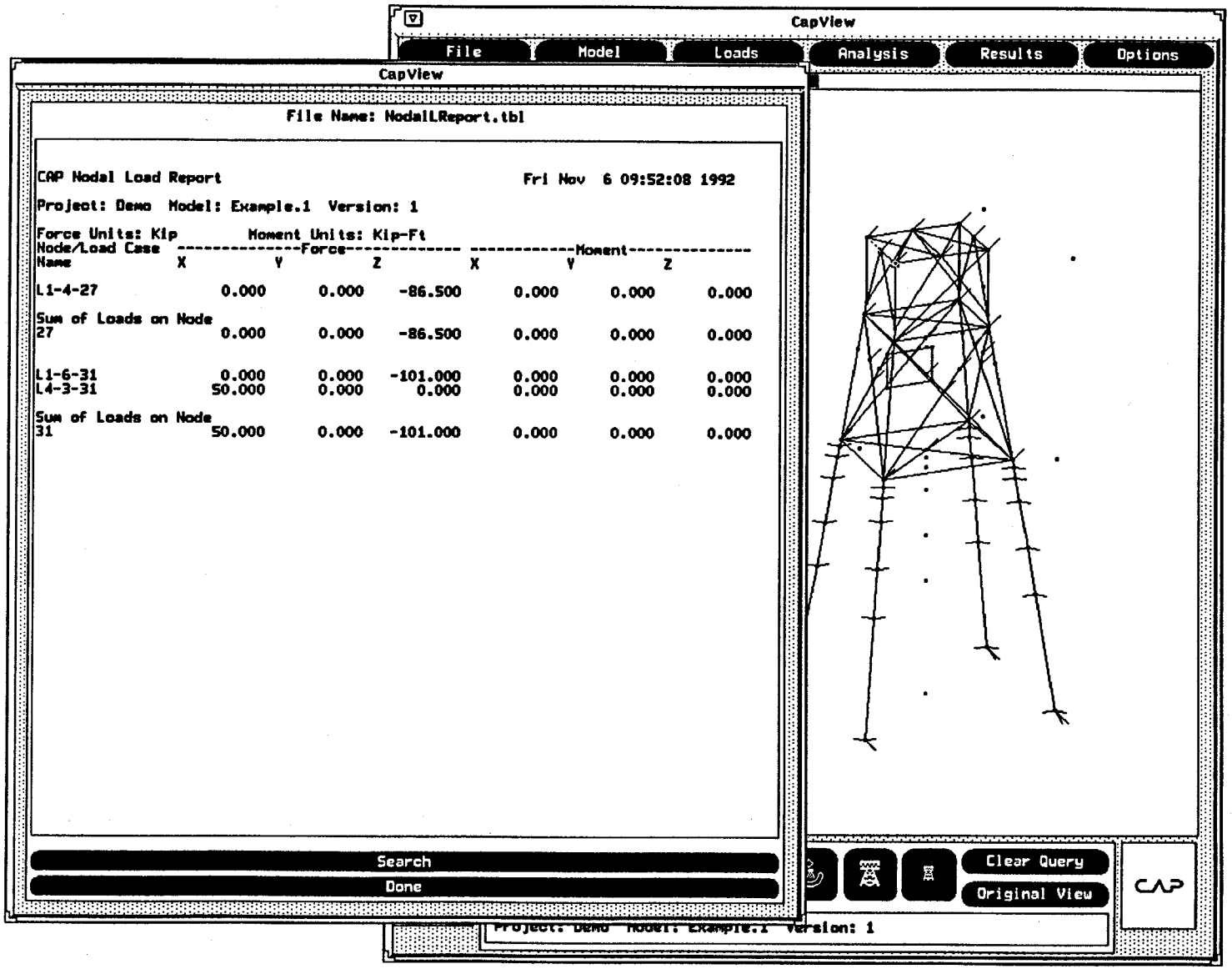

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: NodalLReport.tbl
CAP Nodal Load Report
Fri Nov 6 09:52:08 1992
Project: Demo Model: Example.1 Version: 1
Force Units: Kip
Moment Units: Kip-Ft
Node/Load Case
Force
Moment
Name X Y Z X Y Z
L1-4-27 0.000 0.000 -86.500 0.000 0.000 0.000
Sum of Loads on Node 27 0.000 0.000 -86.500 0.000 0.000 0.000
L1-6-31 0.000 0.000 -101.000 0.000 0.000 0.000
L4-3-31 50.000 0.000 0.000 0.000 0.000 0.000
Sum of Loads on Node 31 50.000 0.000 -101.000 0.000 0.000 0.000
Search
Done
Clear Query
Original View
CAP
Project: Demo Model: Example.1 Version: 1

Figure 8-73

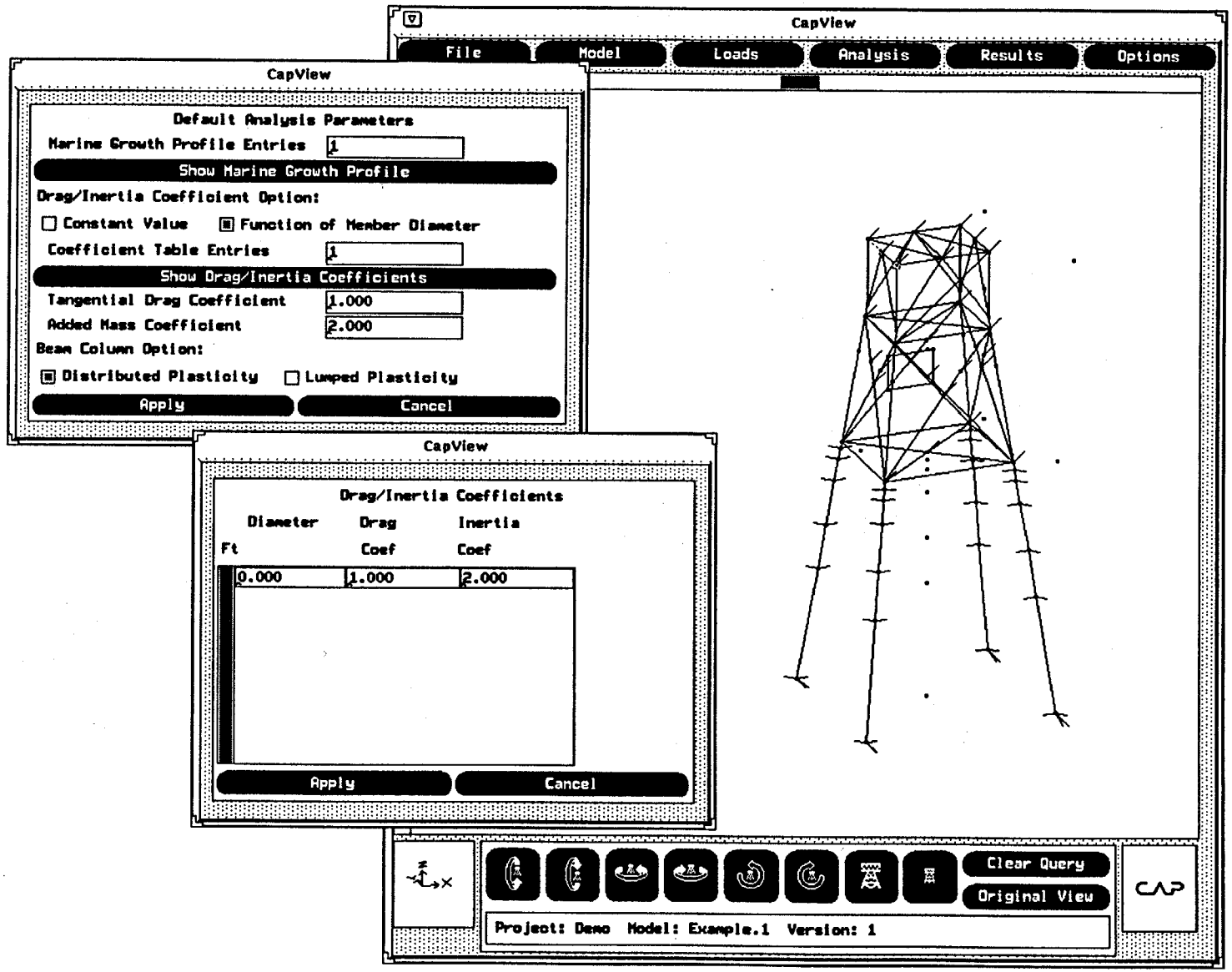

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Default Analysis Parameters
Marine Growth Profile Entries
1
Show Marine Growth Profile
Drag/Inertia Coefficient Option:
Constant Value
Function of Member Diameter
Coefficient Table Entries
1
Show Drag/Inertia Coefficients
Tangential Drag Coefficient
1.000
Added Mass Coefficient
2.000
Beam Column Option:
Distributed Plasticity
Lumped Plasticity
Apply
Cancel
CapView
Drag/Inertia Coefficients
Diameter
Ft
Drag
Coef
Inertia
Coef
0.000
1.000
2.000
Apply
Cancel
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 8-74

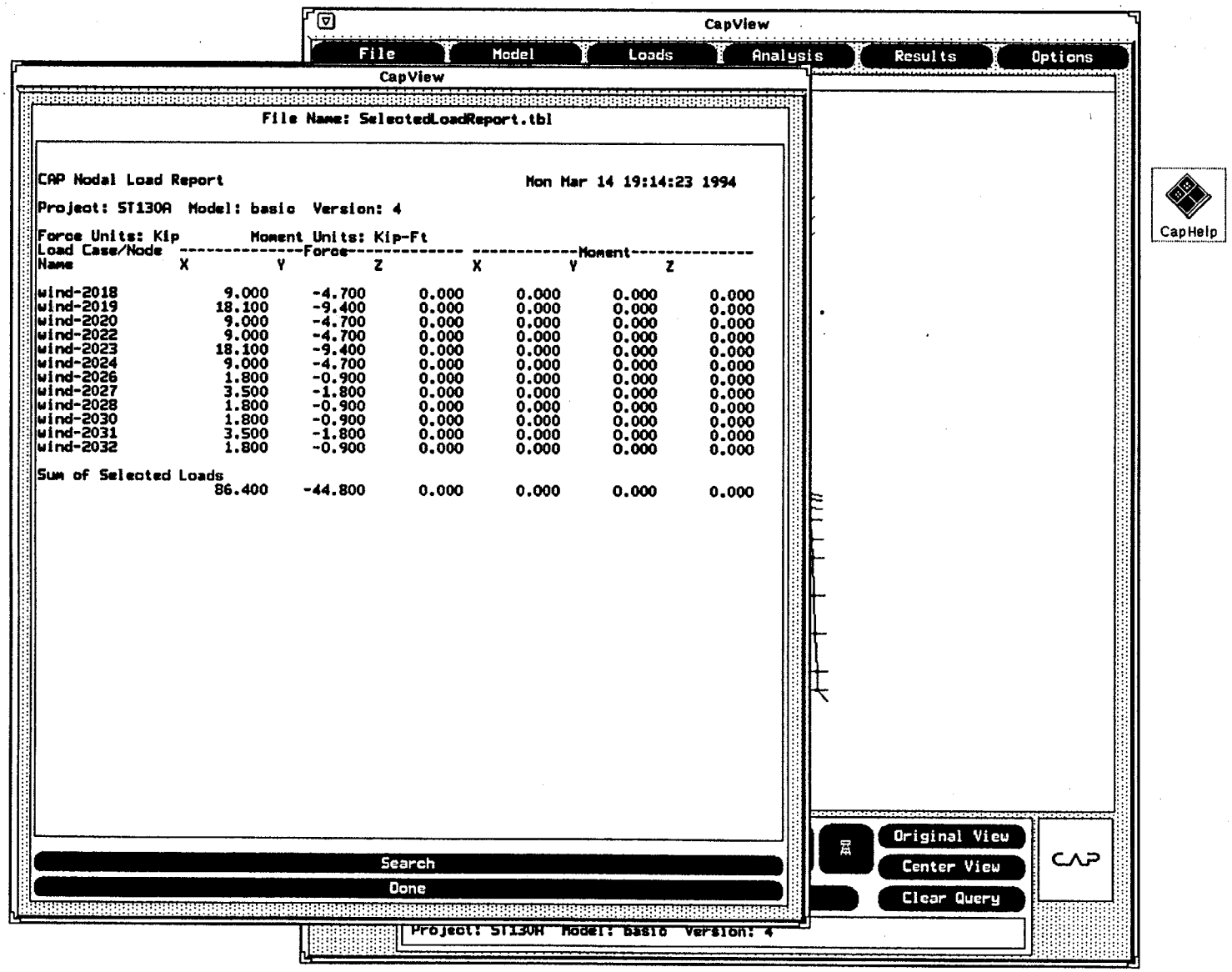

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
File Name: SelectedLoadReport.tbl
CAP Nodal Load Report
Mon Mar 14 19:14:23 1994
Project: ST130A Model: basic Version: 4
Force Units: Kip
Moment Units: Kip-Ft
Load Case/Node
Force
Moment
Name X Y Z X Y Z
wind-2018 9.000 -4.700 0.000 0.000 0.000 0.000
wind-2019 18.100 -9.400 0.000 0.000 0.000 0.000
wind-2020 9.000 -4.700 0.000 0.000 0.000 0.000
wind-2022 9.000 -4.700 0.000 0.000 0.000 0.000
wind-2023 18.100 -9.400 0.000 0.000 0.000 0.000
wind-2024 9.000 -4.700 0.000 0.000 0.000 0.000
wind-2026 1.800 -0.900 0.000 0.000 0.000 0.000
wind-2027 3.500 -1.800 0.000 0.000 0.000 0.000
wind-2028 1.800 -0.900 0.000 0.000 0.000 0.000
wind-2030 1.800 -0.900 0.000 0.000 0.000 0.000
wind-2031 3.500 -1.800 0.000 0.000 0.000 0.000
wind-2032 1.800 -0.900 0.000 0.000 0.000 0.000
Sum of Selected Loads
86.400 -44.800 0.000 0.000 0.000 0.000
Search
Done
Original View
Center View
Clear Query
CAP
Project: ST130A Model: basic Version: 4

Figure 8-75

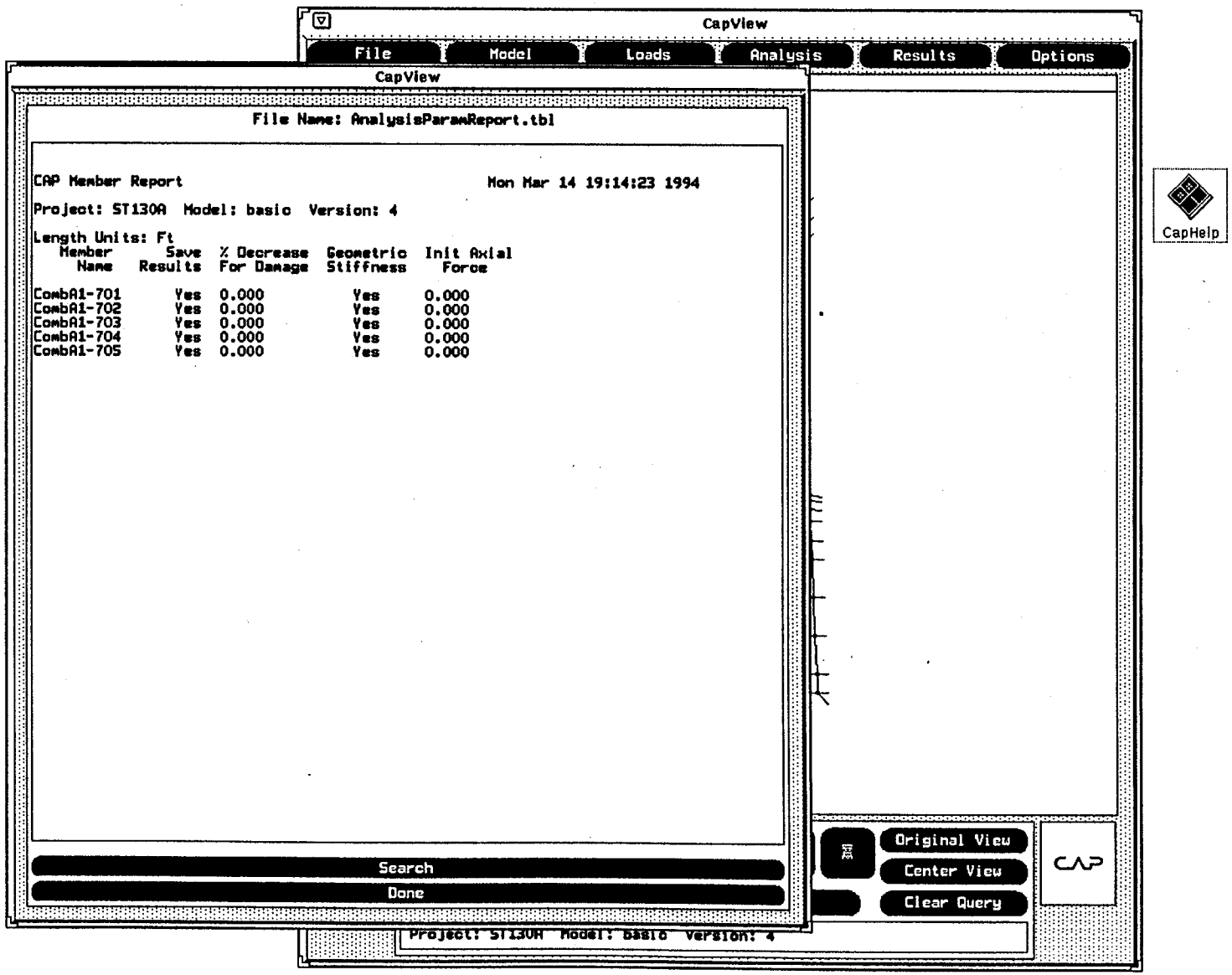

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: AnalysisParamReport.tbl
CAP Member Report
Mon Mar 14 19:14:23 1994
Project: ST130A Model: basic Version: 4
Length Units: Ft
Member Name | Save Results | % Decrease For Damage | Geometric Stiffness | Init Axial Force
CombA1-701 Yes 0.000 Yes 0.000
CombA1-702 Yes 0.000 Yes 0.000
CombA1-703 Yes 0.000 Yes 0.000
CombA1-704 Yes 0.000 Yes 0.000
CombA1-705 Yes 0.000 Yes 0.000
Search
Done
Original View
Center View
Clear Query
CapHelp
Project: ST130A Model: basic Version: 4

Figure 8-76

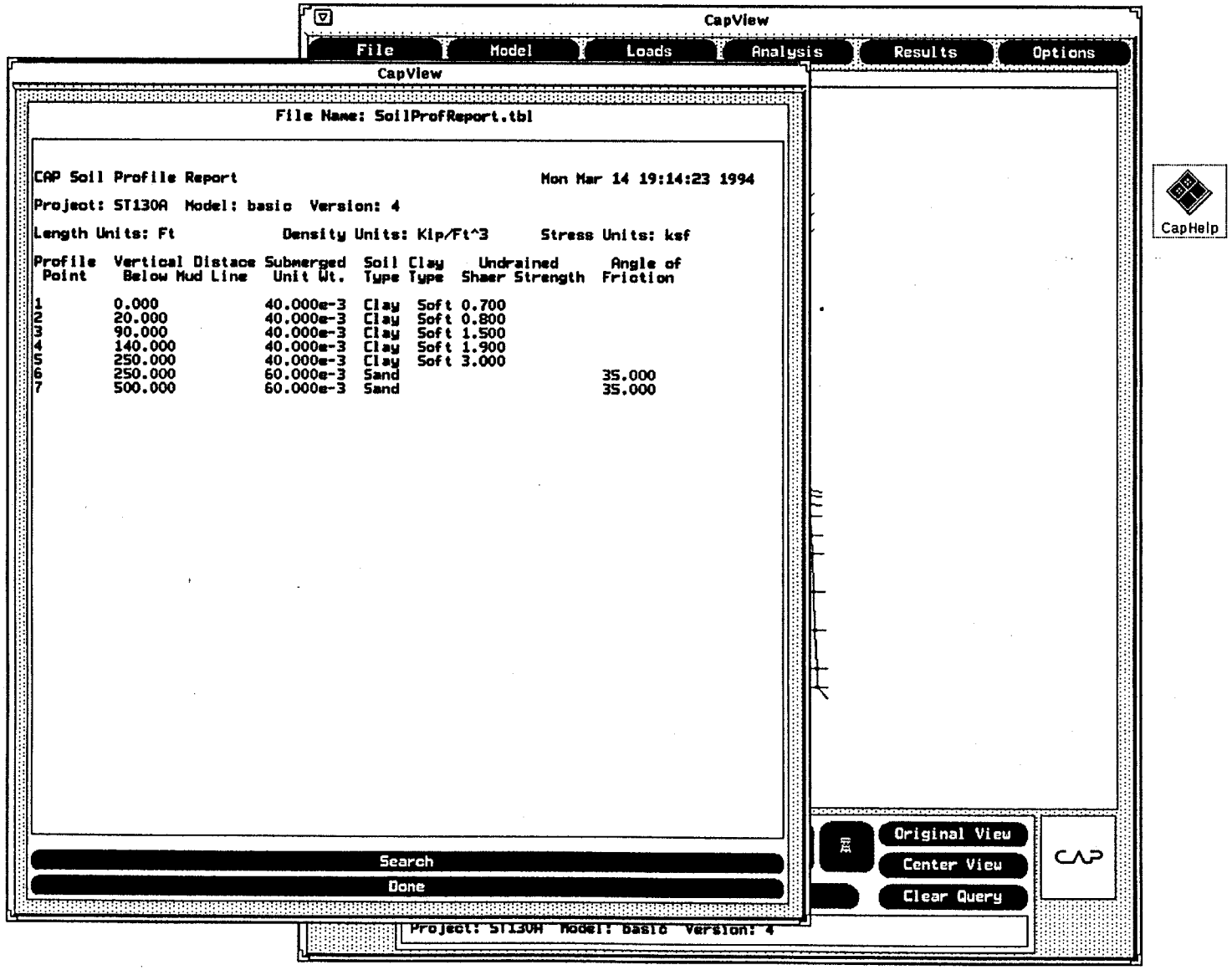

CapView
File
Model
Loads
Analysis
Results
Options
CapView
File Name: SoilProfReport.tbl
CAP Soil Profile Report
Mon Mar 14 19:14:23 1994
Project: ST130A Model: basic Version: 4
Length Units: Ft
Density Units: Kip/Ft^3
Stress Units: ksf
Profile Point
Vertical Distace Below Mud Line
Submerged Unit Wt.
Soil Type
Clay Type
Undrained Shaer Strength
Angle of Friction
1 0.000 40.000e-3 Clay Soft 0.700
2 20.000 40.000e-3 Clay Soft 0.800
3 90.000 40.000e-3 Clay Soft 1.500
4 140.000 40.000e-3 Clay Soft 1.900
5 250.000 40.000e-3 Clay Soft 3.000
6 250.000 60.000e-3 Sand 35.000
7 500.000 60.000e-3 Sand 35.000
Search
Done
Original View
Center View
Clear Query
CapHelp
Project: ST130A Model: basic Version: 4

Figure 8-77

9.0 THE LOADS MENU

The Loads menu presents the user with selections that will help in adding and editing loads. When the Loads menu is selected, the following menu is displayed (Figure 9-1):

Add Nodal Load...
Add Nodal Displacement...
Add Pushover Load
Nodal Load Select/Change...
Nodal Displacement Select/Change...
Show Selected Nodal Loads
Show Selected Nodal Displacements
Wave Load Control...
Regular Wave...
Irregular Sea...
Current Load...
Ground Motion...

9.1 Add Nodal Load

The first selection of the Loads menu will display the Add Nodal Loads dialogue box (Figure 9-2). This dialogue box allows the user to add nodal loads.

The dialogue box has the following:

- A button labeled Load Case followed by an edit field. This field is used by CAP to determine to which load case the new nodal load will be added. When the button is pressed CAP will try to get information about a load case that has a name similar to that in the edit field (Figure 9-3).
- Three edit fields representing the X, Y, and Z components of the load to be added.
- Three edit fields representing the X, Y, and Z components of the moment to be added.
- An instructions box. This box displays instructions on keys to press to apply a load to a node and to undo it.
- A button labeled "Done", that if pressed, will close the Add Nodal Load dialogue box.

Note that a node may only have one load on it for each load case. Thus if a node already has a load on it and you want to add to it, you must either add the new load as part of a new load case or you must edit the load value already on the node by clicking on the load object with the left mouse button and add the appropriate number to it yourself.

9.2 Add Nodal Displacement

The second selection of the Loads menu will display the Add Nodal Displacements dialogue box (Figure 9-4). This dialogue box allows the user to add nodal displacements.

The dialogue box has the following:

- A button labeled Displacement Case followed by an edit field. This field is used by CAP to determine to which displacement case the new nodal displacement will be added. When the button is pressed CAP will try to get information about a displacement case that has a name similar to that in the edit field (figure 9-5).
- Three edit fields representing the X, Y, and Z components of the translation to be added.
- Three edit fields representing the X, Y, and Z components of the rotation to be added.
- An instructions box. This box displays instructions on keys to press to apply a displacement to a node and to undo it.
- A button labeled "Done", that if pressed, will close the Add Nodal Displacement dialogue box.

Note that a node may only have one displacement on it for each displacement case. Thus if a node already has a displacement on it and you want to add to it, you must either add the new displacement as part of a new displacement case or you must edit the displacement value already on the node by clicking on the displacement object with the left mouse button and add the appropriate number to it yourself.

9.3 Add Pushover Load

The third selection of the Loads menu will display the Add Pushover Load Pattern dialogue box. The user can use this dialogue box to convert a pushover load pattern into a nodal load pattern that can be applied statically or dynamically on the structure to perform static or dynamic pushover analyses. Note that before you can use this dialogue box, you must run a Generate Pushover Load Analysis (see section 10.1.1).

The dialogue box has the following:

- A button labeled Load Case followed by an edit field. This field is used by CAP to determine to which load case the new pushover nodal load will be added. When the button is pressed CAP will try to get information about a load case that has a name similar to that in the edit field (figure 9-3).

Apply button. Selecting this button will delete any nodal load that belongs to the specified load case and generate a series of nodal loads that represent the pushover load pattern under the specified nodal load case.

Done button. Selecting this button will close the Add Pushover Load Pattern dialogue box.

9.4 Nodal Load Select/Change

The fourth selection of the Loads menu will display the Nodal Load Select/Change dialogue box (Figure 9-6). This dialogue box allows the user to select specific nodal loads, change nodal load attributes or delete nodal loads. The user can specify a single query or group of queries that can be logically combined for complex searches.

The dialogue box has two criteria sections. The upper section of the box contains criteria that can be searched for or changed. The second section controls select-only criteria - criteria that can be the basis for a search but not for a change operation.

Below the criteria sections is the values section. In this part of the dialogue box the user specifies the values used for the search, or the values that will be assigned to the nodal loads in a change operation. The two check boxes below the values section control logical functions used in selecting and changing.

In addition to these sections, there is a button at the bottom of the dialogue box labeled "Done", that if pressed, will close the Nodal Load Select/Change dialogue box.

Note that when this dialogue box is active the rest of CAP is still active, allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

9.4.1 Logical Operations

A logical "OR" function is achieved by running a second (or further) query with the "Clear Previous Selections" check box not selected. CAP will add those nodal loads to the ones found in a previous search.

The "Exclude Matching Node / Apply Changes to Not Selected Nodes" flag implements a logical "NOT" function.

CAP implements a logical "AND" function by allowing the user to select nodal loads meeting several criteria in a single query. To use this feature, choose several search criteria for one search (i.e. location and slaving) and CAP will select only those nodal loads that match all of the criteria.

9.4.2 Selecting Nodal Loads

A nodal load can be selected based on its:

- Force Values
- Moment Values
- Load Visibility
- Node Name
- Load Case Name

All but the last two can also be change criteria. Once a user chooses the criteria for the selection, he can then choose values for the criteria.

Force and Moment Values can be specified for any combination of X, Y and Z. The changes may be absolute or relative. If they are absolute the new values specified become the new values for the selected nodes. If they are relative the values specified are added to the current values of the selected nodes.

Load Visibility options are hidden or shown.

Node Names can be entered or picked from a list.

Load Case Names can be entered or picked from a list.

Once the criteria values are chosen, clicking on the select button will make the selections, and nodal loads which satisfy the selection criteria will be selected.

9.4.3 Changing Nodal Loads

When you finish selecting the nodal loads desired, changing them is accomplished by changing criteria values and clicking the change button. First, choose a criteria to change. The criteria changed does not have to be the same as that used in the select operation. Second, change the values for this criteria in the values section. Third, click on the change button. The selected nodal loads now have the new values.

9.4.4 Deleting Nodal Loads

To delete selected nodal loads, click on the Delete button. The CapView undo command (<Control> Right Mouse Button) works for undoing delete operations.

9.5 Nodal Displacement Constraint Select/Change

The fifth selection of the Loads menu will display the Nodal Displacement Constraint Select/Change dialogue box (Figure 9-7). This dialogue box allows the user to select specific nodal displacement constraints, change nodal displacement constraint attributes or delete nodal displacement constraints. The user can specify a single query or group of queries that can be logically combined for complex searches.

The dialogue box has two criteria sections. The upper section of the box contains criteria that can be searched for or changed. The second section controls select-only criteria - criteria that can be the basis for a search but not for a change operation.

Below the criteria sections is the values section. In this part of the dialogue box the user specifies the values used for the search, or the values that will be assigned to the nodal displacement constraints in a change operation. The two check boxes below the values section control logical functions used in selecting and changing.

In addition to these sections, there is a button at the bottom of the dialogue box labeled "Done", that if pressed, will close the Nodal Displacement Constraint Select/Change dialogue box.

Note that when this dialogue box is active the rest of CAP is still active, allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

9.5.1 Logical Operations

A logical "OR" function is achieved by running a second (or further) query with the "Clear Previous Selections" check box not selected. CAP will add those nodal displacement constraints to the ones found in a previous search.

The "Exclude Matching Node / Apply Changes to Not Selected Nodes" flag implements a logical "NOT" function.

CAP implements a logical "AND" function by allowing the user to select nodal displacement constraints meeting several criteria in a single query. To use this feature, choose several search criteria for one search (i.e. location and slaving) and CAP will select only those nodal displacement constraints that match all of the criteria.

9.5.2 Selecting Nodal Displacement Constraints

A nodal displacement constraint can be selected based on its:

- Translation Values
- Rotation Values
- Constraint Visibility
- Node Name
- Disp. Case Name

All but the last two can also be change criteria. Once a user chooses the criteria for the selection, he can then choose values for the criteria.

Translation and Rotation Values can be specified for any combination of X, Y and Z. The changes may be absolute or relative. If they are absolute the new values specified become the new values for the selected nodes. If they are relative the values specified are added to the current values of the selected nodes.

Constraint Visibility options are hidden or shown.

Node Names can be entered or picked from a list.

Disp. Case Names can be entered or picked from a list.

Once the criteria values are chosen, clicking on the select button will make the selections, and nodal displacement constraints which satisfy the selection criteria will be selected.

9.5.3 Changing Nodal Displacement Constraints

When you finish selecting the nodal displacement constraints desired, changing them is accomplished by changing criteria values and clicking the change button. First, choose a criteria to change. The criteria changed does not have to be the same as that used in the select operation. Second, change the values for this criteria in the values section. Third, click on the change button. The selected nodal displacement constraints now have the new values.

9.5.4 Deleting Nodal Displacement Constraints

To delete selected nodal displacement constraints, click on the Delete button. The CapView undo command (<Control> Right Mouse Button) works for undoing delete operations.

9.6 Show Selected Nodal Loads

The sixth selection of the Load menu will display the print file browser dialogue box labeled Show Selected Nodal Loads (Figure 9-34). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file with a list of selected nodal loads. If there are no selected nodal loads an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

9.7 Show Selected Nodal Displacements

The seventh selection of the Load menu will display the print file browser dialogue box labeled Show Selected Nodal Displacements (Figure 9-35). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file with a list of selected nodal displacements. If there are no selected nodal displacements an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

9.8 Wave Load Control

The eighth selection of the Loads menu will display the Wave Load Control dialogue box (Figure 9-8). This dialogue box allows the user to select and control the wave loads.

The dialogue box consists of the following:

A set of radio buttons labeled "Wave Load Type", which are used to choose between:

- Regular Wave
- Irregular Sea
- Current Only

A checkbox to set whether or not CapView should "Apply Current Load". If Wave Load Type is Current Only, then the status of this checkbox is not relevent.

A set of radio buttons labeled "Wave Kinetics Computation", which are used to choose between:

- Undeformed Position, where wave kinematics are computed at undeformed position.
- Deformed Position, where wave kinematics are computed at deformed position.

A set of radio buttons labeled "Drag Force Velocity", which are used to choose between:

- Absolute, where the velocity of the structure is ignored in drag force computations
- Relative, where the velocity of the structure is included in drag force computations.

A checkbox to set whether or not CapView should "Include Variable Added Mass".

An edit field to enter the "Ramp Time" value for the time over which to build up to the maximum kinematics.

An edit field to enter the "Kinematic Scale Factor" value for the wave loads.

Apply button. Selecting this button will apply the selections currently displayed and close the Wave Load Control dialogue box. This button must be selected to make any changes effective.

Cancel button. Selecting this button will close the Wave Load Control dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.9 Regular Wave Data

The ninth selection of the loads menu will display the Regular Wave Data dialogue box (Figure 9-9). This dialogue box allows the user to describe regular waves.

The dialogue box has the following:

- An edit field labeled "Height" that is used to set the height of the regular wave.
- An edit field labeled "Period" that is used to set the period of the regular wave.
- An edit field labeled "Distance Between Crest and Origin" that is used to set the distance between the crest of the wave and the origin of the structure's coordinate system. In a dynamic analysis you may want to place the wave in such a position that at the beginning of the analysis the structure is not suddenly hit with the maximum possible wave force, so you minimize the amount of time required to achieve steady static ??? to do that.
- An edit field labeled "Still Water Depth" that is used to set the still water depth of the regular wave.
- An edit field labeled "Cut-off Elevation above MWL" that is used to set the cut-off elevation above MWL for calculation of wave forces.
- An edit field labeled "Cut-off Elevation below MWL" that is used to set the cut-off elevation below MWL for calculation of wave forces. For deep structures where the wave forces would be small you can set this above the mudline to prevent computation of wave forces in the lower members and thereby reduce computation times.
- An edit field labeled "Tolerance for Wave Length Computation" that is used to set the tolerance for the wave length computation of the regular wave.
- An edit field labeled "Max Iterations for Wave Length Computation" that is used to set the maximum number of iterations used to compute the wave length computation of the regular wave.
- An edit field labeled "Order for Stream Function" that is used to set the order for the stream function of the regular wave.
- An edit field labeled "Points on Free Surface" that is used to set the number of points on the free surface at which errors are evaluated.
- Three edit fields labeled "Direction Vector" that are used to set the X, Y and Z components of the regular waves direction vector. Which defines the orientation of the wave with respect to the structure's coordinate system.
- A set of radio buttons labeled "Wave Type" that are used to choose between an Airy, Stokes 5th or a Stream Function wave.
- A checkbox used to set whether or not to "Stretch Kinematics". Stretching is recommended for Stream Function and Stoke's 5th order but not for Airy waves.
- A pair of radio buttons that are used to toggle between Automatic Convergence and Set Maximum Iterations. Automatic convergence stops further iterations when the total error (defined as the sum of the absolute values of the mean sea level error, the wave height error, and the RMS dynamic free surface boundary condition error) is less than 0.01 Ft.

- When Set Maximum Iterations is selected, there is an edit field that is used to set the maximum number of iterations.

Apply button. Selecting this button will apply the selections currently displayed and close the Regular Wave Data dialogue box. This button must be selected to make any changes effective.

Cancel button. Selecting this button will close the Regular Wave Data dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.10 Irregular Sea Data

The tenth selection of the Loads menu will display the Irregular Sea Data dialogue box (Figure 9-10). This dialogue box allows the user to describe an irregular sea.

9.10.1 General Parameters

The first part of the dialogue box has the following:

- An edit field labeled "Still Water Depth" that is used to set the still water depth of the irregular sea.
- An edit field labeled "Cut-off Elevation below MWL" that is used to set the cut-off elevation below MWL of wave forces. For deep structures where the wave forces would be small you can set this above the mudline to prevent computation of wave forces in the lower members and thereby reduce computation times.
- A set of radio buttons labeled "Stretched Type" that are used to choose between Unstretched, Stretched, or Vertical Stretching.
- A set of radio buttons labeled "Multiple Components" that are used to choose between One Time Block, First Block, or All But 1st Block. Use one time block only.
- A set of radio buttons labeled "Wave Component Generation" that are used to choose between Wave Spectra, Wave Components, or Multi-Directional Wave Components.
- A set of radio buttons labeled "Wave Height Spectrum Definition" that are used to choose between Pierson-Muscowitz and User Defined. These buttons are only visible when Wave Component Generation is set to Wave Spectra.
- An edit field labeled "Max Iterations for Sea Length Computation" that is used to set the maximum iterations used for wave length computation of the irregular sea.
- An edit field labeled "Tolerance for Sea Length Computation" that is used to set the tolerance for the wave length computation of the irregular sea.

9.10.2 Generation Option - Specific Parameters

9.10.2.1 When Wave Component Generation is set to Wave Spectra and Wave Height Spectrum Definition is set to Pierson-Moscowitz, the second part consists of (Figure 9-10):

- An edit field labeled "Mean Wind Velocity" that is used to set the mean wind velocity of the irregular sea.
- Three edit fields labeled "Wave Direction Vectors" that are used to set the X, Y and Z components of the irregular sea direction vectors which define the orientation of the irregular sea with respect to the structure's coordinate system.

9.10.2.2 When Wave Component Generation is set to Wave Spectra and Wave Height Spectrum Definition is set to User-Defined, the second part consists of (Figure 9-11):

- An edit field labeled "Number of Points Defining Wave Height Specta" that is used to set the number of points that the user will input to define the wave height specta of the irregular sea.
- A button labeled Edit Spectra which when pressed opens a Definition Points dialogue box (Figure 9-12) to define as many points as were set in

the previous field.

The Definition Points dialogue box consists of two scrollable lists of edit fields, one for the Frequency and one for the Height of each point. There is an Apply button to accept the points as set and close the dialogue, and there is a Cancel button to close the dialogue box without using any changes.

- Three edit fields labeled "Wave Direction Vectors" that are used to set the X, Y and Z components of the irregular sea direction vectors which define the orientation of the irregular sea with respect to the structure's coordinate system.

9.10.2.3 When Wave Component Generation is set to Wave Components the second part consists of (Figure 9-13):

- An edit field labeled "Number of Wave Components" that is used to set the number of components that the user will input to define the irregular sea.
- A button labeled "Edit Components" which when pressed opens a Wave Components dialogue box (Figure 9-14) to define as many points as were set in the previous field.

The Wave Components dialogue box consists of three scrollable lists of edit fields, one for the Period, one for the Height, and one for the Phase of each component. Each set of three values is used to define one wave component. There is an Apply button to accept the components as set and close the dialogue, and there is a Cancel button to close the dialogue box without using any changes.

- Three edit fields labeled "Wave Direction Vectors" that are used to set the X, Y and Z components of the irregular sea direction vectors which define the orientation of the irregular sea with respect to the structure's coordinate system.
- A button labeled "Generate Components" which when pressed opens a Generate Irregular Sea Wave Components dialogue box.

9.10.2.3.1 Generate Irregular Sea Wave Components dialogue box (Figure 9-15)

The first part of the Generate Irregular Sea Wave Components dialogue box has the following:

- A pair of edit fields labeled "Max Wave Height From" and "To" that are used to set the range of the maximum wave heights of the irregular sea.
- An edit field labeled "Calculation Time Range" that is used to set the range of time to be generated for the irregular sea.
- An edit field labeled "Time at Max Wave Height" that is used to set the time when the maximum wave height will be generated for the irregular sea.
- An edit field labeled "Calculation Time Step" that is used to set the size of the time steps to be generated for the irregular sea.
- A set of radio buttons labeled "Generate Using" that are used to choose between Periods, Heights and Phases, Periods and Heights, and Pierson-Moscowitz.

The second part of the Generate Irregular Sea Wave Components dialogue box changes depending on the values that are given in the first part. When "Generate Using" is set to Periods, Heights and Phases, there are no entries in the second area.

When "Generate Using" is set to Periods and Heights, the second part consists of:

- An edit field labeled "Random Seed" that is used to set the seed for random number generation. A prime number should be used.

When "Generate Using" is set to Pierson-Moscowitz, the second part consists of:

- An edit field labeled "Random Seed" that is used to set the seed for random number generation. A prime number should be used.
- An edit field labeled "Number of Wave Components" that is used to set the number of wave components to be generated for the irregular sea.
- An edit field labeled "Significant Wave Height" that is used to set the significant height for the waves generated for the irregular sea.
- An edit field labeled "Zero Crossing Period" that is used to set the zero crossing period for the waves generated for the irregular sea.

The third part of the Generate Irregular Sea Wave Components dialogue box has the following:

- An edit field labeled "Plot File Name" is used to name a file which will be saved and can be printed or viewed later.
- An edit field labeled "User Title" is used to enter a title for the water surface plot.
- The Overwrite File and Append to File radio buttons are used to decide whether to overwrite or append to an existing file.
- A button labeled "Generate Components/Stop" will start the generation of wave components. While the generation is proceeding, it is possible to stop the generation by pressing this button again. Note that all other buttons are inactive. Stop is included as an option simce this operation can take some time and can use significant computer resources.
- A button labeled "Plot Water Surface" will produce a plot file of the water surface. A dialogue box will be displayed showing the water surface plot.
- A button labeled "Show Wave Components" will display a searchable viewing box that contains the generated wave components. The period, height and phase of each wave component generated are displayed.
- A button labeled "Show Statistics of Largest Waves" will display a searchable viewing box that contains the largest generated wave components. The period, height and phase are displayed.
- Apply button. Selecting this button will replace the old components with the newly generated ones using the selections currently displayed and close the Generate Irregular Sea Wave Components dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Generate Irregular Sea Wave Components dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.10.2.4 When Wave Component Generation is set to Multi-Directional Wave Components the second part consists of (Figure 9-16): :

- An edit field labeled "Number of Wave Components" that is used to set the number of components that the user will input to define the irregular sea.
- A button labeled "Edit Components" which when pressed opens a Wave Components dialogue box to define as many points as were set in the

previous field.

The Wave Components dialogue box (Figure 9-17) consists of six scrollable lists of edit fields, one for the Period, one for the Height, one for the Phase and one for each of the component vectors (X, Y and Z) of each component. Each set of six values is used to define one wave component. There is an Apply button to accept the components as set and close the dialogue, and there is a Cancel button to close the dialogue box without using any changes.

9.10.3 Control Buttons

- Apply button. Selecting this button will apply the selections currently displayed and close the Irregular Sea Data dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Irregular Sea Data dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.11 Current Load

The eleventh selection of the Load menu will display the Current Data dialogue box (Figure 9-18). This dialogue box allows the user to define, review, and/or change current profile data.

The Current Data dialogue box has the following:

- An edit field for the time at which current profile occurs. Before this time the value of the current is zero. Note that for generating a push-over load pattern, the time must be set to zero.
- An edit field for the current blockage factor.
- An edit field for the number of elevations describing the current profile. You must specify a minimum of two.
- A button labeled "Show Current Profile" that when pressed will display the Current Profile dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Current Data dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Current Data dialogue box and use the settings that were already in effect. Any changes made will be lost.

Current Profile

The current profile is a set of five values defining the current in terms of speed and X, Y, and Z components of a direction vector at various depths below the water surface. The direction vector defines the orientation of the irregular sea with respect to the structure's coordinate system.

The Current Profile dialogue box has a scrollable list of edit fields (Figure 9-19). Each line represents a profile point and has 5 edit fields. These edit fields represent the following:

- Depth. This is the absolute distance below still water surface (MWL + tide + storm surge) where current velocity is defined. If two profile points have the same depth, the analysis may terminate prematurely.
- X-Component of current direction vector.
- Y-Component of current direction vector.
- Z-Component of current direction vector.
- Current speed. Note that if you want to specify a zero current speed, you must also specify zero values for the components of the direction vector. If you specify zero value for the speed and nonzero values for any of the components of the direction vector, the values of the components of the direction vector will be assumed to be the actual current speeds in the corresponding direction.

In addition the dialogue box has two buttons, Apply and Cancel.

- Apply button. Selecting this button will apply the selections currently displayed and close the Current Profile dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Current Profile dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.12 Ground Motion Record Dialogue Box

The twelfth selection of the Loads menu will display the Ground Motion Record dialogue box (Figure 9-20). This dialogue box allows the user to specify the ground motion record data. The Ground Motion Record dialogue box has the following:

- A button labeled "Read Record" which will activate the Read Record dialogue box (see below). This allows the user to read the ground motion data from ascii files that the user provides.
- A button labeled "Get Library Record" which will activate the Get Library Record dialogue box (see below). This allows the user to retrieve ground motion record libraries that they have stored.
- A button labeled "Scale Record" which will activate the Scale Record dialogue box (see below). This allows the user to scale the ground motion data currently in memory.
- A button labeled "Plot Record" which will activate the Plot Record dialogue box (see below). This allows the user to plot the ground motion data currently in memory.
- A button labeled "Save Library" along with an edit field, which will save the current ground motion record data into a library with the name given.
- A button labeled "Edit Record Notes" which will activate the Edit Record Notes dialogue box. This allows the user to view, edit or add notes to the current ground motion record.
- A button labeled "Done" which will close the Ground Motion Record dialogue box using any changes that have been made.

9.12.1 Read Record Dialogue Box (Figure 9-21)

The Read Record dialogue box allows the user to create a ground motion record based on the data contained in an ascii file. The Read Record dialogue box consists of the following:

- An edit field labeled "Path" that is used to set the path of the file for reading the ground motion record data.
- An edit field labeled "Primary - File Name" that is used to set the file name for reading the primary ground motion record data.
- An edit field labeled "Secondary - File Name" that is used to set the file name for reading the secondary ground motion record data.
- An edit field labeled "Vertical - File Name" that is used to set the file name for reading the vertical ground motion record data.
- An edit field labeled "Number of Records" that is used to set the number of records to read from the ground motion record data file.
- A set of radio buttons labeled "File Format" that lets the user choose between format A and format B.

When format A has been selected (Figure 9-21):

- An edit field labeled "Assumed Time Step" that is used to set the time step increment of the ground motion record in this file.
- An edit field labeled "Acceleration Scale Factor" that is used to set the acceleration scale factor that will be used on the ground motion record data when reading the file.
- An edit field labeled "Number of Acceleration Records / Line" that is used to set the number of acceleration records that are on each line of the

ground motion record data file that will be read.

- An edit field labeled "Acceleration Field Width" that is used to set the width (number of characters) of each acceleration field in the the ground motion record data file.

When format B has been selected (Figure 9-22):

- An edit field labeled "Acceleration Field Width" that is used to set the width (number of characters) of each acceleration field in the the ground motion record data file.
- An edit field labeled "Acceleration Scale Factor" that is used to set the acceleration scale factor that will be used on the ground motion record data when reading the file.

Control Buttons

- Apply button. Selecting this button will apply the selections currently displayed and close the Read Record dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Read Record dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.12.2 Get Library Record Dialogue Box (Figure 9-23)

The Get Library Record dialogue box is used to retrive a ground motion record library that that the user has previously saved. The Get Library Record dialogue box consists of the following:

- A scrollable list labeled "Record" that contains the names of the stored records. Click the mouse on one of the names to select it.
- A button labeled "Edit Record Notes" that when pressed will display any notes for the highlighted ground motion record in a searchable text viewer. New text may be added and existing text may be edited.
- Apply button. Selecting this button will apply the selections currently displayed and close the Get Library Record dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Get Library Record dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.12.3 Scale Record Dialogue Box (Figure 9-24)

The Scale Record dialogue box is used to create a ground motion record by scaling the data in an existing ground motion record. The scaling is done on two stages. The first stage involves computing scale factors. The second stage actually scales the ground motion record components using the scale factors. Scale factors are computed by comparing the response spectrum of each of component of the ground motion record with a target response spectrum and minimizing the error between the actual spectrum and the target spectrum within a set range. Note that the user can bypass the first stage by inputing the desired scale factors. The Scale Record dialogue box consists of the following:

- There are three buttons labeled "Primary Spectrum", "Secondary Spectrum" and "Vertical Spectrum" which when pressed will activate a dialog box (see 9.12.3.1) where the target response spectra can be defined.
- Three sets of radio buttons that let the user choose between the Primary, Secondary and Vertical target response spectra for each ground motion

record component (Primary, Secondary and Vertical components). The response specrum of each ground motion record component will be compared with the target response spectrum selected by the user in order to calculate the scale factor.

- An edit field labeled "Scale Factor" that is used to display the computed scale factor and let the user set it.
- An edit field labeled "Scaling Range Minimum Period" that is used to set the minimum period of the ground motion record response spectrum scaling range.
- An edit field labeled "Scaling Range Maximum Period" that is used to set the maximum period of the ground motion record response spectrum scaling range.
- An edit field labeled "Structure Damping" that is used to set the structure damping during the ground motion record response spectrum calculations.
- A button labeled "Compute Scale" which when pressed will compute ground motion record scale factors.
- Apply button. Selecting this button will scale the components of the ground motion record using the displayed scale factors.
- Done button. Selecting this button will close the Scale Record dialogue box.

9.12.3.1 Primary, Secondary and Vertical Response Spectrum Dialogue Boxes (Figure 9-25)

The Primary, Secondary and Vertical Response Spectrum Dialogue Boxes are used to set the target reposnse spectra data to be used in ground motion record scaling The Primary, Secondary and Vertical Response Spectrum Dialogue Boxes consist of the following:

- The button labeled "Read File" will activate the Read File dialogue box. The Read File dialogue box is used to read an existing sprectrum data file.
- The button labeled "Interactive Input" will activate the Interactive Input dialogue box. The Interactive Input dialogue box is used to interactively set data points to describe the response spectrum.
- The button labeled "Generate From Ground Motion" will activate the Generate From Ground Motion dialogue box. The Generate From Ground Motion dialogue box is used to generate a response spectrum based on the existing ground motion record.
- The button labeled "Generate From Nodal Results" will activate the Generate From Nodal Results dialogue box. The Generate From Nodal Results dialogue box is used to generate a response spectrum based on a nodal displacements record that is part of the results of a previous analysis.
- The button labeled "Plot Spectrum" will activate the Plot Spectrum dialogue box. The Plot Spectrum dialogue box is used to create and view a plot of the current spectrum data.
- Done button. Selecting this button will close the Primary, Secondary or Vertical Response Spectrum Dialogue Box

9.12.3.2 Read File Dialogue Box (Figure 9-26)

The Read File dialogue box is used to read an existing response spectrum data file. The Read File dialogue box consists of the following:

- An edit field labeled "File Name" that is used to set the file name of the response spectrum data to be read.
- An edit field labeled "File Path" that is used to set the file path of the ground motion record response spectrum data to be read.
- A set of radio buttons labeled "Response Type" that lets the user choose between Acceleration, Velocity or Displacement response types.
- An edit field labeled "Number of Points Defining the Resonse Spectrum" that is used to set the number of points in the file to be read.
- Apply button. Selecting this button will apply the selections currently displayed and close the Primary, Secondary or Vertical Response Spectrum dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Primary, Secondary or Vertical Response Spectrum dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.12.3.3 Interactive Input Dialogue Box (Figure 9-27)

The Interactive Input dialogue box is used to interactively set data points to describe the response spectrum. The Interactive Input dialogue box consists of the following:

- A set of radio buttons labeled "Response Type" that lets the user choose between Acceleration, Velocity or Displacement response spectrum data to input.
- An edit field labeled "Number of Points Defining the Response Spectrum" that is used to set the number of points defining the resonse spectrum.
- A button labeled "Edit Points" that when pressed will display a Response Spectrum Points dialogue box (Figure 9-28) with a pair of scrollable edit fields that can be used to set the data for the response spectrum points. There will be a pair of fields for each point, one for the period and one for either pseudo acceleration, pseudo velocity or pseudo displacement. There is an apply button to accept the changes made and a cancel button to discard them.
- Apply button. Selecting this button will apply the selections currently displayed and close the Interactive Input dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Interactive Input dialogue box and use the settings that were already in effect. Any changes made will be lost.

9.12.3.4 Generate From Ground Motion Dialogue Box (Figure 9-29)

The Generate From Ground Motion dialogue box is used to generate a response spectrum based on the existing ground motion record. The Generate From Ground Motion dialogue box consists of the following:

- A set of radio buttons labeled "Ground Motion Direction" that lets the user choose between generating from Primary, Secondary or Vertical ground motion components.
- An edit field labeled "Response Spectra Start Period" that is used to set the response spectra start period.
- An edit field labeled "Response Spectra End Period" that is used to set the response spectra end period.

- An edit field labeled "Number of Response Spectra Period" that is used to set the number of response spectra period to be computed.
- An edit field labeled "Structure Damping" that is used to set the structure damping to be used in response spectra computations.
- Apply button. Selecting this button will apply the selections currently displayed, generate a response spectrum and close the Generate From Ground Motion dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Generate From Ground Motion dialogue box with out generating a response spectrum. Any changes made will be lost.

9.12.3.5 Generate From Nodal Results Dialogue Box (Figure 9-30)

The Generate From Nodal Results dialogue box is used to generate a response spectrum based on a nodal displacement record. The Generate From Nodal Results dialogue box consists of the following:

- An edit field labeled "Node Name" that is used to set the node name.
- An edit field labeled "First Load Step" that is used to set the first load step of the node displacement record record to be used in generating the response spectrum.
- An edit field labeled "Last Load Step" that is used to set the last load step of the node displacement record record to be used in generating the response spectrum.
- Tha Maximum Load Step that is used in the first and last load step fields is shown next. Note that the minimum load step is 1.
- A set of radio buttons labeled "Displacement Direction" that lets the user choose between the X, Y or Z direction for the node displacement.
- An edit field labeled "Response Spectra Start Period" that is used to set the response spectra start period.
- An edit field labeled "Response Spectra End Period" that is used to set the response spectra end period.
- An edit field labeled "Number of Response Spectra Period" that is used to set the number of response spectra period to be computed.
- An edit field labeled "Structure Damping" that is used to set the structure damping to be used in response spectra computations.
- Apply button. Selecting this button will apply the selections currently displayed, generate a response spectrum and close the Generate From Nodal Results dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Generate From Nodal Results dialogue box with out generating a response spectrum. Any changes made will be lost.

9.12.3.6 Plot Spectrum Dialogue Box (Figure 9-31)

The Plot Spectrum dialogue box is used to create and view a plot of the current spectrum data. The Plot Spectrum dialogue box consists of the following:

- An edit field labeled "File Name" that is used to set the file name of the ground motion record plot.

- An edit field labeled "User Title" that is used to set the user title of the ground motion record plot.
- A pair of radio buttons labeled "Overwrite" and "Append" which determine whether an existing file of the same name should be overwritten or appended (allowing multiple plots in one file).
- A pair of radio buttons labeled "Plot Type" that lets the user choose between Linear and Four Way ground motion record plots.
- A set of radio buttons labeled "Response Type" that lets the user choose between Acceleration, Velocity or Displacement response spectrum ground motion record data to plot. This set is only visible when the selected plot type is Linear.
- A checkbox labeled "Show Detailed Grid" that lets the user toggle a detailed log-log grid for the Four-Way plot. This checkbox is only visible when the selected plot type is Four-Way.
- A checkbox labeled "Show API Values" toggles whether API values are shown.
- A set of radio buttons labeled "Soil Type" that lets the user choose between Rock, Shallow Strong Alluvium and Deep Strong Alluvium soils. The soil type is used to evaluate the recommended API response spectrum.
- Apply button. Selecting this button will create a plot file with the given name and display the plot in a viewer.
- Cancel button. Selecting this button will close the Plot Spectrum dialogue box.

9.12.4 Plot Record Dialogue Box (Figure 9-32)

The Plot Record dialogue box is used to create and view a plot of the current ground motion record data. The Plot Record dialogue box consists of the following:

- An edit field labeled "File Name" that is used to set the file name of the ground motion record plot.
- An edit field labeled "User Title" that is used to set the user title of the ground motion record plot.
- A pair of radio buttons labeled "Overwrite" and "Append" which determine whether an existing file of the same name should be overwritten or appended (allowing multiple plots in one file).
- A set of radio buttons labeled "Component" that lets the user choose between Primary, Secondary or Vertical ground motion record data to plot.
- An edit field labeled "Max Time Record" that is used to set the max time record of the ground motion record plot.
- An edit field labeled "Start Time" that is used to set the start time of the ground motion record plot.
- An edit field labeled "End Time" that is used to set the end time of the ground motion record plot.
- The maximum time of the ground motion record is shown next.
- Apply button. Selecting this button will create a plot file with the given name and display the plot in a viewer (Figure 9-33).
- Cancel button. Selecting this button will close the Plot Record dialogue box.

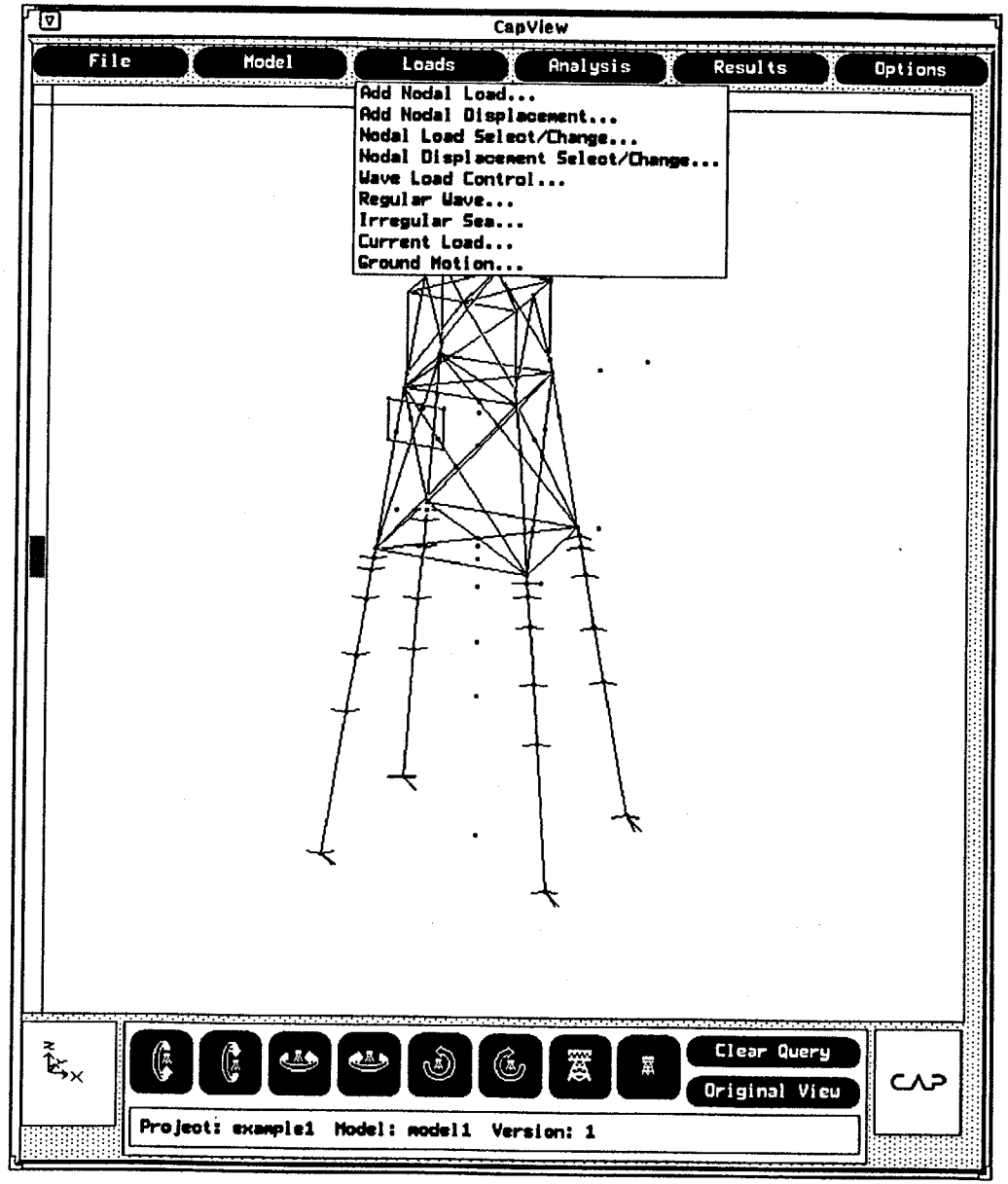

CapView
File
Model
Loads
Analysis
Results
Options
Add Nodal Load...
Add Nodal Displacement...
Nodal Load Select/Change...
Nodal Displacement Select/Change...
Wave Load Control...
Regular Wave...
Irregular Sea...
Current Load...
Ground Motion...
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 9-1

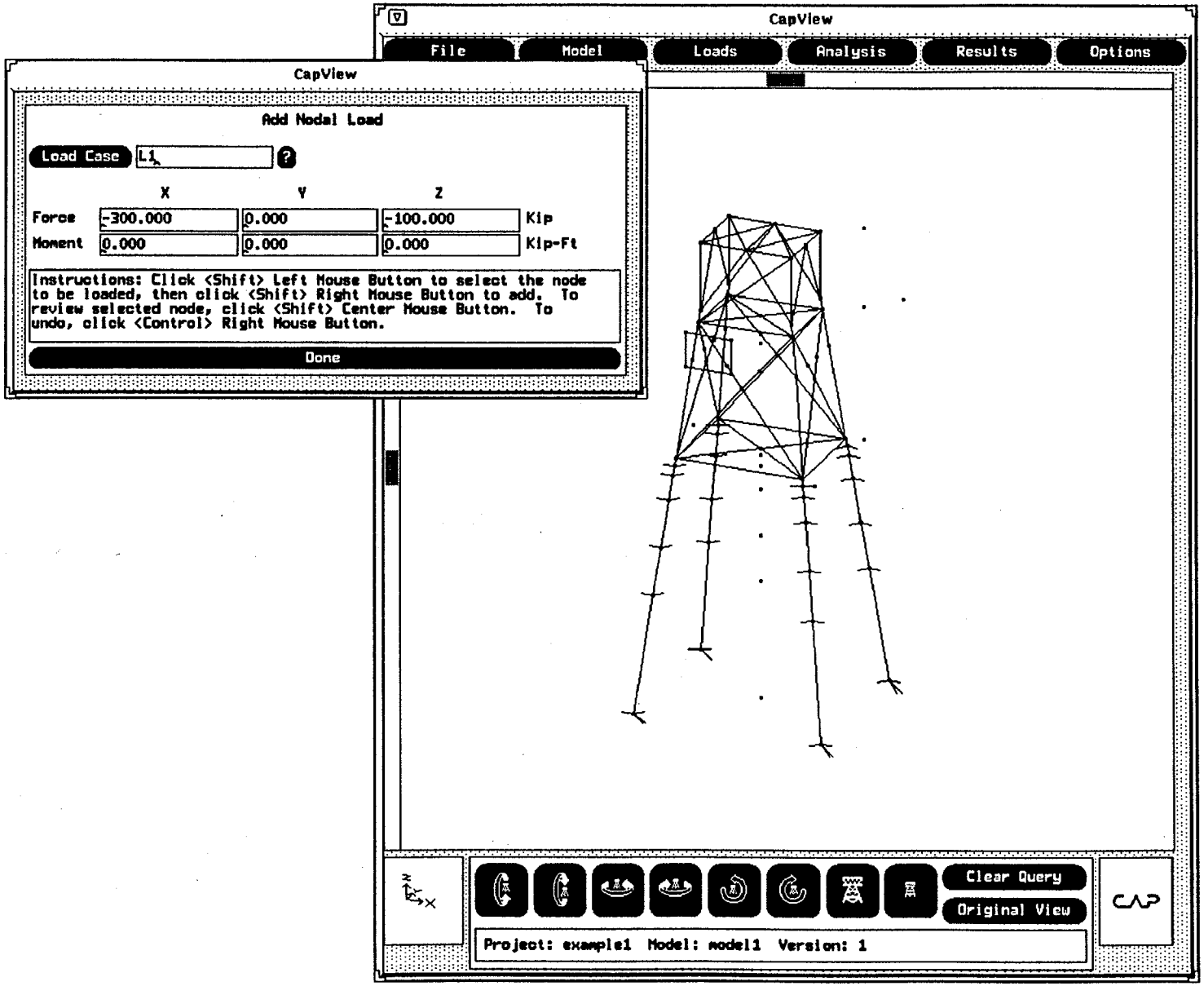

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Nodal Load
Load Case
L1
X
Y
Z
Force
-300.000
0.000
-100.000
Kip
Moment
0.000
0.000
0.000
Kip-Ft
Instructions: Click <Shift> Left Mouse Button to select the node to be loaded, then click <Shift> Right Mouse Button to add. To review selected node, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-2

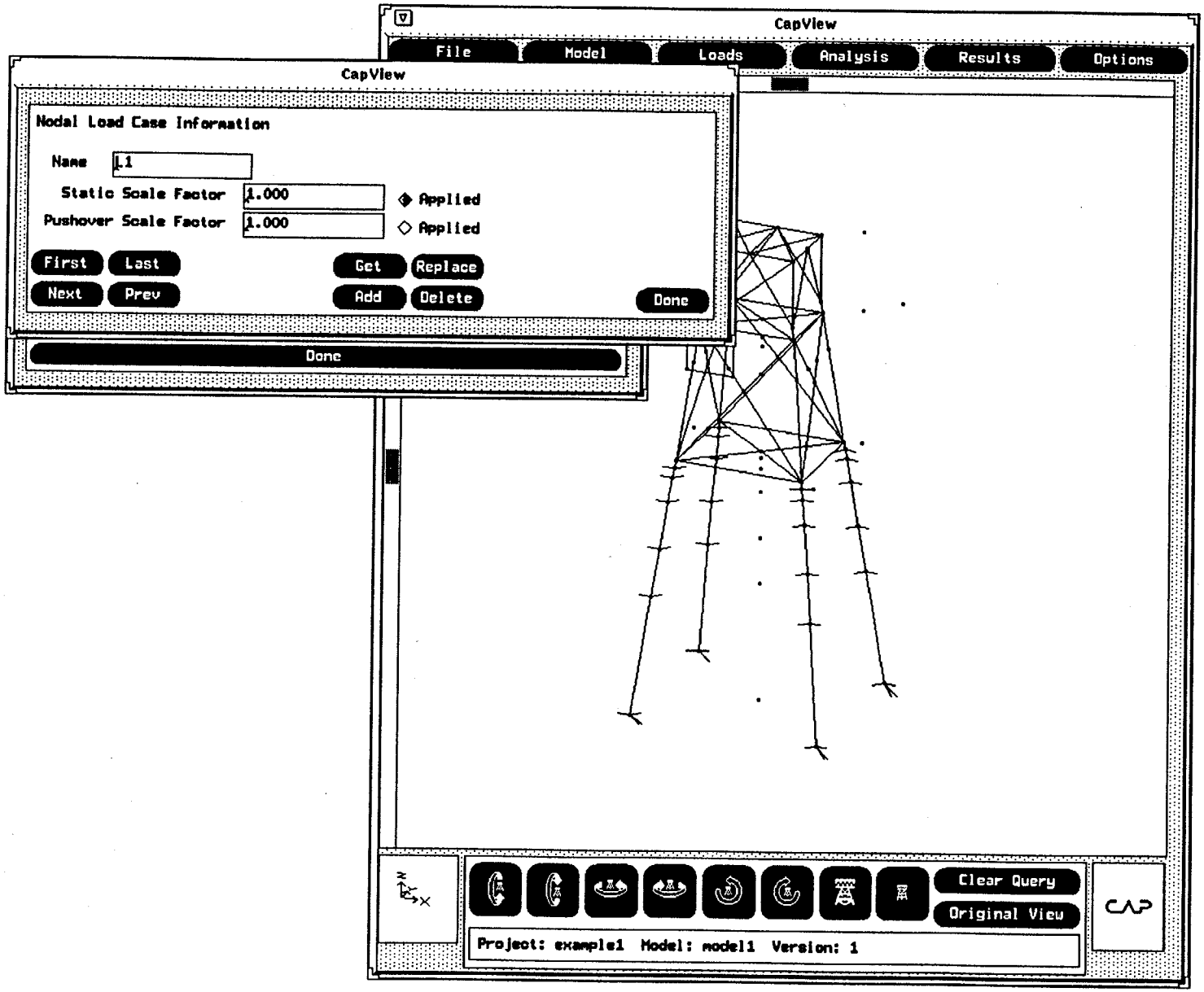

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Nodal Load Case Information
Name
1.1
Static Scale Factor
1.000
Applied
Pushover Scale Factor
1.000
Applied
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-3

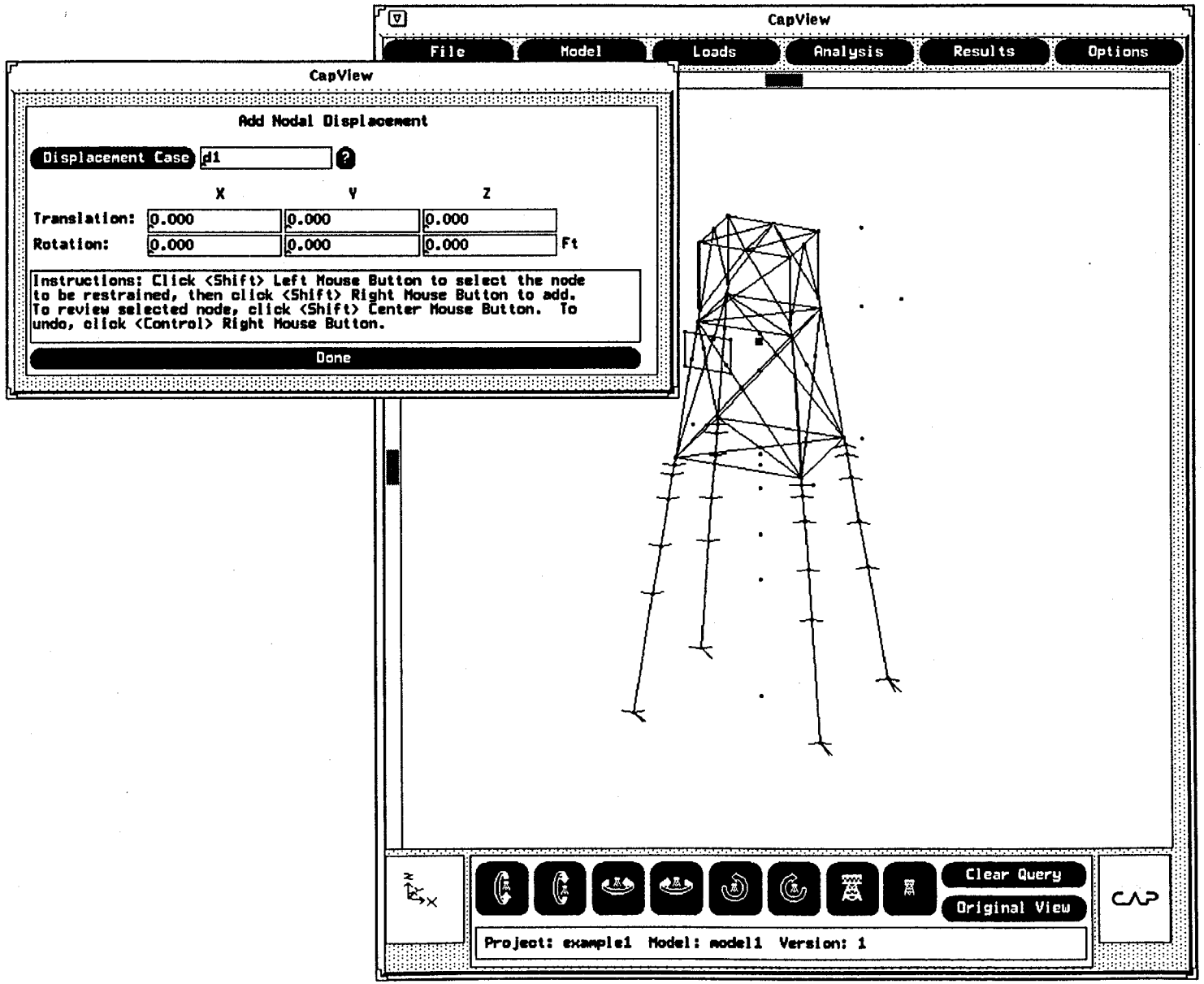

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Add Nodal Displacement
Displacement Case
d1
X
Y
Z
Translation:
0.000
0.000
0.000
Rotation:
0.000
0.000
0.000
Ft
Instructions: Click <Shift> Left Mouse Button to select the node to be restrained, then click <Shift> Right Mouse Button to add. To review selected node, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-4

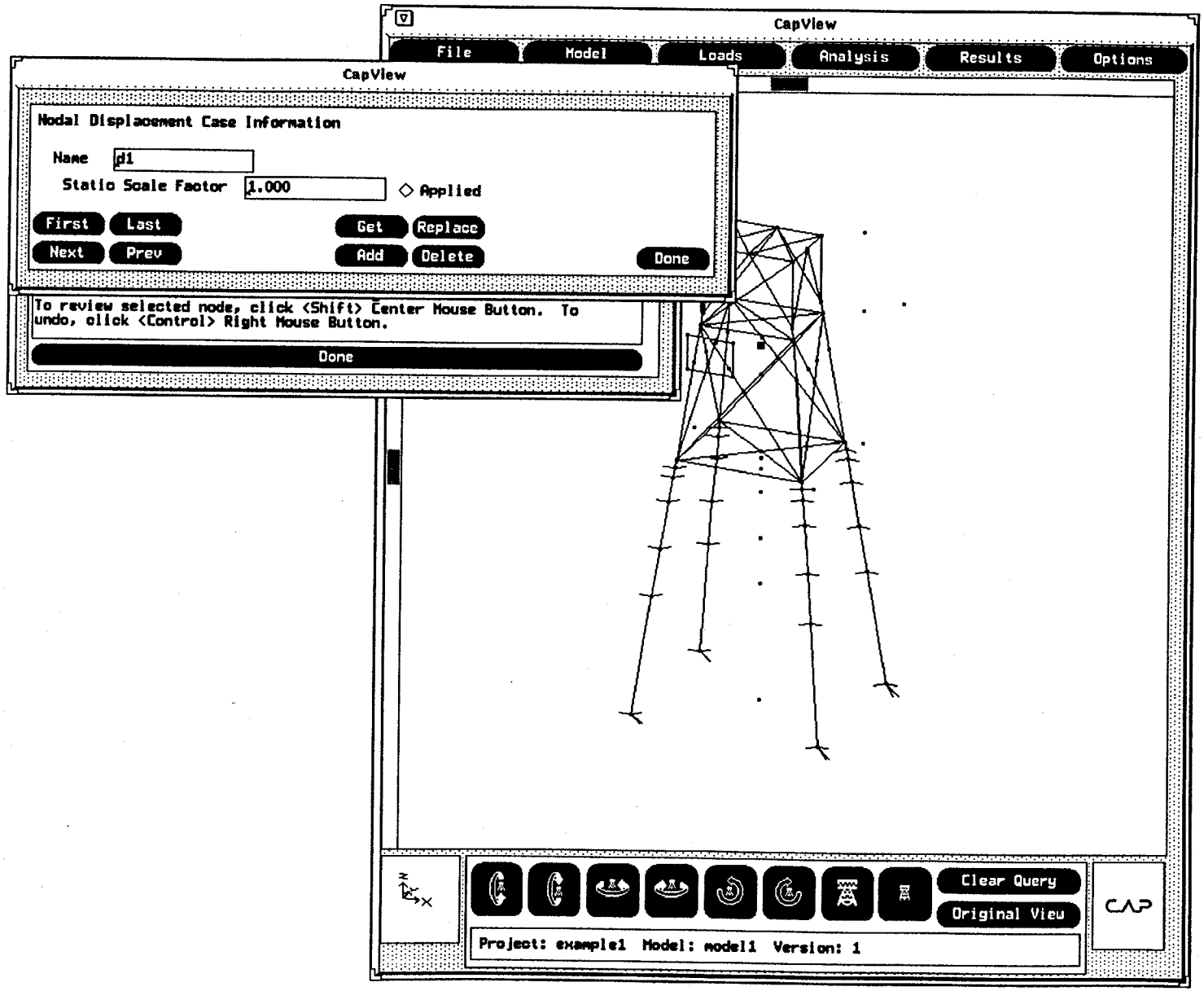

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Nodal Displacement Case Information
Name
d1
Static Scale Factor
1.000
Applied
First
Last
Get
Replace
Next
Prev
Add
Delete
Done
To review selected node, click <Shift> Center Mouse Button. To undo, click <Control> Right Mouse Button.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-5

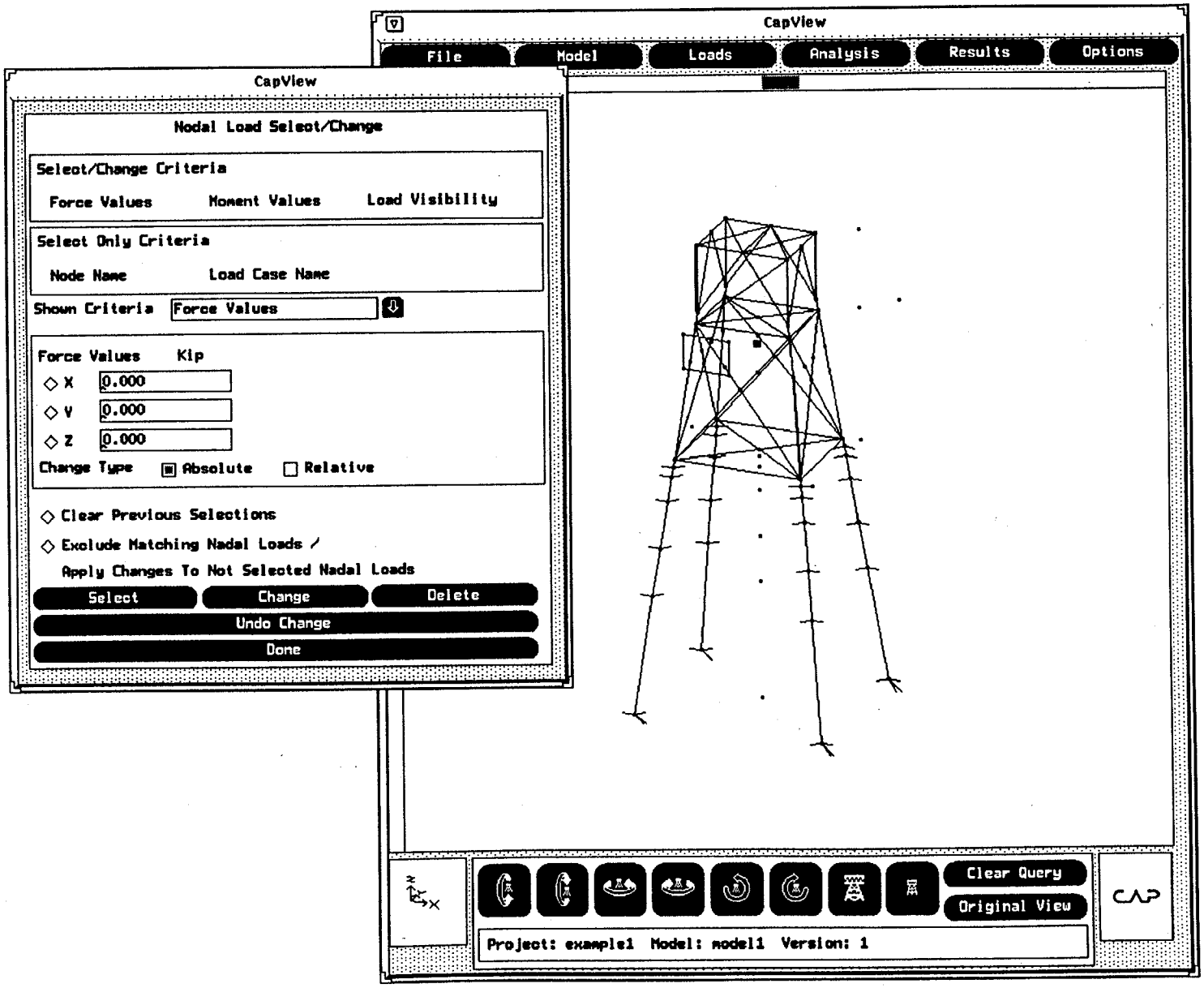

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Nodal Load Select/Change
Select/Change Criteria
Force Values
Moment Values
Load Visibility
Select Only Criteria
Node Name
Load Case Name
Shown Criteria
Force Values
Force Values
Kip
X
0.000
Y
0.000
Z
0.000
Change Type
Absolute
Relative
Clear Previous Selections
Exclude Matching Nodal Loads
Apply Changes To Not Selected Nodal Loads
Select
Change
Delete
Undo Change
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-6

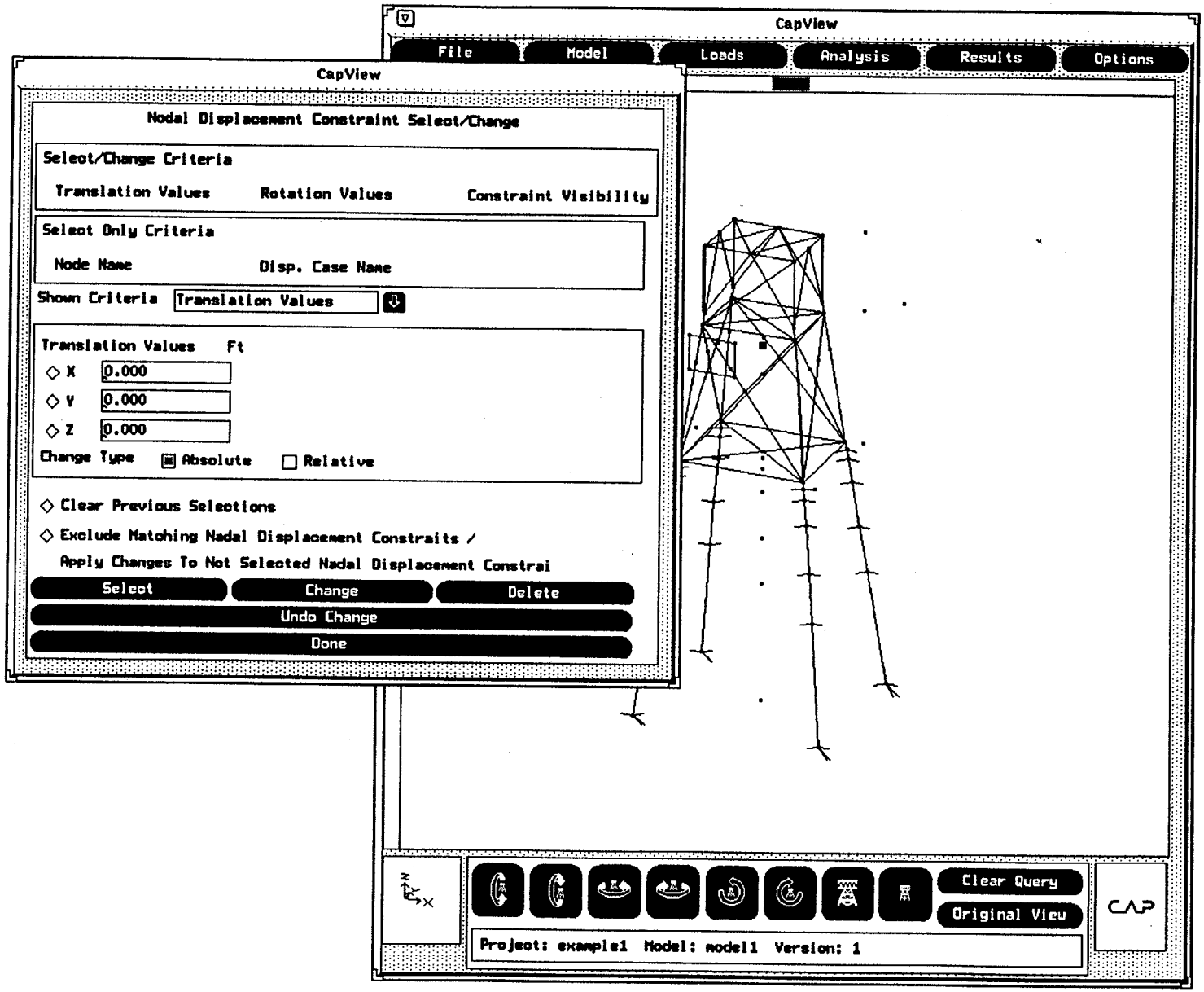

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Nodal Displacement Constraint Select/Change
Select/Change Criteria
Translation Values
Rotation Values
Constraint Visibility
Select Only Criteria
Node Name
Disp. Case Name
Shown Criteria
Translation Values
Translation Values
Ft
X
0.000
Y
0.000
Z
0.000
Change Type
Absolute
Relative
Clear Previous Selections
Exclude Matching Nodal Displacement Constraits /
Apply Changes To Not Selected Nodal Displacement Constrai
Select
Change
Delete
Undo Change
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 9-7

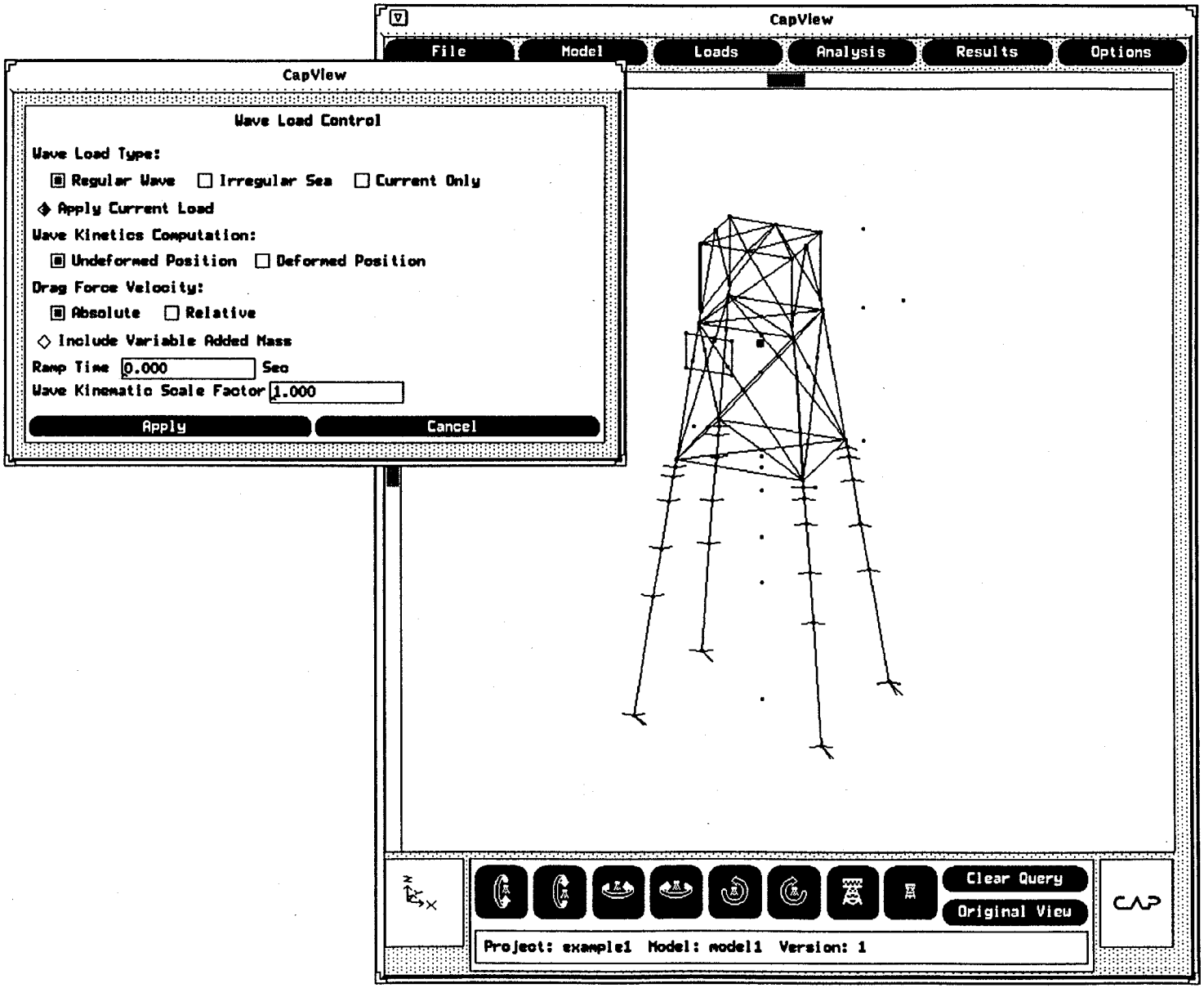

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Wave Load Control
Wave Load Type:
Regular Wave
Irregular Sea
Current Only
Apply Current Load
Wave Kinetics Computation:
Undeformed Position
Deformed Position
Drag Force Velocity:
Absolute
Relative
Include Variable Added Mass
Ramp Time 0.000 Sec
Wave Kinematic Scale Factor 1.000
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-8

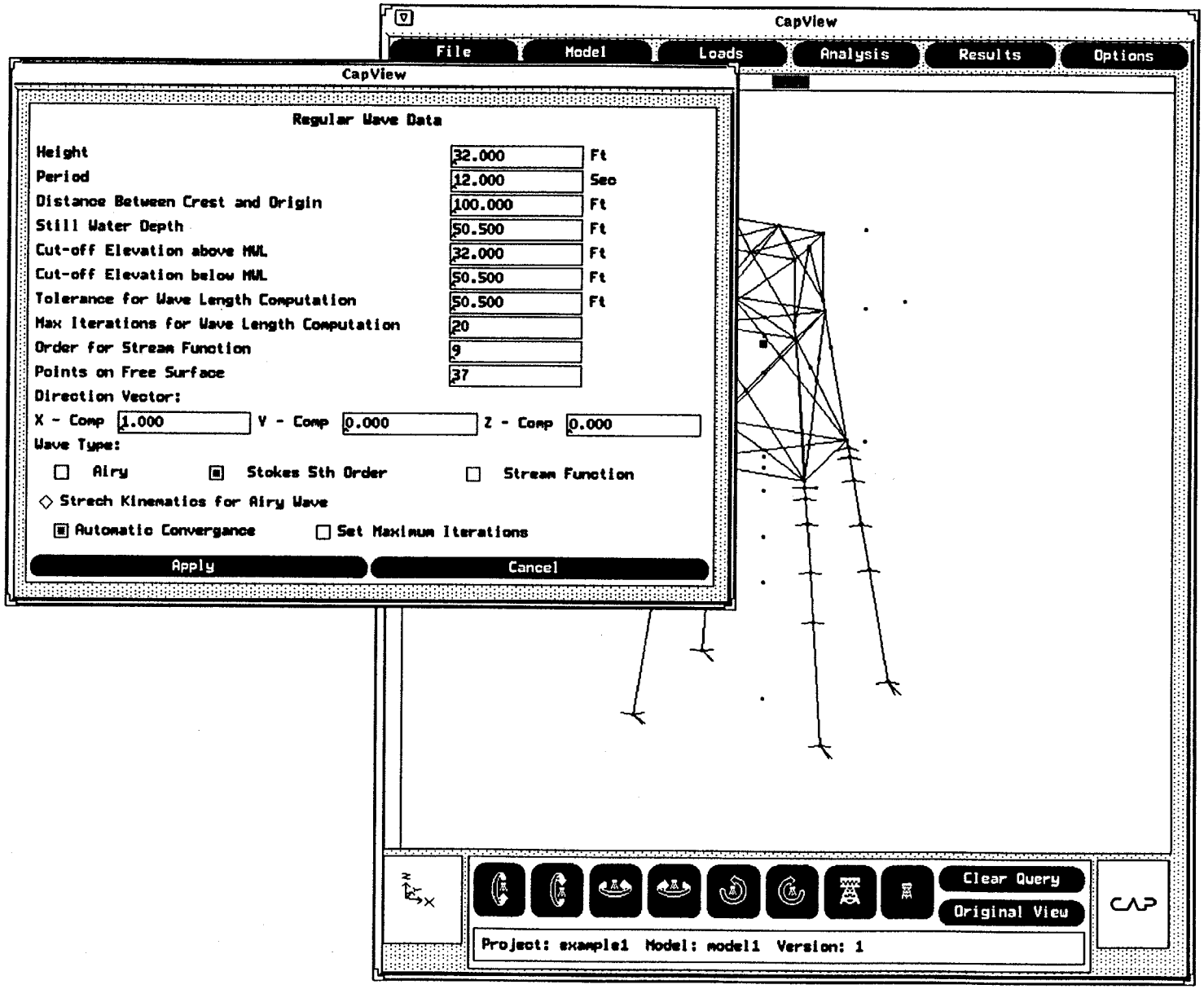

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Regular Wave Data
Height 32.000 Ft
Period 12.000 Sec
Distance Between Crest and Origin 100.000 Ft
Still Water Depth 50.500 Ft
Cut-off Elevation above MWL 32.000 Ft
Cut-off Elevation below MWL 50.500 Ft
Tolerance for Wave Length Computation 50.500 Ft
Max Iterations for Wave Length Computation 20
Order for Stream Function 9
Points on Free Surface 37
Direction Vector:
X - Comp 1.000 Y - Comp 0.000 Z - Comp 0.000
Wave Type:
Airy
Stokes 5th Order
Stream Function
Strech Kinematics for Airy Wave
Automatic Convergance
Set Maximum Iterations
Apply
Cancel
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 9-9

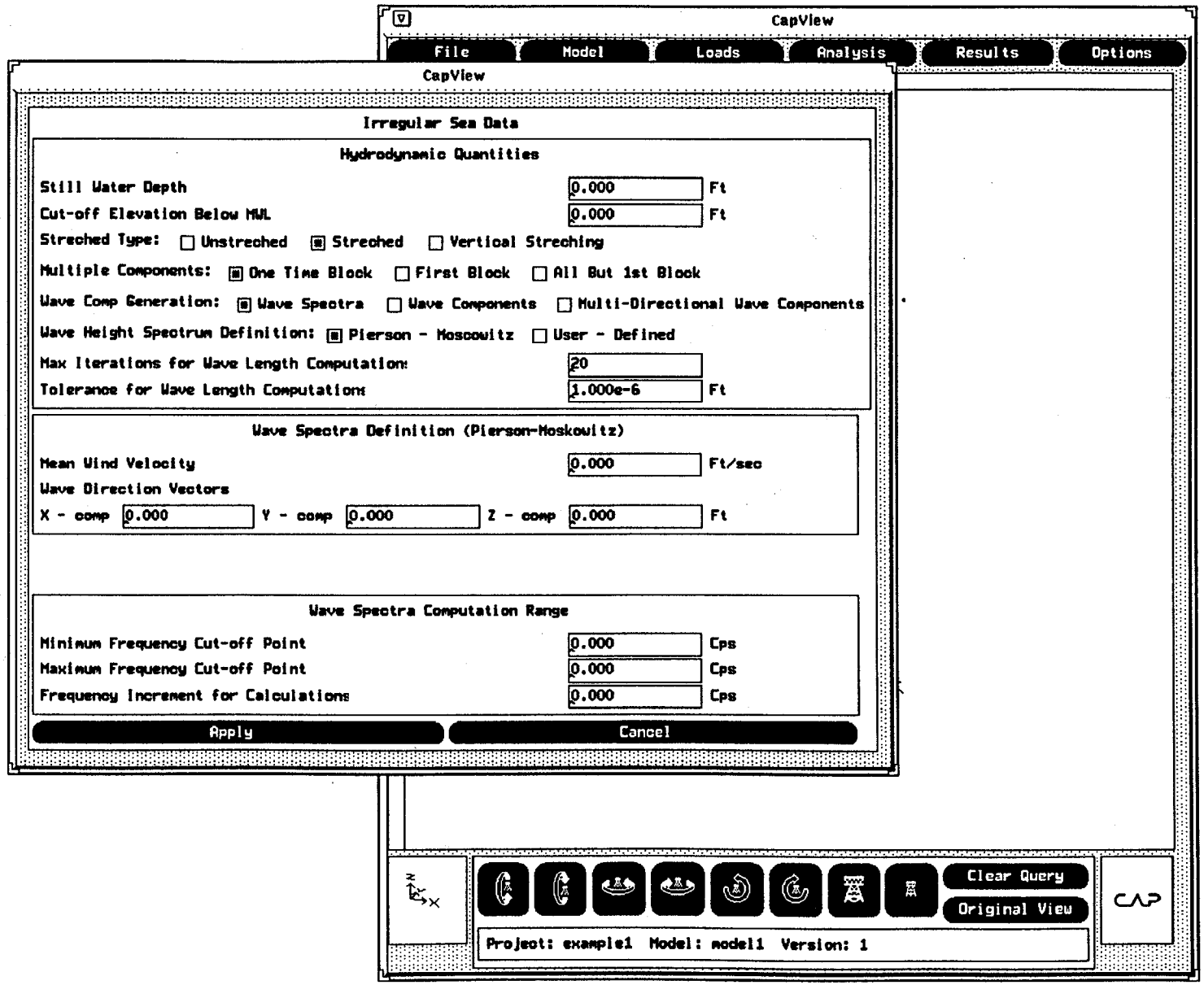

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Irregular Sea Data
Hydrodynamic Quantities
Still Water Depth
0.000
Ft
Cut-off Elevation Below MWL
0.000
Ft
Streched Type:
Unstreched
Streched
Vertical Streching
Multiple Components:
One Time Block
First Block
All But 1st Block
Wave Comp Generation:
Wave Spectra
Wave Components
Multi-Directional Wave Components
Wave Height Spectrum Definition:
Pierson - Moscowitz
User - Defined
Max Iterations for Wave Length Computation:
20
Tolerance for Wave Length Computation:
1.000e-6
Ft
Wave Spectra Definition (Pierson-Moskowitz)
Mean Wind Velocity
0.000
Ft/sec
Wave Direction Vectors
X - comp
0.000
Y - comp
0.000
Z - comp
0.000
Ft
Wave Spectra Computation Range
Minimum Frequency Cut-off Point
0.000
Cps
Maximum Frequency Cut-off Point
0.000
Cps
Frequency Increment for Calculations
0.000
Cps
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-10

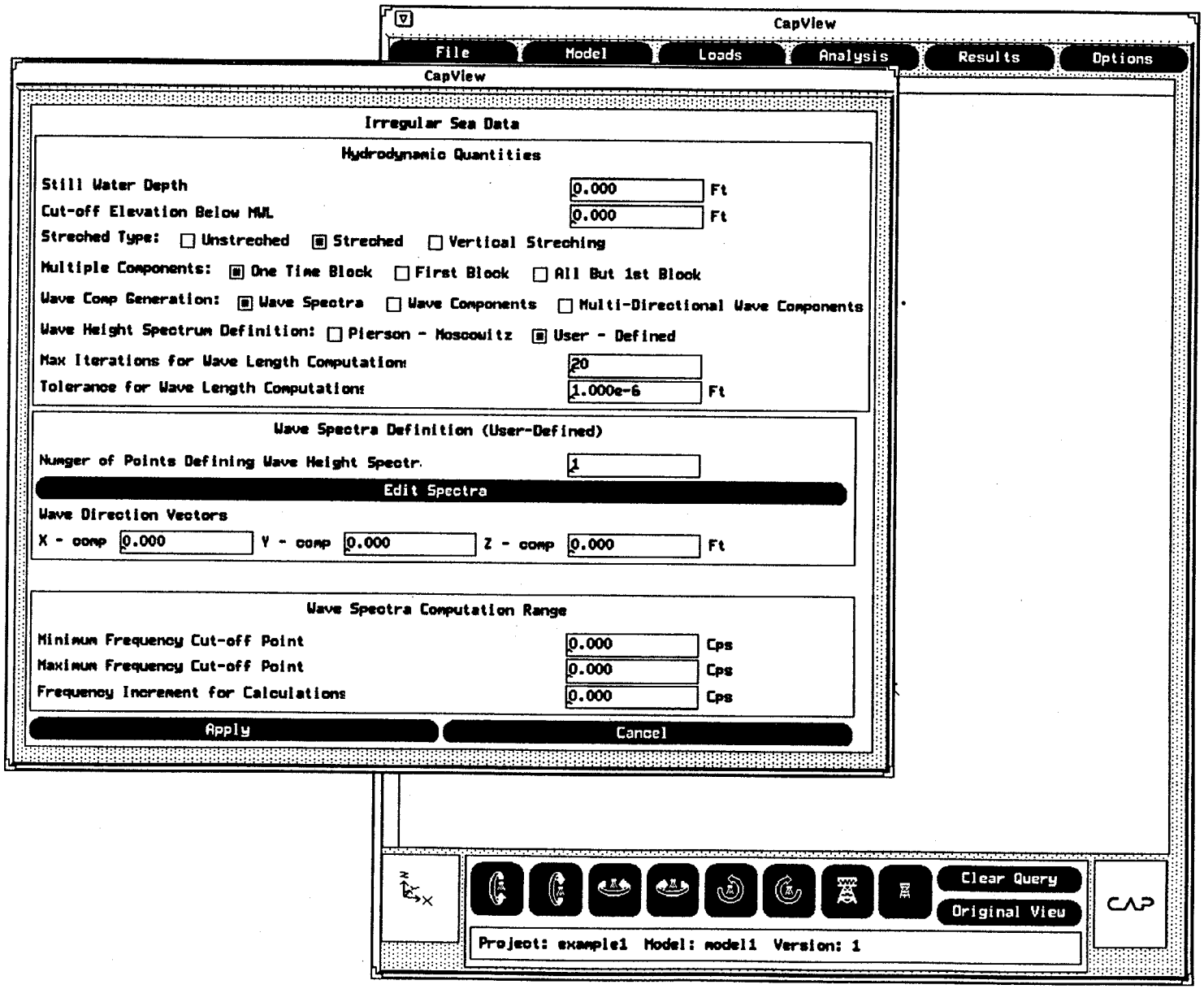

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Irregular Sea Data
Hydrodynamic Quantities
Still Water Depth
0.000
Ft
Cut-off Elevation Below MWL
0.000
Ft
Streched Type:
Unstreched
Streched
Vertical Streching
Multiple Components:
One Time Block
First Block
All But 1st Block
Wave Comp Generation:
Wave Spectra
Wave Components
Multi-Directional Wave Components
Wave Height Spectrum Definition:
Pierson - Moscowitz
User - Defined
Max Iterations for Wave Length Computation
20
Tolerance for Wave Length Computation
1.000e-6
Ft
Wave Spectra Definition (User-Defined)
Number of Points Defining Wave Height Spectr.
1
Edit Spectra
Wave Direction Vectors
X - comp
0.000
Y - comp
0.000
Z - comp
0.000
Ft
Wave Spectra Computation Range
Minimum Frequency Cut-off Point
0.000
Cps
Maximum Frequency Cut-off Point
0.000
Cps
Frequency Increment for Calculations
0.000
Cps
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-11

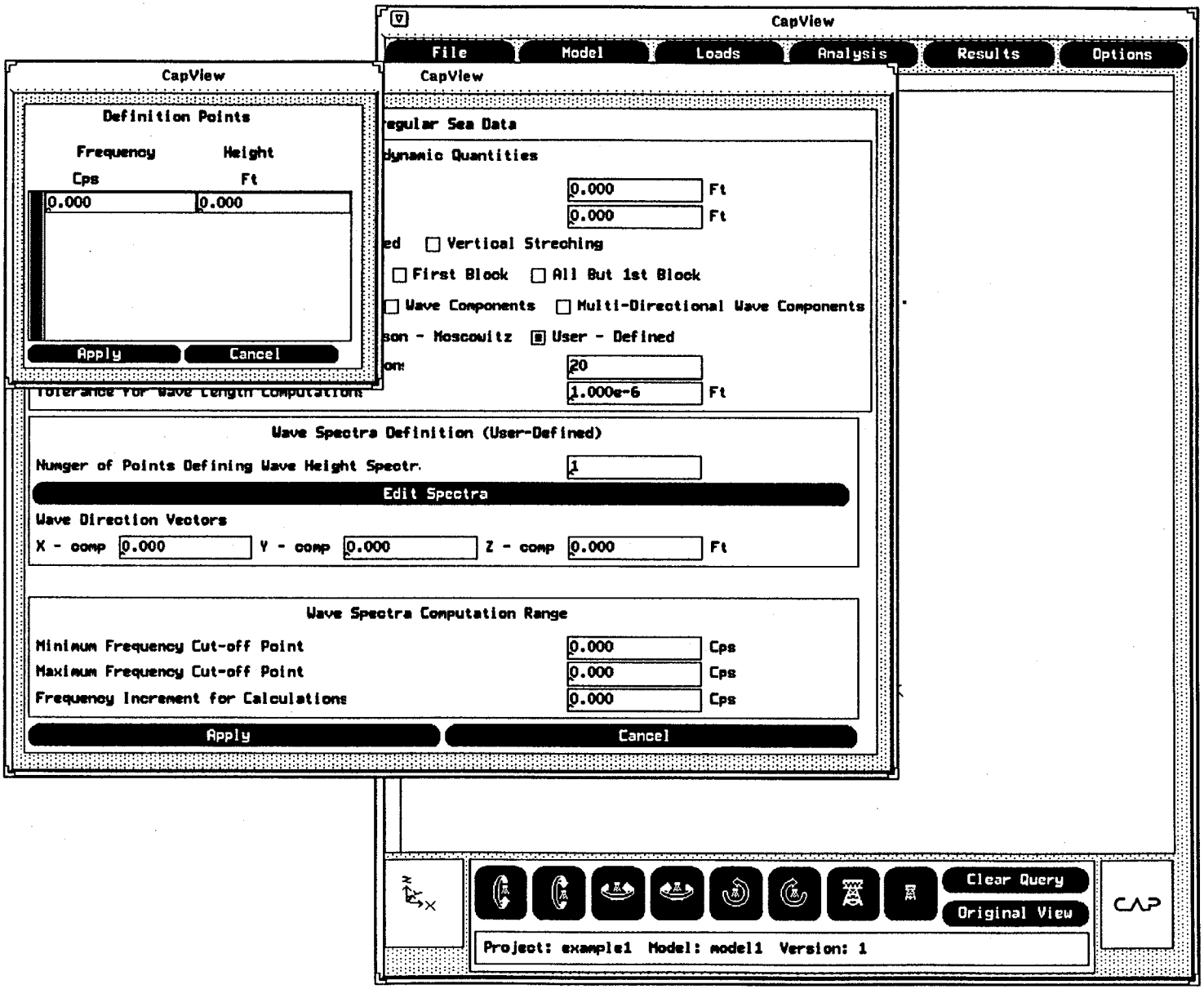

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Definition Points
Frequency
Height
Cps
Ft
0.000
0.000
Apply
Cancel
egular Sea Data
dynamic Quantities
0.000
Ft
0.000
Ft
ed
Vertical Streching
First Block
All But 1st Block
Wave Components
Multi-Directional Wave Components
son - Moscowitz
User - Defined
on:
20
1.000e-6
Ft
Wave Spectra Definition (User-Defined)
Number of Points Defining Wave Height Spectr.
1
Edit Spectra
Wave Direction Vectors
X - comp
0.000
Y - comp
0.000
Z - comp
0.000
Ft
Wave Spectra Computation Range
Minimum Frequency Cut-off Point
0.000
Cps
Maximum Frequency Cut-off Point
0.000
Cps
Frequency Increment for Calculations
0.000
Cps
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-12

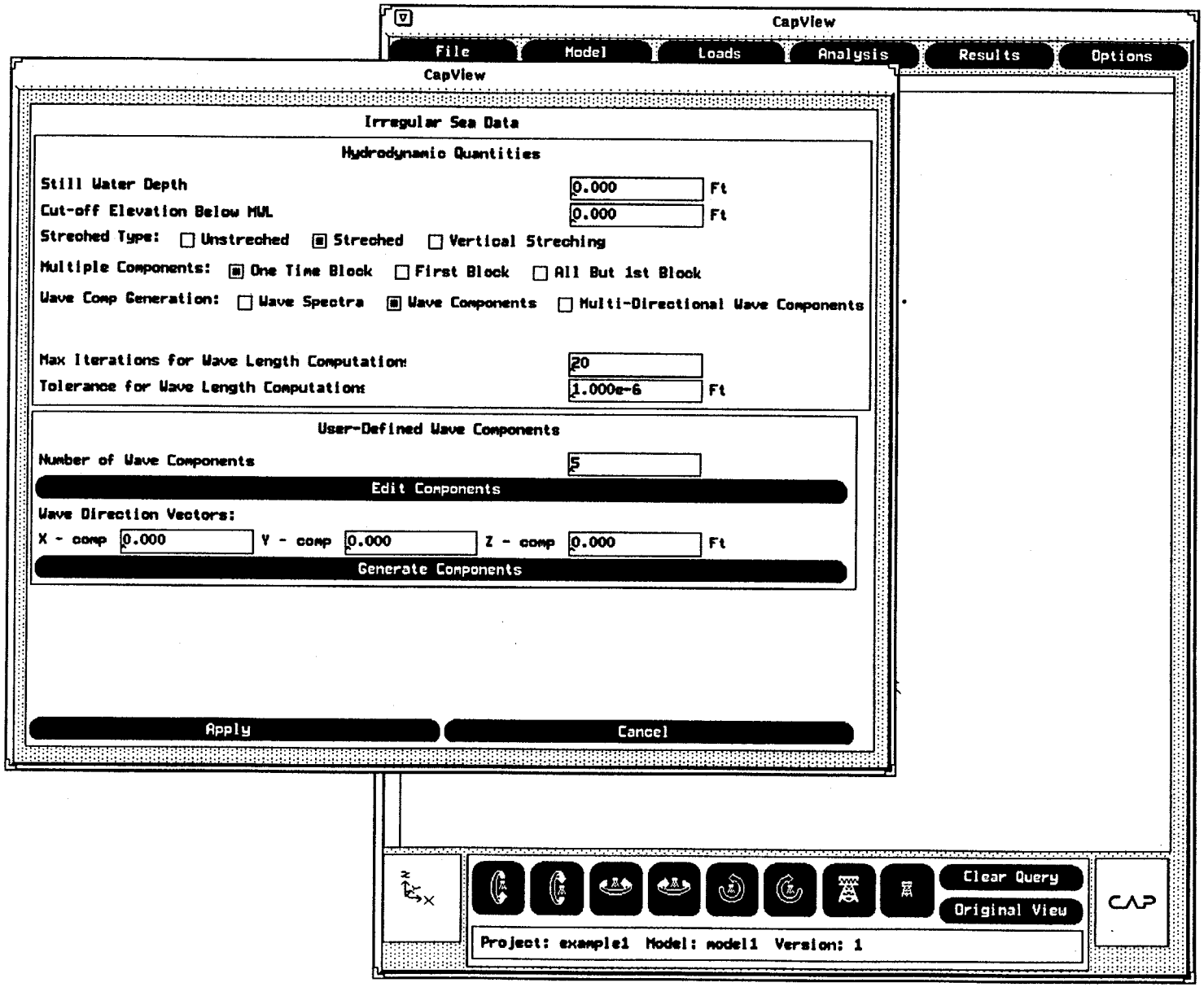

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Irregular Sea Data
Hydrodynamic Quantities
Still Water Depth
0.000
Ft
Cut-off Elevation Below MWL
0.000
Ft
Streched Type:
Unstreched
Streched
Vertical Streching
Multiple Components:
One Time Block
First Block
All But 1st Block
Wave Comp Generation:
Wave Spectra
Wave Components
Multi-Directional Wave Components
Max Iterations for Wave Length Computation:
20
Tolerance for Wave Length Computation:
1.000e-6
Ft
User-Defined Wave Components
Number of Wave Components
5
Edit Components
Wave Direction Vectors:
X - comp
0.000
Y - comp
0.000
Z - comp
0.000
Ft
Generate Components
Apply
Cancel
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 9-13

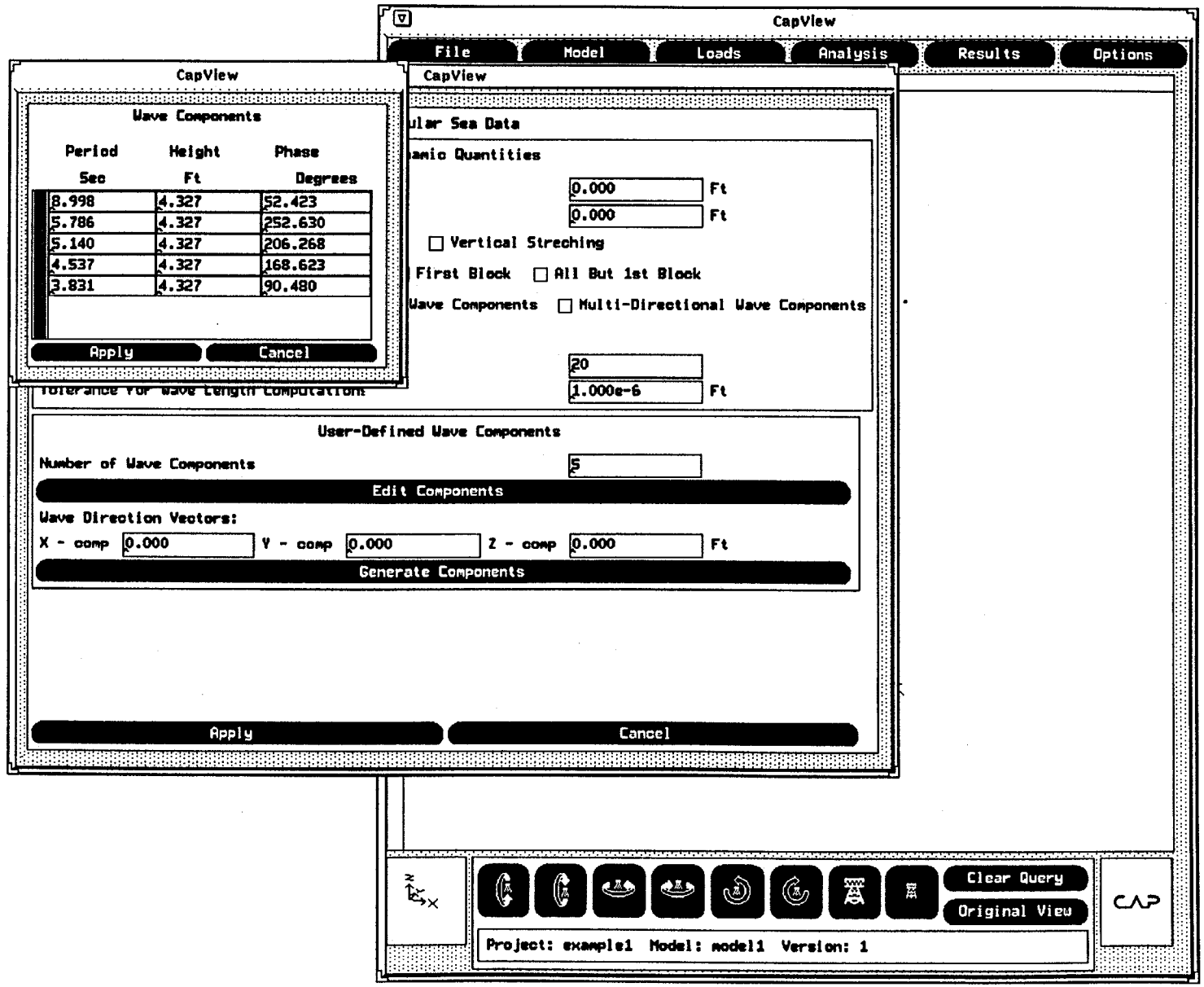

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Wave Components
Period Sec
Height Ft
Phase Degrees
8.998 4.327 52.423
5.786 4.327 252.630
5.140 4.327 206.268
4.537 4.327 168.623
3.831 4.327 90.480
Apply
Cancel
CapView
ular Sea Data
amic Quantities
0.000 Ft
0.000 Ft
Vertical Streching
First Block
All But 1st Block
Wave Components
Multi-Directional Wave Components
20
1.000e-6 Ft
User-Defined Wave Components
Number of Wave Components
5
Edit Components
Wave Direction Vectors:
X - comp 0.000
Y - comp 0.000
Z - comp 0.000 Ft
Generate Components
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-14

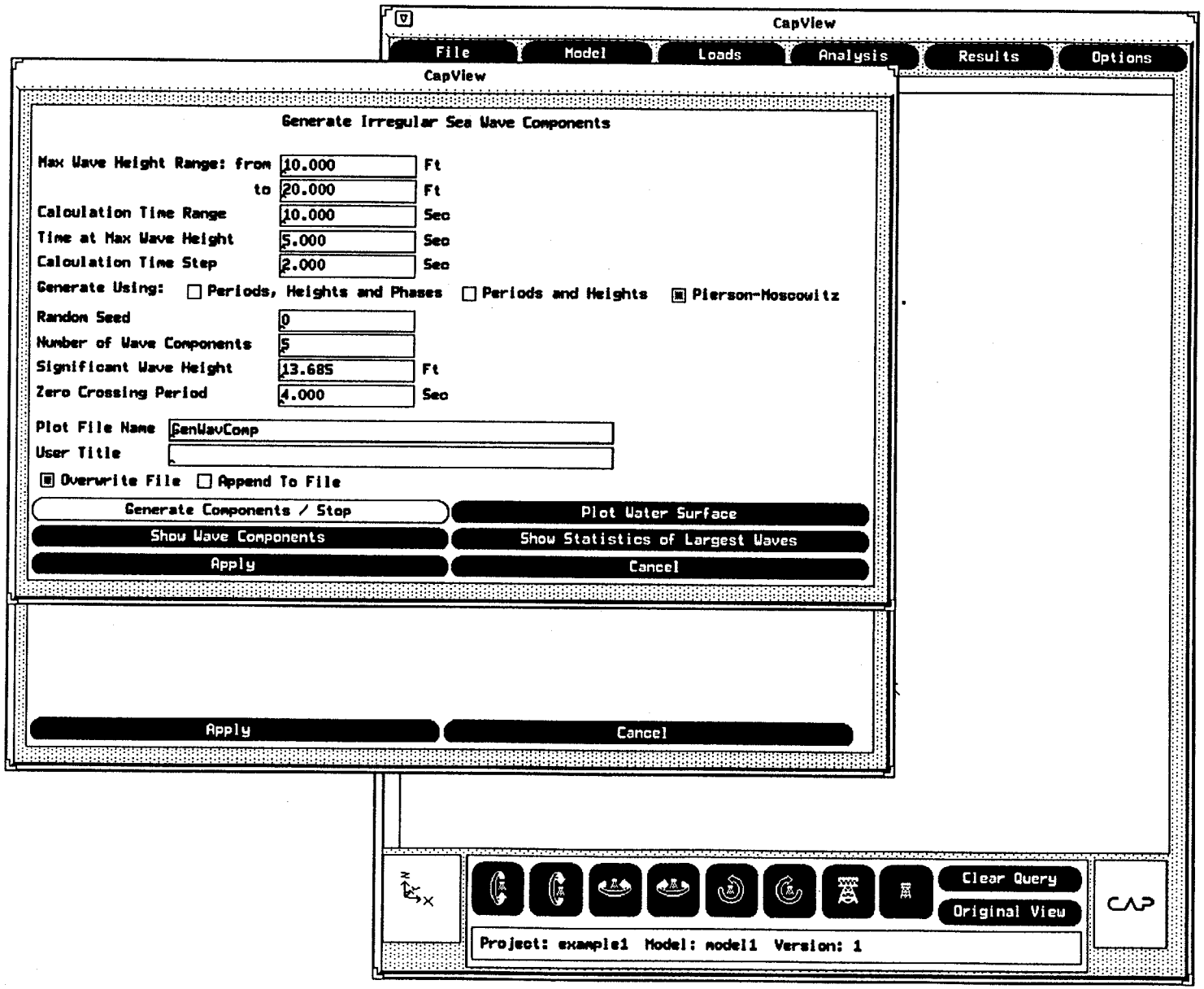

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Generate Irregular Sea Wave Components
Max Wave Height Range: from 10.000 Ft
to 20.000 Ft
Calculation Time Range 10.000 Sec
Time at Max Wave Height 5.000 Sec
Calculation Time Step 2.000 Sec
Generate Using: Periods, Heights and Phases Periods and Heights Pierson-Moscowitz
Random Seed 0
Number of Wave Components 5
Significant Wave Height 13.685 Ft
Zero Crossing Period 4.000 Sec
Plot File Name GenWavComp
User Title
Overwrite File Append To File
Generate Components / Stop
Plot Water Surface
Show Wave Components
Show Statistics of Largest Waves
Apply
Cancel
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-15

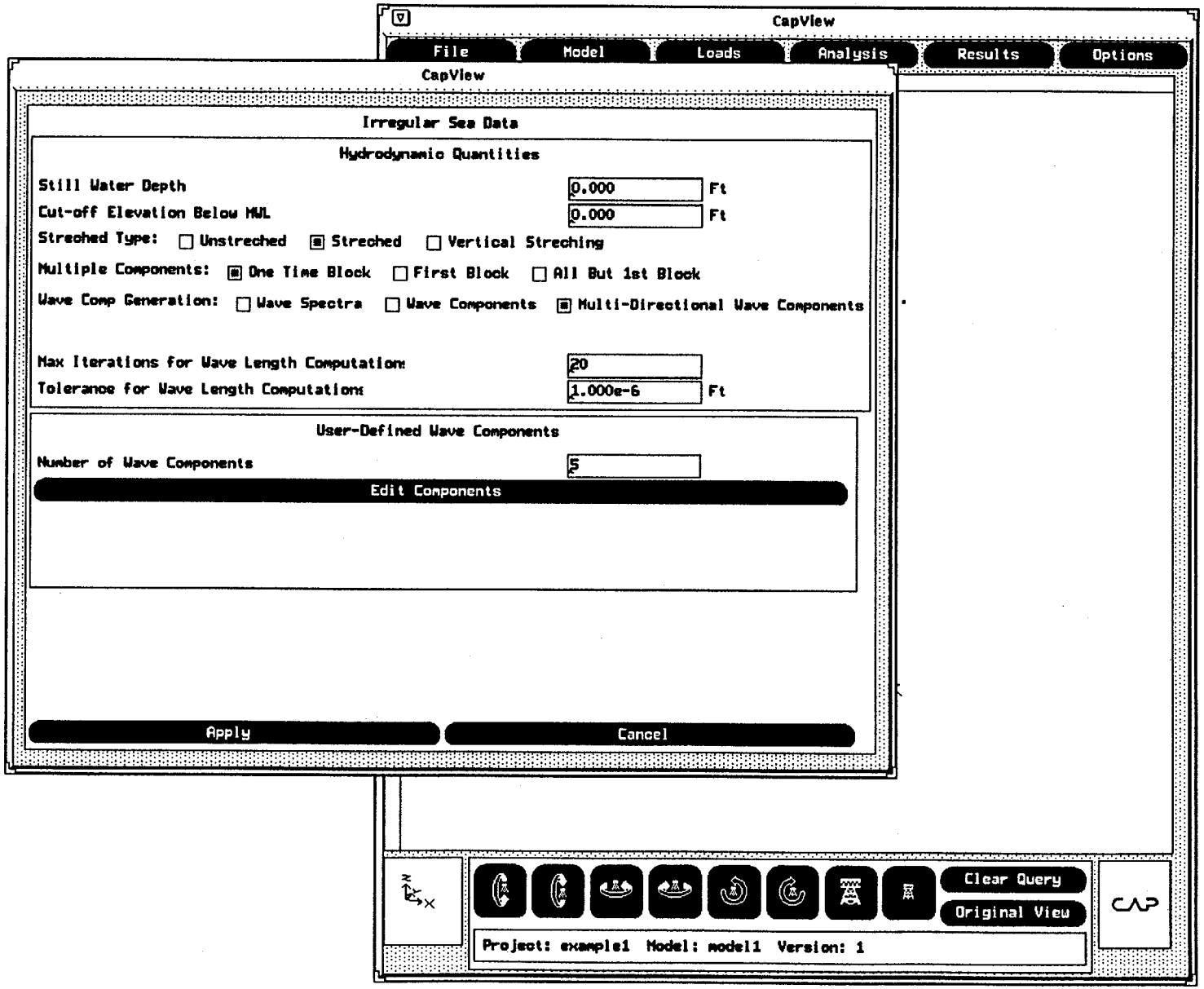

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Irregular Sea Data
Hydrodynamic Quantities
Still Water Depth
0.000
Ft
Cut-off Elevation Below MWL
0.000
Ft
Streched Type:
Unstreched
Streched
Vertical Streching
Multiple Components:
One Time Block
First Block
All But 1st Block
Wave Comp Generation:
Wave Spectra
Wave Components
Multi-Directional Wave Components
Max Iterations for Wave Length Computation
20
Tolerance for Wave Length Computation
1.000e-6
Ft
User-Defined Wave Components
Number of Wave Components
5
Edit Components
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-16

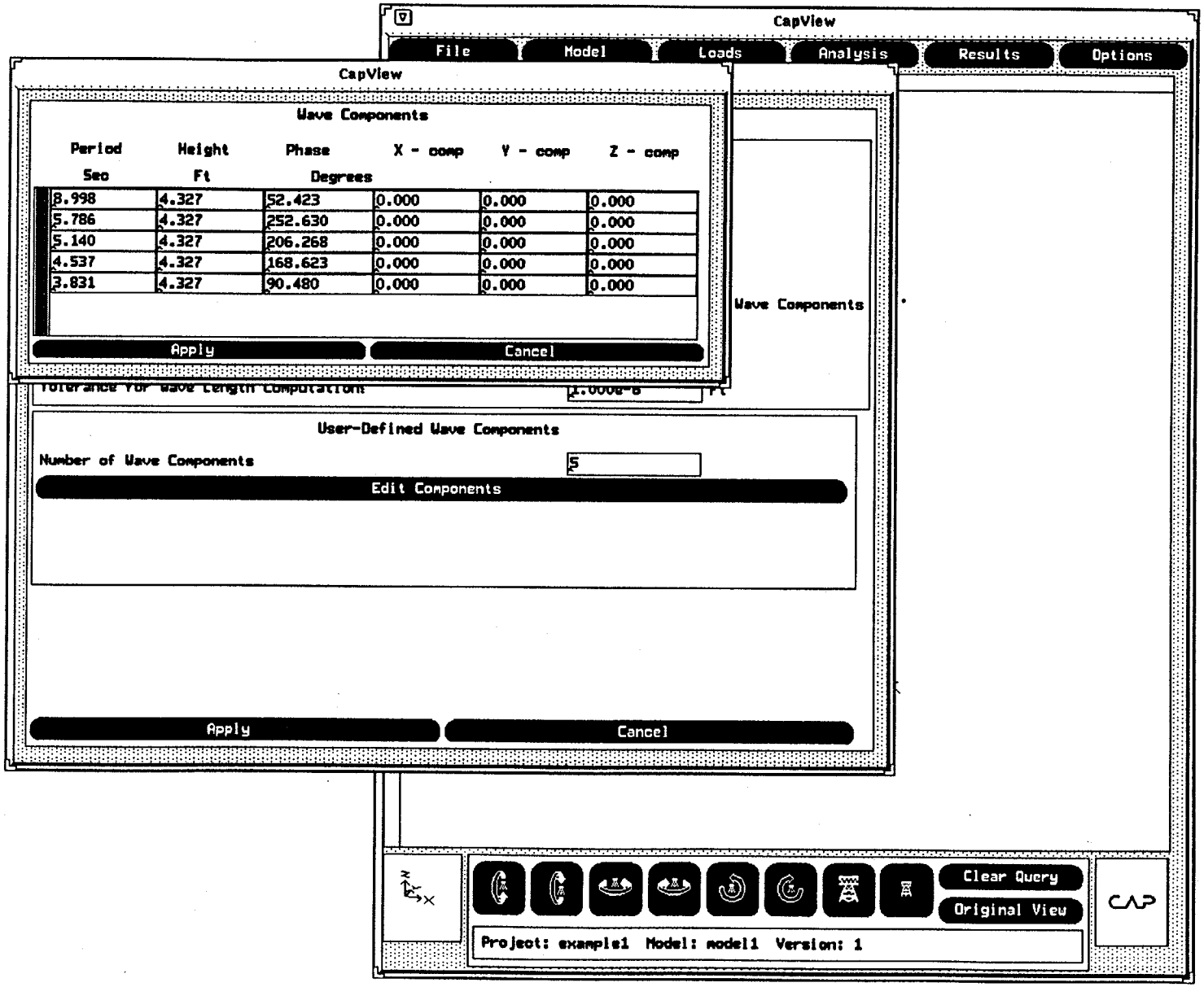

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Wave Components
Period Sec
Height Ft
Phase Degrees
X - comp
Y - comp
Z - comp
8.998 4.327 52.423 0.000 0.000 0.000
5.786 4.327 252.630 0.000 0.000 0.000
5.140 4.327 206.268 0.000 0.000 0.000
4.537 4.327 168.623 0.000 0.000 0.000
3.831 4.327 90.480 0.000 0.000 0.000
Apply
Cancel
Wave Components
User-Defined Wave Components
Number of Wave Components
5
Edit Components
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-17

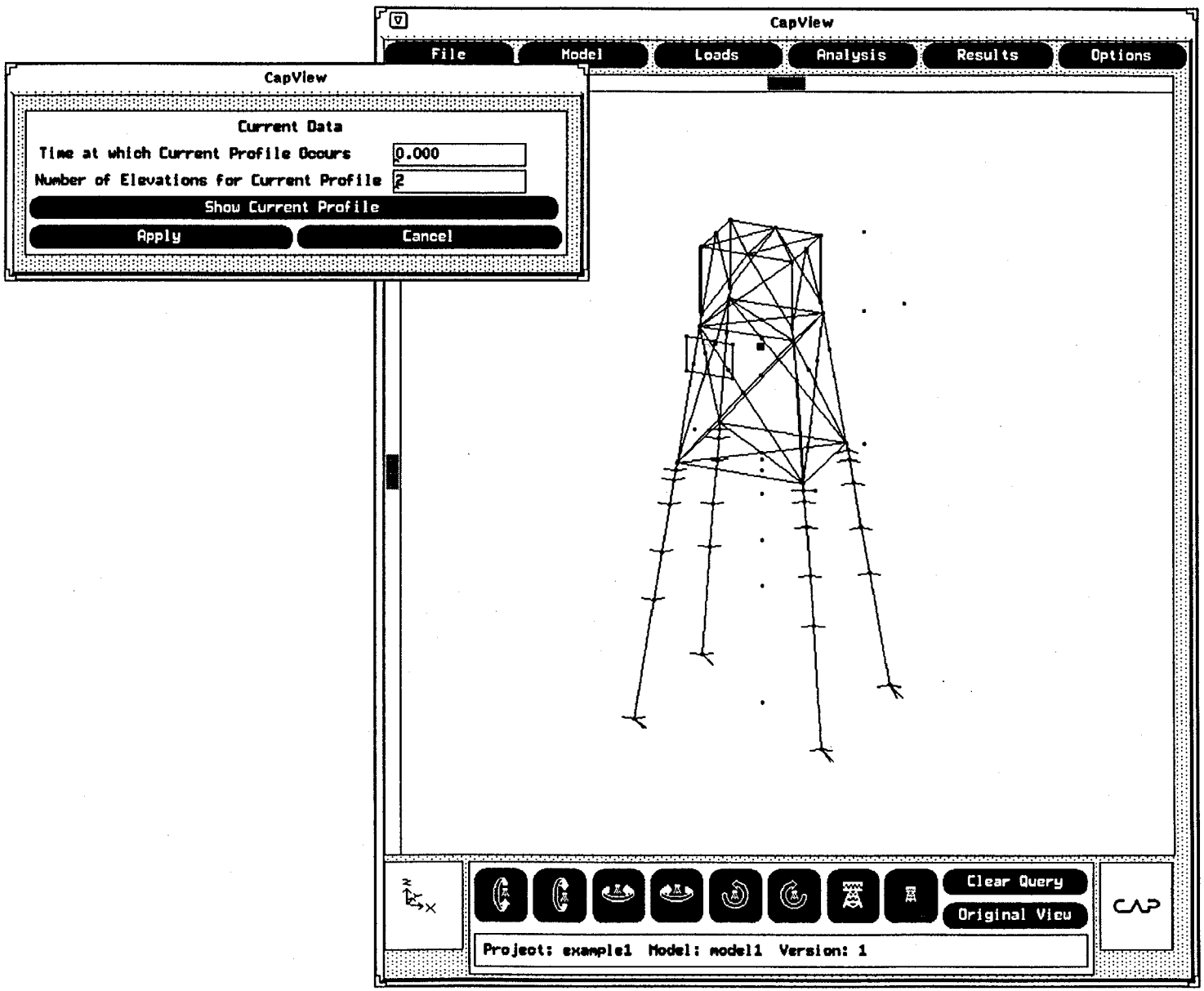

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Current Data
Time at which Current Profile Occurs
0.000
Number of Elevations for Current Profile
2
Show Current Profile
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-18

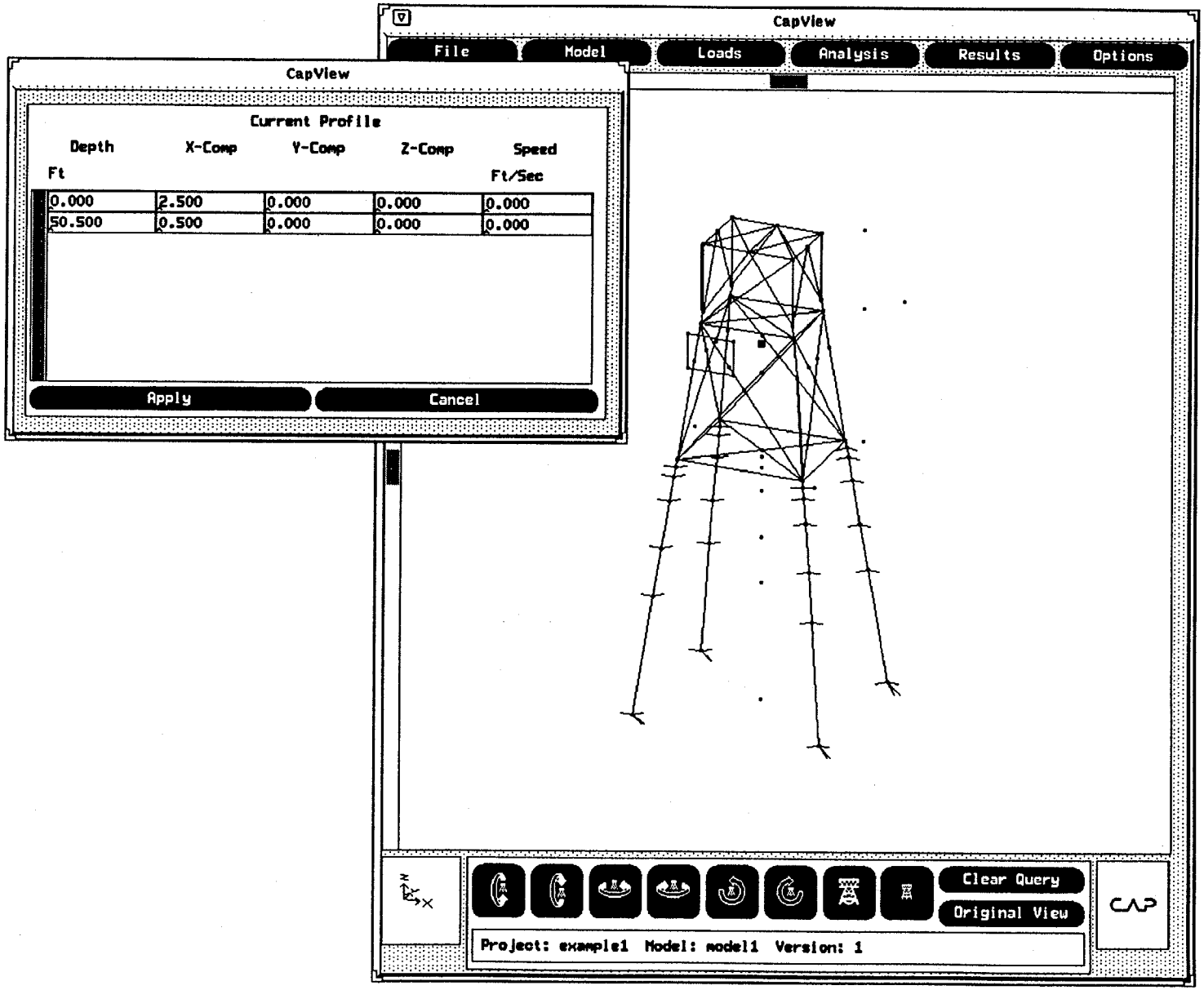

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Current Profile
Depth
X-Comp
Y-Comp
Z-Comp
Speed
Ft
Ft/Sec
0.000
2.500
0.000
0.000
0.000
50.500
0.500
0.000
0.000
0.000
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-19

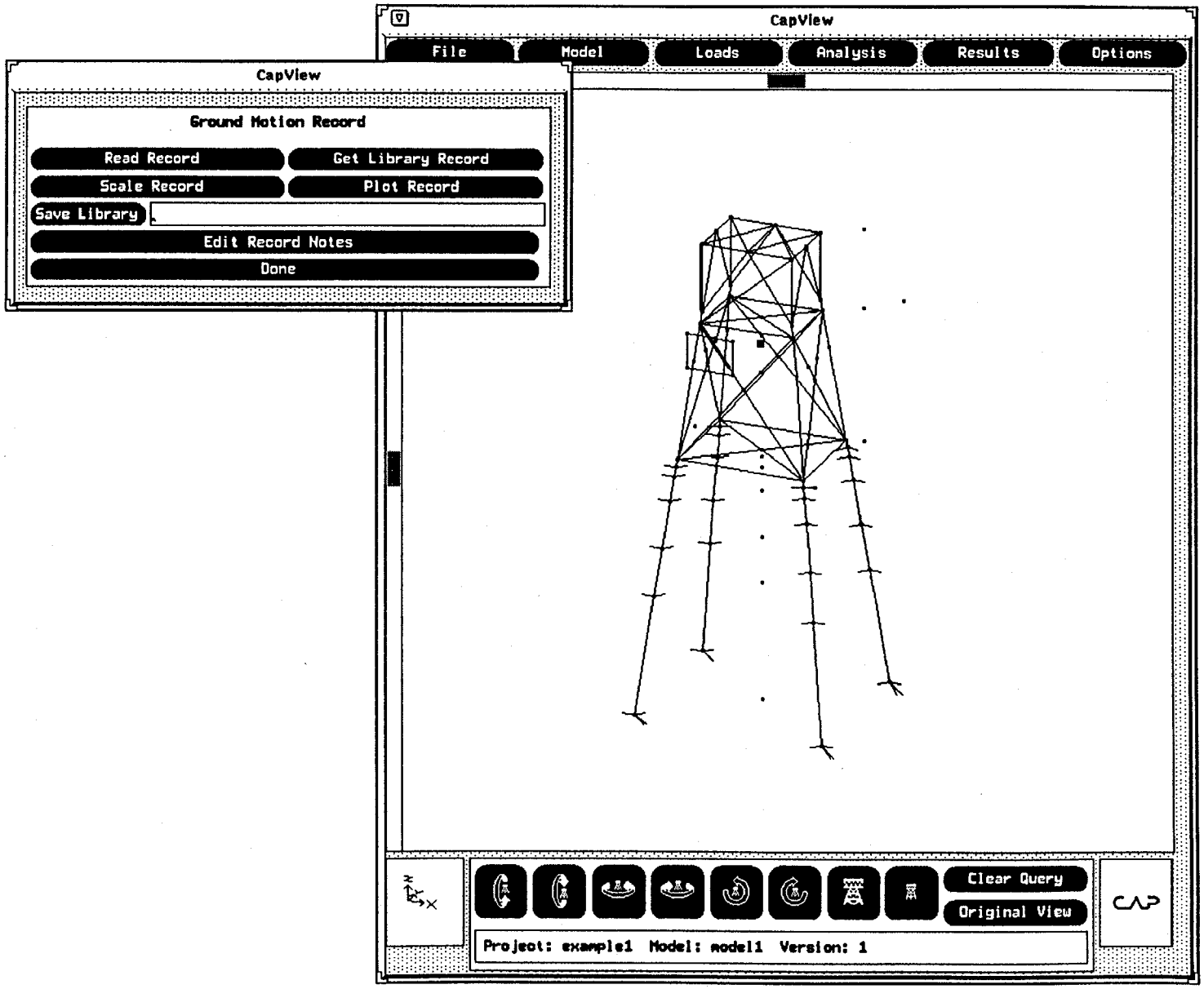

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Motion Record
Read Record
Get Library Record
Scale Record
Plot Record
Save Library
Edit Record Notes
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-20

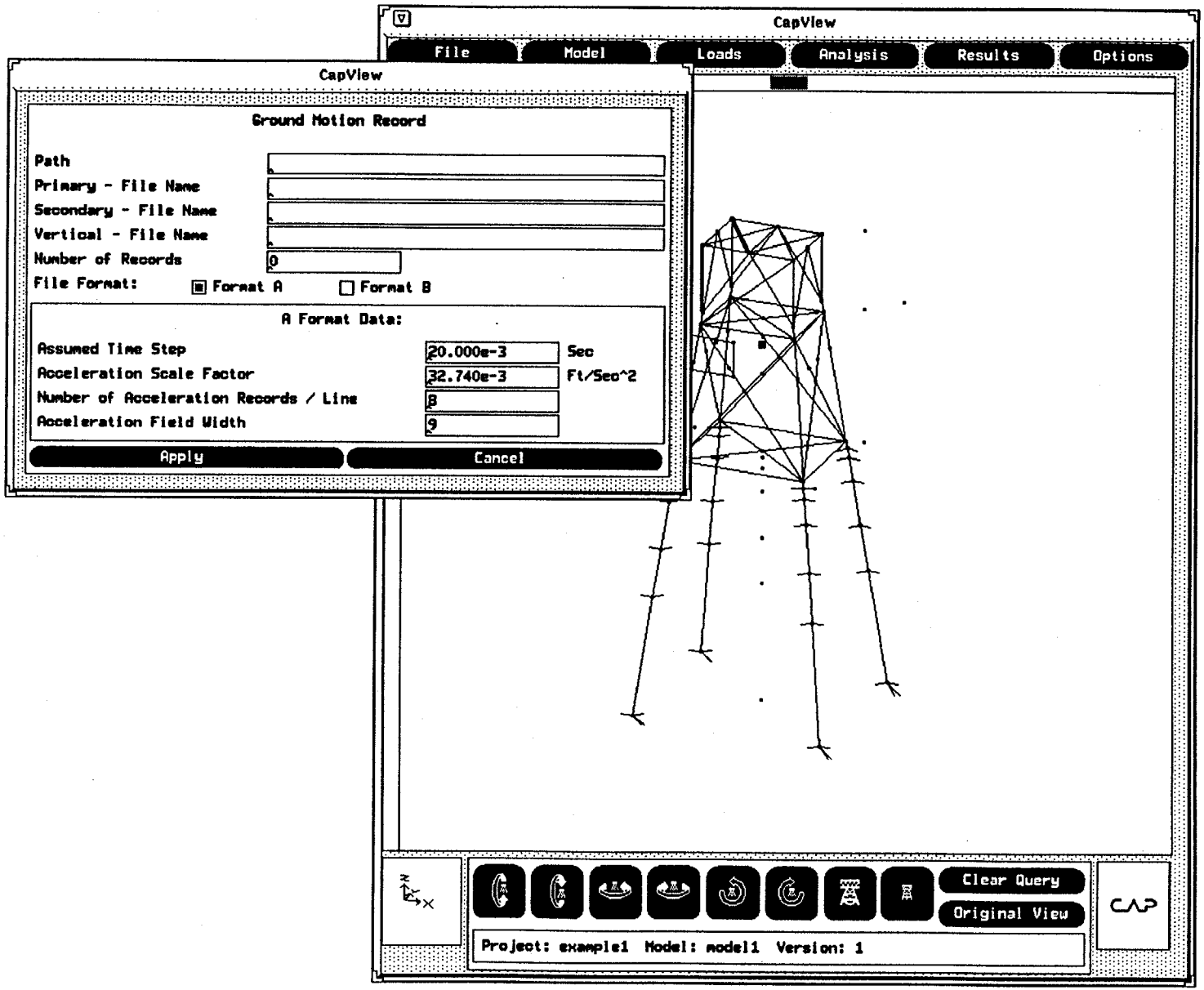

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Motion Record
Path
Primary - File Name
Secondary - File Name
Vertical - File Name
Number of Records
0
File Format:
Format A
Format B
A Format Data:
Assumed Time Step
20.000e-3
Sec
Acceleration Scale Factor
32.740e-3
Ft/Sec^2
Number of Acceleration Records / Line
8
Acceleration Field Width
9
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-21

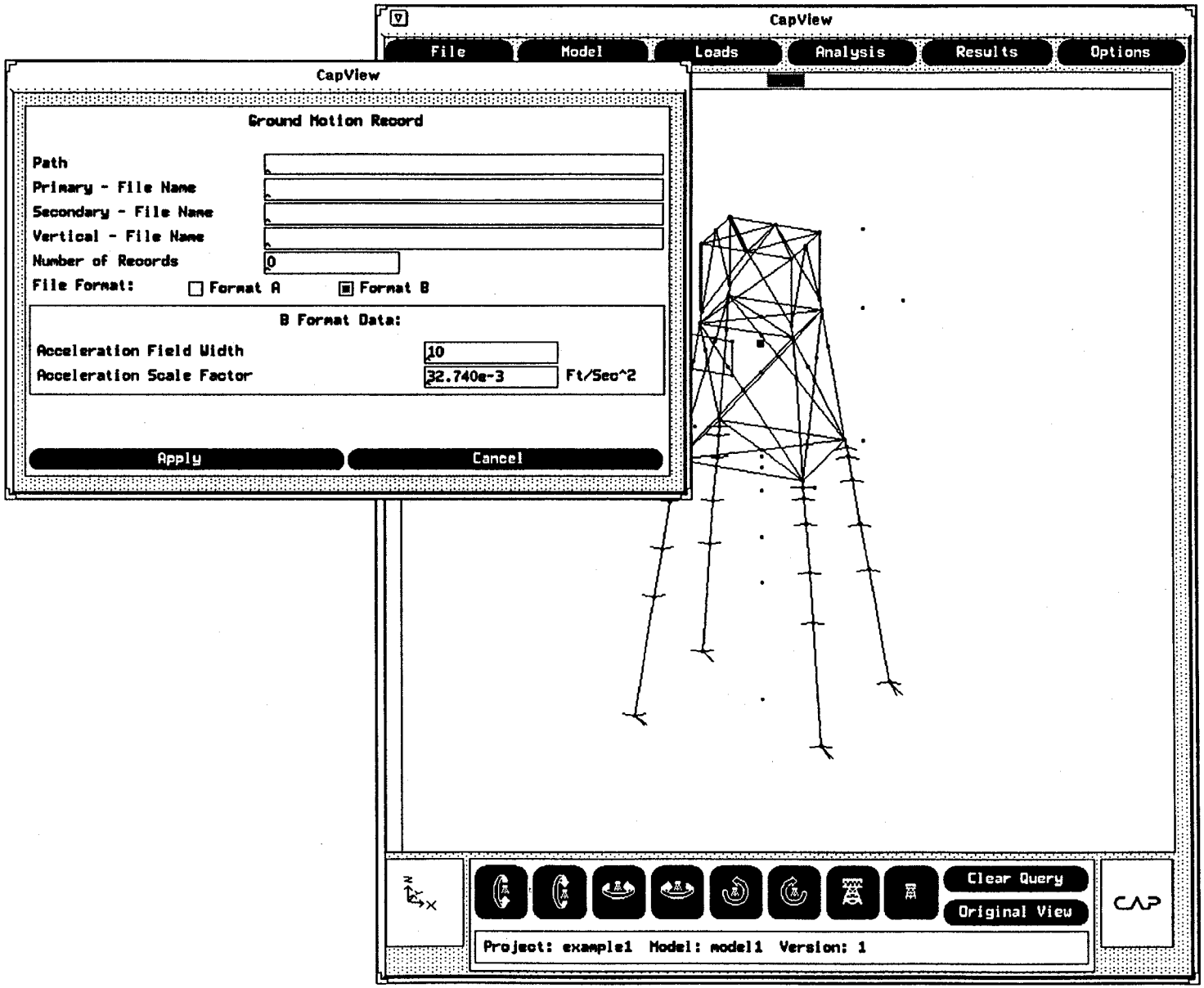

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Motion Record
Path
Primary - File Name
Secondary - File Name
Vertical - File Name
Number of Records
0
File Format:
Format A
Format B
B Format Data:
Acceleration Field Width
10
Acceleration Scale Factor
32.740e-3
Ft/Sec^2
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-22

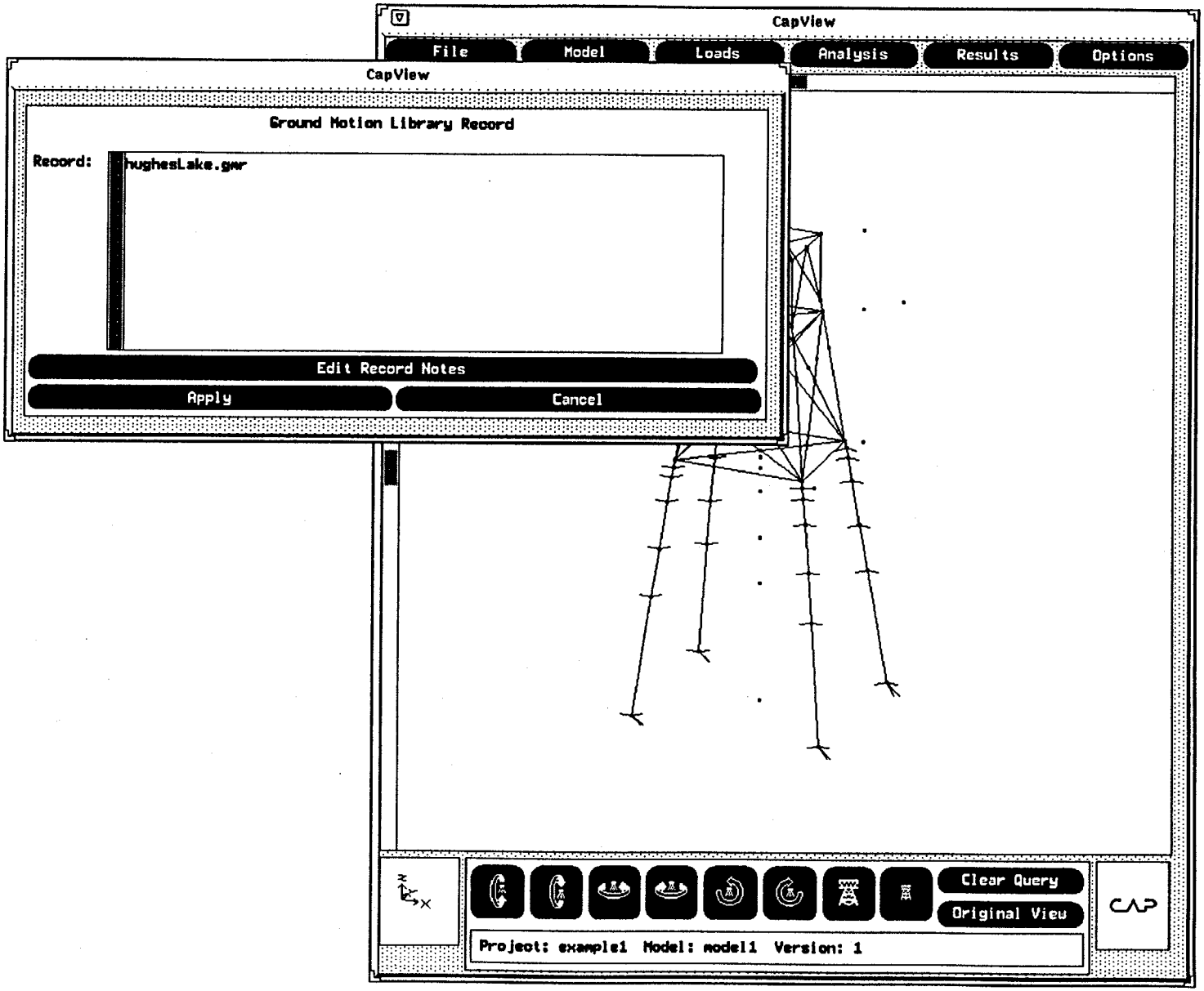

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Motion Library Record
Record:
hughesLake.gmr
Edit Record Notes
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-23

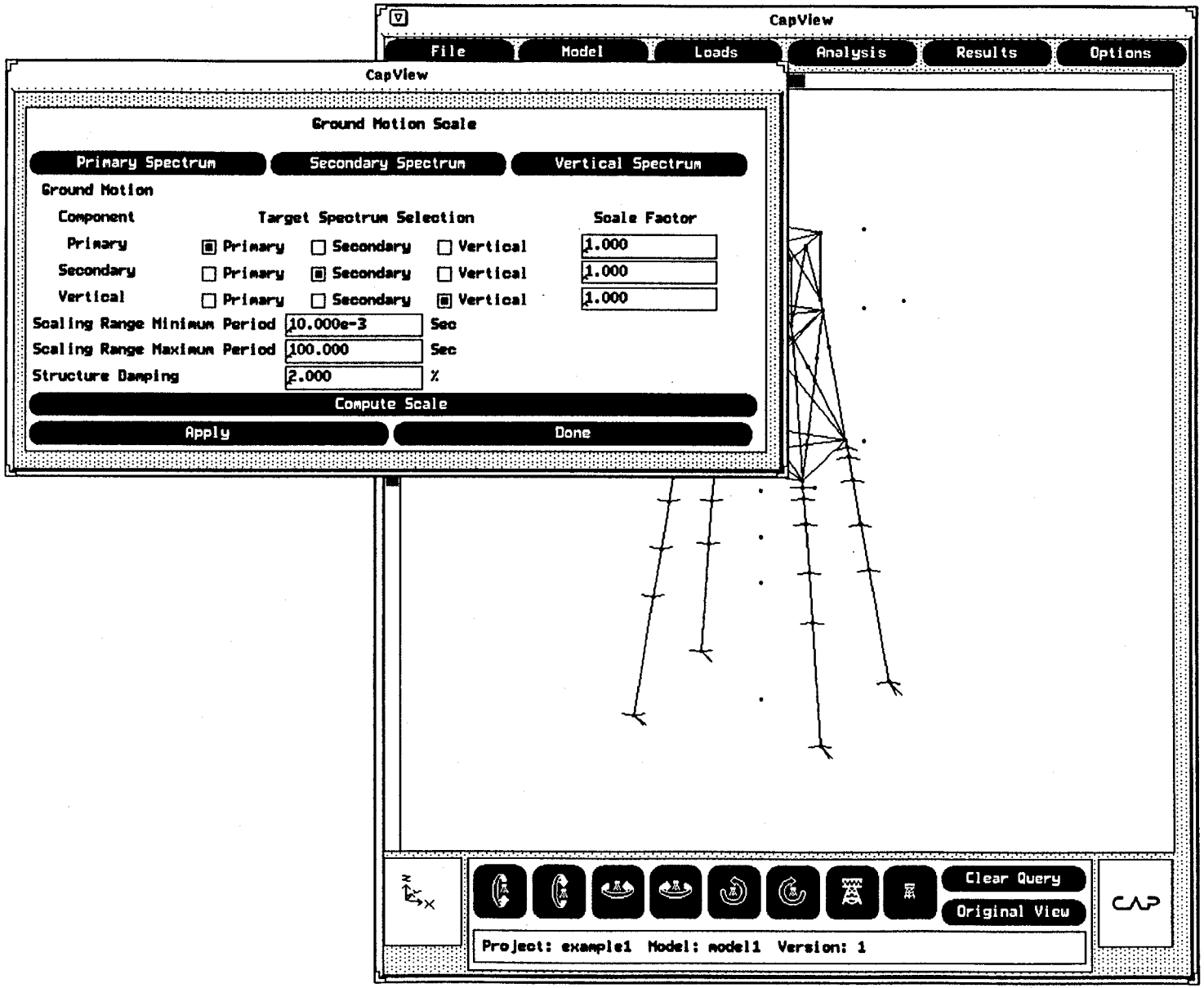

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Motion Scale
Primary Spectrum
Secondary Spectrum
Vertical Spectrum
Ground Motion
Component
Target Spectrum Selection
Scale Factor
Primary
Primary
Secondary
Vertical
1.000
Secondary
Primary
Secondary
Vertical
1.000
Vertical
Primary
Secondary
Vertical
1.000
Scaling Range Minimum Period
10.000e-3
Sec
Scaling Range Maximum Period
100.000
Sec
Structure Damping
2.000
%
Compute Scale
Apply
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-24

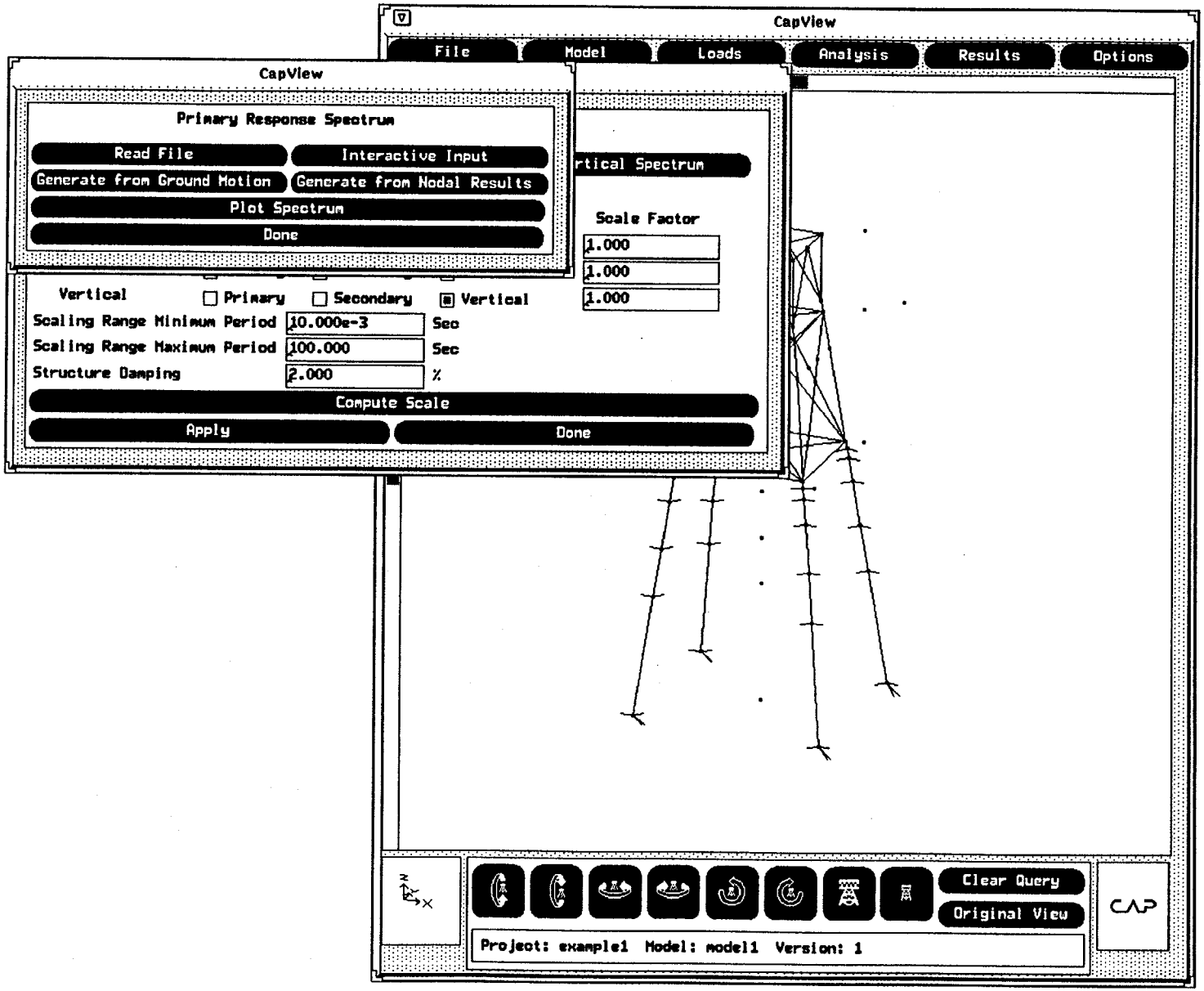

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Primary Response Spectrum
Read File
Interactive Input
Generate from Ground Motion
Generate from Nodal Results
Plot Spectrum
Done
rtical Spectrum
Scale Factor
1.000
1.000
1.000
Vertical
Primary
Secondary
Vertical
Scaling Range Minimum Period
10.000e-3
Sec
Scaling Range Maximum Period
100.000
Sec
Structure Damping
2.000
%
Compute Scale
Apply
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-25

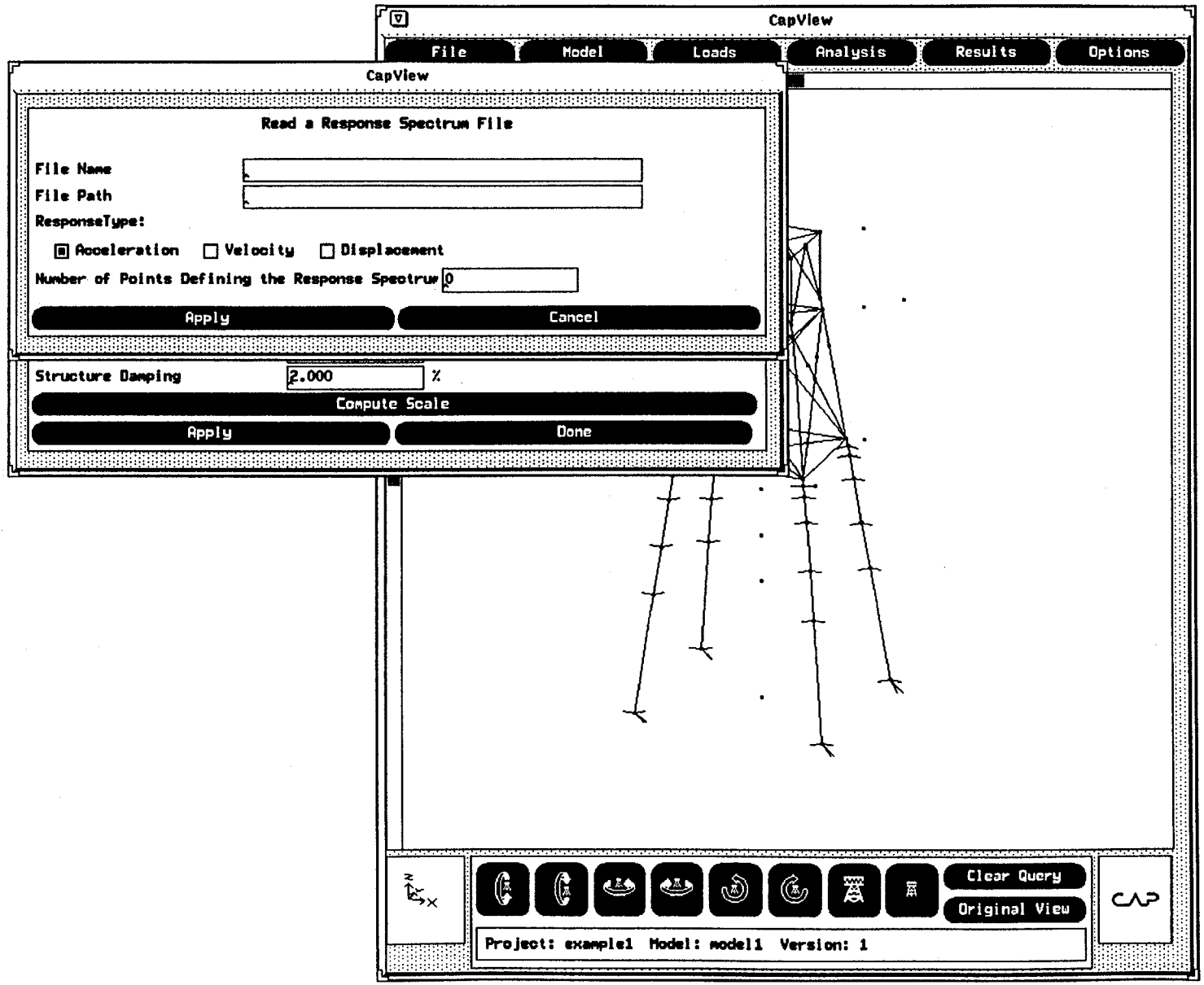

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Read a Response Spectrum File
File Name
File Path
ResponseType:
Acceleration
Velocity
Displacement
Number of Points Defining the Response Spectrum
0
Apply
Cancel
Structure Damping
2.000
%
Compute Scale
Apply
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-26

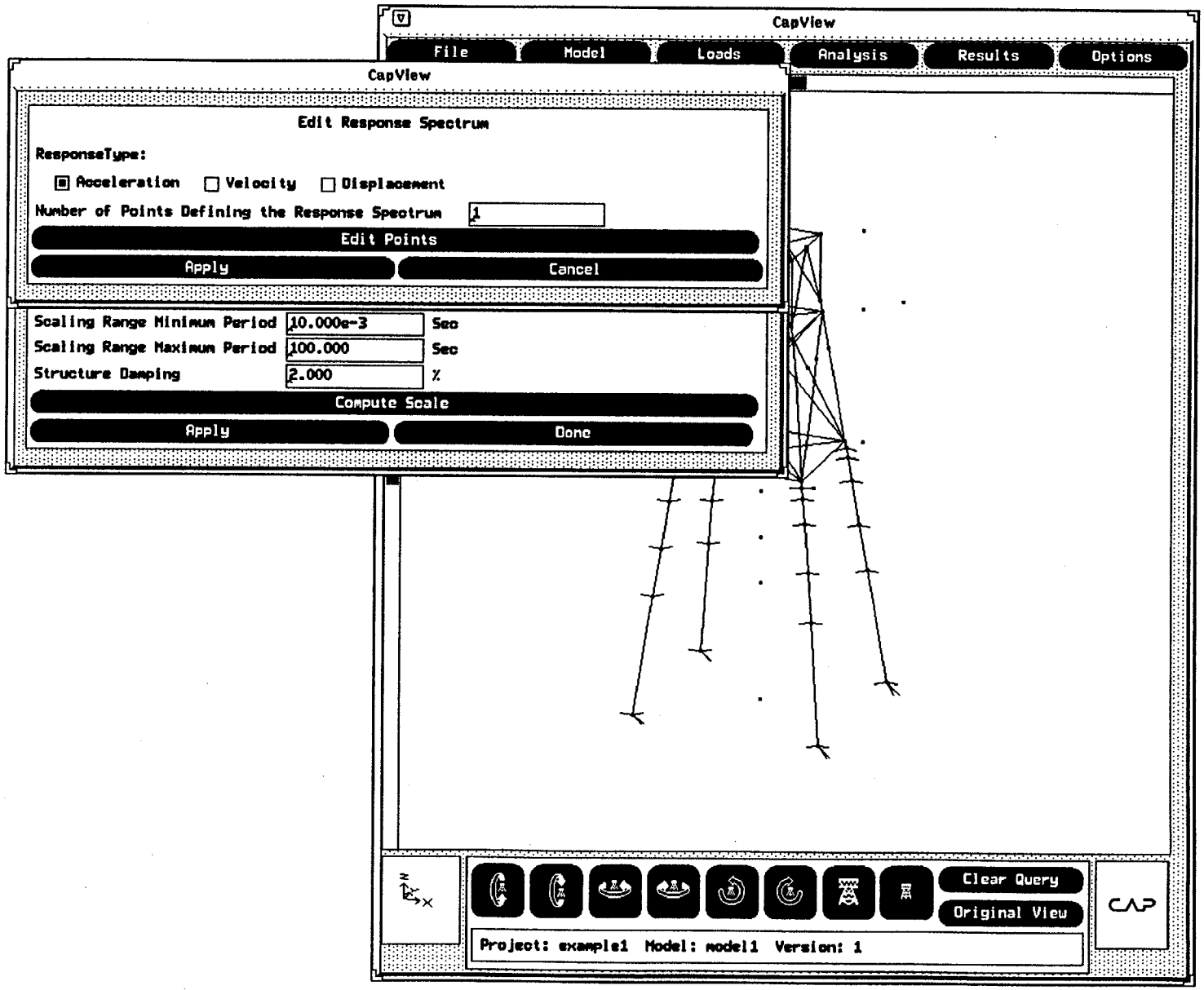

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Edit Response Spectrum
ResponseType:
Acceleration
Velocity
Displacement
Number of Points Defining the Response Spectrum
1
Edit Points
Apply
Cancel
Scaling Range Minimum Period
10.000e-3
Sec
Scaling Range Maximum Period
100.000
Sec
Structure Damping
2.000
%
Compute Scale
Apply
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 9-27

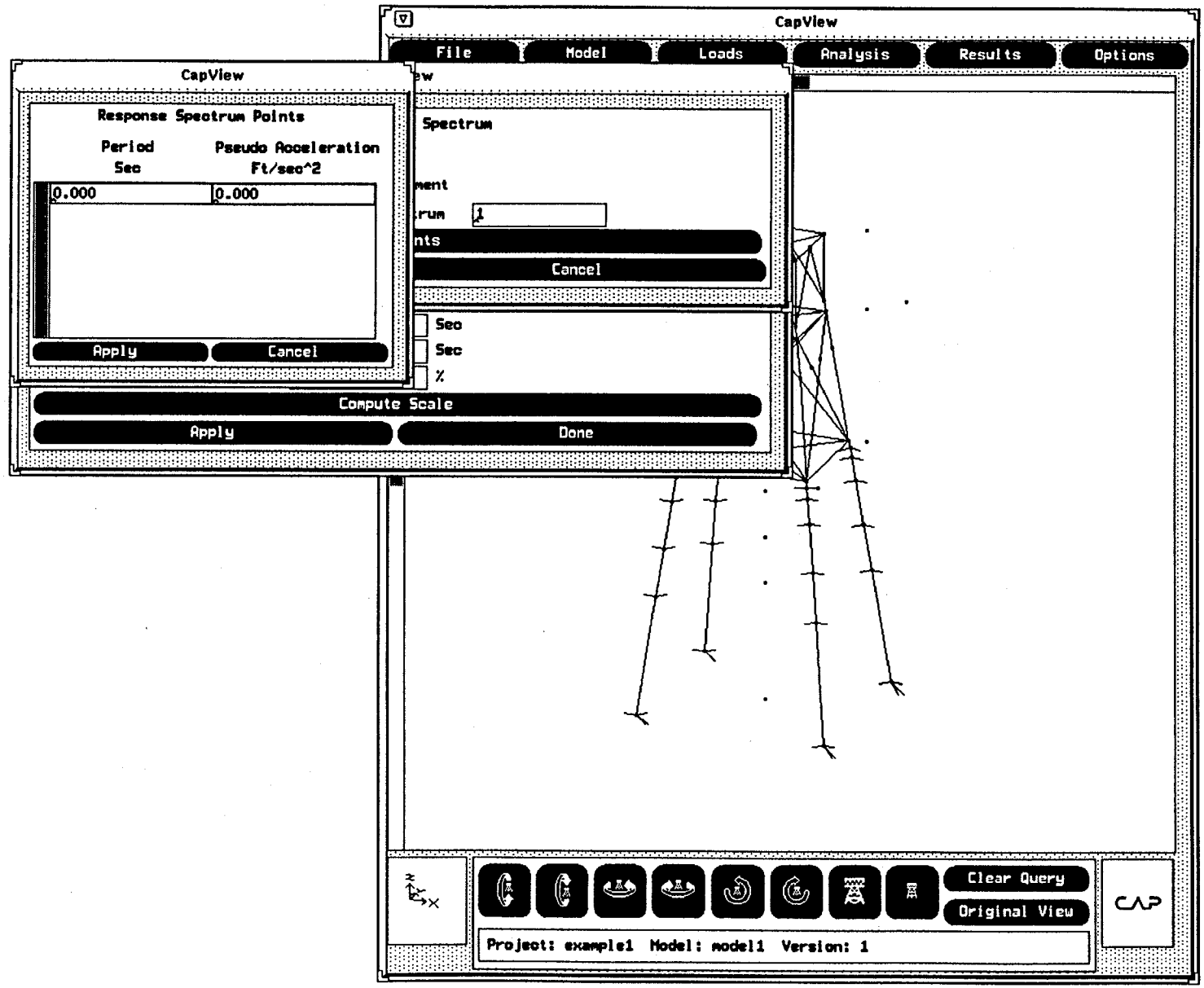

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Response Spectrum Points
Period
Sec
Pseudo Acceleration
Ft/sec^2
0.000
0.000
Apply
Cancel
Spectrum
1
Cancel
Sec
Sec
%
Compute Scale
Apply
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-28

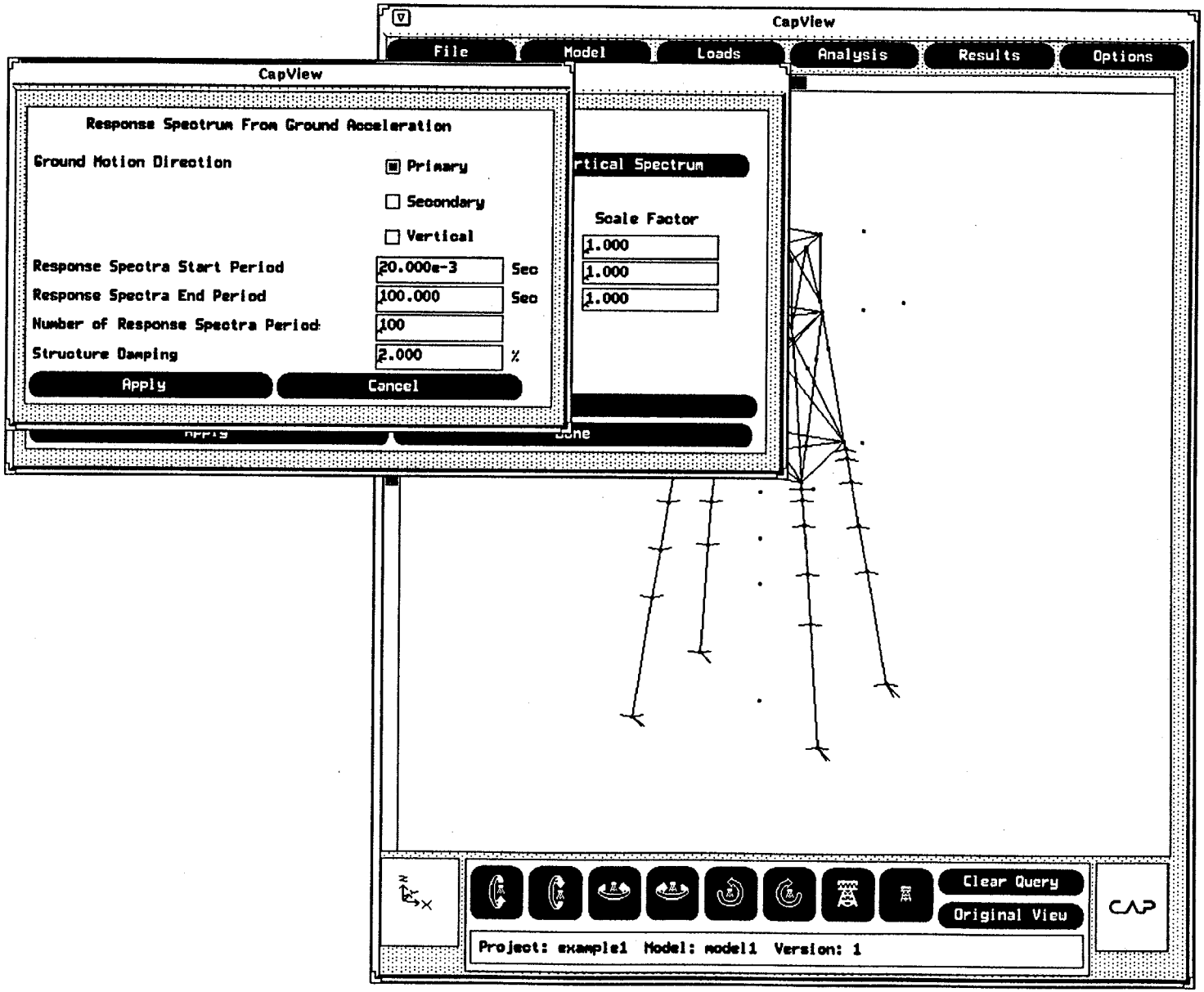

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Response Spectrum From Ground Acceleration
Ground Motion Direction
Primary
Secondary
Vertical
Response Spectra Start Period
20.000e-3
Sec
Response Spectra End Period
100.000
Sec
Number of Response Spectra Period
100
Structure Damping
2.000
%
Apply
Cancel
rtical Spectrum
Scale Factor
1.000
1.000
1.000
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 9-29

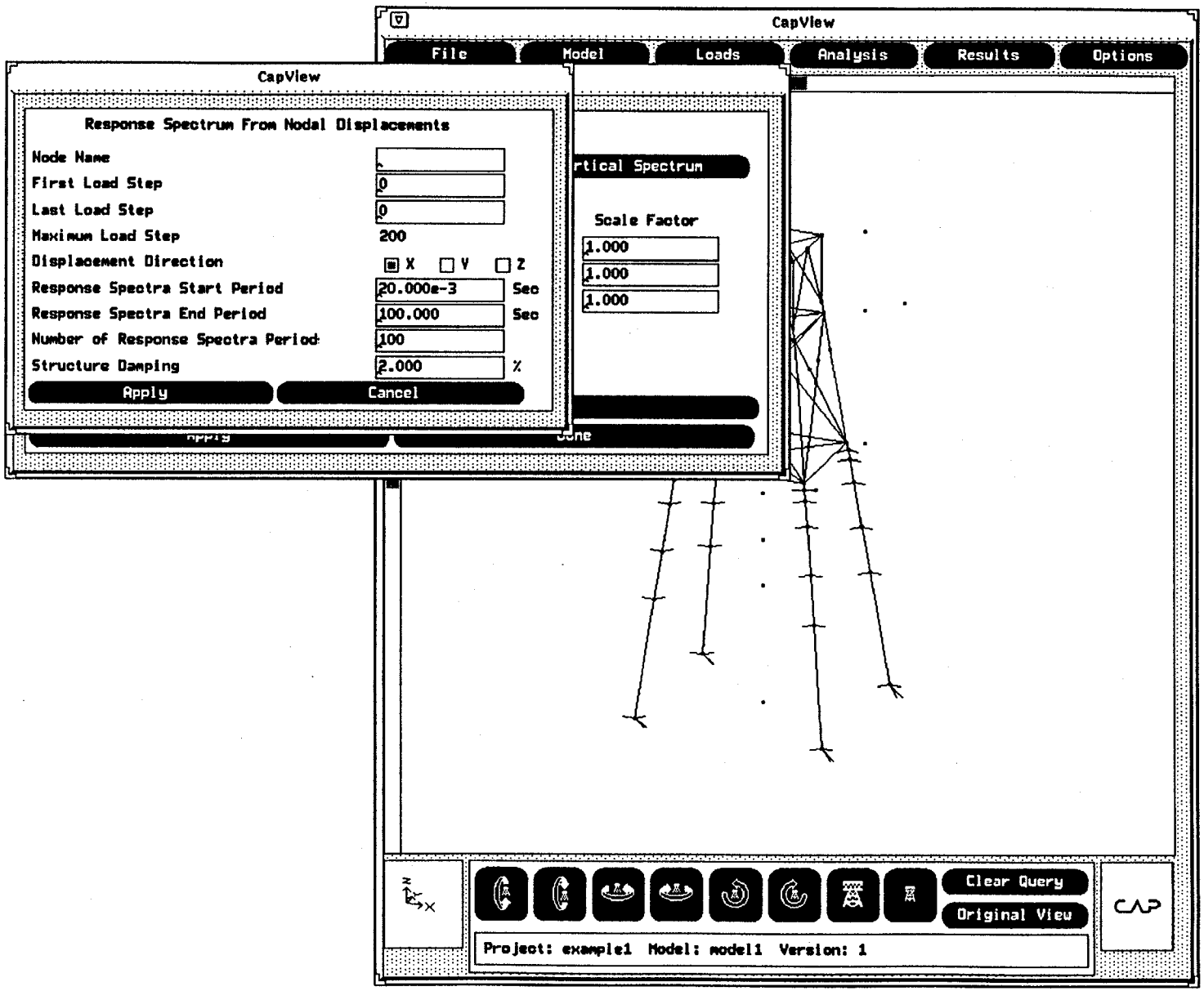

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Response Spectrum From Nodal Displacements
Node Name
First Load Step
0
Last Load Step
0
Maximum Load Step
200
Displacement Direction
X
Y
Z
Response Spectra Start Period
20.000e-3
Sec
Response Spectra End Period
100.000
Sec
Number of Response Spectra Period
100
Structure Damping
2.000
%
Apply
Cancel
rtical Spectrum
Scale Factor
1.000
1.000
1.000
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-30

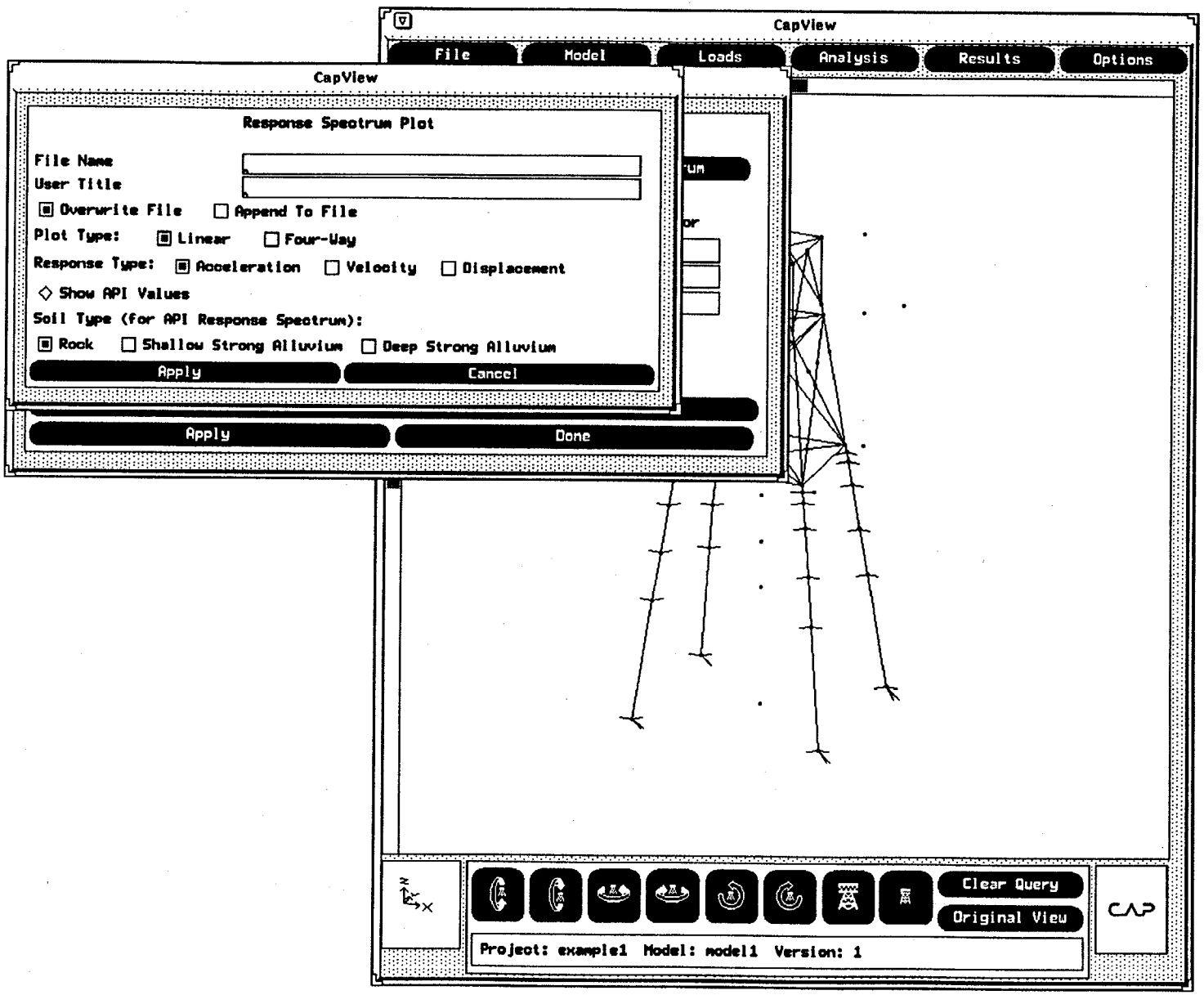

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Response Spectrum Plot
File Name
User Title
Overwrite File
Append To File
Plot Type:
Linear
Four-Way
Response Type:
Acceleration
Velocity
Displacement
Show API Values
Soil Type (for API Response Spectrum):
Rock
Shallow Strong Alluvium
Deep Strong Alluvium
Apply
Cancel
Apply
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-31

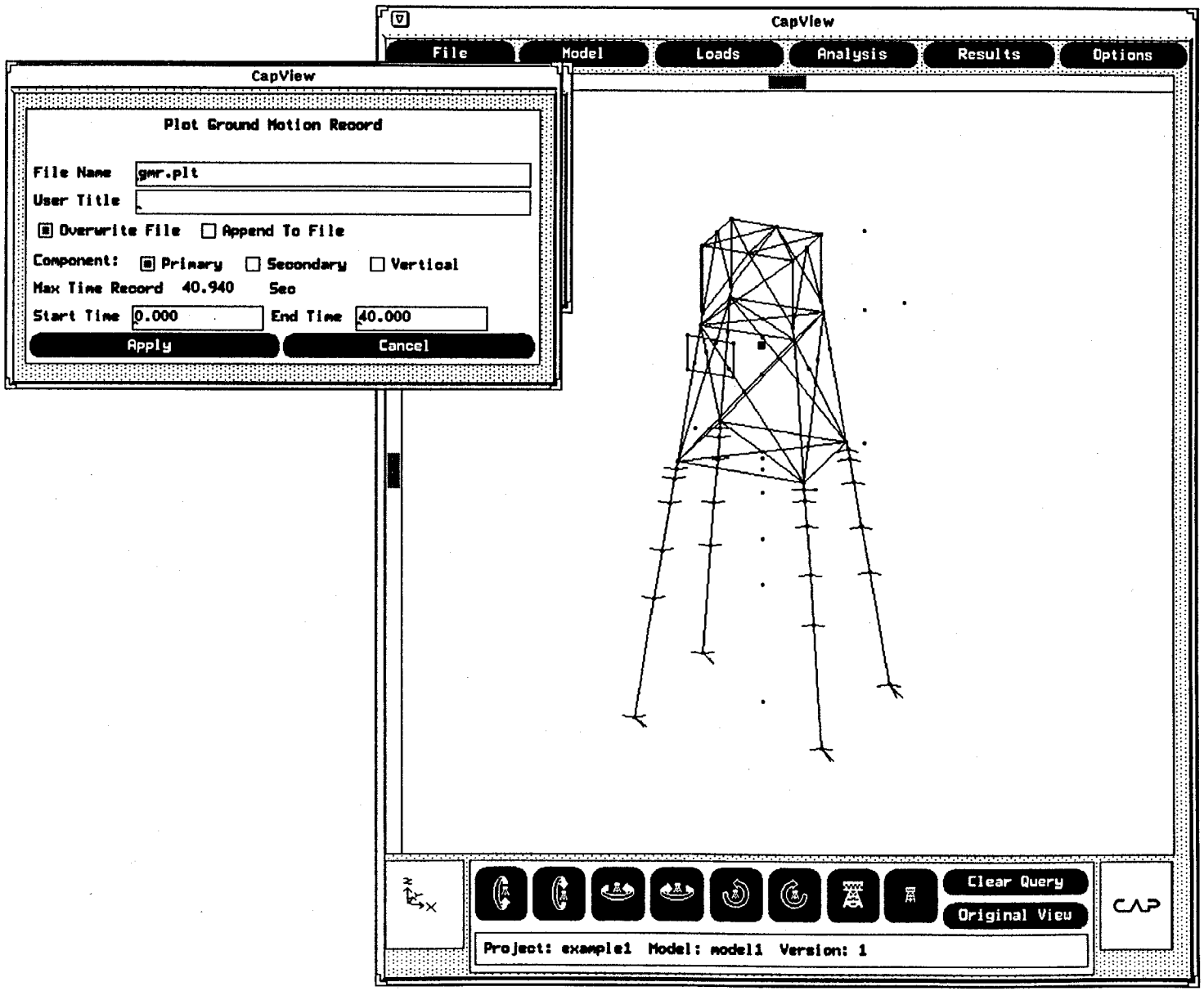

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Plot Ground Motion Record
File Name
gmr.plt
User Title
Overwrite File
Append To File
Component:
Primary
Secondary
Vertical
Max Time Record 40.940 Sec
Start Time
0.000
End Time
40.000
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 9-32

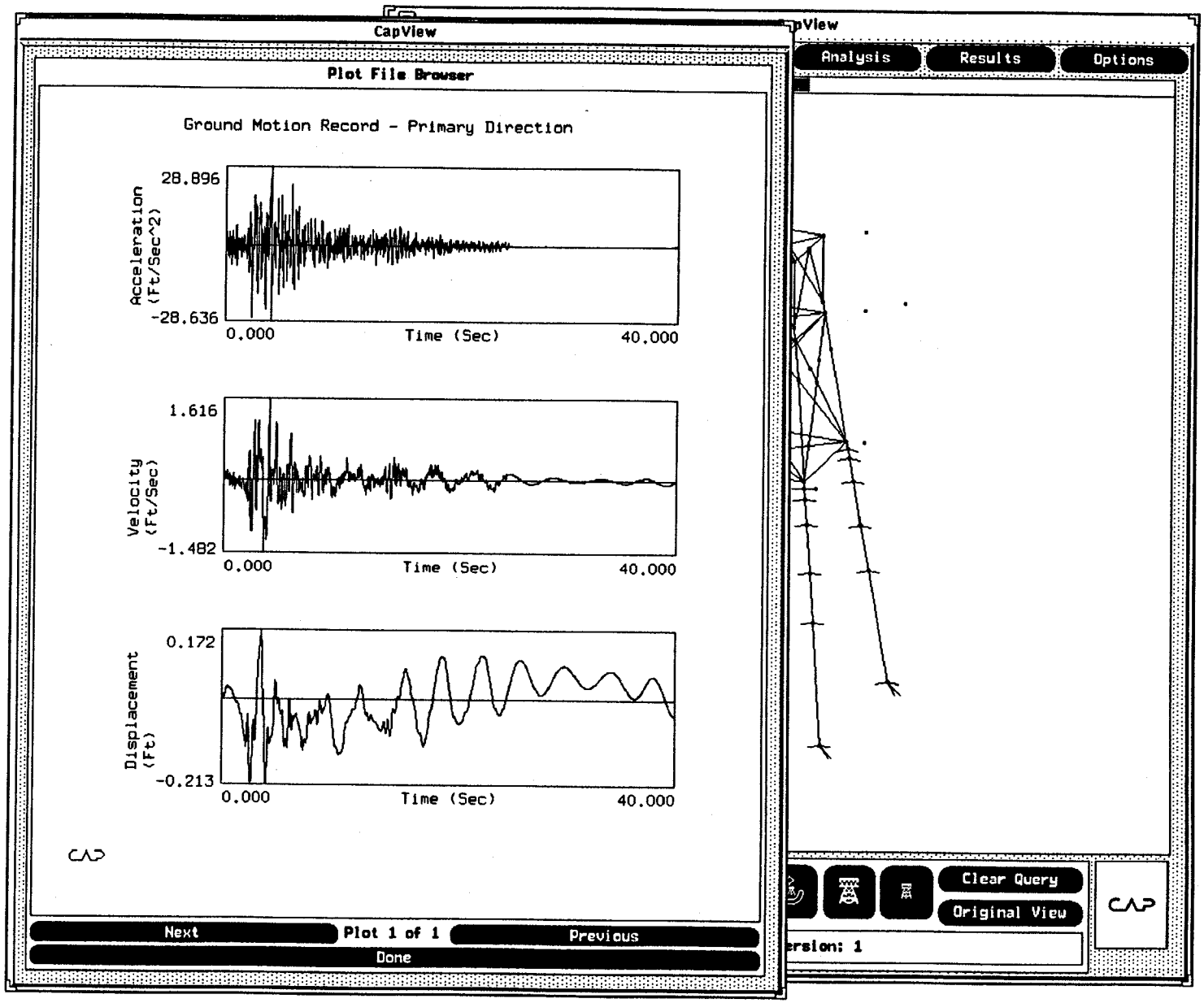

CapView
Plot File Browser
Ground Motion Record - Primary Direction
28.896
Acceleration (Ft/Sec^2)
-28.636
0.000
Time (Sec)
40.000
1.616
Velocity (Ft/Sec)
-1.482
0.000
Time (Sec)
40.000
0.172
Displacement (Ft)
-0.213
0.000
Time (Sec)
40.000
Next
Plot 1 of 1
Previous
Done
Analysis
Results
Options
Clear Query
Original View

Figure 9-33

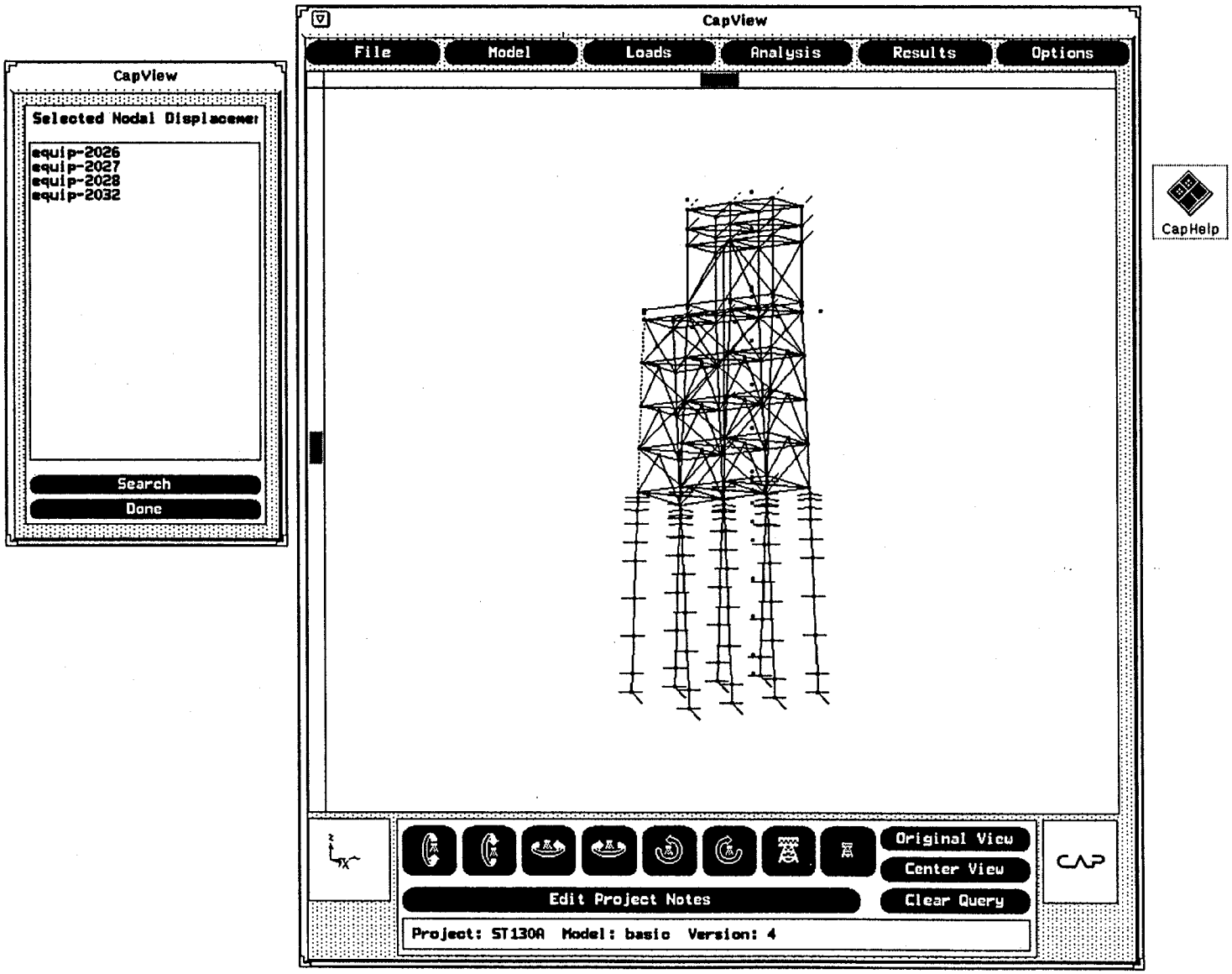

CapView
Selected Nodal Displacement
equip-2026
equip-2027
equip-2028
equip-2032
Search
Done
CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 9-34

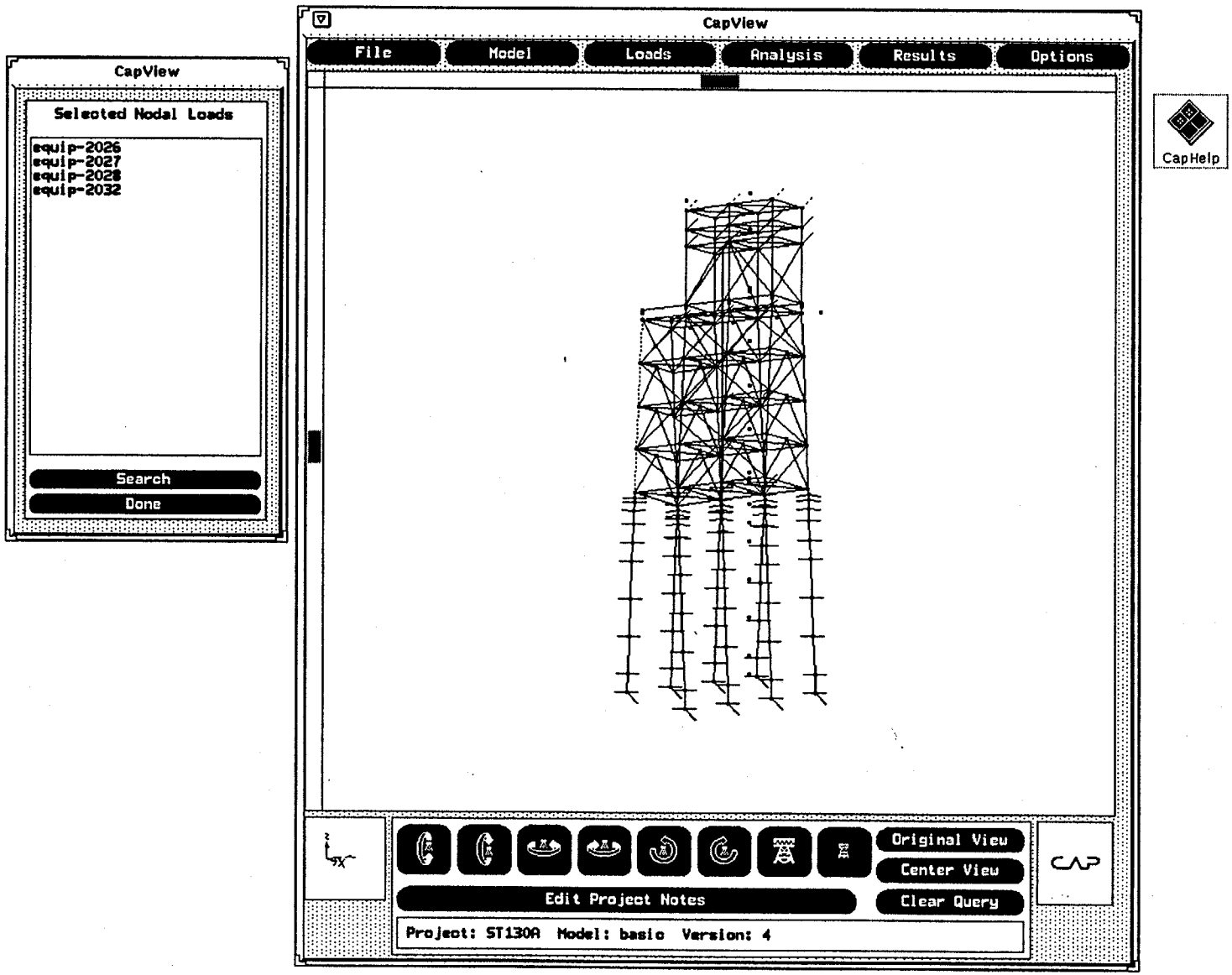

CapView
Selected Nodal Loads
equip-2026
equip-2027
equip-2028
equip-2032
Search
Done
CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 9-35

10.0 THE ANALYSIS MENU

The Analysis menu presents the user with selections that will help in defining analysis options, performing data checks, running a SeaStar analysis, creating a SeaStar input file, and editing various analysis files. When the Analysis menu is selected, the following menu is displayed (Figure 10-1):

Edit Macros...
Perform Data Check
Run SeaStar Analysis
Create SeaStar File
Edit Files...

10.1 Edit Analysis Macros

The first selection of the Analysis menu will display the Edit Analysis Macros dialogue box (Figure 10-2). This dialogue box allows the user to add, edit and delete analysis macros. The Edit Analysis Macros dialogue box consists of the following:

- A button labeled "Preconfigued Macros" which when pressed will activate an Analysis Options dialogue box (see 10.1.1) where a preconfigured analysis macro can be selected from a list of macros.
- A combo box labeled "Restart Mode" where the user can choose between one of the following resart options:

  New Run That Can't be Continued

  New Run That Can be Continued

  Continue a Previous Run
- An edit field labeled "Restart Title" where the restart title is entered. This edit field is hidden when the Restart Mode is a New Run That Can't Be Continued. Note that in order to be able to continue a run, the Restart Title must be the same as in the run to be continued.
- A combo box labeled "Analysis Mode" where the user can choose betweenone of the following analysis modes:

  Generate Pushover Load Pattern

  Dynamic Pushover Analysis

  Response Spectrun Analysis

  Other
- A button labeled "Edit Response Spectrum Analysis Data" which when pressed will activate a Response Spectrum Analysis Data dialogue box (10.1.1.14.1). This button is only visible when the Analysis Mode is Response Spectrun Analysis.
- A checkbox labeled "Apply Wave Loads" where the application of wave loads can be toggled on or off.
- A combo box labeled "Add Macro Type" where the user can choose between the following analysis macro types:

  Header/End Analysis (NONE)

  Static Analysis (STAT)

  Event to Event Dynamic Analysis (DYN1)

  Iterative Dynamic Analysis (DYN2)

  Frequency Analysis (EIGN)

  Save Results for Restart (SAVE)
- A combo box labeled "Add Macro Location" where the user can choose the location on the list of analysis macros to insert the new analysis macro:

  Begining

  End

  Before Current

  After Current
- A button labeled "Add Macro" which when pressed will add the analysis macro type selected in the add macro type combo box to the analysis macro list. It will be added to the list at the location specified in the add

macro location combo box.

- A button labeled "Edit Macro" which when pressed will bring up a dialogue box where the user can edit the analysis macro that is currently highlighted in the macro list combo box.
- A button labeled "Delete Macro" which when pressed will delete the currently highlighted analysis macro from the analysis macro list.
- A combo box labeled "Macro List" where the user can view the names of the analysis macros on the list. This is where analysis macros may be highlighted for editing or deletion.
- A button labeled "Done" which when pressed will close the Edit Analysis Macros dialogue box.

10.1.1 Analysis Options Dialogue Box (Figure 10-3)

When the Preconfigured Macros button is pressed the Analysis Options dialogue box is displayed. This is where the user may select one of a number of preconfigured analysis macros. The Analysis Options dialogue box consists of the following:

- A radio list labeled "Analysis Macro" where the user can select between one of the following preconfigured analysis macros:

  Static Analysis

  Generate Pushover Load Pattern

  Static Pushover Analysis

  Dynamic Pushover Analysis (1 Step Solution)

  Dynamic Pushover Analysis (2 Step Solution)

  Response Spectrum Analysis

  Variable Time Step Dynamic

  Constant Time Step Dynamic
- A button labeled "Load Conditions" which will activate the Load Conditions dialogue box (see below). This is where the load conditions are set.
- A button labeled "Run Parameters" which will activate the Run Parameters dialogue box (see below). This is where the run parameters are set.
- Apply button. Selecting this button will apply the selections currently displayed close the Analysis Options dialogue box, and run Data Check. This button must be selected to make any changes effective.
- Default button. Selecting this button will set all of the preconfigured analysis macros to their default values.
- Cancel button. Selecting this button will close the Analysis Options dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.1 Load Cases Dialogue Box (when the predefined macro type is Static Analysis) (Figure 10-4)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Static Analysis. The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

An edit field labeled "Scale Factors" where the value for Scale Factors is entered.

- A checkbox labeled "Gravity Loads" which toggles Gravity Loads on and off.
- An edit field labeled "Gravity Loads" where the value for Gravity Loads scale factoe is entered.
- A checkbox labeled "Buoyancy Loads" which toggles Buoyancy Loads on and off.
- An edit field labeled "Buoyancy Loads" where the value for Buoyancy Loads scale factor is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.2 Load Cases Dialogue Box (when the predefined macro type is Generate Pushover Load Pattern) (Figure 10-5)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Generate Pushover Load Pattern. The Load Cases dialogue box consists of the following:

- A radio button list labeled "Dynamic Load Cases" which allows the user to choose between:

  Wave Load

  Current Load
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.3 Load Cases Dialogue Box (when the predefined macro type is Static Pushover Analysis) (Figure 10-54)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Static Pushover Analysis. The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- A checkbox labeled "Gravity Loads" which toggles Gravity Loads on and off.

- An edit field labeled "Gravity Loads" where the value for Gravity Loads scale factor is entered.
- A checkbox labeled "Buoyancy Loads" which toggles Buoyancy Loads on and off.
- An edit field labeled "Buoyancy Loads" where the value for Buoyancy Loads scale factor is entered.
- A scrollable list labeled "Pushover Load Cases" where for each load case there is:

  A checkbox labeled "Pushover Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.4 Load Cases Dialogue Box (when the predefined macro type is Dynamic Pushover Analysis (1 Step Solution)) (Figure 10-6)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Dynamic Pushover Analysis (1 Step Solution). The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- A checkbox labeled "Gravity Loads" which toggles Gravity Loads on and off.
- An edit field labeled "Gravity Loads" where the value for Gravity Loads scale factor is entered.
- A checkbox labeled "Buoyancy Loads" which toggles Buoyancy Loads on and off.
- An edit field labeled "Buoyancy Loads" where the value for Buoyancy Loads scale factor is entered.
- A checkbox labeled "Pushover Load Pattern" which toggles Pushover Load Pattern on and off.
- A scrollable list labeled "Dynamic Load Cases" where for each dynamic load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.

- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.5 Load Cases Dialogue Box (when the predefined macro type is Dynamic Pushover Analysis (2 Step Solution) (Figure 10-6)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Dynamic Pushover Analysis (2 Step Solution). The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- A checkbox labeled "Gravity Loads" which toggles Gravity Loads on and off.
- An edit field labeled "Gravity Loads" where the value for Gravity Loads scale factor is entered.
- A checkbox labeled "Buoyancy Loads" which toggles Buoyancy Loads on and off.
- An edit field labeled "Buoyancy Loads" where the value for Buoyancy Loads scale factors is entered.
- A checkbox labeled "Pushover Load Pattern" which toggles Pushover Load Pattern on and off.
- A scrollable list labeled "Dynamic Load Cases" where for each Dynamic load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.6 Load Cases Dialogue Box (when the predefined macro type is Response Spectrum Analysis) (Figure 10-7)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Response Spectrum Analysis. The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.

- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.7 Load Cases Dialogue Box (when the predefined macro type is Variable Time Step Dynamic) (Figure 10-8)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Variable Time Step Dynamic. The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- A checkbox labeled "Gravity Loads" which toggles Gravity Loads on and off.
- An edit field labeled "Gravity Loads" where the value for Gravity Loads scale factor is entered.
- A checkbox labeled "Buoyancy Loads" which toggles Buoyancy Loads on and off.
- An edit field labeled "Buoyancy Loads" where the value for Buoyancy Loads scale factor is entered.
- A radio button list labeled "Loading Type" which allows the user to choose between:

  Ground Acceleration

  Wave Load

  When the loading type is ground acceleration the following are displayed for each of X, Y and Z Directions
- A radio button list labeled "Applied Record" which allows the user to choose the ground motion record component applied at the X, Y, or Z direction. Available selections are:

  None

  Primary

  Secondary

  Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in ground motion record at step zero is entered.
- An edit field labeled "Acceleration Factor" where the value for a ground acceleration scale factor is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will

be lost.

10.1.1.8 Load Cases Dialogue Box (when the predefined macro type is Constant Time Step Dynamic) (Figure 10-8)

The Load Cases dialogue box is displayed when the preconfigured analysis macro type is Constant Time Step Dynamic. The Load Cases dialogue box consists of the following:

- A scrollable list labeled "Static Load Cases" where for each static load case there is:

  A checkbox labeled "Nodal Load Cases" which toggles Nodal Load Cases on and off.

  An edit field labeled "Scale Factors" where the value for Scale Factors is entered.
- A checkbox labeled "Gravity Loads" which toggles Gravity Loads on and off.
- An edit field labeled "Gravity Loads" where the value for Gravity Loads scale factor is entered.
- A checkbox labeled "Buoyancy Loads" which toggles Buoyancy Loads on and off.
- An edit field labeled "Buoyancy Loads" where the value for Buoyancy Loads scale factor is entered.
- A radio button list labeled "Loading Type" which allows the user to choose between:

  Ground Acceleration

  Wave Load

  When the loading type is ground acceleration the following are displayed for each of X, Y and Z Directions
- A radio button list labeled "Applied Record" which allows the user to choose the ground motion record component applied at the X, Y, or Z direction. Available selections are:

  None

  Primary

  Secondary

  Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in ground motion record at step zero is entered.
- An edit field labeled "Acceleration Factor" where the value for a ground acceleration scale factor is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the load cases dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the load cases dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.9 Static Run Parameters Dialogue Box (when the predefined macro type is Static Analysis) (Figure 10-9)

The Static Run Parameters dialogue box is displayed when the preconfigured analysis macro type is Static Analysis. The Static Run Parameters dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.10 Static Run Parameters Dialogue Box (when the predefined macro type is Generate Pushover Load Pattern) (Figure 10-9)

The Static Run Parameters dialogue box is displayed when the preconfigured analysis macro type is Generate Pushover Load Pattern. The Static Run Parameters dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.11 Static Run Parameters Dialogue Box (when the predefined macro type is Statis Pushover Analysis) (Figure 10-9)

The Static Run Parameters dialogue box is displayed when the preconfigured analysis macro type is Static Pushover Analysis. The Static Run Parameters dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.12 Pushover Run Dialogue Box (when the predefined macro type is Dynamic Pushover Analysis (1 Step Solution) (Figure 10-10)

The Pushover Run dialogue box is displayed when the preconfigured analysis macro type is Dynamic Pushover Analysis (1 Step Solution). The Pushover Run dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.

- An edit field labeled "Load Increment Ratio" where the value for Load Increment Ratio is entered.
- An edit field labeled "Maximum Load Ratio" where the value for Maximum Load Ratio is entered.
- An edit field labeled "Maximum Number of Load Steps" where the value for Maximum Number of Load Steps is entered.
- An edit field labeled "Maximum Allowable Displacement" where the value for Maximum Allowable Displacement is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.13 Pushover Run Dialogue Box (when the predefined macro type is Dynamic Pushover Analysis (2 Step Solution) (Figure 10-11)

The Pushover Run dialogue box is displayed when the preconfigured analysis macro type is Dynamic Pushover Analysis (2 Step Solution). The Pushover Run dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.

Then two sets of pushover parameters for step 1 and step 2:

- An edit field labeled "Load Increment Ratio" where the value for Load Increment Ratio is entered.
- An edit field labeled "Maximum Load Ratio" where the value for Maximum Load Ratio is entered.
- An edit field labeled "Maximum Number of Load Steps" where the value for Maximum Number of Load Steps is entered.
- An edit field labeled "Maximum Allowable Displacement" where the value for Maximum Allowable Displacement is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.14 Response Spectrum Run Parameters Dialogue Box (when the predefined macro type is Response Spectrum Analysis) (Figure 10-12)

The Response Spectrum Run Parameters dialogue box is displayed when the preconfigured analysis macro type is Response Spectrum Analysis. The Response Spectrum Run Parameters dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.
- An edit field labeled "Number of Mode Shapes" where the value for Number of Mode Shapes is entered.
- A button labeled "Edit Response Spectrum Analysis Data" which when pressed will activate a Response Spectrum Analysis Data dialogue box

(10.1.1.14.1).

- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.14.1 Response Spectrum Analysis Data Dialogue Box (Figure 10-13)

The Response Spectrum Analysis Data dialogue box is used to define response spectrum analysis data. The Response Spectrum Analysis Data dialogue box consists of the following:

- Three buttons labeled "Primary Spectrum", "Secondary Spectrum" and "Vertical Spectrum" which when pressed will activate a Response Spectrum dialogue box (see 10.1.1.14.1.1) where the data corresponding to the name can be set.
- A set of radio buttons labeled "Modal Combination Type" that let the user choose between CQC and RSS.
- An edit field labeled "Damping Ratio Between Mode Shapes" where the Damping Ratio Between Mode Shapes is entered.
- Three sets of radio buttons that let the user sellect a response spectrum from the Primary, Secondary, or Vertical spectrums and assign it to the structure's X, Y, and Z directions.
- Three edit field labeled "Scale Factor" that is used to set the scale factor of the selected response spectrum for the structures X, Y, and Z directions.
- Apply button. Selecting this button will apply the selections currently displayed and close the Response Spectrum Analysis Data dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Response Spectrum Analysis Data dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.14.1.1 Primary, Secondary and Vertical Response Spectrum Dialogue Boxes (Figure 10-14)

The Primary, Secondary and Vertical Response Spectrum Dialogue Boxes are used to set the data corresponding to the name. These dialogue boxes are similar to the Primary, Secondary and Vertical Response Spectrum Dialogue Boxes described in section 9.12.3.1.

10.1.1.15 Variable Time Step Dynamic Run Dialogue Box (when the predefined macro type is Variable Time Step Dynamic) (Figure 10-15)

The Variable Time Step Dynamic Run dialogue box is displayed when the preconfigured analysis macro type is Variable Time Step Dynamic. The Variable Time Step Dynamic Run dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.
- An edit field labeled "Target Damping Ratio" where the value for Target Damping Ratio is entered.
- An edit field labeled "Fundamental Period" where the value for Fundamental Period is entered.

- An edit field labeled "Period of Highest Mode of Interest" where the value for Period of Highest Mode of Interest is entered.
- An edit field labeled "Analysis Time" where the value for Analysis Time is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.16 Constant Time Step Dynamic Run Dialogue Box (when the predefined macro type is Constant Time Step Dynamic) (Figure 10-16)

The Constant Time Step Dynamic Run dialogue box is displayed when the preconfigured analysis macro type is Constant Time Step Dynamic. The Constant Time Step Dynamic Run dialogue box consists of the following:

- An edit field labeled "Unbalance Tolerance" where the value for Unbalance Tolerance is entered.
- An edit field labeled "Target Damping Ratio" where the value for Target Damping Ratio is entered.
- An edit field labeled "Fundamental Period" where the value for Fundamental Period is entered.
- An edit field labeled "Period of Highest Mode of Interest" where the value for Period of Highest Mode of Interest is entered.
- An edit field labeled "Number of Macros" where the value for Number of Macros is entered.
- Show Macro Times button. Selecting this button will display the Constant Time Step Dynamic Macro Times Dialogue box (Figure 10-17).
- Apply button. Selecting this button will apply the selections currently displayed and close the run parameters dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the run parameters dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.1.16.1 Constant Time Step Dynamic Macro Times Dialogue Box (Figure 10-17)

The Constant Time Step Dynamic Macro Times Dialogue Box is displayed when the Show Macro Times button from the Constant Time Step Dynamic Run Dialogue Box is pressed. The Constant Time Step Dynamic Macro Times Dialogue Box consists of the following:

- A scrollable list of constatnt time step dynamic macro time fields. These fields let the user specify the duration of each constatnt time step dynamic macro.
- Apply button. Selecting this button will apply the selections currently displayed and close the constant time step dynamic macro times dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the constant time step dynamic macro times dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.2 Header Dialogue Box (Figure 10-18)

The Header dialogue box. The Header dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Header dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Header dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.3 End Analysis (NONE) Dialogue Box (Figure 10-19)

The End Analysis (NONE) dialogue box. The End Analysis (NONE) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the End Analysis (NONE) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the End Analysis (NONE) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4 Static Analysis (STAT) Dialogue Box (Figure 10-20)

The Static Analysis (STAT) dialogue box. The Static Analysis (STAT) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- An edit field labeled "Number of Segments" where the Number of Segments is entered.
- A button labeled "Edit Segment ->" which when pressed will activate a Static Analysis Segment dialogue box.
- An edit field labeled "Edit Segment ->" where the segment number to be edited is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Static Analysis (STAT) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Static Analysis (STAT) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4.1 Static Analysis (STAT) Segment Dialogue Box (Figure 10-21)

The Static Analysis Segment dialogue box. The Static Analysis Segment dialogue box consists of the following:

- A button labeled "Analysis Control" which when pressed will activate a Analysis Control dialogue box.
- A button labeled "Output Control" which when pressed will activate a Output Control dialogue box.
- A button labeled "Load Conditions" which when pressed will activate a Load Conditions dialogue box.
- A button labeled "Convergence Criteria" which when pressed will activate a Convergence Criteria dialogue box.

- A button labeled "Restepping Criteria" which when pressed will activate a Restepping Criteria dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Static Analysis (STAT) Segment dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Static Analysis (STAT) Segment dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4.2 Analysis Control (STAT) Dialogue Box (Figure 10-22)

The Analysis Control (STAT) dialogue box. The Analysis Control (STAT) dialogue box consists of the following:

- A radio button list labeled "Step Type" which allows the user to choose between:

  Load Stepping

  Displacement Stepping
- A radio button list labeled "Stepping Parameter" which allows the user to choose between:

  Euclidean Norm

  Maximum Value

  Specified Stepping Vectors
- An edit field labeled "Number of Stepping Vectors" where the Number of Stepping Vectors is entered.
- A button labeled "Edit Stepping Vectors" which when pressed will activate a Edit Stepping Vectors dialogue box (Figure 10-23).
- An edit field labeled "Segment Value" where the Segment Value is entered.
- A radio button list labeled "Step Direction" which allows the user to choose between:

  Load Magnitude

  Scalar Displacement in Critical Direction

  Scalar Displacement

  Based on Stiffness Matrix Determinant Sign
- A radio button list labeled "Element Event Check" which allows the user to choose between:

  No Event Check

  No Limit

  Up to N Events
- An edit field labeled "N" where the number of events is entered.
- A radio button list labeled "Iteration Type" which allows the user to choose between:

  Constant Load

  Constant Scalar Displacement
- A radio button list labeled "Iteration Parameter" which allows the user to choose between:

Constant Work
Constant 'arc length'
Critical Stepping Vector
Iteration Vector

- A radio button list labeled "Termination Action" which allows the user to choose between:
  Restep
  Stop
  Continue
- A radio button list labeled "BFGS Code" which allows the user to choose between:
  None
  Between Stiffness Updates
- An edit field labeled "Initial Step Factor" where the value for Initial Step Factor is entered.
- An edit field labeled "Maximum Number of Steps" where the value for Maximum Number of Steps is entered.
- An edit field labeled "Maximum Number of Iterations per Step" where the value for Maximum Number of Iterations per Step is entered.
- An edit field labeled "Step Control for Stiffness Updates" where the value for Step Control for Stiffness Updates is entered.
- An edit field labeled "Iteration Control for Stiffness Updates" where the value for Iteration Control for Stiffness Updates is entered.
- An edit field labeled "Iteration Control for State Update" where the value for Iteration Control for State Update is entered.
- An edit field labeled "Initial Arc Length (fot Static Pushover)" where the value for Initial Arc Length is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Analysis Control (STAT) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Analysis Control (STAT) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4.3 Stepping Vector (STAT) Dialogue Box (Figure 10-24)

The Stepping Vector (STAT) dialogue box. The Stepping Vector (STAT) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- An edit field labeled "Number of Terms Included" where the Number of Terms Included is entered.
- An edit field labeled "Step Size Increment Scale Factor" where the Step Size Increment Scale Factor is entered. for each node
- An edit field labeled "Node Number" where the Node Number is entered.
- A radio button list labeled "Translation" which allows the user to choose between:
  X

Y

Z

- A radio button list labeled "Rotation" which allows the user to choose between:

  X

  Y

  Z

- An edit field labeled "Weighting Factor" where the value for Weighting Factor is entered.
- Next: Selecting this button will advance to the next stepping vector. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous stepping vector. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.4.4 Output Control (STAT) Dialogue Box (Figure 10-25)

The Output Control (STAT) dialogue box. The Output Control (STAT) dialogue box consists of the following:

- A radio button list labeled "Print Results" which allows the user to choose between:

  Never

  Every Segment

  Every Step

  Every Iteration

- A checkbox labeled "Save to Restart File" which toggles Save to Restart File on and off.
- A checkbox labeled "Save to Postprocessor File" which toggles Save to Postprocessor File on and off.
- A checkbox labeled "Save Linear Beam Force Envelopes for API/AISC Code Checking (IR)" which toggles Save Linear Beam Force Envelopes for API/AISC Code Checking (IR) on and off.(IR2)
- An edit field labeled "Step Interval for Saving" where the Step Interval for Saving is entered.
- An edit field labeled "Results Save Factor (for Static Pushover)" where the value for Results Save Factor (for Static Pushover) is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Output Control (STAT) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Output Control (STAT) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4.5 Load Conditions (STAT) Dialogue Box (Figure 10-26)

The Load Conditions (STAT) dialogue box. The Load Conditions (STAT) dialogue box consists of the following:

for each nodal load case

- A radio button list labeled ""nodal load case name"(r2)
- An edit field labeled "Scale Factor" where the Scale Factor is entered.
- A checkbox labeled "Gravity Load" which toggles Gravity Load on and off.
- An edit field labeled "Gravity Load" where the Gravity Load is entered.
- A checkbox labeled "Buoyancy Load" which toggles Buoyancy Load on and off.
- An edit field labeled "Buoyancy Load" where the value for Buoyancy Load is entered.
- A checkbox labeled "Static Current" which toggles Static Current on and off.
- An edit field labeled "Static Current" where the value for Static Current is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Load Conditions (STAT) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Load Conditions (STAT) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4.6 Convergence Criteria (STAT) Dialogue Box (Figure 10-27)

The Convergence Criteria(STAT) dialogue box. The Convergence Criteria(STAT) dialogue box consists of the following:

- A radio button list labeled "Unbalance Type" which allows the user to choose between:

  Maximum Value

  Euclidean Norm

  Work Norm Norm
- A checkbox labeled "Include Moments for Unbalance" which toggles Include Moments for Unbalance on and off.
- An edit field labeled "Unbalance Tolerance (all but last step of segment)" where the value for Unbalance Tolerance (all but last step of segment) is entered.
- An edit field labeled "Unbalance Tolerance (last step of segment)" where the value for Unbalance Tolerance (last step of segment) is entered.
- An edit field labeled "Unbalance for Step Size Scaling" where the value for Unbalance for Step Size Scaling is entered.
- An edit field labeled "Maximum Number of Line Searches for any Iteration" where the value for Maximum Number of Line Searches for any Iteration is entered.
- An edit field labeled "Upper Line Search Limit" where the value for Upper Line Search Limit is entered.
- An edit field labeled "Lower Line Search Limit" where the value for Lower Line Search Limit is entered.

- An edit field labeled "Convergence Tolerance for Line Search" where the value for Convergence Tolerance for Line Search is entered.
- An edit field labeled "Number of Largest Out-of-Balance Nodal Forces to Save" where the Number of Largest Out-of-Balance Nodal Forces to Save is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Convergence Criteria (STAT) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Convergence Criteria (STAT) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.4.7 Restepping Criteria (STAT) Dialogue Box (Figure 10-28)

The Restepping Criteria dialogue box. The Restepping Criteria dialogue box consists of the following:

- An edit field labeled "Multiplier for Step Factor Reduction" where the value for Multiplier for Step Factor Reduction is entered.
- An edit field labeled "Minimum Allowable Step Factor" where the value for Minimum Allowable Step Factor is entered.
- A radio button list labeled "Termination" which allows the user to choose between:

  Continue Using Minimum

  Stop after N Attempts
- An edit field labeled "Attempts" where the value for Attempts is entered.
- An edit field labeled "Lower Limit on Number of Iterations" where the value for Lower Limit on Number of Iterations is entered.
- An edit field labeled "Multiplier for Step Factor Increase" where the value for Multiplier for Step Factor Increase is entered.
- An edit field labeled "Maximum Allowable Step Factor" where the value for Maximum Allowable Step Factor is entered.
- A radio button list labeled "Convergence Prediction" which allows the user to choose between:

  None

  Start at Iteration N
- An edit field labeled "Iteration N" where the value for Iteration N is entered.
- An edit field labeled "Grace Ratio" where the value for Grace Ratio is entered.
- An edit field labeled "Weight Factor for Iteration Prediction" where the value for Weight Factor for Iteration Prediction is entered.
- An edit field labeled "Maximum Allowable Unbalance Force" where the value for Maximum Allowable Unbalance Force is entered.
- An edit field labeled "Initial Step Size (for Static Pushover)" where the value for Initial Step Size is entered.
- An edit field labeled "Ideal Number of Iterations" where the value for Ideal Number of Iterations is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Restepping Criteria (STAT) dialogue box. This

button must be selected to make any changes effective.

- Cancel button. Selecting this button will close the Restepping Criteria (STAT) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5 Event-to-Event Dynamic Analysis (DYN1) Dialogue Box (Figure 10-29)

The Event-to-Event Dynamic Analysis (DYN1) dialogue box. The Event-to-Event Dynamic Analysis (DYN1) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- A button labeled "Analysis Control" which when pressed will activate a Analysis Control dialogue box.
- A button labeled "Output Control" which when pressed will activate a Output Control dialogue box.
- A button labeled "Load Conditions" which when pressed will activate a Load Conditions dialogue box.
- A button labeled "Convergence Criteria" which when pressed will activate a Convergence Criteria dialogue box.
- A button labeled "Time Step Selection" which when pressed will activate a Time Step Selection dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Event-to-Event Dynamic Analysis (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Event-to-Event Dynamic Analysis (DYN1) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5.1 Analysis Control (DYN1) Dialogue Box (Figure 10-30)

The Analysis Control (DYN1) dialogue box. The Analysis Control (DYN1) dialogue box consists of the following:

- A radio button list labeled "Solution Type" which allows the user to choose between:

  Event to Event with Variable Time Step

  Event to Event with Constant Time Step

  Variable Time Step with no Event Calculation
- A radio button list labeled "Equilibrium Correction" which allows the user to choose between:

  At End of Time Step

  In Next Time Step
- An edit field labeled "Initial Time Step" where the value for Initial Time Step is entered.
- An edit field labeled "Alpha for Integration Scheme" where the value for Alpha for Integration Scheme is entered.
- An edit field labeled "Beta for Integration Scheme" where the value for Beta for Integration Scheme is entered.
- An edit field labeled "Mass Proportional Damping Factor" where the value for Mass Proportional Damping Factor is entered.

- Apply button. Selecting this button will apply the selections currently displayed and close the Analysis Control (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Analysis Control (DYN1) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5.2 Output Control (DYN1) Dialogue Box (Figure 10-31)

The Output Control (DYN1) dialogue box. The Output Control (DYN1) dialogue box consists of the following:

- An edit field labeled "Nodal Displacement Printout: Time Interval" where the value for Nodal Displacement Printout Time Interval is entered.
- An edit field labeled "Nodal Displacement Printout: Step Interval" where the value for Nodal Displacement Printout Step Interval is entered.
- An edit field labeled "Element Printout: Time Interval" where the value for Element Printout Time Interval is entered.
- An edit field labeled "Element Printout: Step Interval" where the value for Element Printout Step Interval is entered.
- An edit field labeled "Restart File Printout: Time Interval" where the value for Restart File Printout Time Interval is entered.
- An edit field labeled "Restart File Printout: Step Interval" where the value for Restart File Printout Step Interval is entered.
- An edit field labeled "Postprocessor File Saving: Time Interval" where the value for Postprocessor File Saving Time Interval is entered.
- A checkbox labeled "Postprocessor File Saving: Step Intervals" which toggles Postprocessor File Saving Step Intervals on and off.
- An edit field labeled "Postprocessor File Saving: Step Intervals" where the value for Postprocessor File Saving Step Intervals is entered.
- A checkbox labeled "Save Linear Beam Force Envelopes for API/AISC Code Checking (IR)" which toggles Save Linear Beam Force Envelopes for API/AISC Code Checking (IR) on and off.(IR2)
- A radio button list labeled "Envelope Initialization" which allows the user to choose between:

  None

  Initialize to Current State
- An edit field labeled "Time Interval" where the value for Time Interval is entered.
- An edit field labeled "Step Interval " where the value for Step Interval is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Output Control (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Output Control (DYN1) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5.3 Load Conditions (DYN1) Dialogue Box (Figure 10-32)

The Load Conditions (DYN1) dialogue box. The Load Conditions (DYN1) dialogue box consists of the following:

- A radio button list labeled "Initial Conditions Code" which allows the user to choose between:

  Zero Initial Velocities and Accelerations

  If First Analysis use Restart Data otherwise use conditions at end of previous analysis

  N Initial Velocities and Accelerations Specified ->
- An edit field labeled "Initial Velocities and Accelerations" where the value for Initial Velocities and Accelerations is entered.
- A button labeled "Edit" which when pressed will activate an Initial Conditions Record dialogue box.
- A radio button list labeled "Number of Dynamic Force Commands" which allows the user to choose between:

  Forces not Applied

  N Forces Applied ->
- An edit field labeled "Forces" where the value for Forces is entered.
- A button labeled "Edit" which when pressed will activate a Force Record dialogue box.
- A radio button list labeled "Number of Dynamic Imposed Displacement Commands" which allows the user to choose between:

  Imposed Displacements not Applied

  N Imposed Displacements Applied ->
- An edit field labeled "N Imposed Displacements" where the number of Imposed Displacements is entered.
- A button labeled "Edit" which when pressed will activate a Displacement Record dialogue box.
- A radio button list labeled "Ground Acceleration Code" which allows the user to choose between:

  Ground Accelerations not Applied

  Ground Accelerations Applied
- A button labeled "Edit" which when pressed will activate a Ground Acceleration Record Application dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Load Conditions (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Load Conditions (DYN1) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5.4 Initial Conditions Record (DYN1) Dialogue Box (Figure 10-33)

The Initial Conditions Record (DYN1) dialogue box. The Initial Conditions Record (DYN1) dialogue box consists of the following:

- A radio button list labeled "Translation" which allows the user to choose between:

X
Y
Z

- A radio button list labeled "Rotation" which allows the user to choose between:
  X
  Y
  Z
- An edit field labeled "Velocity" where the value for Velocity is entered.
- An edit field labeled "Acceleration" where the value for Acceleration is entered.
- A button labeled "Node" which when pressed will activate a Node dialogue box.
- An edit field labeled "Node" where the Node is entered.
- Next: Selecting this button will advance to the next initial condition. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous initial condition. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.5.5 Force Record (DYN1) Dialogue Box (Figure 10-34)

The Force Record (DYN1) dialogue box. The Force Record (DYN1) dialogue box consists of the following:

- An edit field labeled "Record Number" where the Record Number is entered.
- A radio button list labeled "Force" which allows the user to choose between:
  X
  Y
  Z
- A radio button list labeled "Moment" which allows the user to choose between:
  X
  Y
  Z
- An edit field labeled "Time" where the value for Time is entered.
- An edit field labeled "Force" where the value for Force is entered.
- A button labeled "Node" which when pressed will activate a Node dialogue box.
- An edit field labeled "Node" where the Node is entered.

- Next: Selecting this button will advance to the next force record. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous force record. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.5.6 Displacement Record (DYN1) Dialogue Box (Figure 10-35)

The Displacement Record (DYN1) dialogue box. The Displacement Record (DYN1) dialogue box consists of the following:

- An edit field labeled "Record Number" where the Record Number is entered.
- A radio button list labeled "DOF Translation" which allows the user to choose between:

  X
  Y
  Z
- A radio button list labeled "DOF Rotation" which allows the user to choose between:

  X
  Y
  Z
- An edit field labeled "Time" where the value for Time is entered.
- An edit field labeled "Translation" where the value for Translation is entered.
- A button labeled "Node" which when pressed will activate a Node dialogue box.
- An edit field labeled "Node" where the Node is entered.
- Next: Selecting this button will advance to the next displacement record. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous displacement record. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.5.7 Ground Acceleration Record Application (DYN1) Dialogue Box (Figure 10-36)

The Ground Acceleration Record Application (DYN1) dialogue box. The Ground Acceleration Record Application (DYN1) dialogue box consists of the

following:

X Direction

- A radio button list labeled "Applied Record" which allows the user to choose between:
  - None
  - Primary
  - Secondary
  - Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in Record at Step Zero is entered.
- An edit field labeled "Acceleration Factor" where the value for Acceleration Factor is entered.

Y Direction

- A radio button list labeled "Applied Record" which allows the user to choose between:
  - None
  - Primary
  - Secondary
  - Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in Record at Step Zero is entered.
- An edit field labeled "Acceleration Factor" where the value for Acceleration Factor is entered.

Z Direction

- A radio button list labeled "Applied Record" which allows the user to choose between:
  - None
  - Primary
  - Secondary
  - Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in Record at Step Zero is entered.
- An edit field labeled "Acceleration Factor" where the value for Acceleration Factor is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Ground Acceleration Record Application (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Ground Acceleration Record Application (DYN1) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5.8 Convergence Criteria (DYN1) Dialogue Box (Figure 10-37)

The Convergence Criteria(DYN1) dialogue box. The Convergence Criteria(DYN1) dialogue box consists of the following:

- An edit field labeled "Maximum Number of Steps" where the Maximum Number of Steps is entered.

- An edit field labeled "Analysis Time" where the value for Analysis Time is entered.
- An edit field labeled "Maximum Allowable Unbalanced Force" where the value for Maximum Allowable Unbalanced Force is entered.
- An edit field labeled "Maximum Allowable Displacement" where the value for Maximum Allowable Displacement is entered.
- An edit field labeled "Maximum Displacement Increment" where the value for Maximum Displacement Increment is entered.
- An edit field labeled "Maximum Allowable Number of Substeps" where the value for Maximum Allowable Number of Substeps is entered.
- An edit field labeled "Cap Cut-off Velocity" where the value for Cap Cut-off Velocity is entered.
- An edit field labeled "Number of Largest Out-of-Balance Nodal Forces to Save" where the Number of Largest Out-of-Balance Nodal Forces to Save is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Convergence Criteria (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Convergence Criteria (DYN1) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.5.9 Time Step Selection (DYN1) Dialogue Box (Figure 10-38)

The Time Step Selection dialogue box. The Time Step Selection dialogue box consists of the following:

- An edit field labeled "Minimum Allowable Step Size" where the value for Minimum Allowable Step Size is entered.
- An edit field labeled "Maximum Allowable Step Size" where the value for Maximum Allowable Step Size is entered.
- An edit field labeled "Step Size Reduction Factor" where the value for Step Size Reduction Factor is entered.
- An edit field labeled "Step Size Increment Factor" where the value for Step Size Increment Factor is entered.
- An edit field labeled "Upper Tolerance on Midstep Error" where the value for Upper Tolerance on Midstep Error is entered.
- An edit field labeled "Lower Tolerance on Midstep Error" where the value for Lower Tolerance on Midstep Error is entered.
- A checkbox labeled "Include Moment for Midstep Error" which toggles Include Moment for Midstep Error on and off.
- An edit field labeled "Load Increment Ratio" where the value for Load Increment Ratio is entered.
- An edit field labeled "Maximum Load Ratio" where the value for Maximum Load Ratio is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Time Step Selection (DYN1) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Time Step Selection (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.6 Iterative Dynamic Analysis (DYN2) Dialogue Box (Figure 10-39)

The Iterative Dynamic Analysis (DYN2) dialogue box. The Iterative Dynamic Analysis (DYN2) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- A button labeled "Analysis Control" which when pressed will activate a Analysis Control dialogue box.
- A button labeled "Output Control" which when pressed will activate a Output Control dialogue box.
- A button labeled "Load Conditions" which when pressed will activate a Load Conditions dialogue box.
- A button labeled "Convergence Criteria" which when pressed will activate a Convergence Criteria dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Iterative Dynamic Analysis (DYN2) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Iterative Dynamic Analysis (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.6.1 Analysis Control (DYN2) Dialogue Box (Figure 10-40)

The Analysis Control (DYN2) dialogue box. The Analysis Control (DYN2) dialogue box consists of the following:

- An edit field labeled "Time Step" where the Time Step is entered.
- An edit field labeled "Alpha for Integration Scheme" where the value for Alpha for Integration Scheme is entered.
- An edit field labeled "Beta for Integration Scheme" where the value for Beta for Integration Scheme is entered.
- An edit field labeled "Mass Proportional Damping Factor" where the value for Mass Proportional Damping Factor is entered.
- An edit field labeled "Number of Steps" where the Number of Steps is entered.
- A radio button list labeled "Initial Conditions Code" which allows the user to choose between:

  Zero Initial Conditions and Accelerations

  If First Analysis use Restart Data otherwise use conditions at end of previous analysis

  N Initial Velocities and Accelerations Specified ->
- An edit field labeled "N Initial Velocities and Accelerations Specified" where the number of Initial Velocities and Accelerations Specified is entered.
- A button labeled "Edit" which when pressed will activate an Initial Conditions Record dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Analysis Control (DYN2) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Analysis Control (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.6.2 Initial Conditions Record (DYN2) Dialogue Box (Figure 10-41)

The Initial Conditions Record (DYN2) dialogue box. The Initial Conditions Record (DYN2) dialogue box consists of the following:

- A radio button list labeled "Translation" which allows the user to choose between:
  - X
  - Y
  - Z
- A radio button list labeled "Rotation" which allows the user to choose between:
  - X
  - Y
  - Z
- An edit field labeled "Velocity" where the value for Velocity is entered.
- An edit field labeled "Acceleration" where the value for Acceleration is entered.
- A button labeled "Node" which when pressed will activate a Node dialogue box.
- An edit field labeled "Node" where the Node is entered.
- Next: Selecting this button will advance to the next initial conditions record. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous initial conditions record. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.6.3 Output Control (DYN2) Dialogue Box (Figure 10-42)

The Output Control (DYN2) dialogue box. The Output Control (DYN2) dialogue box consists of the following:

- An edit field labeled "Time Step Interval for Nodal Disp. Printout" where the value for Time Step Interval for Nodal Disp. Printout is entered.
- An edit field labeled "Time Step Interval for Element Printout" where the value for Time Step Interval for Element Printout is entered.
- An edit field labeled "Time Step Interval for Envelope Printout" where the value for Time Step Interval for Envelope Printout is entered.
- An edit field labeled "Time Step Interval for Postprocessor Saving" where the value for Time Step Interval for Postprocessor Saving is entered.
- A checkbox labeled "Save Nodal Wave and Inertia Loads" which toggles Save Nodal Wave and Inertia Loads on and off.
- A checkbox labeled "Save Linear Beam Force Envelopes for Fatigue and API/AISC Code Checking (IR)" which toggles Save Linear Beam Force Envelopes for Fatigue and API/AISC Code Checking (IR) on and off.(IR2)

- A radio button list labeled "Envelope Initialization" which allows the user to choose between:

  None

  Initialized to Current State
- Apply button. Selecting this button will apply the selections currently displayed and close the Output Control (DYN2) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Output Control (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.6.4 Load Conditions (DYN2) Dialogue Box (Figure 10-43)

The Load Conditions (DYN2) dialogue box. The Load Conditions (DYN2) dialogue box consists of the following:

- A radio button list labeled "Number of Dynamic Force Commands" which allows the user to choose between:

  Forces not Applied

  N Forces Applied ->
- An edit field labeled "Forces" where the value for Forces is entered.
- A button labeled "Edit" which when pressed will activate a Force Record dialogue box.
- A radio button list labeled "Number of Dynamic Imposed Displacement Commands" which allows the user to choose between:

  Imposed Displacements not Applied

  N Imposed Displacements Applied ->
- An edit field labeled "Imposed Displacements" where the value for Imposed Displacements is entered.
- A button labeled "Edit" which when pressed will activate a Displacement Record dialogue box.
- A radio button list labeled "Ground Acceleration Code" which allows the user to choose between:

  Ground Accelerations not Applied

  Ground Accelerations Applied
- A button labeled "Edit" which when pressed will activate a Ground Acceleration Record Application dialogue box.
- Apply button. Selecting this button will apply the selections currently displayed and close the Load Conditions (DYN2) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Load Conditions (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.6.5 Force Record (DYN2) Dialogue Box (Figure 10-44)

The Force Record (DYN2) dialogue box. The Force Record (DYN2) dialogue box consists of the following:

- An edit field labeled "Record Number" where the Record Number is entered.

- A radio button list labeled "Force" which allows the user to choose between:
  X
  Y
  Z
- A radio button list labeled "Moment" which allows the user to choose between:
  X
  Y
  Z
- An edit field labeled "Time" where the value for Time is entered.
- An edit field labeled "Force" where the value for Force is entered.
- A button labeled "Node" which when pressed will activate a Node dialogue box.
- An edit field labeled "Node" where the Node is entered.
- Next: Selecting this button will advance to the next force record. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous force record. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.6.6 Displacement Record (DYN2) Dialogue Box (Figure 10-45)

The Displacement Record (DYN2) dialogue box. The Displacement Record (DYN2) dialogue box consists of the following:

- An edit field labeled "Record Number" where the Record Number is entered.
- A radio button list labeled "DOF Translation" which allows the user to choose between:
  X
  Y
  Z
- A radio button list labeled "DOF Rotation" which allows the user to choose between:
  X
  Y
  Z
- An edit field labeled "Time" where the value for Time is entered.
- An edit field labeled "Translation" where the value for Translation is entered.
- A button labeled "Node" which when pressed will activate a Node dialogue box.

- An edit field labeled "Node" where the Node is entered.
- Next: Selecting this button will advance to the next displacement record. If at the last, selecting this button will roll back to the first.
- Prev: Selecting this button will roll back to the previous displacement record. If at the first, selecting this button will roll back to the last.
- Apply: Selecting this button will apply the selections currently displayed. This button must be selected to make any changes effective.
- Cancel: Selecting this button will recover the old selections since the dialogue box was displayed or the last Apply button selection. Any changes made will be lost.
- Done: Selecting this button will close the dialogue box.

10.1.6.7 Ground Acceleration Record Application (DYN2) Dialogue Box (Figure 10-46)

The Ground Acceleration Record Application (DYN2) dialogue box. The Ground Acceleration Record Application (DYN2) dialogue box consists of the following:

X Direction

- A radio button list labeled "Applied Record" which allows the user to choose between:

  None

  Primary

  Secondary

  Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in Record at Step Zero is entered.
- An edit field labeled "Acceleration Factor" where the value for Acceleration Factor is entered.

Y Direction

- A radio button list labeled "Applied Record" which allows the user to choose between:

  None

  Primary

  Secondary

  Vertical
- An edit field labeled "Time in Record at Step Zero" where the value for Time in Record at Step Zero is entered.
- An edit field labeled "Acceleration Factor" where the value for Acceleration Factor is entered.

Z Direction

- A radio button list labeled "Applied Record" which allows the user to choose between:

  None

  Primary

  Secondary

  Vertical

- An edit field labeled "Time in Record at Step Zero" where the value for Time in Record at Step Zero is entered.
- An edit field labeled "Acceleration Factor" where the value for Acceleration Factor is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Ground Acceleration Record Application (DYN2) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Ground Acceleration Record Application (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.6.8 Convergence Criteria (DYN2) Dialogue Box (Figure 10-47)

The Convergence Criteria(DYN2) dialogue box. The Convergence Criteria(DYN2) dialogue box consists of the following:

- An edit field labeled "Maximum Number of Iterations in Any Step" where the value for Maximum Number of Iterations in Any Step is entered.
- An edit field labeled "Number of Time Step for Stiffness Update" where the Number of Time Step for Stiffness Update is entered.
- An edit field labeled "Number of Iterations for Stiffness Update" where the Number of Iterations for Stiffness Update is entered.
- An edit field labeled "Number of Iterations for State Update" where the Number of Iterations for State Update is entered.
- A radio button list labeled "Unbalance Type" which allows the user to choose between:

  Maximum Value

  Euclidean Norm
- A checkbox labeled "Include Moments in Unbalance Calculations" which toggles Include Moments in Unbalance Calculations on and off.
- A radio button list labeled "Solution Termination" which allows the user to choose between:

  Continue with Next Step if no Convergence

  Stop Analysis if no Convergence
- An edit field labeled "No. of Largest Out-of-Balance Nodal Forces to Save" where the Number of Largest Out-of-Balance Nodal Forces to Save is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Convergence Criteria (DYN2) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Convergence Criteria (DYN2) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.7 Frequency Analysis (EIGN) Dialogue Box (Figure 10-48)

The Frequency Analysis (EIGN) dialogue box. The Frequency Analysis (EIGN) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.

- A checkbox labeled "Print Intermediate Norms/Matrices" which toggles Print Intermediate Norms/Matrices on and off.
- An edit field labeled "Max Number of Iterations Allowed / Vector" where the Max Number of Iterations Allowed / Vector is entered.
- A checkbox labeled "Perform Strum Sequence Check" which toggles Perform Strum Sequence Check on and off.
- An edit field labeled "Number of Frequencies and Mode Shapes" where the Number of Frequencies and Mode Shapes is entered.
- An edit field labeled "Top Cut-Off Frequencies" where the value for Top Cut-Off Frequencies is entered.
- An edit field labeled "Subspace Shift" where the value for Subspace Shift is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Frequency Analysis (EIGN) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Frequency Analysis (EIGN) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.1.8 Save Results for Restart (SAVE) Dialogue Box (Figure 10-49)

The Save Results for Restart (SAVE) dialogue box. The Save Results for Restart (SAVE) dialogue box consists of the following:

- An edit field labeled "Title" where the title is entered.
- Apply button. Selecting this button will apply the selections currently displayed and close the Save Results for Restart (SAVE) dialogue box. This button must be selected to make any changes effective.
- Cancel button. Selecting this button will close the Save Results for Restart (SAVE) dialogue box and use the settings that were already in effect. Any changes made will be lost.

10.2 Perform Data Check

The second selection of the Analysis menu will perform data checks on the model. The detected problems will be saved to an error log file. If there are problems to report, the contents of the error log file will be displayed using the Print File Browser dialogue box automatically, and members or nodes which have caused errors will be query selected (Figure 10-50). While the data check operation is in progress, the status line will display an appropriate message.

The data check module will perform the following tests:

1- Connectivity checks. The data check module will determine whether nodes referenced by elements are defined.

2- Joint checks. The data check module will determine whether any node is not connected to any members and has free DOFs or if a node is connected only to truss members and is not fixed against rotation. Also it checks for existance of master nodes if any of the joints dof's are slaved.

3- Section Properties checks. The data check module will determine whether any of the tubular or generic section properties has a value that is less than or equal to zero and whether an AISC section ID data is not available in the AISC table.

4- Torsional Stability checks. The data check module will determine whether a linear beam member is fully pinned at both ends.

5- Strut and Buckling Beam Section checks. The data check module will determine whether a strut or a buckling beam element has a nontubular section.

6- Strut Wave Data check. The data check module will determine whether there are strut element but there are no wave data defined.

7- Material Properties checks. The data check module will determine whether any of the materials referenced in the elements are not defined and determine whether any of the defined materials has a value that is less than or equal to zero.

8- Wave Data checks. The data check module will determine whether the defined wave attributes are zero or negative.

9- Current Data checks. The data check module will determine whether the defined current attributes are zero or negative and whether the specified number of profile points is less than two.

10- Group checks. The data check module will determine whether there is an element group with more elements than SeaStar can accommodate.

11- Load checks. The data check module will determine whether there are no defined loads.

10.3 Run SeaStar Analysis

If the user selects the third selection of the Analysis menu when there is an open project, CAP will display a dialogue box to verify that the user wants to run a SeaStar analysis on the open project (Figure 10-51). If a SeaStar input file already exists, a check box is provided to allow the user to toggle the generation of a new SeaStar input file on and off. This feature allows the user to run the analysis from CAP after editing the input file generated by CAP. The user can either select the button labeled Yes and run the analysis, or the button labeled No and ignore the analysis request. Note that when the user selects the Yes button, CAP will generate a SeaStar input file only if the user selected so (by selecting the check box).

If the user selects the third selection of the Analysis menu when there is no open project, CAP will display a dialogue box similar to the open project dialogue box. The user is presented with a list of projects in the current working directory. As is the case with the open project dialogue box, the user must select a project, a model, and a version on which the analysis will be performed. The user is also presented with a check box that will make CAP generate the input file and stop rather than continue performing the analysis. Finally, the user can select one of two buttons, the first, labeled Accept, will run the analysis (or generate the input file only) and the second, labeled Cancel, will ignore the analysis request.

When an analysis (or input file generation) is selected, the status line will display messages reflecting the various stages of creating a temporary copy of the project, generating the SeaStar input file, and if the analysis is to proceed, the analysis, and finally, updating the project directory.

10.4 Create SeaStar File

If the user selects the fourth selection of the Analysis menu when there is an open project, CAP will display a dialogue box to verify that the user wants to create a SeaStar input file on the open project (Figure 10-52). The user can either select the button labeled Yes and create the file, or the button labeled No and ignore the request.

If the user selects the fourth selection of the Analysis menu when there is no open project, CAP will display a dialogue box similar to the open project dialogue box. The user is presented with a list of projects in the current working directory. As is the case with the open project dialogue box, the user must select a project, a model, and a version on which the analysis will be performed. The user is also presented with a check box that will make CAP generate the input file and stop rather than continue performing the analysis. Finally, the user can select one of two buttons, the first, labeled Accept, will run the analysis (or generate the input file only) and the second, labeled Cancel, will ignore the analysis request.

When an analysis (or input file generation) is selected, the status line will display messages reflecting the various stages of creating a temporary copy of the project, generating the SeaStar input file, and if the analysis is to proceed, the analysis, and finally, updating the project directory.

10.5 Edit Files

The fifth selection of the Analysis menu will display the Edit Files dialogue box (Figure 10-53). This dialogue box allows the user to edit a file. The user can select the file from one of the following checkboxes.

- SeaStar Input File
- Code Check Configuration File
- Code Check Member File
- Response Spectrum Analysis Input File
- Response Spectrum Force Profile Input File

The File Editor edit field is used to specify the editor. Enter the name of the editor you wish to use (e.g. vi, emacs, etc.).

Buttons:

- Edit button. Selecting this button run the editor with the specified file.
- Cancel button. Selecting this button will close the Edit Files dialogue box.

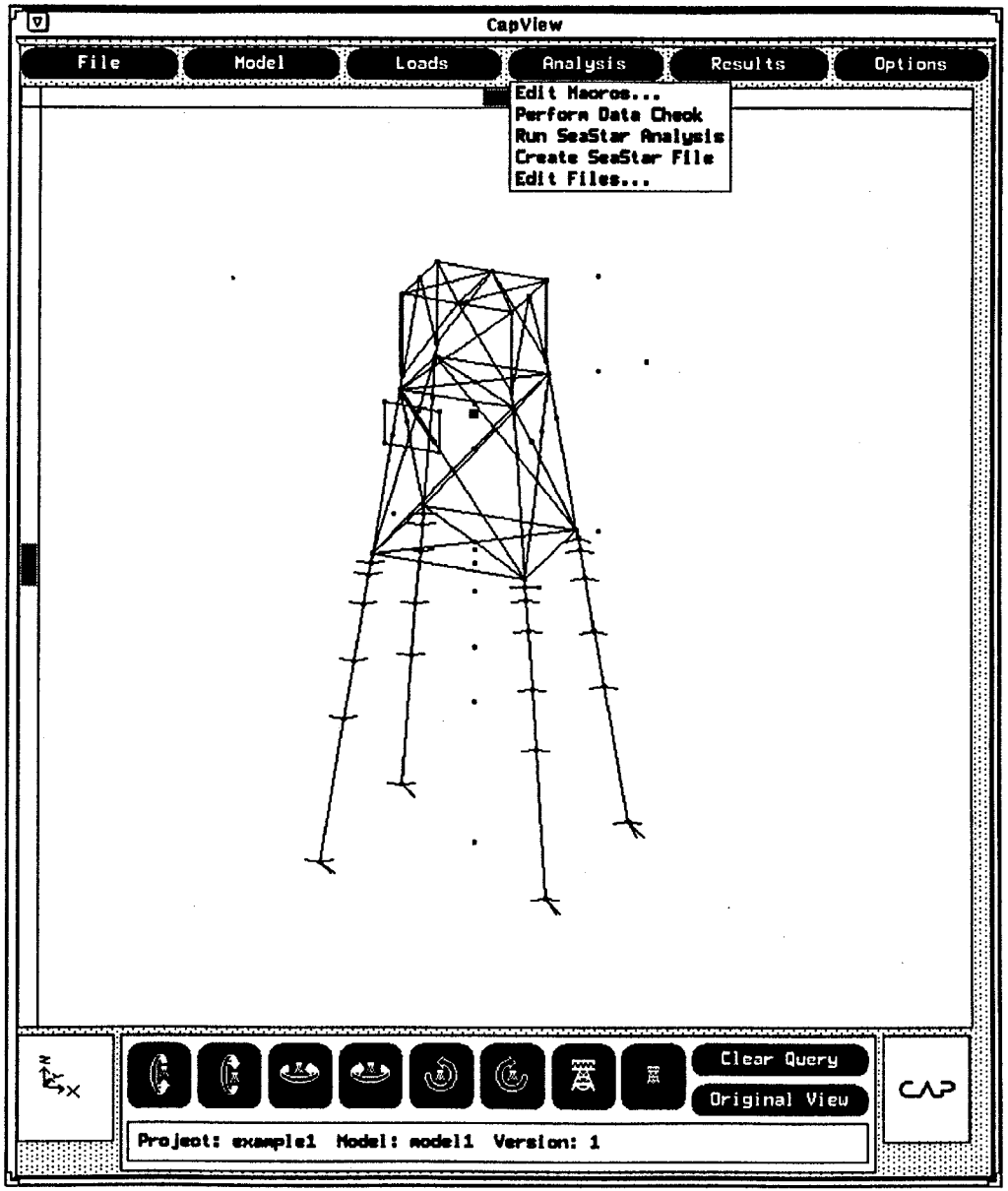

CapView
File
Model
Loads
Analysis
Results
Options
Edit Macros...
Perform Data Check
Run SeaStar Analysis
Create SeaStar File
Edit Files...
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-1

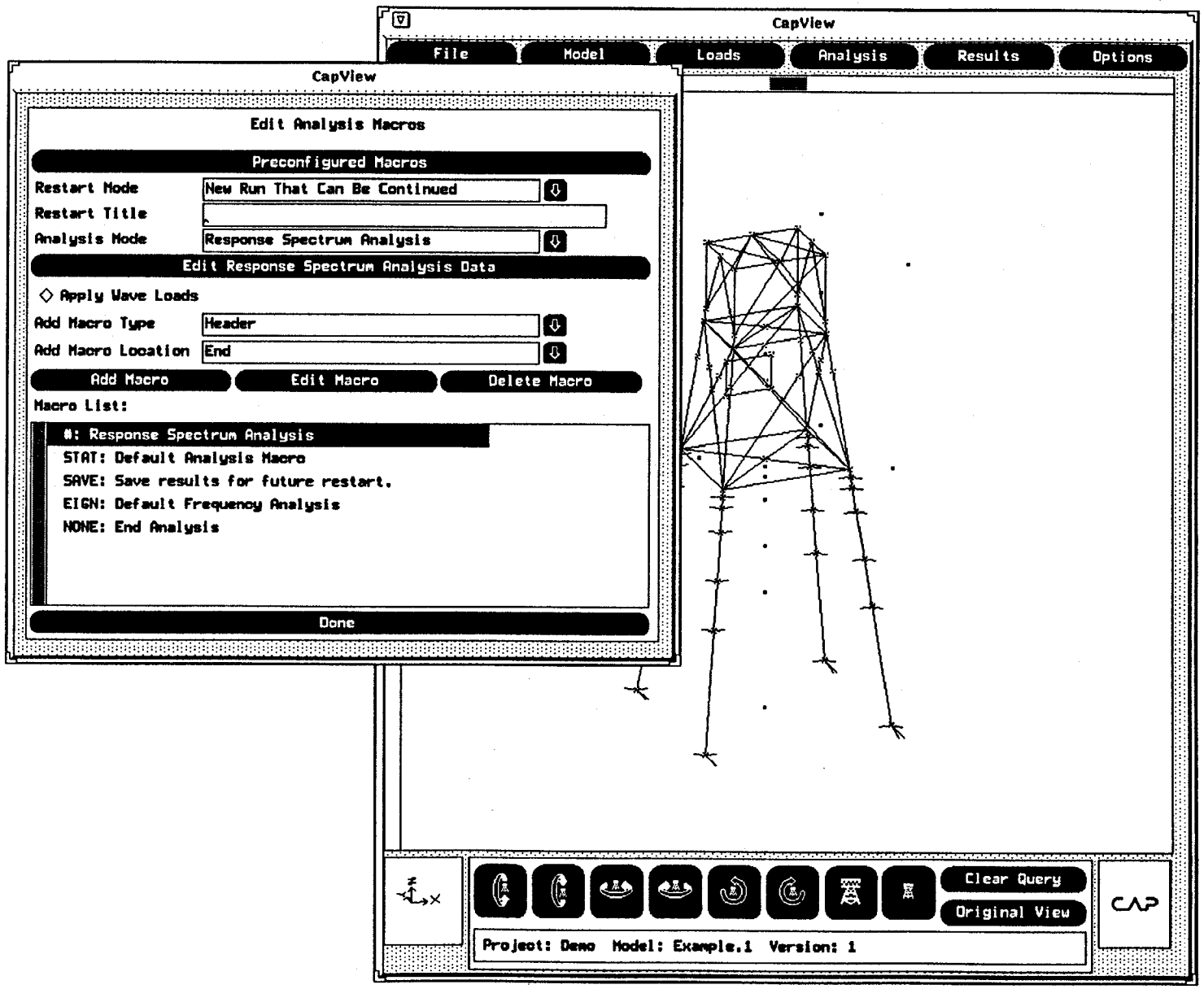

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Edit Analysis Macros
Preconfigured Macros
Restart Mode
New Run That Can Be Continued
Restart Title
Analysis Mode
Response Spectrum Analysis
Edit Response Spectrum Analysis Data
Apply Wave Loads
Add Macro Type
Header
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Response Spectrum Analysis
STAT: Default Analysis Macro
SAVE: Save results for future restart.
EIGN: Default Frequency Analysis
NONE: End Analysis
Done
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 10-2

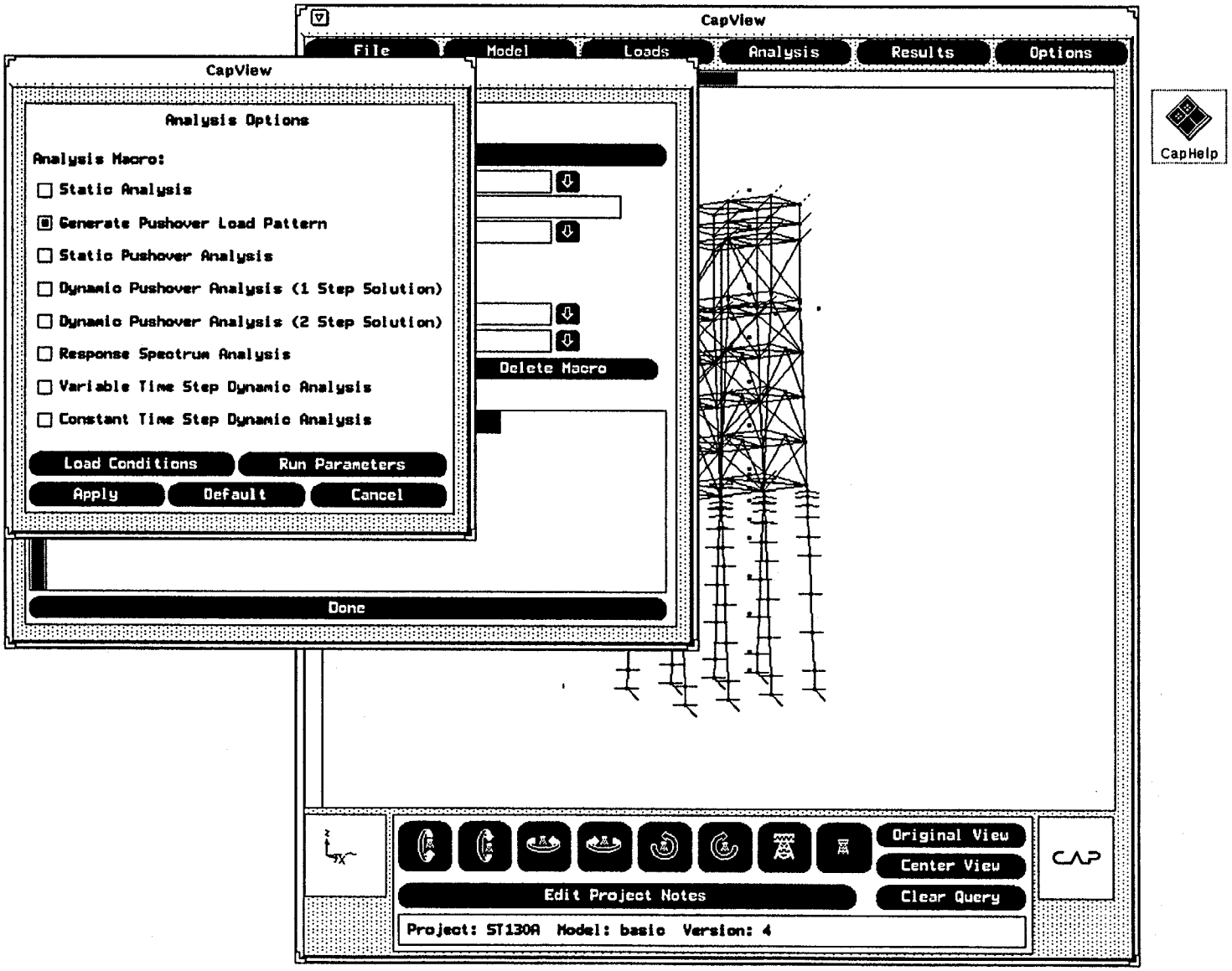

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Analysis Options
Analysis Macro:
Static Analysis
Generate Pushover Load Pattern
Static Pushover Analysis
Dynamic Pushover Analysis (1 Step Solution)
Dynamic Pushover Analysis (2 Step Solution)
Response Spectrum Analysis
Variable Time Step Dynamic Analysis
Constant Time Step Dynamic Analysis
Load Conditions
Run Parameters
Apply
Default
Cancel
Delete Macro
Done
Original View
Center View
Edit Project Notes
Clear Query
Project: ST130A Model: basic Version: 4

Figure 10-3

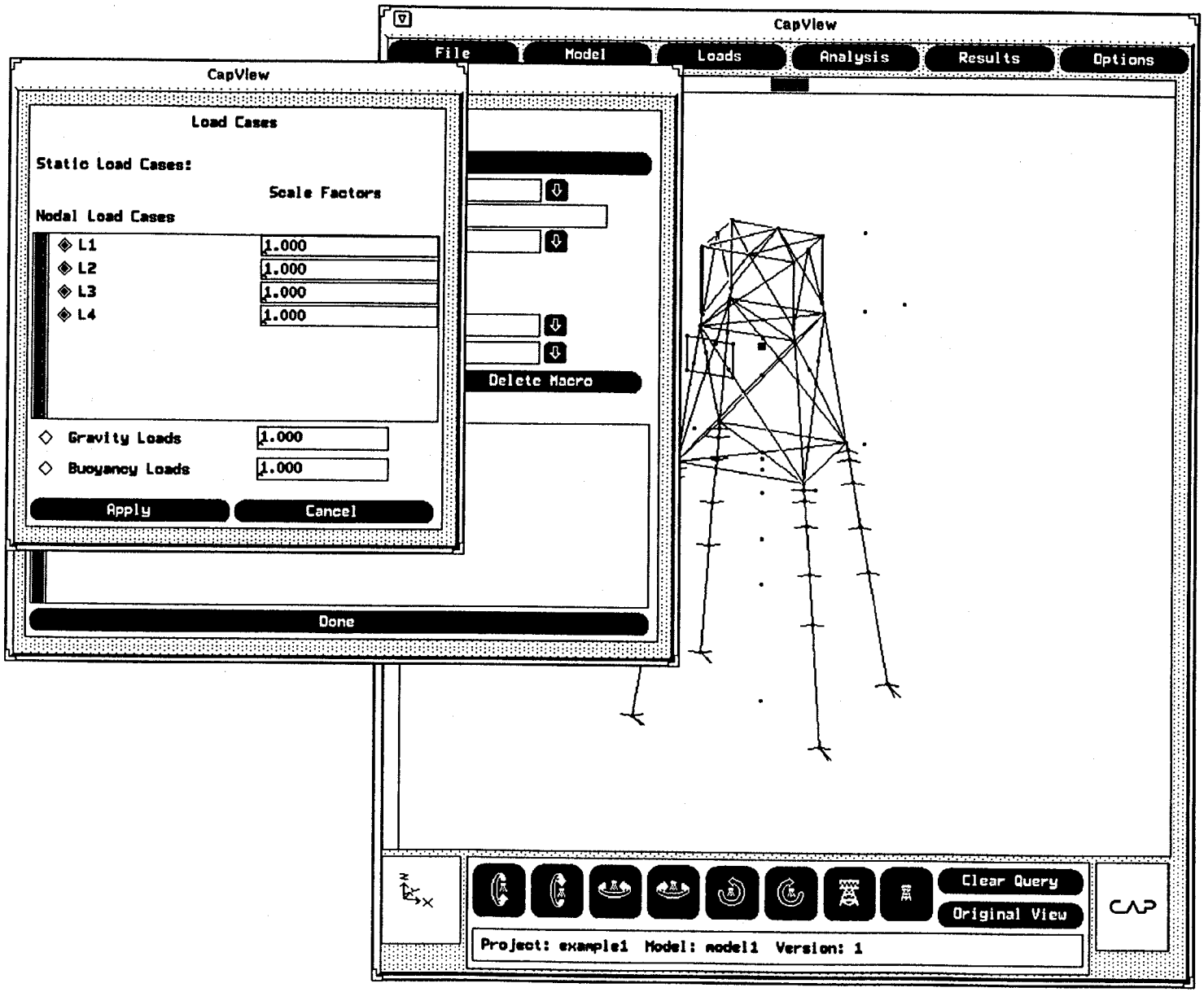

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Cases
Static Load Cases:
Scale Factors
Nodal Load Cases
L1
1.000
L2
1.000
L3
1.000
L4
1.000
Gravity Loads
1.000
Buoyancy Loads
1.000
Apply
Cancel
Delete Macro
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-4

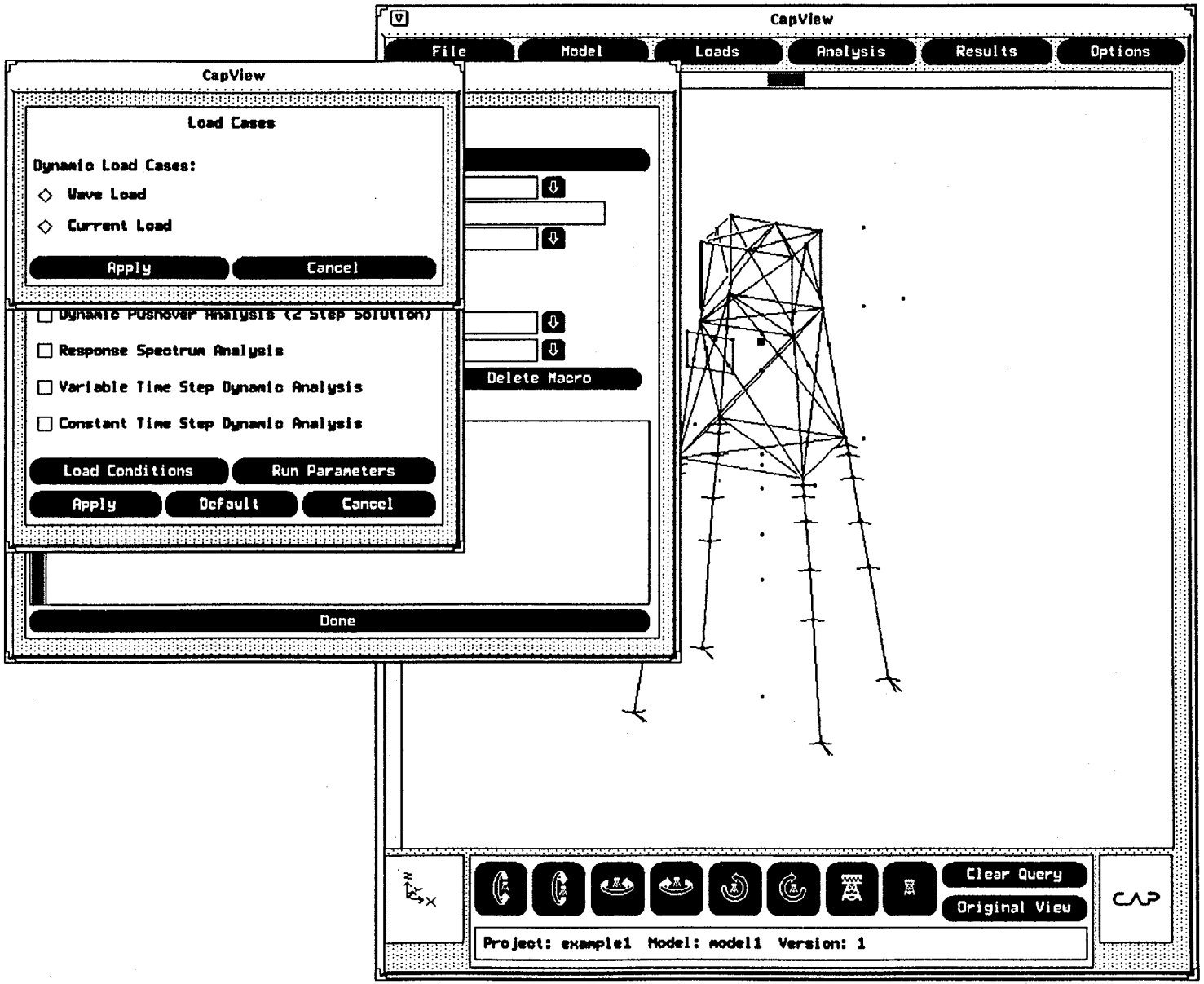

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Cases
Dynamic Load Cases:
Wave Load
Current Load
Apply
Cancel
Dynamic Pushover Analysis (2 Step Solution)
Response Spectrum Analysis
Variable Time Step Dynamic Analysis
Constant Time Step Dynamic Analysis
Load Conditions
Run Parameters
Apply
Default
Cancel
Delete Macro
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-5

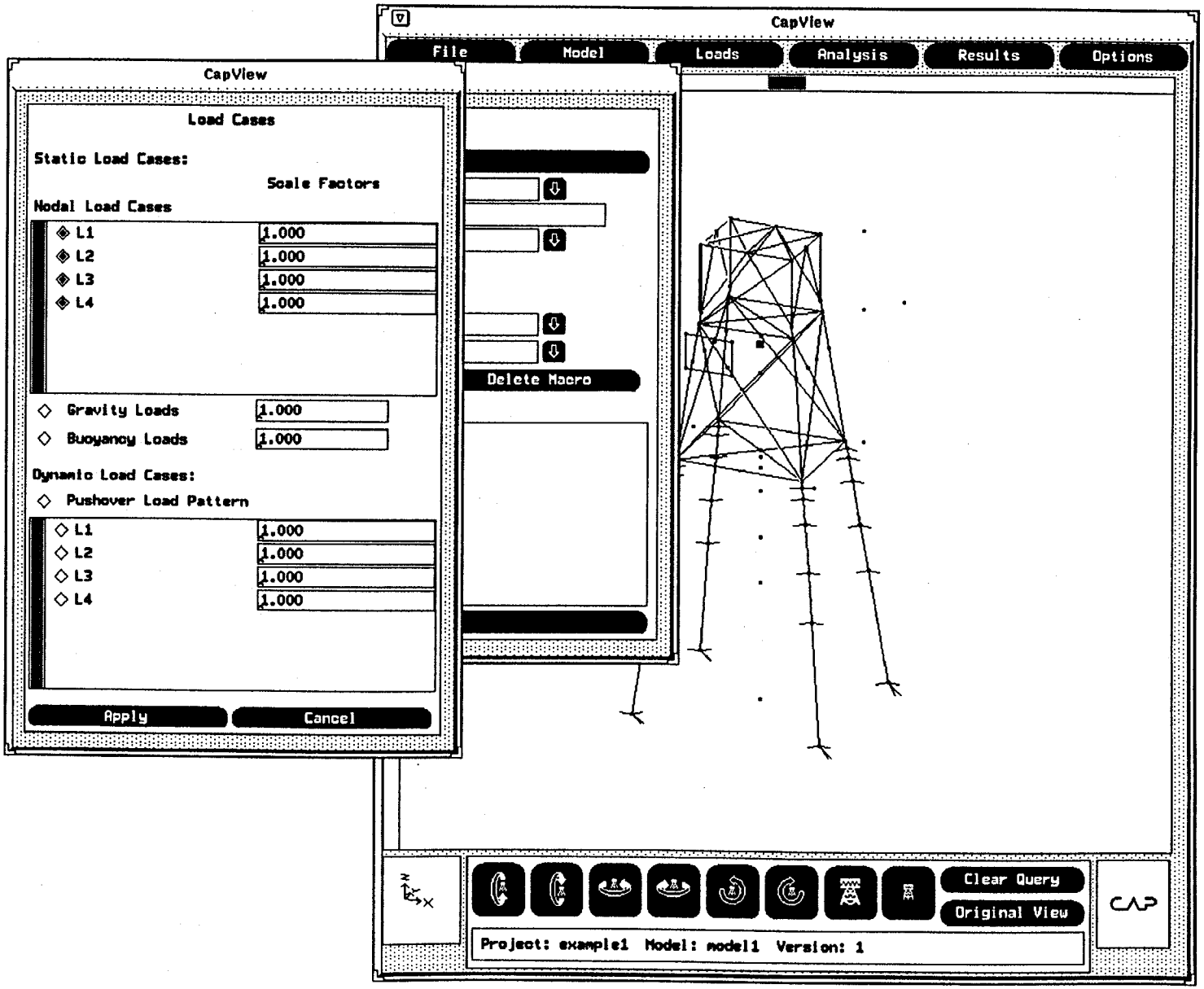

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Cases
Static Load Cases:
Scale Factors
Nodal Load Cases
L1 1.000
L2 1.000
L3 1.000
L4 1.000
Gravity Loads 1.000
Buoyancy Loads 1.000
Dynamic Load Cases:
Pushover Load Pattern
L1 1.000
L2 1.000
L3 1.000
L4 1.000
Apply
Cancel
Delete Macro
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-6

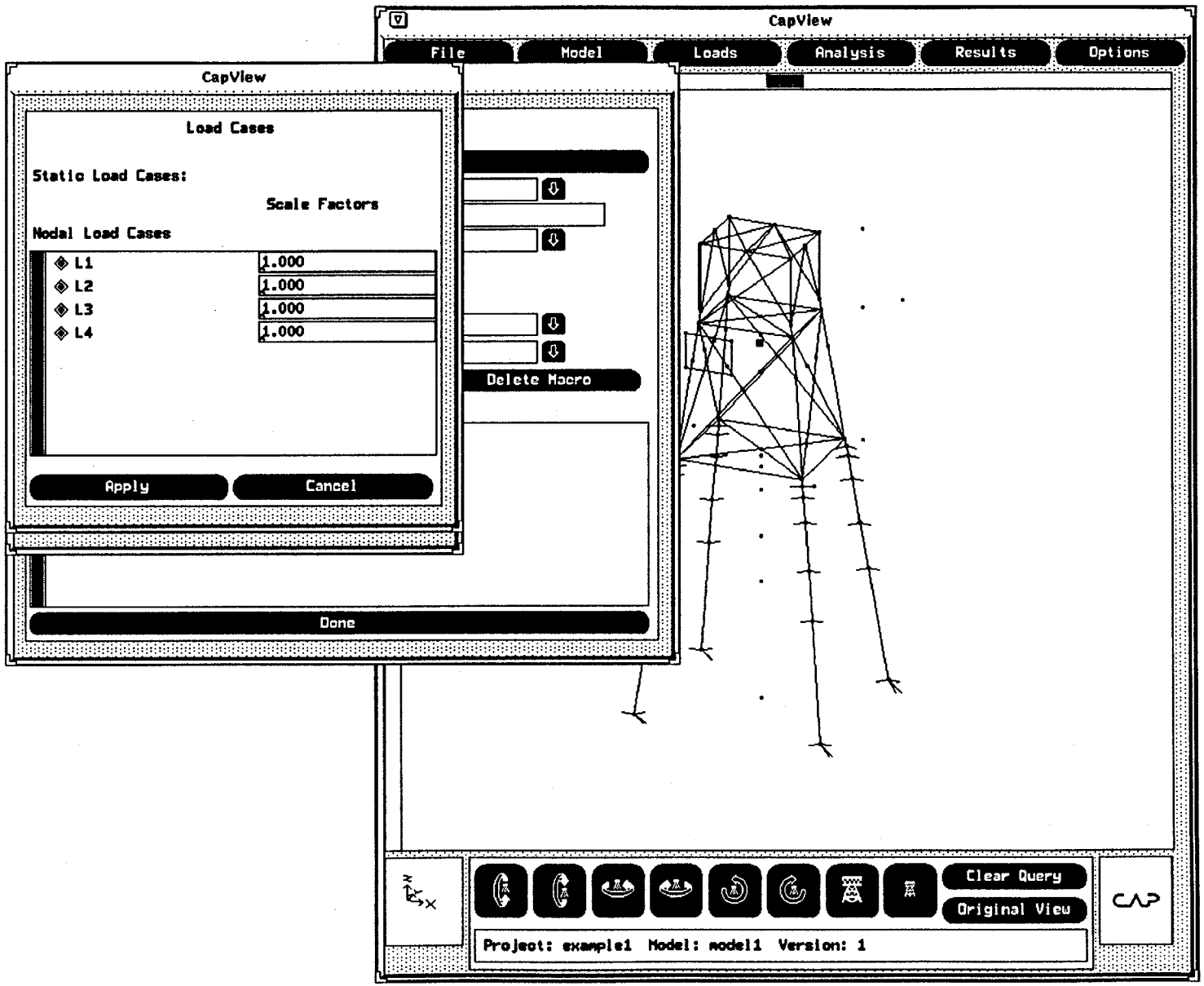

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Cases
Static Load Cases:
Scale Factors
Nodal Load Cases
L1
1.000
L2
1.000
L3
1.000
L4
1.000
Apply
Cancel
Delete Macro
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-7

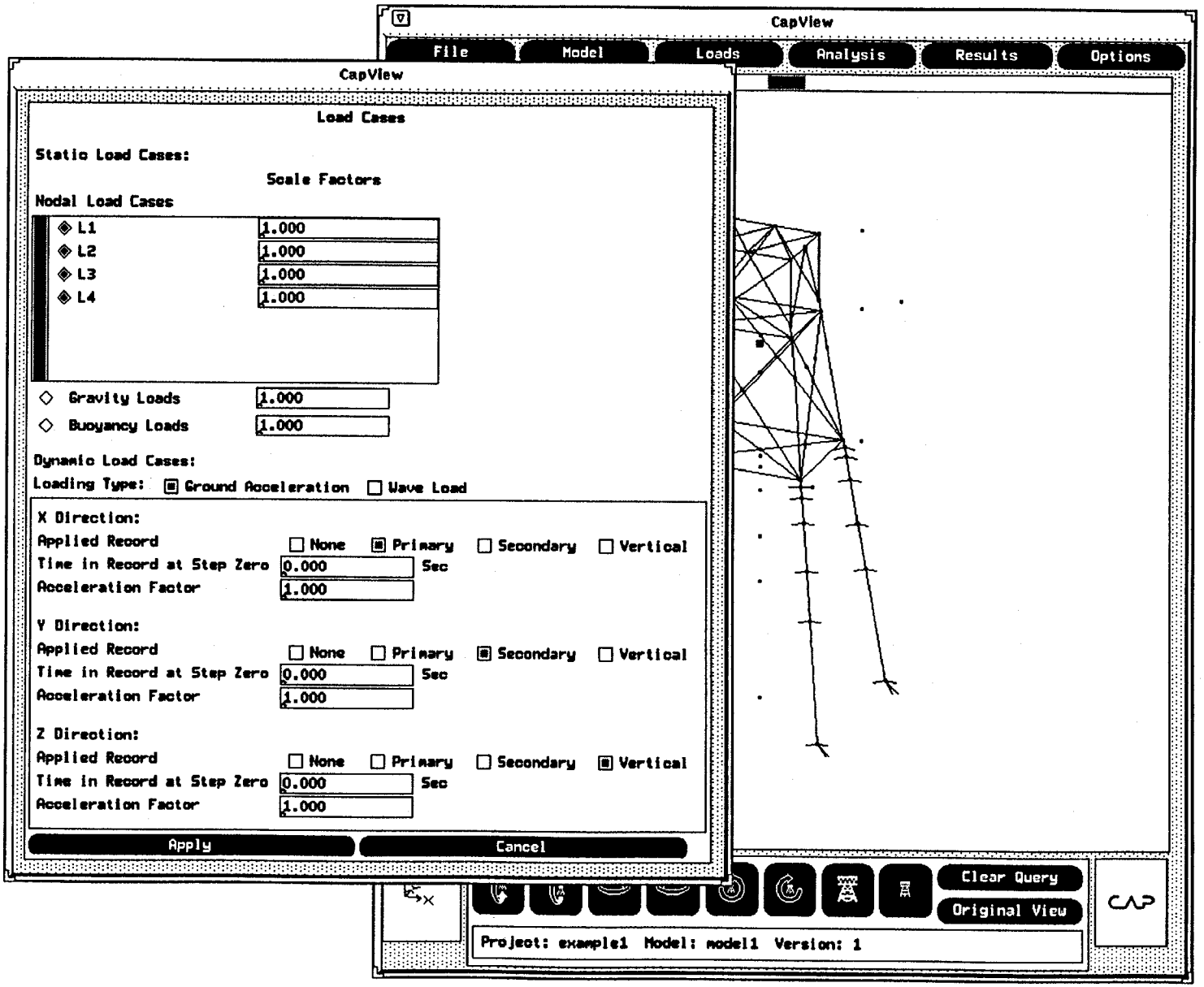

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Cases
Static Load Cases:
Scale Factors
Nodal Load Cases
L1 1.000
L2 1.000
L3 1.000
L4 1.000
Gravity Loads 1.000
Buoyancy Loads 1.000
Dynamic Load Cases:
Loading Type: Ground Acceleration Wave Load
X Direction:
Applied Record None Primary Secondary Vertical
Time in Record at Step Zero 0.000 Sec
Acceleration Factor 1.000
Y Direction:
Applied Record None Primary Secondary Vertical
Time in Record at Step Zero 0.000 Sec
Acceleration Factor 1.000
Z Direction:
Applied Record None Primary Secondary Vertical
Time in Record at Step Zero 0.000 Sec
Acceleration Factor 1.000
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-8

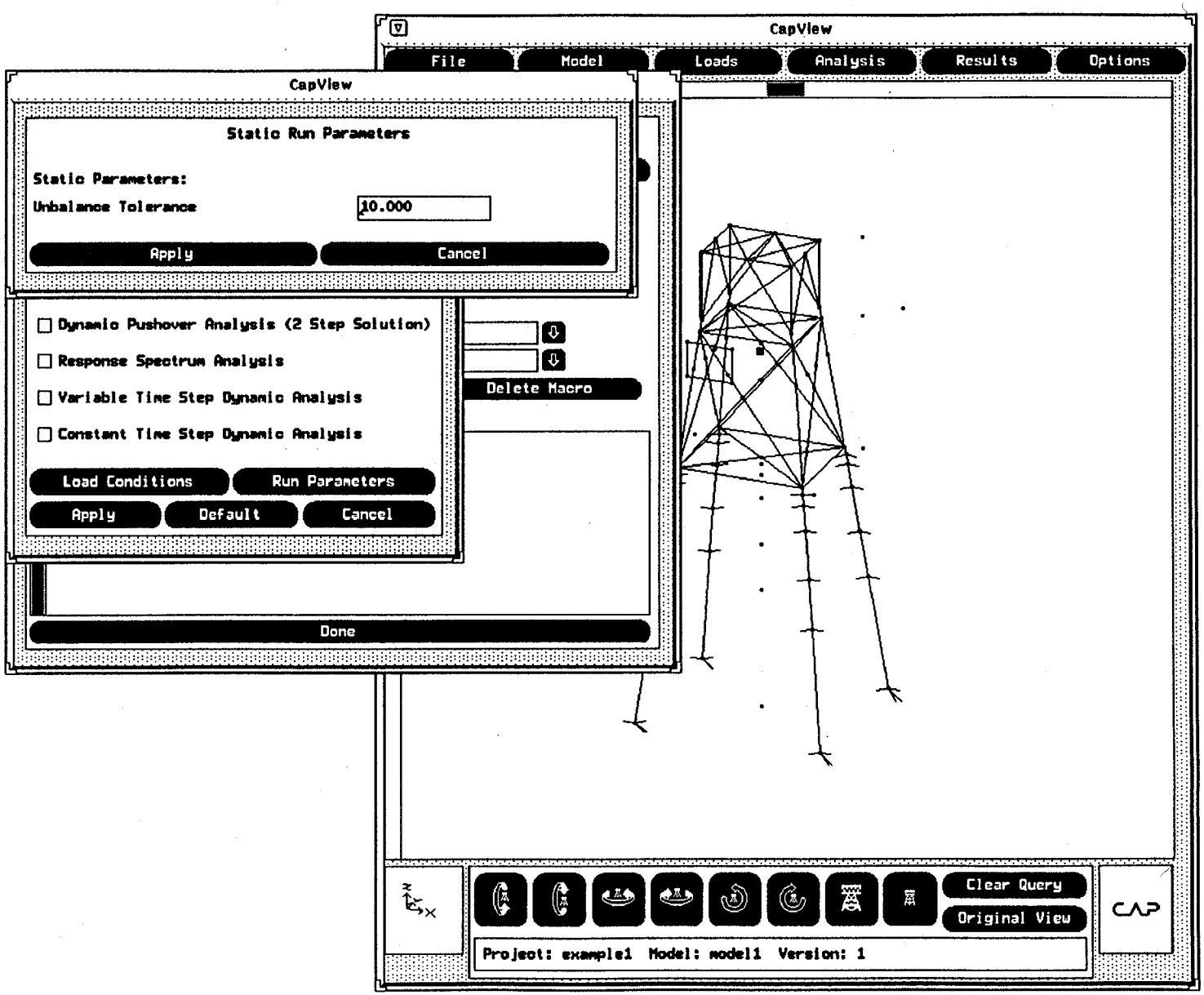

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Static Run Parameters
Static Parameters:
Unbalance Tolerance
10.000
Apply
Cancel
Dynamic Pushover Analysis (2 Step Solution)
Response Spectrum Analysis
Variable Time Step Dynamic Analysis
Constant Time Step Dynamic Analysis
Load Conditions
Run Parameters
Apply
Default
Cancel
Delete Macro
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-9

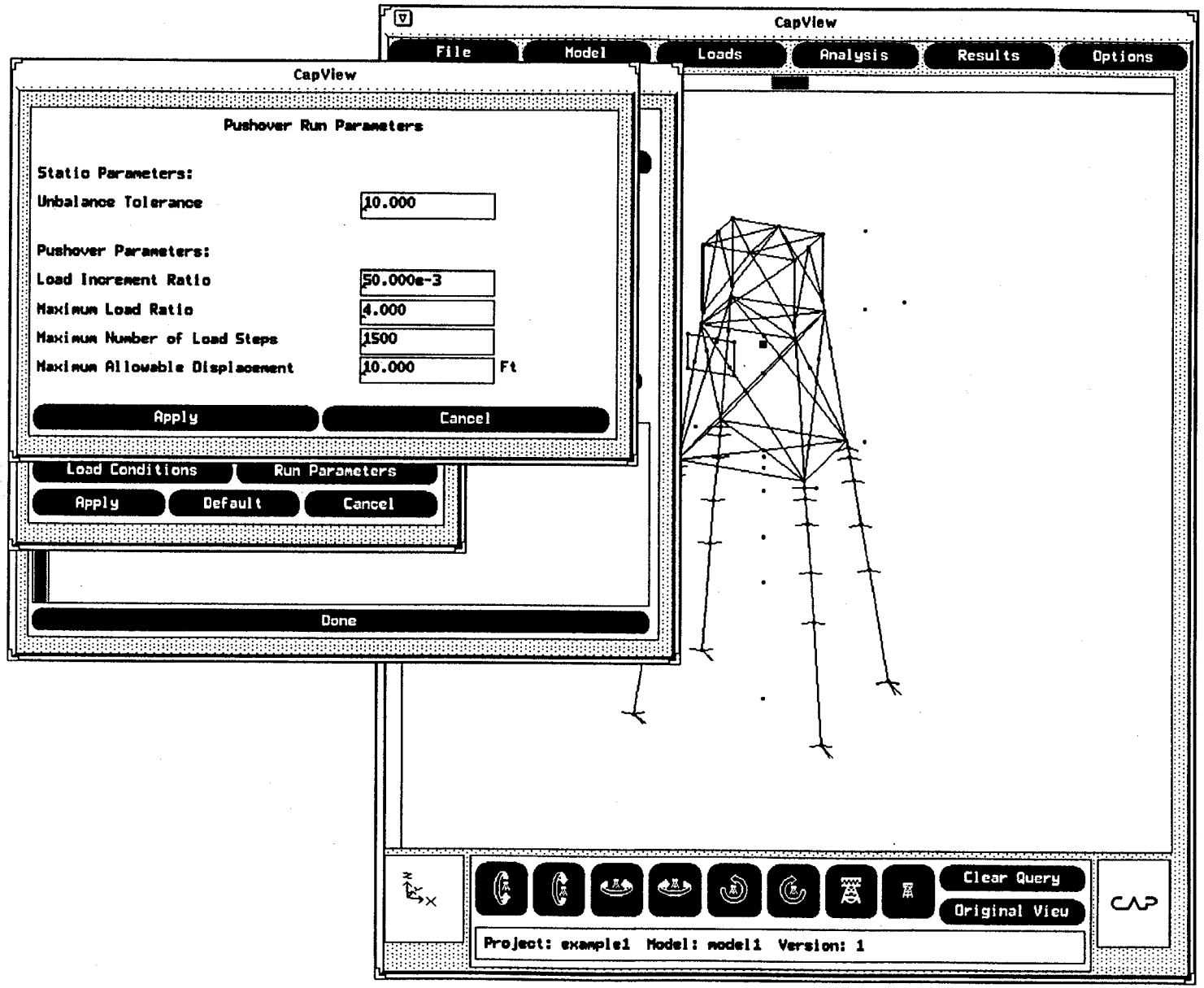

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Pushover Run Parameters
Static Parameters:
Unbalance Tolerance
10.000
Pushover Parameters:
Load Increment Ratio
50.000e-3
Maximum Load Ratio
4.000
Maximum Number of Load Steps
1500
Maximum Allowable Displacement
10.000
Ft
Apply
Cancel
Load Conditions
Run Parameters
Apply
Default
Cancel
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-10

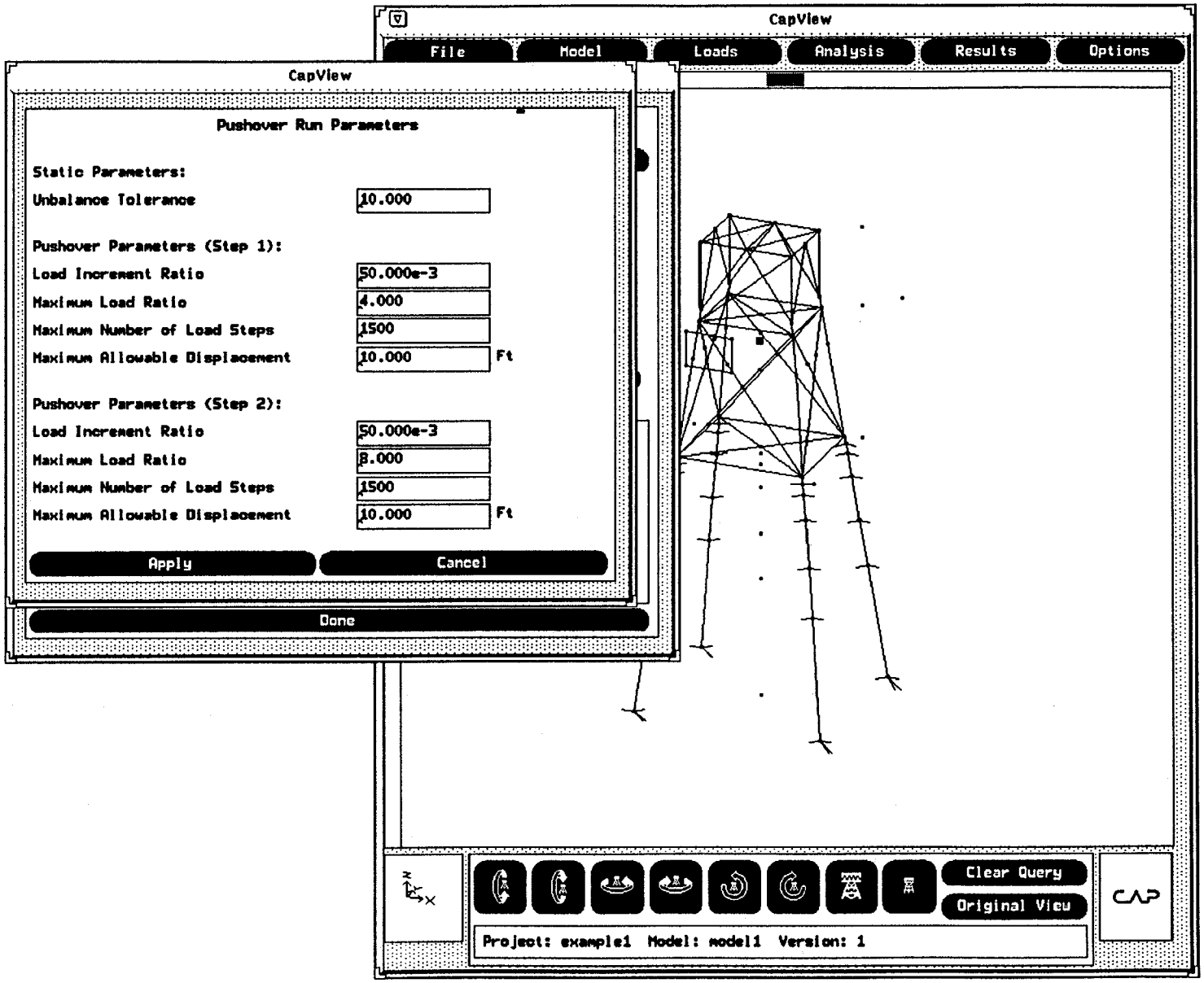

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Pushover Run Parameters
Static Parameters:
Unbalance Tolerance
10.000
Pushover Parameters (Step 1):
Load Increment Ratio
50.000e-3
Maximum Load Ratio
4.000
Maximum Number of Load Steps
1500
Maximum Allowable Displacement
10.000
Ft
Pushover Parameters (Step 2):
Load Increment Ratio
50.000e-3
Maximum Load Ratio
8.000
Maximum Number of Load Steps
1500
Maximum Allowable Displacement
10.000
Ft
Apply
Cancel
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-11

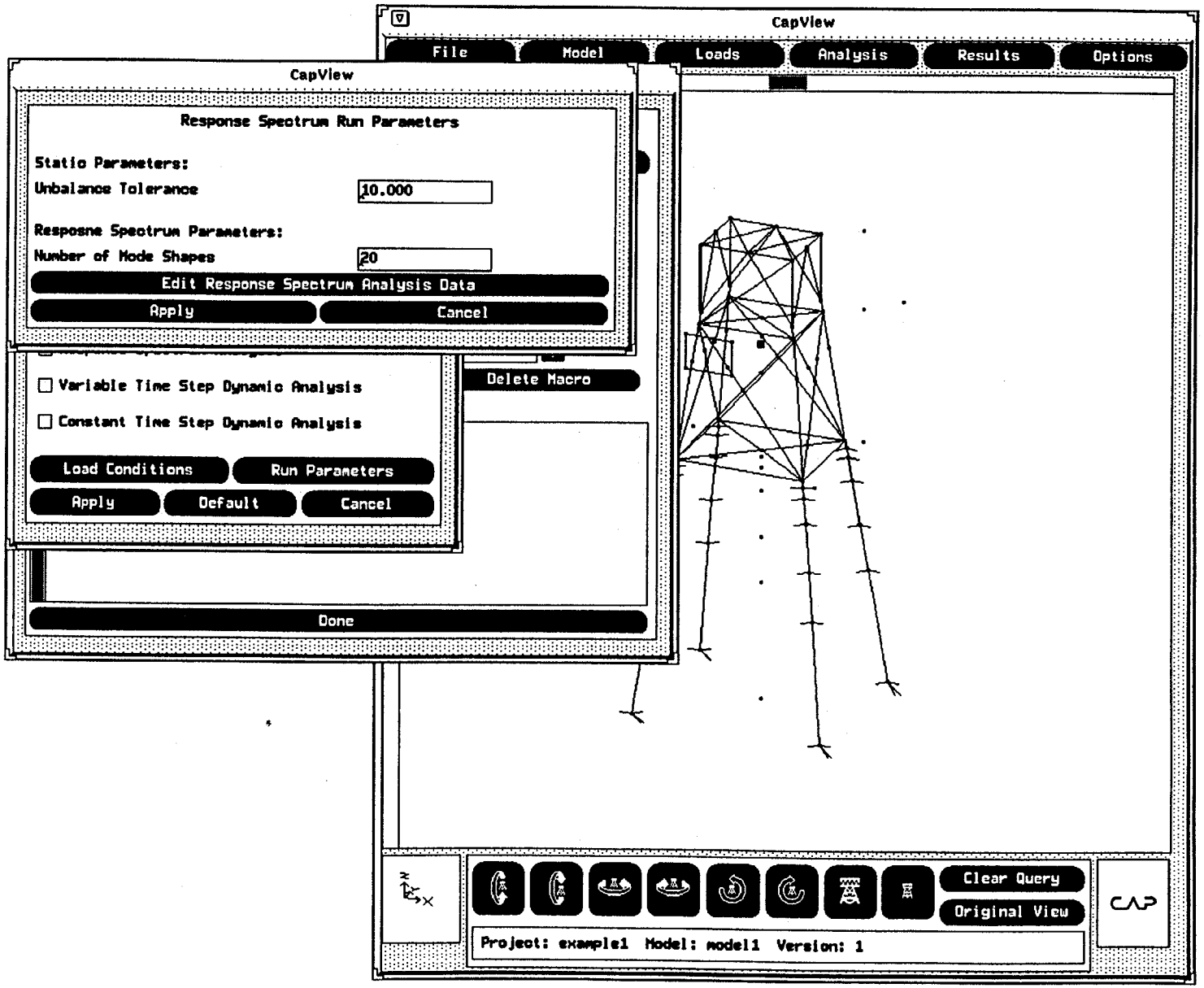

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Response Spectrum Run Parameters
Static Parameters:
Unbalance Tolerance
10.000
Response Spectrum Parameters:
Number of Mode Shapes
20
Edit Response Spectrum Analysis Data
Apply
Cancel
Variable Time Step Dynamic Analysis
Constant Time Step Dynamic Analysis
Load Conditions
Run Parameters
Apply
Default
Cancel
Delete Macro
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-12

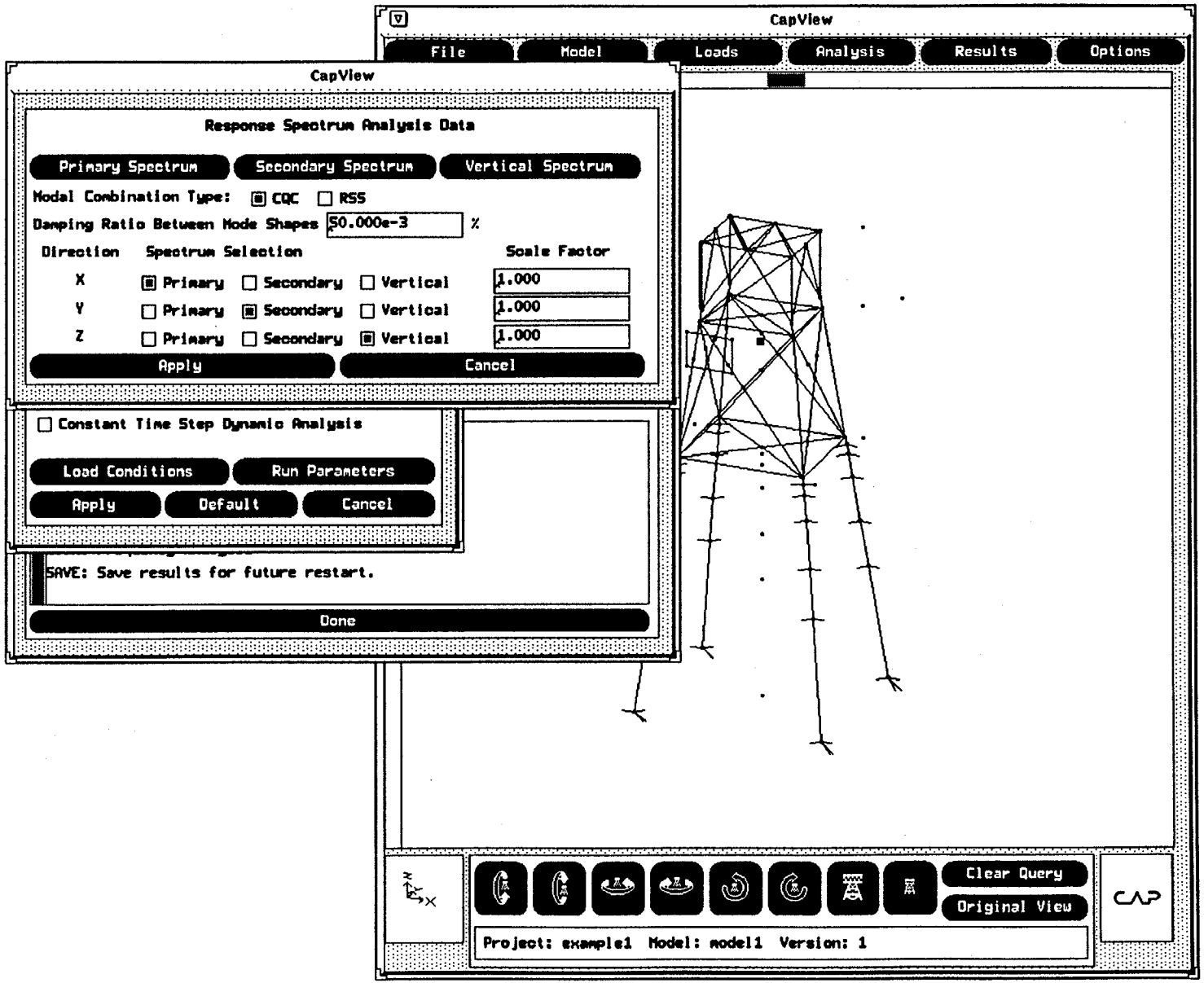

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Response Spectrum Analysis Data
Primary Spectrum
Secondary Spectrum
Vertical Spectrum
Modal Combination Type:
CQC
RSS
Damping Ratio Between Mode Shapes
50.000e-3
%
Direction
Spectrum Selection
Scale Factor
X
Primary
Secondary
Vertical
1.000
Y
Primary
Secondary
Vertical
1.000
Z
Primary
Secondary
Vertical
1.000
Apply
Cancel
Constant Time Step Dynamic Analysis
Load Conditions
Run Parameters
Apply
Default
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-13

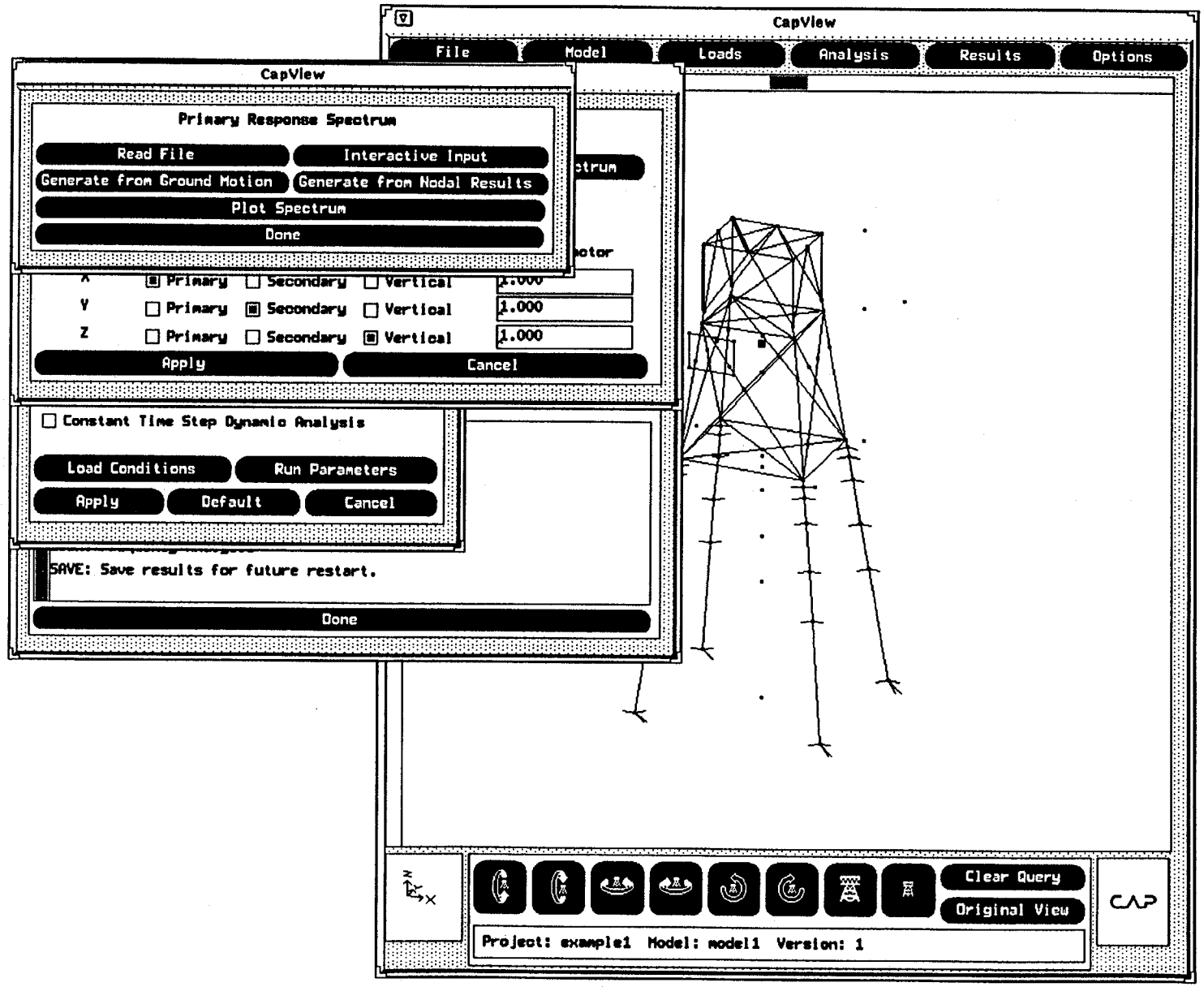

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Primary Response Spectrum
Read File
Interactive Input
Generate from Ground Motion
Generate from Nodal Results
Plot Spectrum
Done
X
Y
Z
Primary
Secondary
Vertical
1.000
1.000
Apply
Cancel
Constant Time Step Dynamic Analysis
Load Conditions
Run Parameters
Apply
Default
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-14

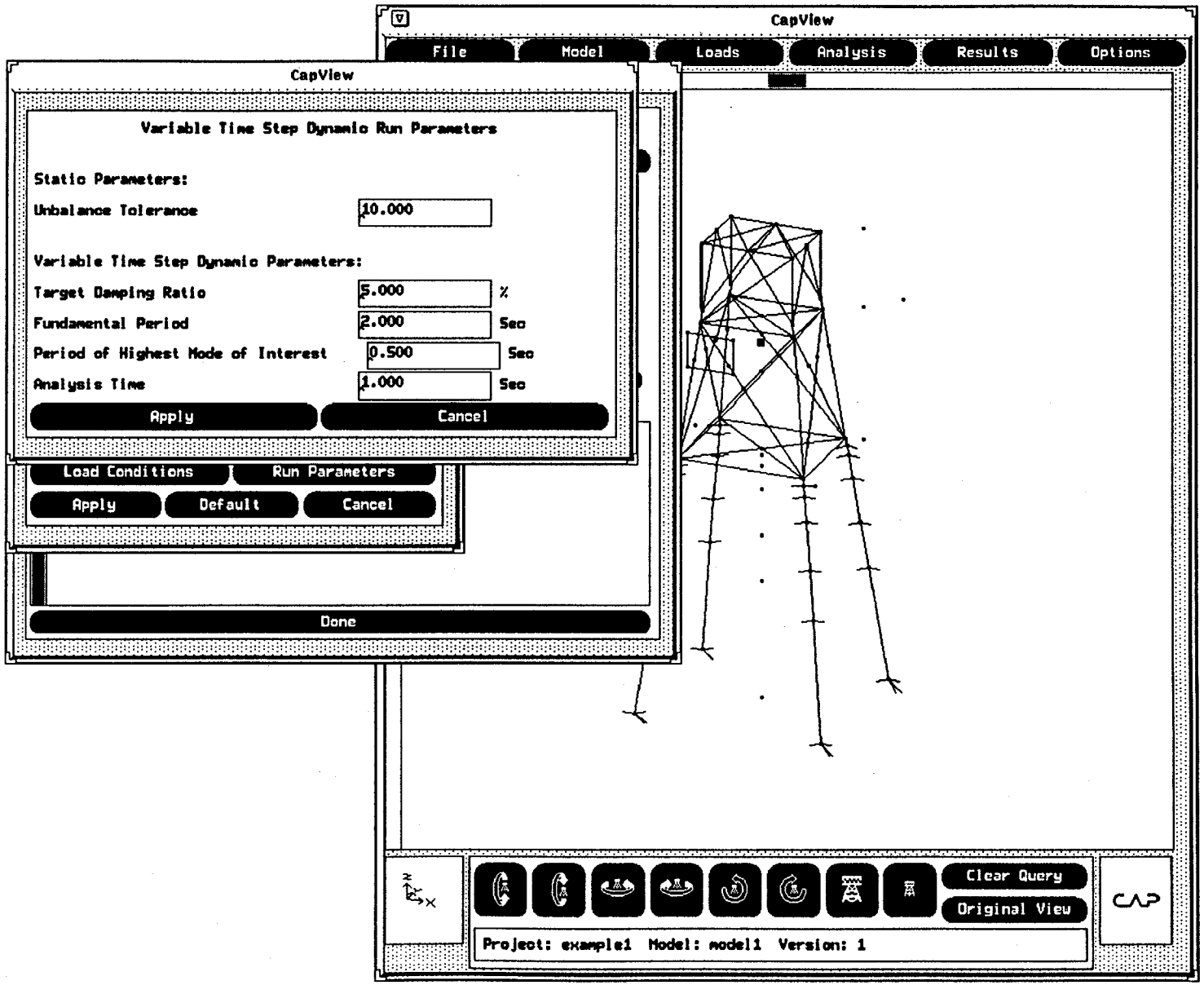

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Variable Time Step Dynamic Run Parameters
Static Parameters:
Unbalance Tolerance
10.000
Variable Time Step Dynamic Parameters:
Target Damping Ratio
5.000
%
Fundamental Period
2.000
Sec
Period of Highest Mode of Interest
0.500
Sec
Analysis Time
1.000
Sec
Apply
Cancel
Load Conditions
Run Parameters
Apply
Default
Cancel
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-15

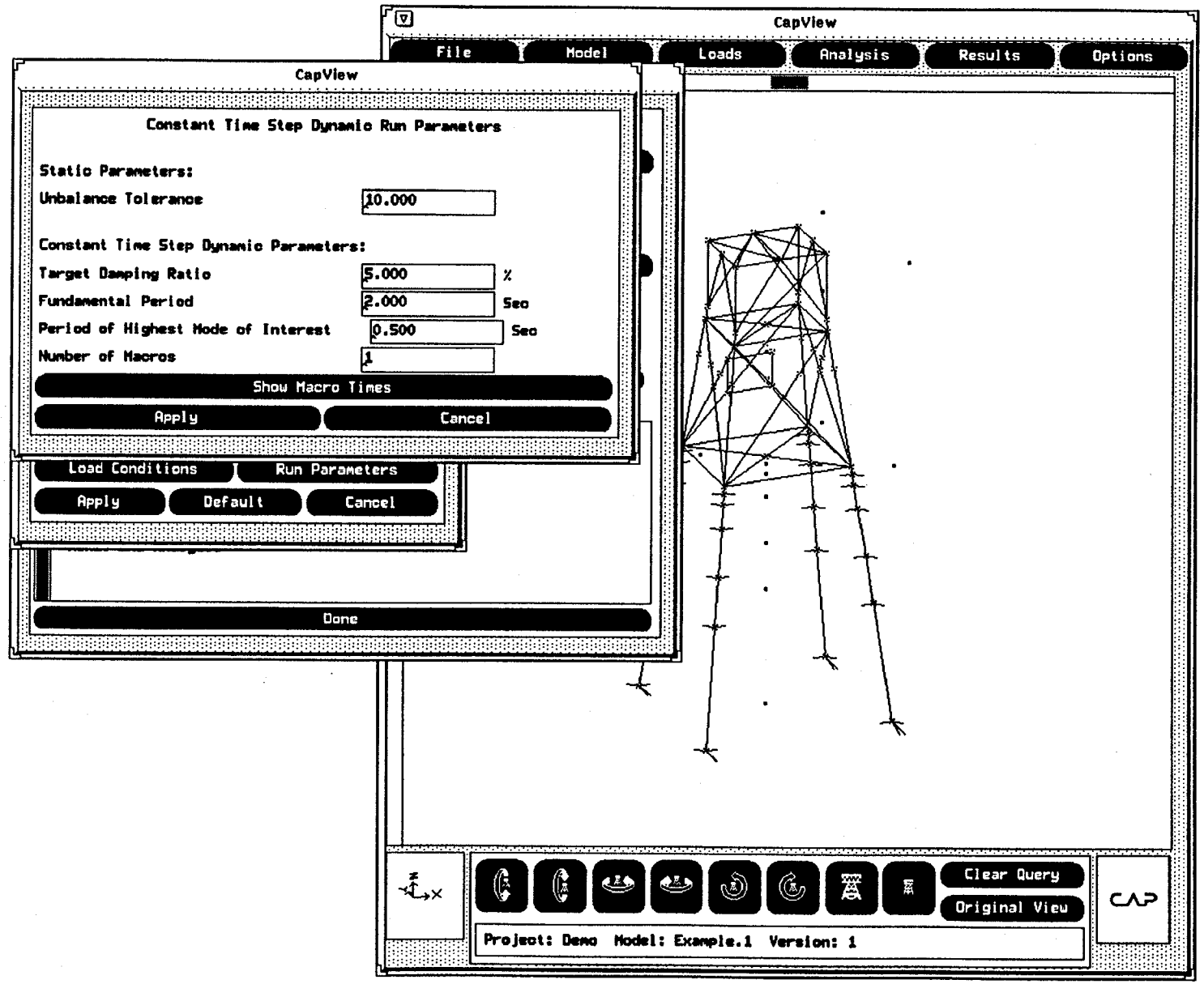

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Constant Time Step Dynamic Run Parameters
Static Parameters:
Unbalance Tolerance
10.000
Constant Time Step Dynamic Parameters:
Target Damping Ratio
5.000
%
Fundamental Period
2.000
Sec
Period of Highest Mode of Interest
0.500
Sec
Number of Macros
1
Show Macro Times
Apply
Cancel
Load Conditions
Run Parameters
Apply
Default
Cancel
Done
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 10-16

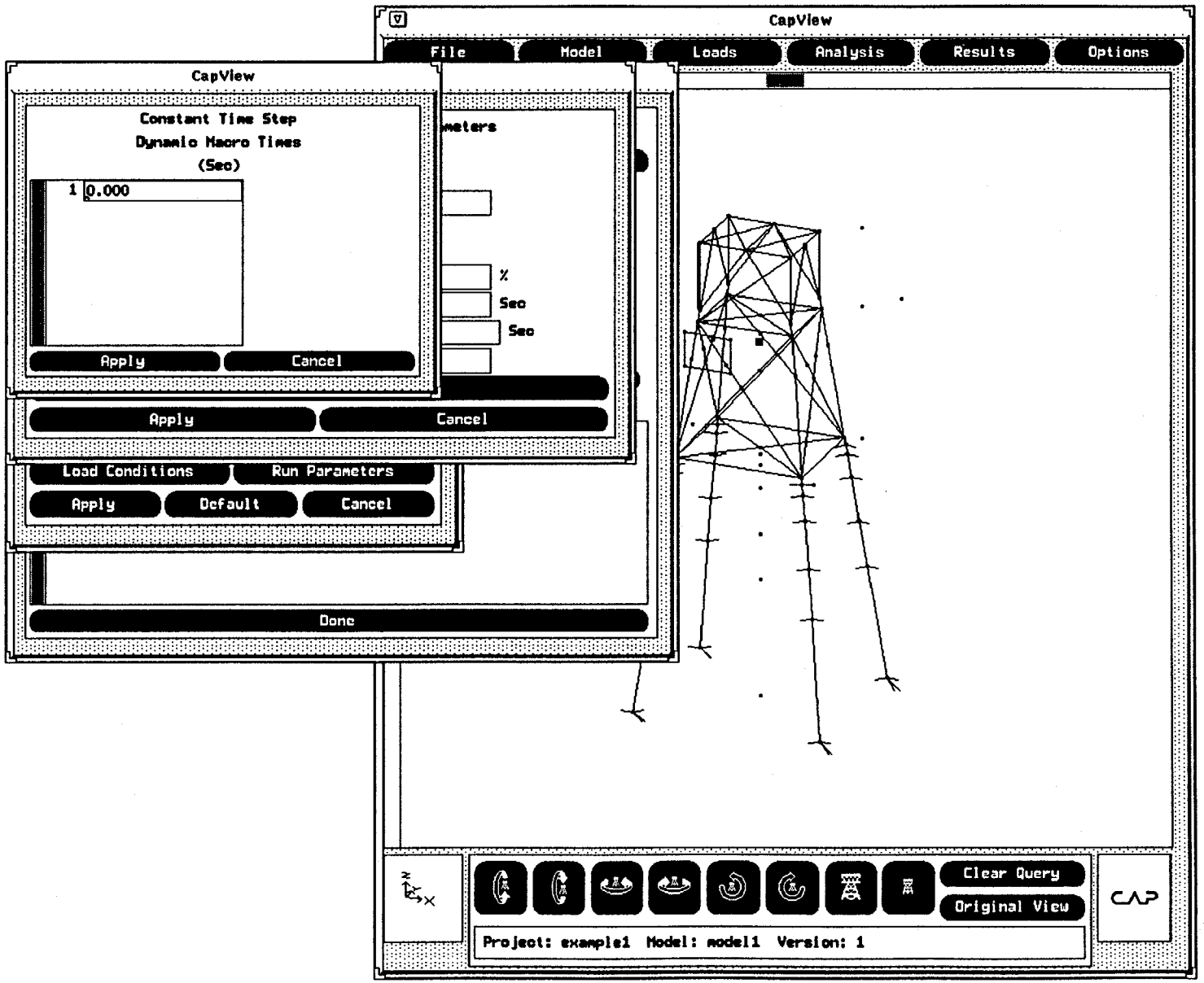

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Constant Time Step
Dynamic Macro Times
(Sec)
1 0.000
Apply
Cancel
meters
%
Sec
Sec
Apply
Cancel
Load Conditions
Run Parameters
Apply
Default
Cancel
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-17

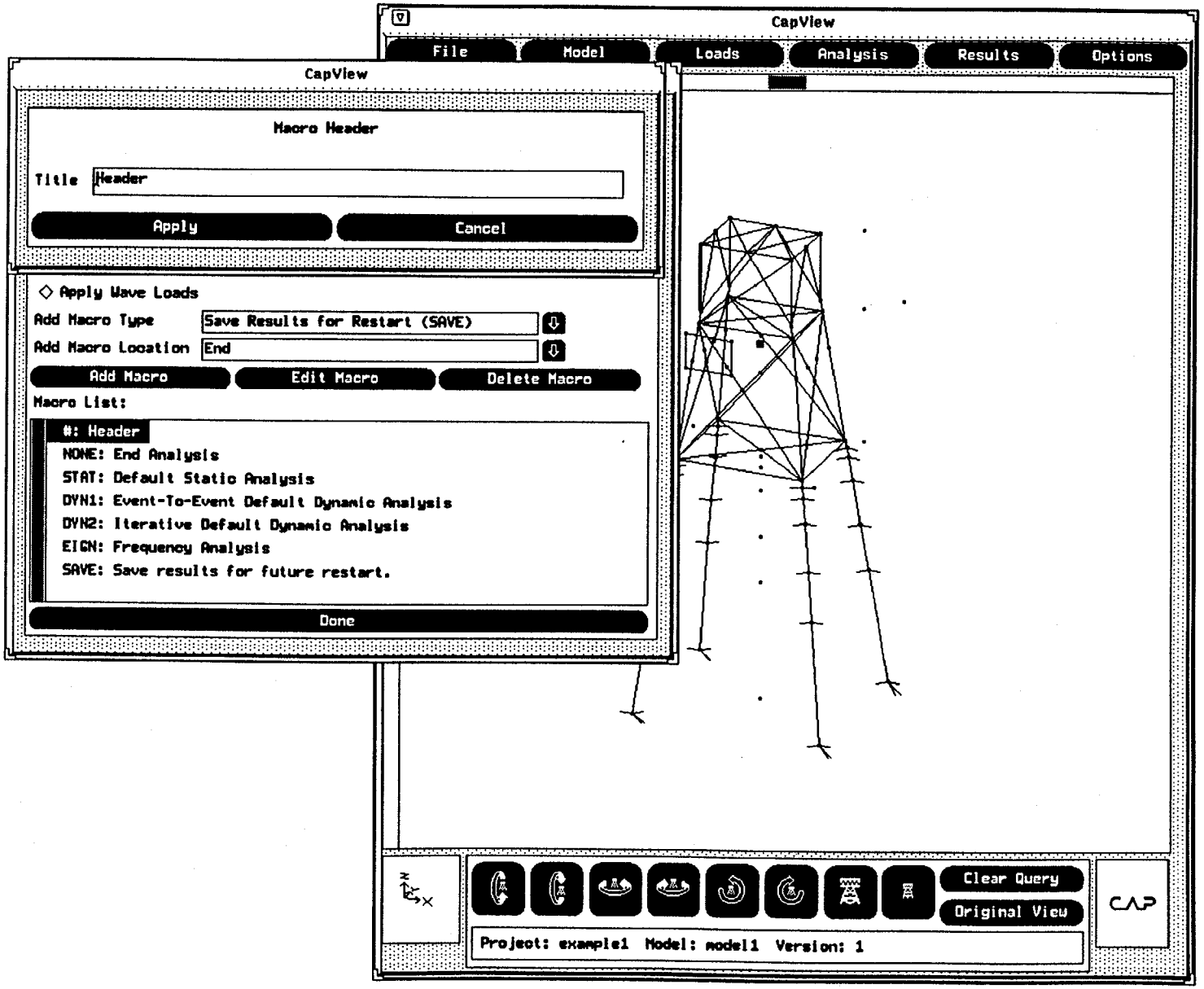

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Macro Header
Title
Header
Apply
Cancel
Apply Wave Loads
Add Macro Type
Save Results for Restart (SAVE)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-18

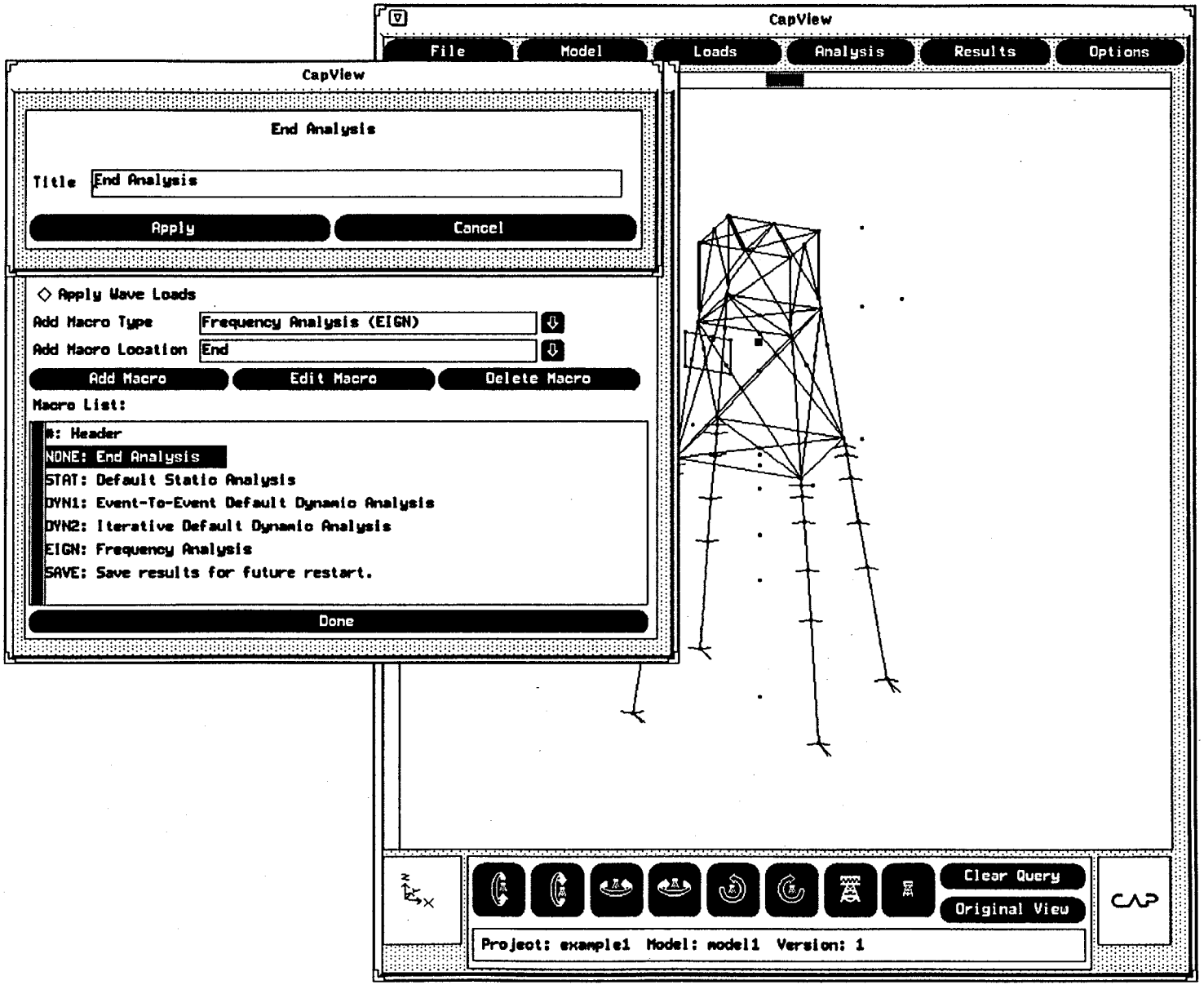

CapView
File
Model
Loads
Analysis
Results
Options
CapView
End Analysis
Title
End Analysis
Apply
Cancel
Apply Wave Loads
Add Macro Type
Frequency Analysis (EIGN)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-19

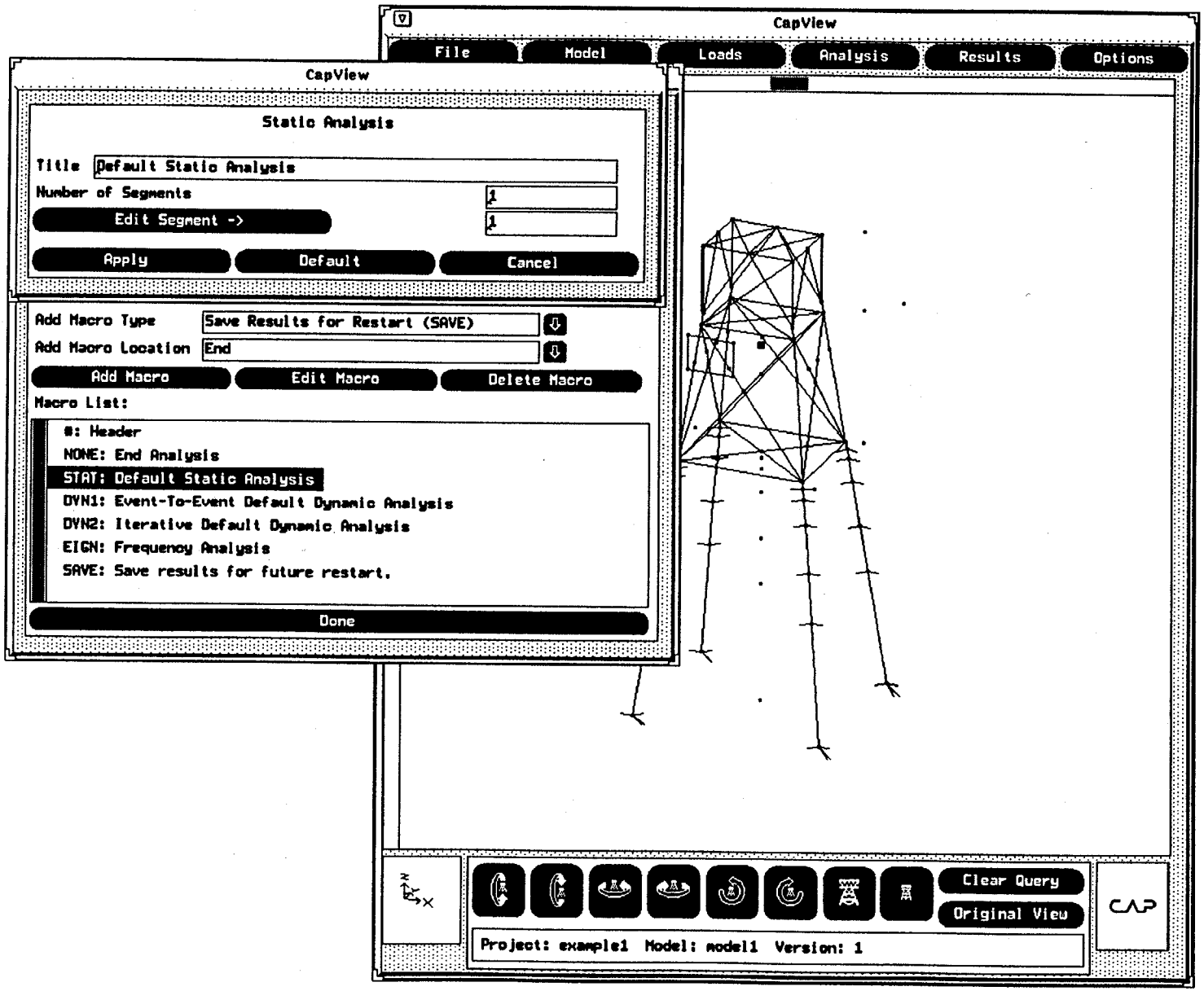

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Static Analysis
Title
Default Static Analysis
Number of Segments
1
Edit Segment ->
1
Apply
Default
Cancel
Add Macro Type
Save Results for Restart (SAVE)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-20

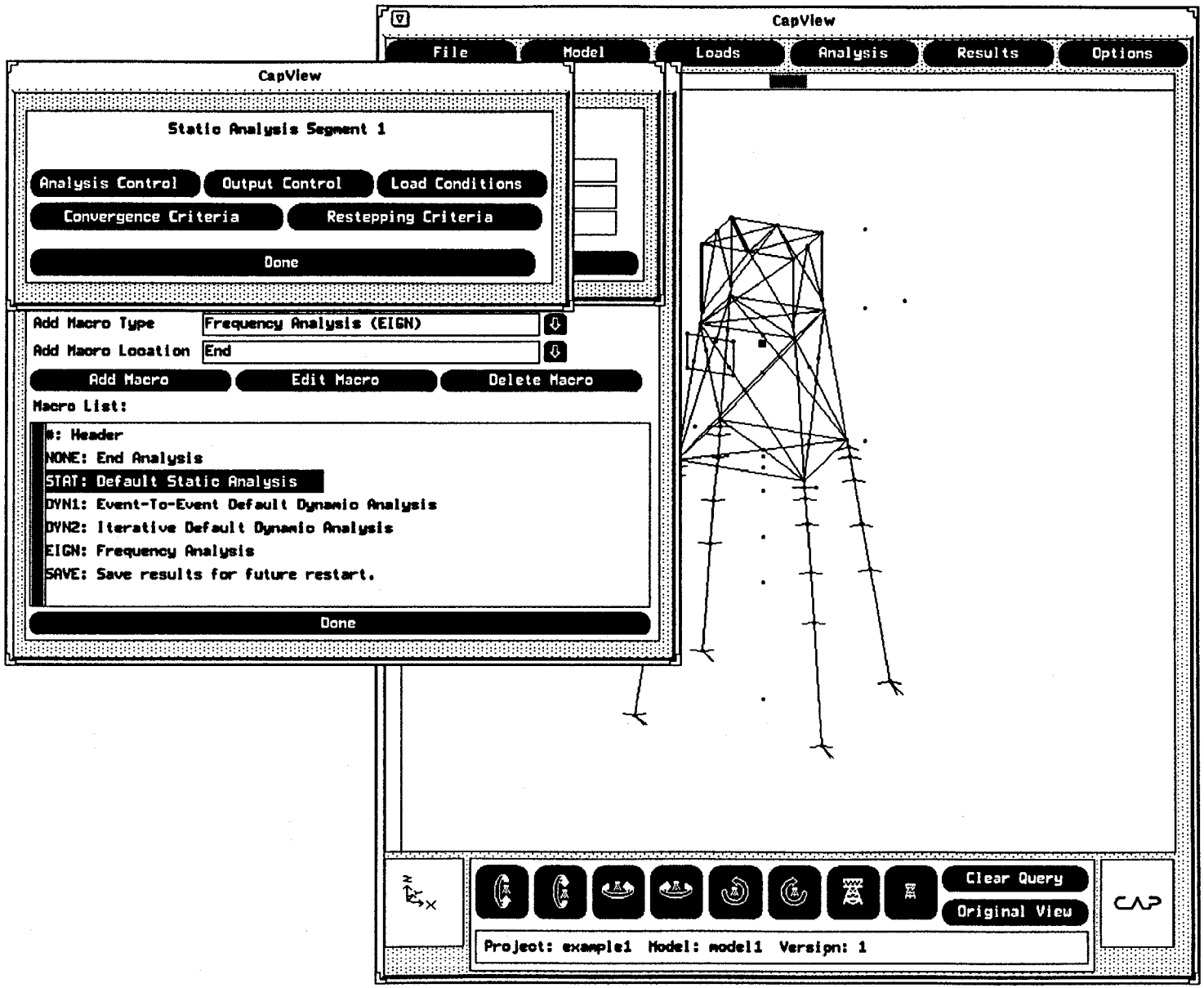

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Static Analysis Segment 1
Analysis Control
Output Control
Load Conditions
Convergence Criteria
Restepping Criteria
Done
Add Macro Type
Frequency Analysis (EIGN)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-21

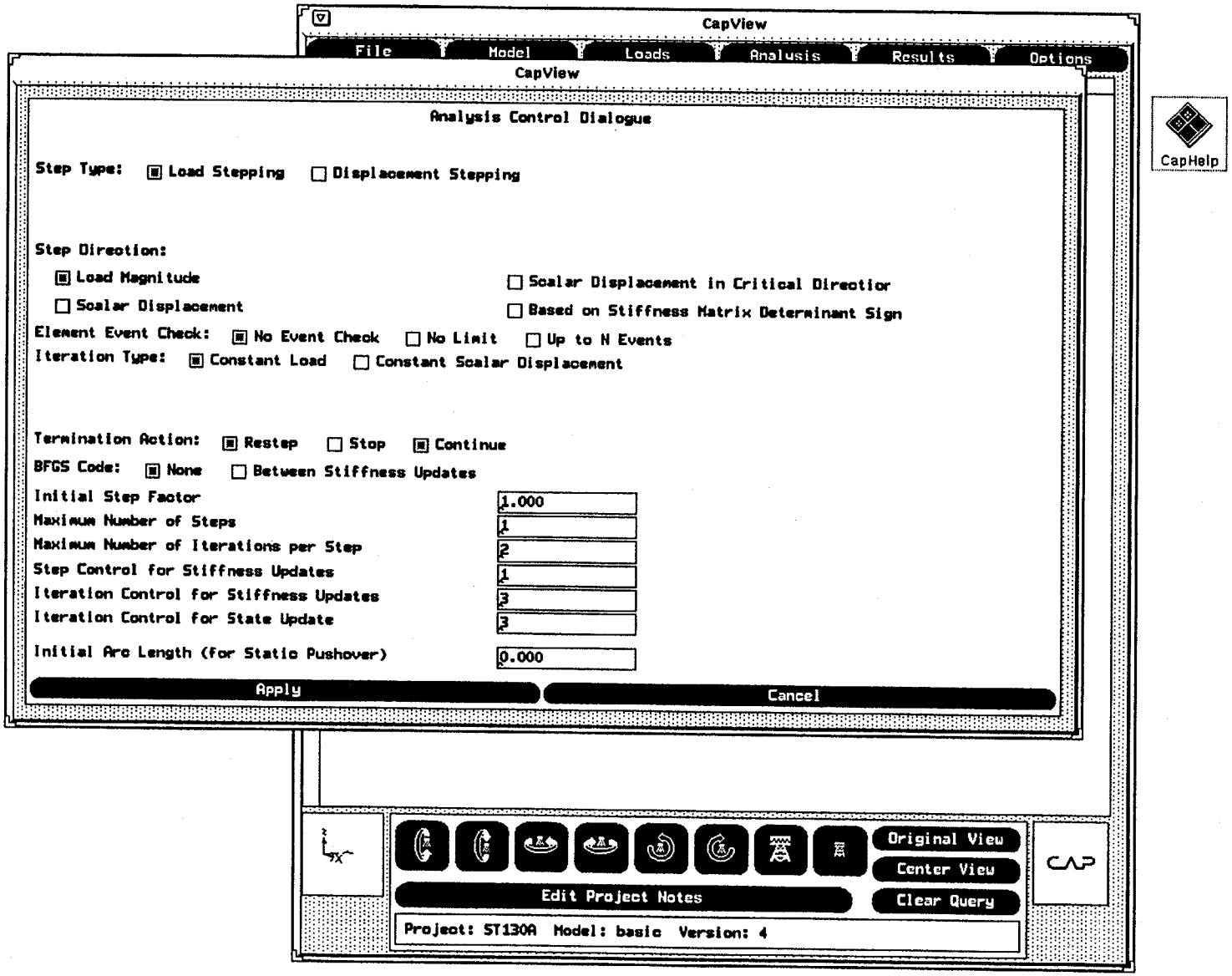

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Analysis Control Dialogue
Step Type:
Load Stepping
Displacement Stepping
Step Direction:
Load Magnitude
Scalar Displacement
Scalar Displacement in Critical Directior
Based on Stiffness Matrix Determinant Sign
Element Event Check:
No Event Check
No Limit
Up to N Events
Iteration Type:
Constant Load
Constant Scalar Displacement
Termination Action:
Restep
Stop
Continue
BFGS Code:
None
Between Stiffness Updates
Initial Step Factor
1.000
Maximum Number of Steps
1
Maximum Number of Iterations per Step
2
Step Control for Stiffness Updates
1
Iteration Control for Stiffness Updates
3
Iteration Control for State Update
3
Initial Arc Length (for Static Pushover)
0.000
Apply
Cancel
CapHelp
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 10-22

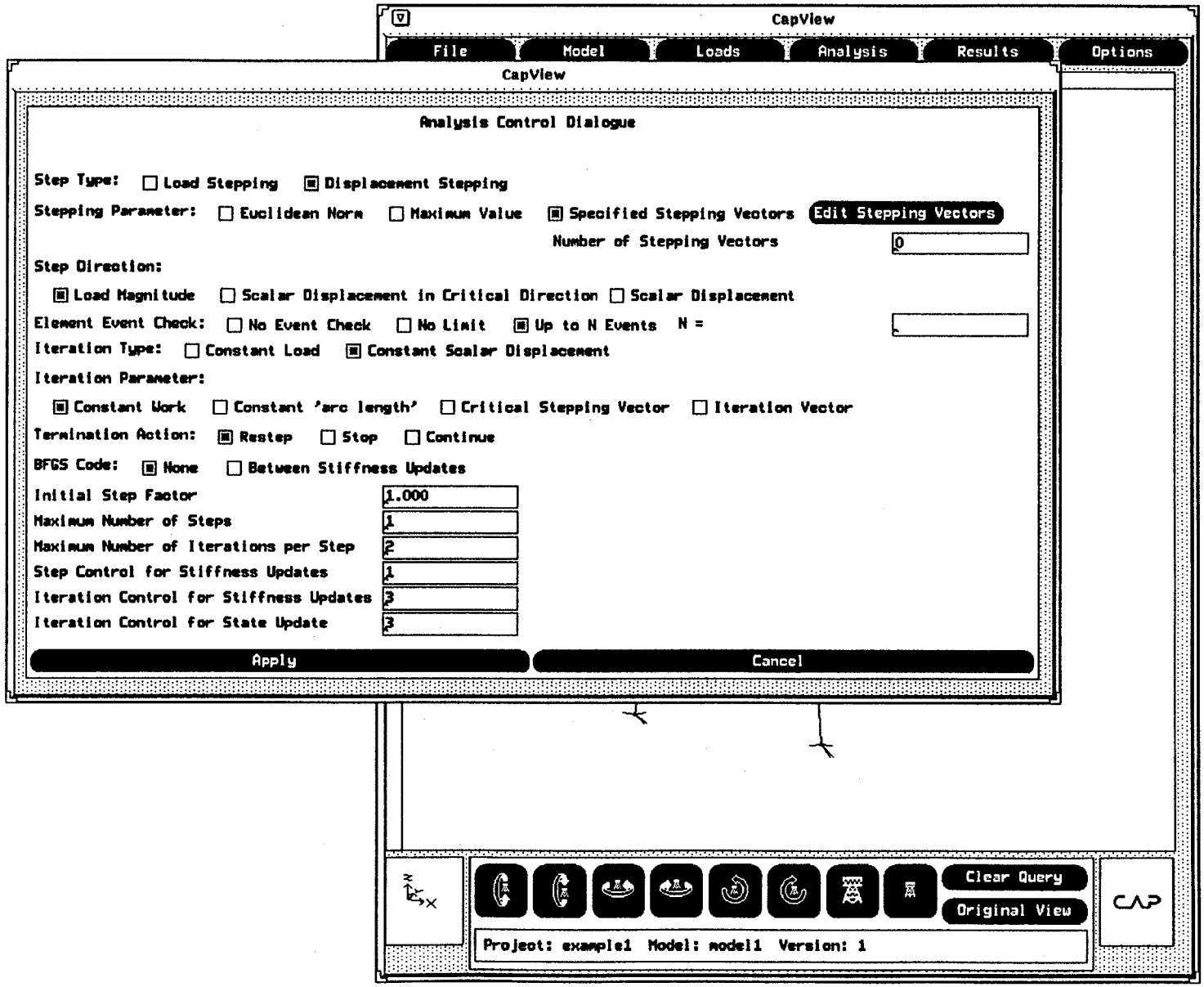

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Analysis Control Dialogue
Step Type: Load Stepping Displacement Stepping
Stepping Parameter: Euclidean Norm Maximum Value Specified Stepping Vectors Edit Stepping Vectors
Number of Stepping Vectors 0
Step Direction:
Load Magnitude Scalar Displacement in Critical Direction Scalar Displacement
Element Event Check: No Event Check No Limit Up to N Events N =
Iteration Type: Constant Load Constant Scalar Displacement
Iteration Parameter:
Constant Work Constant 'arc length' Critical Stepping Vector Iteration Vector
Termination Action: Restep Stop Continue
BFGS Code: None Between Stiffness Updates
Initial Step Factor 1.000
Maximum Number of Steps 1
Maximum Number of Iterations per Step 2
Step Control for Stiffness Updates 1
Iteration Control for Stiffness Updates 3
Iteration Control for State Update 3
Apply
Cancel
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 10-23

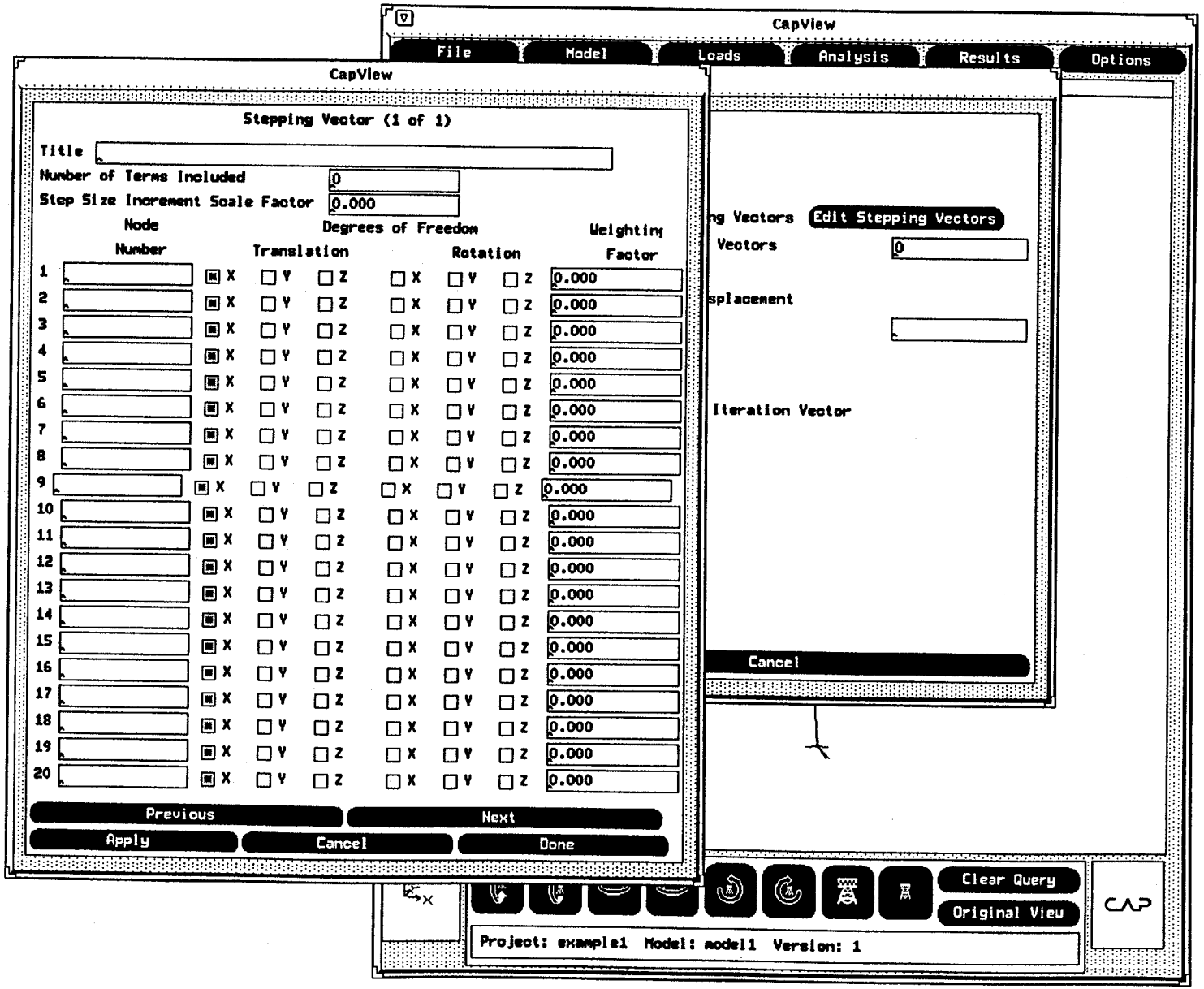

CapView
File
Model
Loads
Analysis
Results
Options
Stepping Vector (1 of 1)
Title
Number of Terms Included
0
Step Size Increment Scale Factor
0.000
Node Number
Degrees of Freedom
Translation
Rotation
Weighting Factor
Previous
Next
Apply
Cancel
Done
Edit Stepping Vectors
Vectors
Iteration Vector
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-24

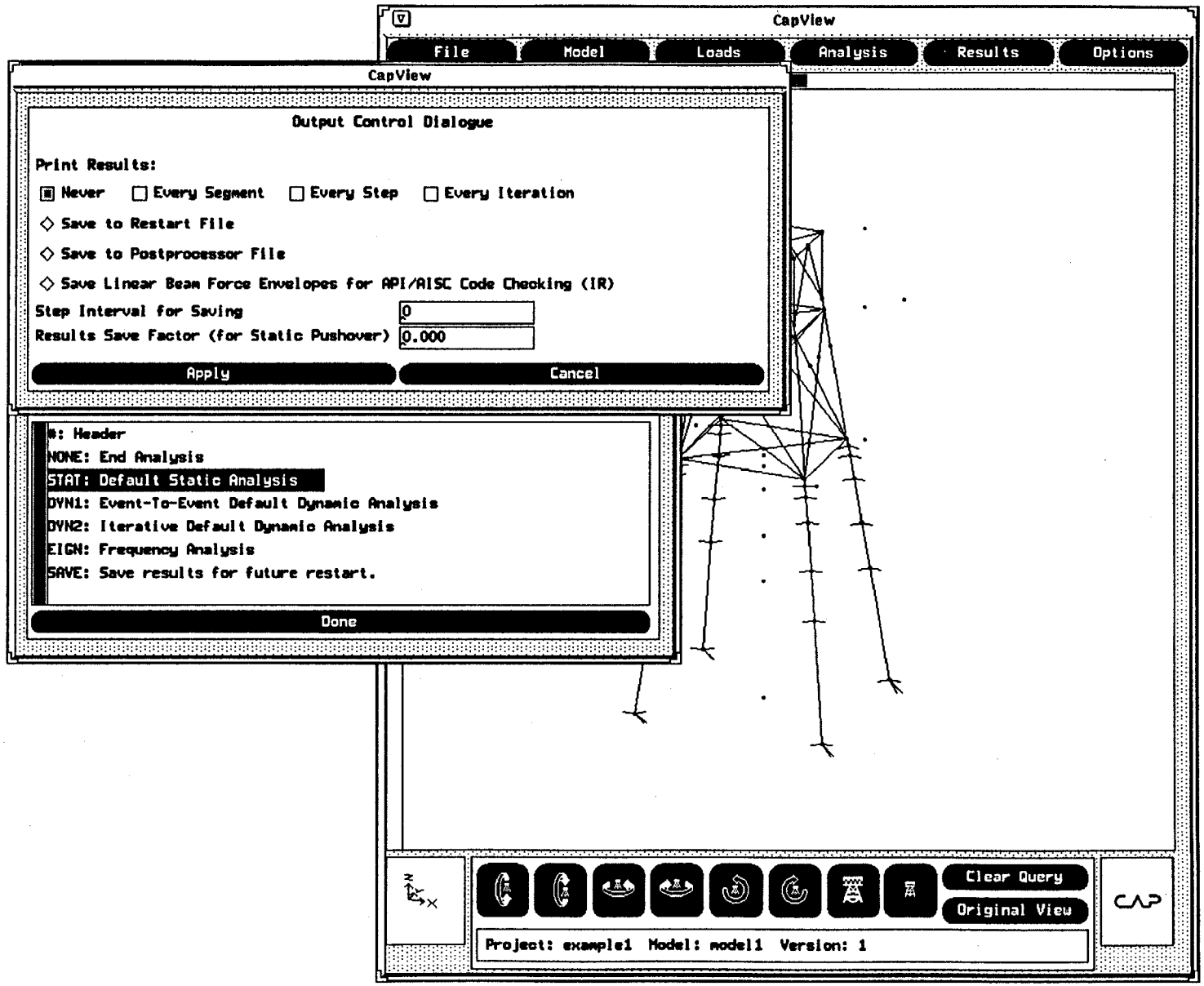

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Output Control Dialogue
Print Results:
Never
Every Segment
Every Step
Every Iteration
Save to Restart File
Save to Postprocessor File
Save Linear Beam Force Envelopes for API/AISC Code Checking (IR)
Step Interval for Saving
0
Results Save Factor (for Static Pushover)
0.000
Apply
Cancel
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-25

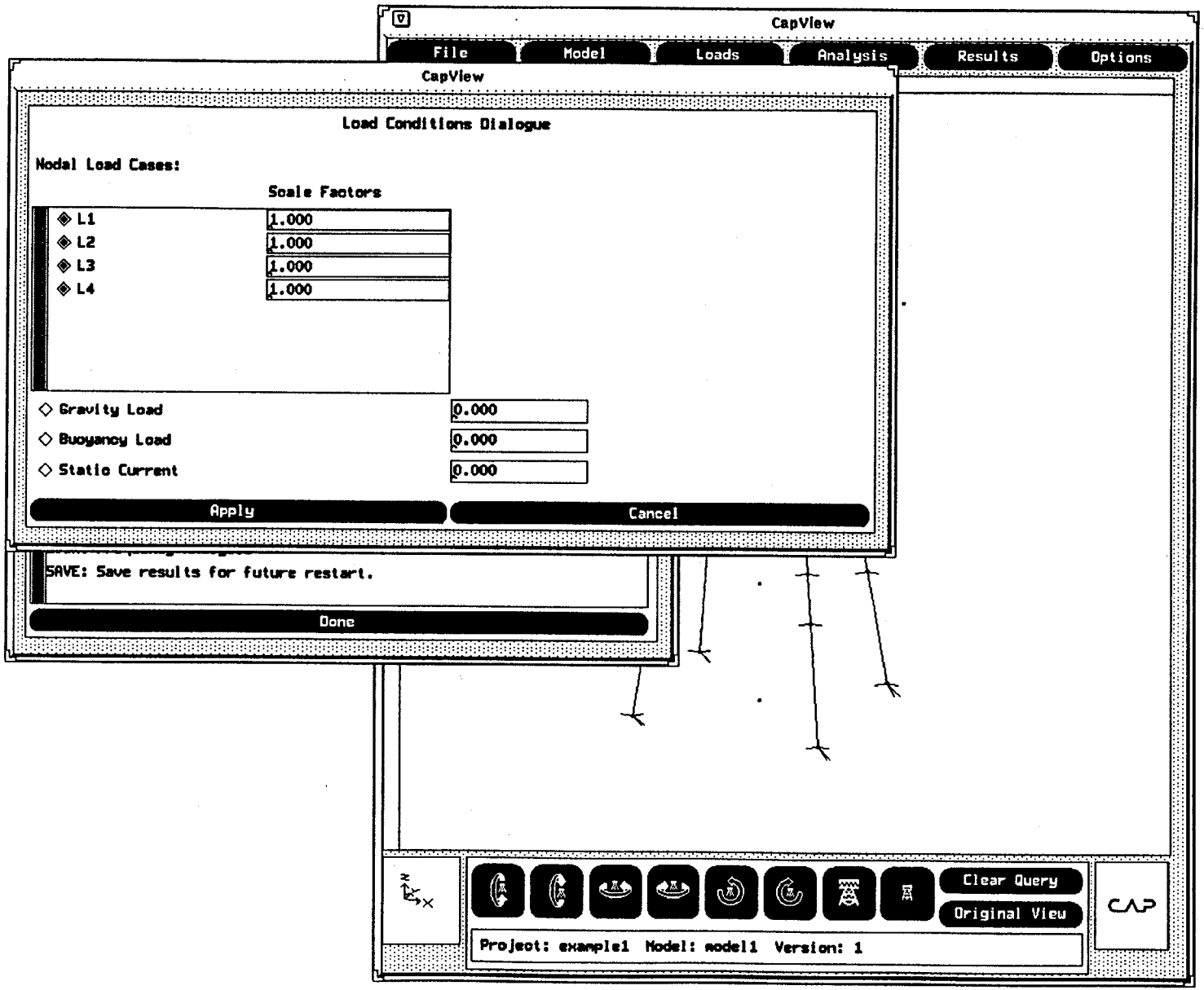

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Conditions Dialogue
Nodal Load Cases:
Scale Factors
L1
1.000
L2
1.000
L3
1.000
L4
1.000
Gravity Load
0.000
Buoyancy Load
0.000
Static Current
0.000
Apply
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-26

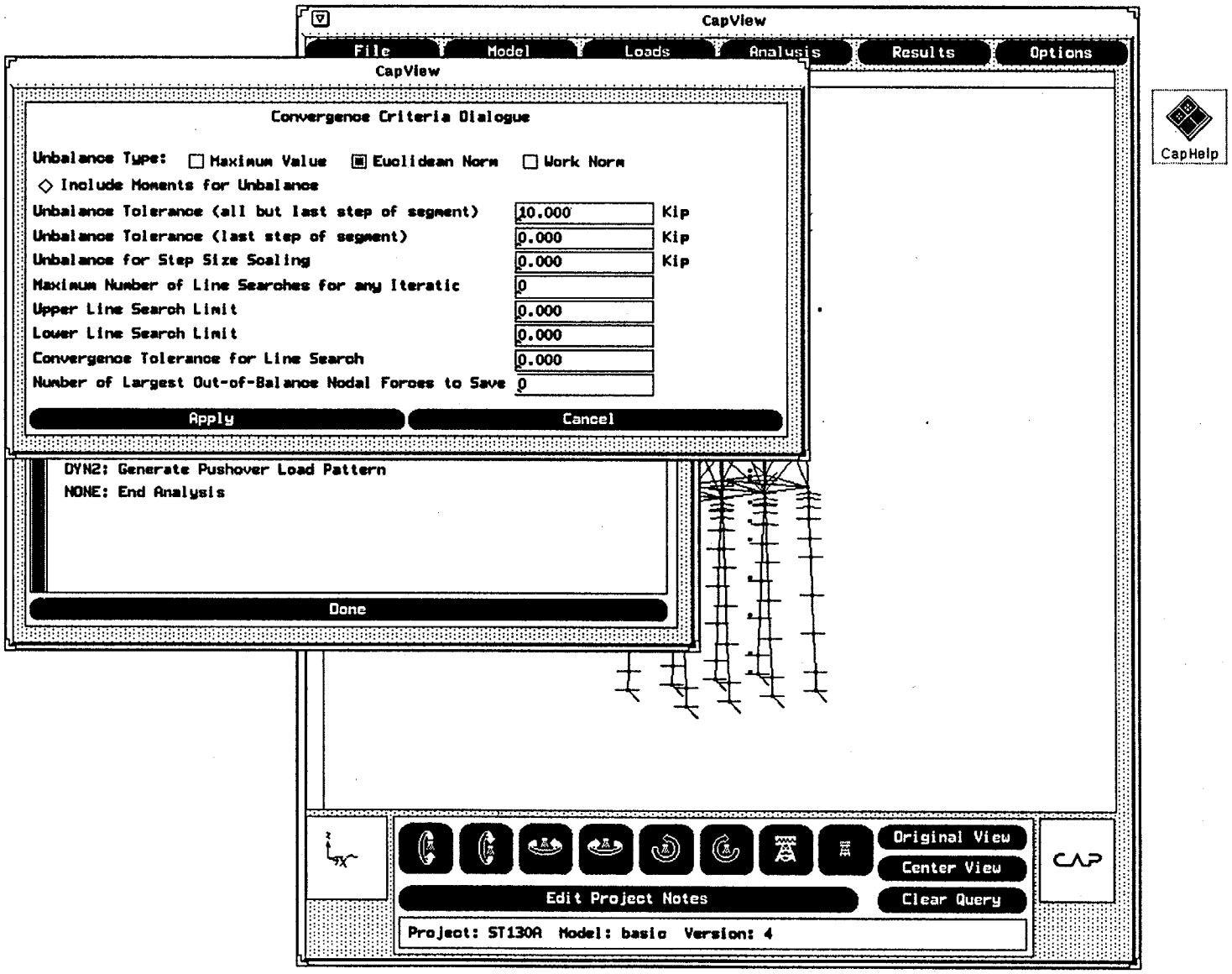

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Convergence Criteria Dialogue
Unbalance Type:
Maximum Value
Euclidean Norm
Work Norm
Include Moments for Unbalance
Unbalance Tolerance (all but last step of segment)
10.000
Kip
Unbalance Tolerance (last step of segment)
0.000
Kip
Unbalance for Step Size Scaling
0.000
Kip
Maximum Number of Line Searches for any Iteratic
0
Upper Line Search Limit
0.000
Lower Line Search Limit
0.000
Convergence Tolerance for Line Search
0.000
Number of Largest Out-of-Balance Nodal Forces to Save
0
Apply
Cancel
DYN2: Generate Pushover Load Pattern
NONE: End Analysis
Done
Original View
Center View
Edit Project Notes
Clear Query
Project: ST130A Model: basic Version: 4

Figure 10-27

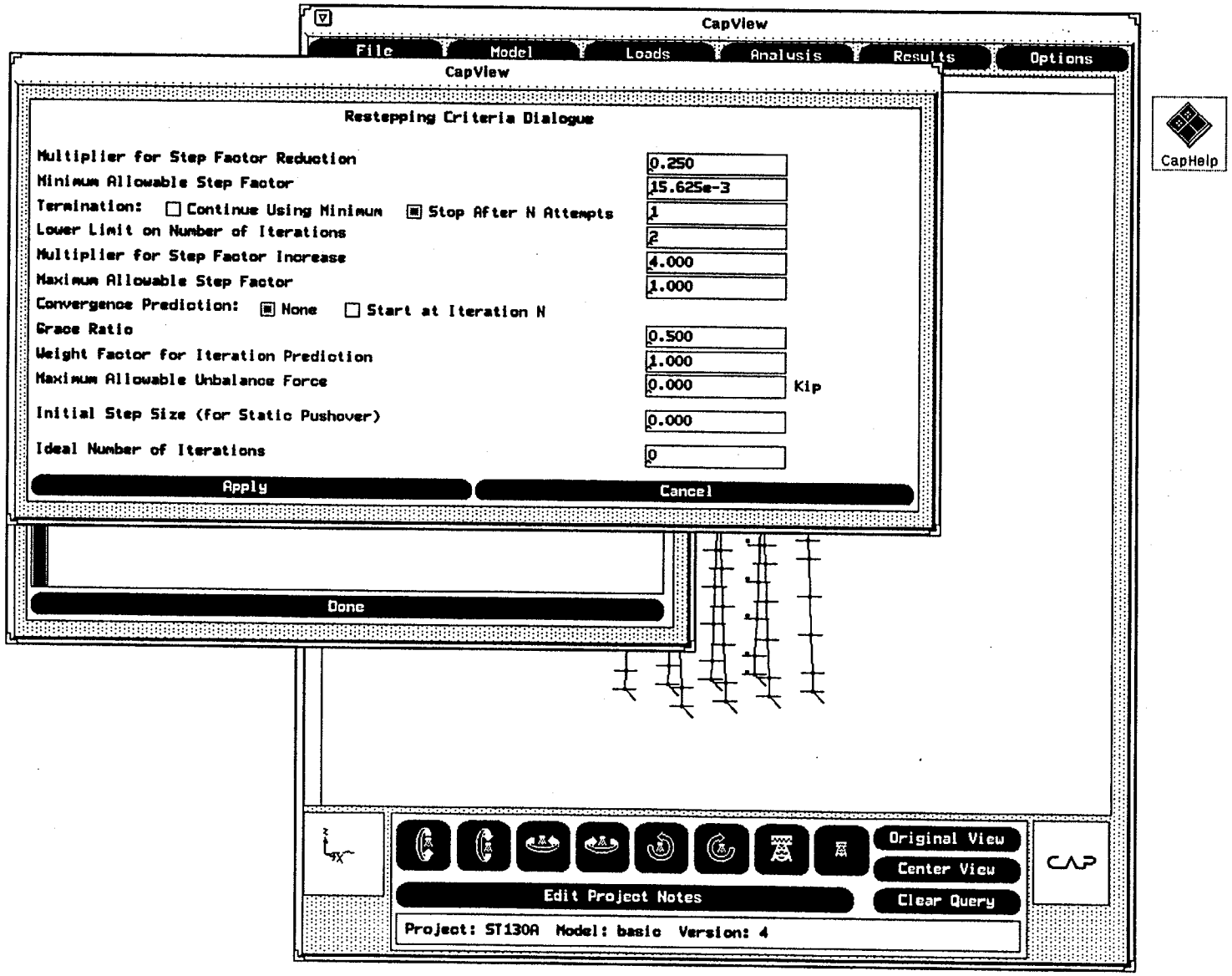

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Restepping Criteria Dialogue
Multiplier for Step Factor Reduction
0.250
Minimum Allowable Step Factor
15.625e-3
Termination:
Continue Using Minimum
Stop After N Attempts
1
Lower Limit on Number of Iterations
2
Multiplier for Step Factor Increase
4.000
Maximum Allowable Step Factor
1.000
Convergence Prediction:
None
Start at Iteration N
Grace Ratio
0.500
Weight Factor for Iteration Prediction
1.000
Maximum Allowable Unbalance Force
0.000
Kip
Initial Step Size (for Static Pushover)
0.000
Ideal Number of Iterations
0
Apply
Cancel
Done
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4
CAP

Figure 10-28

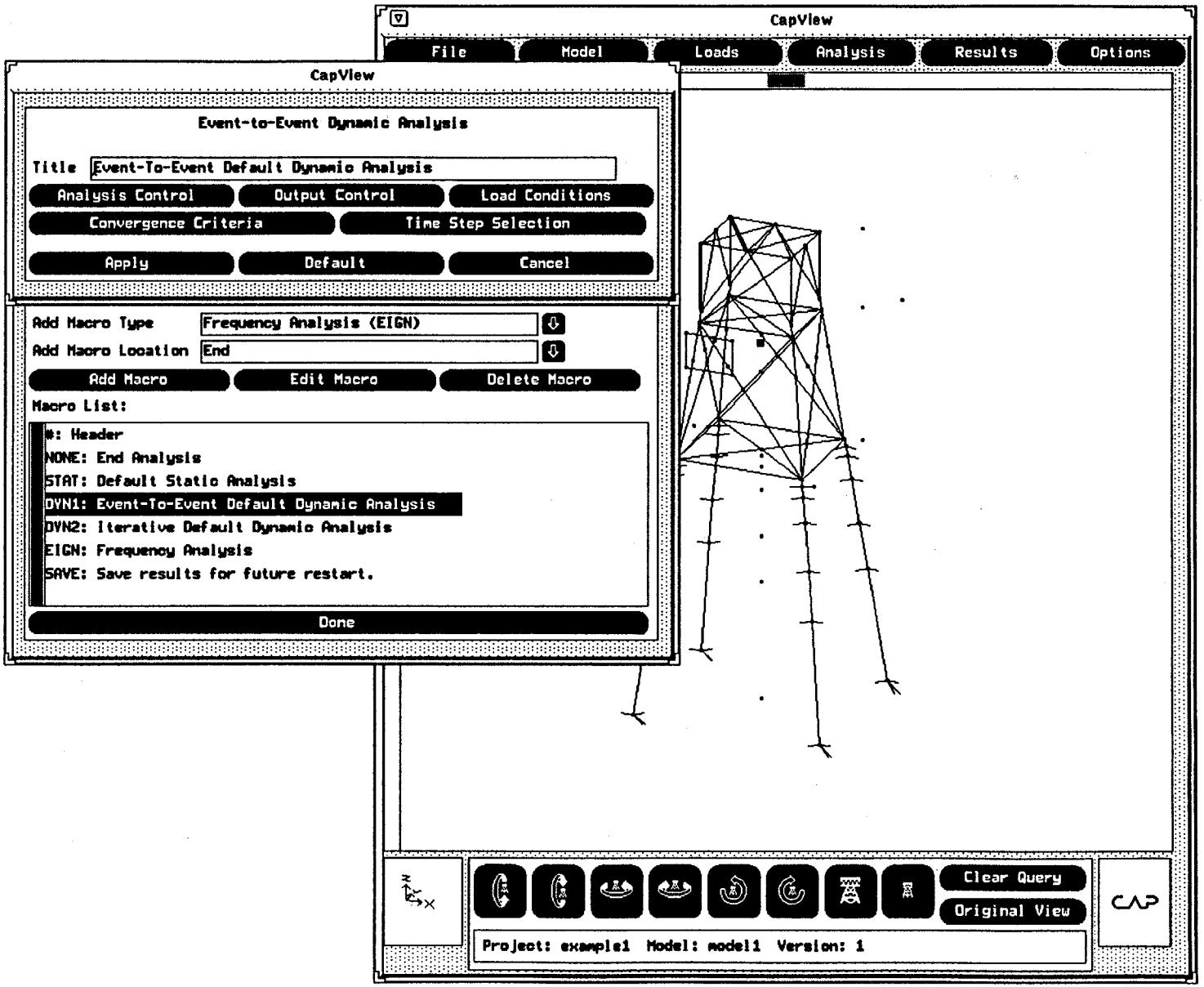

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Event-to-Event Dynamic Analysis
Title
Event-To-Event Default Dynamic Analysis
Analysis Control
Output Control
Load Conditions
Convergence Criteria
Time Step Selection
Apply
Default
Cancel
Add Macro Type
Frequency Analysis (EIGN)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-29

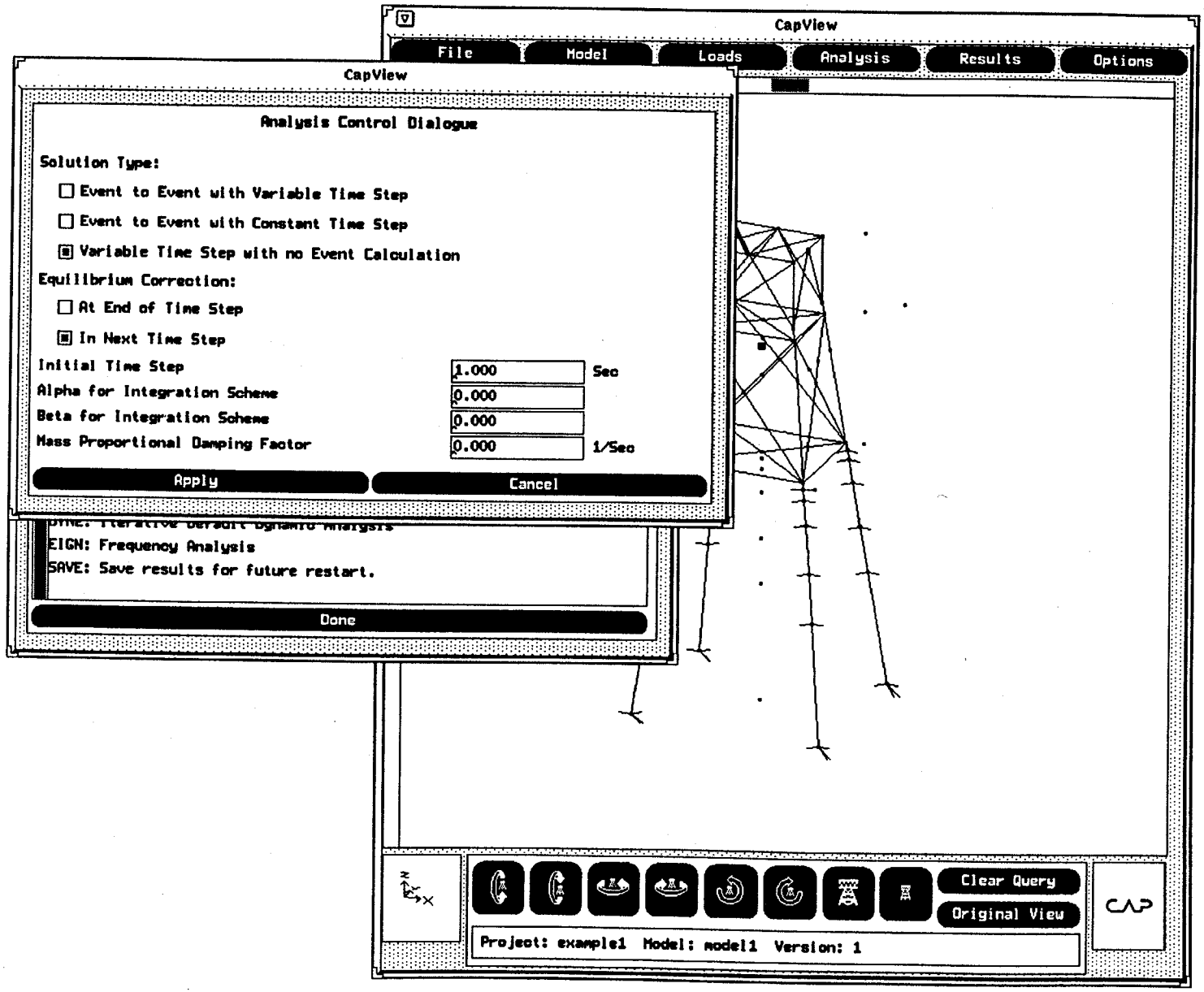

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Analysis Control Dialogue
Solution Type:
Event to Event with Variable Time Step
Event to Event with Constant Time Step
Variable Time Step with no Event Calculation
Equilibrium Correction:
At End of Time Step
In Next Time Step
Initial Time Step
1.000
Sec
Alpha for Integration Scheme
0.000
Beta for Integration Scheme
0.000
Mass Proportional Damping Factor
0.000
1/Sec
Apply
Cancel
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-30

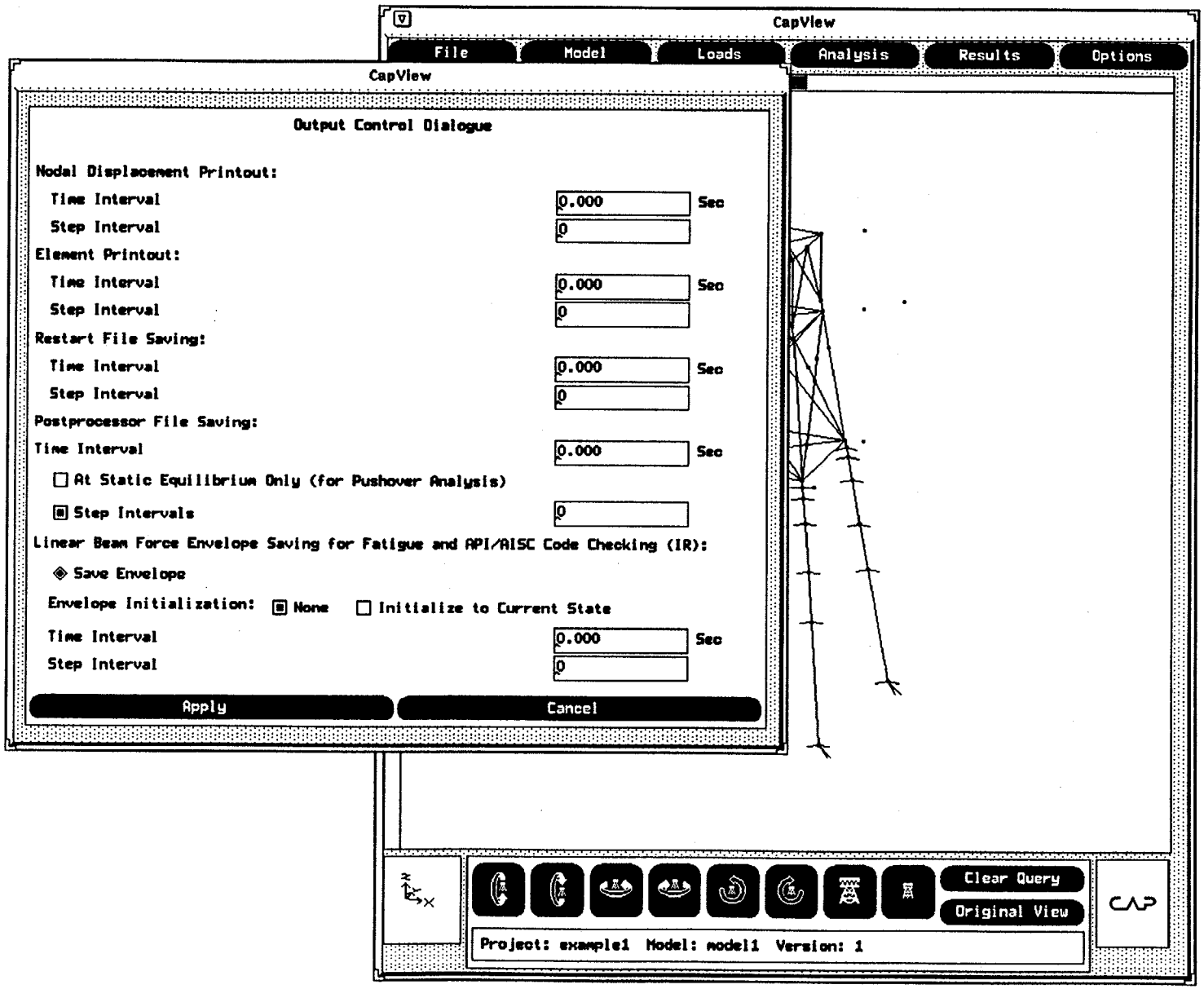

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Output Control Dialogue
Nodal Displacement Printout:
Time Interval
0.000
Sec
Step Interval
0
Element Printout:
Time Interval
0.000
Sec
Step Interval
0
Restart File Saving:
Time Interval
0.000
Sec
Step Interval
0
Postprocessor File Saving:
Time Interval
0.000
Sec
At Static Equilibrium Only (for Pushover Analysis)
Step Intervals
0
Linear Beam Force Envelope Saving for Fatigue and API/AISC Code Checking (IR):
Save Envelope
Envelope Initialization:
None
Initialize to Current State
Time Interval
0.000
Sec
Step Interval
0
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-31

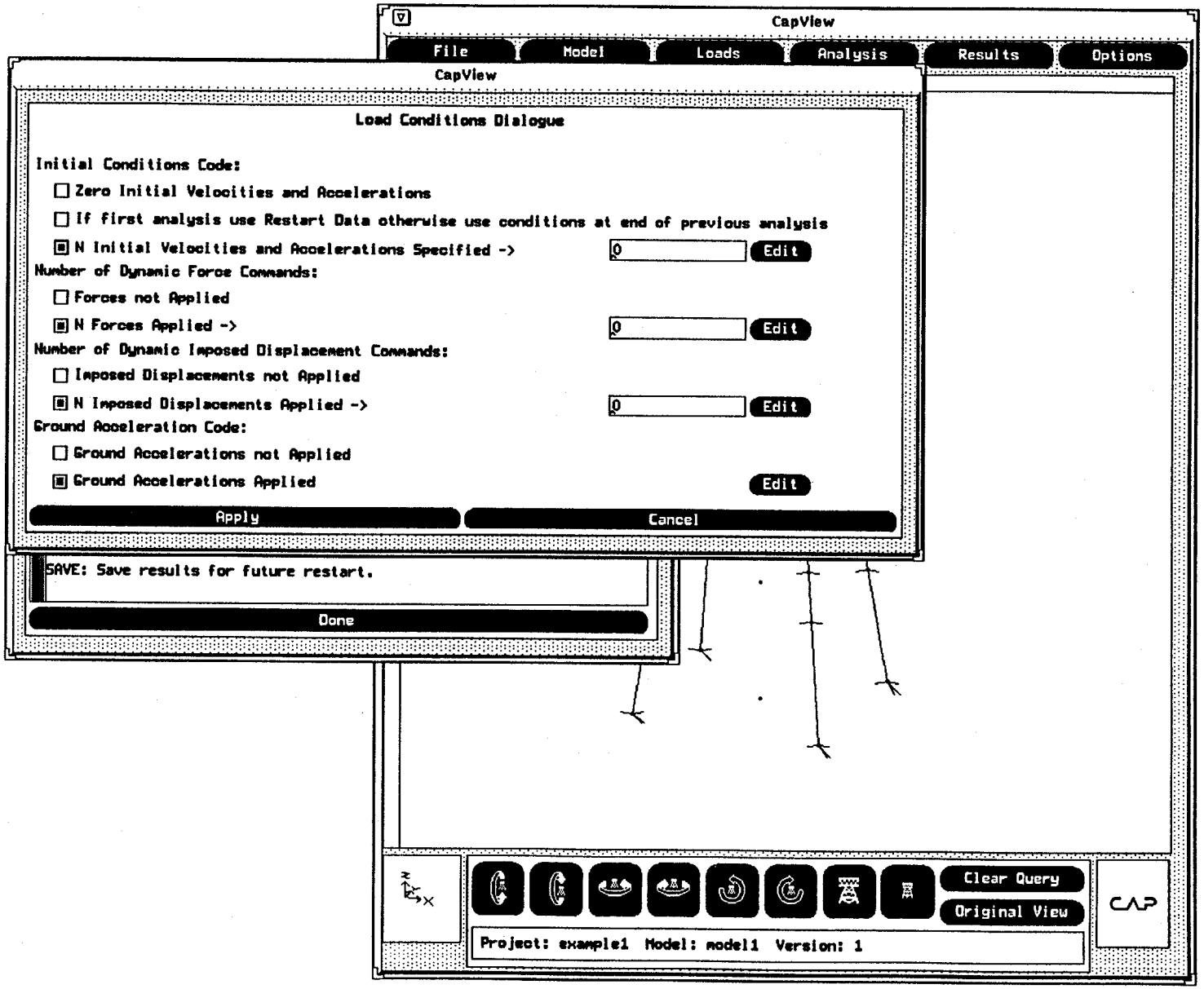

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Conditions Dialogue
Initial Conditions Code:
Zero Initial Velocities and Accelerations
If first analysis use Restart Data otherwise use conditions at end of previous analysis
N Initial Velocities and Accelerations Specified ->
0
Edit
Number of Dynamic Force Commands:
Forces not Applied
N Forces Applied ->
0
Edit
Number of Dynamic Imposed Displacement Commands:
Imposed Displacements not Applied
N Imposed Displacements Applied ->
0
Edit
Ground Acceleration Code:
Ground Accelerations not Applied
Ground Accelerations Applied
Edit
Apply
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 10-32

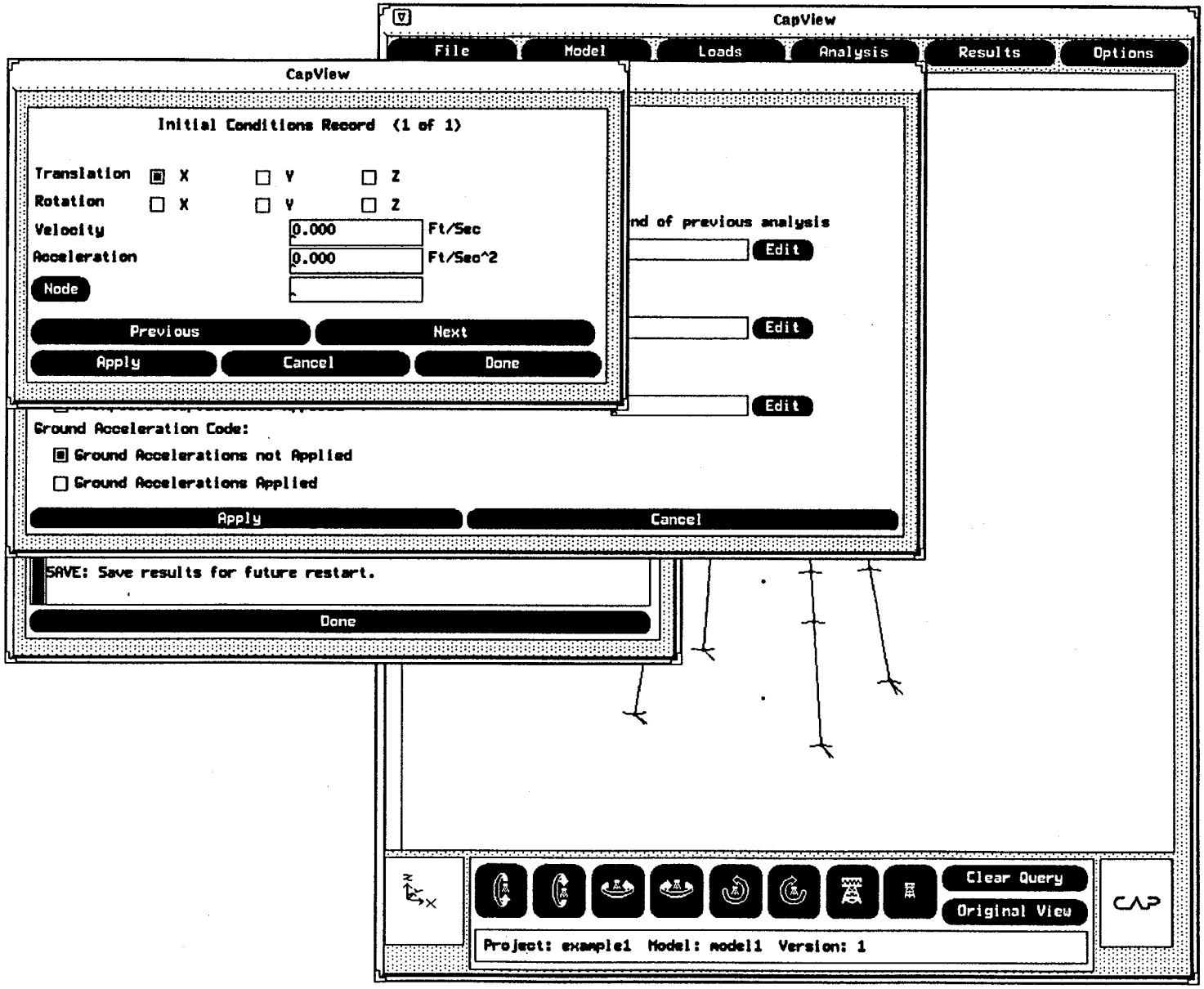

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Initial Conditions Record (1 of 1)
Translation X Y Z
Rotation X Y Z
Velocity 0.000 Ft/Sec
Acceleration 0.000 Ft/Sec^2
Node
Previous
Next
Apply
Cancel
Done
nd of previous analysis
Edit
Edit
Edit
Ground Acceleration Code:
Ground Accelerations not Applied
Ground Accelerations Applied
Apply
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-33

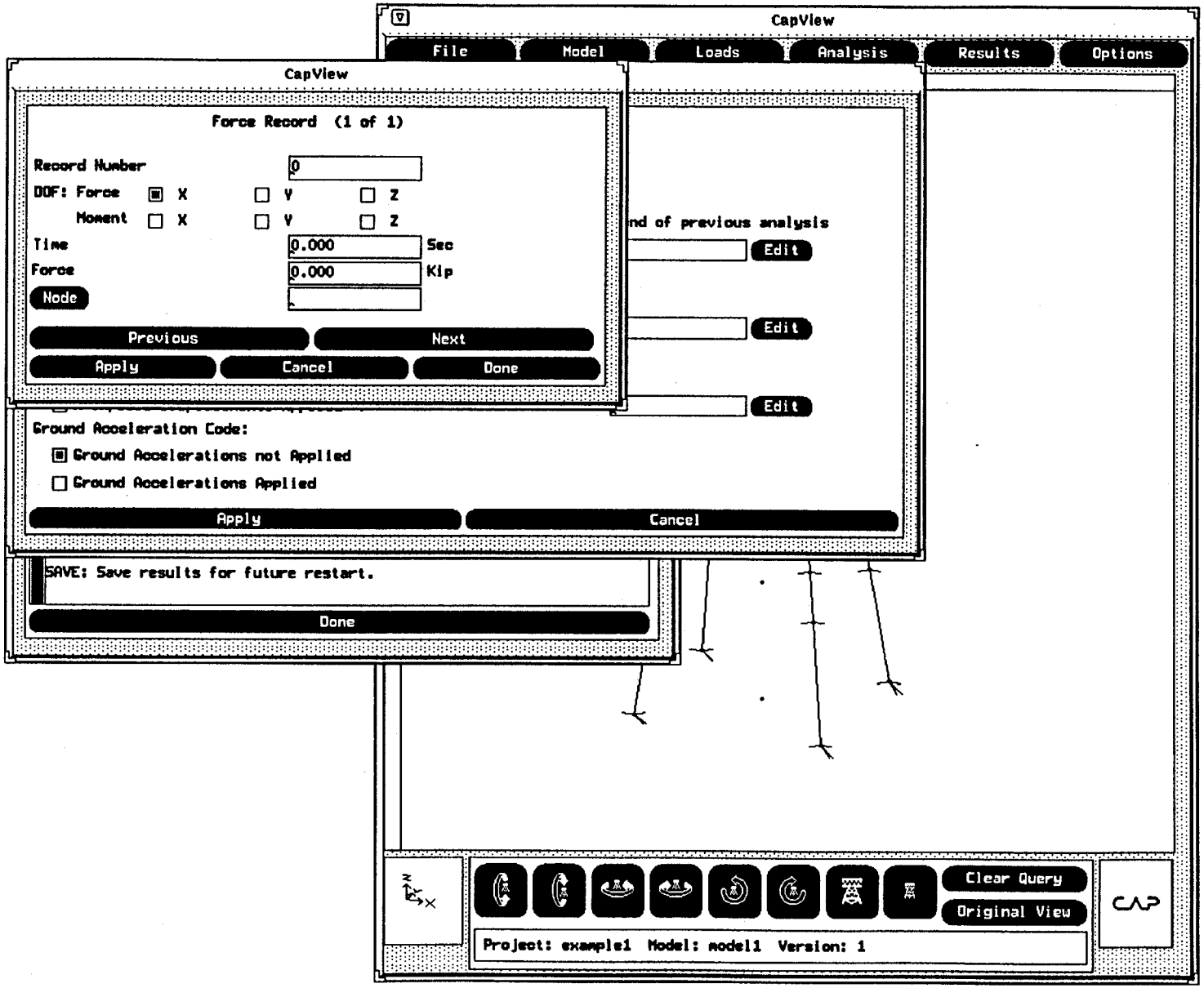

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Force Record (1 of 1)
Record Number
0
DOF: Force
X
Y
Z
Moment
X
Y
Z
Time
0.000
Sec
Force
0.000
Kip
Node
Previous
Next
Apply
Cancel
Done
nd of previous analysis
Edit
Edit
Edit
Ground Acceleration Code:
Ground Accelerations not Applied
Ground Accelerations Applied
Apply
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-34

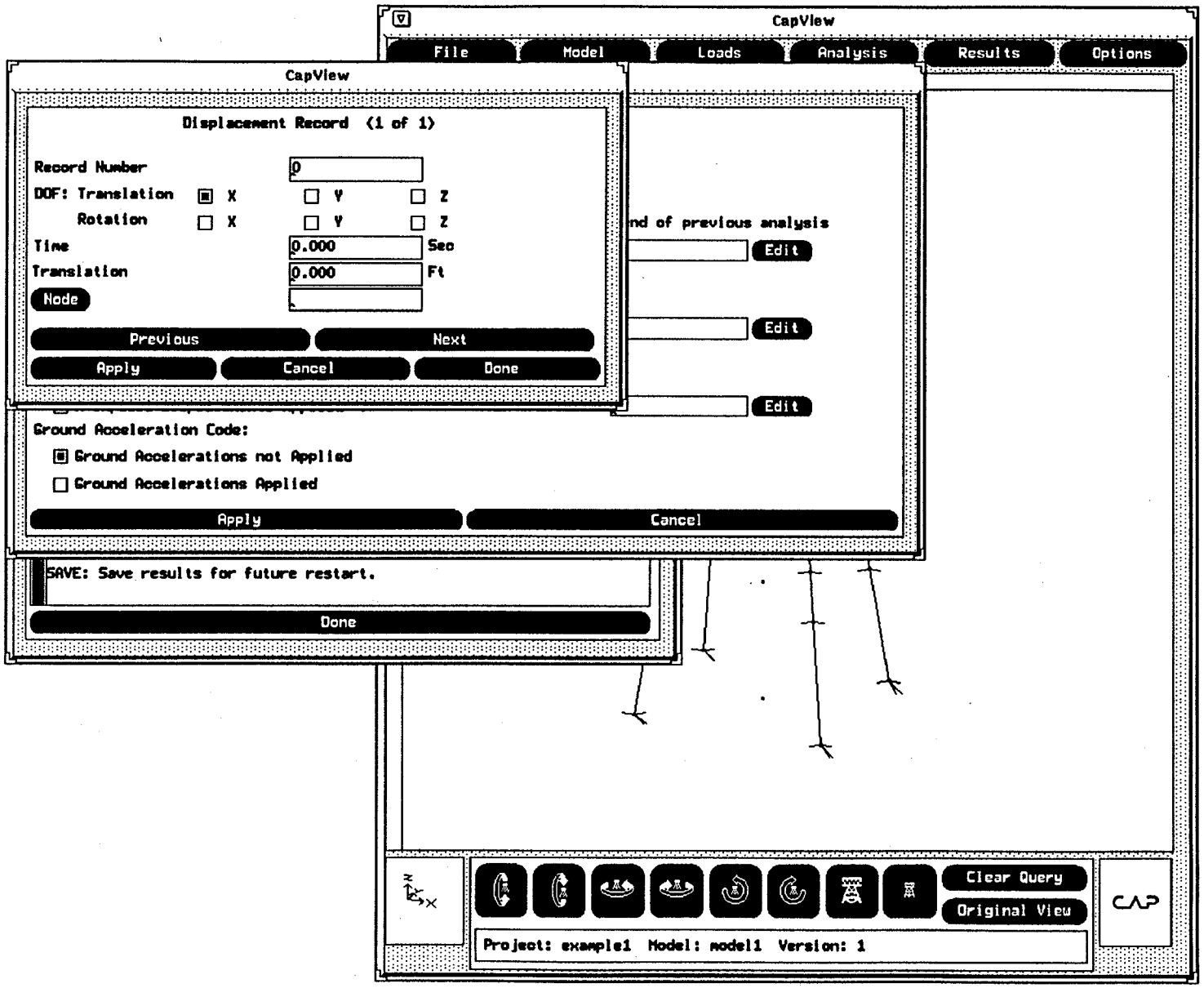

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Displacement Record (1 of 1)
Record Number
0
DOF: Translation
X
Y
Z
Rotation
X
Y
Z
Time
0.000
Sec
Translation
0.000
Ft
Node
Previous
Next
Apply
Cancel
Done
nd of previous analysis
Edit
Edit
Edit
Ground Acceleration Code:
Ground Accelerations not Applied
Ground Accelerations Applied
Apply
Cancel
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-35

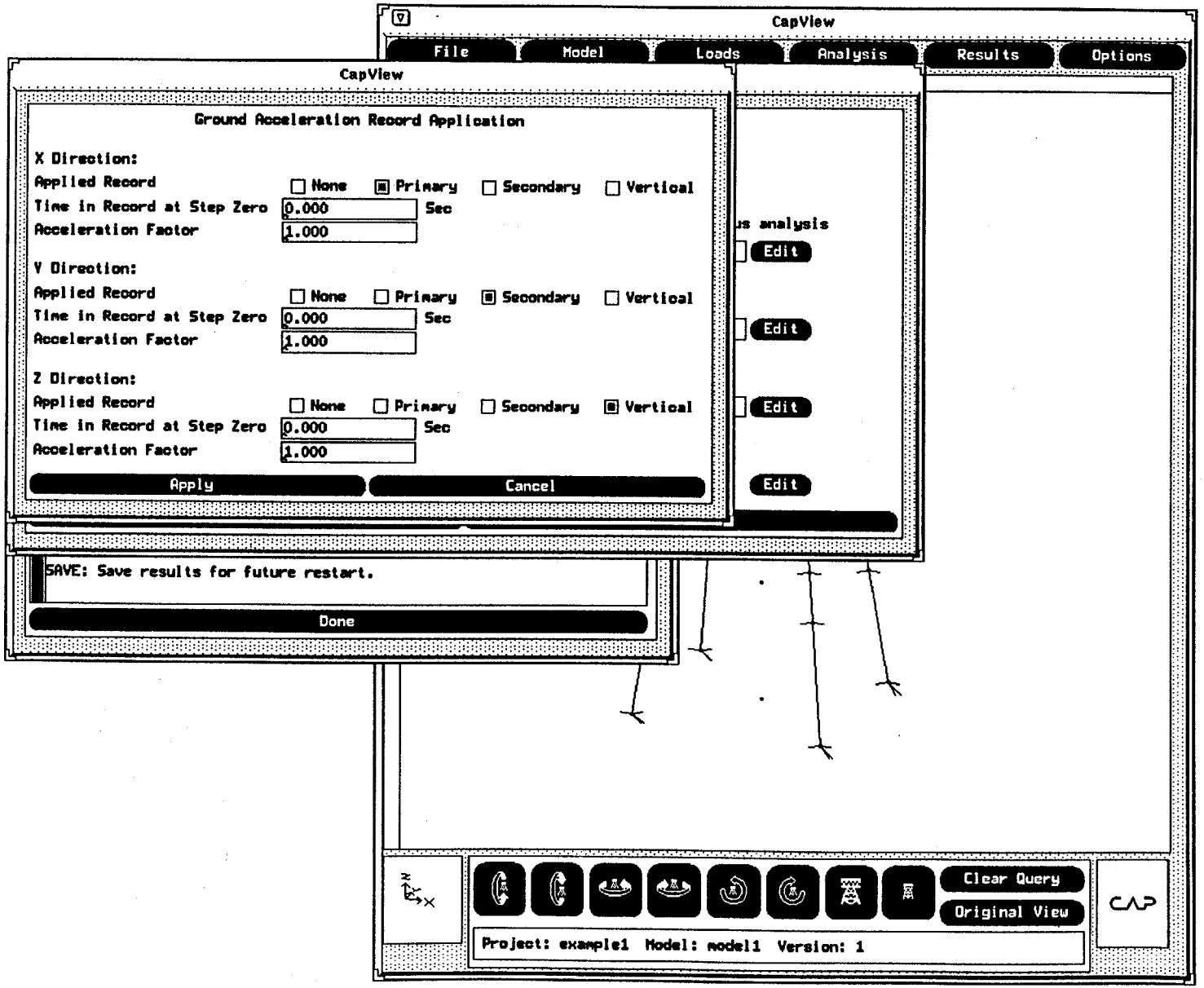

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Acceleration Record Application
X Direction:
Applied Record
None
Primary
Secondary
Vertical
Time in Record at Step Zero
0.000
Sec
Acceleration Factor
1.000
Y Direction:
Applied Record
None
Primary
Secondary
Vertical
Time in Record at Step Zero
0.000
Sec
Acceleration Factor
1.000
Z Direction:
Applied Record
None
Primary
Secondary
Vertical
Time in Record at Step Zero
0.000
Sec
Acceleration Factor
1.000
Apply
Cancel
us analysis
Edit
Edit
Edit
Edit
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-36

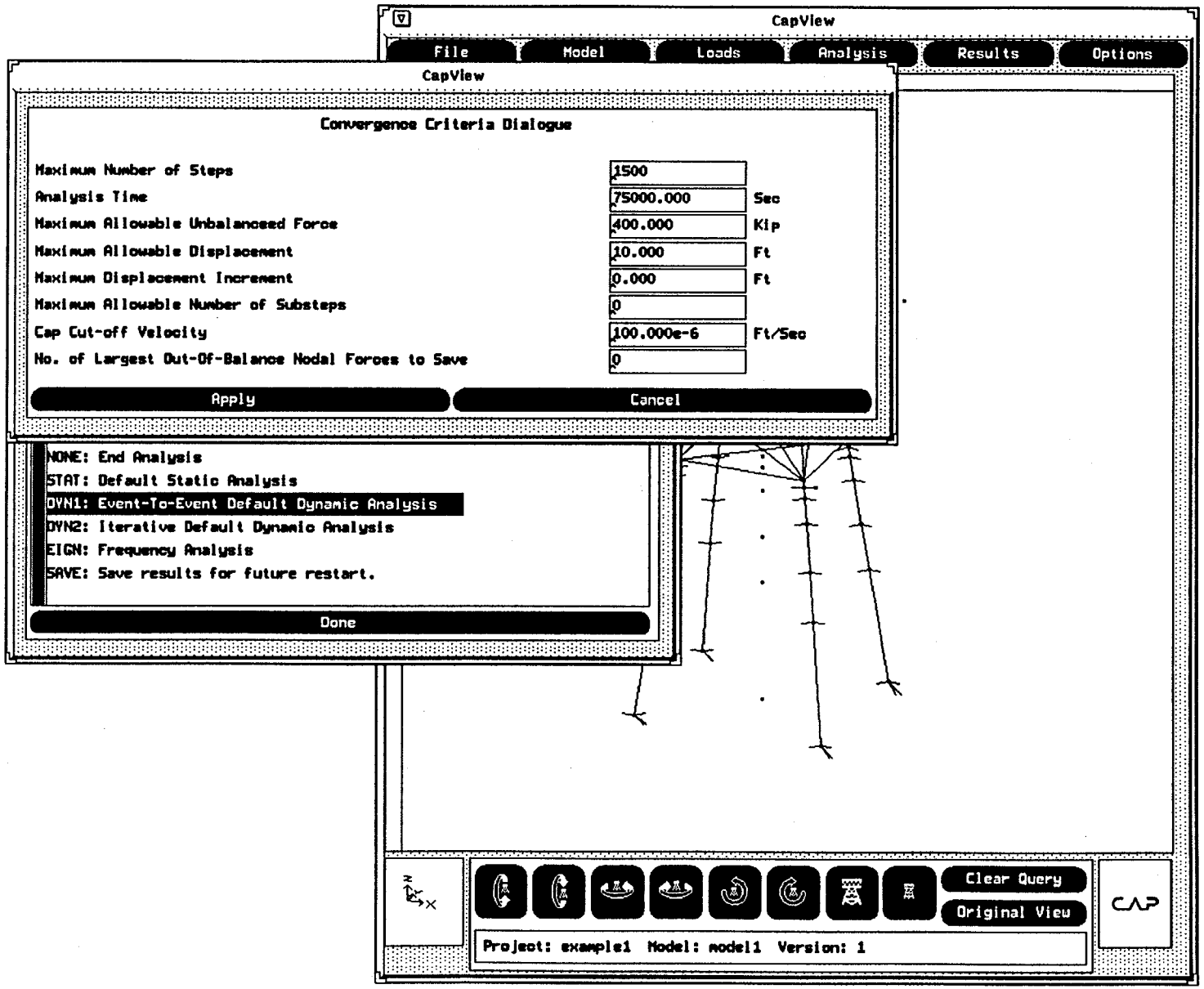

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Convergence Criteria Dialogue
Maximum Number of Steps
1500
Analysis Time
75000.000
Sec
Maximum Allowable Unbalanceed Force
400.000
Kip
Maximum Allowable Displacement
10.000
Ft
Maximum Displacement Increment
0.000
Ft
Maximum Allowable Number of Substeps
0
Cap Cut-off Velocity
100.000e-6
Ft/Sec
No. of Largest Out-Of-Balance Nodal Forces to Save
0
Apply
Cancel
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 10-37

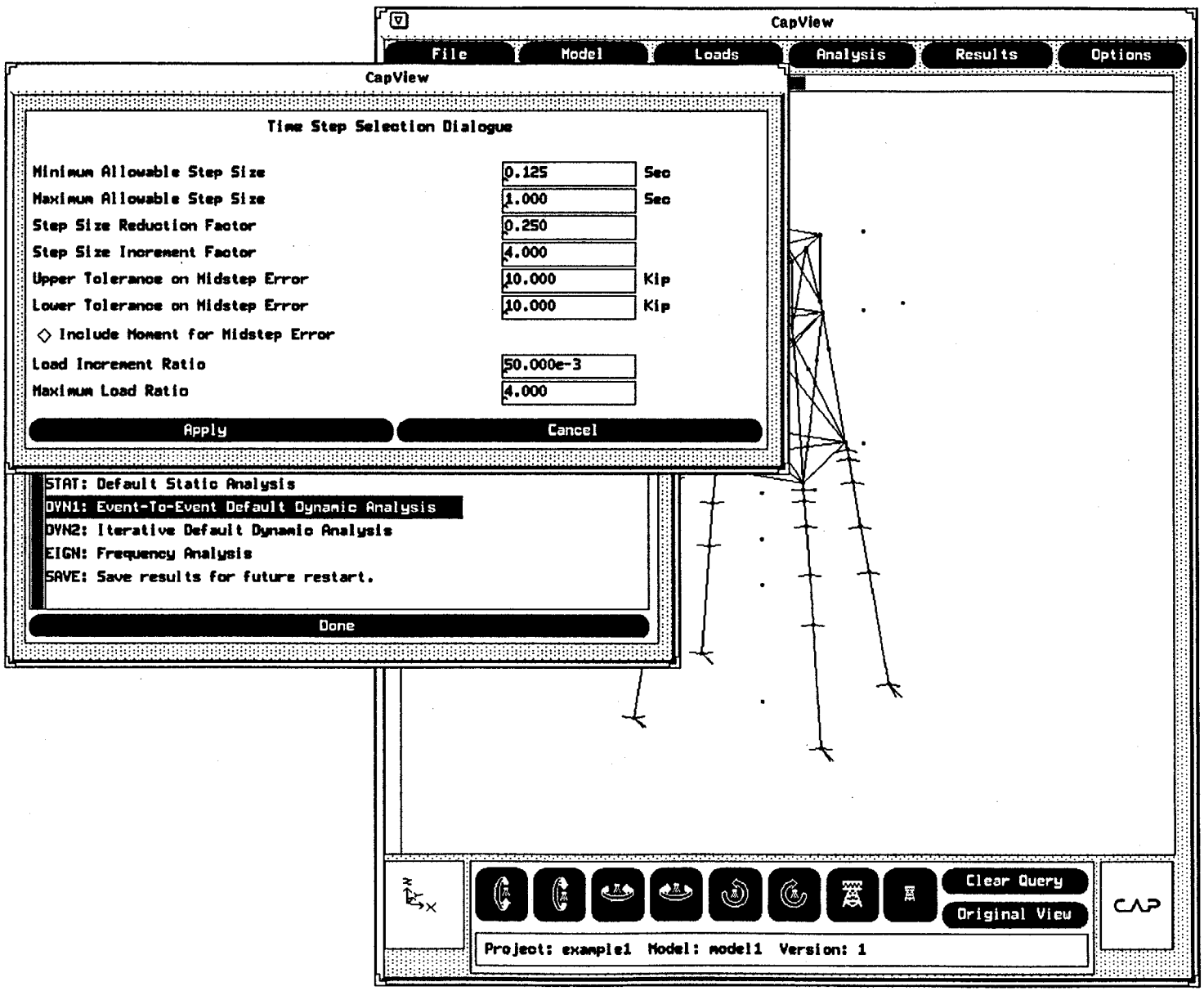

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Time Step Selection Dialogue
Minimum Allowable Step Size
0.125
Sec
Maximum Allowable Step Size
1.000
Sec
Step Size Reduction Factor
0.250
Step Size Increment Factor
4.000
Upper Tolerance on Midstep Error
10.000
Kip
Lower Tolerance on Midstep Error
10.000
Kip
◇ Include Moment for Midstep Error
Load Increment Ratio
50.000e-3
Maximum Load Ratio
4.000
Apply
Cancel
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-38

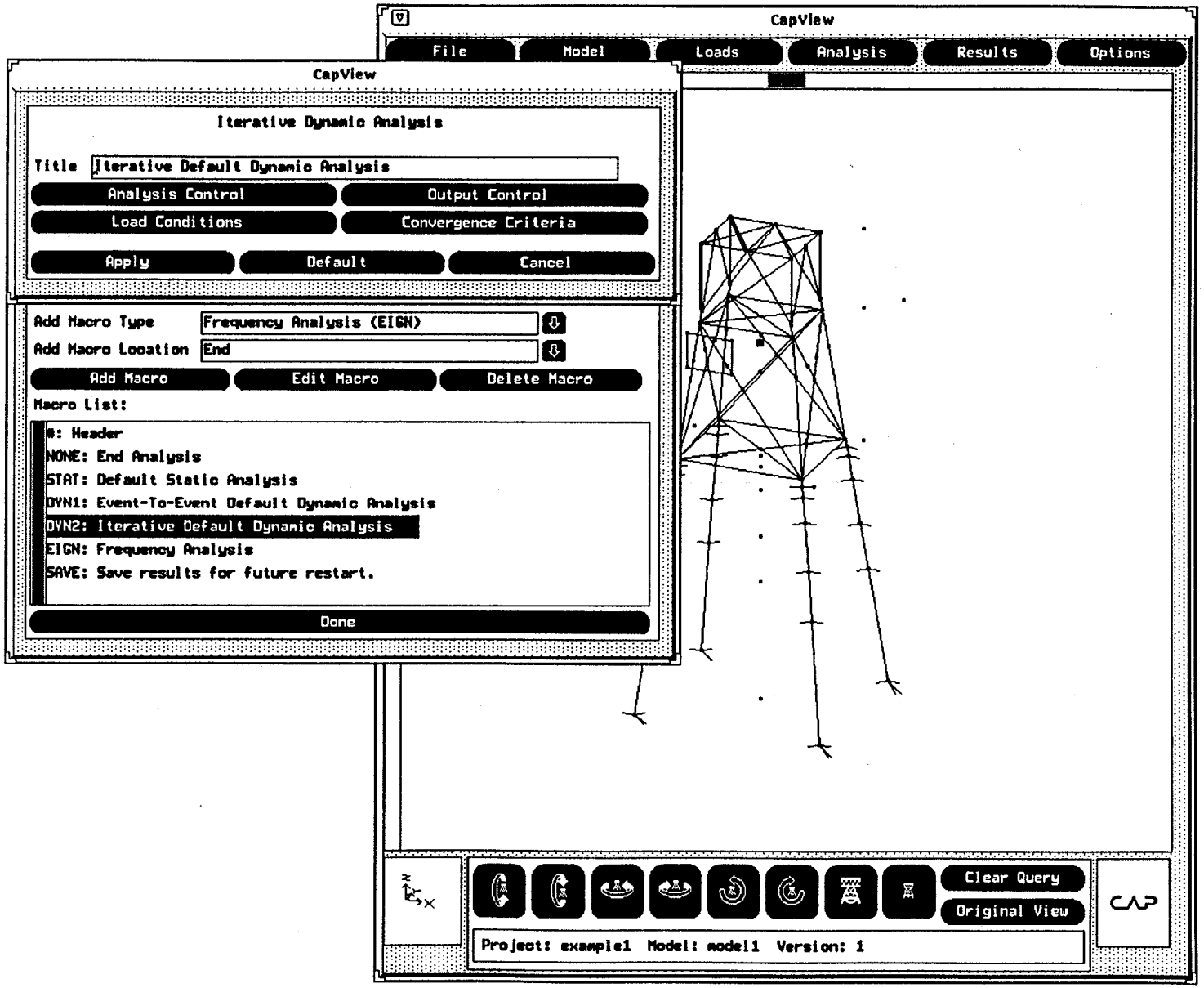

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Iterative Dynamic Analysis
Title
Iterative Default Dynamic Analysis
Analysis Control
Output Control
Load Conditions
Convergence Criteria
Apply
Default
Cancel
Add Macro Type
Frequency Analysis (EIGN)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-39

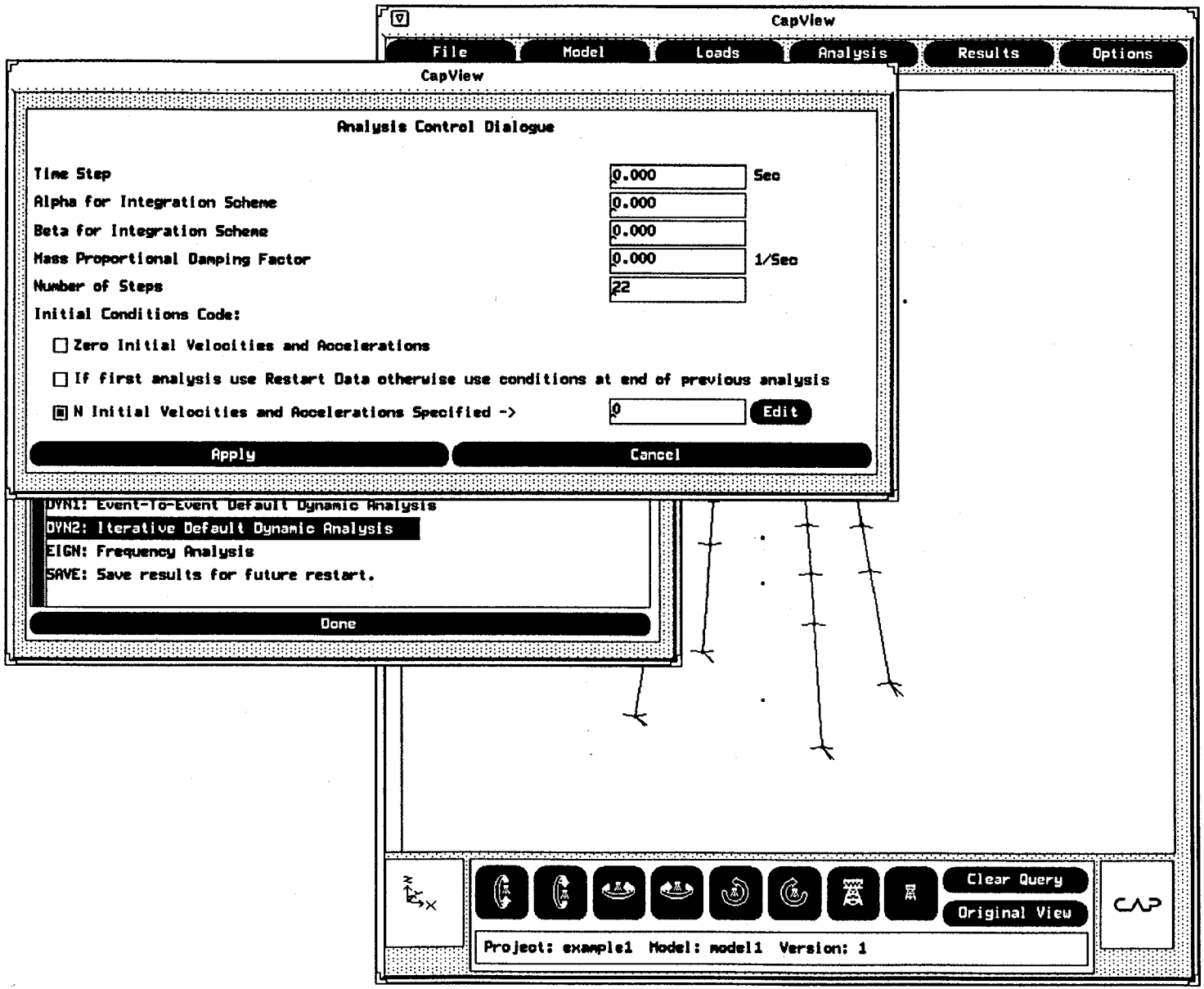

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Analysis Control Dialogue
Time Step
0.000
Sec
Alpha for Integration Scheme
0.000
Beta for Integration Scheme
0.000
Mass Proportional Damping Factor
0.000
1/Sec
Number of Steps
22
Initial Conditions Code:
Zero Initial Velocities and Accelerations
If first analysis use Restart Data otherwise use conditions at end of previous analysis
N Initial Velocities and Accelerations Specified ->
0
Edit
Apply
Cancel
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-40

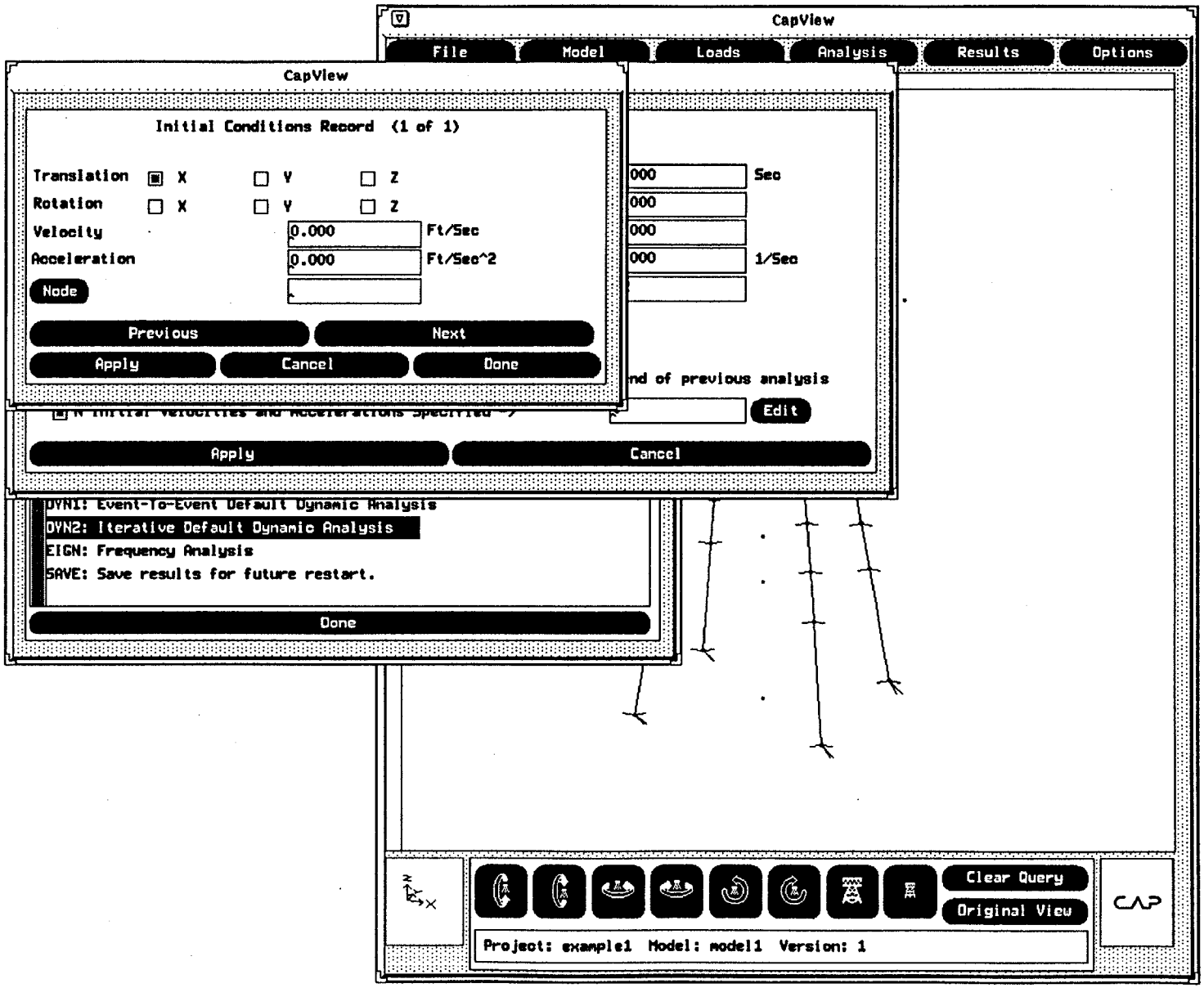

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Initial Conditions Record (1 of 1)
Translation X Y Z
Rotation X Y Z
Velocity 0.000 Ft/Sec
Acceleration 0.000 Ft/Sec^2
Node
Previous
Next
Apply
Cancel
Done
000 Sec
000
000
000 1/Sec
nd of previous analysis
Edit
Apply
Cancel
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-41

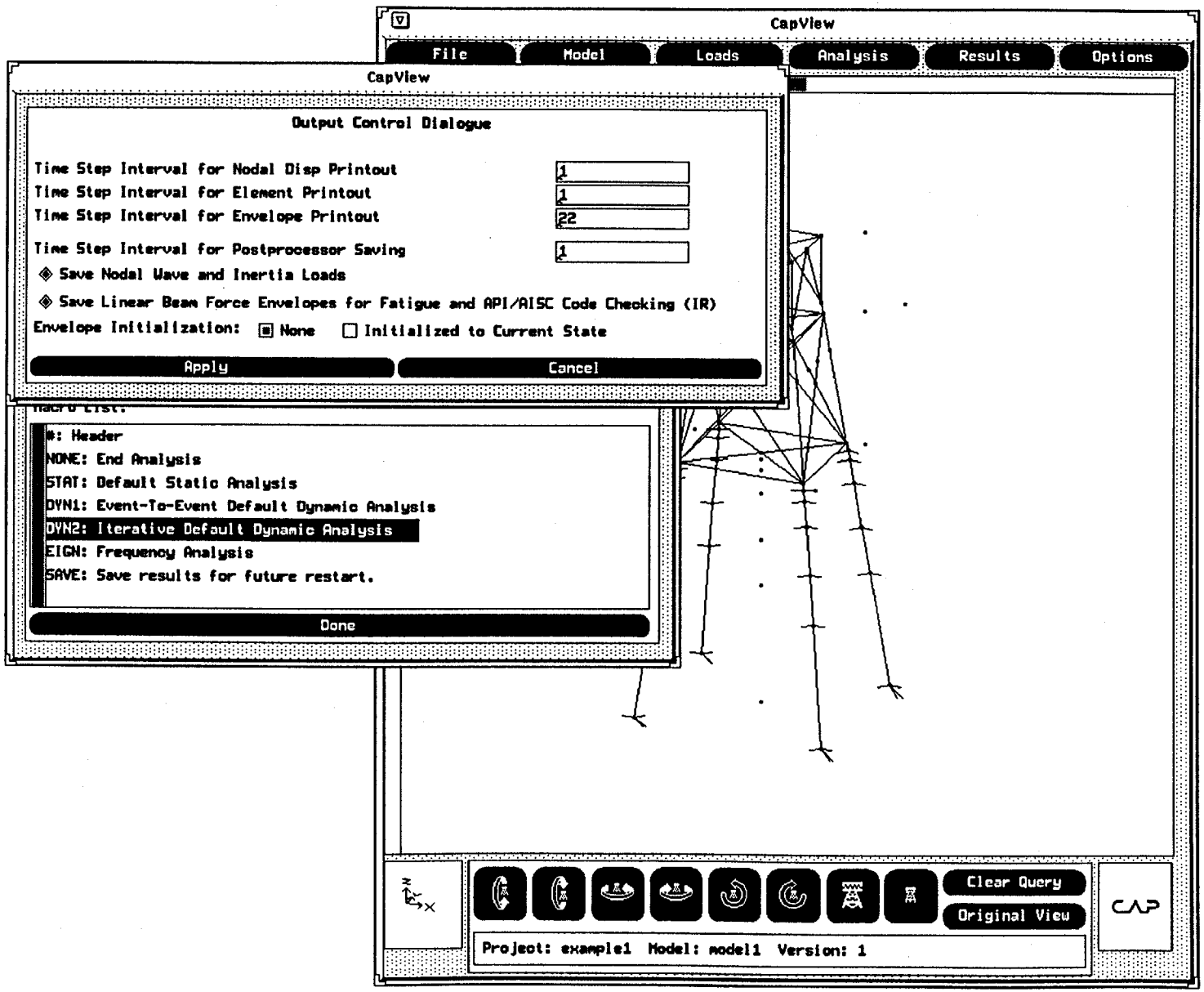

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Output Control Dialogue
Time Step Interval for Nodal Disp Printout
1
Time Step Interval for Element Printout
1
Time Step Interval for Envelope Printout
22
Time Step Interval for Postprocessor Saving
1
Save Nodal Wave and Inertia Loads
Save Linear Beam Force Envelopes for Fatigue and API/AISC Code Checking (IR)
Envelope Initialization:
None
Initialized to Current State
Apply
Cancel
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-42

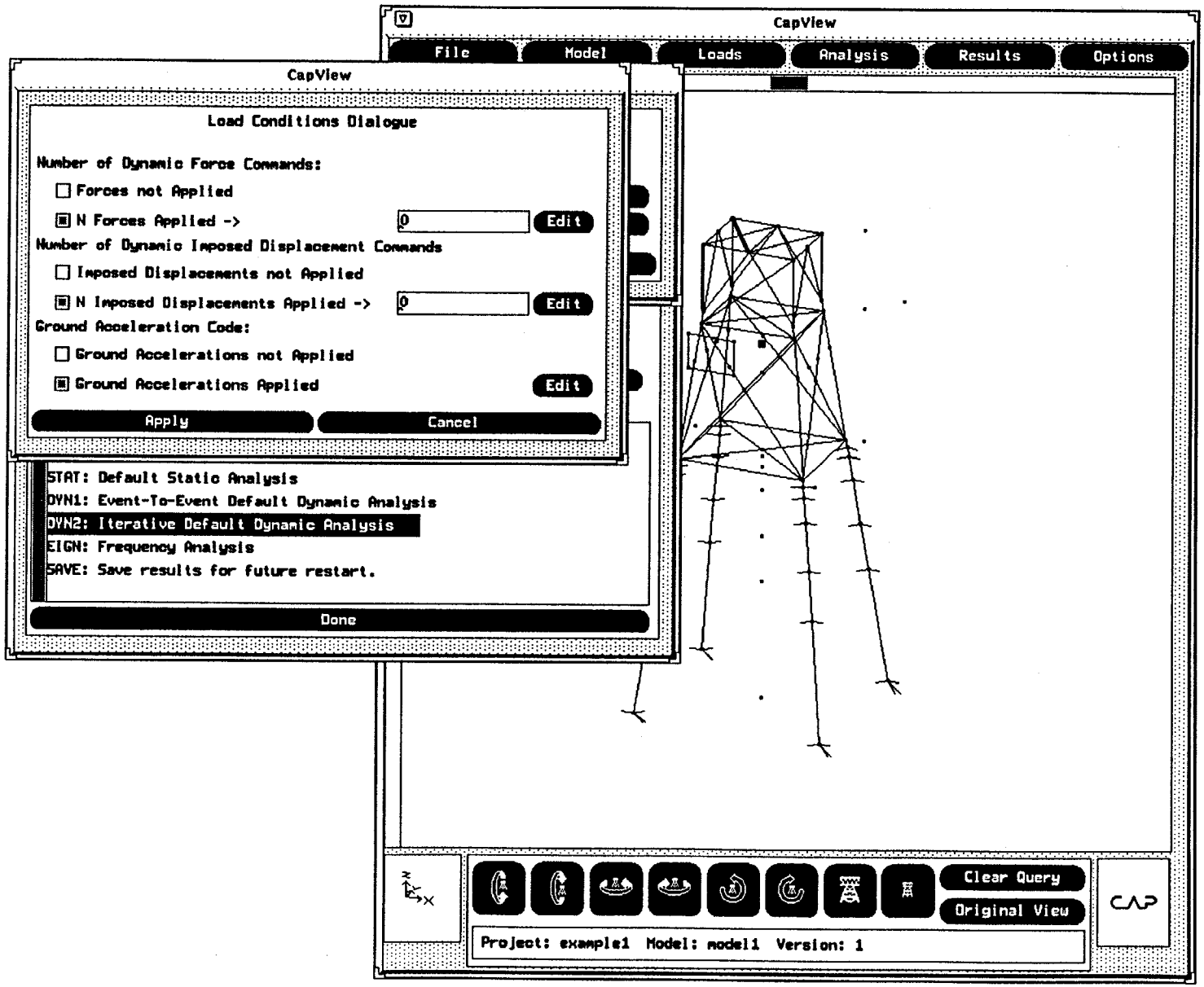

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Load Conditions Dialogue
Number of Dynamic Force Commands:
Forces not Applied
N Forces Applied ->
0
Edit
Number of Dynamic Imposed Displacement Commands
Imposed Displacements not Applied
N Imposed Displacements Applied ->
0
Edit
Ground Acceleration Code:
Ground Accelerations not Applied
Ground Accelerations Applied
Edit
Apply
Cancel
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-43

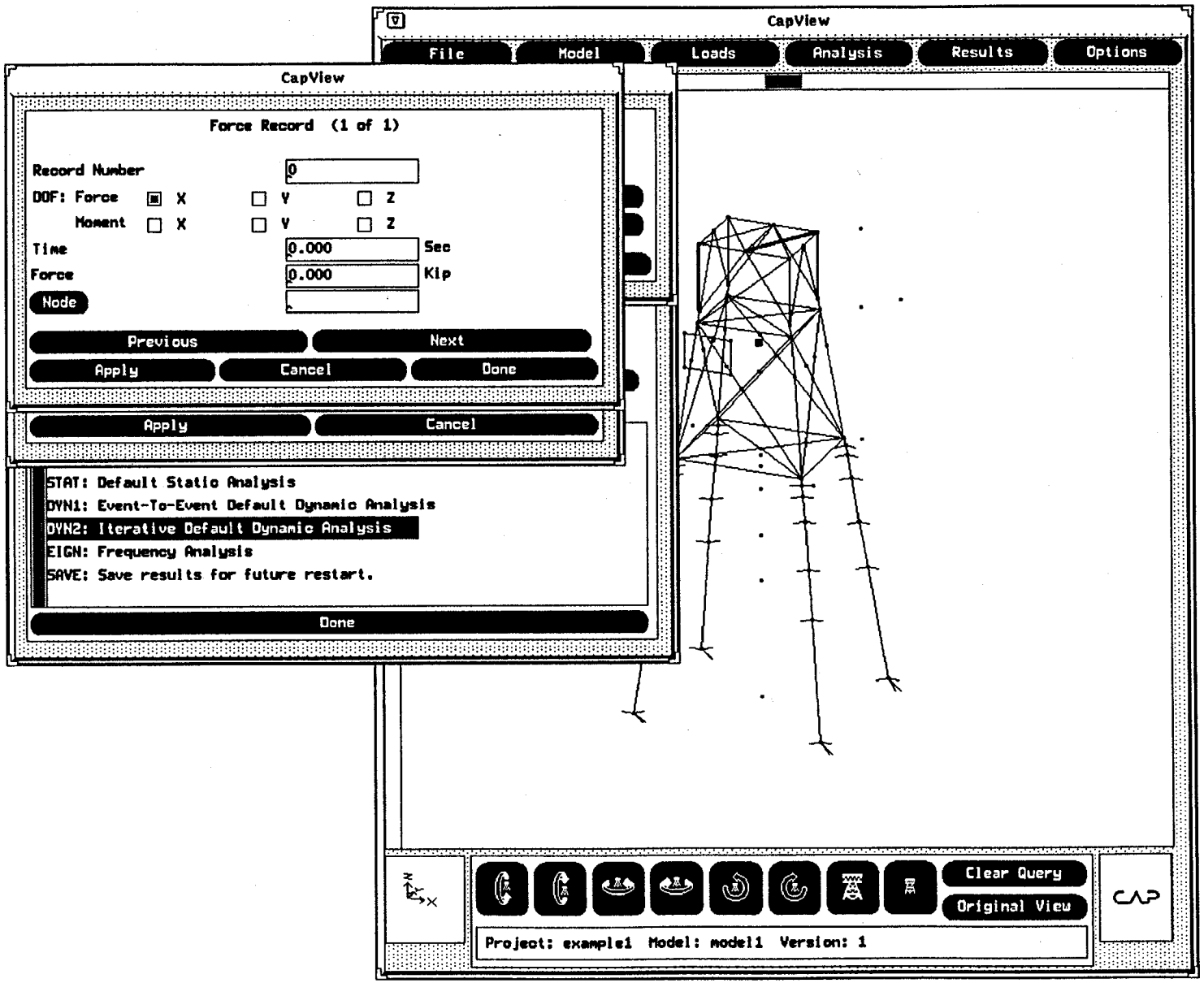

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Force Record (1 of 1)
Record Number
0
DOF: Force
X
Y
Z
Moment
X
Y
Z
Time
0.000
Sec
Force
0.000
Kip
Node
Previous
Next
Apply
Cancel
Done
Apply
Cancel
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-44

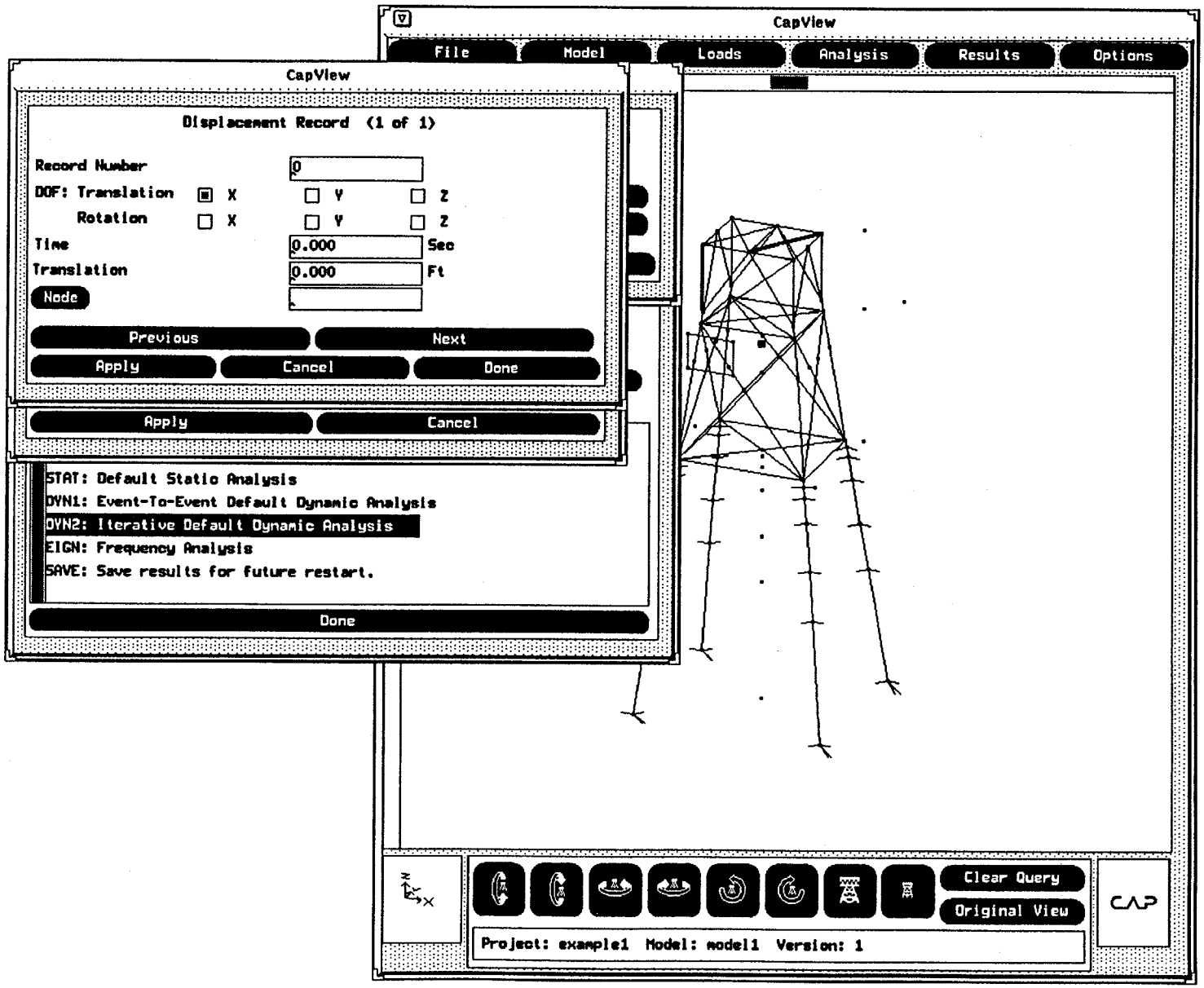

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Displacement Record (1 of 1)
Record Number
0
DOF: Translation
X
Y
Z
Rotation
X
Y
Z
Time
0.000
Sec
Translation
0.000
Ft
Node
Previous
Next
Apply
Cancel
Done
Apply
Cancel
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 10-45

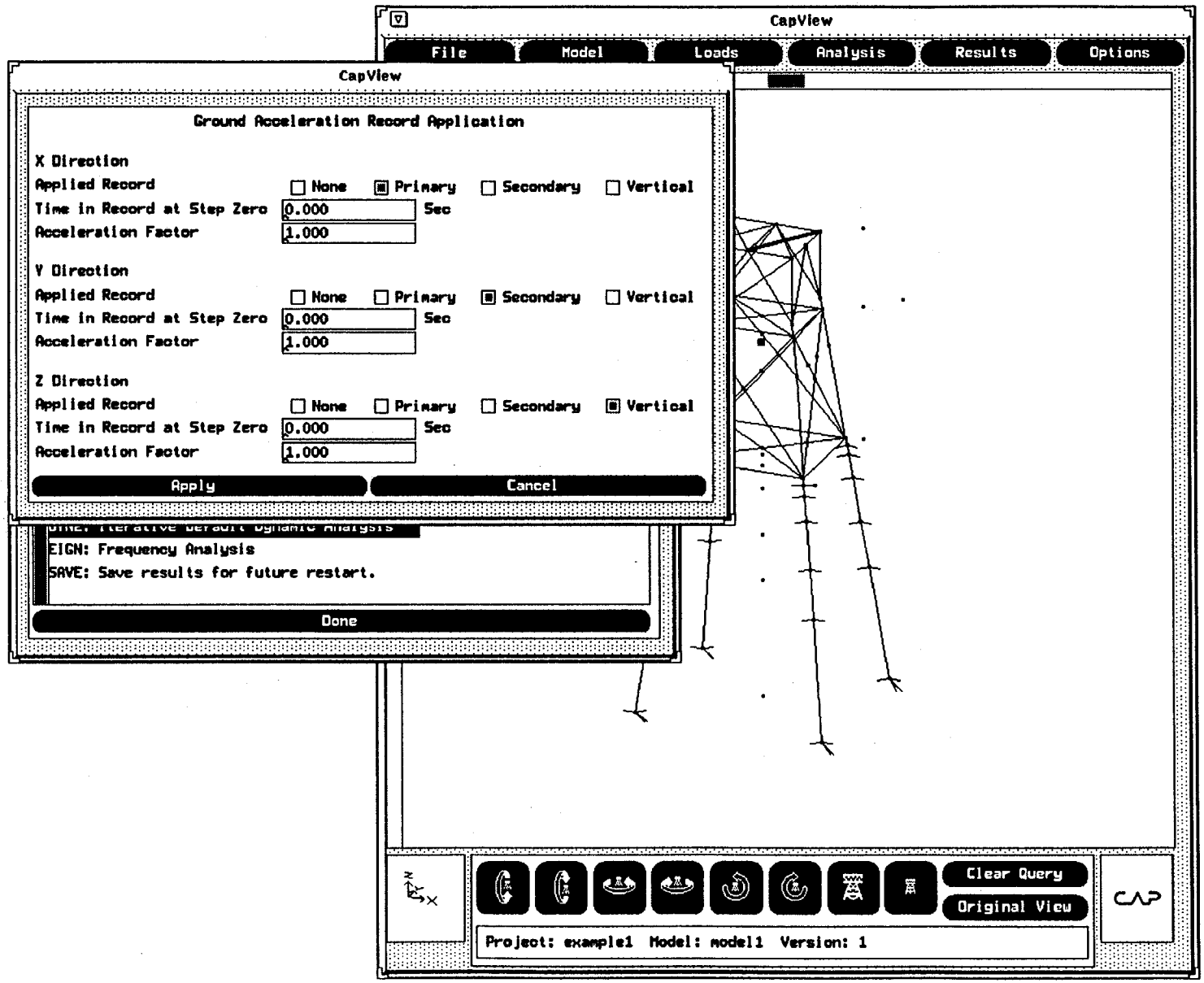

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Ground Acceleration Record Application
X Direction
Applied Record
None
Primary
Secondary
Vertical
Time in Record at Step Zero
0.000
Sec
Acceleration Factor
1.000
Y Direction
Applied Record
None
Primary
Secondary
Vertical
Time in Record at Step Zero
0.000
Sec
Acceleration Factor
1.000
Z Direction
Applied Record
None
Primary
Secondary
Vertical
Time in Record at Step Zero
0.000
Sec
Acceleration Factor
1.000
Apply
Cancel
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-46

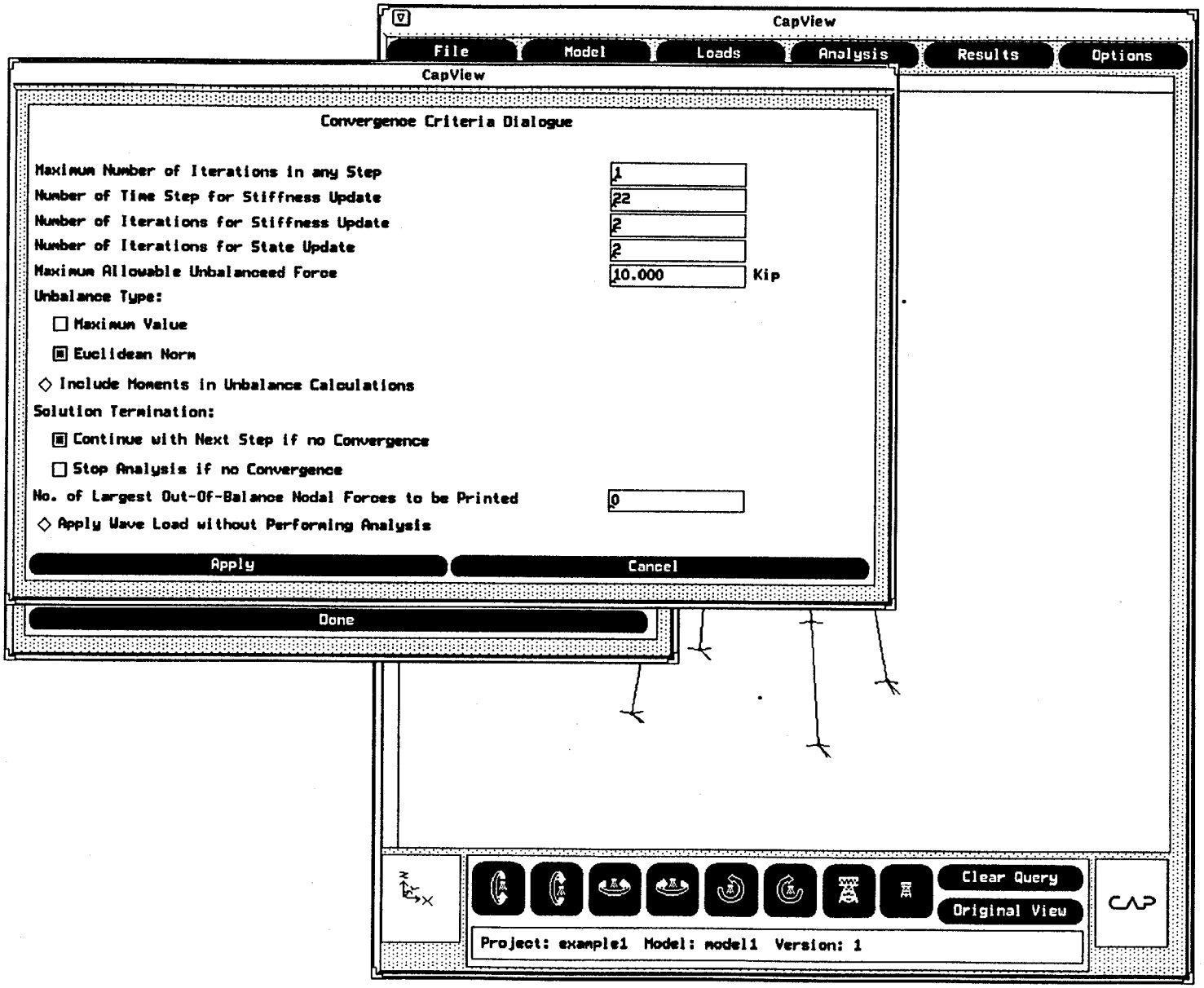

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Convergence Criteria Dialogue
Maximum Number of Iterations in any Step
1
Number of Time Step for Stiffness Update
22
Number of Iterations for Stiffness Update
2
Number of Iterations for State Update
2
Maximum Allowable Unbalanceed Force
10.000
Kip
Unbalance Type:
Maximum Value
Euclidean Norm
Include Moments in Unbalance Calculations
Solution Termination:
Continue with Next Step if no Convergence
Stop Analysis if no Convergence
No. of Largest Out-Of-Balance Nodal Forces to be Printed
0
Apply Wave Load without Performing Analysis
Apply
Cancel
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-47

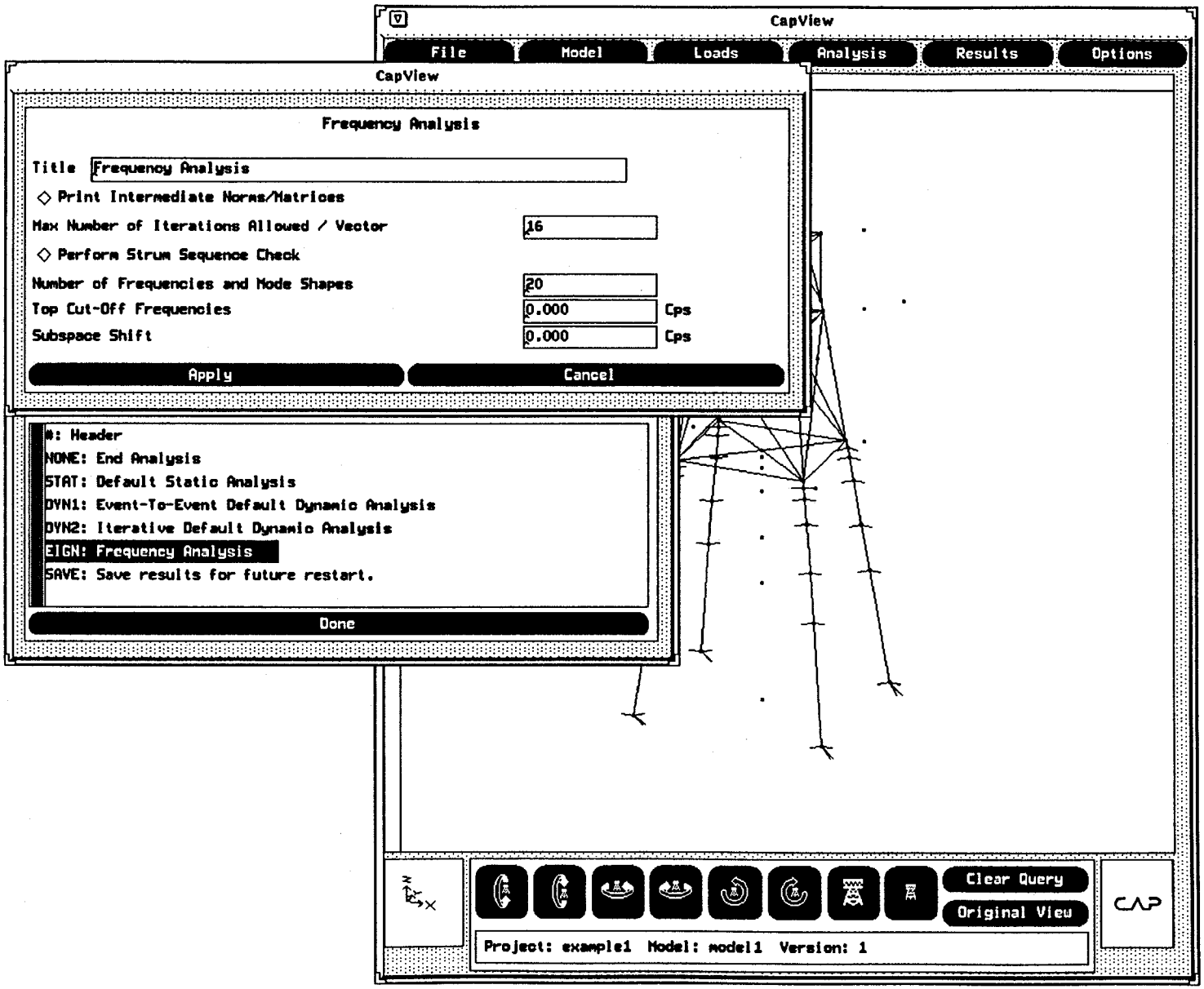

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Frequency Analysis
Title
Frequency Analysis
◇ Print Intermediate Norms/Matrices
Max Number of Iterations Allowed / Vector
16
◇ Perform Strum Sequence Check
Number of Frequencies and Mode Shapes
20
Top Cut-Off Frequencies
0.000
Cps
Subspace Shift
0.000
Cps
Apply
Cancel
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-48

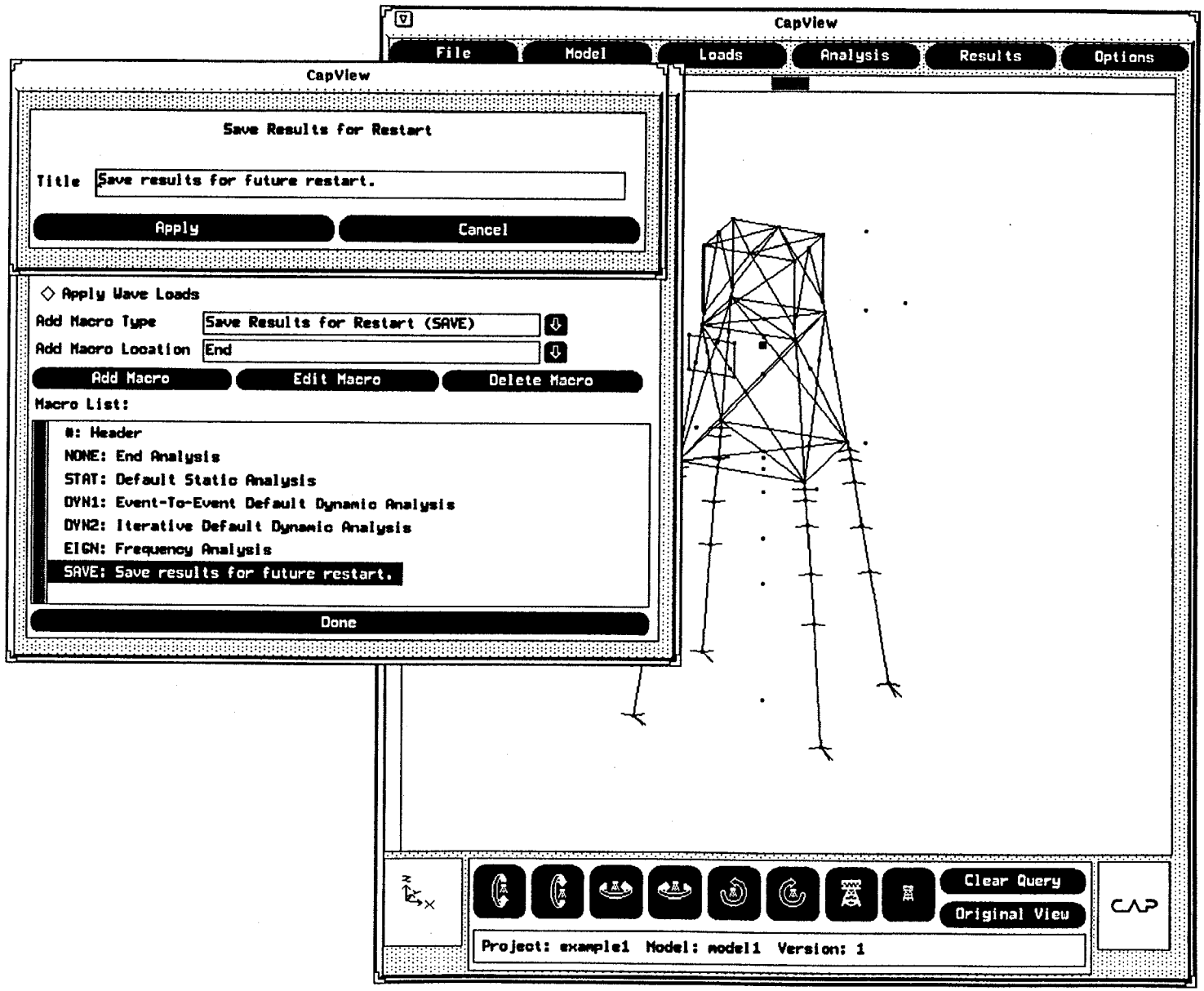

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Save Results for Restart
Title
Save results for future restart.
Apply
Cancel
Apply Wave Loads
Add Macro Type
Save Results for Restart (SAVE)
Add Macro Location
End
Add Macro
Edit Macro
Delete Macro
Macro List:
#: Header
NONE: End Analysis
STAT: Default Static Analysis
DYN1: Event-To-Event Default Dynamic Analysis
DYN2: Iterative Default Dynamic Analysis
EIGN: Frequency Analysis
SAVE: Save results for future restart.
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1
CAP

Figure 10-49

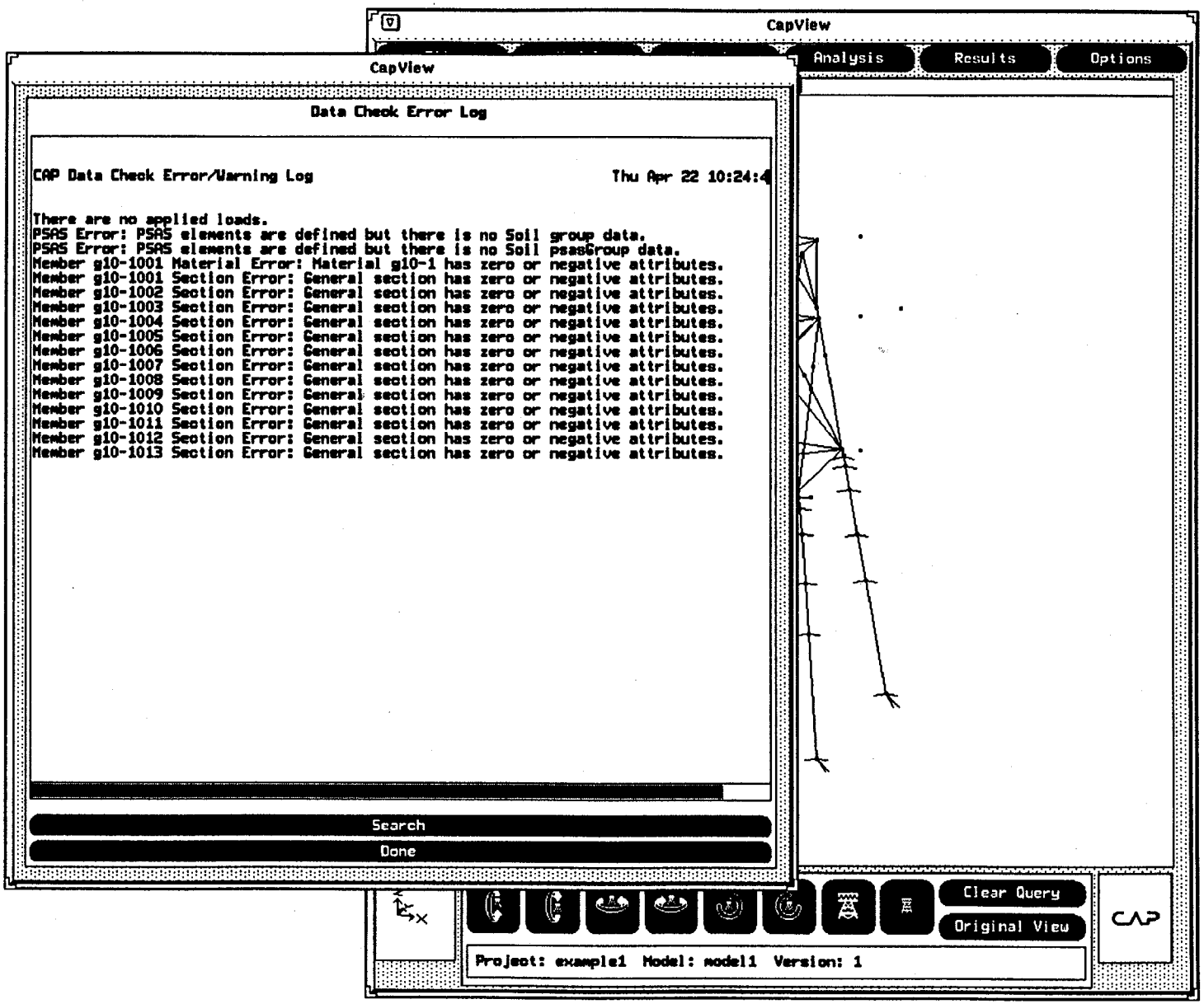

CapView
Analysis
Results
Options
CapView
Data Check Error Log
CAP Data Check Error/Warning Log
Thu Apr 22 10:24:4
There are no applied loads.
PSAS Error: PSAS elements are defined but there is no Soil group data.
PSAS Error: PSAS elements are defined but there is no Soil psasGroup data.
Member g10-1001 Material Error: Material g10-1 has zero or negative attributes.
Member g10-1001 Section Error: General section has zero or negative attributes.
Member g10-1002 Section Error: General section has zero or negative attributes.
Member g10-1003 Section Error: General section has zero or negative attributes.
Member g10-1004 Section Error: General section has zero or negative attributes.
Member g10-1005 Section Error: General section has zero or negative attributes.
Member g10-1006 Section Error: General section has zero or negative attributes.
Member g10-1007 Section Error: General section has zero or negative attributes.
Member g10-1008 Section Error: General section has zero or negative attributes.
Member g10-1009 Section Error: General section has zero or negative attributes.
Member g10-1010 Section Error: General section has zero or negative attributes.
Member g10-1011 Section Error: General section has zero or negative attributes.
Member g10-1012 Section Error: General section has zero or negative attributes.
Member g10-1013 Section Error: General section has zero or negative attributes.
Search
Done
Clear Query
Original View
CAP
Project: example1 Model: model1 Version: 1

Figure 10-50

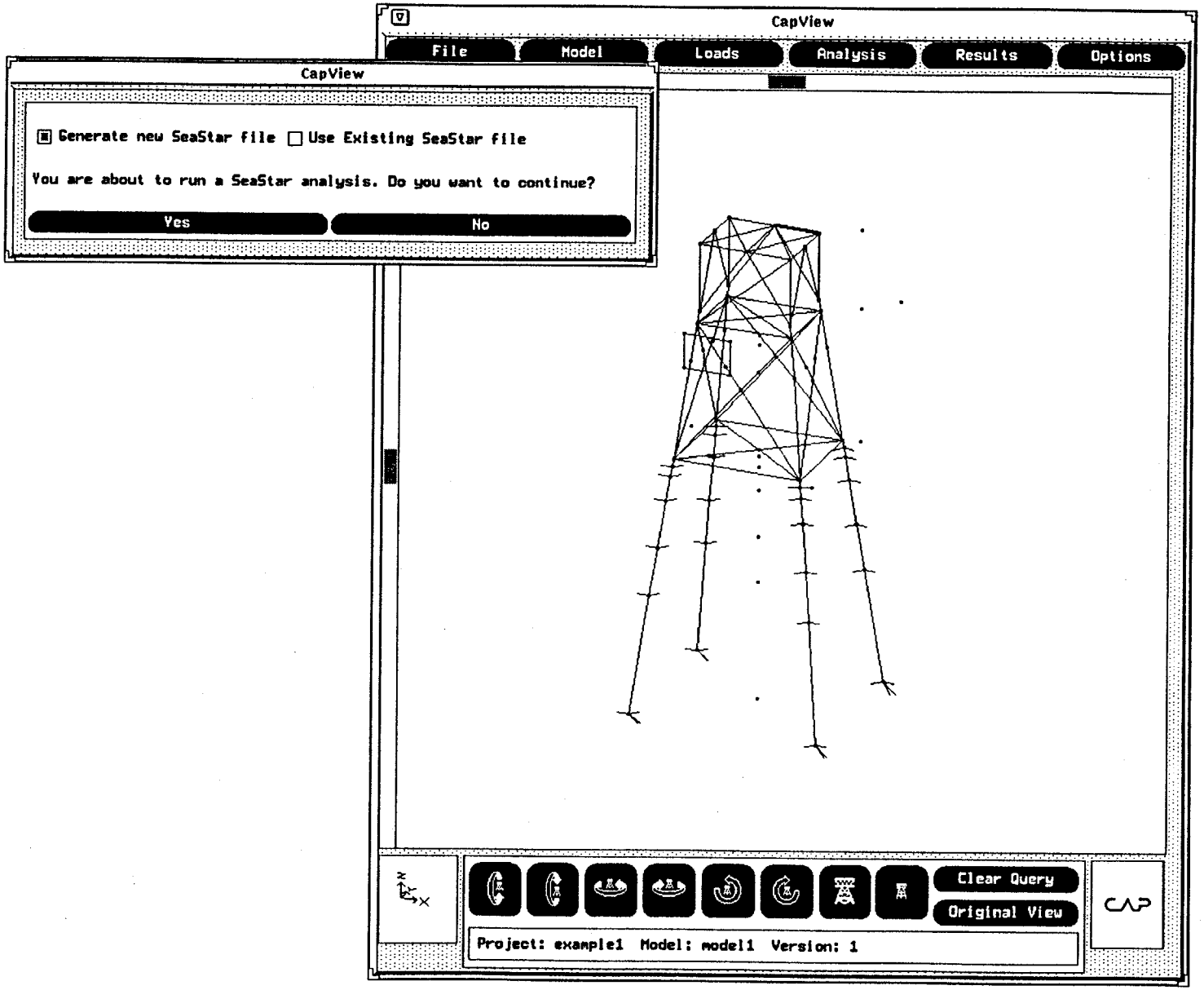

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Generate new SeaStar file
Use Existing SeaStar file
You are about to run a SeaStar analysis. Do you want to continue?
Yes
No
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-51

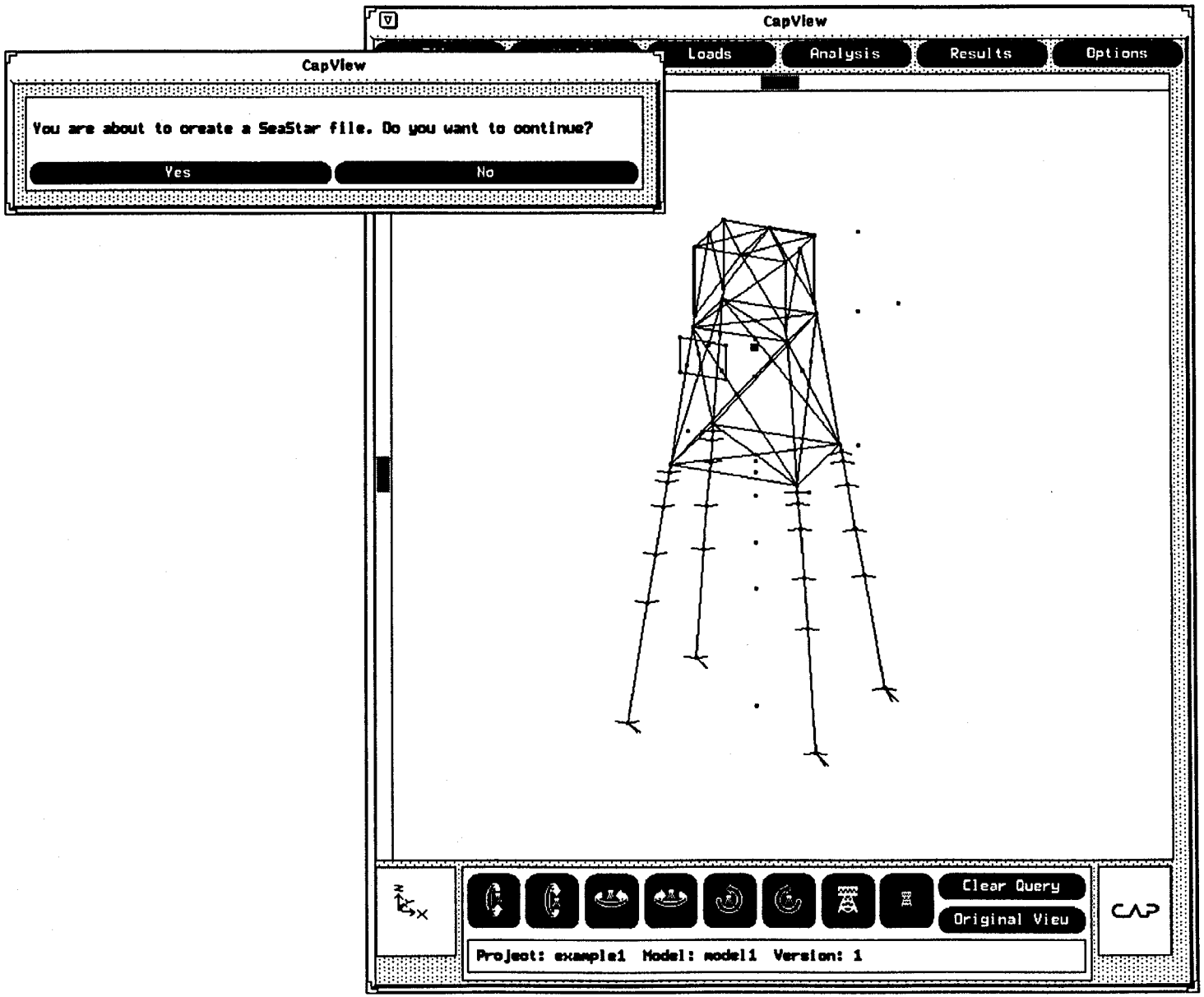

CapView
Loads
Analysis
Results
Options
CapView
You are about to create a SeaStar file. Do you want to continue?
Yes
No
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-52

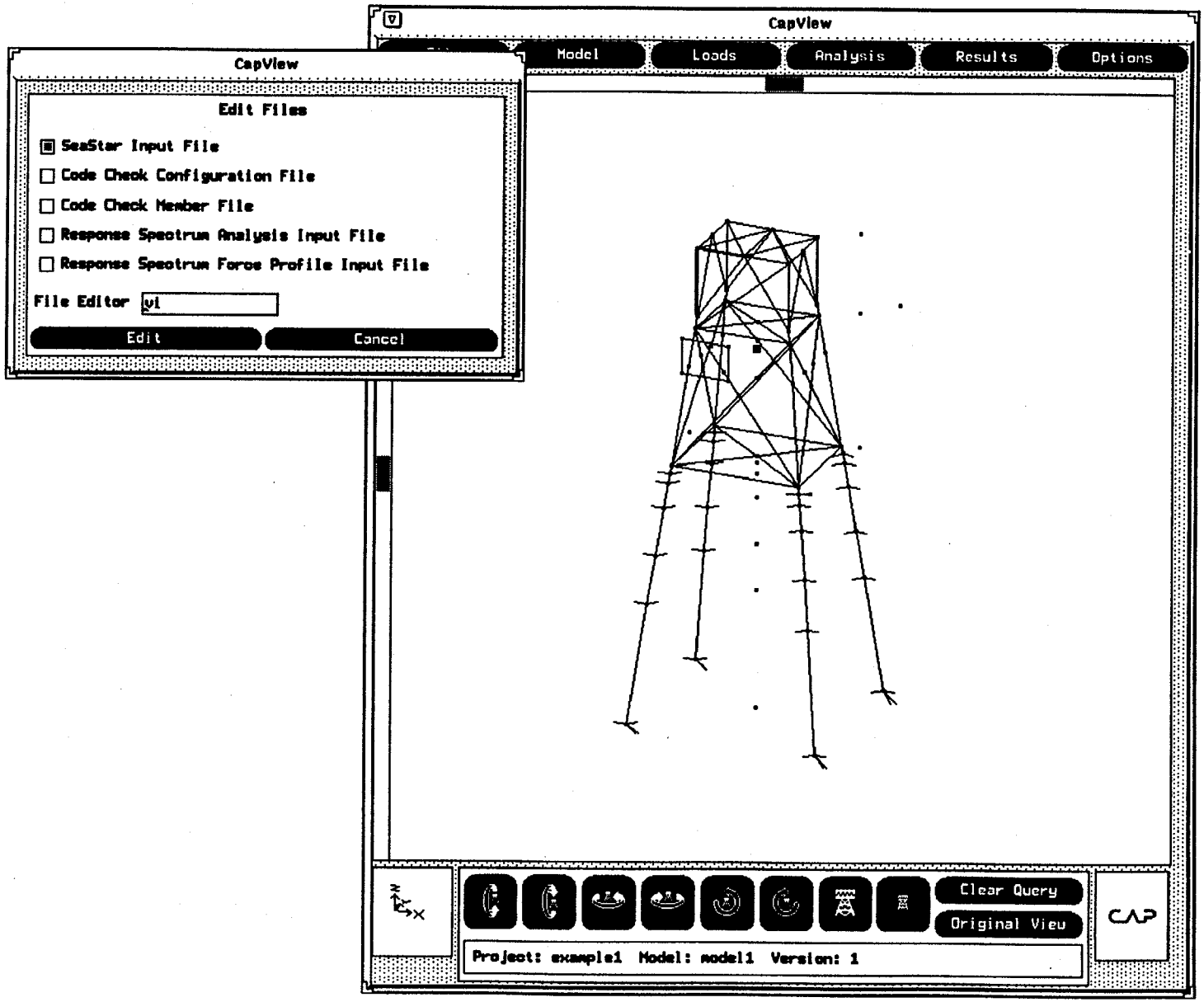

CapView
Model
Loads
Analysis
Results
Options
CapView
Edit Files
SeaStar Input File
Code Check Configuration File
Code Check Member File
Response Spectrum Analysis Input File
Response Spectrum Force Profile Input File
File Editor
vi
Edit
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 10-53

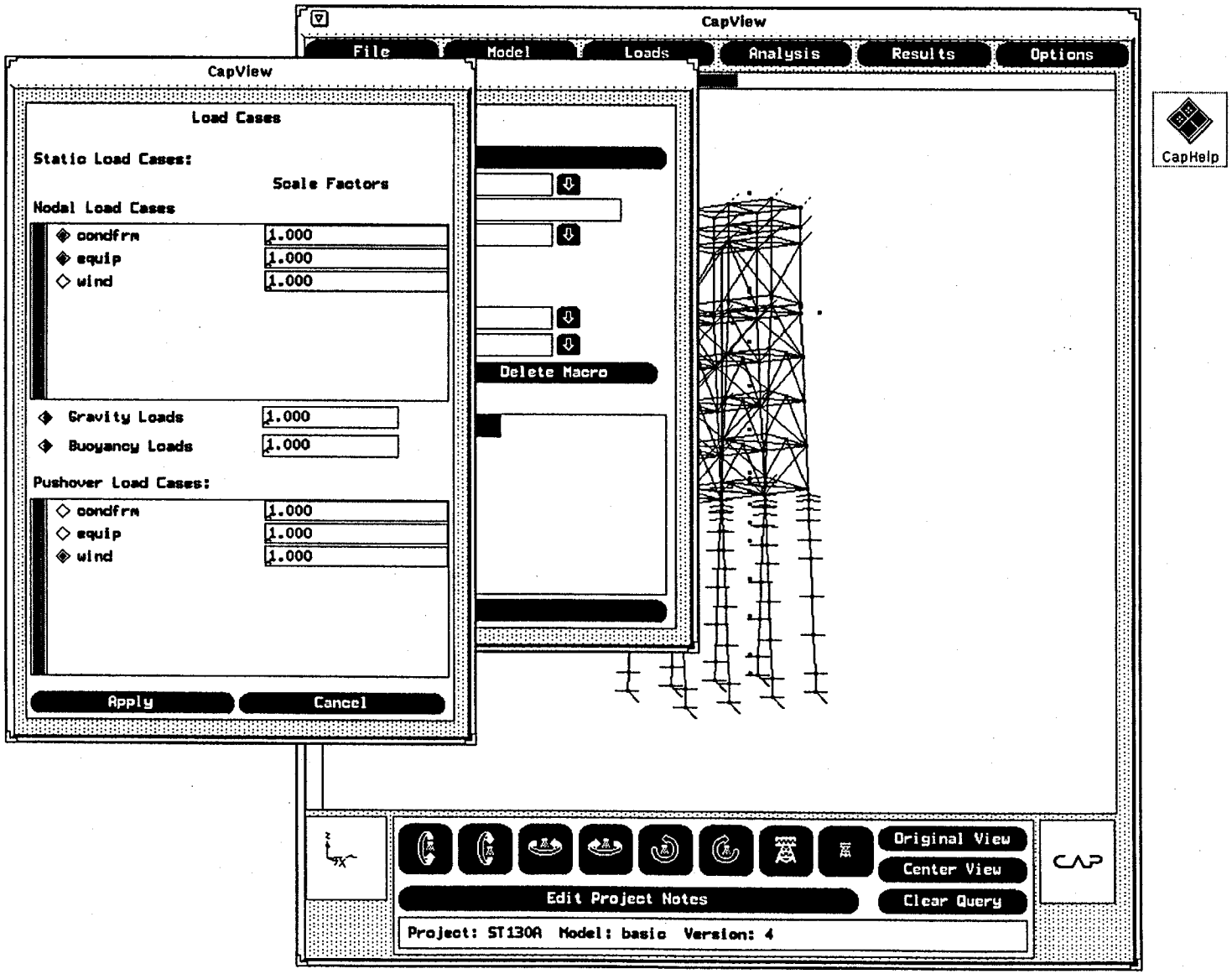

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Load Cases
Static Load Cases:
Scale Factors
Nodal Load Cases
condfrm 1.000
equip 1.000
wind 1.000
Gravity Loads 1.000
Buoyancy Loads 1.000
Pushover Load Cases:
condfrm 1.000
equip 1.000
wind 1.000
Apply
Cancel
Delete Macro
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 10-54

11.0 THE RESULTS MENU

The Results menu presents the user with selections that will help in reviewing and processing the results of an analysis, viewing the deformed structure, color coding members based on their interaction ratios and inelastic events and creating X-Y plots of member forces, IRs, node displacements, etc. When the Results menu is selected, the following menu is displayed (Figure 11-1):

Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots ->
Response Spectrum Results ->

When the mouse is at the "Load Step Plots" selection, the following sub-menu is displayed (Figure 11-2):

Node Displacement...
Member IR...
Member Force...
Member Force/Deformation...
Beam Column Interaction...
Buckling Beam Inter. Force...
Buckling Beam Inter. Disp...
Pile Capacity
Cut Plane Force...
Force Profile...
Pushover Load...
Pushover Load Ratio...
Unbalance Load...

When the mouse is at the "Response Spectrum Results" selection, the following sub-menu is displayed (Figure 11-3):

Mode Shape Plot...
Structure Frequencies...
Mass Participation Factors...
Modal Forces...
Envelope IR Plot...
Envelope IR Report...
Nodal Response Spectra Plot...

11.1 Review Input File

The first selection of the Results menu will display a print file browser dialogue box labeled Input File (Figure 11-4). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is the input file SSModel.inp. If the input file does not exist an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

11.2 Review Output File

The second selection of the Results menu will display a print file browser dialogue box labeled Output File (similar to Figure 11-4). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is the output file SSModel.out. If the output file does not exist an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

11.3 Review Analysis Status File

The third selection of the Results menu will display a print file browser dialogue box labeled Analysis Status File (similar to Figure 11-4). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is the analysis status file SSModel.sts. If the input file does not exist an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

11.4 Maximum IR Plot

The fourth selection of the Results menu will display the Maximum Interaction Ratio Plot dialogue box (Figure 11-5). This dialogue box allows the user to display the structure with the members color-coded according to their interaction ratio (IR). The interaction ratio values are automatically calculated after an analysis based on the API 16th edition Member strength and hydrostatic collapse and the AISC 8th edition member strength code checks. Note that IRs are only calculated for linear beam elements.

Note that when this dialogue box is active the rest of CAP is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc.

The Maximum Interaction Ratio Plot dialogue box has the following:

- An edit field for the load step at which the member IR values are to be used. Valid responses are integer numbers from zero up to the maximum load step value shown below this entry field. Note that load step zero refers to the unloaded structure.
- Apply button. Selecting this button will put CAP in the "IR Mode" in which it stops highlighting members and nodes when the mouse moves near them, and instead the members are color-coded according to their IR values. If the load step value is invalid an appropriate error message is displayed. To exit from the "IR Mode" you must use the Display Defaults selection from the Options menu (section 7.1).
- Cancel button. Selecting this button will close the Maximum Interaction Ratio Plot dialogue box. Note that closing the dialogue box will not exit CAP from the IR Mode.
- Previous Step button. Selecting this button is equivalent to reducing the load step value by one and pressing the Apply button. To exit from the "IR Mode" you must use the Display Defaults selection from the Options menu.
- Next Step button. Selecting this button is equivalent to incrementing the load step value by one and pressing the Apply button. To exit from the "IR Mode" you must use the Display Defaults selection from the Options menu.

11.5 Deflected Shape Plot

The fifth selection of the Results menu will display the Deflected Shape Plot dialogue box (Figure 11-6). This dialogue box allows the user to display the structure in its deformed condition.

Note that when this dialogue box is active the rest of CAP is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc. In addition, this dialogue may be active at the same time as the Inelastic Events Plot dialogue box described in section 11.7.

The Deflected Shape Plot dialogue box has the following:

- An edit field for the magnification factor. The displacements of each node will be multiplied by this factor before figuring out the new location of the node.
- An edit field for the load step at which the node displacements are to be used. Valid responses are integer numbers from zero up to the maximum load step value shown below this entry field. Note that load step zero refers to the unloaded structure.
- Apply button. Selecting this button will display the deformed structure using the specified magnification factor and load step. If the load step value is invalid an appropriate error message is displayed.
- Cancel button. Selecting this button will close the Deflected Shape Plot dialogue box. Note that closing the dialogue box will not restore the undeformed configuration. To restore the undeformed configuration, you must display the deformed shape at load step zero.
- Previous Step button. Selecting this button is equivalent to reducing the load step value by one and pressing the Apply button.
- Next Step button. Selecting this button is equivalent to incrementing the load step value by one and pressing the Apply button.

11.6 Animated Deflected Shape Plot

The sixth selection of the Results menu will display the Deflected Shape Animation Control dialogue box (Figure 11-7). This dialogue box allows the user to display an animation of the deformed structure at a series of load steps.

The Deflected Shape Animation Control dialogue box has the following:

- A load step selection bar that allows the user to choose the range of load steps for which the node displacements are to be animated. To select a range, use the mouse to place the cursor at the time step where the animation will start. Then, while holding down the left mouse button, drag the cursor to the time step where the animation will stop. The range selected will be indicated by a solid bar. When the animation starts a vertical line on the bar will move through the time step indicating the progress of the animation (Figure 11-8). Note that if you press the "Animate" button (described below) without selecting a load step range, CAP will animate through all load steps by default.
- An edit field for the deformation magnification factor. The displacements of each node will be multiplied by this factor before figuring out the new location of the node at each load step.
- A group of radio buttons to select between showing Inelastic Events, Maximum IR ranges or none. If inelastic events are included CAP will color-code all nonlinear members based on the events (buckling, hinging, etc.) they see at that point in the analysis. If Max IRs are included CAP will color-code all linear beam members based on the IR range at that point in the analysis. The colors used are defined with the pen attributes option on the Options menu.
- A checkbox that toggles whether a representation of the water surface is included in the animation.
- An edit field for wave amplitude magnification factor. This value is used to exaggerate the displayed wave height, if shown.
- A pair of radio buttons to choose between displaying the water surface as either a single line or a grid of lines.

Control Buttons

- Animate/Stop button. Pressing this button will display the deflected shape animation of the structure. After an animation has started, pressing this button again will stop the animation.
- Done button. Selecting this button will close the Animated Deflected Shape Plot dialogue box.

CAP can also display graphically other load step results while performing the animation (Figure 11-9). These results must first be extracted and stored in animation files before the animated deflected shape dialogue box is selected. There are two types of animation files. Horizontal animation files (*.had) and vertical animation files (*.vad). Data in a horizontal animation file is a simple plot of a particular result value (e.g. node displacement) as a function of load step. This fuction is displayed in the load step selection bar. Horizontal animation files can be created by any of the Load Step Plots submenu selections except the Force Profile selection using the animation file selection on the respective dialogue boxes. The Force Profile selection creates a vertical animation file. This file has a force profile as a function of load step. A new window is opened showing the force profile paralled to the main display window. Note that the envelope force profile is displayed all the time in dashed lines. As the animation proceeds, the force profile corresponding to the current animation

load step is displayed in solid lines. If the force profile for the current load step is not included in the animation file, a zero force profile is shown.

If animation files were created before the animated deflected shape plot is requested the user is promped to selecte animation files by the Select Animated Plots dialogue box (Figure 11-10). This has the following:

- A scrollable list of checkboxes each representing an animation file. Files with animation files.
- Animate button. Selecting this button will display the Deflected Shape Animation Control dialogue box along with a Profile Animation window if needed.
- Cancel button. Selecting this button will close the dialogue box and will not display the Deflected Shape Animation Control dialogue box.

11.7 Inelastic Events Plot

The seventh selection of the Results menu will display the Inelastic Events Plot dialogue box (Figure 11-11). This dialogue box allows the user to display the structure with the members that went through inelastic events (buckling, hinging, etc.) color-coded. Note that any color-coded members are also query selected so that you can easily perform load step plots (see section 11.10) and other operations on them.

Note that when this dialogue box is active the rest of CAP is still active allowing the user to rotate, pan, and scale the structure, change the display projection, etc. In addition this dialogue may be active at the same time as the Deflected Shape Plot dialogue box described in section 11.5.

The Inelastic Events Plot dialogue box has the following:

- A check box to toggle between highlighting all members that had events up to the current load step or just members that had events at the current load step.
- A check box to toggle between highlighting the worst events up to the current load step or just the current event state.
- An edit field for the load step at which the inelastic events are to be used. Valid responses are integer numbers from zero up to the maximum load step value shown below this entry field. Note that load step zero refers to the unloaded structure.
- Apply button. Selecting this button will display the structure with color-coded members representing members that went through inelastic events. The colors used are defined with the pen attributes option on the Options menu. If the load step value is invalid an appropriate error message is displayed. Note that when CAP processes inelastic events, it automatically switches to Inelastic Event Plot mode described in section 7.1.
- Cancel button. Selecting this button will close the Inelastic Events Plot dialogue box. Note that closing the dialogue box will not exit Inelastic Event Plot mode. To do that, use the Display Defaults selection from the Options mensu (section 7.1).
- Previous Event button. Selecting this button is equivalent to reducing the load step value by the difference to the previous event and pressing the Apply button. Note that when CAP processes inelastic events, it automatically switches to Inelastic Event Plot mode described in section 7.1.
- Next Event button. Selecting this button is equivalent to incrementing the load step value by the difference to the next event and pressing the Apply button. Note that when CAP processes inelastic events, it automatically switches to Inelastic Event Plot mode described in section 7.1.
- Detailed Report button. Selecting this button will make CapView generate a detailed Inelastic Event Report that shows all the Inelastic events during the entire analysis history. This report can be printed.
- Short Report button. Selecting this button will make CapView generate a short Inelastic Event Report that shows for each member that had an inelastic event, the time (and load step) of the first event and the worst event the the member had. This report can be printed.

Note that for Beam Column yield eventsm the I and J node yield surface is listed after the event and that for worst event state the I and J ends are processed independantly.

11.8 Node Displacement LS Plot

The first selection of the Results menu's Load Step Plots submenu will display the Node Displacement LS Plot dialogue box (Figure 11-12). This dialogue box allows the user to create X-Y plots of one or more node displacements in any of the three global directions versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information. Before the user can use this dialogue box, the nodes for which the plots are to be created must be query selected. For more information about query selection, see section 5.1.5.

The Node Displacement LS Plot dialogue box has the following:

- A scrollable list of selected nodes.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the component of the displacements to be plotted or saved in a table file. Available selections are X, Y, and Z.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID, and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6). Note that the plot and table files can be displayed or printed later using the Print Files option on the options menu.
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Node Displacement LS Plot dialogue box.

11.9 Member IR LS Plot

The second selection of the Results menu's Load Step Plots submenu will display the Member Interaction Ratio LS Plot dialogue box (Figure 11-13). This dialogue box allows the user to create X-Y plots of one or more member IR values versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information. Before the user can use this dialogue box, the members for which the plots are to be created must be query selected. For more information about query selection, see section 5.1.5. Note that IR values are only calculated for linear beams.

The Member Interaction Ratio LS Plot dialogue box has the following:

- A scrollable list of selected members.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Member Interaction Ratio LS Plot dialogue box.

11.10 Member Force LS Plot

The third selection of the Results menu's Load Step Plots submenu will display the Member Force LS Plot dialogue box (Figure 11-14). This dialogue box allows the user to create X-Y plots of one or more member forces versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information. Before the user can use this dialogue box, the members for which the plots are to be created must be query selected. For more information about query selection, see section 5.1.5.

The Member Force LS Plot dialogue box has the following:

- A scrollable list of selected members.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of 12 radio buttons to select the force component to be plotted or saved in a table file. The available selections are: Fx, Fy, Fz, Mx, My, and Mz at either end I or end J. Note that Fx represents axial force, Fy represents shear force in a direction parallel to the plane which contains the I, J, and K nodes, Fz represents shear force in a direction perpendicular to the plane which contains the I, J, and K nodes, Mx represents torsion, My represents moment about an axis parallel to the plane which contains the I, J, and K nodes, and Mz represents moment about an axis perpendicular to the plane which contains the I, J, and K nodes (Figure 11-29). Also, note that if the element type of a selected element is Nonlinear Truss or Marshall Strut, then the only valid force component is Fx at end I (axial force).
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Member Force LS Plot dialogue box.

11.11 Member Force/Deformation LS Plot

The fourth selection of the Results menu's Load Step Plots submenu will display the Member Force/Deformation LS Plot dialogue box (Figure 11-30). This dialogue box allows the user to create X-Y plots of one or more member forces versus deformation. It also allows the user to create tables of this information. Before the user can use this dialogue box, the members for which the plots are to be created must be query selected. For more information about query selection, see section 5.1.5.

The Member Force/Deformation LS Plot dialogue box has the following:

- A scrollable list of selected members.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of 12 radio buttons to select the force component to be plotted or saved in a table file. The available selections are: Fx, Fy, Fz, Mx, My, and Mz at either end I or end J. Note that Fx represents axial force, Fy represents shear force in a direction parallel to the plane which contains the I, J, and K nodes, Fz represents shear force in a direction perpendicular to the plane which contains the I, J, and K nodes, Mx represents torsion, My represents moment about an axis parallel to the plane which contains the I, J, and K nodes, and Mz represents moment about an axis perpendicular to the plane which contains the I, J, and K nodes (Figure 11-29). Also, note that if the element type of a selected element is Nonlinear Truss or Marshall Strut, then the only valid force component is Fx at end I (axial force).
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Member Force/Deformation LS Plot dialogue box.

11.12 Beam Column Interaction LS Plot

The fifth selection of the Results menu's Load Step Plots submenu will display the Beam Column Interaction LS Plot dialogue box (Figure 11-32). This dialogue box allows the user to create X-Y plots of one member's Fx/Fxu versus M/Mu (where Fx is axial force, Fxu is the ultimate axial force, M is a moment about y or z local axes and Mu it the coresponding ultimate moment). It also allows the user to create tables of this information. Before the user can use this dialogue box, the member for which the plot is to be created must be query selected. For more information about query selection, see section 5.1.5.

The Beam Column Interaction LS Plot dialogue box has the following:

- The member id.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of 4 radio buttons to select the moment component to be plotted or saved in a table file. The available selections are: My, and Mz at either end I or end J. Note that My represents moment about an axis parallel to the plane which contains the I, J, and K nodes, and Mz represents moment about an axis perpendicular to the plane which contains the I, J, and K nodes (Figure 11-29).
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Member Force/Deformation LS Plot dialogue box.

11.13 Buckling Beam Inter. Force LS Plot

The sixth selection of the Results menu's Load Step Plots submenu will display the Buckling Beam Intermediate Force LS Plot dialogue box (Figure 11-15). This dialogue box allows the user to create X-Y plots of the intermediate force of one buckling beam member versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information. Note that buckling beam members are divided into several segments. The plotted forces are the forces at the segments' ends. Before the user can use this dialogue box, the buckling beam member for which the plots are to be created must be query selected. If more than one member is selected, an error message will be displayed. For more information about query selection, see section 5.1.5.

The Buckling Beam Intermediate Force LS Plot dialogue box has the following:

- Member name.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of 12 radio buttons to select the force component to be plotted or saved in a table file. The available choices are: Fx, Fy, Fz, Mx, My, and Mz at either end I or end J for each segment.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Buckling Beam Intermediate Force LS Plot dialogue box.

11.14 Buckling Beam Inter. Displacement LS Plot

The seventh selection of the Results menu's Load Step Plots submenu will display the Buckling Beam Intermediate Node Displacement LS Plot dialogue box (Figure 11-16). This dialogue box allows the user to create X-Y plots of the intermediate nodes' displacements of one buckling beam member versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information. Note that buckling beam members are divided into several segments. The plotted displacements are the displacements at the intermediate nodes. Before the user can use this dialogue box, the buckling beam member for which the plots are to be created must be query selected. If more than one member is selected, an error message will be displayed. For more information about query selection, see section 5.1.5.

The Buckling Beam Intermediate Node Displacement LS Plot dialogue box has the following:

- Member name.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the component of the displacements to be plotted or saved in a table file. Available selections are X, Y, and Z.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Buckling Beam Intermediate Node Displacement LS Plot dialogue box.

11.15 Pile Capacity

The eighth selection of the Results menu's Load Step Plots submenu will display the Pile Capacity LS Plot dialogue box (Figure 11-17). This dialogue box allows the user to create X-Y plots of one or more pile capacities in any of the three global directions versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information.

Before the user can use this dialogue box, the elements for which the plots are to be created must be query selected. For more information about query selection, see section 5.1.5. The elements that should be selected for these plots are those at the top of the pile (i.e. the pile element that has a PSAS element at the bottom, but not at the top).

The Pile Capacity LS Plot dialogue box has the following:

- A scrollable list of selected pile head elements.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot on the screen, saving a plot in a print file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Pile Capacity LS Plot dialogue box.

11.16 Cut Plane Force LS Plot

The ninth selection of the Results menu's Load Step Plots submenu will display the Cut Plane Force LS Plot dialogue box (Figure 11-18). This dialogue box allows the user to create X-Y plots of the force at at cut plane versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information.

The Cut Plane Force LS Plot dialogue box has the following:

- A pair of radio buttons that allows the user to select Cut Mode. Available options are a Horizontal Plane and a General Plane specification.
- An edit field defining the horizontal plane (visible when the cut mode is Horizontal Plane). The horizontal plane is defined by an elevation (Figure 11-18).
- A group of edit fields defining the general plane (visible when the cut mode is General Plane). The plane is defined by a point that is in the plane, a direction vector that is perpendicular to the plane, a direction vector for the major axis of the plane, and the plane width and plane length (Figure 11-33).
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of 6 radio buttons to select the force component to be plotted or saved in a table file. The available choices are: Fx, Fy, Fz, Mx, My, and Mz. Note that when the Cut Mode is General Plane, then Fx represents force along the major axis of the plane (shear), Fy represents force along the minor axis of the plane (shear), Fz represents force along the normal axis of the plane (axial), Mx represents moment about the major axis of the plane (overturning), My represents moment about the minor axis of the plane (overturning), and Mz represents moment about the normal axis of the plane (torsion). When the Cut Mode is Horizontal Plane, then the cut plane coordinates are parallel to the global coordinate system.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID, a radio button list to select the component of the deck node displacement, and a flag to select between displacement in cut plane coordinate or global coordinates (Figure 11-33).
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the

plot and table files can be displayed or printed later using the Print Files option on the Options menu.

- Cancel button. Selecting this button will close the Cut Plane Force LS Plot dialogue box.

11.17 Force Profile LS Plot

The tenth selection of the Results menu's Load Step Plots submenu will display the Force Profile LS Plot dialogue box (Figure 11-19). This dialogue box allows the user to create X-Y plots of a force profile versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information.

The Force Profile LS Plot dialogue box has the following:

- A scrollable radio list of elevations that are to be included in the force profile.
- A scrollable radio list of element groups that are to be excluded from the force profile.
- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of 6 radio buttons to select the force component to be plotted or saved in a table file. The available choices are: Fx, Fy, Fz, Mx, My, and Mz. Note that Fx represents axial force, Fy represents shear force in a direction parallel to the plane which contains the I, J, and K nodes, Fz represents shear force in a direction perpendicular to the plane which contains the I, J, and K nodes, Mx represents torsion, My represents moment about an axis parallel to the plane which contains the I, J, and K nodes, and Mz represents moment about an axis perpendicular to the plane which contains the I, J, and K nodes (Figure 11-29).
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot in a vertical window next to the structure in the Animated Deflected Shape Plot (see section 11.6). This allows you to animate story shears along with and in sync with animated structure deformation.
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Cut Plane Force LS Plot dialogue box.

11.18 Pushover Load LS Plot

The eleventh selection of the Results menu's Load Step Plots submenu will display the Pushover Load LS Plot dialogue box (Figure 11-20). This dialogue box allows the user to create X-Y plots of the magnitude of the applied pushover load versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information.

The Pushover Load LS Plot dialogue box has the following:

- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the pushover analysis macro type. Available selections are: Dynamic, and Static. Select the Dynamic if you ran a dynamic pushover analysis. If you ran a static pushover analysis, then you need to select Static option.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Pushover Load LS Plot dialogue box.

11.19 Pushover Load Ratio LS Plot

The twelfth selection of the Results menu's Load Step Plots submenu will display the Pushover Load Ratio LS Plot dialogue box (Figure 11-21). This dialogue box allows the user to create X-Y plots of the magnitude of the applied push-over load ratio versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information.

The Pushover Load Ratio LS Plot dialogue box has the following:

- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the pushover analysis macro type. Available selections are: Dynamic, and Static. Select the Dynamic if you ran a dynamic pushover analysis. If you ran a static pushover analysis, then you need to select Static option.
- The value of the Pushover load.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Pushover Load Ratio LS Plot dialogue box.

11.20 Unbalance Force LS Plot

The thirteenth selection of the Results menu's Load Step Plots submenu will display the Unbalance Force LS Plot dialogue box (Figure 11-22). This dialogue box allows the user to create X-Y plots of the norm of the unbalance force versus time, or versus the displacement of a deck node. It also allows the user to create tables of this information.

The Unbalance Force LS Plot dialogue box has the following:

- Two edit fields for the first and last load steps. Valid responses are integer numbers from zero up to the maximum load step value shown below these fields, and the last load step must be larger than the first load step. Note that load step zero refers to the unloaded structure.
- A list of radio buttons to select the pushover analysis macro type. Available selections are: Dynamic, and Static. Select the Dynamic if you ran a dynamic pushover analysis. If you ran a static pushover analysis, then you need to select Static option.
- A list of radio buttons to select the abscissa. Available selections are: load step, deck displacement, and time. If the user selects the abscissa to be the deck displacement, then an edit field is shown for the user to enter a deck node ID and a radio button list to select the component of the deck node displacement.
- A list of check boxes labeled "To:" allow the user to select the destination of the data. The user can toggle on and off: creating a plot file, creating an animation file, and creating a table file. If You create an animation file you can tell CAP to display this plot along with the load step bar in the Animated Deflected Shape Plot (see section 11.6).
- An edit field for a print file name. This name is used for plot and table files.
- A radio button list to toggle between overwriting the print file or appending to its contents. The latter selection allows the user to save more than one plot in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to a plot or table.
- Apply button. Selecting this button will make CAP create the selected plots and/or tables and possibly show the plot on the screen. Note that the plot and table files can be displayed or printed later using the Print Files option on the Options menu.
- Cancel button. Selecting this button will close the Unbalance Force LS Plot dialogue box.

11.21 Mode Shape Plot

The first selection of the Results menu's Response Spectrum submenu will display the Mode Shape Plot dialogue box (Figure 11-23). The Mode Shape Plot dialogue box consists of the following:

- An edit field labeled "Magnification Factor" which is used to set the magnification factor for the plot.
- An edit field labeled "Cycle Count" which is used to set the number of cycles to plot.
- An edit field labeled "Mode Shape" which is used to set the mode shape for the plot. Following this the maximum mode shape is indicated.
- A checkbox labeled "Vibrate" where vibration can be toggled on or off.
- Apply button. Selecting this button will apply the selections currently displayed and create a Mode Shape Plot file which will be shown in a plot viewing dialogue box.
- Cancel button. Selecting this button will close the Mode Shape Plot dialogue box.
- Previous Mode button. Selecting this button will redisplay the Mode Shape Plot dialogue box with the values of the previous mode shape plot.
- Next Mode button. Selecting this button will redisplay the Mode Shape Plot dialogue box with the values of the next mode shape plot.

11.22 Structure Frequencies

The second selection of the Results menu's Response Spectrum submenu will display a print file browser dialogue box labeled Structure Frequencies (Figure 11-24). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file extracted from the output file SSModel.out. This file has a list of structure frequencies (in radians/sec and cycles/sec) and periods (in seconds). If the frequencies were not computed during the most recent analysis, an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

11.23 Mass Participation Factors

The third selection of the Results menu's Response Spectrum submenu will display a print file browser dialogue box labeled Mass Participation Factors (Figure 11-25). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file extracted from the output file CapResp.out. This file lists the mass captured by each computed mode shape and compares the sum of the mass captured by each mode with total mass in the structure. If the response spectrum analysis was not performed during the most recent analysis, an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

11.24 Modal Forces

The fourth selection of the Results menu's Response Spectrum submenu will display a print file browser dialogue box labeled Modal Forces (Figure 11-26). This dialogue box allows the user to review the contents of a file. In this case the reviewed file is a temporary file extracted from the output file CapCut.Forces. This file lists the modal forces at the major elevations of the structure. If the response spectrum analysis was not performed during the most recent analysis, an appropriate error message is displayed instead of the dialogue box. The dialogue box has two buttons, a Search button and a Done button.

- Search button. Selecting this button will pop-up a text search dialogue box.
- Done button. Selecting this button will close the dialogue box.

11.25 Envelope Interaction Ratio Plot

The fifth selection of the Results menu's Response Spectrum submenu will display the Envelope Interaction Ratio Plot dialogue box (Figure 11-27). The Envelope Interaction Ratio Plot dialogue box consists of the following:

- A pair of radio buttons labeled Load Case which is used to choose between Gravity+Dynamic and Gravity-Dynamic load cases.
- Apply button. Selecting this button will put CAP in the "IR Mode" in which it stops highlighting members and nodes and the members are color-coded according to their IR values. To exit from the "IR Mode" you must use the Display Defaults selection from the Options menu (section 7.1).
- Cancel button. Selecting this button will close the Envelope Interaction Ratio Plot dialogue box.

11.26 Member Envelope Interaction Ratio Report

The sixth selection of the Results menu's Response Spectrum Results submenu will display the Member Envelope Interaction Ratio Report dialogue box (Figure 11-28). This dialogue box allows the user to create a report of the Member Envelope Interaction Ratio of one or more member IR values. Before the user can use this dialogue box, the members for which the plots are to be created must be query selected. For more information about query selection, see section 5.1.5.

The Member Envelope Interaction Ratio dialogue box has the following:

- A scrollable list of selected members.
- An edit field for a file name. This name is used for the report file.
- A radio button list to toggle between overwriting the report file or appending to its contents. The latter selection allows the user to save more than one report in a single file.
- An edit field labeled title. The user can use this entry field to add an additional title to the report.
- Apply button. Selecting this button will make CAP create the selected report.
- Cancel button. Selecting this button will close the Member Envelope Interaction Ratio Report dialogue box.

11.27 Nodal Response Spectra Plot

The seventh selection of the Results menu's Response Spectrum Results submenu will display the Nodal Response Spectra Plot dialogue box (Figure 11-31) This dialogue box allows the user to create a response spectrum plot based on a nodal displacement record.

The Nodal Response Spectra Plot dialogue box consists of the following:

- An edit field labeled "Node Name" that is used to set the node name.
- An edit field labeled "First Load Step" that is used to set the first load step of the node displacement record record to be used in generating the response spectrum.
- An edit field labeled "Last Load Step" that is used to set the last load step of the node displacement record record to be used in generating the response spectrum.
- Tha Maximum Load Step that is used in the first and last load step fields is shown next. Note that the minimum load step is 1.
- A set of radio buttons labeled "Displacement Direction" that lets the user choose between the X, Y or Z direction for the node displacement.
- An edit field labeled "Response Spectra Start Period" that is used to set the response spectra start period.
- An edit field labeled "Response Spectra End Period" that is used to set the response spectra end period.
- An edit field labeled "Number of Response Spectra Period" that is used to set the number of response spectra period to be computed.
- An edit field labeled "Structure Damping" that is used to set the structure damping to be used in response spectra computations.
- A pair of radio buttons labeled "Plot Type" that lets the user choose between Linear and Four Way ground motion record plots.
- A set of radio buttons labeled "Response Type" that lets the user choose between Acceleration, Velocity or Displacement response spectrum ground motion record data to plot. This set is only visible when the selected plot type is Linear.
- A checkbox labeled "Show Detailed Grid" that lets the user toggle a detailed log-log grid for the Four-Way plot. This checkbox is only visible when the selected plot type is Four-Way.
- A checkbox labeled "Show API Values" toggles whether API values are shown.
- A set of radio buttons labeled "Soil Type" that lets the user choose between Rock, Shallow Strong Alluvium and Deep Strong Alluvium soils. The soil type is used to evaluate the recommended API response spectrum.
- An edit field labeled "File Name" that is used to set the file name of the ground motion record plot.
- A pair of radio buttons labeled "Overwrite" and "Append" which determine whether an existing file of the same name should be overwritten or appended (allowing multiple plots in one file).
- An edit field labeled "Title" that is used to set the user title of the ground motion record plot.
- Apply button. Selecting this button will create a plot file with the given name and display the plot in a viewer.

- Cancel button. Selecting this button will close the dialogue box.

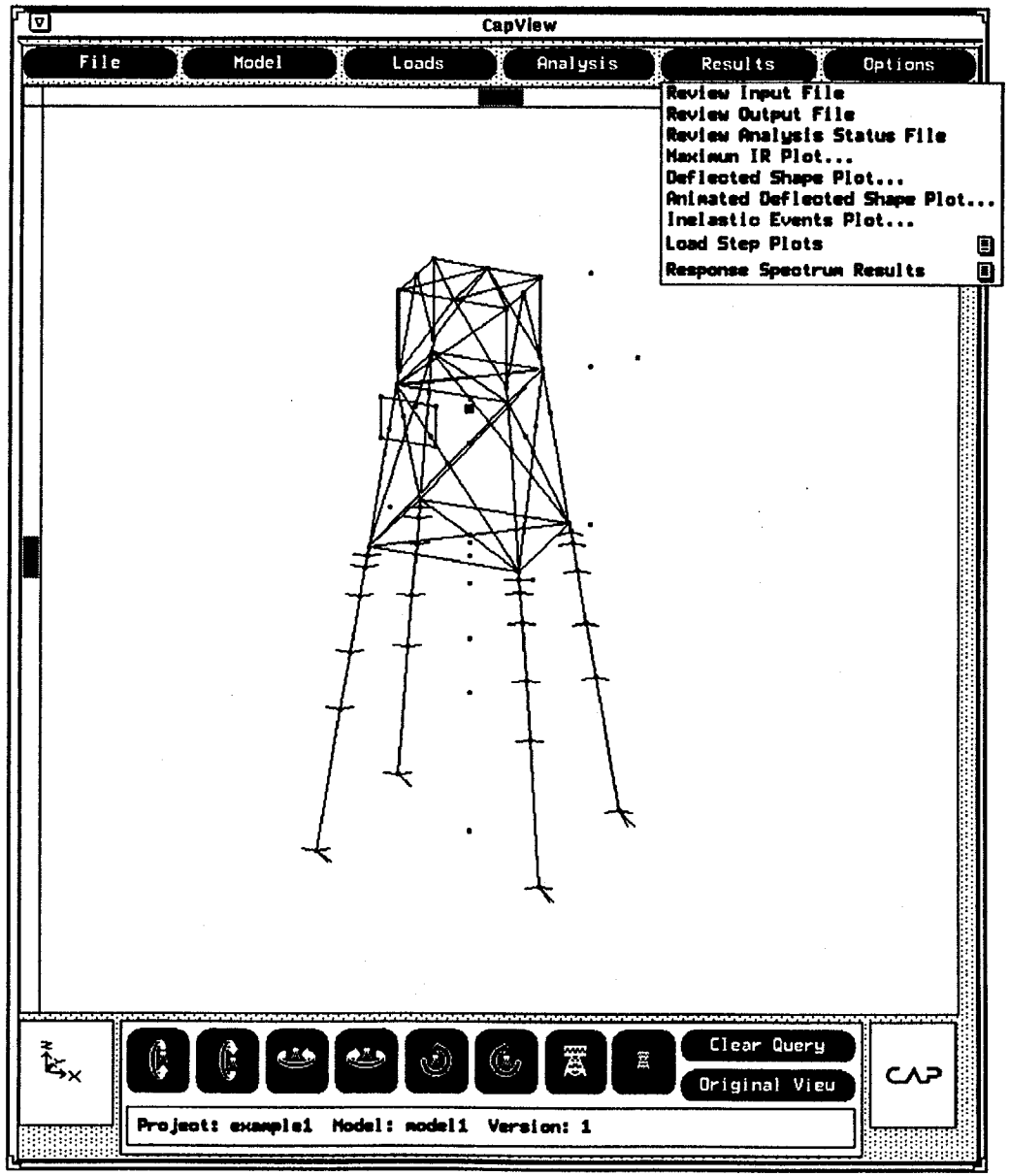

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-1

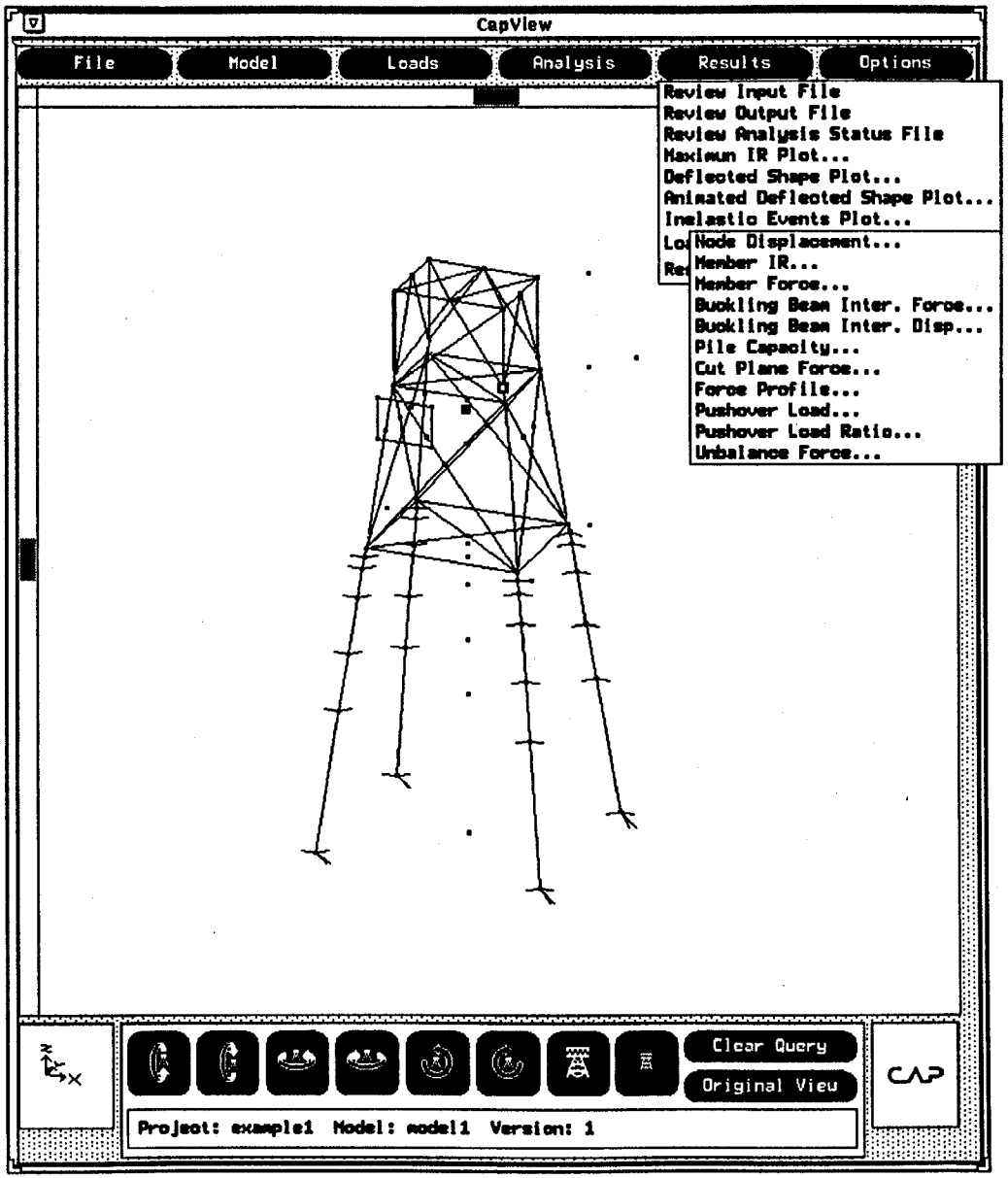

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Node Displacement...
Member IR...
Member Force...
Buckling Beam Inter. Force...
Buckling Beam Inter. Disp...
Pile Capacity...
Cut Plane Force...
Force Profile...
Pushover Load...
Pushover Load Ratio...
Unbalance Force...
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-2

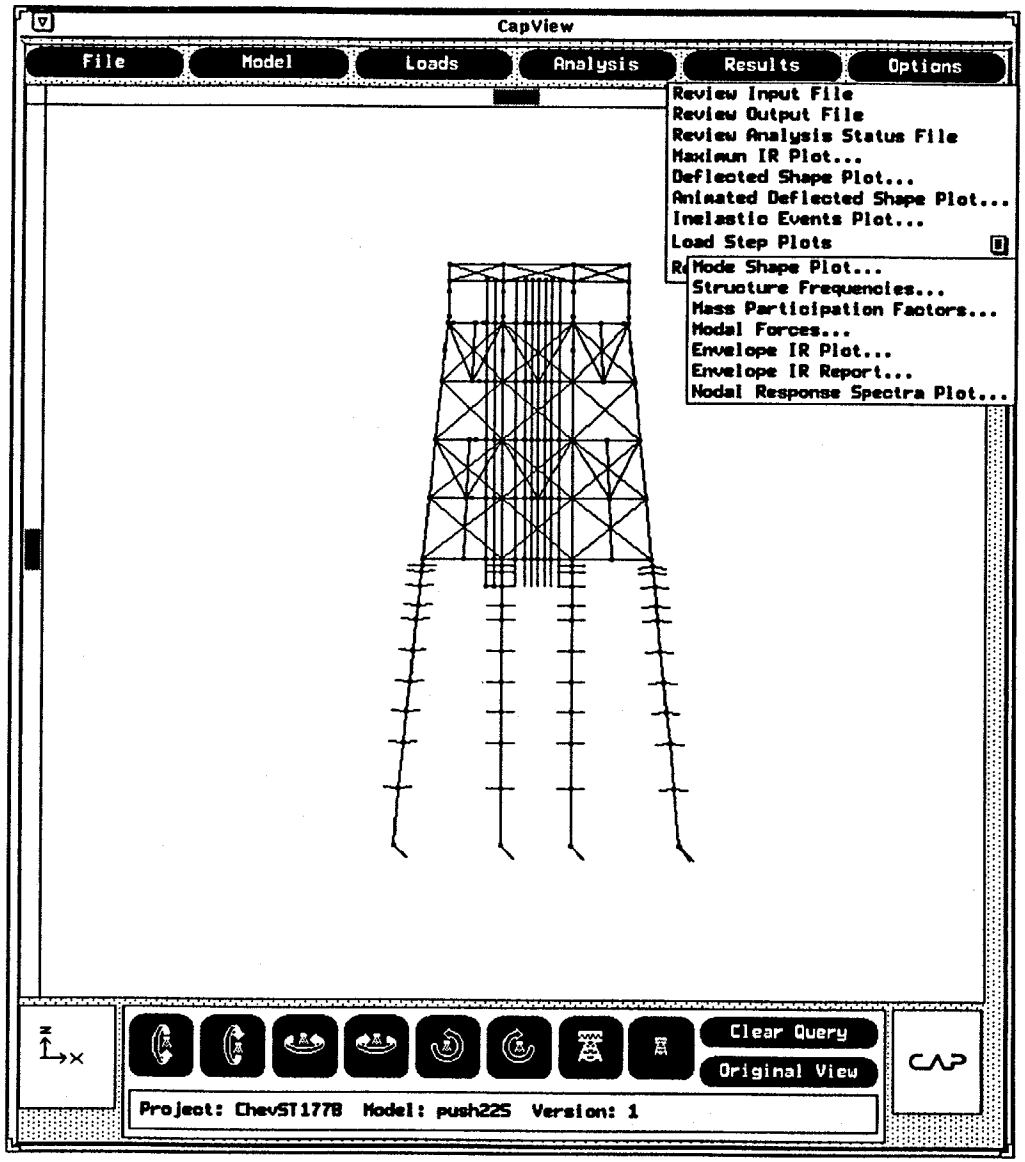

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Mode Shape Plot...
Structure Frequencies...
Mass Participation Factors...
Modal Forces...
Envelope IR Plot...
Envelope IR Report...
Nodal Response Spectra Plot...
Clear Query
Original View
Project: ChevST177B Model: push22S Version: 1

Figure 11-3

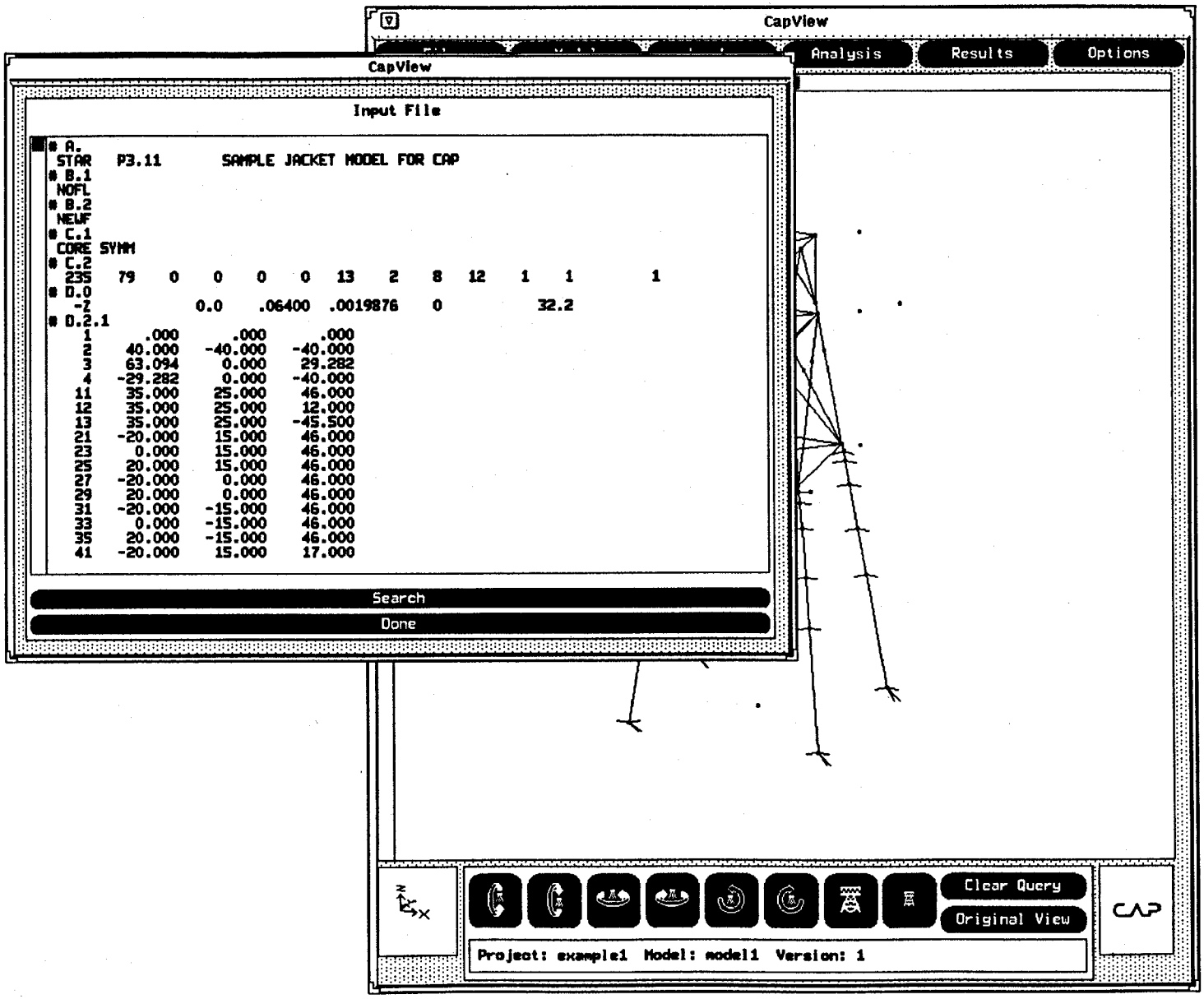

CapView
Analysis
Results
Options
CapView
Input File
A.
STAR P3.11 SAMPLE JACKET MODEL FOR CAP
B.1
NOFL
B.2
NEWF
C.1
CORE SYMM
C.2
235 79 0 0 0 0 13 2 8 12 1 1 1
D.0
-2 0.0 .06400 .0019876 0 32.2
D.2.1
1 .000 .000 .000
2 40.000 -40.000 -40.000
3 63.094 0.000 29.282
4 -29.282 0.000 -40.000
11 35.000 25.000 46.000
12 35.000 25.000 12.000
13 35.000 25.000 -45.500
21 -20.000 15.000 46.000
23 0.000 15.000 46.000
25 20.000 15.000 46.000
27 -20.000 0.000 46.000
29 20.000 0.000 46.000
31 -20.000 -15.000 46.000
33 0.000 -15.000 46.000
35 20.000 -15.000 46.000
41 -20.000 15.000 17.000
Search
Done
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-4

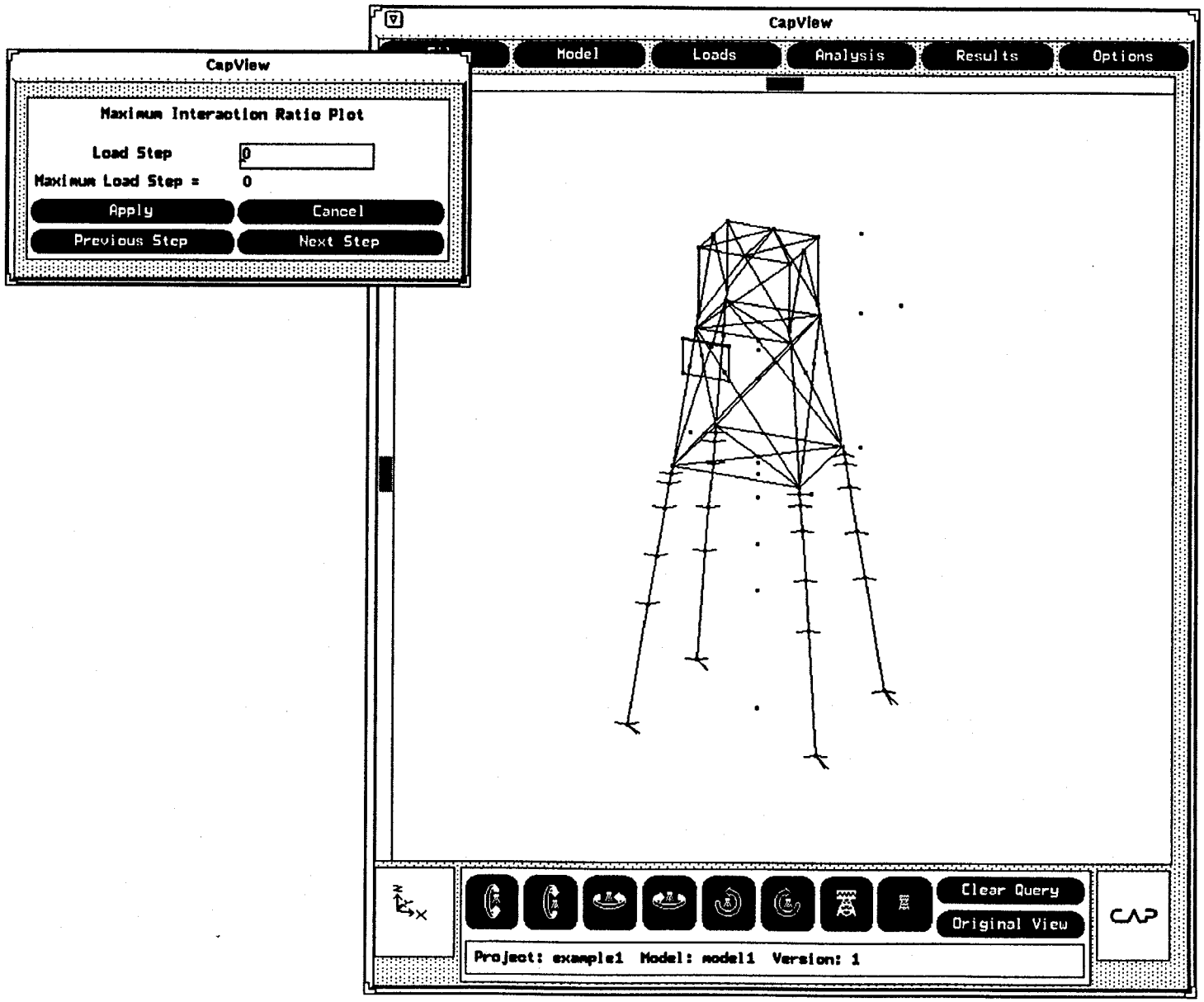

CapView
Maximum Interaction Ratio Plot
Load Step
0
Maximum Load Step =
0
Apply
Cancel
Previous Step
Next Step
CapView
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-5

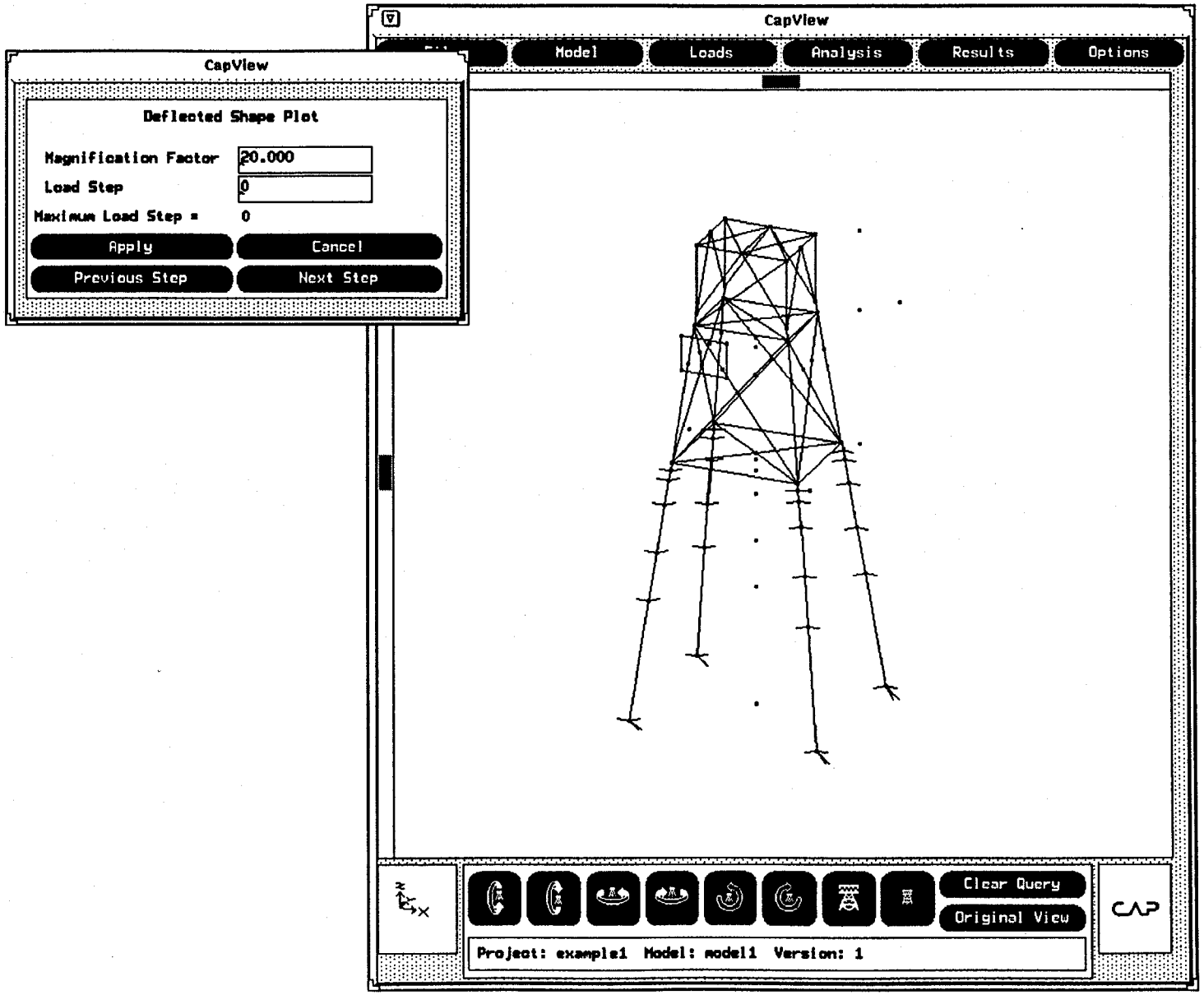

CapView
Model
Loads
Analysis
Results
Options
CapView
Deflected Shape Plot
Magnification Factor
20.000
Load Step
0
Maximum Load Step =
0
Apply
Cancel
Previous Step
Next Step
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-6

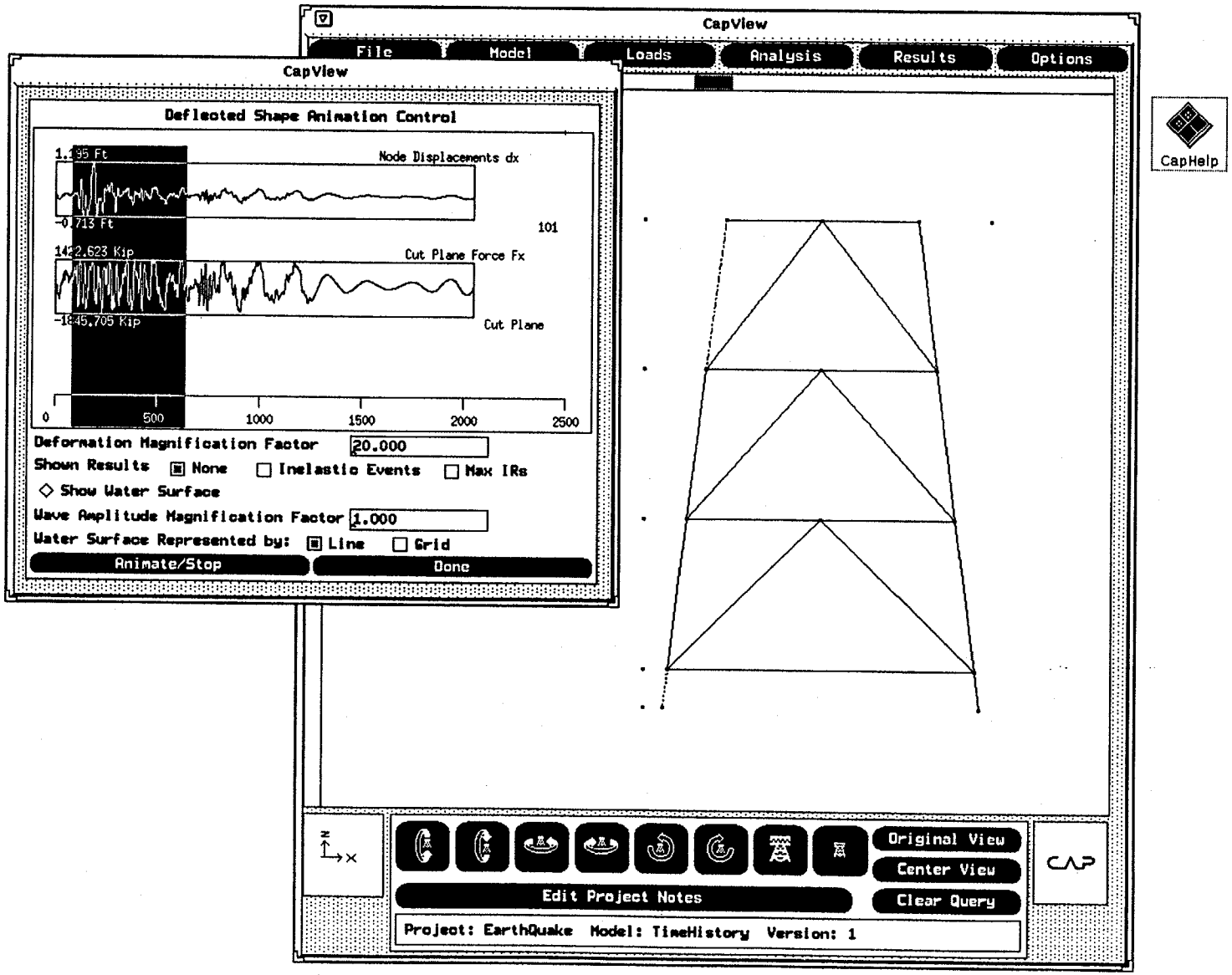

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Deflected Shape Animation Control
Node Displacements dx
101
Cut Plane Force Fx
Cut Plane
0
500
1000
1500
2000
2500
Deformation Magnification Factor
20.000
Shown Results
None
Inelastic Events
Max IRs
Show Water Surface
Wave Amplitude Magnification Factor
1.000
Water Surface Represented by:
Line
Grid
Animate/Stop
Done
Original View
Center View
Clear Query
Edit Project Notes
Project: EarthQuake Model: TimeHistory Version: 1

Figure 11-7

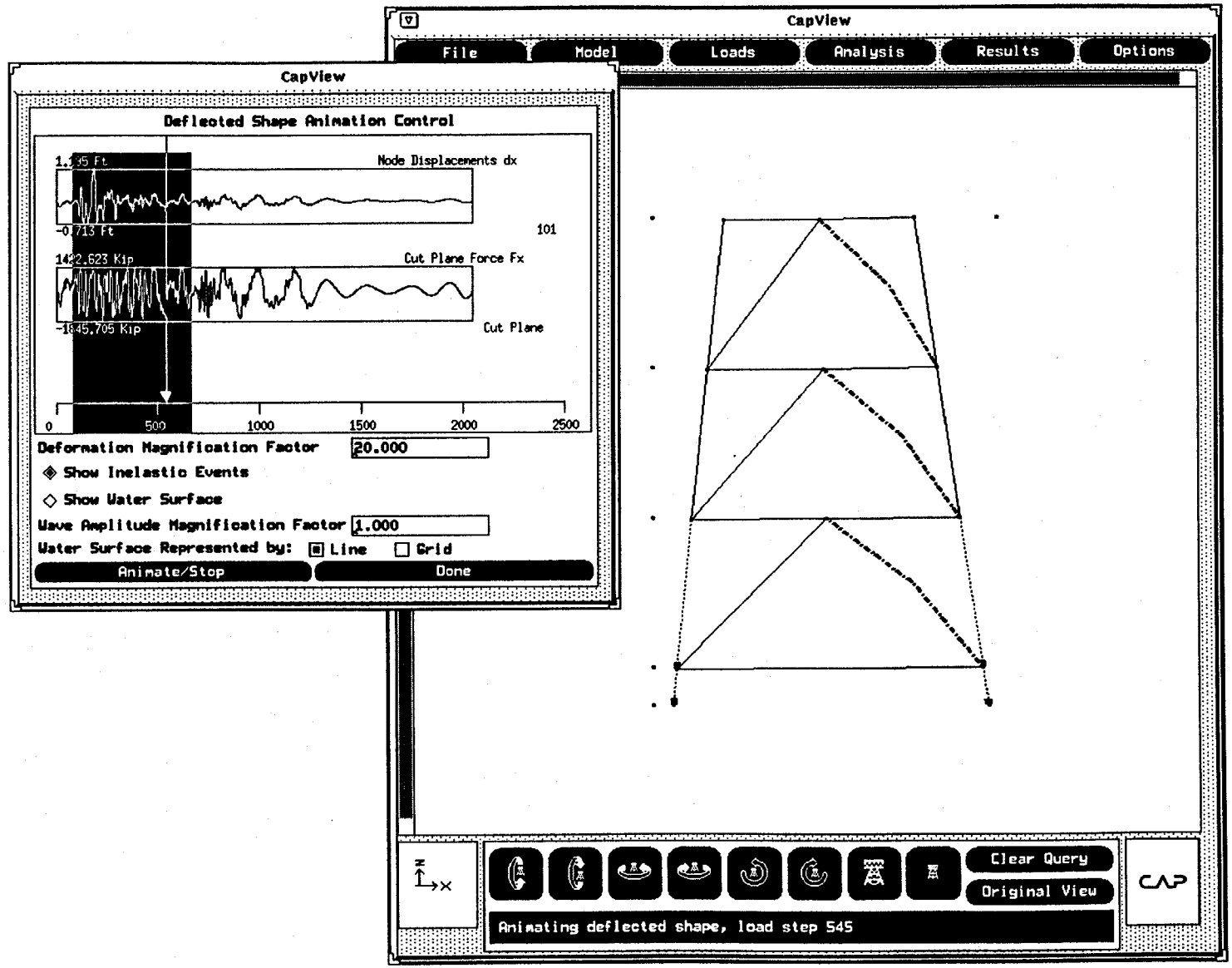

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Deflected Shape Animation Control
Node Displacements dx
101
Cut Plane Force Fx
Cut Plane
0
500
1000
1500
2000
2500
Deformation Magnification Factor
20.000
Show Inelastic Events
Show Water Surface
Wave Amplitude Magnification Factor
1.000
Water Surface Represented by:
Line
Grid
Animate/Stop
Done
Clear Query
Original View
Animating deflected shape, load step 545

Figure 11-8

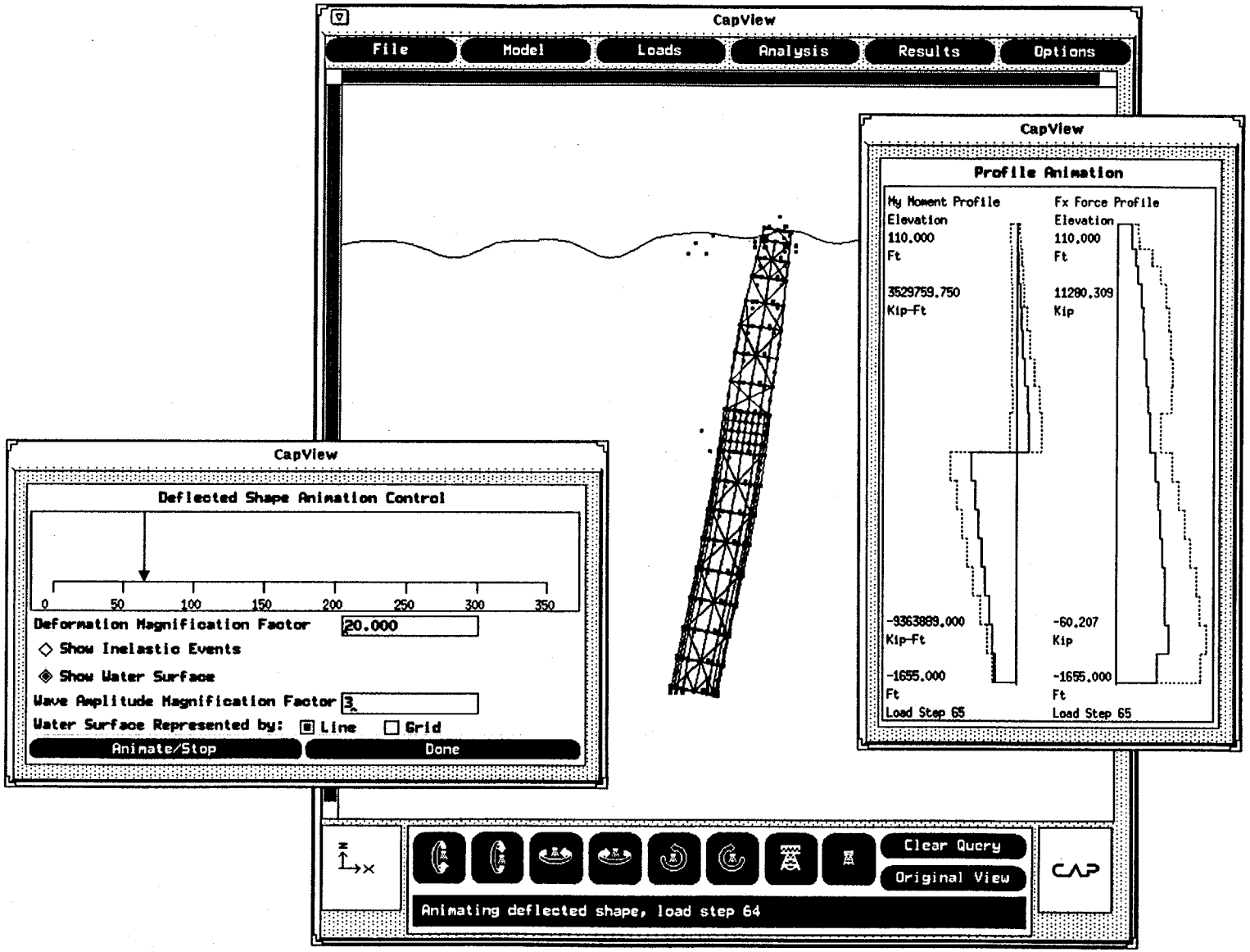

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Profile Animation
My Moment Profile
Elevation
110.000
Ft
3529759.750
Kip-Ft
-9363889.000
Kip-Ft
-1655.000
Ft
Load Step 65
Fx Force Profile
Elevation
110.000
Ft
11280.309
Kip
-60.207
Kip
-1655.000
Ft
Load Step 65
CapView
Deflected Shape Animation Control
0
50
100
150
200
250
300
350
Deformation Magnification Factor
20.000
Show Inelastic Events
Show Water Surface
Wave Amplitude Magnification Factor
3
Water Surface Represented by:
Line
Grid
Animate/Stop
Done
Clear Query
Original View
Animating deflected shape, load step 64

Figure 11-9

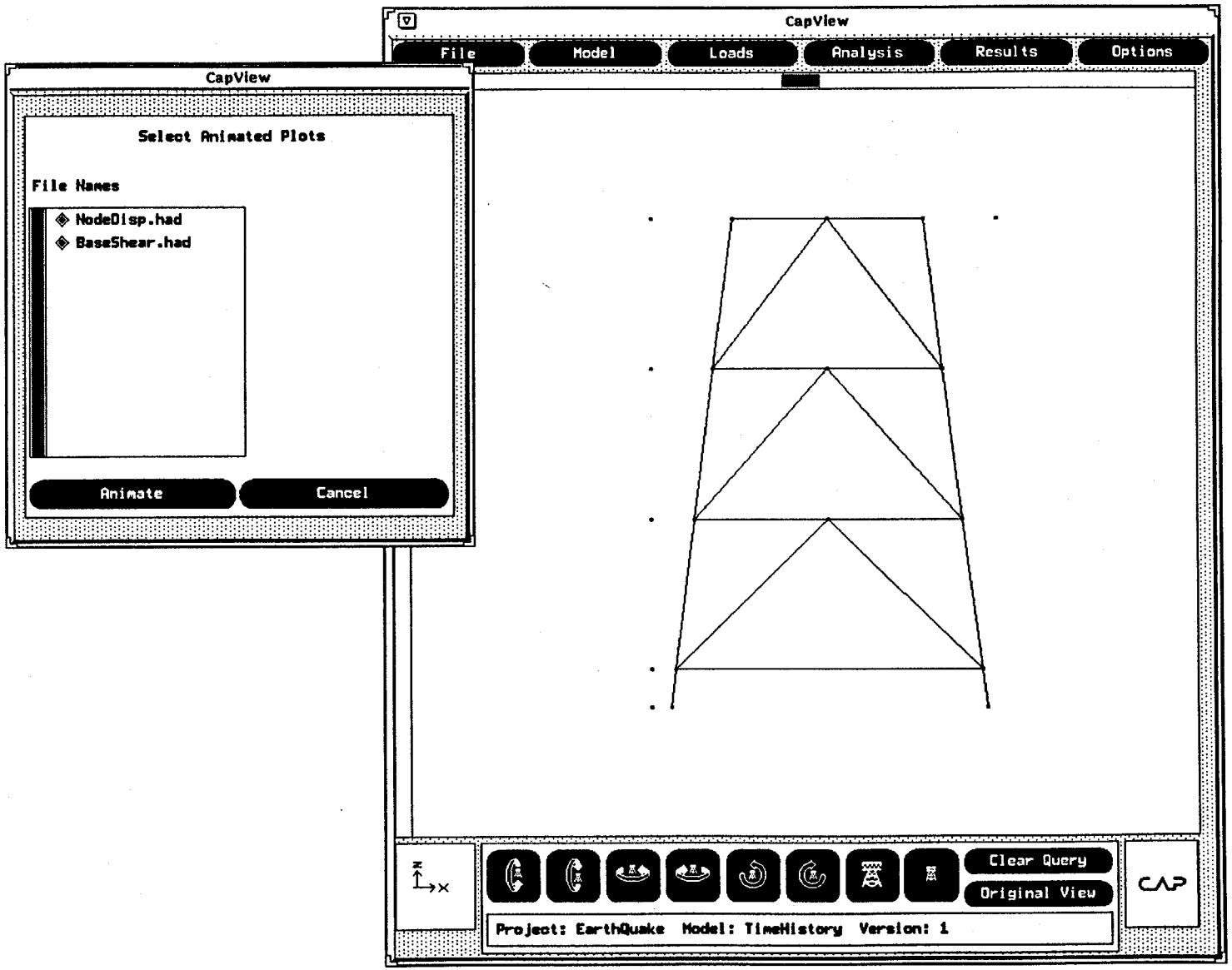

CapView
File
Model
Loads
Analysis
Results
Options
CapView
Select Animated Plots
File Names
NodeDisp.had
BaseShear.had
Animate
Cancel
Clear Query
Original View
Project: EarthQuake Model: TimeHistory Version: 1

Figure 11-10

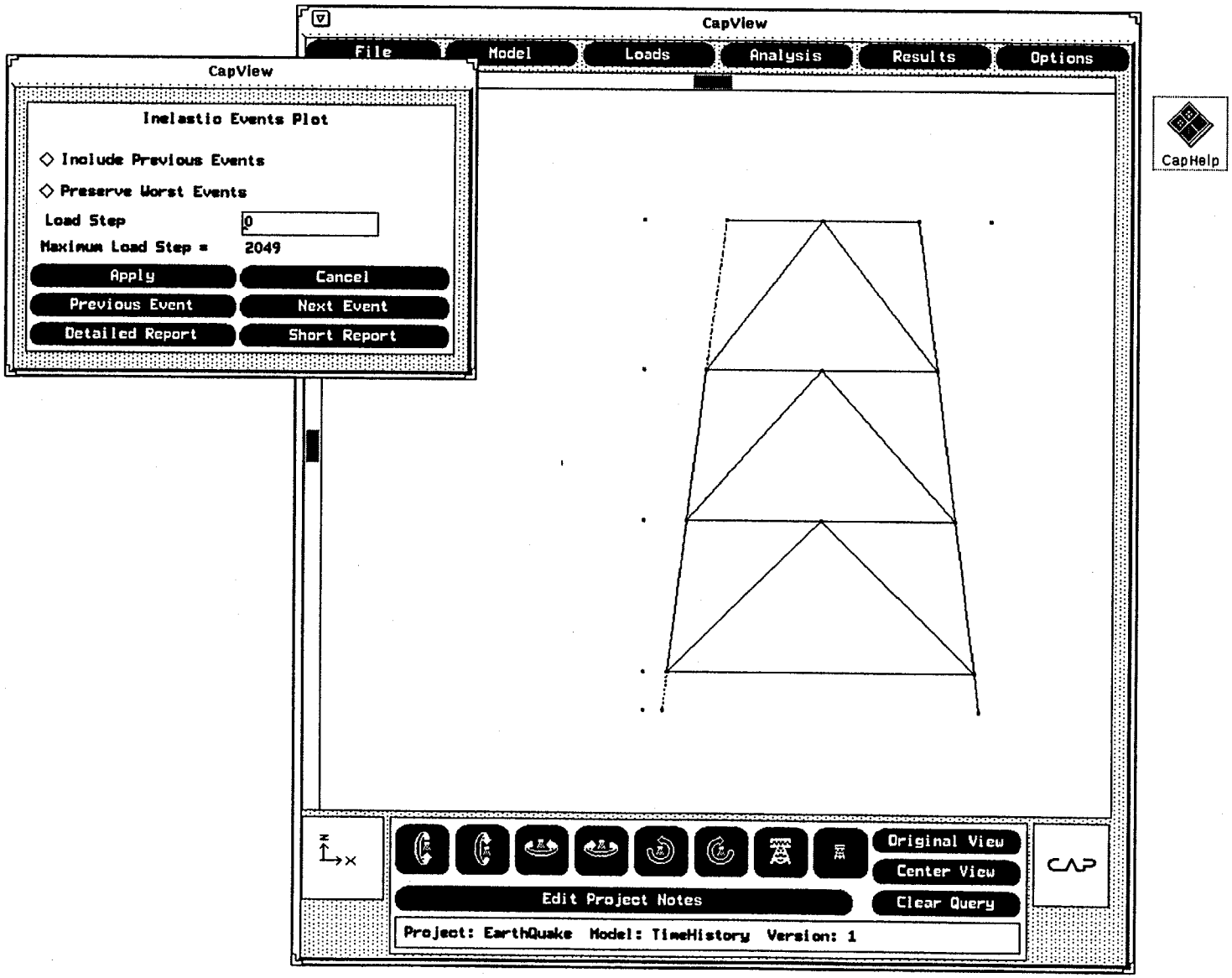

CapView
File
Model
Loads
Analysis
Results
Options
CapHelp
CapView
Inelastic Events Plot
Include Previous Events
Preserve Worst Events
Load Step
0
Maximum Load Step =
2049
Apply
Cancel
Previous Event
Next Event
Detailed Report
Short Report
Original View
Center View
Clear Query
Edit Project Notes
Project: EarthQuake Model: TimeHistory Version: 1

Figure 11-11

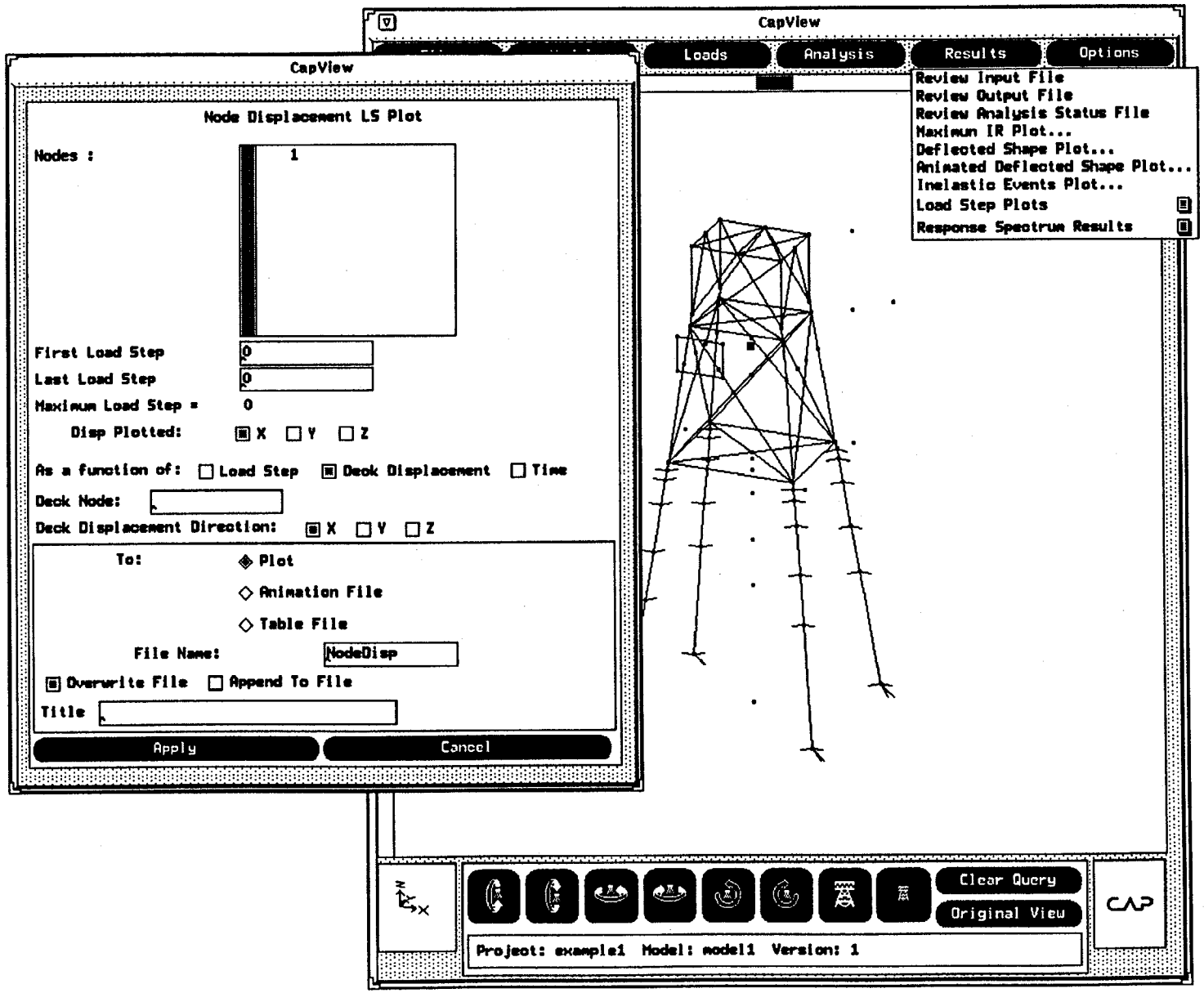

CapView
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Node Displacement LS Plot
Nodes :
1
First Load Step
0
Last Load Step
0
Maximum Load Step =
0
Disp Plotted:
X
Y
Z
As a function of:
Load Step
Deck Displacement
Time
Deck Node:
Deck Displacement Direction:
X
Y
Z
To:
Plot
Animation File
Table File
File Name:
NodeDisp
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-12

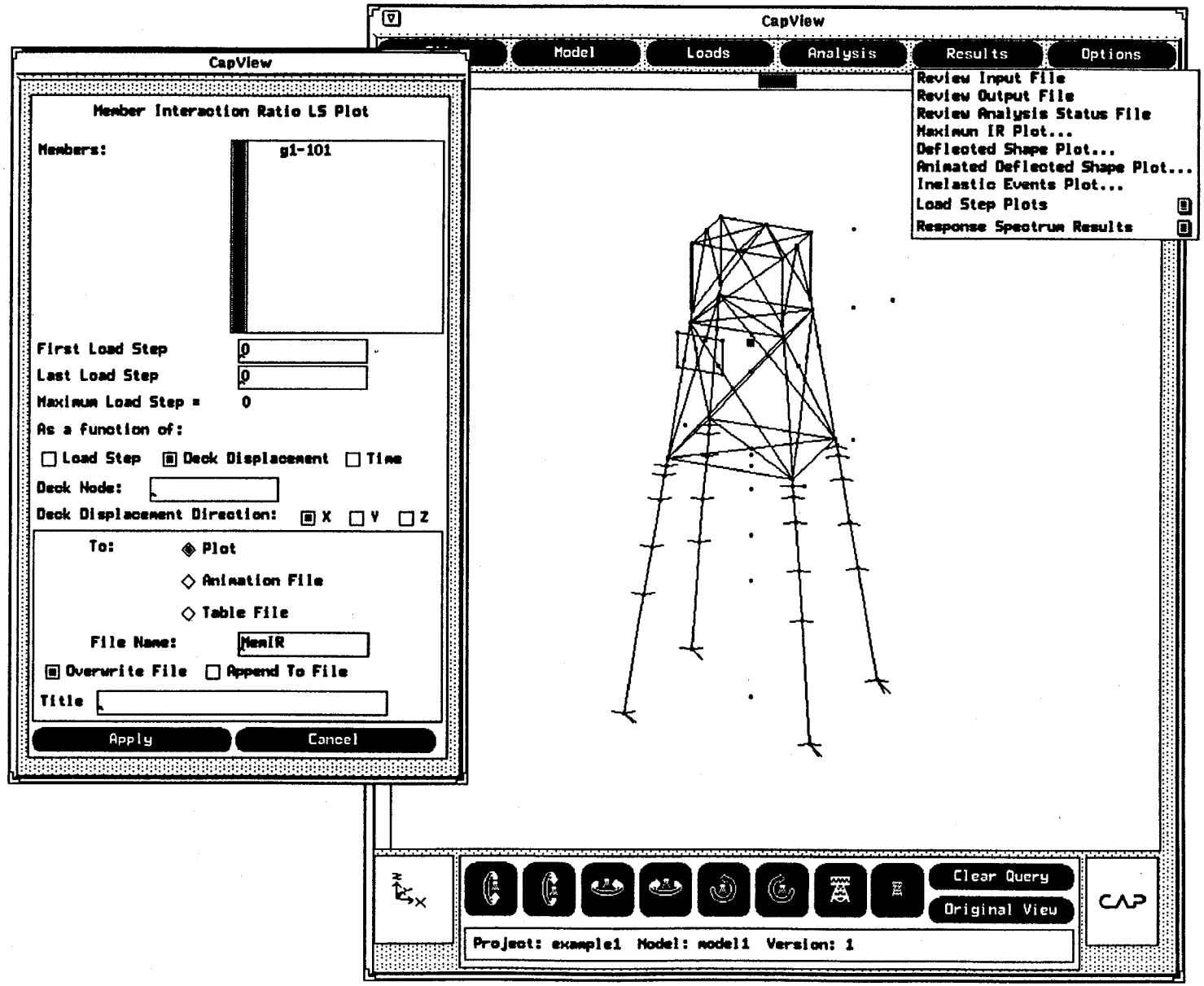

CapView
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Member Interaction Ratio LS Plot
Members:
g1-101
First Load Step
0
Last Load Step
0
Maximum Load Step =
0
As a function of:
Load Step
Deck Displacement
Time
Deck Node:
Deck Displacement Direction:
X
Y
Z
To:
Plot
Animation File
Table File
File Name:
MemIR
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-13

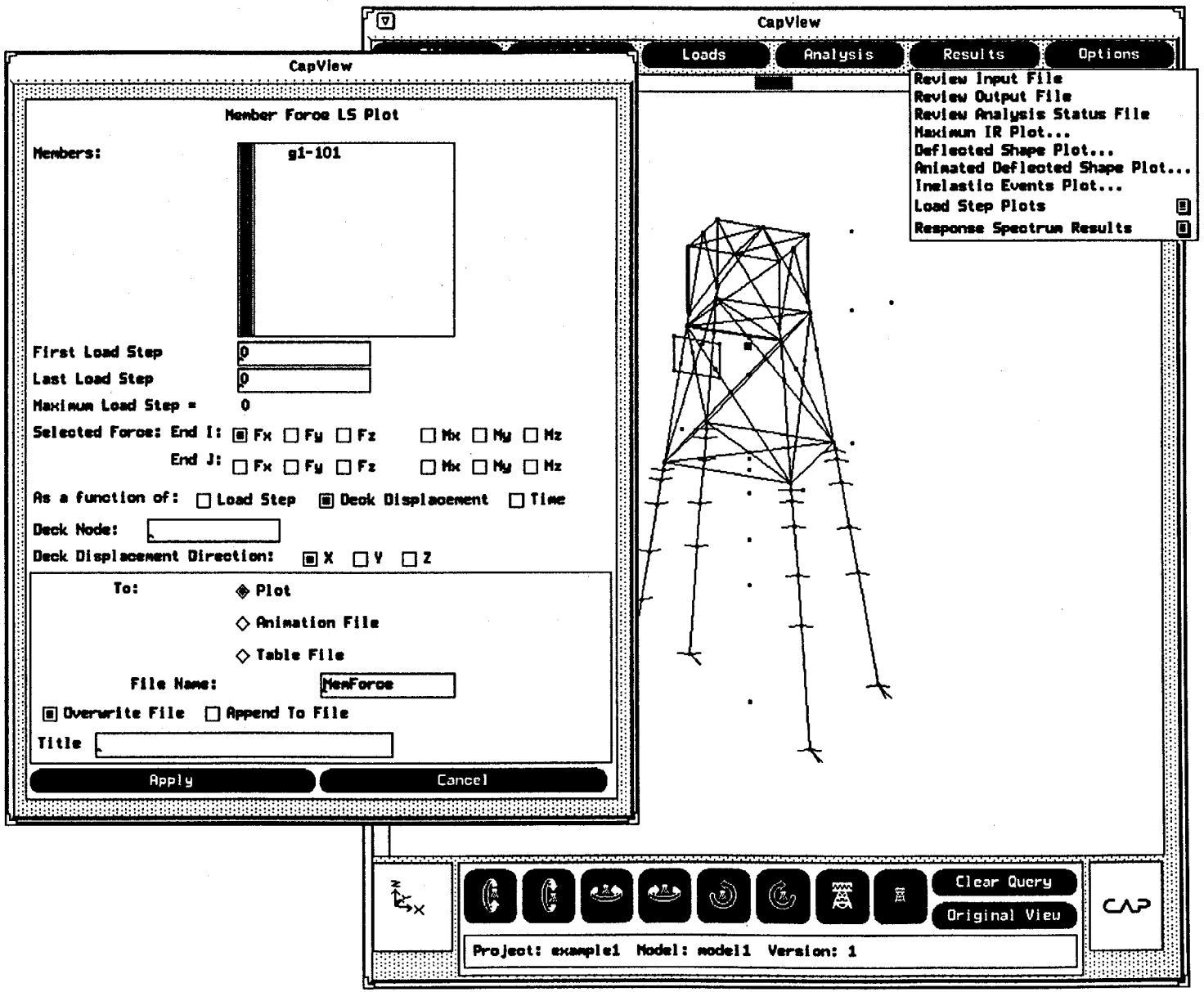

CapView
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Member Force LS Plot
Members:
g1-101
First Load Step
0
Last Load Step
0
Maximum Load Step =
0
Selected Force: End I: Fx Fy Fz Mx My Mz
End J: Fx Fy Fz Mx My Mz
As a function of: Load Step Deck Displacement Time
Deck Node:
Deck Displacement Direction: X Y Z
To:
Plot
Animation File
Table File
File Name:
MemForce
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-14

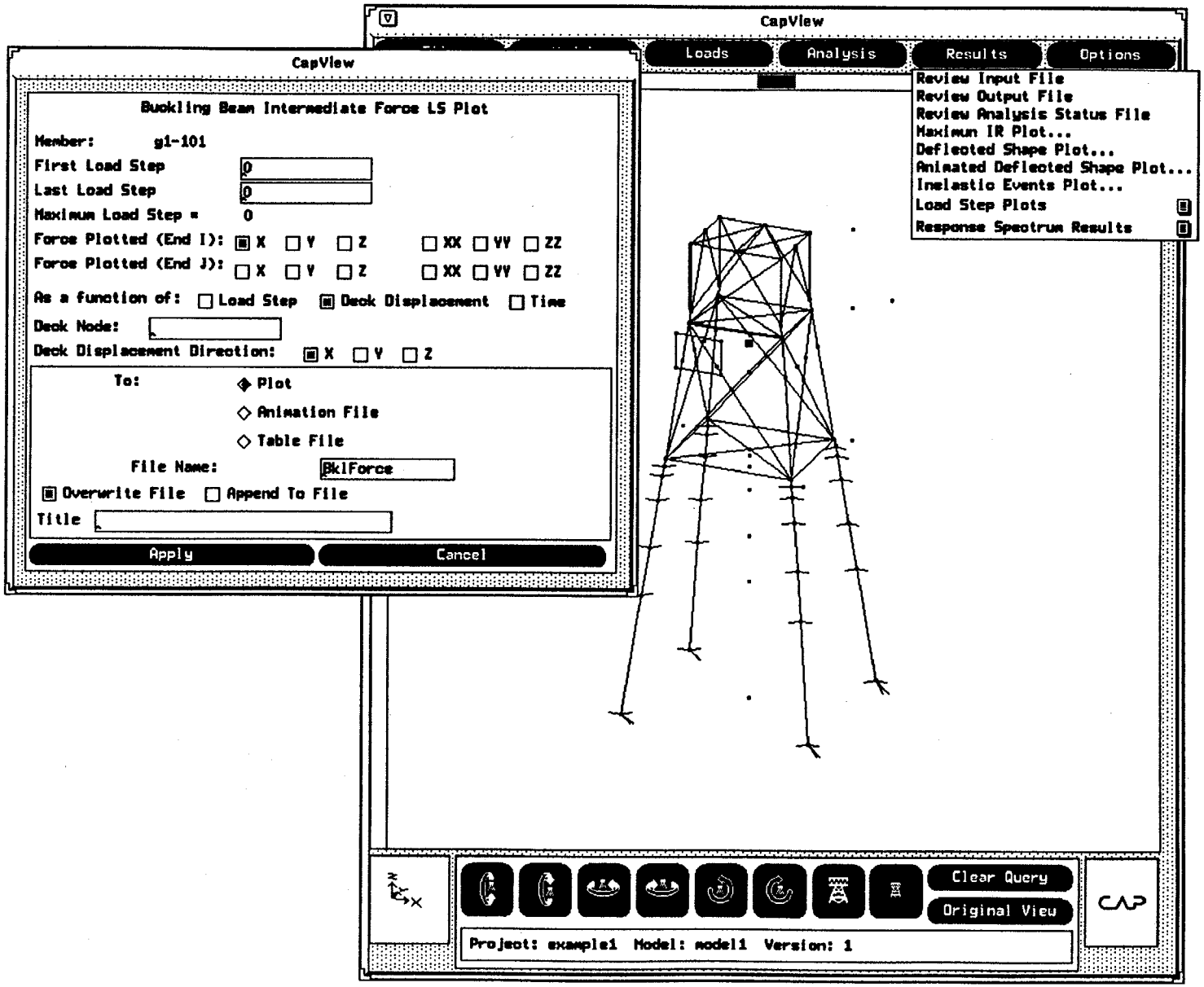

CapView
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Buckling Beam Intermediate Force LS Plot
Member: g1-101
First Load Step 0
Last Load Step 0
Maximum Load Step = 0
Force Plotted (End I): X Y Z XX YY ZZ
Force Plotted (End J): X Y Z XX YY ZZ
As a function of: Load Step Deck Displacement Time
Deck Node:
Deck Displacement Direction: X Y Z
To: Plot
Animation File
Table File
File Name: BklForce
Overwrite File Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-15

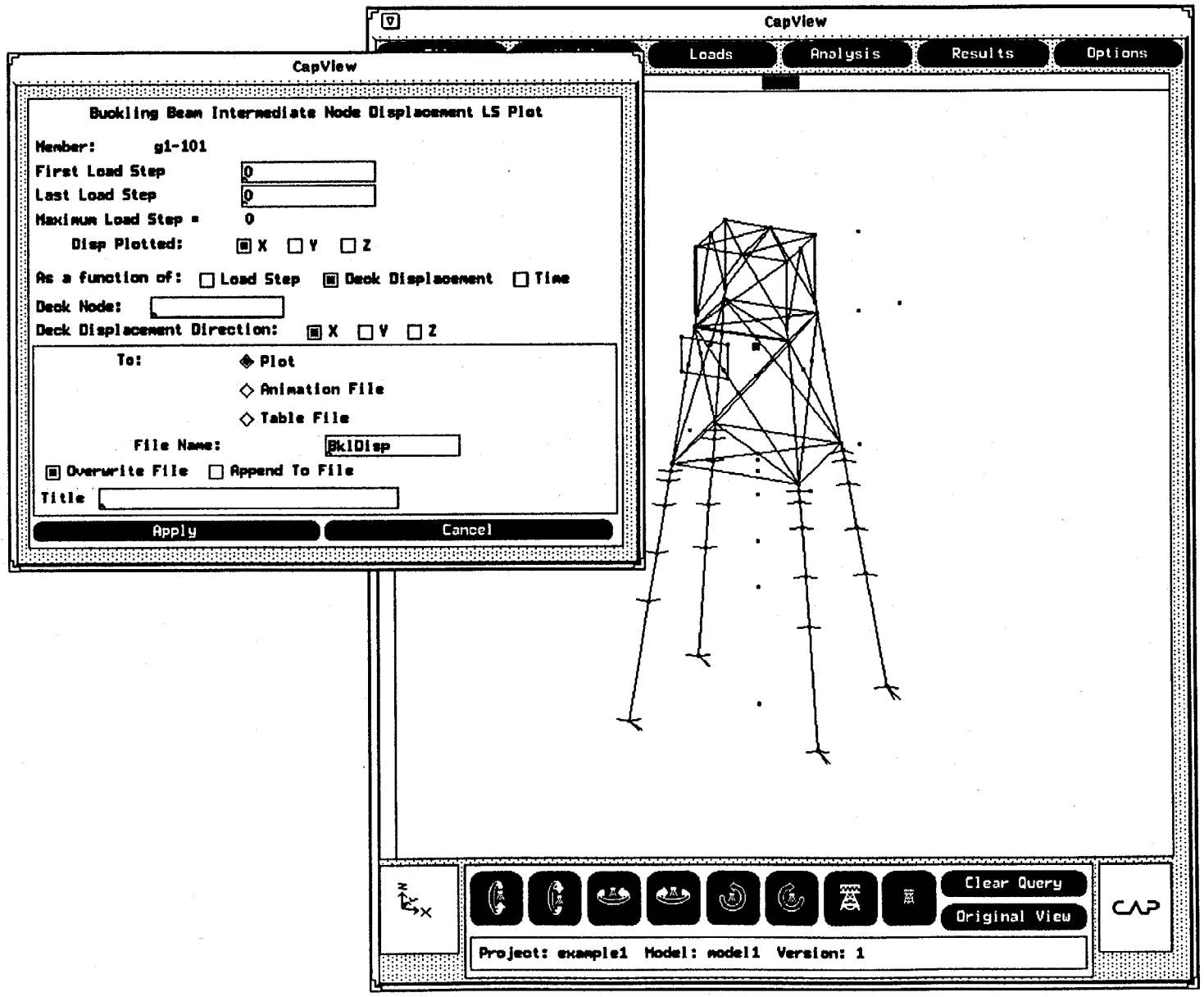

CapView
Loads
Analysis
Results
Options
CapView
Buckling Beam Intermediate Node Displacement LS Plot
Member: g1-101
First Load Step 0
Last Load Step 0
Maximum Load Step = 0
Disp Plotted: X Y Z
As a function of: Load Step Deck Displacement Time
Deck Node:
Deck Displacement Direction: X Y Z
To: Plot
Animation File
Table File
File Name: BklDisp
Overwrite File Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-16

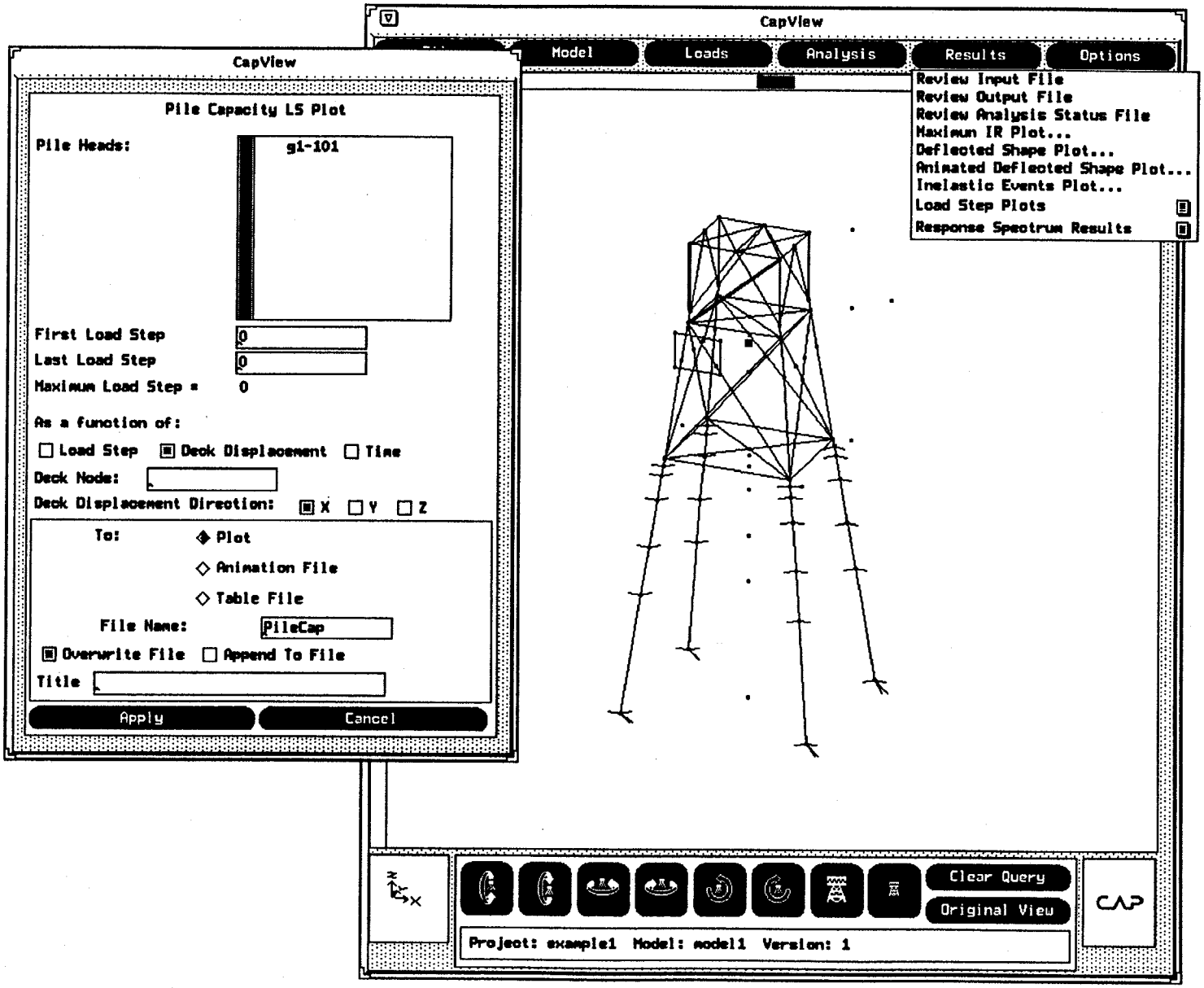

CapView
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Pile Capacity LS Plot
Pile Heads:
g1-101
First Load Step
0
Last Load Step
0
Maximum Load Step =
0
As a function of:
Load Step
Deck Displacement
Time
Deck Node:
Deck Displacement Direction:
X
Y
Z
To:
Plot
Animation File
Table File
File Name:
PileCap
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-17

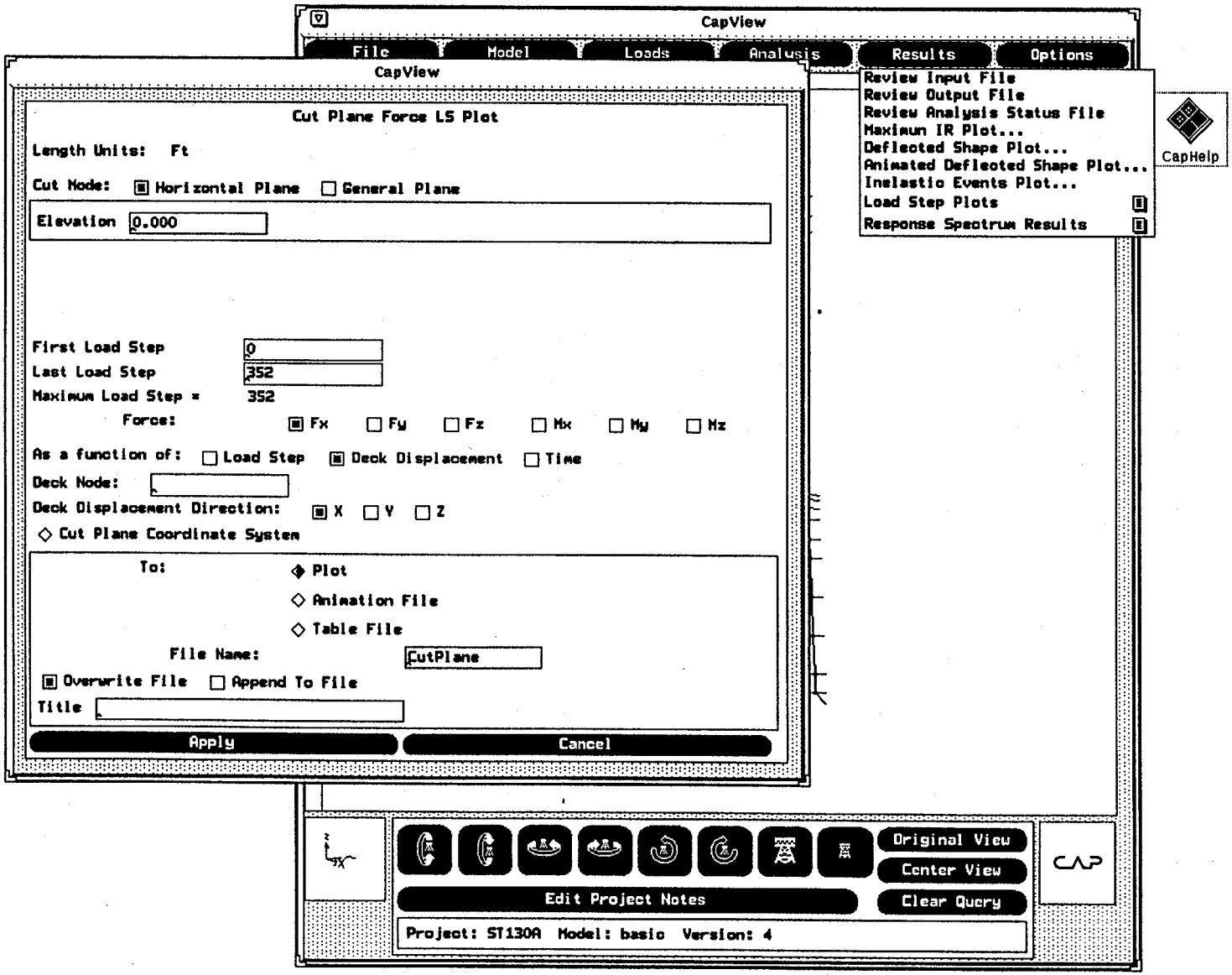

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapHelp
CapView
Cut Plane Force LS Plot
Length Units: Ft
Cut Mode: Horizontal Plane General Plane
Elevation 0.000
First Load Step 0
Last Load Step 352
Maximum Load Step = 352
Force: Fx Fy Fz Mx My Mz
As a function of: Load Step Deck Displacement Time
Deck Node:
Deck Displacement Direction: X Y Z
Cut Plane Coordinate System
To: Plot
Animation File
Table File
File Name: CutPlane
Overwrite File Append To File
Title
Apply
Cancel
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 11-18

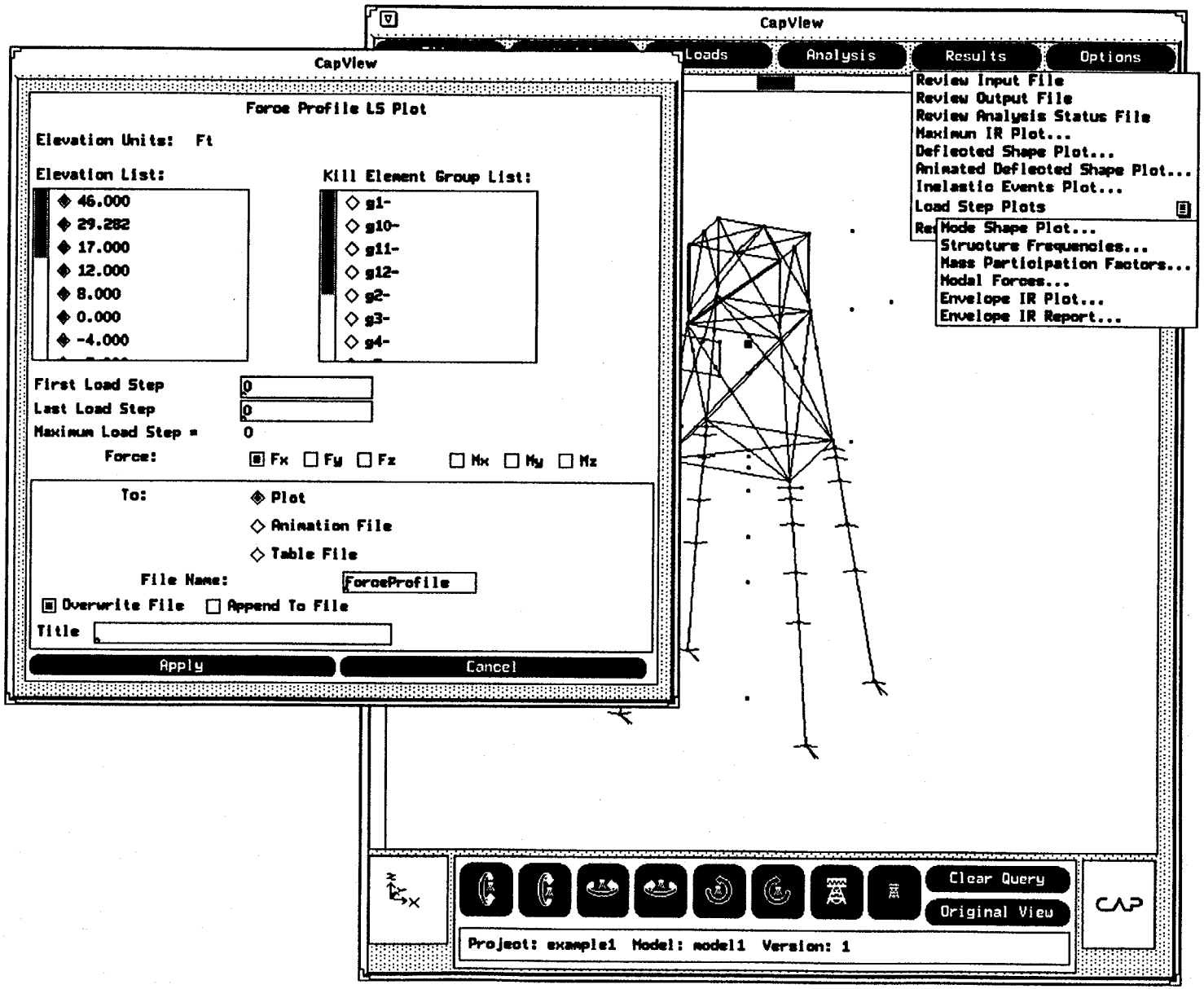

CapView
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Mode Shape Plot...
Structure Frequencies...
Mass Participation Factors...
Modal Forces...
Envelope IR Plot...
Envelope IR Report...
CapView
Force Profile LS Plot
Elevation Units: Ft
Elevation List:
46.000
29.282
17.000
12.000
8.000
0.000
-4.000
Kill Element Group List:
g1-
g10-
g11-
g12-
g2-
g3-
g4-
First Load Step 0
Last Load Step 0
Maximum Load Step = 0
Force: Fx Fy Fz Mx My Mz
To: Plot
Animation File
Table File
File Name: ForceProfile
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-19

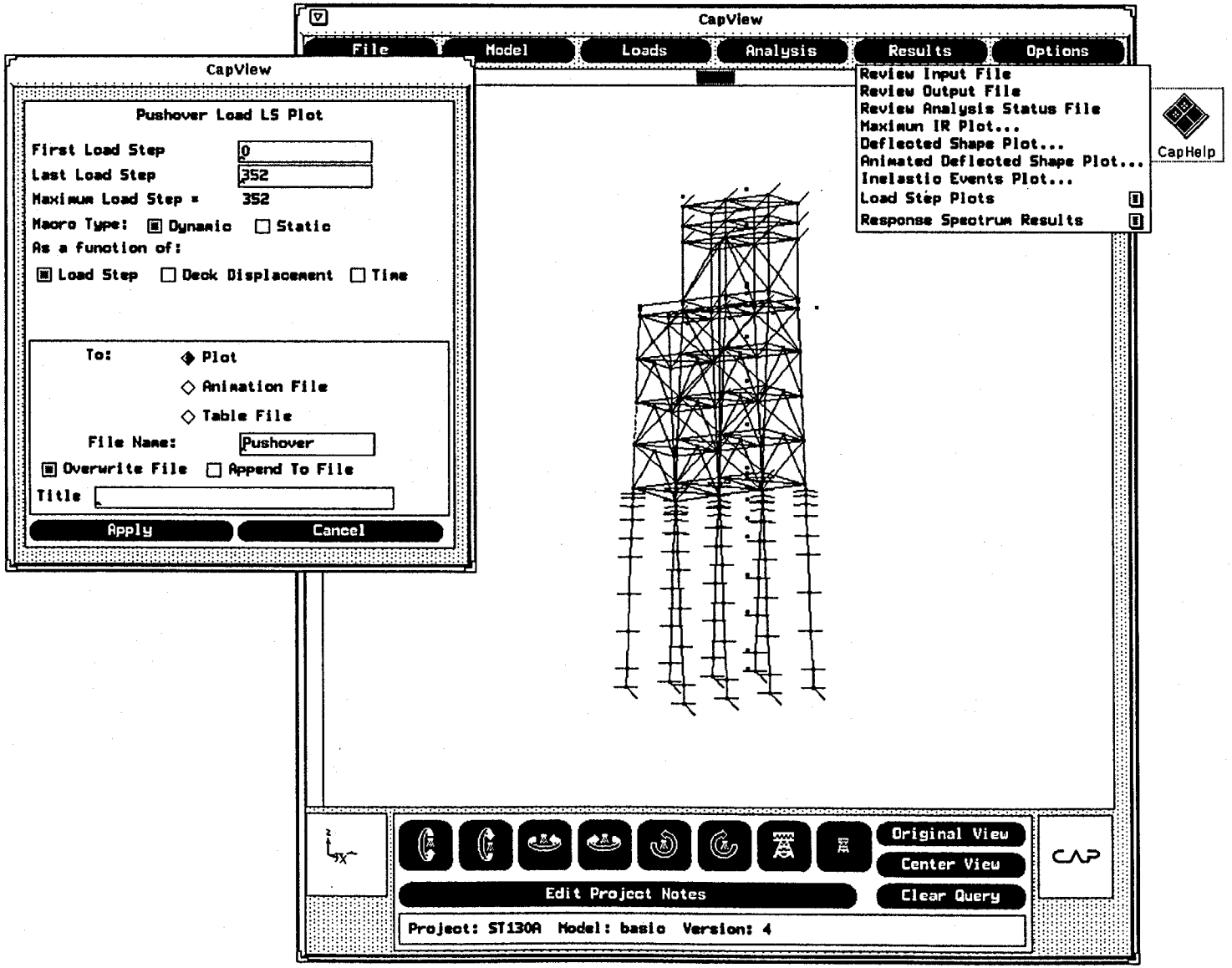

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapHelp
CapView
Pushover Load LS Plot
First Load Step
0
Last Load Step
352
Maximum Load Step =
352
Macro Type:
Dynamic
Static
As a function of:
Load Step
Deck Displacement
Time
To:
Plot
Animation File
Table File
File Name:
Pushover
Overwrite File
Append To File
Title
Apply
Cancel
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 11-20

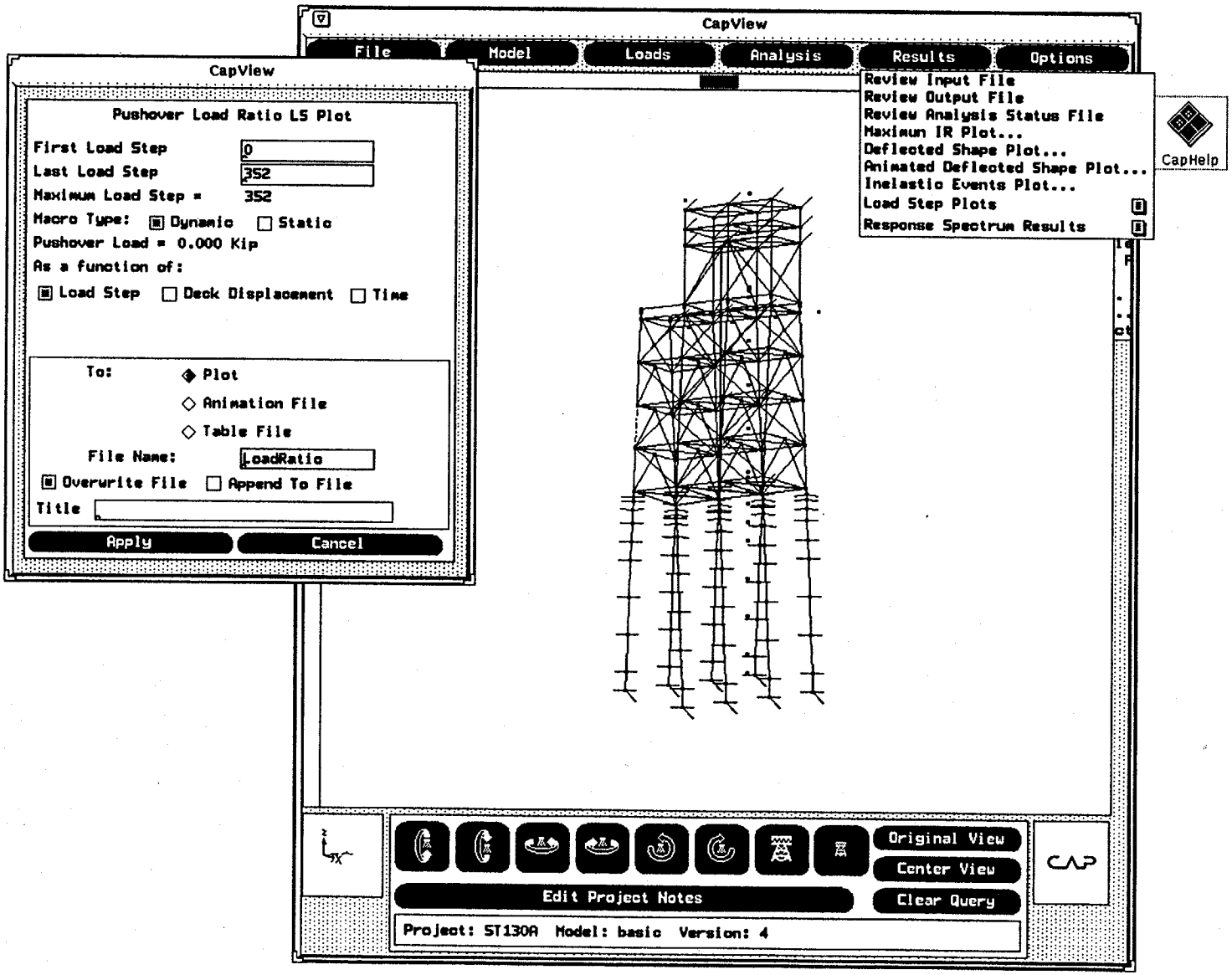

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximun IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapHelp
CapView
Pushover Load Ratio LS Plot
First Load Step 0
Last Load Step 352
Maximum Load Step = 352
Macro Type: Dynamic Static
Pushover Load = 0.000 Kip
As a function of:
Load Step Deck Displacement Time
To: Plot
Animation File
Table File
File Name: LoadRatio
Overwrite File Append To File
Title
Apply
Cancel
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 11-21

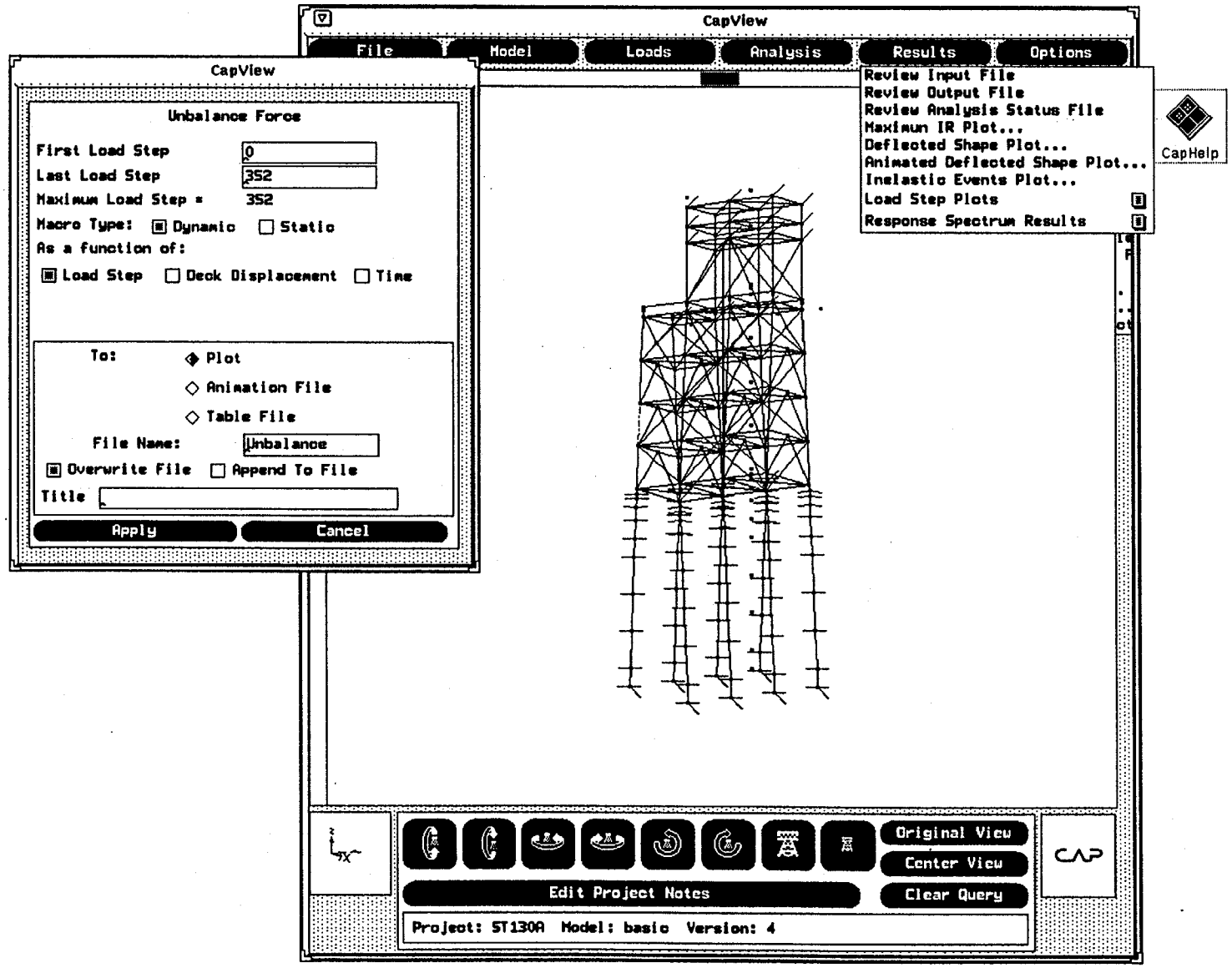

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapHelp
CapView
Unbalance Force
First Load Step
0
Last Load Step
352
Maximum Load Step =
352
Macro Type:
Dynamic
Static
As a function of:
Load Step
Deck Displacement
Time
To:
Plot
Animation File
Table File
File Name:
Unbalance
Overwrite File
Append To File
Title
Apply
Cancel
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 11-22

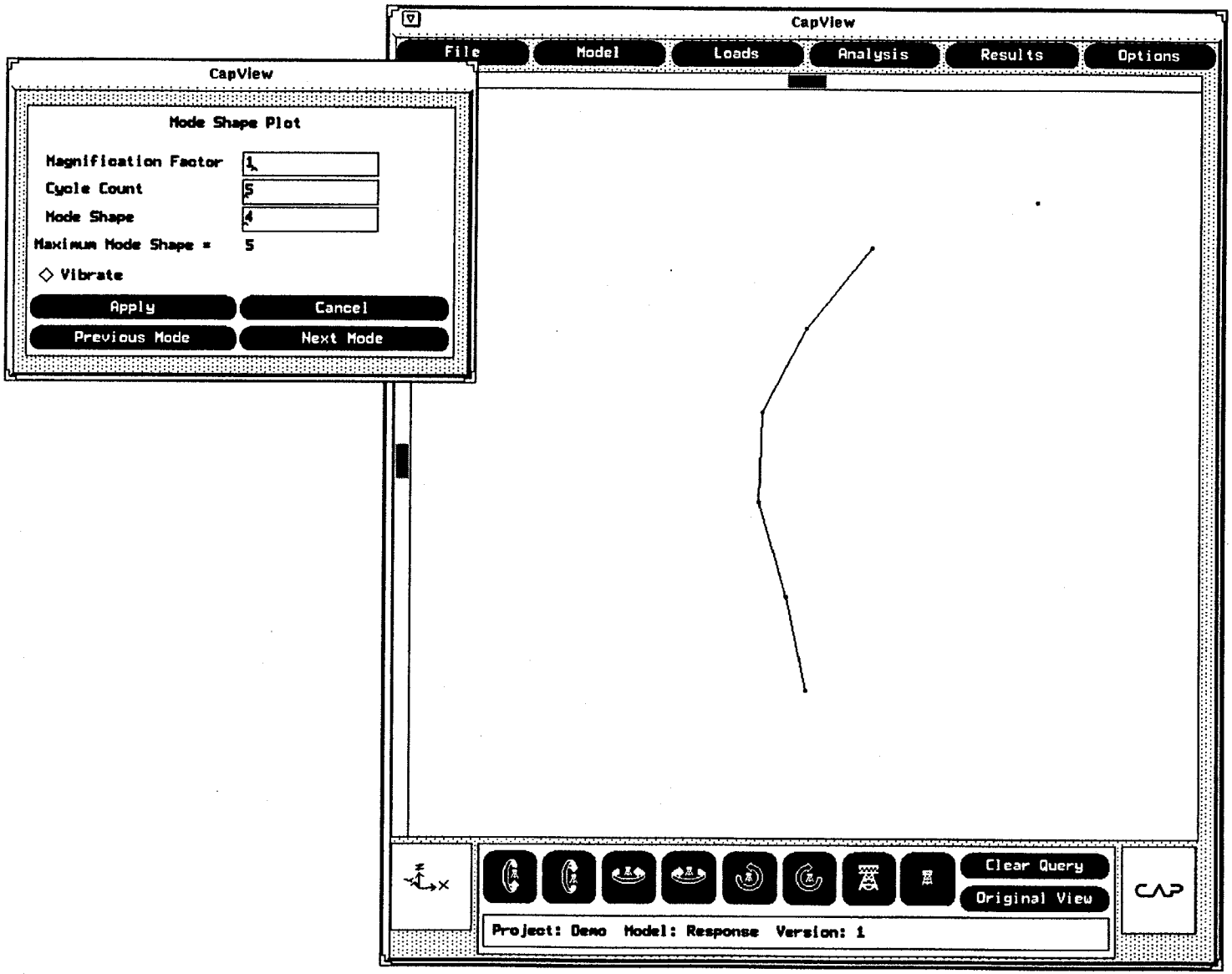

CapView
Mode Shape Plot
Magnification Factor
1
Cycle Count
5
Mode Shape
4
Maximum Mode Shape =
5
Vibrate
Apply
Cancel
Previous Mode
Next Mode
CapView
File
Model
Loads
Analysis
Results
Options
Clear Query
Original View
Project: Demo Model: Response Version: 1

Figure 11-23

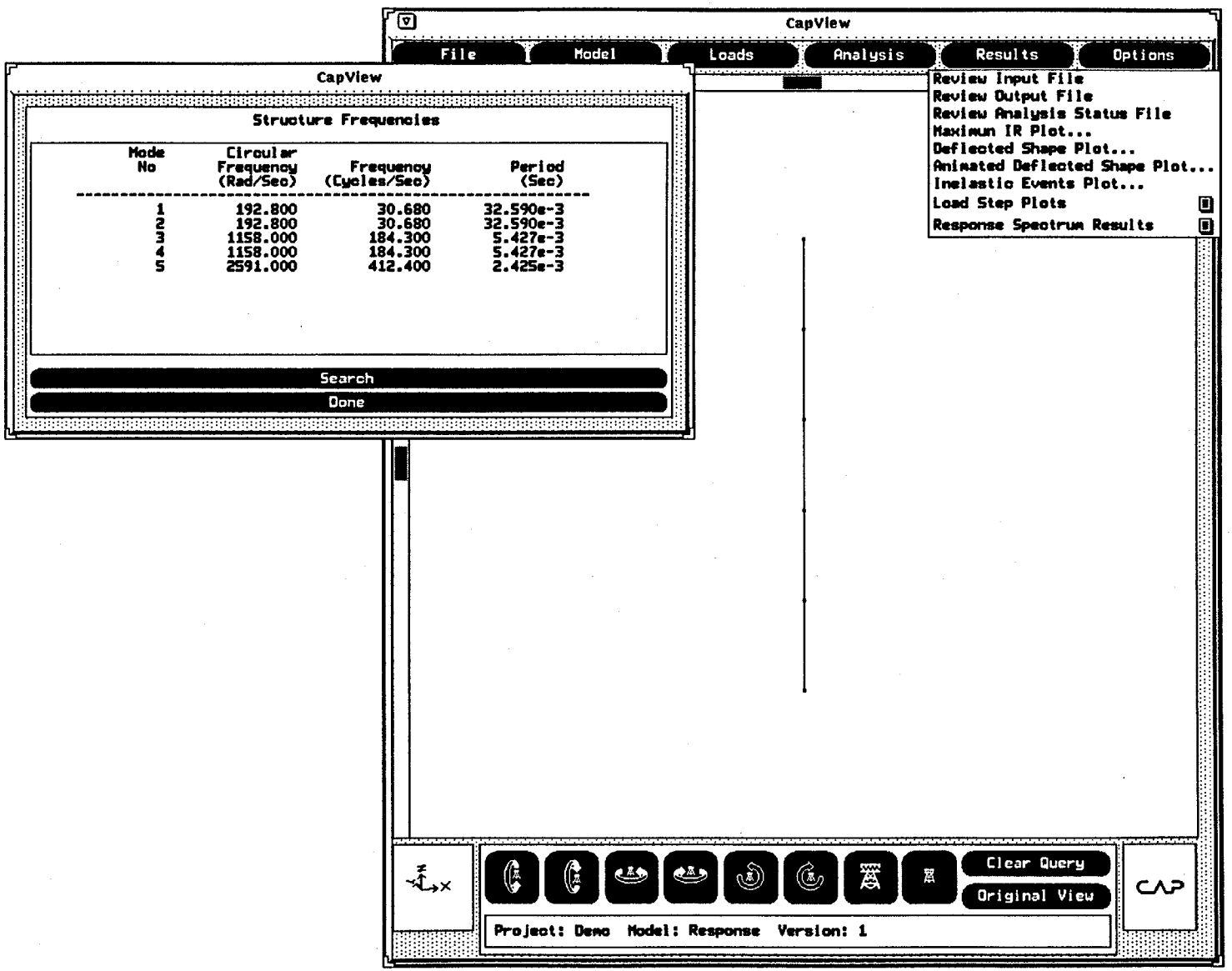

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximun IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Structure Frequencies
Mode No | Circular Frequency (Rad/Sec) | Frequency (Cycles/Sec) | Period (Sec)
1 | 192.800 | 30.680 | 32.590e-3
2 | 192.800 | 30.680 | 32.590e-3
3 | 1158.000 | 184.300 | 5.427e-3
4 | 1158.000 | 184.300 | 5.427e-3
5 | 2591.000 | 412.400 | 2.425e-3
Search
Done
Clear Query
Original View
Project: Demo Model: Response Version: 1

Figure 11-24

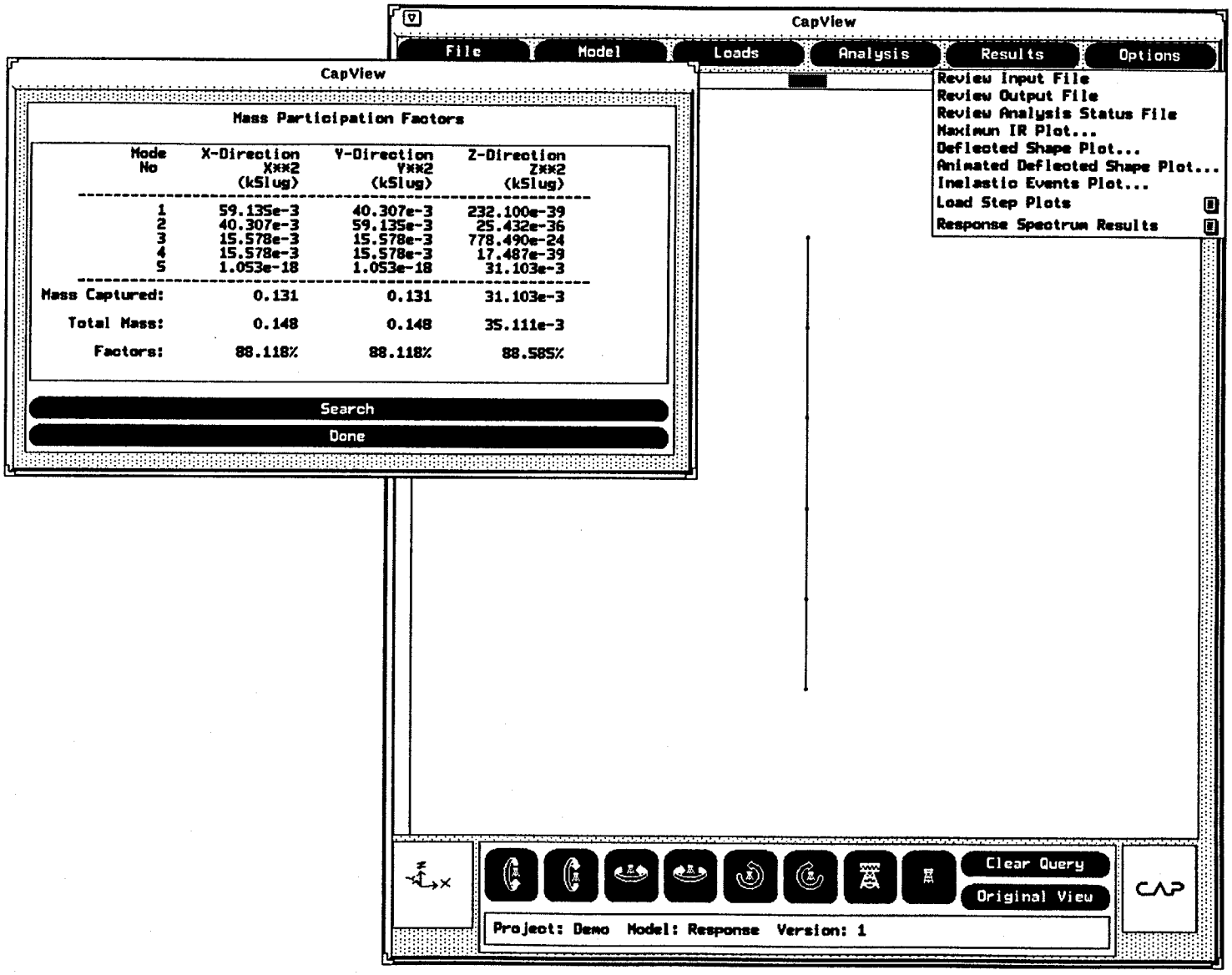

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Mass Participation Factors
Mode No | X-Direction X**2 (kSlug) | Y-Direction Y**2 (kSlug) | Z-Direction Z**2 (kSlug)
1 | 59.135e-3 | 40.307e-3 | 232.100e-39
2 | 40.307e-3 | 59.135e-3 | 25.432e-36
3 | 15.578e-3 | 15.578e-3 | 778.490e-24
4 | 15.578e-3 | 15.578e-3 | 17.487e-39
5 | 1.053e-18 | 1.053e-18 | 31.103e-3
Mass Captured: | 0.131 | 0.131 | 31.103e-3
Total Mass: | 0.148 | 0.148 | 35.111e-3
Factors: | 88.118% | 88.118% | 88.585%
Search
Done
Clear Query
Original View
Project: Demo Model: Response Version: 1

Figure 11-25

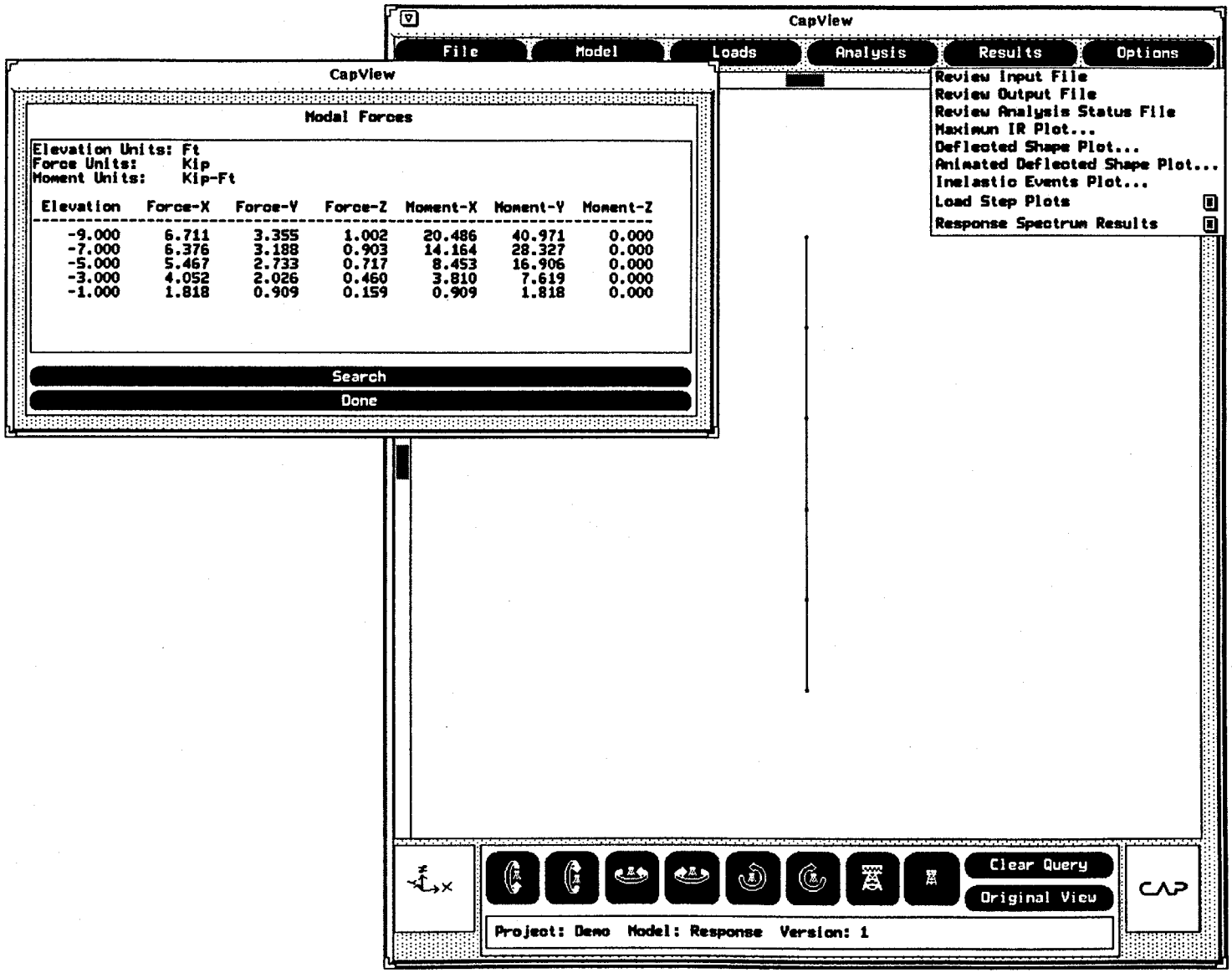

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximun IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Modal Forces
Elevation Units: Ft
Force Units: Kip
Moment Units: Kip-Ft
Elevation Force-X Force-Y Force-Z Moment-X Moment-Y Moment-Z
-9.000 6.711 3.355 1.002 20.486 40.971 0.000
-7.000 6.376 3.188 0.903 14.164 28.327 0.000
-5.000 5.467 2.733 0.717 8.453 16.906 0.000
-3.000 4.052 2.026 0.460 3.810 7.619 0.000
-1.000 1.818 0.909 0.159 0.909 1.818 0.000
Search
Done
Clear Query
Original View
Project: Demo Model: Response Version: 1

Figure 11-26

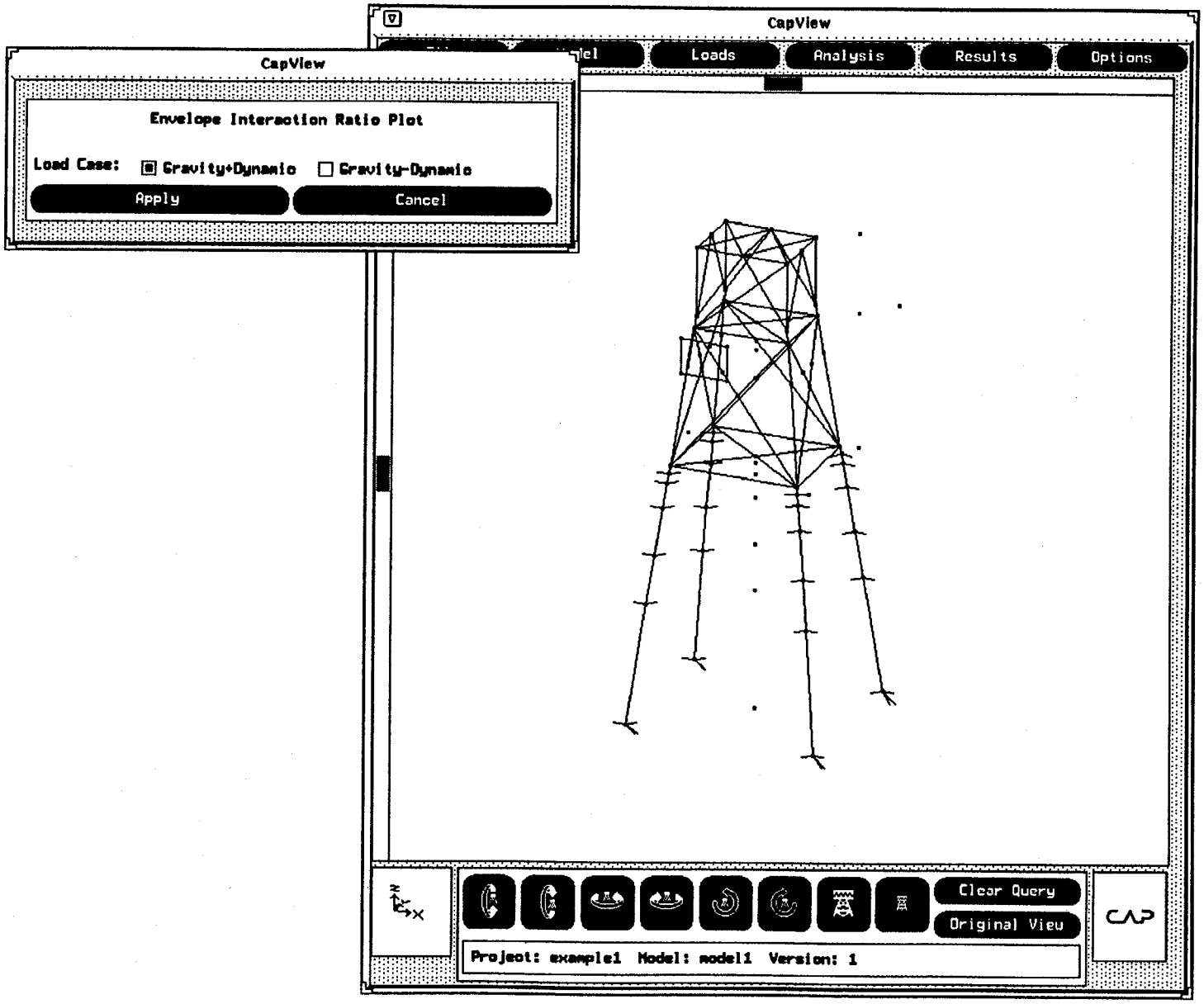

CapView
Loads
Analysis
Results
Options
CapView
Envelope Interaction Ratio Plot
Load Case:
Gravity+Dynamic
Gravity-Dynamic
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-27

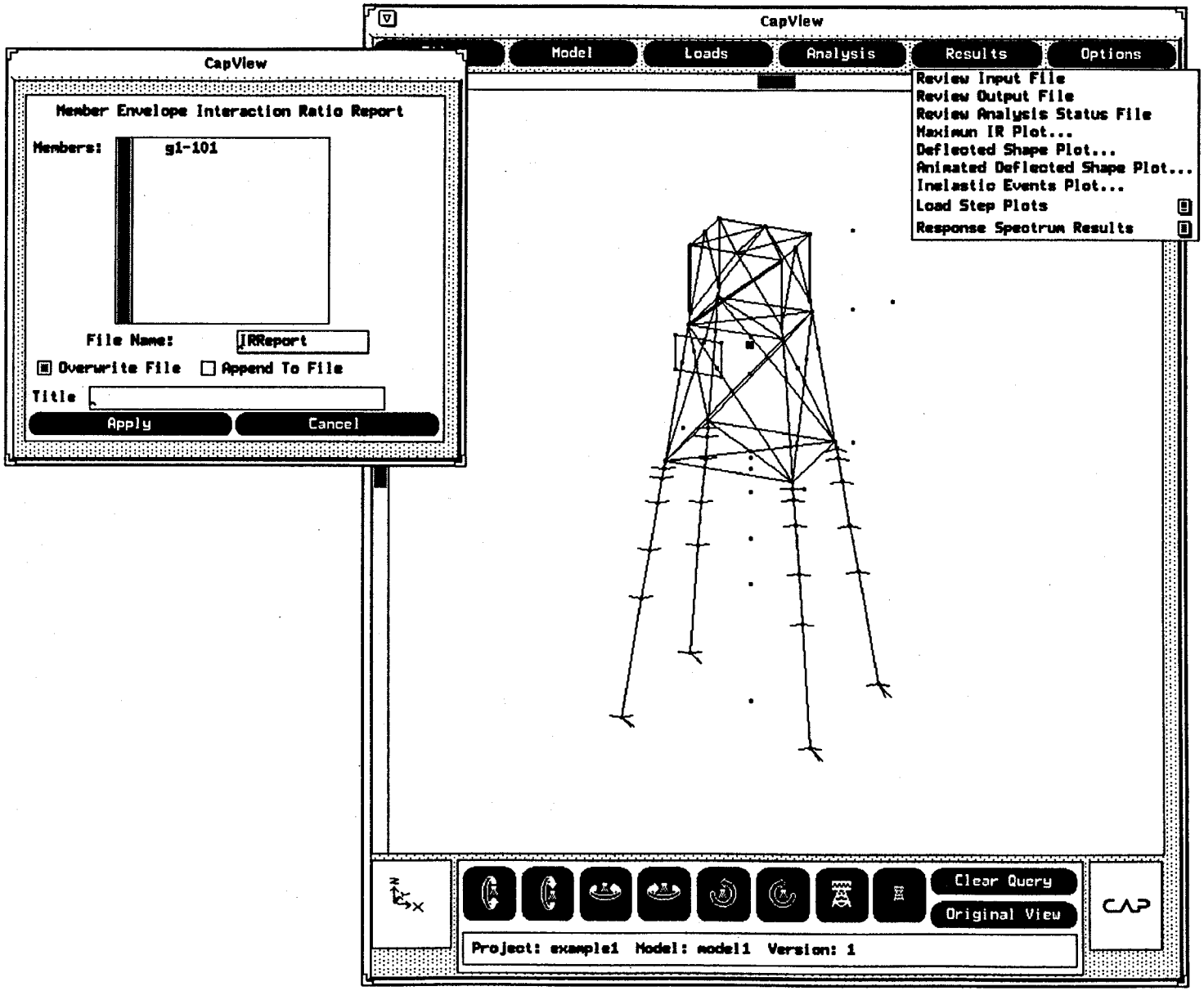

CapView
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Member Envelope Interaction Ratio Report
Members:
g1-101
File Name:
IRReport
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: example1 Model: model1 Version: 1

Figure 11-28

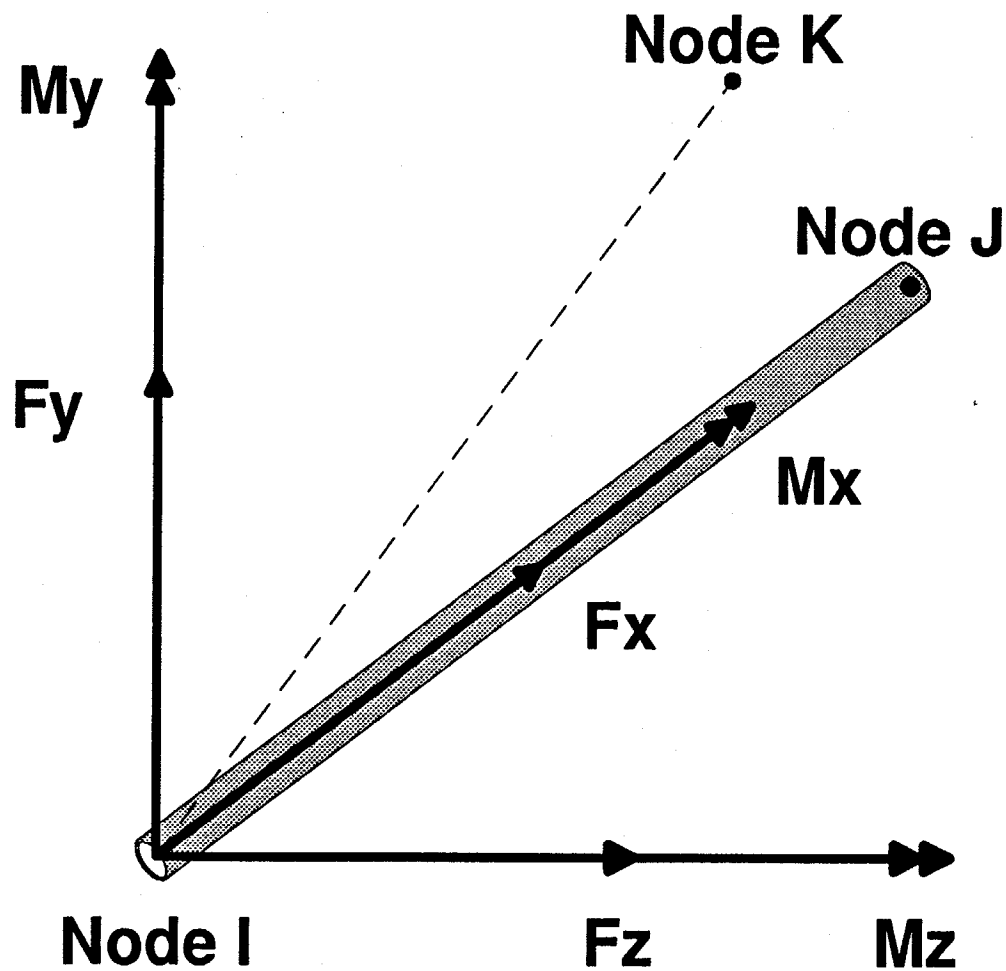


Figure 11-29

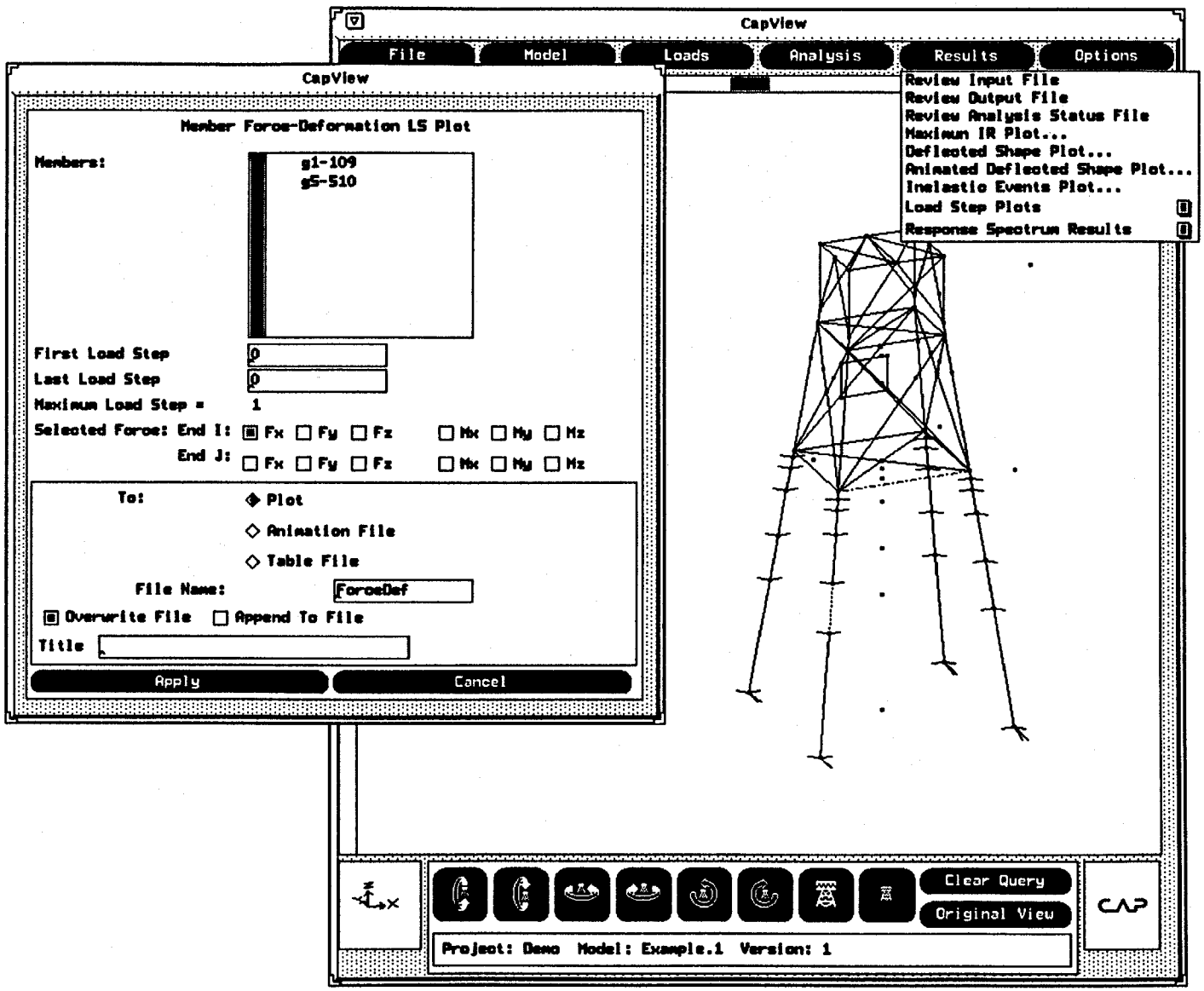

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Member Force-Deformation LS Plot
Members:
g1-109
gS-510
First Load Step
0
Last Load Step
0
Maximum Load Step =
1
Selected Force: End I:
Fx Fy Fz Mx My Mz
End J:
Fx Fy Fz Mx My Mz
To:
Plot
Animation File
Table File
File Name:
ForceDef
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 11-30

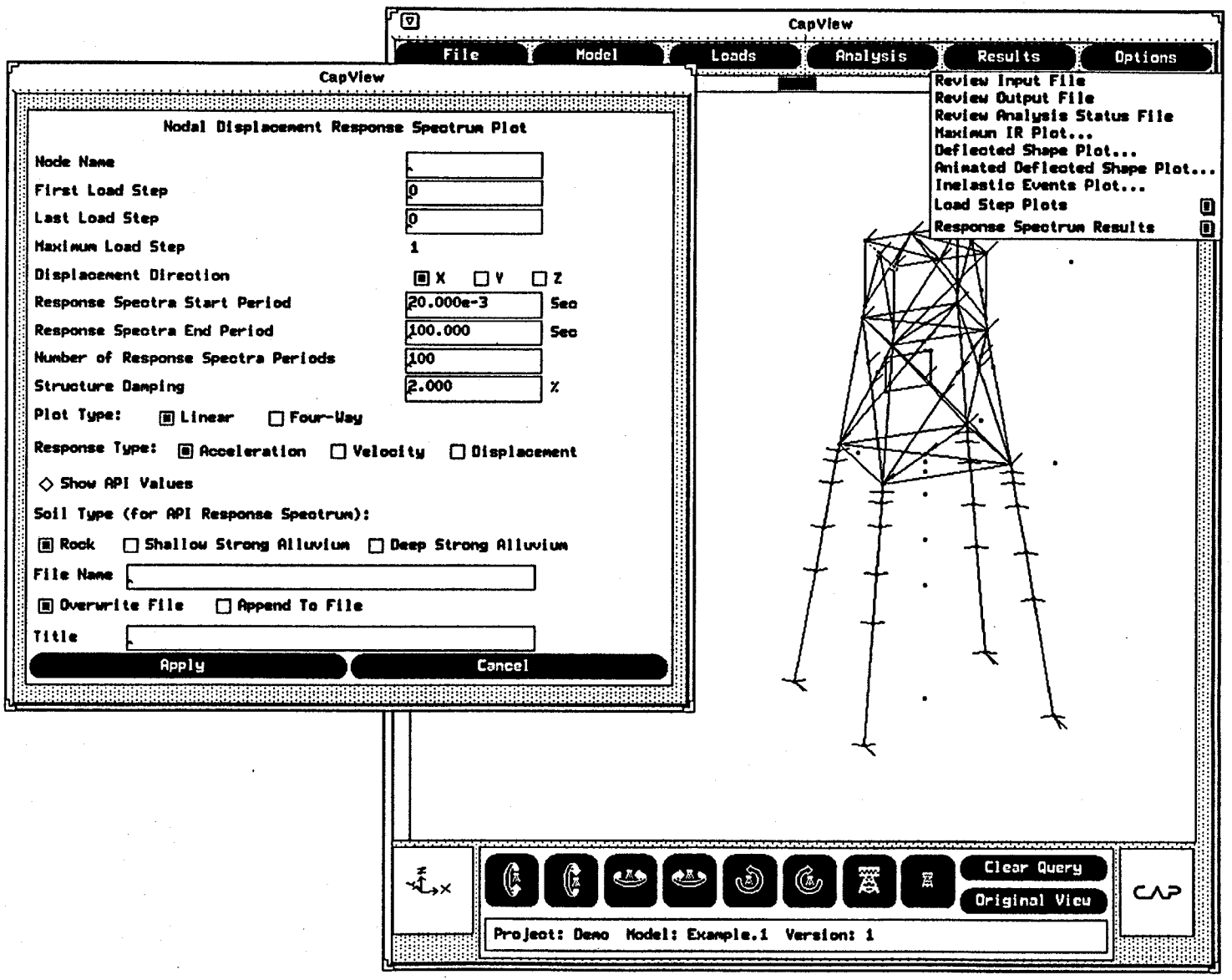

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximum IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapView
Nodal Displacement Response Spectrum Plot
Node Name
First Load Step
0
Last Load Step
0
Maximum Load Step
1
Displacement Direction
X
Y
Z
Response Spectra Start Period
20.000e-3
Sec
Response Spectra End Period
100.000
Sec
Number of Response Spectra Periods
100
Structure Damping
2.000
%
Plot Type:
Linear
Four-Way
Response Type:
Acceleration
Velocity
Displacement
Show API Values
Soil Type (for API Response Spectrum):
Rock
Shallow Strong Alluvium
Deep Strong Alluvium
File Name
Overwrite File
Append To File
Title
Apply
Cancel
Clear Query
Original View
Project: Demo Model: Example.1 Version: 1

Figure 11-31

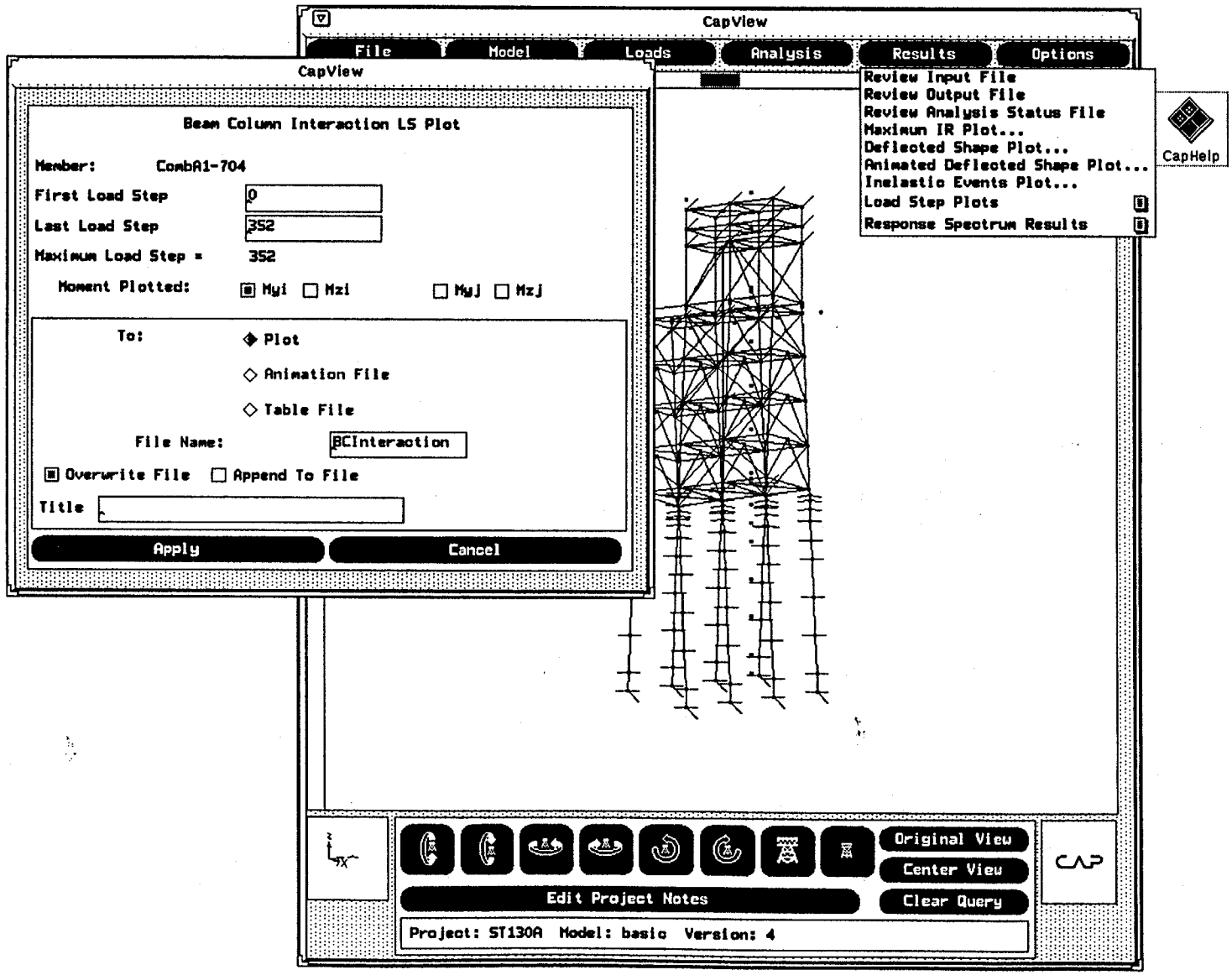

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximun IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapHelp
CapView
Beam Column Interaction LS Plot
Member: CombA1-704
First Load Step 0
Last Load Step 352
Maximum Load Step = 352
Moment Plotted: Myi Mzi Myj Mzj
To: Plot
Animation File
Table File
File Name: BCInteraction
Overwrite File Append To File
Title
Apply
Cancel
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 11-32

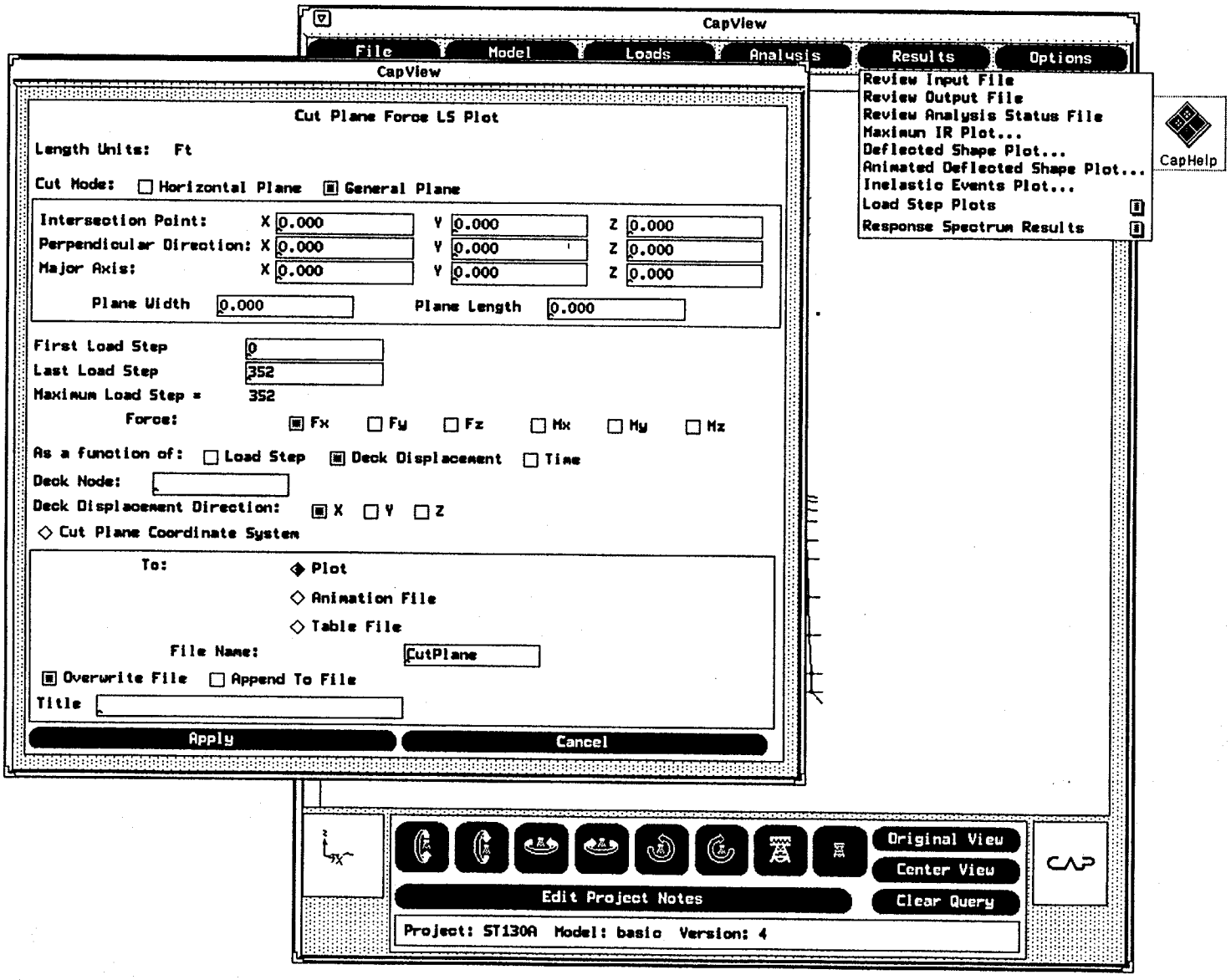

CapView
File
Model
Loads
Analysis
Results
Options
Review Input File
Review Output File
Review Analysis Status File
Maximun IR Plot...
Deflected Shape Plot...
Animated Deflected Shape Plot...
Inelastic Events Plot...
Load Step Plots
Response Spectrum Results
CapHelp
CapView
Cut Plane Force LS Plot
Length Units: Ft
Cut Mode: Horizontal Plane General Plane
Intersection Point: X 0.000 Y 0.000 Z 0.000
Perpendicular Direction: X 0.000 Y 0.000 Z 0.000
Major Axis: X 0.000 Y 0.000 Z 0.000
Plane Width 0.000
Plane Length 0.000
First Load Step 0
Last Load Step 352
Maximum Load Step = 352
Force: Fx Fy Fz Mx My Mz
As a function of: Load Step Deck Displacement Time
Deck Node:
Deck Displacement Direction: X Y Z
Cut Plane Coordinate System
To: Plot
Animation File
Table File
File Name: CutPlane
Overwrite File Append To File
Title
Apply
Cancel
Original View
Center View
Clear Query
Edit Project Notes
Project: ST130A Model: basic Version: 4

Figure 11-33

12.0 HOW TO...?

This section explains the basic steps involved in performing a push-over load analysis on an offshore platform. The first subsection describes the basic steps. The second subsection deals with some advanced topics.

12.1 Basic Steps

To find the ultimate capacity of an offshore platform you have to (1) create the stiffness model of the platform, (2) specify the load conditions, (3) run an over-load analysis, and (4) examine the analysis results to find out the ultimate capacity load, plot deformed shape of the platform, check member failures, etc. In the second step, to specify the push-over load pattern, the user must run an analysis to determine the most critical combination of a user defined wave and/or current loading condition.

12.1.1 Creating the Stiffness Model

When you start a new project (version), you have two ways to build the model. You either quickly generate a platform skeleton then (manually) add additional nodes and members, or you jump to (manually) adding nodes and members.

The Generate Platform Skeleton selection from the Model menu (section 8.1), is ideal if you are dealing with stick models, tripods and regular rectangular platforms. Regular rectangular platforms satisfy the following condition. Each side of the platform has no batter, or a single batter starting from the working point and extending all the way down the legs. If the piles are inclosed inside the legs, then the pile extensions beyond the legs will have the same batter as the legs. If the platform you are modeling does not satisfy this condition, you might be better off adding nodes and members manually. The decision on when to use the Generate Platform Skeleton feature depends on the platform you are modeling.

The Generate Platform Skeleton selection allows you to define the geometry of the platform, the section properties of the legs and the horizontal braces, and the soil profile if any. Remember that all of this can be done manually and that the Generate Platform Skeleton feature is provided for the convenience of the user. Also, keep in mind that in most of the cases after you have generated a skeleton, you still have to add/delete members and nodes manually. More details on this can be found in section 8.1.

The Add Members/Nodes selection from the Model menu allows the user to add members and nodes in different configuration, split members, connect members, and add soil spring (PSAS) elements. More details on this can be found in section 8.2.

The Soil Profile selection from the Model menu allows the user to define a soil profile. The user can also review and/or update a profile that was created using this selection or using the Generate Platform Skeleton selection. More details on this can be found in section 8.4.

The Water Data selection from the Model menu allows the user to review and/or update certain water parameters. This is a must if you are not using the Generate Platform Skeleton selection. More details on this can be found in section 8.5.

The Default Analysis Parameters and Default Code Check Parameters selections from the Model menu allow the user to set global values for certain analysis and code check parameters. More details on this can be found in sections 8.6 and 8.7.

12.1.2 Specifying Loads

In addition to the gravity and buoyancy loads of the members, the user can define nodal loads and wave and current loads. The nodal loads are static concentrated forces and mements. These can be used to model the gravity loads of nonstructural components of the platform. Nonstructural components are members which are of no significant structural stiffness (e.g. flooring system, drilling or production equipment, etc...). Note that if a nonstructural member is subjected to wave or current loads, then it must be modeled by a member of the Wave Load Element type.

The Add Nodal Load selection from the Load menu allows the user to add nodal loads. More details on this can be found in section 9.1

The Define Current and Define Wave selections from the Load menu allow the user to define a current velocity profile and a wave. More details on this can be found in sections 9.1 and 9.2.

To generate a push-over load pattern, the user must specify a wave and/or a current load. Then the user must run a special analysis that will find the most critical distribution of wave and/or current loads on the platform. Mode details can be found in the next subsection.

12.1.3 Running an Analysis

Before you can run an analysis you must first select an analysis macro and the loads to be applied (from the set of defined loads). Also, it is a good idea to run a few data checks to make sure the model is defined properly.

The Analysis Options selection from the Analysis menu allows the user to select an analysis macro, customize certain parameters, and select the applied loads. The user can select from a list of macros that perform static analysis, push-over analysis, and an analysis to create a push-over load pattern. More details on this can be found in section 10.1.

The Perform Data Check selection from the Analysis menu runs a series of data checks on the model and reports problems (if any) to the user. Note that the same data checks are performed after selecting analysis options. More details on this can be found in section 10.2.

The Run SeaStar Analysis selection from the Analysis menu creates a SeaStar input file and perform an analysis. The Create SeaStar File selection from the Analysis menu creates a SeaStar input file so the user may edit it. More details on this can be found in sections 10.3 and 10.4.

12.1.4 Examining Results

After running an analysis the user may browse through the output file, display the deflected shape, inelastic member events, and/or member IRs at any (saved) load step. The user may also create X-Y plots of the nodal displacements, member forces, member IRs, push-over load and unbalance to list a few. More details on this can be found in sections 11.0 through 11.14.

12.2 Advanced Topics

In the previous subsections, various basic steps of the push-over analysis process have been described. In this section, some advanced steps are presented.

12.2.1 Adding Skirt Piles.

The Generate Platform Skeleton selection does not generate skirt piles. Fortunately, skirt pile can be modeled. However, there is no simple way of

modeling them. To add skirt piles to your model, you must follow these steps:

1- Make sure that you have modeled the soil profile. If not, then you have to model it. To do that, use the Soil Profile selection from the Model menu (section 8.4).

2- Add a line of nodes for each skirt pile. Use the Add Members/Nodes selection of the Model menu to generate a line of nodes for each pile (section 8.2).

3- Add the pile elements. Using the Add Members/Nodes selection of the Model menu, add single elements connecting the nodes of each pile (section 8.2).

4- Add soil spring (PSAS) elements to the pile nodes which are below the mud line (section 8.2).

5- Model the connection between skirt piles and the rest of the platform. This depends on the nature of the connection. The Add Members/Nodes selection of the Model menu can be used for this (section 8.2).

12.2.2 Changing Element Type of Members and Groups

Each member in a model has an element type. Every member in a model belongs to a group. For each group, all the members in the group have the same element type. To change the element type of a member, you must also change the group to which the member belongs. The new group must have members with the new element type or is a brand new group.

Also, to change the group affiliation of a member, the new group must have the same element type or you have to change the element type of the member, too. The Select/Change Members selection of the Model menu, will not perform a change that involves group affiliation, or element type unless both group affiliation and element type are changed.

Therefore, to change the element type of all members in a group, you must also change change the group affiliation to a new group. Keep in mind that a group with no members is ignored during the analysis.

12.2.3 Editing Pile Members, Nodes and Shims

When working on a model with ungrouted piles, the model will have pile members and leg members which occupy the same space. (The model will also have pile nodes and leg nodes which are at the same location.) When you highlight objects occupying the same space, one and only one object is highlighted. By default, the leg member (node) is highlighted.

To edit the pile members, you must first hide the leg members. The Select/Change Member selection from the Model menu (section 8.8.9) can be used to select the leg member, then change their Member Visibility to Hidden. Once a leg member is hidden, and you attempt to highlight a member in its space, the pile member will be highlighted. Note that at the moment you hide the leg member, if the member is highlighted, it will retain that status until you highlight a new object. It will also continue to be query selected until you cancel query selection by either running a query that clear previous selections or select the Cancel Query button. To change the Member Visibility of a hidden member (to shown), you must some run a query to query select the member. Then you can change its Member Visibility. Note that although a member is not shown, it can still be query selected and changed. Therefore you must exercise great care when you change member visibility.

Unfortunately, you can not change the visibility of a node. Therefore, in order to edit a pile node, you must change the location of the leg node hiding it. To change the node location, highlight the node and bring its information dialogue box. Change the node location to a new one and press the Replace button followed by the Done button. After you have done this, you will see the leg node has been moved in the display window. Note that the leg node is connected to at least three members. Two are the leg members. The third member connects the leg node to the pile node. This latter member is the shim member. Now you can edit the pile node and the shim element. Remember to change the leg node back to its original location after you have finished editing the pile node. To edit a shim element, move the leg node to expose the shim element.

APPENDIX A: HARDWARE AND SOFTWARE REQUIREMENTS

CAP runs on SPARC compatible workstations equiped with a SPARC-based IU, and floating-point hardware (FPU) compatible with the older or newer Sun-4 FPU's. The minimum memory requirements are 12 MB RAM for monochrome systems and 16 MB RAM for color systems. Systems equiped with 24 MB RAM are recommended. The system must be configured with a minimum of 100 MB of swap space. If you are going to run other applications concurrently with CAP, then memory and swap space requirements will be higher.

For your data files, it is recommended that you have 200 MB of free disk space. It is not uncommon for a overload analysis to generate up to 80 MB of data or more. Although CAP works on a monochrome system, a color system is highly recommended.

For loading the software, you must have one of the following: 1/4" QIC-24 compatible tape drive, 1/4" QIC-150 compatible tape drive, 1/2" tape drive, or 3-1/2" floppy disk drive.

Your workstation must be running SunOS 4.1 (or later) compatible operating system (e.g. Solaris 1.0, and SPARC OS 1.1) and an X Window System, version 11 release 4 (X11R4) compatible windowing system (e.g. OpenWindows 2.0 or later).

APPENDIX B: SOFTWARE INSTALLATION

1- Insert the 1/4-inch distribution tape in the tape drive.

2- Log on the target machine as "root".

3- Select an install directory under which to install CAP. This directory should be on a disk device that has more than 25 MBytes of free storage. It is recommended that you use the same install directory you used for previous CAP prototypes (overwrite old prototypes).

4- Change the current working directory to the install directory selected in step 3. This can be done by typing the following command at the unix shell prompt:

   cd *install_directory*

5- Load the software from the tape. This can be done by typing the following command at the unix shell prompt:

   tar xvf /dev/rst0

6- Configure the working environment of every CAP user by doing the following:

   a- Making sure that the install directory (selected in step 3) is part of his/her search path.

   b- Run the ConfigUserEnvironment shell script. This can be done by typing the following command at the unix shell prompt:

      ConfigUserEnvironment

      This shell script will ask for the user id and the CAP install directory.

   c- If you selected not to let ConfigUserEnvironment modify the user's .Xdefaults, then you have to do it manually. Modify the user's .Xdefaults file to include the contents of the CapView.Xdefaults or CapView.Xdefaults.mc file.

APPENDIX C: CAP FILES

Initialization Files in ~/.capLIB

CapView has several initialization files. These files are used to save data from one CapView run to the next. These files provide data that is shared by all CapView models. The following is a list of these files and their uses.

capAddGElem	This file contains the values that the user selected in the Add Members/Nodes dialogue box.
capAddLoad	This file contains the values that the user selected in the Add Loads dialogue box.
capDispDef	This file contains the values that the user selected in the Display Defaults dialogue box.
capDispProj	This file contains the values that the user selected in the Display Projection dialogue box.
capEditorCommand	This file contains the values that the user selected in the Editor Command dialogue box.
capIRRange	This file contains the values that the user selected in the IR Ranges dialogue box.
capInelasticEvent	This file contains the values that the user selected in the Inelastic Event dialogue box.
capPenSelection	This file contains the values that the user selected in the Pen Selection dialogue box.
capPenState	This file contains the values that the user selected in the Pen Attributes dialogue box.
capPrintSelect	This file contains the values that the user selected in the Print/View Files dialogue box.
capSacsDef	This file contains the values that the user selected in the SACS defaults dialogue box.
capStrudlDef	This file contains the values that the user selected in the STRUDL defaults dialogue box.
capUserUnits	This file contains the values that the user selected in the user defined units.
AISCSecRel.dat	This file contains the AISC sections' table data that is used by CapView.
AISCSecRel.idx	This file contains the AISC sections' table data that is used by CapView.
GenericMat.dat	This file contains the Generic materials' table data that is used by CapView.
GenericMat.idx	This file contains the Generic materials' table data that is used by CapView.
STSecs.dat	This file contains the standard tubular sections' table data that is used by CapView.
version_2.1	This files name indicates the users CapView version number.
Help->	This is a link to the directory that contains the CapView help files.

Platform Skeleton Files in ~/.capLIB/capPSD

This is a directory that contains platform skeleton data files. The Generate Platform Skeleton dialogue box reads and writes platform skeleton data in this directory. An example file is provided.

*.psd	These files contain the platform skeleton data files that CapView can use to generate a platform skeleton.
workshop.psd	Sample platform skeleton data file.

Ground Motion Record files in ~/.capLIB/capPSD

This is a directory that contains ground motion data files. An example file is provided.

*.gmr	These files contain ground motion record data to be used by CapView.
*.dat	These files contain notes on the corresponding ground motion record data files.

Files created by SEASTAR

These files are created by SEASTAR and stored in the project/model/version directory.

SSModel.ok	An ASCII file used by CapView that is present if run was successful. If this file does not exist then there was an error during the analysis.
SSModel.sts	An ASCII file used by CapView that contains analysis summary data. If there was an error during the analysis this is the first file that should be checked. It gives a summary of the analysis and may indicate where and why the anlalysis failed.
SSModel.out	An ASCII file that contains formatted SEASTAR output data. If there was an error during the analysis this is the second file that should be checked. It gives a detailed report of the analysis and will indicate where and why the anlalysis failed.
SSModel.acc	A binary file used by CapView that contains acceleration loads data.
SSModel.bid	A binary file used by CapView that contains buckling beam data.
SSModel.chk	A binary file used by CapDesign that contains member forces for code checking data.
SSModel.ele	A binary file used by CapView that contains element data.
SSModel.erg	An ASCII file used by SEASTAR that contains energy data.
SSModel.err	A binary file used by SEASTAR that contains runtime error data.
SSModel.fat	A file used by SEAFAT that contains stress envelopes for fatigue analysis.
SSModel.ldf	An ASCII file used by CapView that contains load level data.

SSModel.mod	A binary file used by CapView that contains mode shapes data.
SSModel.nct	A binary file used by CapView that contains node control data.
SSModel.nds	A binary file used by CapView that contains node displacement history data.
SSModel.ovr	A binary file used by CapView that contains overload data for inelastic event data.
SSModel.pst	A binary file used by CapView that contains general SEASTAR post-processor data.
SSModel.rst	
SSModel.rst2	Binary files used by SeaStar that contain restart data.
SSModel.sno	An ASCII file used by CapView that contains sequence numbers for post processor data.
SSModel.thf	A file used by SeaTimeHis that contains member force time history data.
SSModel.wav	A binary file used by CapView that contains wave load data.

FILES CREATED BY CapView

These fILES are CREATED BY CapView and stored in the project/model/version directory.

SSModel.inp	An ASCII file used by SEASTAR that is the main input file.
SSModel.cnf	An ASCII file used by SeaTimeHis and CapDesign that contain the configuration data.
SSModel.mem	An ASCII file used by SeaTimeHis and CapDesign that contain the Member code check parameter data.
MemberNames.dat	A file used by CapView that contains the CapView - SEASTAR member translation data.
NodeNumbers.dat	A file used by CapView that contains the CapView - SEASTAR nodes translation data.
*.dat	Files used by CapView that contain internal data.
*.idx	Files used by CapView that contain internal data.

FILES CREATED BY CapEvent

These fILES are CREATED BY CapEvent and stored in the project/model/version directory.

SSModel.evt	An ASCII file that contains the standard event information.
SSModel.pvt	An ASCII file used by CapView that contains processed. event data
Event.dat	A file used by CapView that contains the event data and date.

FILES CREATED BY SeaTimeHis

These fILES are CREATED BY SeaTimeHis and stored in the project/model/version directory.

test.out	A file that contains output data.
test.out.ERROR	A file that contains output data.
SSModel.thf.ir	A file used by CapView that contains interaction ratio time history data.
SSModel.thf.ir.max	A file used by CapView that contains interaction ratio time history data.

APPENDIX D: INTRODUCTION TO UNIX

1. Objective

This appendix is not intended to teach you how to use the UNIX operating system. However, it is intended to present the basic components and features of the UNIX system. Basically, it provides you with the information you need to get you started in learning and using UNIX.

2. Getting Started

Loggin In

- To log in, type your username to the system login prompt. Then type your password to the password prompt.

Home Directory

- A *directory file* can contain both files and other directories.
- A directory is similar to a file cabinet: a file cabinet has several drawers (directories), each of which can contain many folders (subdirectories), each holding one or more documents (files).
- A *file* is a collection of data stored under an assigned name.
- Whenever you log in, the UNIX system will place you in your *home directory*.
- Your home directory is your personal work area.

Setting/Changing a Password

- It you were not assigned a passwd when your login account was set up, you should set one right away. It is also a good idea to change your password from time to time.
- A password keeps unauthorized users from tampering with your files by assuring that no one can log into your account on the computer.
- To set a new password, use the UNIX **passwd** command. This will first ask you to type your current (old) password, then it will prompt you for the new one.

Logging Out

- To end your session, you must log out. This will prevent unauthorized use of your account. To log out, simply type **logout**.

3. UNIX Commands

- On UNIX systems the words *command* and *program* are almost synonymous. That is to say, when the user types a command, the UNIX system runs a program that performs the user's command. The idea is to keep UNIX modular by implementing *system commands* by programs.
- The UNIX system is made up of two parts: the *kernel* and the *shell*.
- The kernel is the core of the operating system. It is responsible for managing the computer's hardware and software resources. It performs three major tasks: virtual memory management, resource allocation/scheduling, and interprocess communications.
- The shell is the part of the system that interacts with the user (command interpreter). It is responsible for passing user commands to the kernel for

processing and for returning the results to the terminal screen, into a file, or to another device (e.g. printer).

- The UNIX shell itself is a program/command. There are several popular UNIX shells, among which, the Bourne shell and the C shell are the most commonly used.
- A UNIX command line can have three parts: the *command name*, its *options*, and its arguments.
- The UNIX system is case sensitive, which means that the system distinguishes between upper case and lower case letters.
- Each command, option, or argument consists of one *word* (a group ,or string, of characters surrounded by spaces).
- Options typically begin with a dash (-).

Command Names

- UNIX command names are short or abbreviated words that describe the programs they invoke. The are deliberately kept short to save time and reduce typing errors.
- When you enter a UNIX command, you can not abbreviate it or change its spelling in any way. Some of the commands can be cryptic and confusing for the novice user, and in fact, some of the commands have names which describe part of the functionality of the command (e.g. **cat, uptime**). Because the UNIX system was originally developed for systems programmers, many of the commands were not created with the new user in mind (e.g. **find**).
- If you type **ls** at the UNIX shell prompt, the system will display an alphabetical listing of the file and directory names.

Command Arguments

- An argument gives the system information that is required to process a specific command or to change the default behavior of a command. Some commands require one or more arguments; for others, arguments are optional.
- **ls** is an example of a command that accepts a file or a directory name as an optional argument. If you type **ls** followed by a file name, the system redisplays the file name if the file exits otherwise it will display an error message. If a directory name is used as an argument, the system displays the names of all files and directories in that directory.

Command Options

- An option (flag) is a special kind of argument. It provides additional versatility when used with a command. Most options are preceded by a dash (-). Some UNIX commands can accept both arguments and options.
- Options are specific to each command. For example the **rm** command has a flag to perform a recursive delete (**-r**) meanwhile the **chmod**'s recursive flag is **-R**. On the other hand, the **lpr**'s **-r** flag causes a file to be deleted after printing it. Be Careful.
- The **ls** command has several options. The **-l** or long form option causes **ls** to display additional information about the files and directories.

Environment Configuration Files

- Certain files in a user's home directory contain environment configuration commands. Among these files is the shell configuration file (.cshrc) which contains commands that are executed whenever a new shell is created.
- Another important file is the login script file (.login) which contains a list of commands which are executed each time you login.
- The .logout file contains a list of commands that are executed whenever you logout.
- Other files include: .mailrc .dbxinit .forward .plan .project .rhosts .Xdefaults .uwmrc .twmrc .xinitrc .sunview

4. UNIX Directory Structure

- On a UNIX system, the files are organized in a multi-level hierarchical structure. In which, a single master directory (the root directory) is at the top of level and additional files and directories are defined at various levels below it.
- Each file or directory (except the root directory) is defined within a directory. This directory is called the *parent* of the (*child*) file.
- The root directory is the first directory in the UNIX file system. The root directory contains several directories, which in turn can contain other files and directories. In the UNIX file naming convention, the root directory is referred to by slash (/).
- The UNIX file system is very flexible and the organization of your files and directories can be changed at any time to fit your needs.

File and Directory Naming Convention

- All programs, text, data and directories on disks reside in files. Each file or directory has a unique name. Because UNIX is case sensitive, file and directory names can include both upper case and lower case letters. They can also include numbers and special symbols. However, because certain special symbols have special meaning to the shell, it is recommended that the user does not use these symbols in file/directory names: ( ) * & $? / ! ~ ‘ ’ " ; < > |

Path Names

- A path name is a sequence of directory names separated by slashes followed by a file or directory name. It identifies a file or directory in the file system by specifying the path name to its parent. A path name, therefore, specifies the location of a file or directory.
- The full path name is the path name starting from the root directory (starts by a slash(/)).
- A relative path name starts at the current location or the home directory of a user.

Current Working Directory (pwd)

- To determine the current location, type **pwd** at the UNIX system prompt. This command (**p**rint **w**orking **d**irectory) will tell you where you are in the directory structure.

Creating a New Directory (mkdir)

- It is a good idea to organize files by filing them in directories. This is specially important when working with SEASTAR and its pre- and post- processors. For SEASTAR it is recommended to have a directory for each SEASTAR run that is to be saved.
- To create a new directory use the **mkdir dir-name** command. This command will create the new directory and assigns the name dir-name to it. However, it will not take you to the new directory.

Changing Directories (cd)

- To move from the current working directory to a new directory, use the **cd** command (**cd [pathname]**). The pathname argument is optional the default value for which is the home directory of the user.

Path Name Shortcuts: Dot, Dot Dot, ~, and Wild cards

- The UNIX system provides many shortcuts to help make life a little easier (and confuse the new users) and to minimize typing. A relative path name is one type of shortcut.
- When the first directory word in a relative pathname begins with a dot (./), this indicates that the pathname begins with the current working directory.
- When the first directory word in a relative pathname begins with two dots (../), this indicates that the pathname begins with the parent directory of the current working directory.
- When the first directory word in a relative pathname begins with a tilde (~) followed by a name, this indicates that the pathname begins with the home directory of the user who's name is given after the tilde.
- UNIX also recognizes two wild card characters. The ? symbol represents a single character wild card and the * symbol represents an arbitrary number of characters.

5. File and Directory Permissions

- Each file and directory on a UNIX system is associated with a set of *file access permissions* (protection modes) that specify who will have access to it. There are three types of permissions: *read, write,* and *execute.*

Read permission

Allows a user to view the contents of a file or copy it to another area. Read permission does not allow the user to change the contents of a file.

Write permission

Allows a user to change the contents of a file. Write permission on a directory allows a user to create and delete files in that directory.

Execute permission

Allows a user to execute the file as a UNIX command if the file contains UNIX command lines (or if the file is an executable program). Execute permission on a directory allows a user to **cd** to that directory.

- There are three sets of permissions each applying to one of three categories of users: *owner, group,* and *other.* The *owner* of the file owns the file or directory (generally, the user who created the file or directory in the first place). The *group* often consists of people working on the same project or in the same department and using the same files. Group membership is usually determined by the system administrator. *Other* refers to everyone on the system who is neither the owner

nor in the owner's group.

- The **ls -lg** command displays a listing of the files in a directory showing the permissions associated with every file. The access permissions appear on the left of the display. For example:

```
% ls -lg
total 22

-rwxr-xr-x 1  nashif  engi  9741 Jan  9  12:55  GenSD
drw------- 1  nashif  engi  512  Jan  9  12:56  results
-rw-r--r-- 1  nashif  engi  554  Jan  8  08:46  input1
-rw-r--r-- 1  nashif  engi  560  Jan  8  08:46  input2
```

- The first column specifies the file type: a dash (-) indicates a file and a **d** indicates a directory. Other file types include **l, b, c, p,** and **s**. The next three columns specify the read (r) , write (w), and execute (x) permissions for the owner (nashif). The next three columns specify the group permissions (engi) and the last three columns specify the permissions for the other users.
- To change the permissions of a file or a directory the **chmod** command can be used. This command takes an argument which indicate the permissions in an octal numeric representation. It also requires a list of files to change their permissions. The following table shows the octal equivalents for the basic permissions:

Permissions	Octal Representation
-	0
x	1
w	2
wx	3
r	4
rx	5
rw	6
rwx	7

- To change the ownership of a file the **chown** command can be used. This command takes an argument that specifies the new owner's username. It also requires a list of filenames to change their ownerships.
- To change the group membership of a file the **chgrp** command can be used. This command takes an argument that specifies the new group's. It also requires a list of filenames to change their group membererships.

6. Redirection of input and Output

- UNIX commands normally take their input from the terminal' keyboard (*standard input device*) and direct their output to the terminal's screen (*standard output device*).
- Some programs expect to get their input and/or redirect their output to a file, printer or any other I/O device.
- An important feature of UNIX is the ability to redirect the standard input and output devices to files or devices other than a terminal's keyboard and screen.
- Two symbols are used to indicate I/O redirection. The greater than symbol (>) redirects the output, and the less than symbol (<) redirects the input.
- When the output of a command is directed to a file, the system checks first if a file exists. If it does not, a new file is created. Then the system puts the output of

the command to the file (destroying the old contents of the file). To prevent the destruction of the old contents of a file make sure you have "**set noclobber**" in you .cshrc file.

- To append the output of a command to the contents of a file two grater than symbols (>>) must be used.

7. Pipes and Filters

- The UNIX system was built on the philosophy that utilities on the system should build on the work of other utilities and be reusable. There are two special notions that are important to UNIX: *pipes* and *filters*.
- A filter is a program that accepts input from one source, performs an appointed task on the data, and then writes the results to another place without changing the input file in any way. Examples of filters are **sort, grep, head, tail,** etc..
- A pipe is a connection between two or more commands that redirects the output of one command to be the input of another command. To create a pipe, you execute a command and pass its output (pipe its output) so that it becomes the input for the next command. The pipe is indicated by the vertical bar symbol (|). For example:

  cat east.emp west.emp | sort > all.emp

  cat east.emp west.emp | sort | lpr

8. Process Control

- UNIX is an interactive operating system in which the user can carry on a dialogue with the computer. Normally, you issue a command and wait for the shell to process it and return the shell prompt for you.
- If a command requires a long time to process and does not require any assistance from the user, waiting for the command to complete can be tedious and a waste of time.
- UNIX allows the user to request that a command be run in the background. In this mode the UNIX system starts the commands and allows the user to continue interacting with the shell while it executes the command.
- To run a command in the background type an ampersand (**&**) at the end of the command line. The system will start the command, print the process identification number (PID) associated with it and then returns the shell prompt.
- If you start a process in the foreground and decide to shift it to the background, you first have to stop the process by typing Cntl-Z (**^Z**). The UNIX system will stop the process. Then you must type **bg** to continue running the process in the background. If you type **fg**, UNIX will continue running the process in the foreground.
- To list the jobs which are either stopped or running in the background, type **jobs**.

9. References

The following reference was used as a guideline to create these notes, however, the information was modified to reflect the SunOS version of UNIX.

386/ix Operating System Guide, INTERACTIVE Systems Corporations

APPENDIX E: INTRODUCTION TO X WINDOW SYSTEM

1. What is X Window System

X Window System, Version 11 Release 4 (X11R4) is a network-transparent graphics window system. To achieve network transparency, the architecture of X11R4 is divided into two distinct parts: *display server* and *clients*. This division in the architecture of X11R4 allows clients and the display server to run on the same machine or on different machines connected by a network. For example, you can have a client program running on a supercomputer and using a low-end PC-based workstation as the display server. Even though the client program is running on the more powerful system, all user interaction (I/O) occurs on the display server workstation and is transmitted over the network to the supercomputer.

2. The Display Server

The X11R4 display server is a program that controls a bitmapped display screen. It allows multiple client programs to share the display resource simultaneously. It keeps track of all input devices (e.g. keyboard, mouse) and "input" from all clients that are running. Depending on all the information it receives it updates the display.

Servers are available for workstations, MS-DOS PC's, Macintoshes, and special terminals.

3. Clients

Clients are the application programs that perform specific tasks. X11R4 distribution includes several client programs. The most useful of which are *xterm, twm, xclock, bitmap, xcalc, xdpr*. Other clients are also available from different vendors (e.g. CapView).

The X11R4 display server allows clients to run on different computers on the network. It is important to note that clients may not look and behave the same on different servers because servers differ in their screen resolution, color capabilities and the way users customize their display environments.

4. The *xterm* Terminal Emulator

Since the X11R4 display server is designed to control only bitmapped graphics displays, the most important client program is a terminal emulator. A terminal emulator allows application that are designed to run on alphanumeric terminals to run on the X11R4 display. The standard X11R4 distribution includes a client that brings up a window on the display and emulates a DEC VT102 terminal or a Tektronix 4015 terminal. Since the display server allows several clients to run simultaneously, you can bring up more than one *xterm* window and run several programs at once.

In addition to xterm, OpenWindows has another terminal emulator called shell tool. More information on this can be found in [7].

5. The Window Manager

A window manager is a client that allows you to control the size and location of windows on your display. With a window manager the user can move and resize windows, raise and lower overlapped windows, create new window (by starting new clients) and collapse windows into icons. Window managers read a resource file when they start and bind keyboard and mouse keys to the various operations as specified in the resource file. A resource file allows you to

build menus to move/resize windows, start new clients, etc.

X11R4 distribution includes a window manager client called twm. OpenWindows has another window manager called olwm. More information on this can be found in [7].

6. Other Clients

The following is a list of the most commonly used clients.

xclock

display the time of day.

xcalc

emulates a TI-30, HP-10C or a slide rule.

bitmap

a bitmap editor to create pointer cursors, icons and background patterns.

xload

displays a histogram of the load average.

xwd

dumps the contents of a window into a file.

xpr

translate the contents of a file created by *xwd* to format suitable for printing.

OpenWindows DeskSet has other clients, More information on these can be found in [7].

7. Starting X11R4

To start X11R4, make sure that /usr/bin/X11 is in your path. Then, type at the prompt:

```
% startx
```

which is actually a shell script that looks like the following

```
xinit ; clear ; kbd_mode -a
```

xinit is the display server initialization program. The rest of the script makes sure that the screen and the keyboard are reusable in case the display server crashes.

xinit starts the X11R4 display server and looks for a resource file called ~/.xinitrc. If it can not find this file it brings up an *xterm* window at the upper left corner. Otherwise it starts the clients specified in the resource file. For example the following resource file starts *xterm, xclock, xload* and the *twm* window manager.

```
xterm &
xclock -geometry 100x100-0-0 &
xload -geometry 250x100-100-0 &
twm
```

To start OpenWindows, make sure that /usr/openwin is in your path. Then, type at the prompt:

```
% openwin
```

which is actually a shell script. More information on this can be found in [8].

8. Display Names

From the user's prospective, every X11R4 server has a *display name* of the form:

hostname:displaynumber.screennumber

This information is used by the application to determine how it should connect to the server and which screen it should use by default (on displays with multiple monitors): *hostname* The *hostname* specifies the name of the machine to which the display is physically connected. If the hostname is not given, the most efficient way of communicating to a server on the same machine will be used. *displaynumber* The phrase "display" is usually used to refer to collection of monitors that share a common keyboard and pointer (mouse, tablet, etc.). Most workstations tend to only have one keyboard, and therefore, only one display. Larger, multi-user systems, however, will frequently have several displays so that more than one person can be doing graphics work at once. To avoid confusion, each display on a machine is assigned a *display number* (beginning at 0) when the X11R4 server for that display is started. The display number must always be given in a display name. *screennumber* Some displays share a single keyboard and pointer among two or more monitors. Since each monitor has its own set of windows, each screen is assigned a *screen number* (beginning at 0) when the X11R4 server for that display is started. If the screen number is not given, then screen 0 will be used.

On UNIX systems, the default display name is stored in your DISPLAY environment variable. This variable is set automatically by the *xterm* terminal emulator. However, when you log into another machine on a network, you'll need to set DISPLAY by hand to point to your display. For example,

```
% setenv DISPLAY myws:0
$ DISPLAY=myws:0; export DISPLAY
```

Finally, most X11R4 programs accept a command line option of **-display** *displayname* to temporarily override the contents of DISPLAY. This is most commonly used to pop windows on another person's screen or as part of a "remote shell" command to start an xterm pointing back to your display. For example,

```
% xeyes -display joesws:0 -geometry 1000x1000+0+0
% rsh big xterm -display myws:0 -ls </dev/null &
```

9. Command Line Options

Most X11R4 programs attempt to use the same names for command line options and arguments. All applications written with the X11R4 Toolkit automatically accept the following options:

–display*display*

This option specifies the name of the X11R4 server to use.

–geometry*geometry*

This option specifies the initial size and location of the window.

–bg*color*,

Either option specifies the color to use for the window background.

–bd*color*,

Either option specifies the color to use for the window border.

–bw*number*,

Either option specifies the width in pixels of the window border.

–fg*color*,

Either option specifies the color to use for text or graphics.

–fn*font*,

Either option specifies the font to use for displaying text.

–iconic

This option indicates that the user would prefer that the application's windows initially not be visible as if the windows had be immediately iconified by the user. Window managers may choose not to honor the application's request.

–name

This option specifies the name under which resources for the application should be found. This option is useful in shell aliases to distinguish between invocations of an application, without resorting to creating links to alter the executable file name.

–rv,–reverse

Either option indicates that the program should simulate reverse video if possible, often by swapping the foreground and background colors. Not all programs honor this or implement it correctly. It is usually only used on monochrome displays.

+rv

This option indicates that the program should not simulate reverse video. This is used to override any defaults since reverse video doesn't always work properly.

–synchronous

This option indicates that requests to the X11R4 server should be sent synchronously, instead of asynchronously. Since *Xlib* normally buffers requests to the server, errors do not necessarily get reported immediately after they occur. This option turns off the buffering so that the application can be debugged. It should never be used with a working program.

–title*string*

This option specifies the title to be used for this window. This information is sometimes used by a window manager to provide some sort of header identifying the window.

–xrm*resourcestring*

This option specifies a resource name and value to override any defaults. It is also very useful for setting resources that don't have explicit command line arguments.

10. References

[1] Johnson, E. and Richard, K., "X Window Applications Programming", MIS:press 1989

[2] Jones, O., "Introduction to the X Window System", Prentice Hall 1989

[3] Nye, A., "Xlib Programming Manual", O'Reilly & Associates, Inc. 1988

[4] Nye, A., "Xlib Reference Manual", O'Reilly & Associates, Inc. 1988

[5] O'Reilly, T., Quercia, V. and Lamb, L., "X Window System User's Guide", O'Reilly & Associates, Inc. 1988

[6] Young, D., "X Window Systems Programming and Applications with Xt", Prentice Hall 1989

[7] "OpenWindows Version 3 DeskSet Reference Guide", SunSoft. 1991

[8] "OpenWindows Version 3 User's Guide", SunSoft. 1991