NASA Contractor Report 201653

A Users' Manual and Tutorial for Performing COMET Structural Analyses Using an Advanced User-Friendly Interface

Charles L. Blackburn, Christine G. Lotts, and Brian H. Mason
Analytical Services and Materials, Inc., Hampton, Virginia

Contract NAS1-19700

January 1997

National Aeronautics and
Space Administration
Langley Research Center
Hampton, Virginia 23681-0001

ACE

Advanced COMET-Interface Environment

A USERS' MANUAL AND TUTORIAL FOR PERFORMING COMET STRUCTURAL ANALYSES USING AN ADVANCED USER-FRIENDLY INTERFACE

Charles L. Blackburn, Christine G. Lotts and Brian H. Mason
Analytical Services and Materials, Inc.
Hampton, VA

A report prepared for the
Computational Mechanics Branch
NASA Langley Research Center

Foreword

This users' manual describes the use of a graphical interface developed for the general purpose finite element system, COMET. There has been some considerations given to user-friendly concerns in this development and these considerations will continue and increase in priority as the complexity of the analyses increases. **When the interface is fully user-friendly, it will not be necessary to write this users' manual.**

Introduction

The COMET software system is a highly modular and flexible structural analysis system for research in computational methods and for exploration of new multiprocessor and vector computer hardware. These attributes: namely, modularity and flexibility, make COMET attractive to analysts, researchers, and developers in the structural mechanics discipline. However, these attributes contribute to the complexity of the system and make its usage difficult to grasp. Although the engineer/user who wants to perform an analysis may understand the data requirements and mathematical manipulations required to effect his/her desired results, he/she may not be able to,

- identify the procedures and/or processors required for a particular analysis and their related I/O requirements and
- interact with the computer to execute COMET without extensive training, or by reading applicable documentation and/or attending classes.

Consequently, a valuable tool is lost to the generic user who may be unwilling or unable to afford the time on the learning curve before they can perform meaningful analyses. Therefore, the development of a user-friendly interface will permit immediate execution of the program by a novice user while providing training through an interactive query mode such that the user will become proficient with the software and hardware. This initial development has been directed toward the generic user who will be performing structural analyses on a component or an assembly of structural components. Future development will enhance this user-friendly interface to support the development of the system by researchers and/or computer scientists.

ACE (Advanced COMET-Interface Environment) provides the capability to create and execute a COMET runstream file by interactive query of the user and/or pull down menu selections for defining the I/O and the type of analysis. Two modes of interaction have been implemented; namely, "QUERY" (or novice) and "EXPERT". Each of these modes only addresses the users' capability in exercising the COMET program and does not refer to their expertise in performing finite element analysis or computer system operations. In the QUERY mode, the runstream is created by querying the user and directing them to the proper menu selections. In the "EXPERT" mode, it is assumed that the user is familiar with the program and the use of window-based application software.

Getting Started

The ACE development is being done on a UNIX based DECStation 5000 workstation. The application software can be ported to other UNIX based workstations and mainframes which support X-windows and Motif. The host computer must have sufficient memory,

both RAM and hard disk, to satisfy the requirements of both ACE and COMET. The user can run ACE from a remote X workstation such as a Macintosh with MacX (version 1.2 or higher) or a UNIX based workstation with an X window server running.

This Phase 1 implementation has been restricted to performing linear static analyses on stiffened shell structures made of isotropic materials. It is assumed that the geometry (nodes and elements) of the finite element model have been previously defined by a PATRAN exercise or have been stored in a COMET data base file. Further, the PATRAN neutral file may contain constraint, loads, and element orientation data and the COMET data base file may contain all the above in addition to material and section property data. These files must exist on the same computer where ACE is being executed.

Setup Workstation to Execute ACE

ACE is executed on a UNIX host computer where COMET also resides. It's windows are displayed on a remote X Window workstation. It can be initiated in one of two ways. The first way is to log in to the host computer with the "telnet" or "rlogin" command in an xterm window, then once logged in, execute ACE with the "run_ace" command. The second way is to execute a remote shell to the host computer, for example:

```
rsh blackbird "setenv DISPLAY workstation:0.0; /usr/bin/X11/xterm -ls -e /csm/bin/run_ace working_directory"
```

where *workstation:0.0* should be the network address and screen of your workstation's X-server. The *working_directory* parameter should be the name of the directory you want to work in on the host computer; it can be omitted if you want to work in your home directory.

On a Macintosh workstation running MacX version 1.2, you can set up the command to be executed on blackbird as follows:

```
setenv DISPLAY "workstation:0.0"; /usr/bin/X11/xterm -ls -e /csm/bin/run_ace "working_directory"
```

where "*workstation:0.0*" is the network address and screen of your workstation's X-server. You can substitute "®display" (® is typed by holding down the option key and striking the r-key) in place of "*workstation:0.0*" and MacX will place the correct information for you. The "working_directory" parameter should be the name of the directory you want to work in on the host computer; it can be omitted if you want to work in your home directory .

Selecting the Level of Interaction

As mentioned earlier, ACE can be run in two modes of interaction, "QUERY" or "EXPERT". To begin execution of ACE, type

run_ace -e [C/R] to execute ACE in the "EXPERT" mode

run_ace -q [C/R] to execute ACE in the "QUERY" (default) mode

which then brings up the ACE main window, shown in Figure 1.

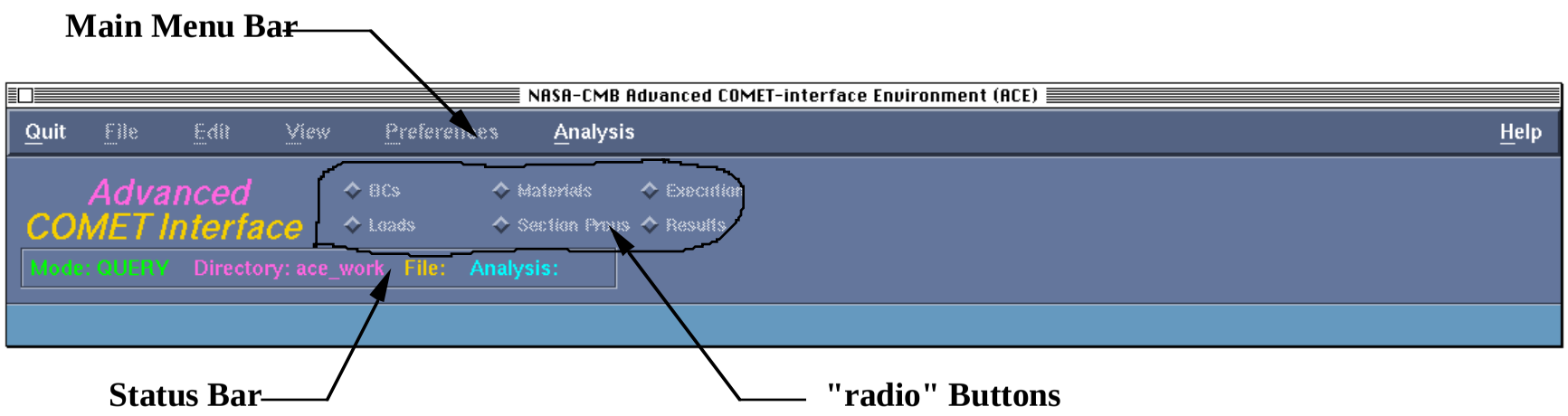

Figure 1

- The horizontal bar which spans the width of the main window is the menu bar with pull-down menus for frequently used functions.
- The intermediate region below the menubar is the control area with "radio" buttons for the sub-groups of problem definition.
- In the lower left corner of the main menu window, below the control area is an information box, called the "status bar". The mode of interaction, the current directory (in this case, work) where the finite element model resides, the name of the model file, and the type of analysis are displayed in the status bar.

The "QUERY" mode will give the user detailed information on how to proceed through the menus in a pre-defined step by step manner to formulate or modify and execute the runstream.

The "EXPERT" mode permits the user to make his/her own selections, without guidance, from the top-level menu and control area.

For the present exercise, execute ACE in the "QUERY" mode by entering,

run_ace -q C/R *

* Bold Lettering indicates a response by the user.

Function	ACE Screen Component
Specify analysis type	Menu Bar/Analysis
Input Model geometry	Menu Bar/File/Open Model Database
Select COMET elements	Define Elements Dialog
Examine initial model (optional)	COMET Graphics Processor Menus
Define material properties	Materials "radio" Button
Define section properties	Section Props. "radio" Button
Define/modify boundary conditions	BC "radio" Button
Select analysis options	Menu Bar/Preferences/Analysis
Execute	Execute "radio" Button
Post-process results (optional)	Results "radio" Button
Done	Menu Bar/Quit

ACE Road Map

The basic procedure to be performed in ACE is outlined in the table above. The steps required for creating and executing a COMET analysis are listed in the order that they should be performed, and the ACE screen components used to initiate the actions are shown. Detailed discussion of the screens and data is presented later in this report.

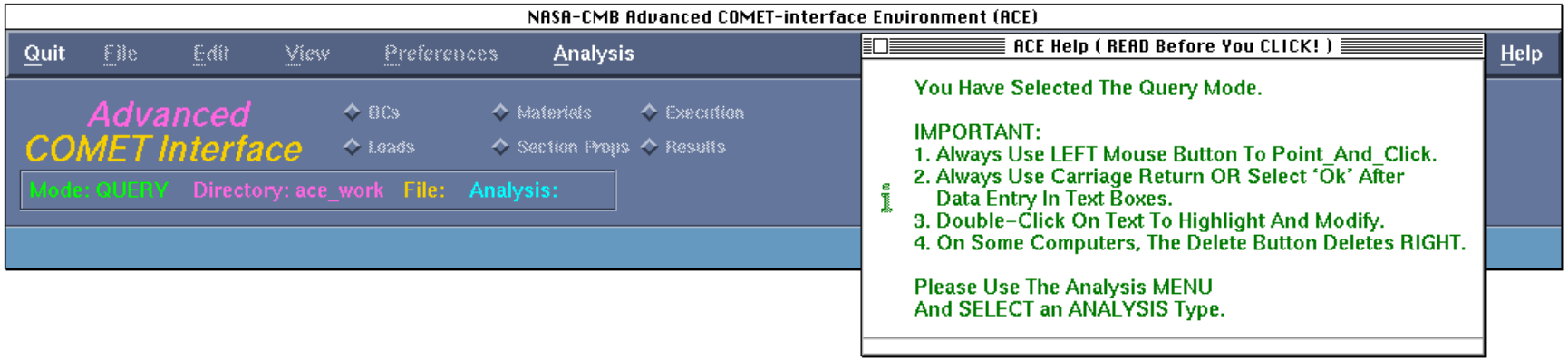

Figure 2

Selecting the Type of Structural Analysis

The screen will now appear as shown in Figure 2 with the "HELP" window in the upper right hand corner which is the usual location.

- The HELP window has a white background and contains detailed instructions.
- It is noted that the "QUERY" selection has been identified in the "status bar".

After reading the operational instructions for the mouse and data entries, the user is instructed to click on the ANALYSIS menu item (which has been highlighted) and then select an ANALYSIS type. **Click on "ANALYSIS".**

Note: On occasions, the "HELP" window may be "lowered" below the main window. **Click on "Help"** button to raise it to the top.

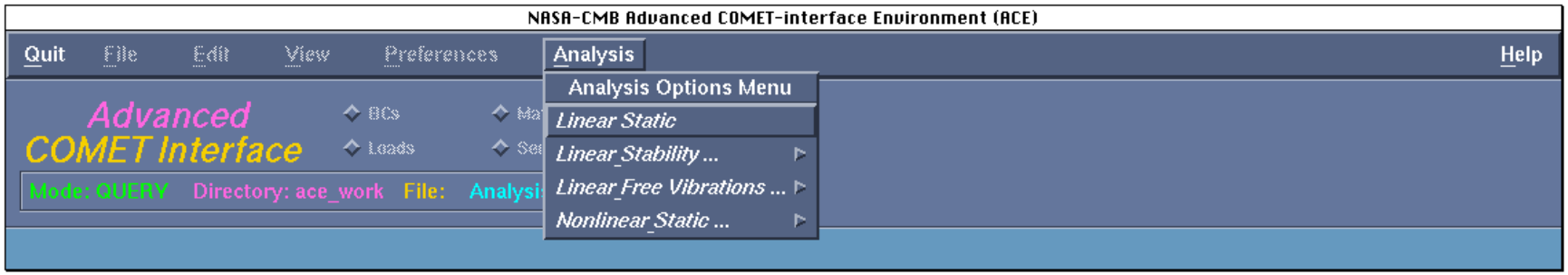

Figure 3

When the mouse is clicked on top of the ANALYSIS menu item, the sub-menu window entitled "ANALYSIS OPTIONS MENU" is opened as shown in Figure 3.

- Several analysis options are listed.
- Some menu items can be selected to reveal another level of menus (as indicated by the three dots following the name of the menu item).
- All items except for LINEAR STATIC appear gray in color as they have not been implemented as yet, but are planned for later releases.

Click on "Linear Static" to specify the analysis type to be used for this session**.**

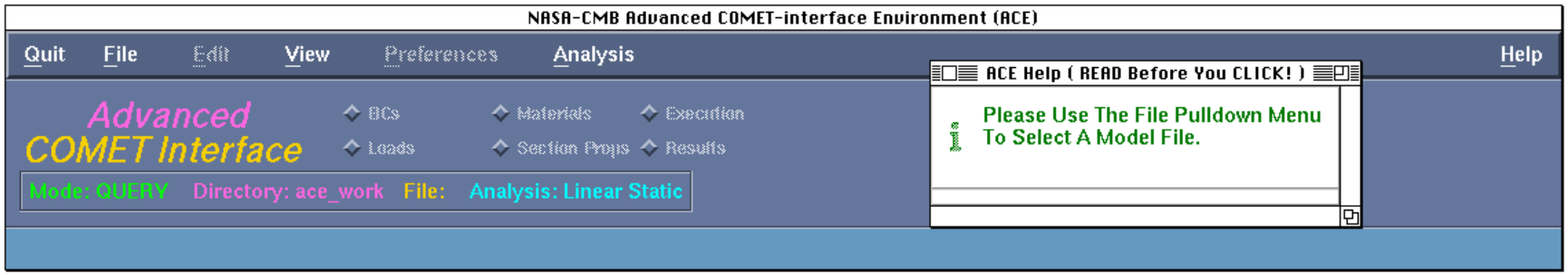

Figure 4

Selecting the Finite Element Model File

The "status bar" in Figure 4 indicates the selection of "Linear Static" analysis and the "HELP" window directs the user to click on the FILE menu item (which is now highlighted) to select a finite element model file that currently exists on this computer where ACE is being executed. **Click on "File".**

Figure 5

After clicking on FILE, the pull-down menu called "FILES MENU" appears as shown in Figure 5. The gray menu items are not available for selection at this time either because they are not required at the present time, or they have not been implemented as yet. The user has been instructed to select a model file; therefore, **click on "Open Model Database"**. This selection will cause all files with a "pat" (PATRAN neutral files) or "dbc" (COMET database files) extension to be selected from the directory identified in the "status bar"(in this example, the directory is "work").

As previously mentioned, the three (3) dots following a menu item indicates that another level of interactive user response follows. For this exercise, the previous action opened up the window shown in Figure 6(a).

- The "HELP" window instructs the user to click on the "radio" button designating the file type containing the model file.
- The default selection is PATRAN Neutral Files.
- Clicking on each displays a list of files of the corresponding type available in the working directory.

- In the event that the model files are contained in a different directory than the working directory, the user can navigate the file system to the desired directory by clicking on the "One Level Above" or "One Level Below" as many times as are required.
- Having located the desired model file (which may require using the scroll bars if there are too many files to appear in the window), **place the cursor on top of the file** (for this example, "demo1" or "demo2") **and click**.
- This will highlight the selected file name. The file name is also displayed in the "Selection" field below the "radio" buttons (Fig. 6 (b)).
- The name and identification of the file will appear in the "HELP" window (Fig. 6 (b)).
- You can change the selection by scrolling through the list of file names and selecting another file by placing the cursor on the file name and clicking to select.

Clicking on "Cancel" causes the file selection process to terminate without a file being selected.

Now, **click on "Select"** to select the identified file

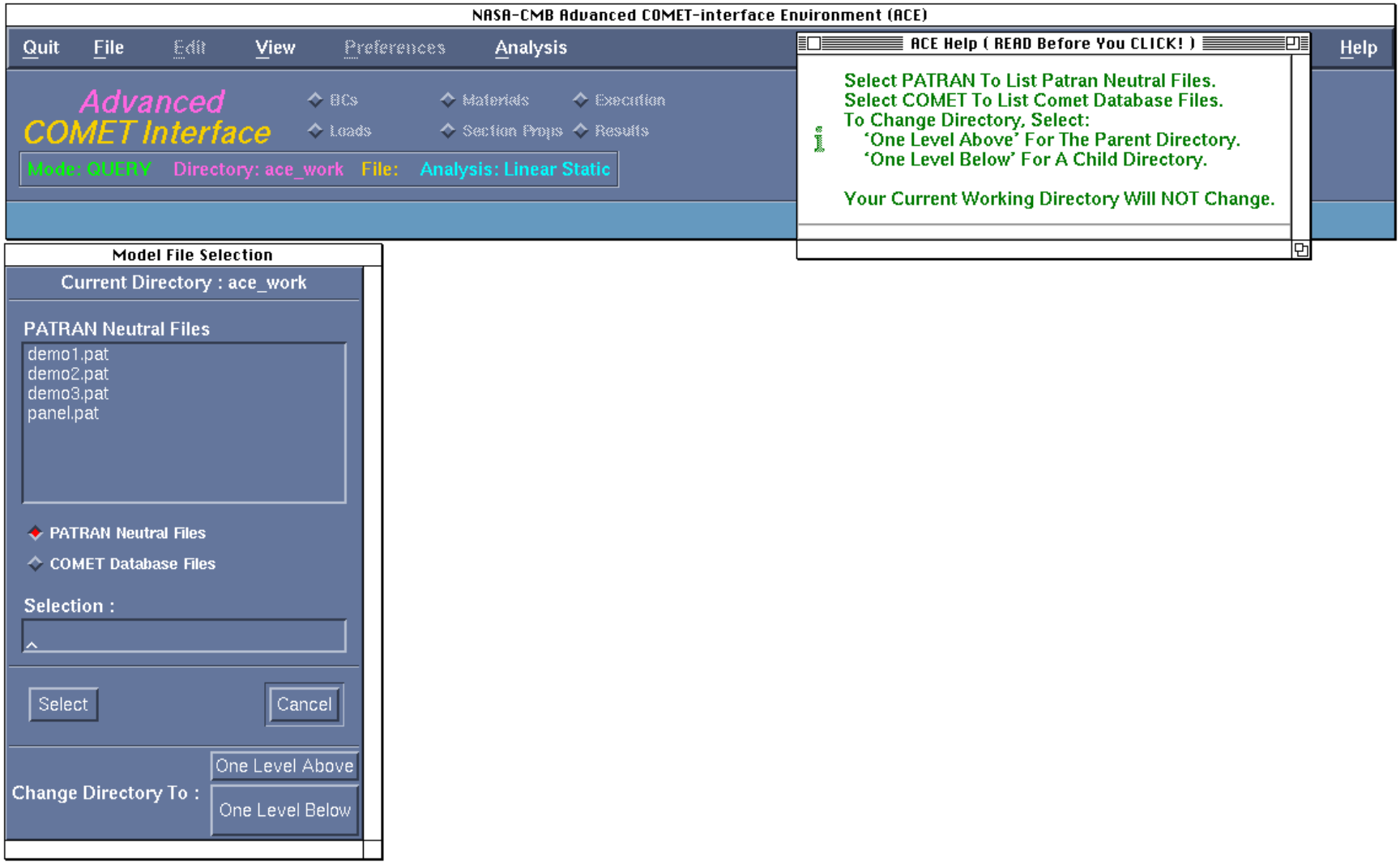
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File:
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
Select PATRAN To List Patran Neutral Files.
Select COMET To List Comet Database Files.
To Change Directory, Select:
'One Level Above' For The Parent Directory.
'One Level Below' For A Child Directory.
Your Current Working Directory Will NOT Change.
Model File Selection
Current Directory : ace_work
PATRAN Neutral Files
demo1.pat
demo2.pat
demo3.pat
panel.pat
PATRAN Neutral Files
COMET Database Files
Selection :
Select
Cancel
Change Directory To :
One Level Above
One Level Below

Figure 6 (a)

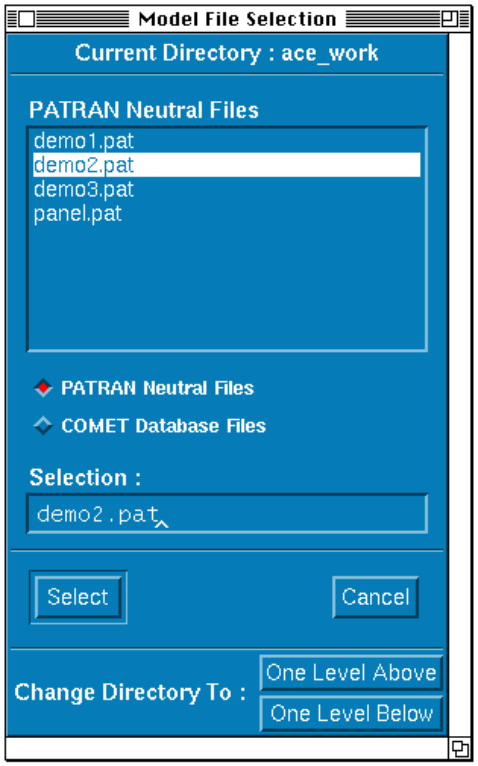

Figure 6 (b)

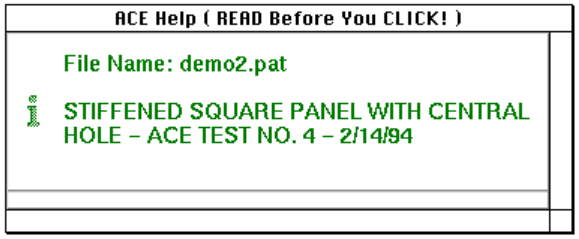

Figure 6 (c)

At this point, a small "info" window, in the center of the screen, directs the user to wait while the designated file is being processed to extract the data describing the finite element model. (The present PATRAN processor does not recognize some loading data such a distributed loads, more than one load or constraint case, material properties, and section properties.) See Appendix A for a discussion of how the neutral file data is associated with COMET modeling capabilities.

Generating the Runstream for Defining the Finite Element Model

Element Selection

As shown in Figure 7 (a), the "status bar" reflects the name of the file selected and the "HELP" window describes the steps to follow in selecting the COMET element formulations. These steps are reiterated in an abbreviated form in the "Define Elements" window along with the name of the model file in which the element types were found.

Since the element types are purely a function of the geometry, it is necessary to select an element formulation that will describe the response of a structural component represented by this finite element model. For example, in some situations a 2-node rod element that carries an axial load only may be preferred over the more complex 2-node beam that carries axial and shear loads and bending and torsional moments. These element types can be

(a) 2-node beam elements,

(b) 3, 4, and 9-node plate and shell elements,

(c) 8, 10 and 20-node solid elements and/or

(d) 12 and 15-node shell-to-solid transition elements (not implemented in the current PATRAN translator).

Select an "element type" (the selection will be highlighted) from the top window. A list of available COMET elements corresponding to this nodal arrangement will appear in the window below the "Element Types" (Fig. 7 (b)). A "Help" window (Fig. 7 (c)) will appear in the usual location.

Clicking on a particular element will highlight that selection (Fig. 7 (d)) and information describing the formulation of the element and the recommended use will be displayed in the "Element Information" window (Fig. 7 (f)) which is in the central part of the screen. As explained in the "Help" window (Fig. 7 (e)), another selection can be made if after reading this information you find that the element does not satisfy your modeling requirements. (At this time, ACE can only accommodate a single element formulation for each element geometry.)

Once a COMET element has been selected for a particular element geometry, return to the "Define Elements" window (Fig. 7 (g)) and continue to specify a COMET element for all remaining elements geometry found in the model file (Figs. 7 (h) and 7 (i)) until the entire list is exhausted. The selections do not have to be done in order and can be changed. Referring to the instructions in the "Help"

window (Fig. 7 (e)), **click on "Accept".** Finally, the user is asked to confirm his/her element selections or repeat some or all of the prior steps. **Click on the appropriate button.**

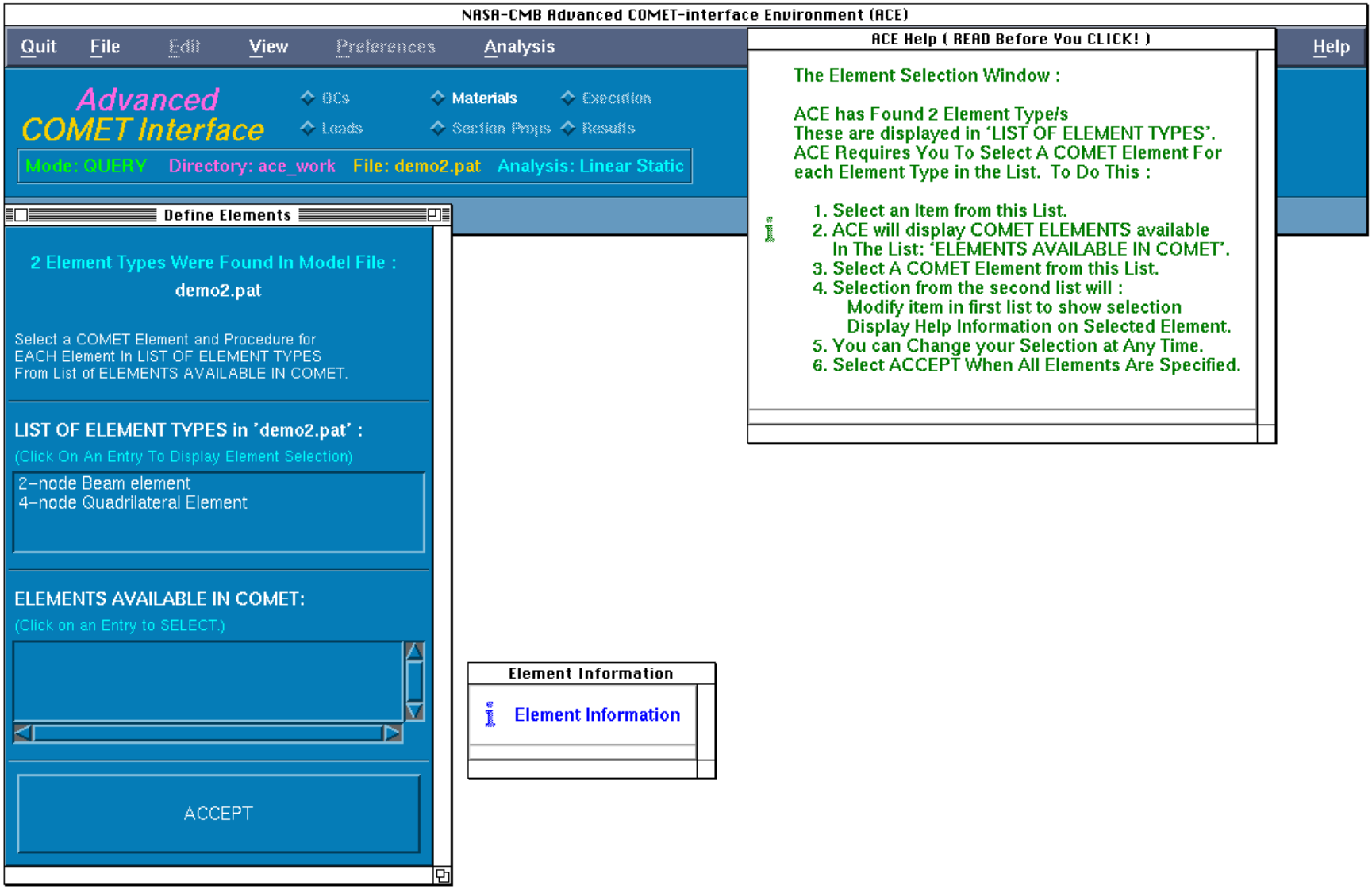
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
Define Elements
2 Element Types Were Found In Model File :
demo2.pat
Select a COMET Element and Procedure for
EACH Element In LIST OF ELEMENT TYPES
From List of ELEMENTS AVAILABLE IN COMET.
LIST OF ELEMENT TYPES in 'demo2.pat' :
(Click On An Entry To Display Element Selection)
2-node Beam element
4-node Quadrilateral Element
ELEMENTS AVAILABLE IN COMET:
(Click on an Entry to SELECT.)
ACCEPT
ACE Help (READ Before You CLICK!)
The Element Selection Window :
ACE has Found 2 Element Type/s
These are displayed in 'LIST OF ELEMENT TYPES'.
ACE Requires You To Select A COMET Element For
each Element Type in the List. To Do This :
1. Select an Item from this List.
2. ACE will display COMET ELEMENTS available
In The List: 'ELEMENTS AVAILABLE IN COMET'.
3. Select A COMET Element from this List.
4. Selection from the second list will :
Modify item in first list to show selection
Display Help Information on Selected Element.
5. You can Change your Selection at Any Time.
6. Select ACCEPT When All Elements Are Specified.
Element Information
Element Information

Figure 7 (a)

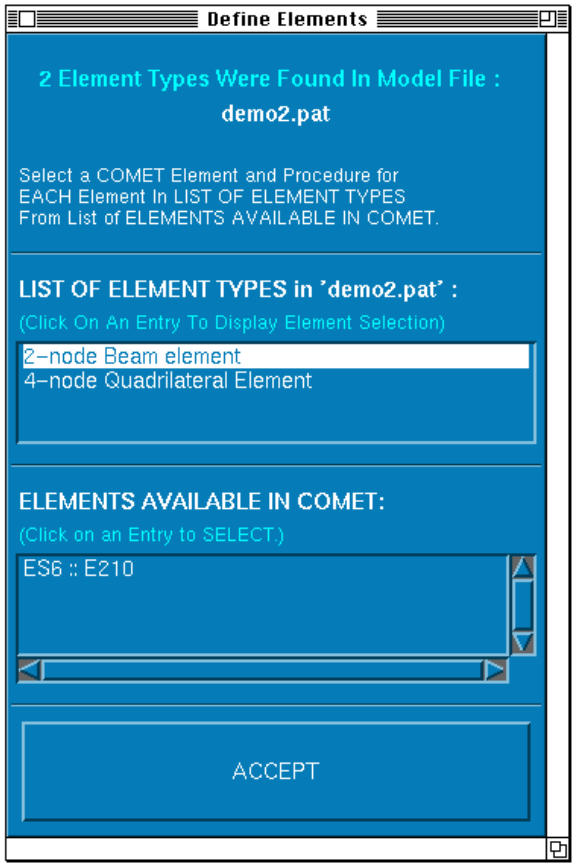

Figure 7(b)

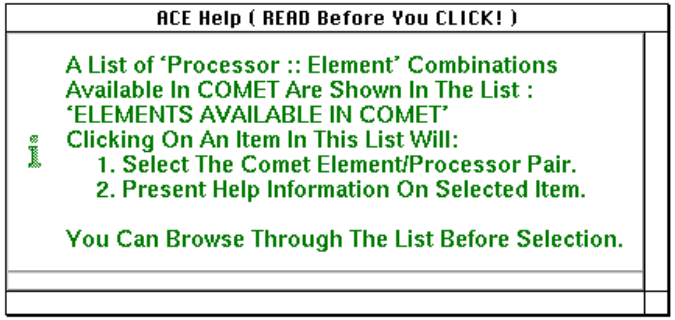

Figure 7(c)

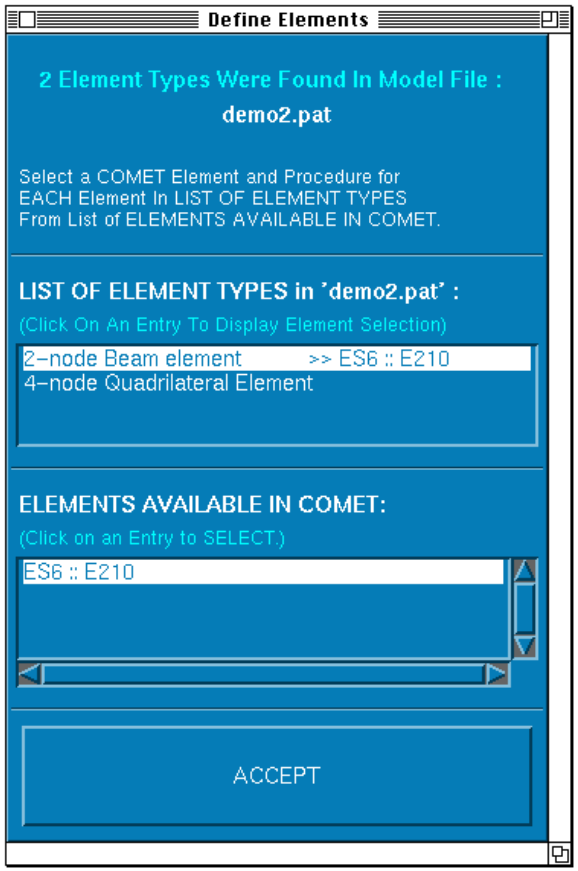

Figure 7(d)

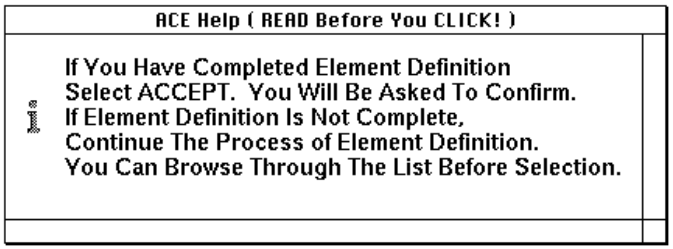

Figure 7(e)

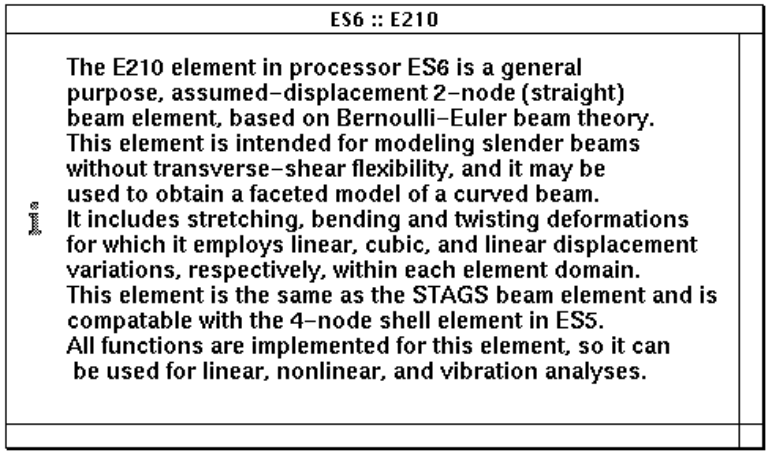

Figure 7(f)

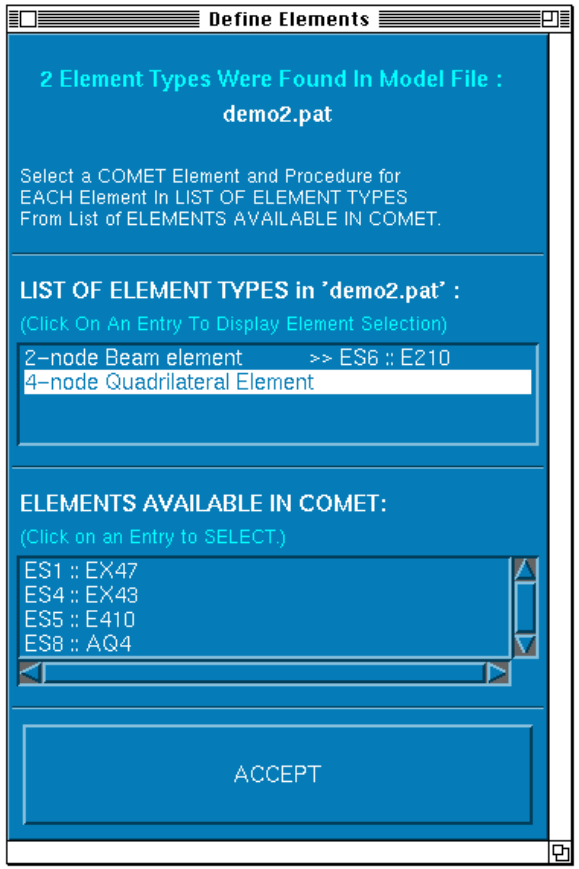

Figure 7 (g)

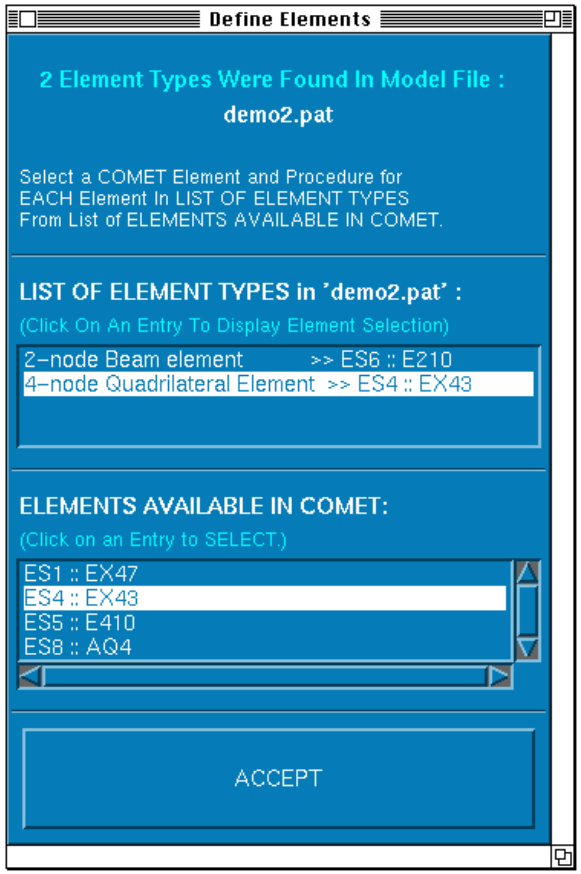

Figure 7 (h)

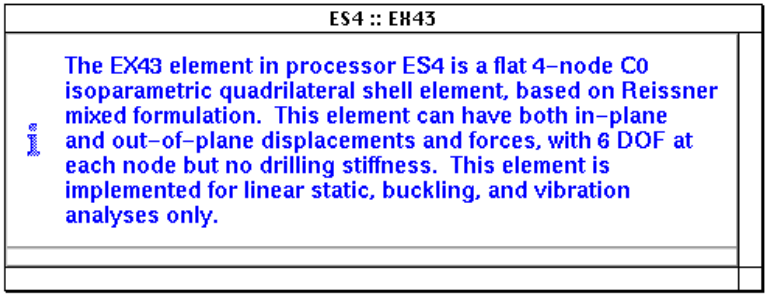

Figure 7 (i)

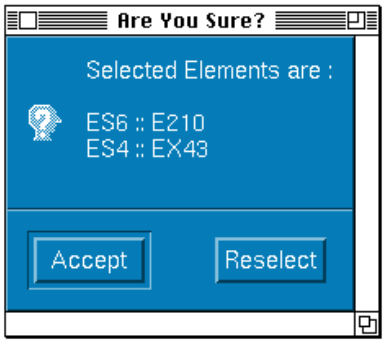

Figure 7 (j)

Assuming that the necessary and relevant elements have been accepted, the user is advised to wait (see Fig. 8) and then the finite

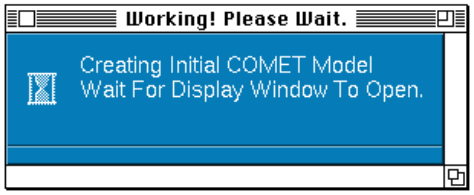

Figure 8

element model is displayed in a COMET Graphics (CGx) window as shown in Figure 9 (a) along with the "main menu" and "Help" windows. The CGx window has its own set of menus and its general use will not be documented herein. The "Help" window informs the user of the number of section types that have been identified (one for "demo1" and two for "demo2") and directs the user to click on the highlighted "Materials" "radio" button in the control area. **Click on the "radio" button adjacent to "Materials"** (it will glow red).

Material Properties Definition

Selecting the Materials "radio" button displays a "MATERIALS SELECTION DIALOG" window, along with an updated "Help" window (Fig. 9 (b)). Following the directions given in the "Help" window,

- To select a material type, **click on the "option" button labeled "Isotropic"**, the default material type.
- An option menu will appear listing several material types such as Isotropic, Orthotropic, Plasticity, 2-D Anisotropic, 3-D Anisotropic, and Composite. The material types shown in gray have not been implemented.
- The user is now asked about the location of the materials data. In other words, does the data reside on a archived database (available in this working directory) or must the data be entered using the keyboard.
- To define a new material (which is the default), answer "Yes" to the second question.
- To select a material from an archived materials database, **click on the "radio" button next to "Yes "** after the first question. The button will glow red. The answer to "Define A New Material?" will automatically be set to "No".
- A modified "MATERIALS SELECTION DIALOG" window (Fig. 9 (d)) that also reflects the new help messages is displayed. The user is instructed to **click on "Files"** menu and then **select the "Open Material Database"** option (Fig. 9 (e)).

This action opens the "Material Database Selection" window (Fig. 9 (f)) which lists all the material database files (those with the file extension, "mat").

- Select the desired materials database file by **clicking on the name of the file** ("materials.mat" for this example)**.** The selection is highlighted (Fig. 9 (g)).
- The names of the materials of the selected material type available in the selected materials file are listed in a scrollable sub-window of the "MATERIALS SELECTION DIALOG" window titled "Materials in Database File:" (Fig 9 (h)).
- To select a material for addition to the runstream, **place cursor on top of that material and click.** The selection will be highlighted (Fig 9 (i)) and will appear immediately in the sub-window called "Materials Selected For Computation:".

- The properties of the highlighted material can be viewed or modified. **Click on "Materials Properties ..."** to open a new window (Fig. 9 (k)) identified by the name of the selected material. All data in this scrollable window, including the material name, can be modified.
- To modify, double-click or drag with mouse button pressed on the text area. Type over the highlighted area to replace text. Click on "OK" button or enter a carriage return to accept data change or "Cancel" button to negate the change. Clicking on the "Apply" button saves the changes and exits and clicking on the "Cancel" button quits without saving the modifications.

Click on the "Apply" button.

After completing the materials specifications, **click on the "Done" button** to return to the main menu.

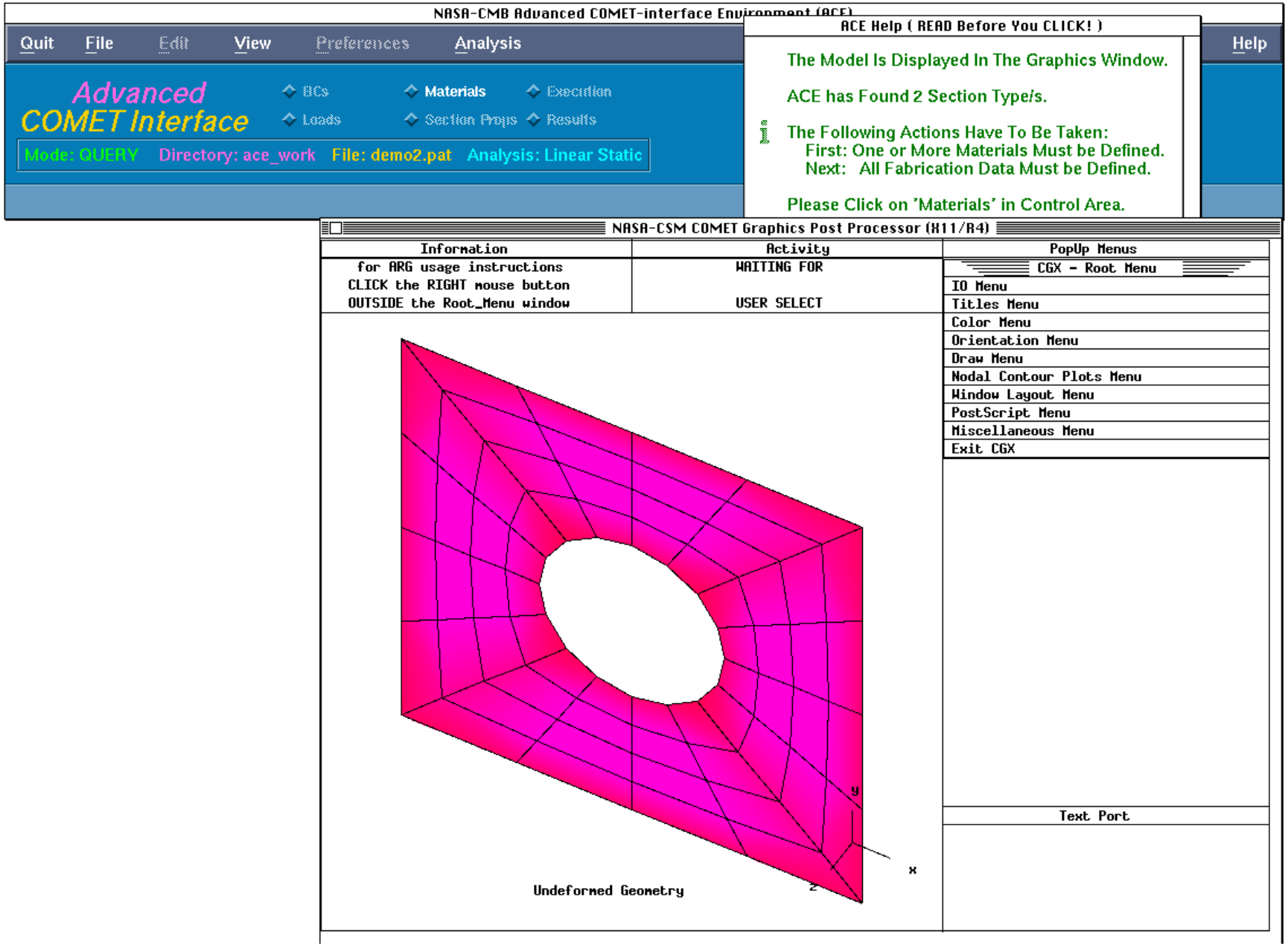
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
The Model Is Displayed In The Graphics Window.
ACE has Found 2 Section Type/s.
The Following Actions Have To Be Taken:
First: One or More Materials Must be Defined.
Next: All Fabrication Data Must be Defined.
Please Click on 'Materials' in Control Area.
NASA-CSM COMET Graphics Post Processor (H11/R4)
Information
Activity
PopUp Menus
for ARG usage instructions
CLICK the RIGHT mouse button
OUTSIDE the Root_Menu window
WAITING FOR
USER SELECT
CGX - Root Menu
IO Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
y
x
z
Undeformed Geometry

Figure 9 (a)

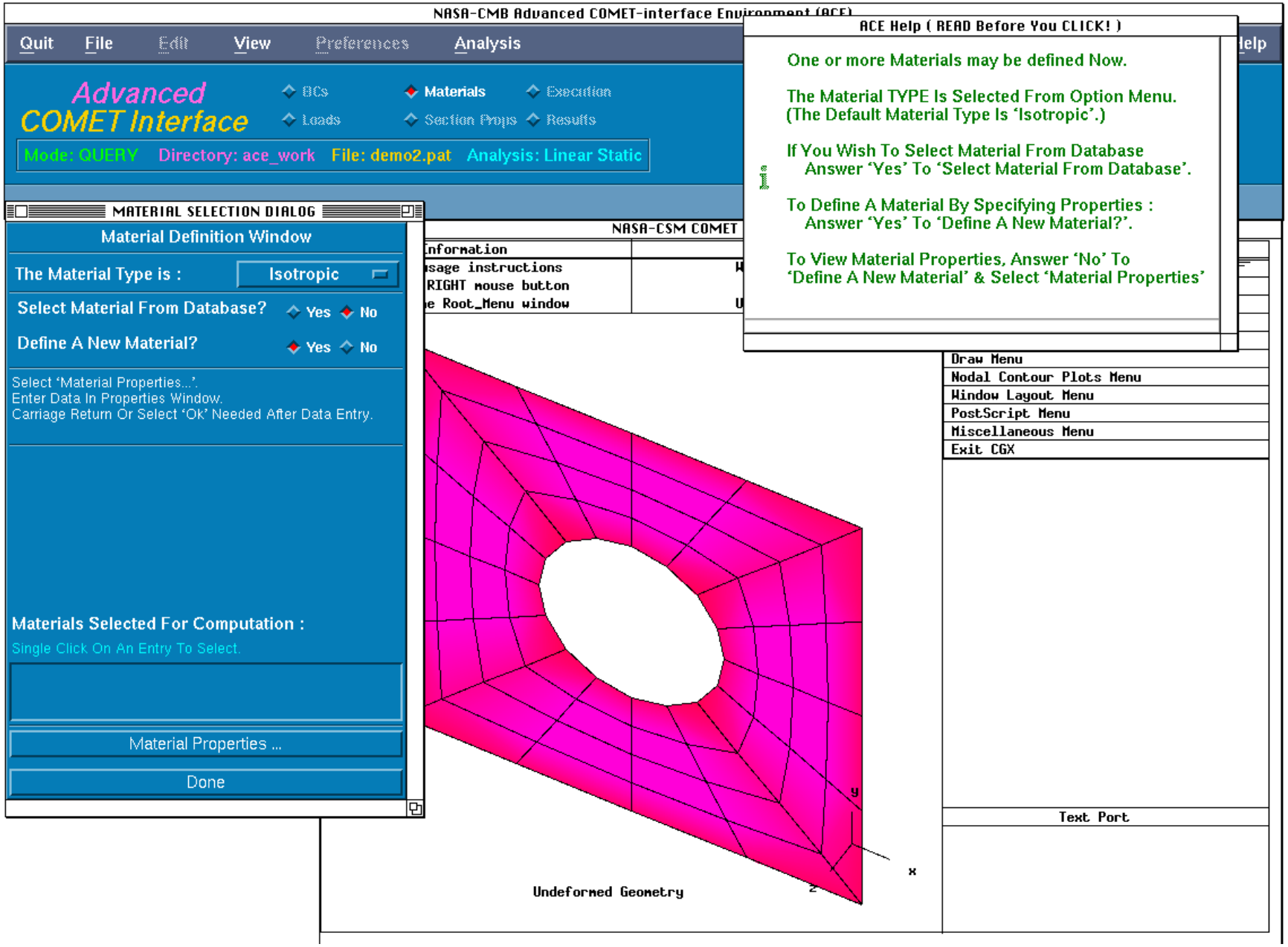
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
One or more Materials may be defined Now.
The Material TYPE Is Selected From Option Menu.
(The Default Material Type Is 'Isotropic'.)
If You Wish To Select Material From Database
Answer 'Yes' To 'Select Material From Database'.
To Define A Material By Specifying Properties :
Answer 'Yes' To 'Define A New Material?'.
To View Material Properties, Answer 'No' To
'Define A New Material' & Select 'Material Properties'
MATERIAL SELECTION DIALOG
Material Definition Window
The Material Type is :
Isotropic
Select Material From Database?
Yes
No
Define A New Material?
Yes
No
Select 'Material Properties...'.
Enter Data In Properties Window.
Carriage Return Or Select 'Ok' Needed After Data Entry.
Materials Selected For Computation :
Single Click On An Entry To Select.
Material Properties ...
Done
NASA-CSM COMET
Information
Usage instructions
RIGHT mouse button
Root_Menu window
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry
y
x
z

Figure 9 (b)

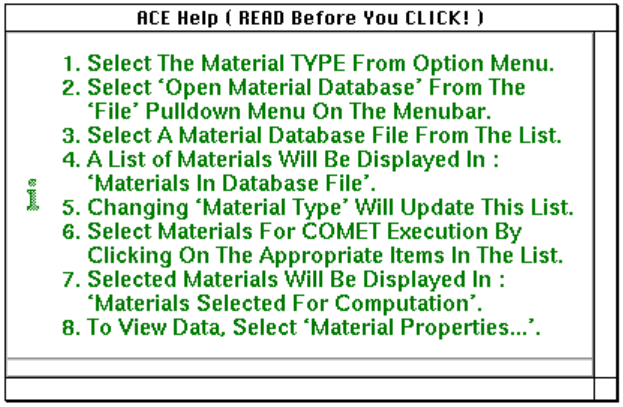

Figure 9(c)

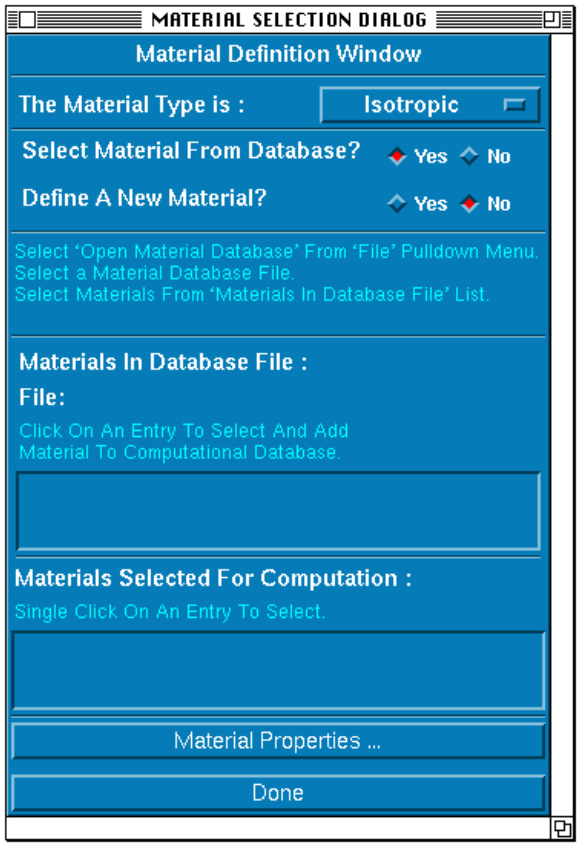

Figure 9(d)

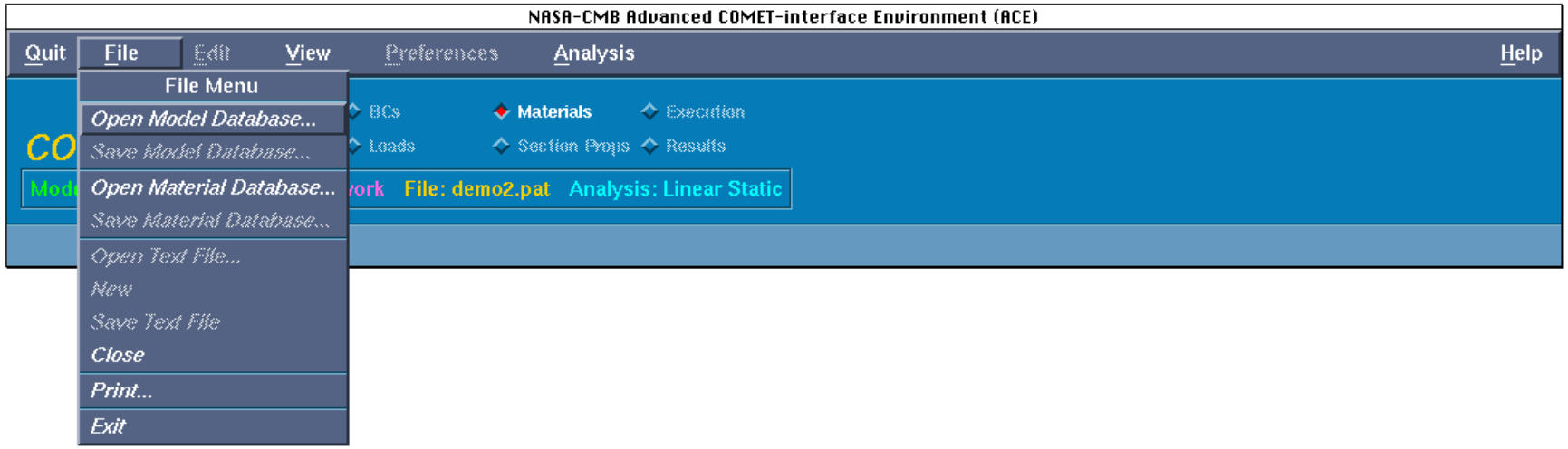
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
File Menu
Open Model Database...
Save Model Database...
Open Material Database...
Save Material Database...
Open Text File...
New
Save Text File
Close
Print...
Exit
BCs
Materials
Execution
Loads
Section Props
Results
File: demo2.pat
Analysis: Linear Static

Figure 9 (e)

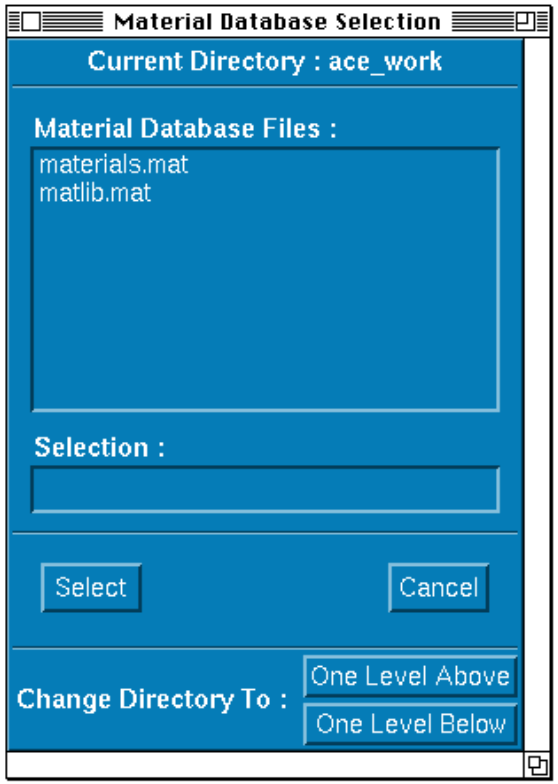

Figure 9(f)

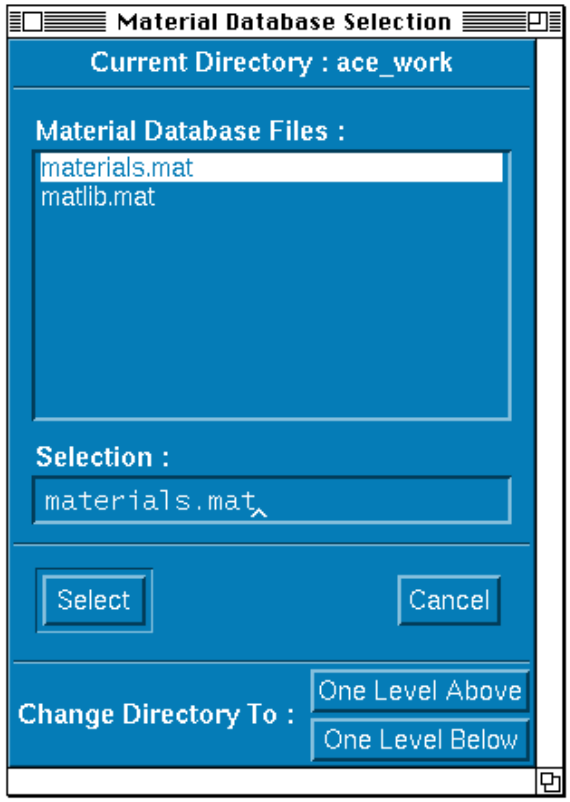

Figure 9(g)

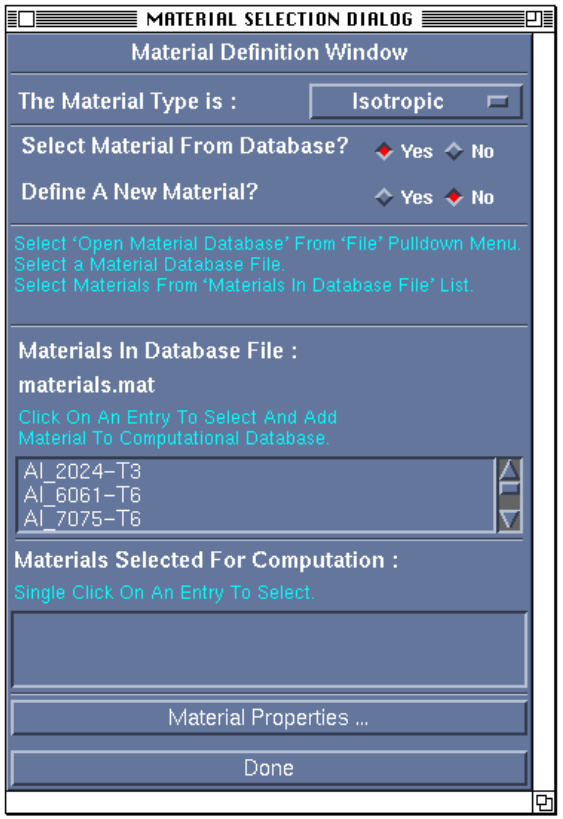

Figure 9(h)

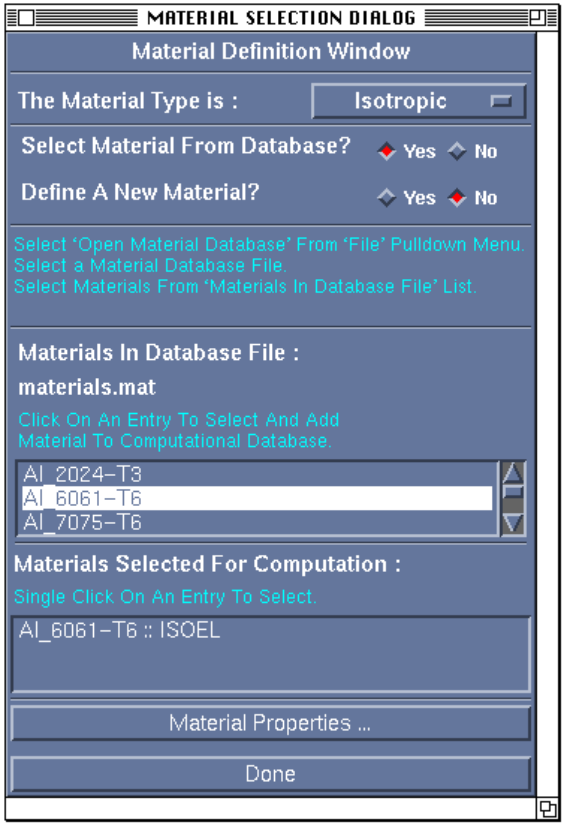

Figure 9(i)

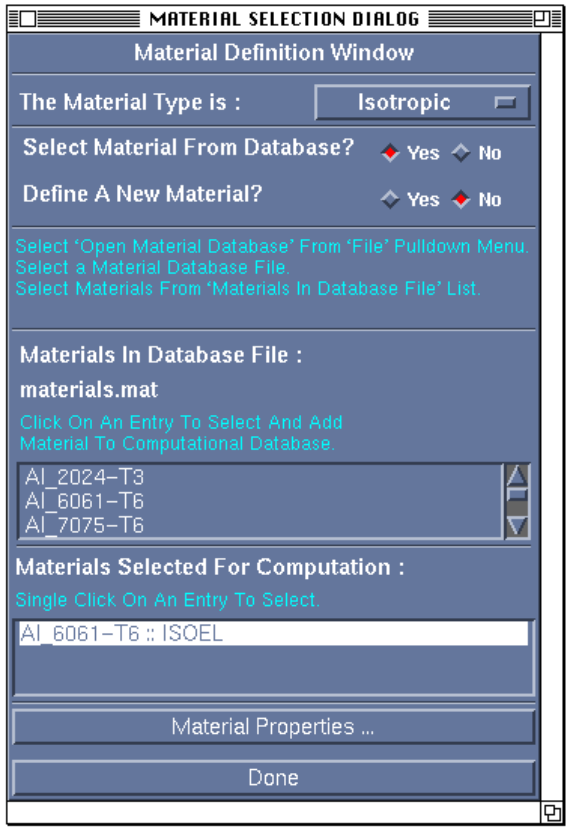

Figure 9(j)

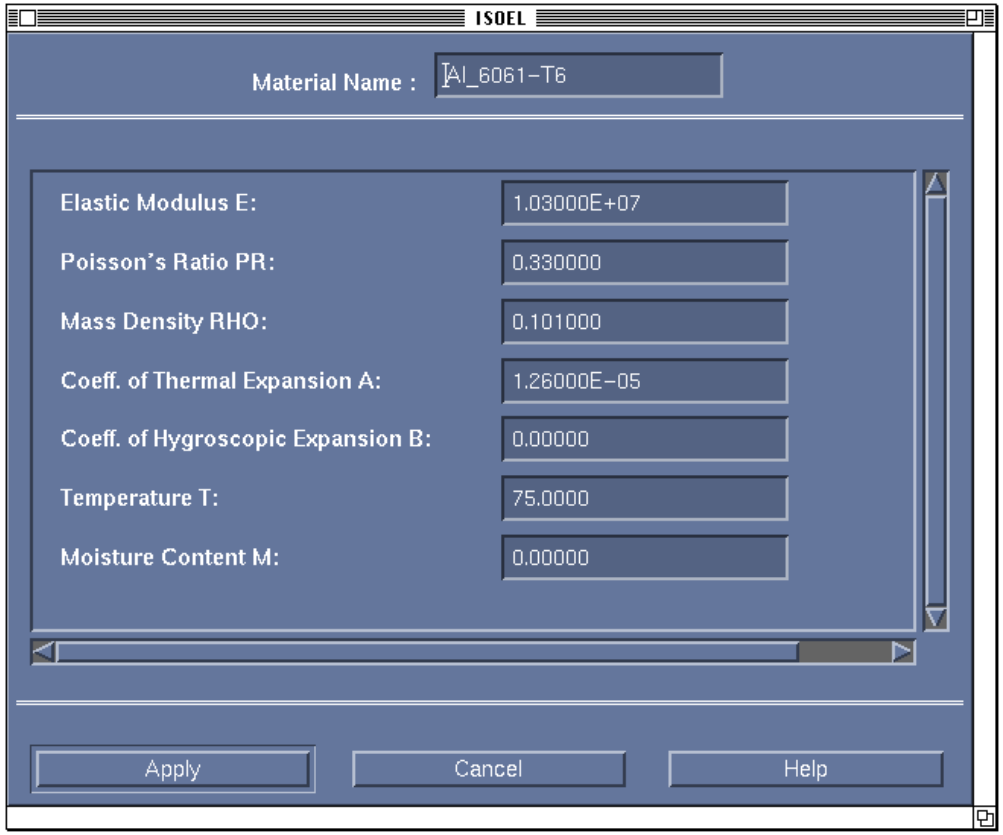

Figure 9(k)

Assigning Material and Section Properties to the Finite Elements

Having selected the materials which will be used in formulating the finite element model, these materials and section properties must be assigned to the previously defined element types. In this example, all EX43 elements will be assigned the same material (Al 6061-T6) and section properties (0.1" thick). The ability to group elements (i.e.; spars, ribs, wing skins, fuselage frames, fuselage skins, floors, bulkheads, etc.) will be included in a later version of ACE. The section property identifiers which must have data defined in this step are associated with the element "pid's" found in the PATRAN neutral file. For the most simple model with only a single section property, only one property identifier must be defined.

The "Help" window (Fig. 10 (a)) informs the user of the number of section types that have been identified (one for "demo1" and two for "demo2") and directs the user to click on "Section Props" in the control area. **Click on the "radio" button next to "Section Props"** (glows red). Selecting the "Section Props" "radio" button displays a "SECTION PROPERTIES" window along with an updated "Help" window (Fig. 10 (b)). The "SECTION PROPERTIES" window has,

- an abbreviated form of the "Help" instructions;
- two scrolled list windows, one to hold the list of section property ids, the other to hold a list of materials selected for computation;
- an option menu to define the section type (Homogeneous, Laminate, Beam, Continuum); and
- two push buttons called "Section Properties..." and "Done", one for assigning material and section properties to the elements and the other to exit.

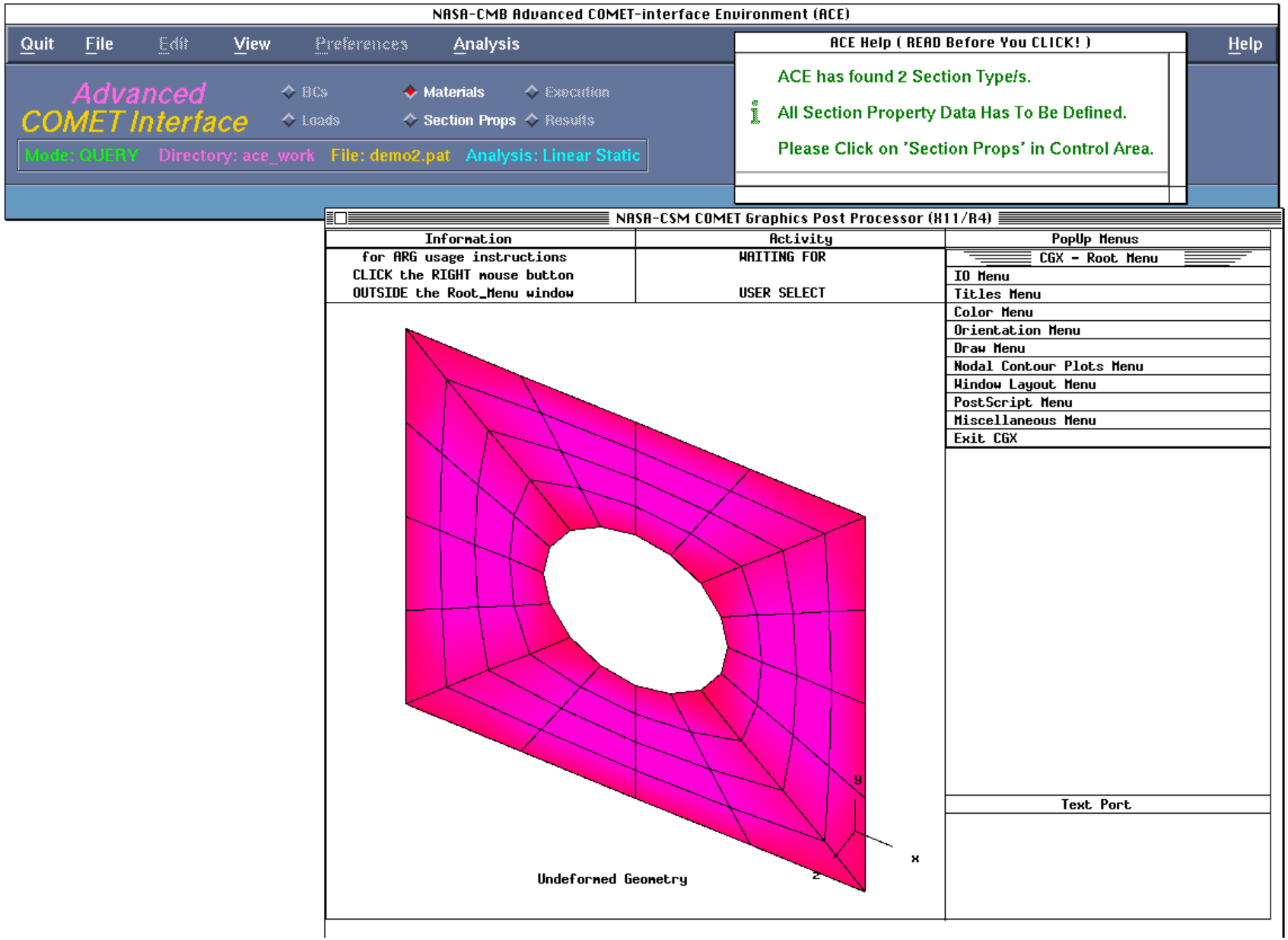
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
ACE has found 2 Section Type/s.
All Section Property Data Has To Be Defined.
Please Click on 'Section Props' in Control Area.
NASA-CSM COMET Graphics Post Processor (X11/R4)
Information
Activity
PopUp Menus
for ARG usage instructions
CLICK the RIGHT mouse button
OUTSIDE the Root_Menu window
WAITING FOR
USER SELECT
CGX - Root Menu
IO Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
y
x
z
Undeformed Geometry

Figure 10(a)

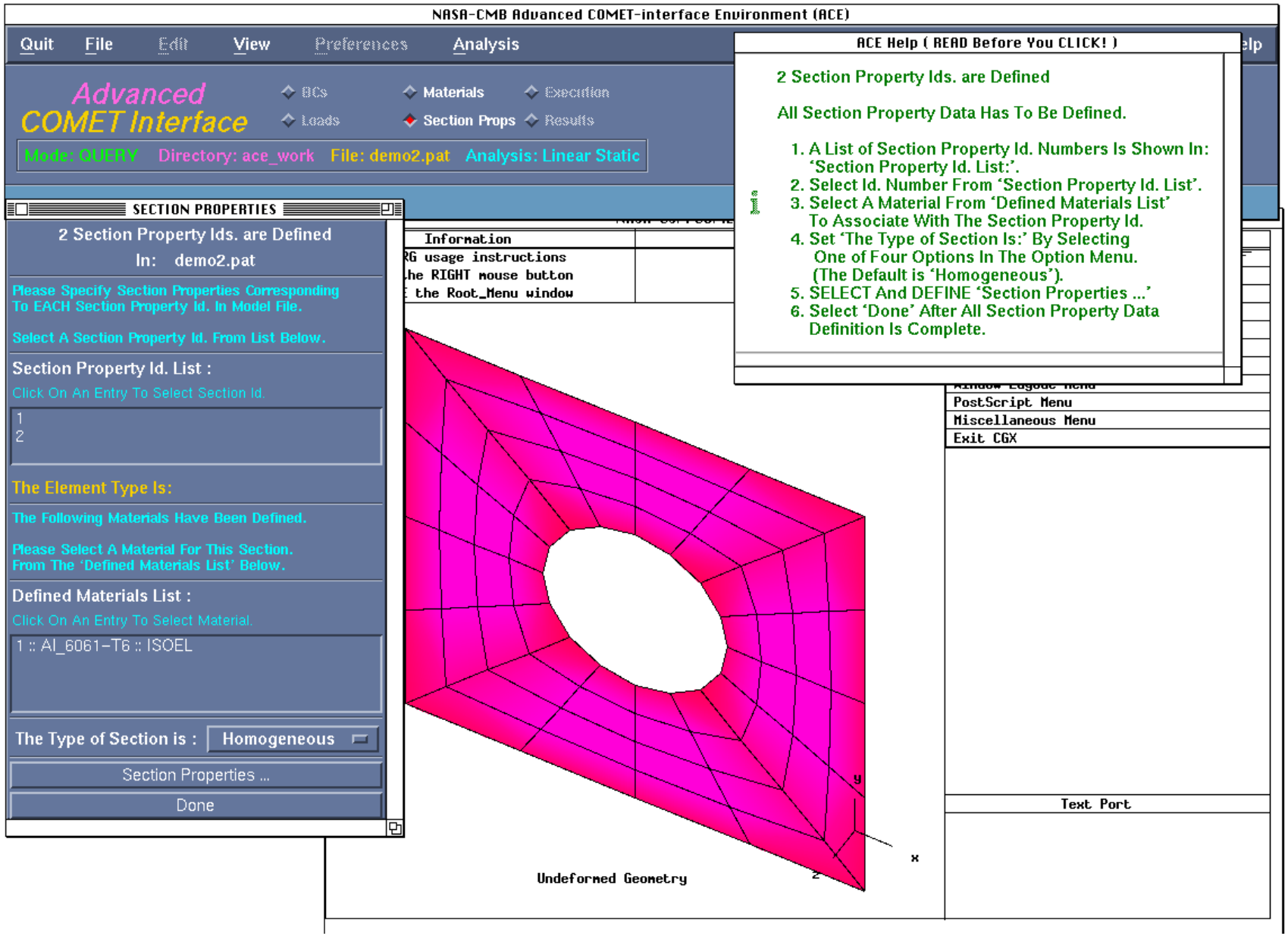
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
SECTION PROPERTIES
2 Section Property Ids. are Defined
In: demo2.pat
Please Specify Section Properties Corresponding
To EACH Section Property Id. In Model File.
Select A Section Property Id. From List Below.
Section Property Id. List :
Click On An Entry To Select Section Id.
1
2
The Element Type Is:
The Following Materials Have Been Defined.
Please Select A Material For This Section.
From The 'Defined Materials List' Below.
Defined Materials List :
Click On An Entry To Select Material.
1 :: Al_6061-T6 :: ISOEL
The Type of Section is :
Homogeneous
Section Properties ...
Done
ACE Help (READ Before You CLICK!)
2 Section Property Ids. are Defined
All Section Property Data Has To Be Defined.
1. A List of Section Property Id. Numbers Is Shown In: 'Section Property Id. List:'.
2. Select Id. Number From 'Section Property Id. List'.
3. Select A Material From 'Defined Materials List' To Associate With The Section Property Id.
4. Set 'The Type of Section Is:' By Selecting One of Four Options In The Option Menu. (The Default is 'Homogeneous').
5. SELECT And DEFINE 'Section Properties ...'
6. Select 'Done' After All Section Property Data Definition Is Complete.
Information
RG usage instructions
the RIGHT mouse button
the Root_Menu window
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry
y
x
z

Figure 10(b)

Figure 10(c)

Figure 10(d)

Figure 10(e)

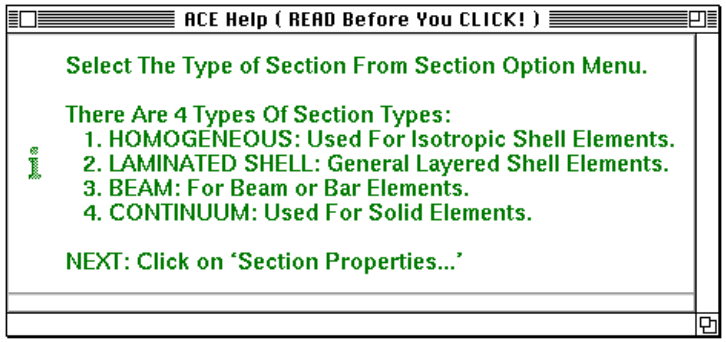

Figure 10(f)

Following the directions given in the "Help" window,

- select a Section ID for definition, **place the cursor on top of a Section ID from the Section Property Id. List and click** (Fig. 10 (c)) which gives the following results:
  - the selected Section ID is highlighted,
  - the associated element type is identified by the label The Element Type Is : (9 Node Quadrilateral in this example),
  - The Type of Section Is**:** "option" menu's default value is set to the default type for that element type (Homogeneous in this example) and
  - the user is instructed to select a material from the "Defined Materials List:" as indicated by Figures 10 (c) and (d).

After highlighting the material selection (Fig. 10 (e)), a new "Help" window (Fig. 10 (f)) advises the user about the section types available and then instructs him/her to **click on the "Section Properties ..." button.**

A data input window (Fig. 10 (g)) titled with the selected section type ("Homogeneous", in this example) is now opened and the section and the material identification numbers are filled in the corresponding fields as described in the "Help" window (Fig. 10 (h)) The "Help" window also lists the remaining input requirements for the section properties of this particular element type .

Instructions for entering or modifying data in these input windows was discussed earlier when describing material properties.

After completing the data input (a thickness of 0.1 in. was used for this example), **click on the "Apply" button.** The user now reverts to the "SECTION PROPERTIES" window and selects another section property ID until the list is exhausted.

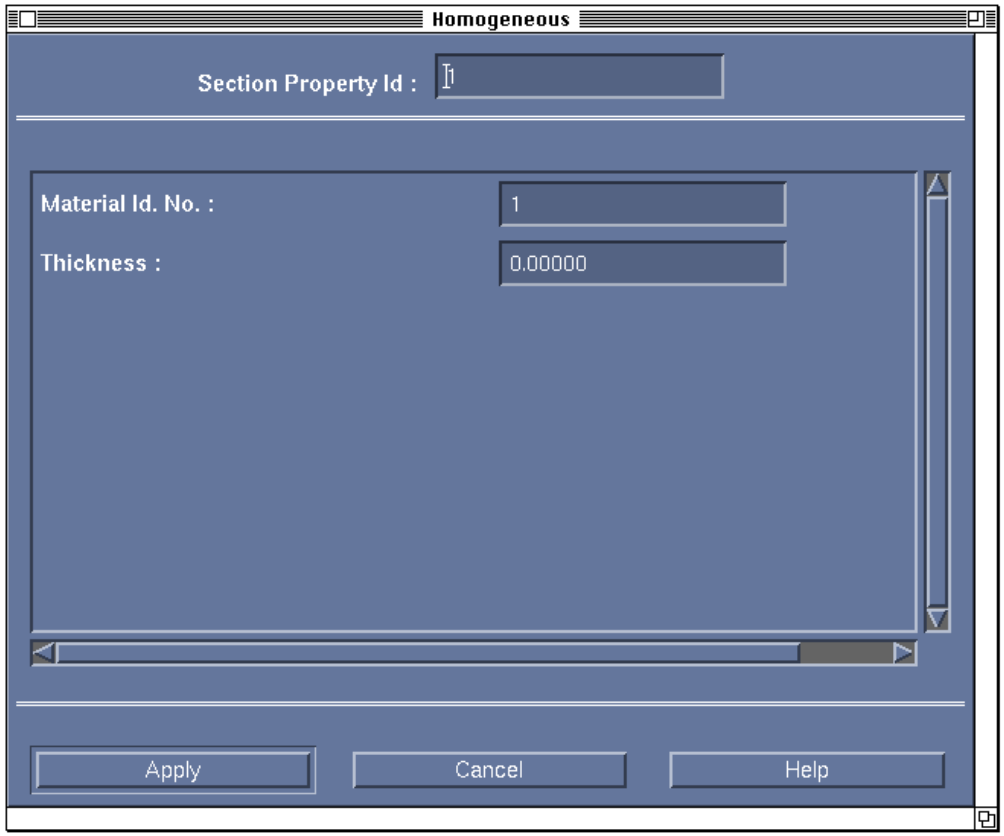

Figure 10(g)

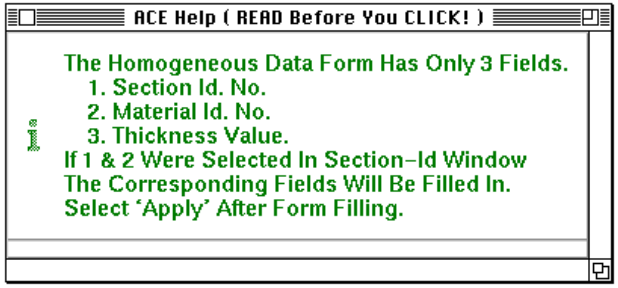

Figure 10(h)

For the "demo2" PATRAN model file, the second section ID represents a Bar/Beam, as indicated by the yellow text, (Fig.. 10 (i)) and "The Type of Section is:" has been set to "Beam". Just like before, **selecting the material** and then **clicking on "Section Properties ..." button** will open the data input window (Fig. 10 (j)) that is titled "Beam" and

- displays the identification numbers of the section and the material properties as previously selected (Fig. 10 (k)) and
- identifies the input requirements required to describe the section properties for this particular element type.

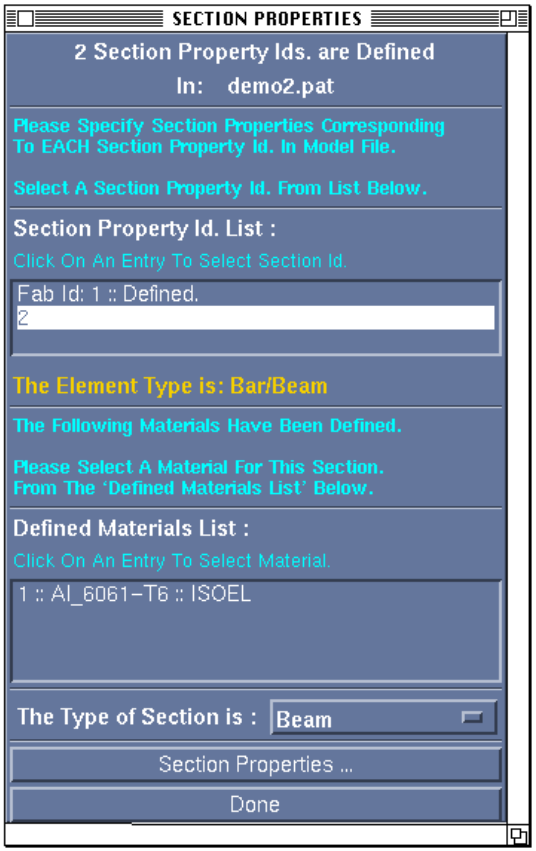

Figure 10(i)

In the next release of ACE, an interactive graphics window will display the beam cross-section, its dimensions and orientation. Although, PATRAN requires that the orientation all 1-dimensional elements (i.e.; beams) be specified at the time of their generation, the next release will show this orientation and permit it's modification. For the present time, it is suggested that $I_{y\text{-}y}$ and $I_{z\text{-}z}$ represent the maximum and minimum principal moments of inertia, respectively, and that the torsional rigidity represents the polar moment of inertia, all moments of inertia with respect to the centroid. Also, the ability of a beam to support torsion is somewhat complex for non-circular sections and this complexity will also be dealt with in a later version. Actually, this later version will accommodate the grouping of one-

dimensional elements such that they can be described as axial rods, torsion bars, shear/moment beams, or full load carrying (axial, shear, torsion, and bending) beams as was mentioned earlier for selecting various plate and/or shell elements (i.e.; shear and membrane/bending elements which would be used to model a wing) that may be needed to define a structure.

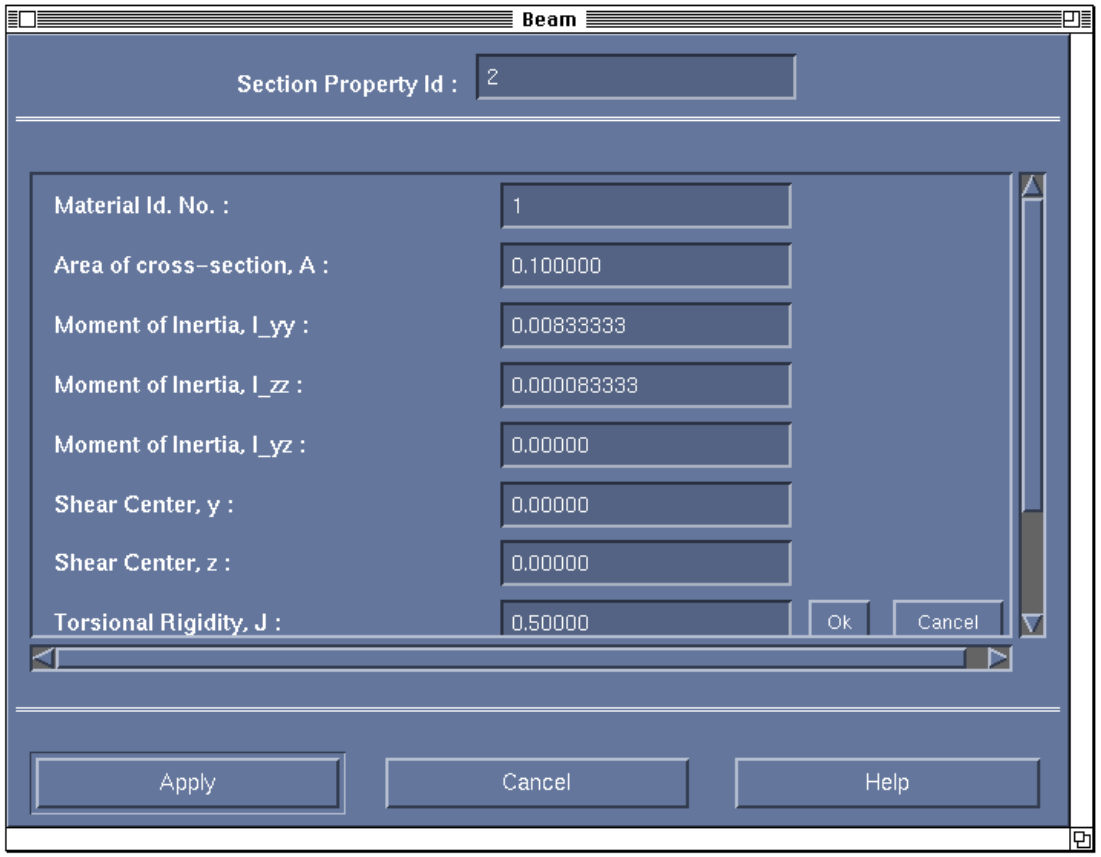

Figure 10(j)

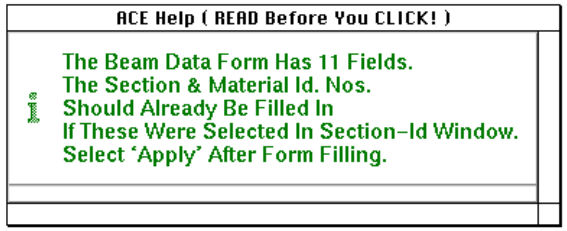

Figure 10(k)

Figure 10 (l)

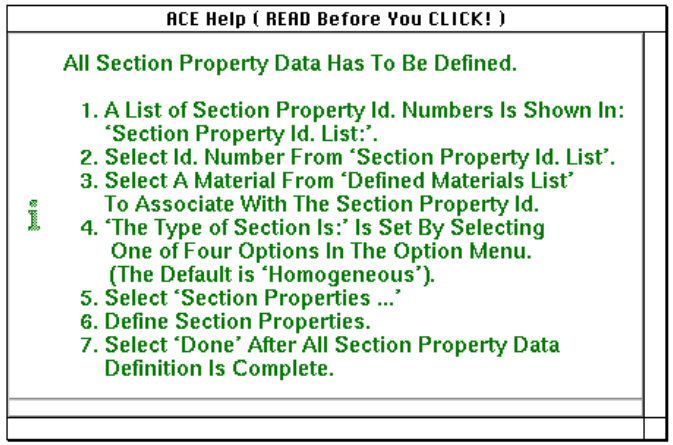

Figure 10 (m)

Having specified the section properties for the beam element, **click on the "Apply" button.** The user is now returned to the "SECTION PROPERTIES" window (Fig. 10 (l)) and a new "Help" window (Fig. 10 (m)) informs the user that if all section property data has been defined for all the Section IDs, **click on "Done.**

Specification of Solution Constraints

The initial size of the stiffness matrix is determined by assigning a full six degrees of freedom to each node of the finite element model. For those models in which the elements connected to the nodes do not represent a stiffness component in one or more of the degrees of freedom, then these particular degrees of freedom should be deleted from the model and the stiffness matrix will be reduced accordingly. For example, a flat panel subjected to an in-plane tensile or compressive load could be modeled with membrane or membrane/bending plate elements, and the degrees of freedom representing the out-of-plane translation and rotation (often called the drilling degree of freedom) about all three axes would be eliminated. In some COMET applications, there is an automatic suppression of the drilling degrees of freedom and more consideration will be given to this in later ACE versions such that the user will be advised of such and offered possible fixes.

After the user has defined the material and cross-sectional properties of the various element types, the screen is refreshed (Fig. 11 (a)) with a new "Help" window that advises the user of whether or not any constraints describing the boundary conditions have been specified. The specification of constraints that represent the boundary conditions of the structure (i.e.; simply supported or fixed edges) can be accomplished at this time or the constraints defined in the imported model file can be modified. For the demonstration problems ("demo1" and "demo2"), the boundary conditions were defined in the PATRAN model and imported via the neutral file.

Due to the predetermined sequencing of actions in the "QUERY" mode, the user is required to **click on the "radio" button adjacent to "BC"** (see "Help" window). The user may review the defined boundary constraints (here, they represent fixed rotational constraints along the edges and translation only in the x-direction) by **clicking on the "Constraint" button in the "Define Constraints" window** (see Fig. 11 (b) and corresponding "Help" window). This will open the "Applied Displacement Dialog" window shown in Figure 11(c).

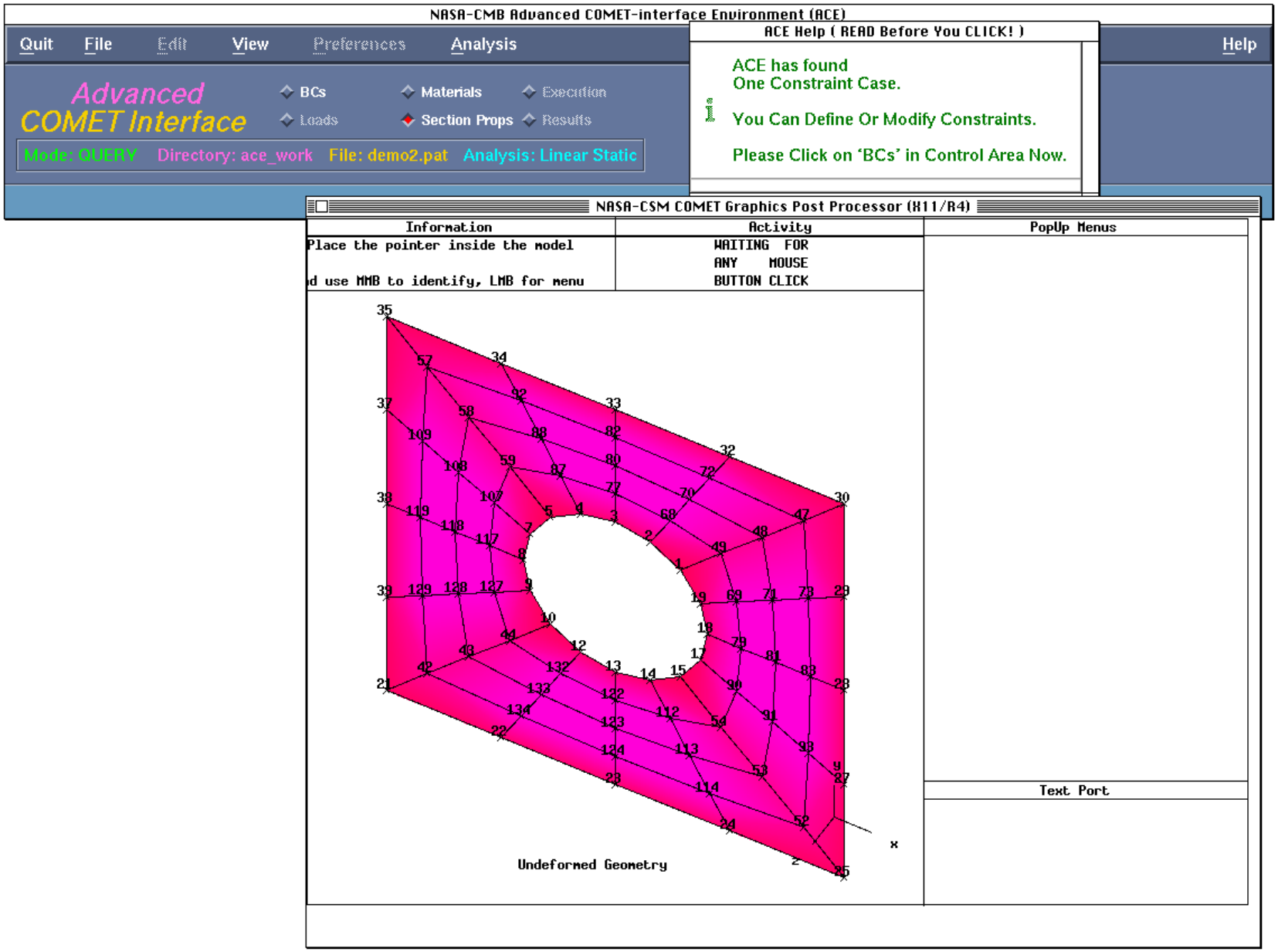
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
ACE has found
One Constraint Case.
You Can Define Or Modify Constraints.
Please Click on 'BCs' in Control Area Now.
NASA-CSM COMET Graphics Post Processor (X11/R4)
Information
Activity
PopUp Menus
Place the pointer inside the model
d use MMB to identify, LMB for menu
WAITING FOR
ANY MOUSE
BUTTON CLICK
Text Port
Undeformed Geometry
x

Figure 11 (a)

For the demonstration problem chosen to illustrate this graphical user interface ("demo-2"), the edges were intended to be totally constrained in the out-of-plane (z) translation and the rotations about the x and y axes. Scrolling down the list of constrained nodes and selecting each node, one at a time, by **placing the cursor on the node number and clicking,** the selected node is shown directly below the list of nodes and its applied constraints are shown below the selected node. The "radio" button for the constrained node is turned on (red glow) and the enforced displacement or rotation is shown to the right. These constraints can be set fixed or free by clicking on the "radio" button and the enforced displacement value can be specified as previously described for data input dialogs. Reviewing the list of constraints will reveal that the edge (nodes 25 and 27 through 30 as shown in Figure 11 (c)), where the in-plane tensile load will be applied, has not been constrained. To restrict displacements along this edge to elongation along the x-axis, we will apply constraints to the interior nodes (nodes 27 through 29) as follows:

- In the bottom data window titled, "Constrain Node Number (and <CR>):" of the "Applied Displacement Dialog" window (Fig. 11 (d)), **enter 27** (see "Help" window as illustrated in Figure 11 (e)).
- Review the constraint setting (Fig. 11 (f)) and **click on "radio" button adjacent to "W", "Θ_x", and "Θ_y"** (red glow of the "radio" button means that particular degree of freedom at node 27 is constrained). If constraints are indicated for other degrees of freedom (i.e.; "U", "V" and/or "Θ_z"), click on the "radio" button adjacent to that degree of freedom (a gray button means unconstrained).
- **Click on the "Add" button** to incorporate the new constrained node to the model. In this example, the enforced displacement at all nodes has been unchanged from the default value of 0.0.
- **Repeat the above three steps for nodes 28 and 29 and then click on the "Done" window.**
- To delete a constrained node from the list, select the node and unconstrain all degrees of freedom by clicking on the red "radio" buttons associated with the node.

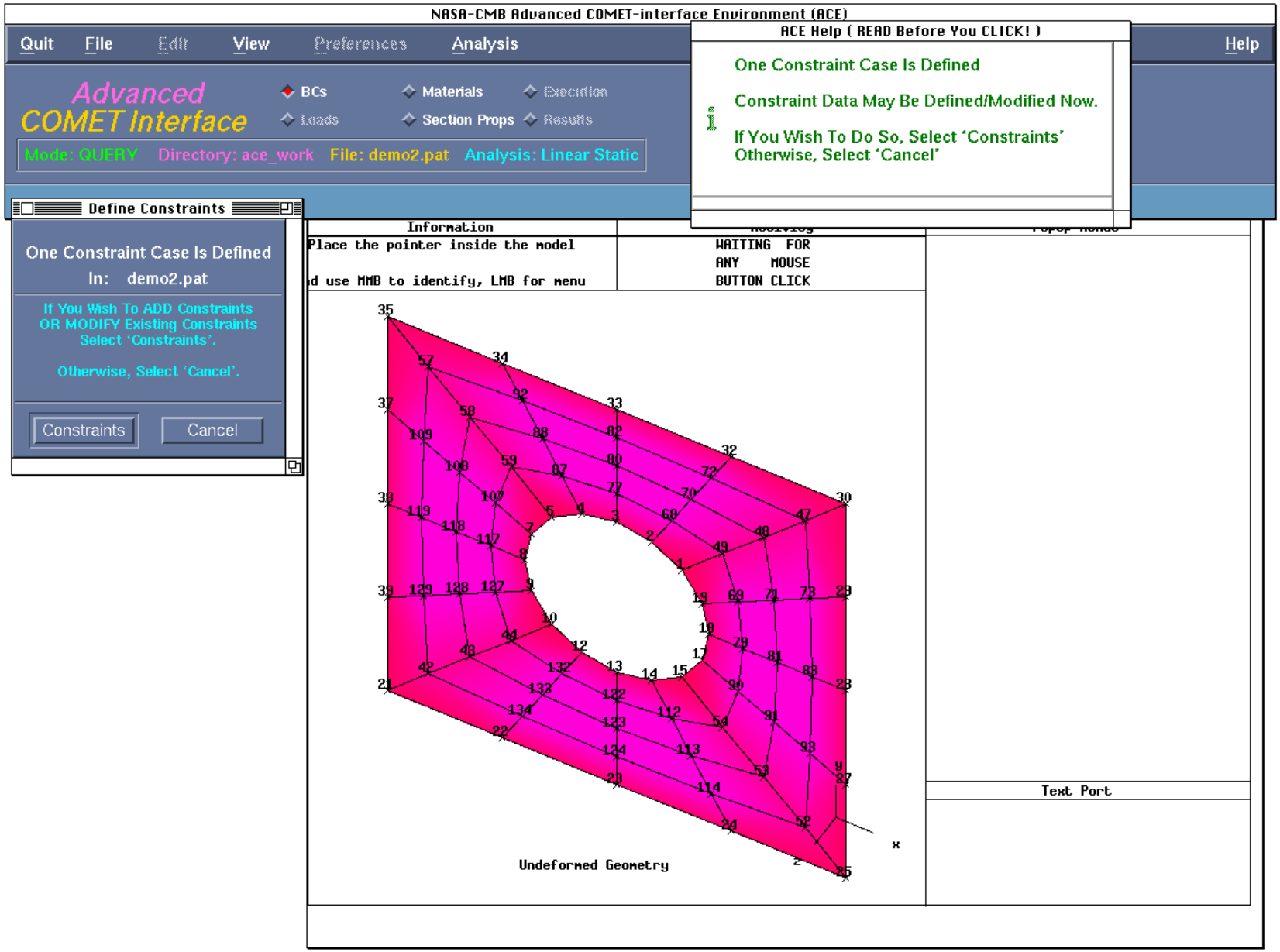
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
One Constraint Case Is Defined
Constraint Data May Be Defined/Modified Now.
If You Wish To Do So, Select 'Constraints'
Otherwise, Select 'Cancel'
Define Constraints
One Constraint Case Is Defined
In: demo2.pat
If You Wish To ADD Constraints
OR MODIFY Existing Constraints
Select 'Constraints'.
Otherwise, Select 'Cancel'.
Constraints
Cancel
Information
Place the pointer inside the model
d use MMB to identify, LMB for menu
WAITING FOR
ANY MOUSE
BUTTON CLICK
Text Port
Undeformed Geometry
x

Figure 11 (b)

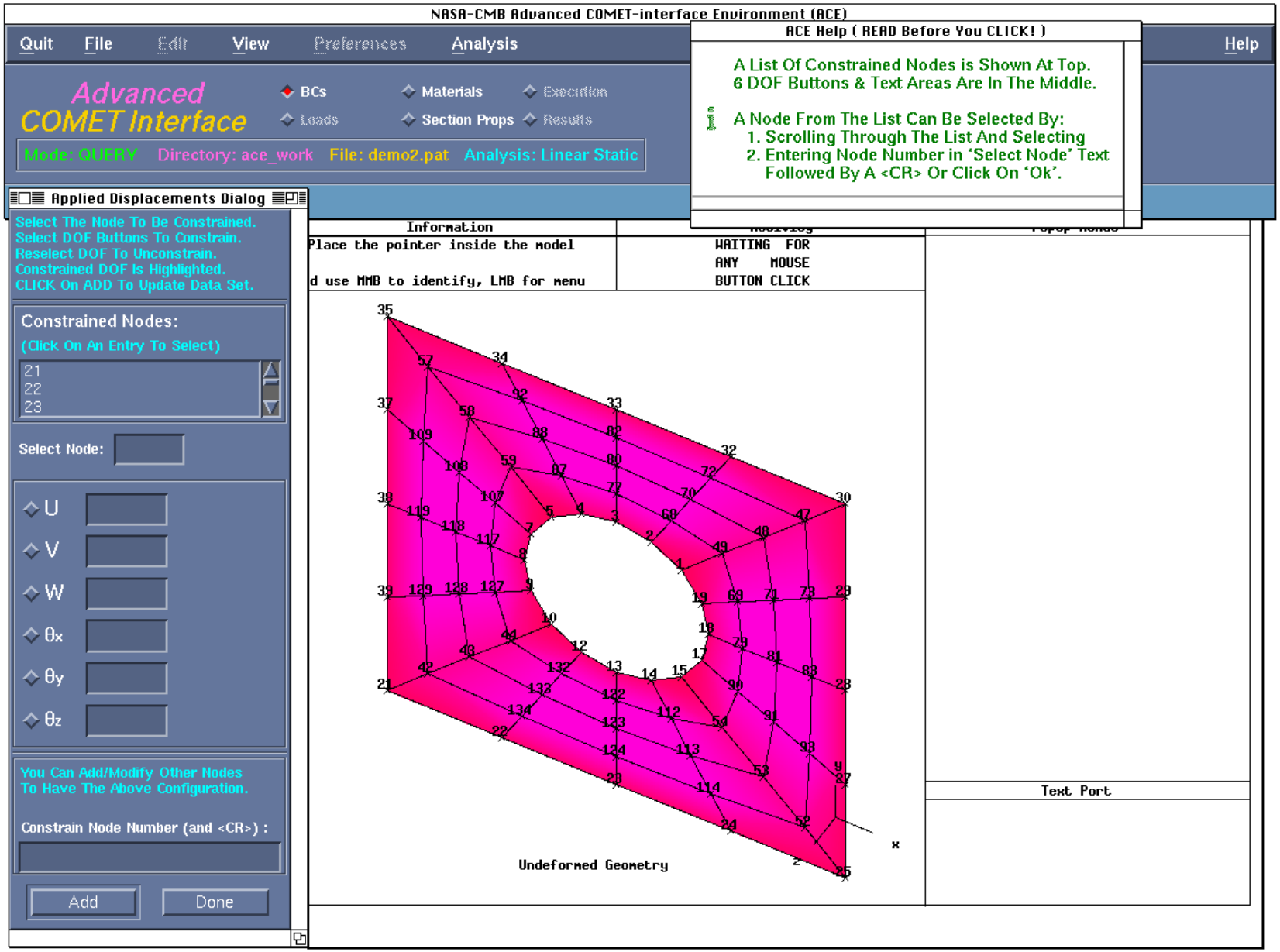
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
A List Of Constrained Nodes is Shown At Top.
6 DOF Buttons & Text Areas Are In The Middle.
A Node From The List Can Be Selected By:
1. Scrolling Through The List And Selecting
2. Entering Node Number in 'Select Node' Text
Followed By A <CR> Or Click On 'Ok'.
Applied Displacements Dialog
Select The Node To Be Constrained.
Select DOF Buttons To Constrain.
Reselect DOF To Unconstrain.
Constrained DOF Is Highlighted.
CLICK On ADD To Update Data Set.
Constrained Nodes:
(Click On An Entry To Select)
21
22
23
Select Node:
U
V
W
θx
θy
θz
You Can Add/Modify Other Nodes
To Have The Above Configuration.
Constrain Node Number (and <CR>) :
Add
Done
Information
Place the pointer inside the model
d use MMB to identify, LMB for menu
WAITING FOR
ANY MOUSE
BUTTON CLICK
Undeformed Geometry
Text Port

Figure 11 (c)

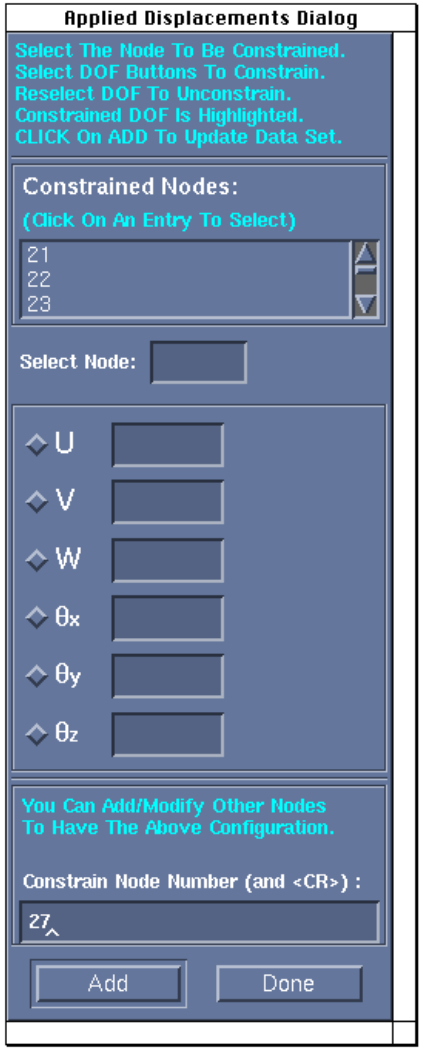

Figure 11 (d)

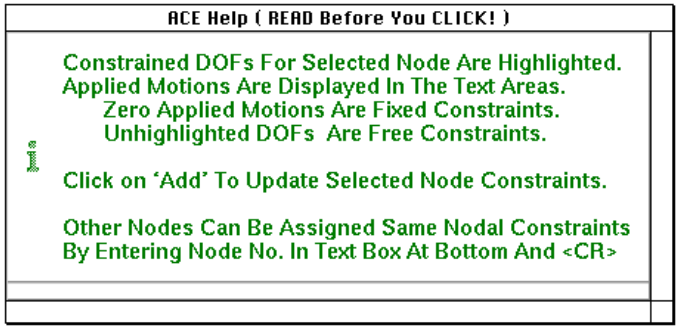

Figure 11 (e)

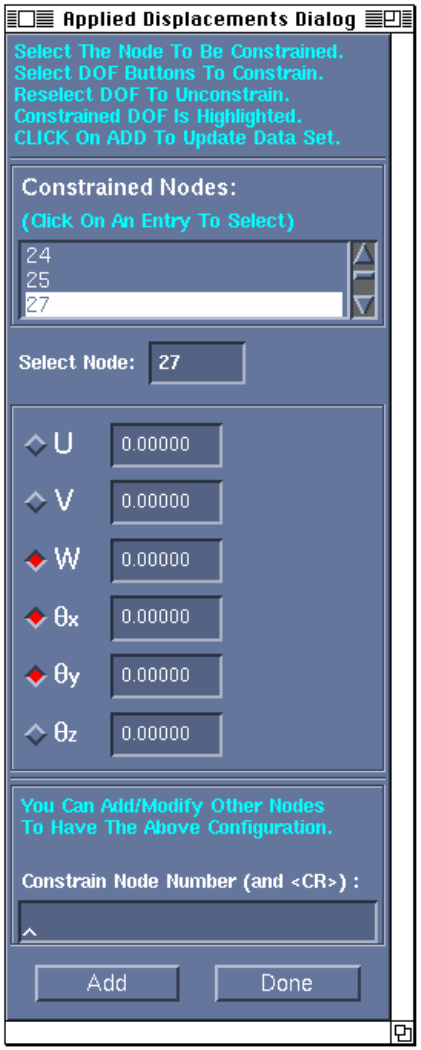
Applied Displacements Dialog
Select The Node To Be Constrained.
Select DOF Buttons To Constrain.
Reselect DOF To Unconstrain.
Constrained DOF Is Highlighted.
CLICK On ADD To Update Data Set.
Constrained Nodes:
(Click On An Entry To Select)
24
25
27
Select Node:
27
U
0.00000
V
0.00000
W
0.00000
θx
0.00000
θy
0.00000
θz
0.00000
You Can Add/Modify Other Nodes
To Have The Above Configuration.
Constrain Node Number (and <CR>) :
Add
Done

Figure 11 (f)

Specification of Applied Loads

As illustrated in Figure 12 (a), the user is now returned to the control area and the "Help" window informs him/her about the existence or non-existence of previous load case(s) defined in PATRAN.

The user is now requested to **click on the "radio" button adjacent to "Loads"** which opens the "Define Loads" window as shown in Figure 12 (b). The procedure for adding new loading conditions or modifying existing loading conditions is identical to the procedure described for boundary conditions.

The "Define Loads" window includes an "Option" menu which allows for the selection of various type of loads. As with all "Option" menus, **place the cursor on top of the "Option" button and depress the mouse button** to display the various types of loads that can be added or modified. While depressing the mouse button, drag the cursor to the desired load type. Only the "Force" option can be selected for this initial version and those shown in gray are planned for later versions. Also, this initial version can only accommodate a single load case but later versions will handle several load cases and combinations of various load cases.

Since a description of this input window is nearly identical to the "Applied Displacement Dialog" window, **click on the "Cancel" button** to move on to the next item in the control area.

This completes the physical description of the finite element model.

Note : The display of the applied forces present in the initial model is possible through interaction with CGx**,** the COMET Graphic processor.

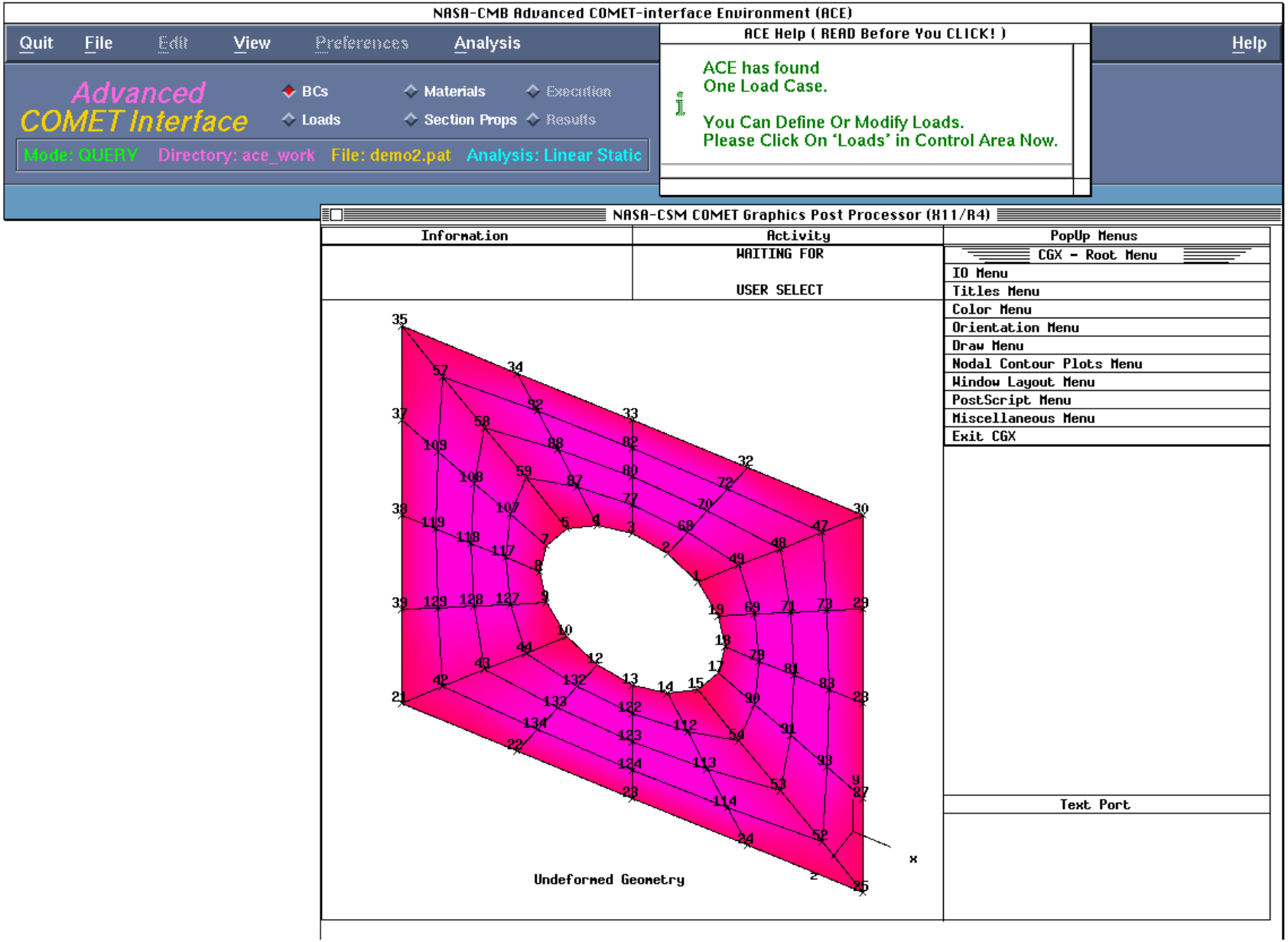
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
ACE has found
One Load Case.
You Can Define Or Modify Loads.
Please Click On 'Loads' in Control Area Now.
NASA-CSM COMET Graphics Post Processor (H11/R4)
Information
Activity
WAITING FOR
USER SELECT
PopUp Menus
CGX - Root Menu
IO Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry
x

Figure 12 (a)

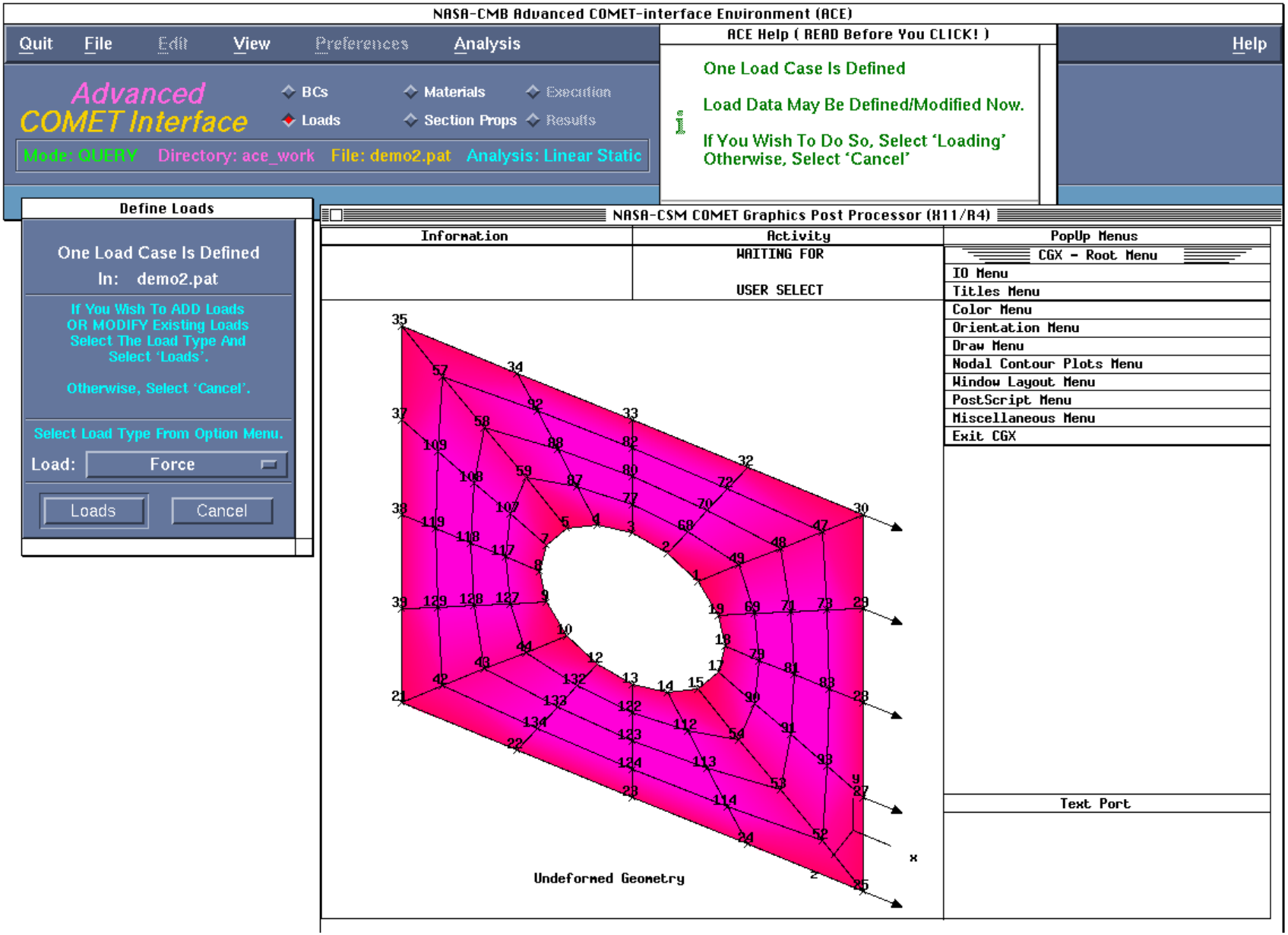
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
One Load Case Is Defined
Load Data May Be Defined/Modified Now.
If You Wish To Do So, Select 'Loading'
Otherwise, Select 'Cancel'
Define Loads
One Load Case Is Defined
In: demo2.pat
If You Wish To ADD Loads
OR MODIFY Existing Loads
Select The Load Type And
Select 'Loads'.
Otherwise, Select 'Cancel'.
Select Load Type From Option Menu.
Load:
Force
Loads
Cancel
NASA-CSM COMET Graphics Post Processor (X11/R4)
Information
Activity
WAITING FOR
USER SELECT
PopUp Menus
CGX - Root Menu
IO Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry
x
y
z

Figure 12 (b)

Specification of the Results

After being returned to the control area (Fig. 13 (a)), the user is requested to **click on "Preferences"** on the menu bar and **select "Analysis ..."** from the pull down menu (Fig. 13 (b)). "Preferences" allows the user to select the type and format of the results and the method for displaying and/or reporting same. Only the "Analysis" option is operational in this initial version.

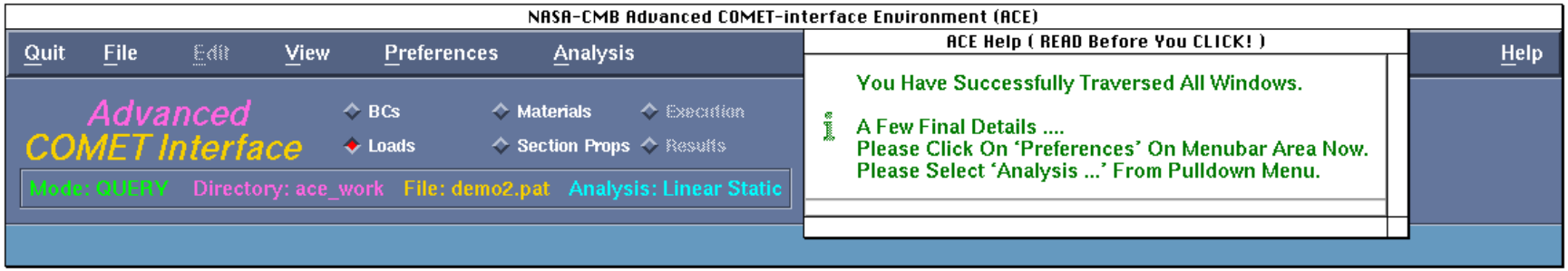

Figure 13 (a)

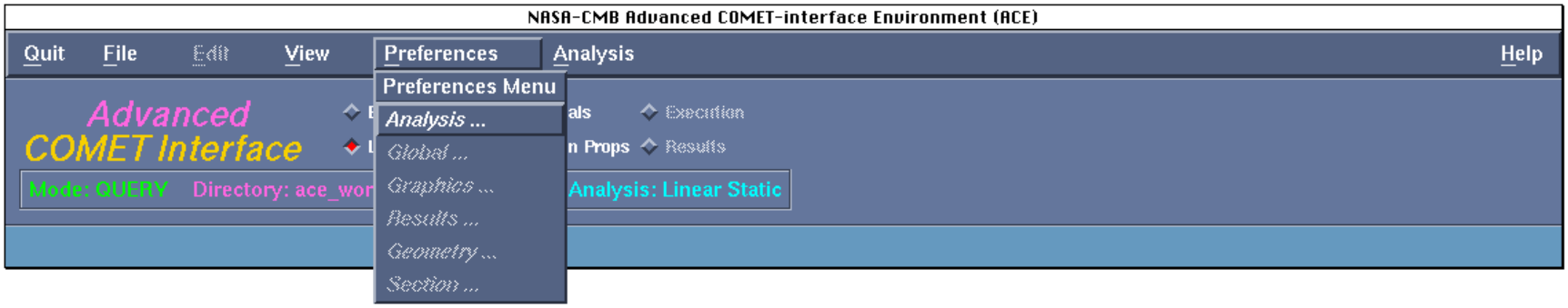

Figure 13 (b)

Selecting the "Analysis ..." option opens the "LINEAR STATIC ANALYSIS" window and corresponding "Help" window shown in Fig. 13 (c). The "LINEAR STATIC ANALYSIS" window makes direct queries of the user which can be answered with a "Yes" or "No". The chosen response will be highlighted with a red glow.

Answering "Yes" to question 1 will cause additional queries to be displayed as shown in Fig. 13 (d). These questions relate to the coordinate system of the computed stresses (i.e.; element or global axes system) and their location (i.e.; nodes, centroids, or integration points). Future versions will allow for the selections of stresses that correspond to various failure theories (e.g.., von Mises, maximum shear stress, etc. for the isotropic materials and Tsai-Wu, Hoffman, etc. for the orthotropic materials).

Once the type and format of the requested output is specified, **click on the "ACCEPT" button**. This completes the development of a COMET runstream.

Note: Placing the cursor and clicking on the questions will provide help on that question in the "Help" window.

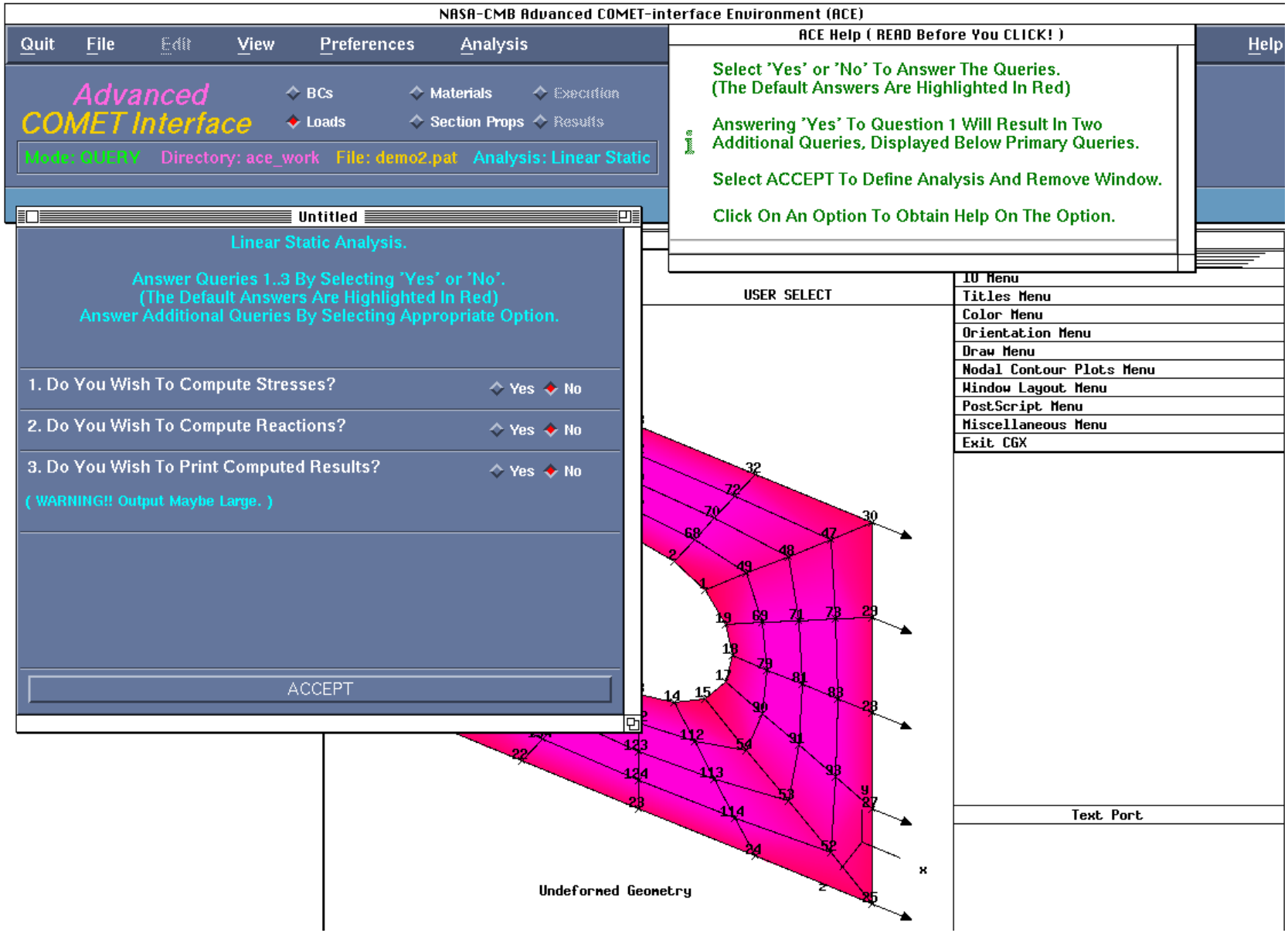
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
Select 'Yes' or 'No' To Answer The Queries.
(The Default Answers Are Highlighted In Red)
Answering 'Yes' To Question 1 Will Result In Two
Additional Queries, Displayed Below Primary Queries.
Select ACCEPT To Define Analysis And Remove Window.
Click On An Option To Obtain Help On The Option.
Untitled
Linear Static Analysis.
Answer Queries 1..3 By Selecting 'Yes' or 'No'.
(The Default Answers Are Highlighted In Red)
Answer Additional Queries By Selecting Appropriate Option.
1. Do You Wish To Compute Stresses?
Yes
No
2. Do You Wish To Compute Reactions?
Yes
No
3. Do You Wish To Print Computed Results?
Yes
No
(WARNING!! Output Maybe Large.)
ACCEPT
USER SELECT
IU Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry

Figure 13 (c)

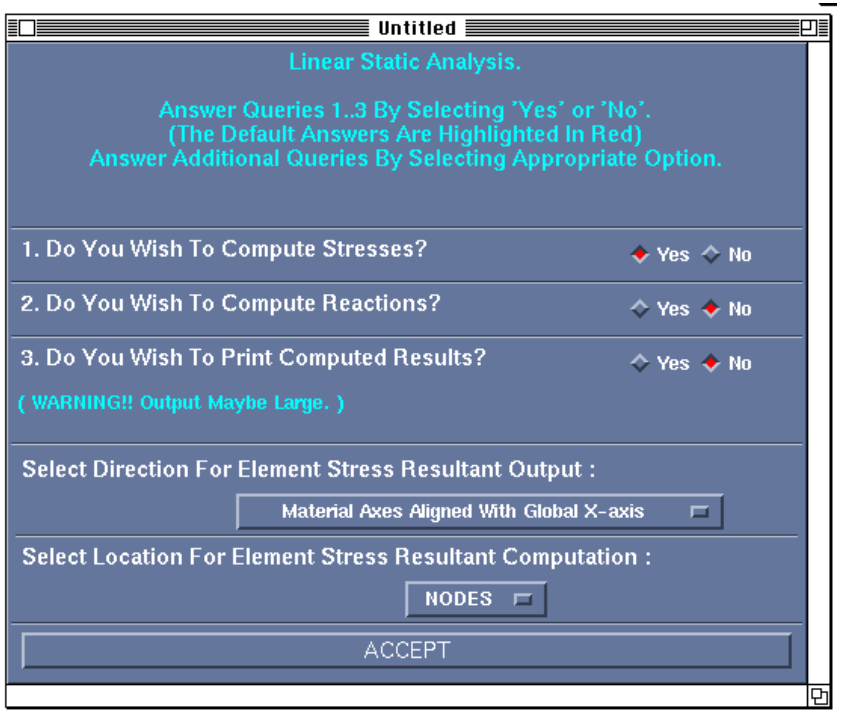

Figure 13 (d)

Execution

Now that the runstream has been developed and accepted, the user is returned to the control area and asked to **click on the "Execution" button** (Fig. 14 (a)). The "Model Execution Dialog" window is now opened, as shown in Figure 14 (b), and the user is requested to review his/her runstream specifications and then **click on the large "Execution" button**. The user is advised (Figure 14 (c)) that the COMET runstream is being assembled and immediate execution will follow. The runstream file is created in the working directory with the filename based on the model input file name and the analysis type with the extension ".com" appended. It is executed as a background job, and after execution is complete, the user is returned to the ACE control area. The log file from the analysis execution is in the current directory with the same root filename as the runstream and the extension of ".log". Both the runstream file and the log file are text files which may be viewed, edited, and printed outside of ACE, if desired.

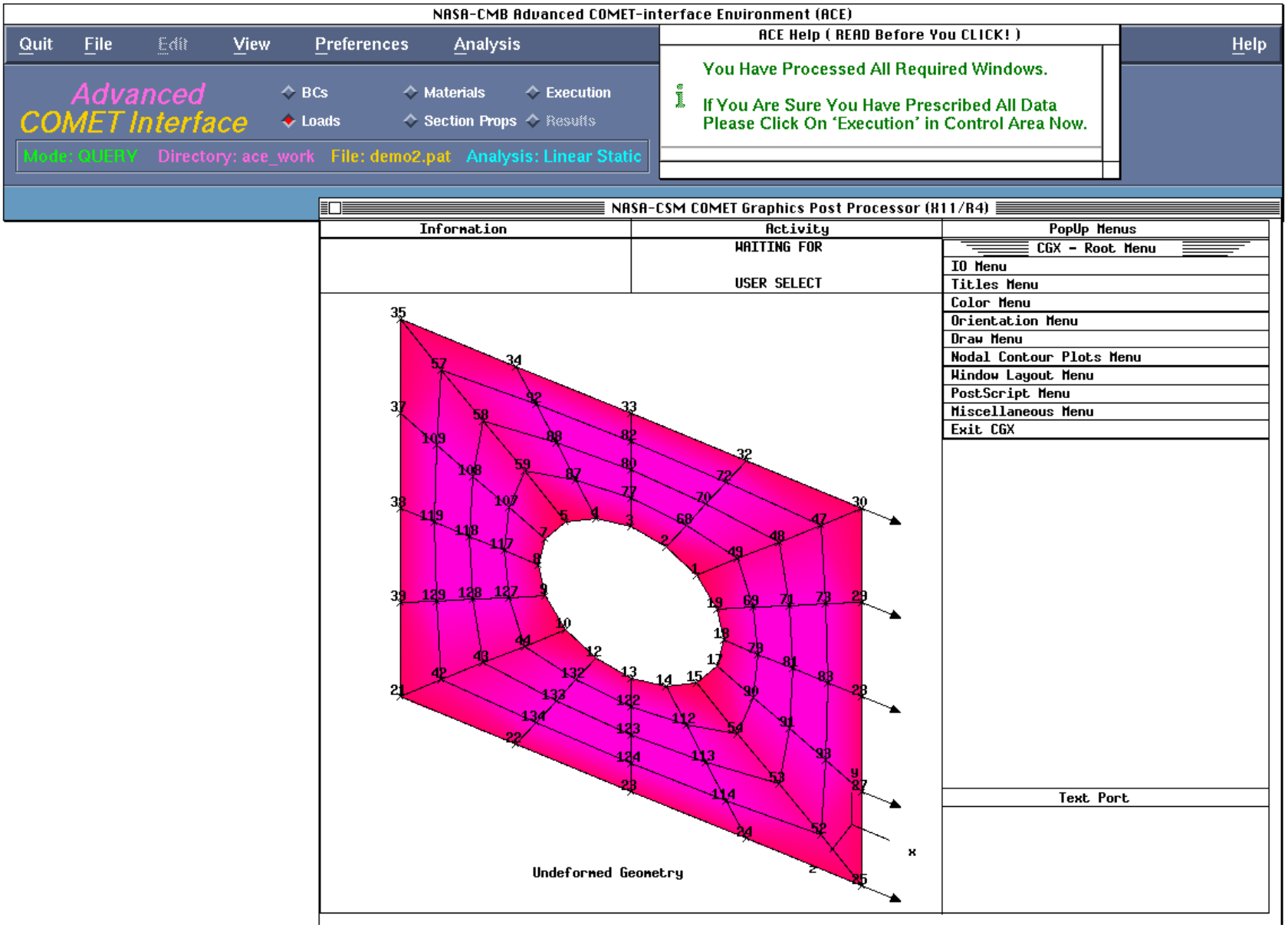
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
You Have Processed All Required Windows.
If You Are Sure You Have Prescribed All Data
Please Click On 'Execution' in Control Area Now.
NASA-CSM COMET Graphics Post Processor (X11/R4)
Information
Activity
WAITING FOR
USER SELECT
PopUp Menus
CGX - Root Menu
IO Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry
x
y
z

Figure 14 (a)

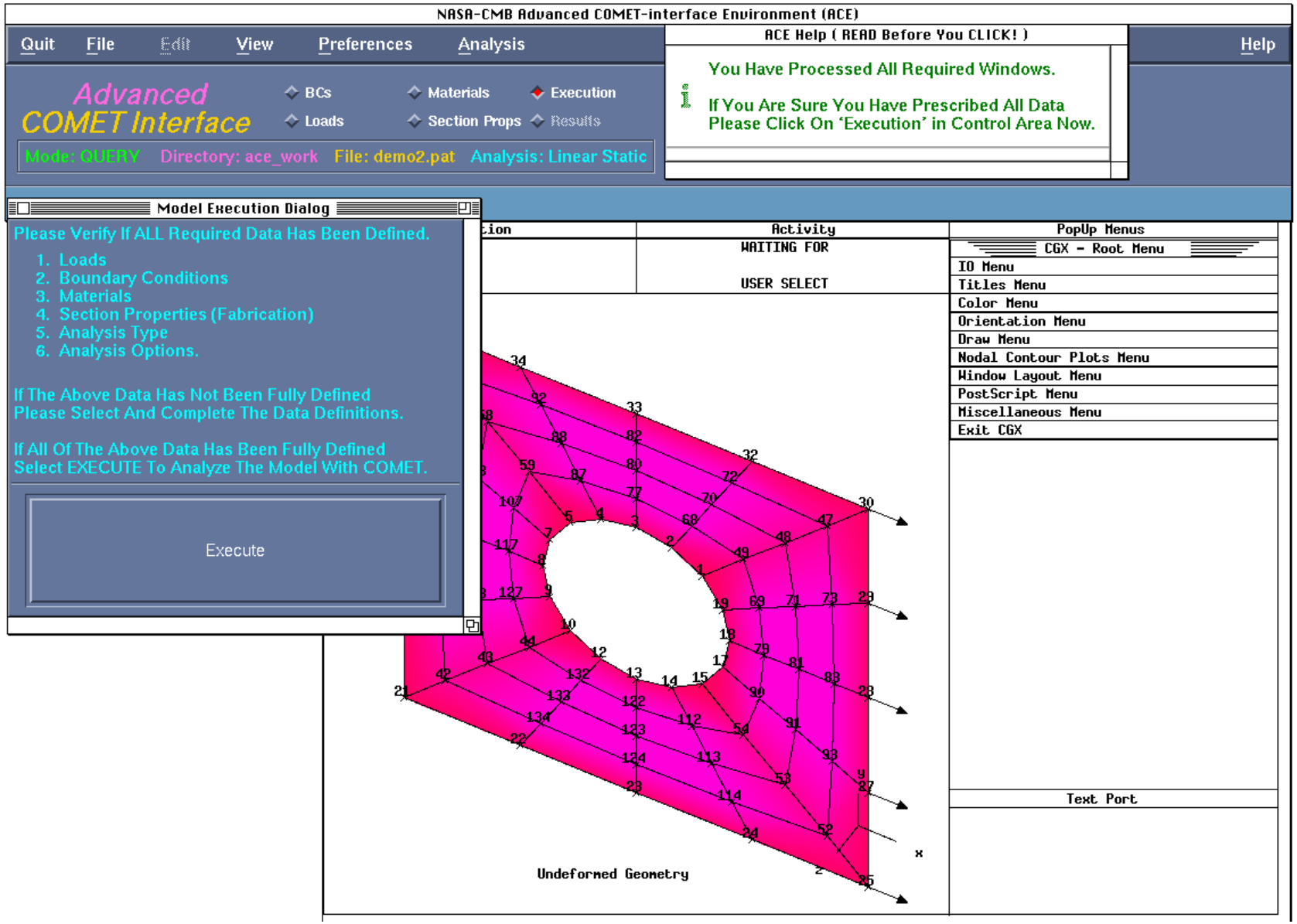
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
ACE Help (READ Before You CLICK!)
You Have Processed All Required Windows.
If You Are Sure You Have Prescribed All Data
Please Click On 'Execution' in Control Area Now.
Model Execution Dialog
Please Verify If ALL Required Data Has Been Defined.
1. Loads
2. Boundary Conditions
3. Materials
4. Section Properties (Fabrication)
5. Analysis Type
6. Analysis Options.
If The Above Data Has Not Been Fully Defined
Please Select And Complete The Data Definitions.
If All Of The Above Data Has Been Fully Defined
Select EXECUTE To Analyze The Model With COMET.
Execute
Activity
WAITING FOR
USER SELECT
PopUp Menus
CGX - Root Menu
IO Menu
Titles Menu
Color Menu
Orientation Menu
Draw Menu
Nodal Contour Plots Menu
Window Layout Menu
PostScript Menu
Miscellaneous Menu
Exit CGX
Text Port
Undeformed Geometry

Figure 14 (b)

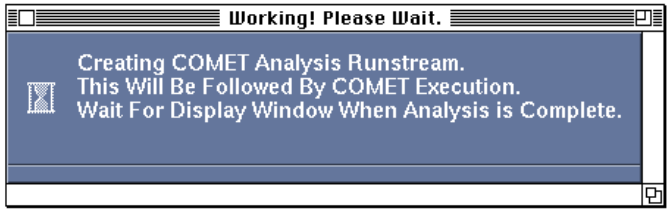

Figure 14 (c)

Evaluating and Reporting the Results

Presently, the "Results" part of ACE has not been implemented. However, graphical views of the results can be obtained via CGx which has been interfaced with the COMET results database. As previously stated, directions for using CGx are not intended to be part of this users' manual. However, later versions of ACE will include a self-contained capability for generating various graphical presentations of data (i.e.; contours, x-y, deformed and undeformed, etc.) and report forms (i.e.; tables).

For the included demonstration problem ("demo2"), a CGx generated contour plot of the normal stresses in the x-direction is shown in Figure 15. All the results requested in "Preferences" are given in the output file called "demo2_lstatic.log". In general, the name of this file identifies the name of the model ("demo2") and the type of analysis (linear static). Since this file is a log of the execution of the ACE generated COMET runstream, it could be reviewed in case of possible execution errors (Heaven Forbid!!!).

Performing Another Analysis or Quitting

As indicated in the "Help" window of Figure (15), a new analysis can begin by selecting a model file from the "File" pull down menu or the interface can be exited by **clicking on "Quit" in the control area**. Another "pop-up" window (Fig. 16) will appear in the upper right hand corner and gives the user another chance to decide. **Let us quit writing and demonstrating by clicking a solid "Yes"!!!!**

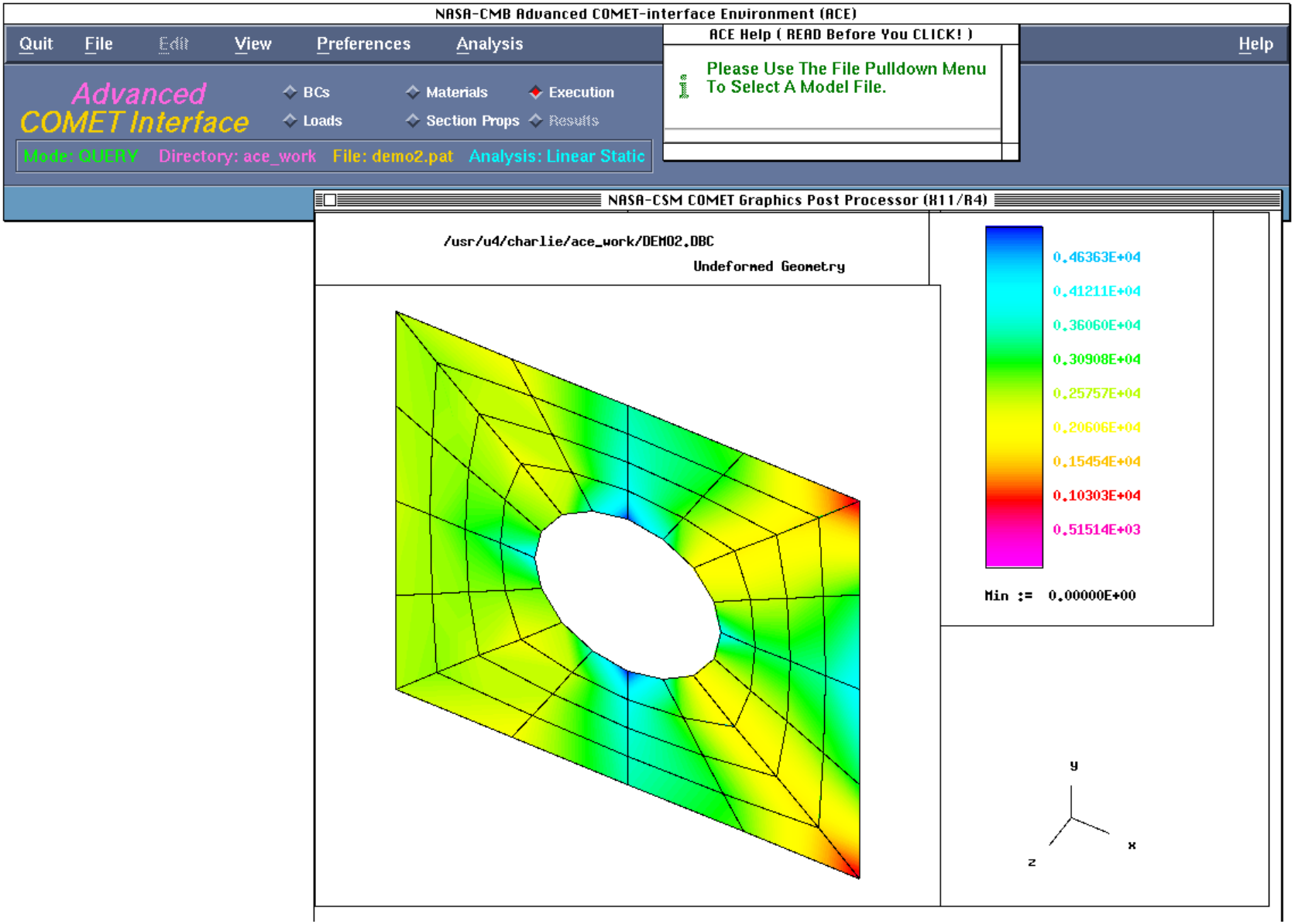
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Help
ACE Help (READ Before You CLICK!)
Please Use The File Pulldown Menu
To Select A Model File.
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
NASA-CSM COMET Graphics Post Processor (X11/R4)
/usr/u4/charlie/ace_work/DEMO2.DBC
Undeformed Geometry
0.46363E+04
0.41211E+04
0.36060E+04
0.30908E+04
0.25757E+04
0.20606E+04
0.15454E+04
0.10303E+04
0.51514E+03
Min := 0.00000E+00
y
x
z

Figure 15

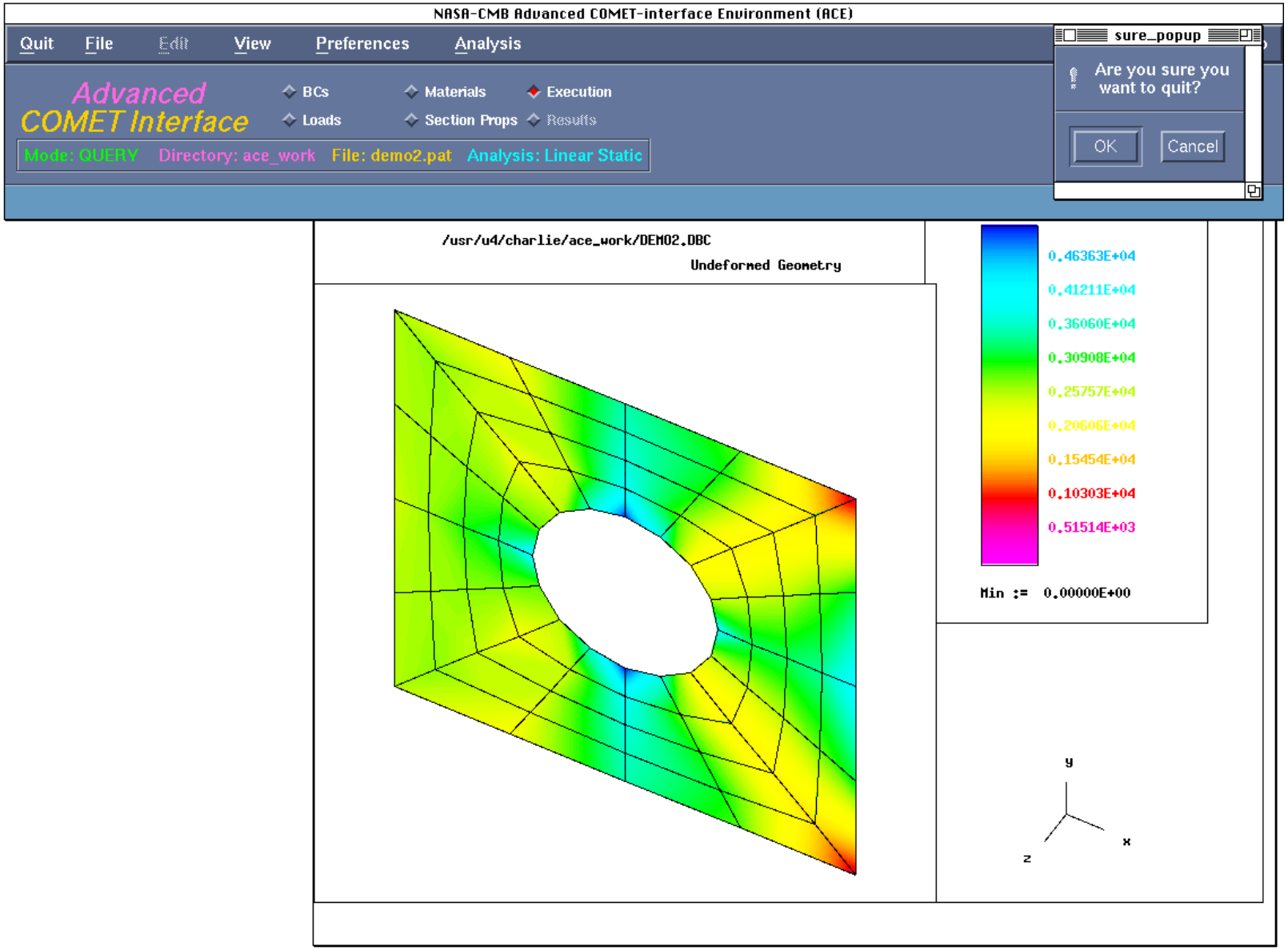
NASA-CMB Advanced COMET-interface Environment (ACE)
Quit
File
Edit
View
Preferences
Analysis
Advanced
COMET Interface
BCs
Materials
Execution
Loads
Section Props
Results
Mode: QUERY
Directory: ace_work
File: demo2.pat
Analysis: Linear Static
sure_popup
Are you sure you want to quit?
OK
Cancel
/usr/u4/charlie/ace_work/DEMO2.DBC
Undeformed Geometry
0.46363E+04
0.41211E+04
0.36060E+04
0.30908E+04
0.25757E+04
0.20606E+04
0.15454E+04
0.10303E+04
0.51514E+03
Min := 0.00000E+00
y
x
z

Figure 16