

OVERVIEW OF ANSYS STRUCTURE AND VISUAL CAPABILITIES

CHAPTER OUTLINE

2.1	INTRODUCTION	37
2.2	STARTING THE PROGRAM	38
2.3	PREPROCESSING STAGE	43
2.4	SOLUTION STAGE	49
2.5	POSTPROCESSING STAGE	50

2.1 INTRODUCTION

ANSYS is a general-purpose finite-element modeling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear), heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

In general, a finite-element solution may be broken into the following three stages.

(1) *Preprocessing: defining the problem*

The major steps in preprocessing are (i) define keypoints/lines/areas/volumes, (ii) define element type and material/geometric properties, and (iii) mesh lines/areas/ volumes as required.

The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, axisymmetric, and 3D.

(2) *Solution: assigning loads, constraints, and solving*

Here, it is necessary to specify the loads (point or pressure), constraints (translational and rotational), and finally solve the resulting set of equations.

(3) *Postprocessing: further processing and viewing of the results*

In this stage one may wish to see (i) lists of nodal displacements, (ii) element forces and moments, (iii) deflection plots, and (iv) stress contour diagrams or temperature maps.

2.2 STARTING THE PROGRAM

2.2.1 PRELIMINARIES

There are two methods to use ANSYS. The first is by means of the graphical user interface or GUI. This method follows the conventions of popular Windows and X-Windows based programs. The GUI method is exclusively used in this book.

The second is by means of command files. The command file approach has a steeper learning curve for many, but it has the advantage that the entire analysis can be described in a small text file, typically in less than 50 lines of commands. This approach enables easy model modifications and minimal file space requirements.

The ANSYS environment contains two windows: the Main Window and an Output Window. Within the Main Window there are five divisions (see Figure 2.1):

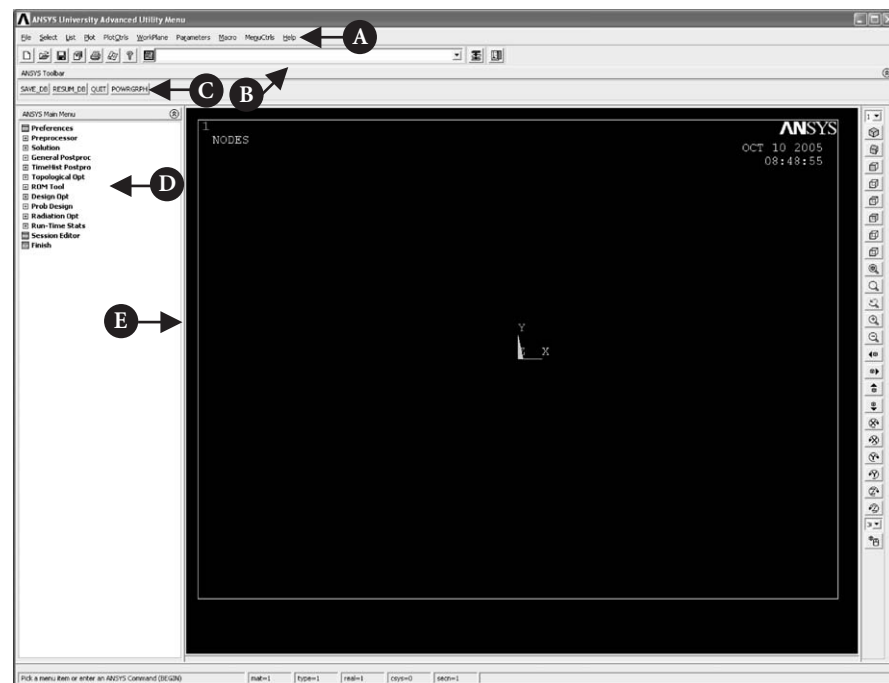


Figure 2.1 Main window of ANSYS.

- (1) *Utility Menu*: The Utility Menu [A] contains functions that are available throughout the ANSYS session, such as file controls, selections, graphic controls, and parameters.
- (2) *Input Line*: The Input Line [B] shows program prompt messages and allows to type in commands directly.
- (3) *Toolbar*: The Toolbar [C] contains push buttons that execute commonly used ANSYS commands. More push buttons can be made available if desired.
- (4) *Main Menu*: The Main Menu [D] contains the primary ANSYS functions, organized by preprocessor, solution, general postprocessor, and design optimizer. It is from this menu that the vast majority of modeling commands are issued.
- (5) *Graphics Windows*: The Graphics Window [E] is where graphics are shown and graphical picking can be made. It is here where the model in its various stages of construction and the ensuing results from the analysis can be viewed.

The Output Window, shown in Figure 2.2, displays text output from the program, such as listing of data, etc. It is usually positioned behind the Graphics Window and can be put to the front if necessary.



```

c:\ ANSYS 9.0 Output Window

ANSYS University Advanced

***** ANSYS COMMAND LINE ARGUMENTS *****
INITIAL JOBNAME          = file

START-UP FILE MODE       = READ
STOP FILE MODE           = READ
GRAPHICS DEVICE REQUESTED = win32
GRAPHICAL ENTRY          = YES
LANGUAGE                 = en-us
INITIAL DIRECTORY        = F:\ansyswork

00217716      VERSION=INTEL NT      RELEASE= 9.0      UP20041
CURRENT JOBNAME=file 08:48:49 OCT 10, 2005 CP=      0.453

/SHOW SET WITH DRIVER NAME= WIN32 , RASTER MODE, GRAPHIC PLANE
RUN SETUP PROCEDURE FROM FILE= C:\Program Files\Ansys Inc\90\ANSYS
.ans
/INPUT FILE= menust.tmp LINE=      0
/INPUT FILE= C:\Program Files\Ansys Inc\90\ANSYS\apdl\start90.ans
0
ACTIVATING THE GRAPHICAL USER INTERFACE <GUI>. PLEASE WAIT...
Creating the GUI panes
Creating the GUI menubar
Creating the GUI status
Creating the GUI Graphics Context
Creating the Application Context
Creating the GUI toolbar
Creating the GUI toolbar - "ANSYS Standard Toolbar"
Creating the GUI toolbar - "ANSYS Main Menu"
Creating the GUI toolbar - "ANSYS Toolbar"
Creating the GUI toolbar - "ANSYS Graphical View"

CUTTING PLANE SET TO THE WORKING PLANE
Loading ANSYS Help System
Welcome to ANSYS

PRODUCE NODAL PLOT IN DSYS= 0
TURN OFF WORKING PLANE DISPLAY
  
```

Figure 2.2 Output window of ANSYS.

2.2.2 SAVING AND RESTORING JOBS

It is a good practice to save the model at various stages during its creation. Very often the stage in the modeling is reached where things have gone well and the model ought to be saved at this point. In that way, if mistakes are made later on, it will be possible to come back to this point.

To save the model, from **ANSYS Utility Menu** select, **File → Save as Jobname.db**.

The model will be saved in a file called Jobname.db, where Jobname is the name that was specified in the Launcher when ANSYS was first started. It is a good idea to save the job at different times throughout the building and analysis of the model to backup the work in case of a system crash or other unforeseen problems.

Alternatively, select **File → Save as**. In response to the second option frame, shown in Figure 2.3, is produced.

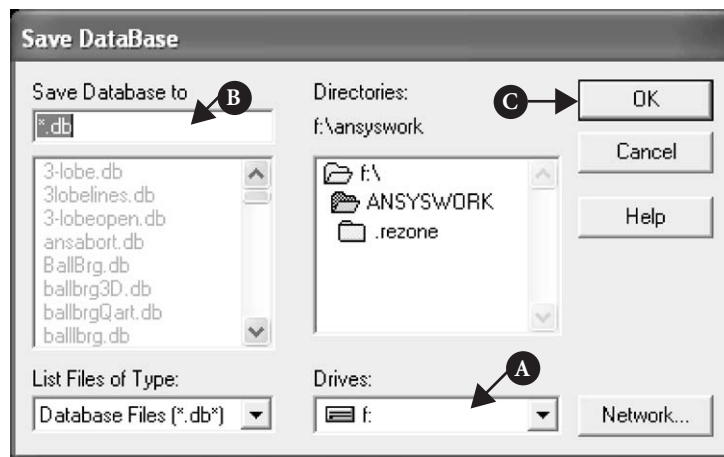


Figure 2.3 Save database of the problem.

Select appropriate drive [A] and give the file a name [B]. Clicking [C] **OK** button saves the model as a database with the name given.

Frequently there is a need to start up ANSYS and recall and continue a previous job. There are two methods to do this:

- (1) Using the Launcher. (i) In the ANSYS Launcher, select Interactive and specify the previously defined jobname. (ii) When ANSYS is running, select Utility Menu: File: Resume Jobname.db. This will restore as much of the database (geometry, loads, solution, etc.) as was previously saved.
- (2) Start ANSYS and select **Utility Menu: File → Resume from** and click on the job from the list that appears. Figure 2.4 shows the resulting frame.

Select appropriate file from the list [A] and click [B] **OK** button to resume the analysis.

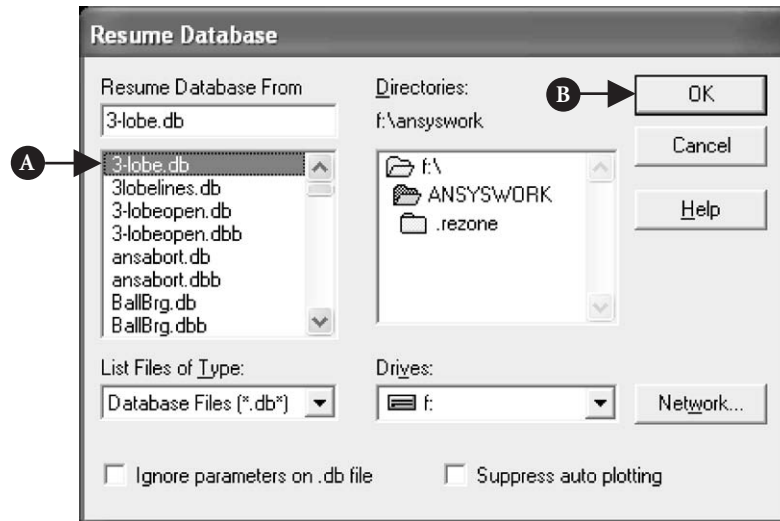


Figure 2.4 Resume problem from saved database.

2.2.3 ORGANIZATION OF FILES

A large number of files are created when ANSYS is run. If ANSYS is started without specifying a jobname, the name of all files created will be File.*, where the * represents various extensions described below. If a jobname is specified, say Beam, then the created files will all have the file prefix, Beam again with various extensions:

beam.db – database file (binary). This file stores the geometry, boundary conditions, and any solutions.

beam.dbb – backup of the database file (binary).

beam.err – error file (text). Listing of all error and warning messages.

beam.out – output of all ANSYS operations (text). This is what normally scrolls in the output window during ANSYS session.

beam.log – log file or listing of ANSYS commands (text). Listing of all equivalent ANSYS command line commands used during the current session.

Depending on the operations carried out, other files may have been written. These files may contain, for example, results. It is important to know what to save when, for instance, there is a need to clean up a directory or to move things from the /scratch directory. If the GUI is always used, then only the .db file is required. This file stores the geometry, boundary conditions, and any solutions. Once the ANSYS program has started, and the job name has been specified, only the resume command has to be activated to proceed from where the model was last left off. If, however, ANSYS command files are planned to be used, then only command file and/or the log file have to be stored. The log file contains a complete list of the ANSYS commands used to get the model to its current stage. That file may be run as is, or edited and rerun as desired.

2.2.4 PRINTING AND PLOTTING

ANSYS produces lists and tables of many types of results that are normally displayed on the screen. However, it is often desired to save the results to a file to be later analyzed or included in a report.

In case of stresses, instead of using **Plot Results** to plot the stresses, choose **List Results**. In the same way as described above. When the list appears on the screen in its own window, select File: **Save As** and give a file name to store the results. Any other solution can be done in the same way. For example, select **Nodal Solution** from the **List Results** menu to get displacements. Preprocessing and Solution data can be listed and saved from the **List Menu** in the **ANSYS Utility Menu**. Save the resulting list in the same way described above.

When there is a need to quickly save an image of the entire screen or the current Graphics Window, from **ANSYS Utility Menu** select **PlotCtrls → Hard Copy**. There are two options, namely **To Printer** (see Figure 2.5) and **To File** (see Figure 2.6).

In the frame shown in Figure 2.5, select the printer [A] from the list of printers and press [B] **Print** button.

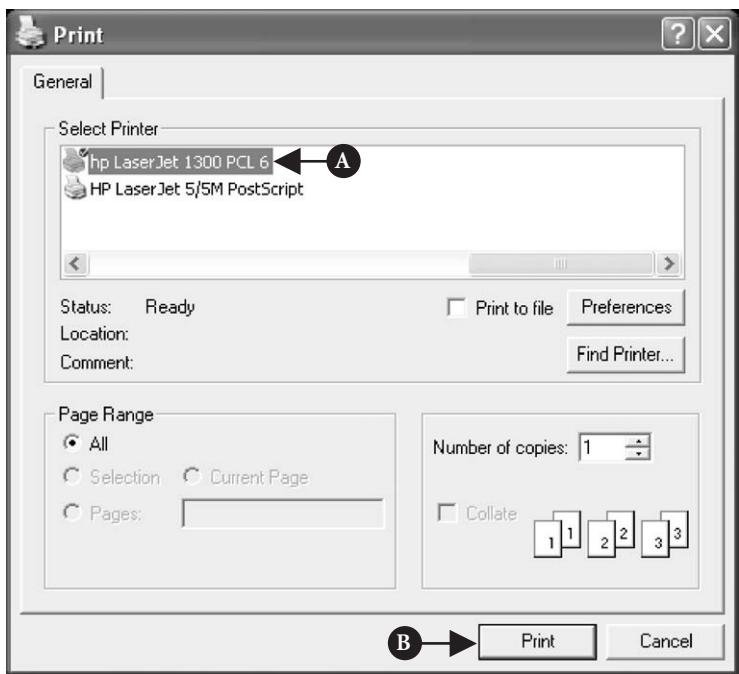


Figure 2.5 Print to printer.

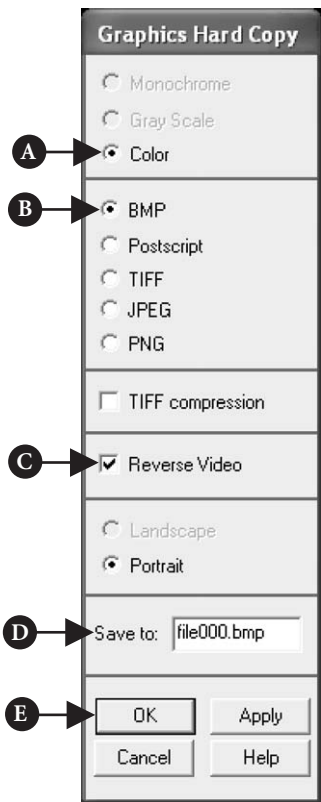


Figure 2.6 Print to file.

In the frame that is shown in Figure 2.6, an example selection could be [A] **Color**, [B] **BMP**, [C] **Reverse Video**, and finally [D] **Save to**. Then enter an appropriate file name and click [E] **OK** button. This image file may now be printed on a PostScript printer or included in a document.

2.2.5 EXITING THE PROGRAM

The ANSYS program can be exited in a number of different ways. When the current analysis is saved from **Utility Menu** select **File** → **Exit**. A frame shown in Figure 2.7 appears.

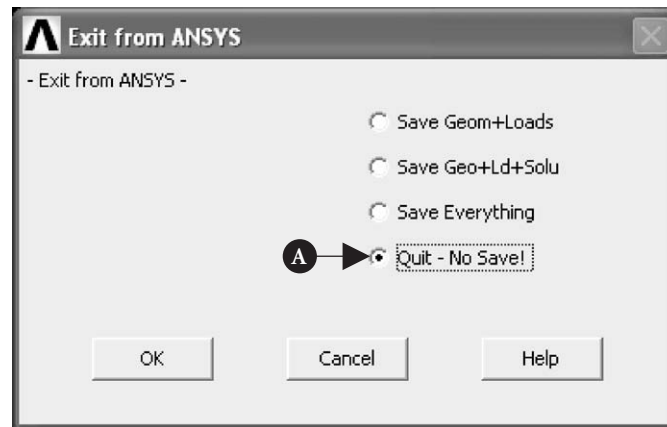


Figure 2.7 Exit from ANSYS.

There are four options, depending on what is important to be saved. If nothing is to be saved then select [A] **Quit – No Save** as indicated.

2.3 PREPROCESSING STAGE

2.3.1 BUILDING A MODEL

The ANSYS program has many finite-element analysis capabilities, ranging from a simple linear static analysis to a complex nonlinear transient dynamic analysis. Building a finite-element model requires more time than any other part of the analysis. First, a jobname and analysis title have to be specified. Next, the PREP7 preprocessor is used to define the element types, element real constants, material properties, and the model geometry. It is important to remember that ANSYS does not assume a system of units for intended analysis. Except in magnetic field analyses, any system of units can be used so long as it is ensured that units are consistent for all input data. Units

cannot be set directly from the GUI. In order to set units as the international system of units (SI) from **ANSYS Main Menu**, select **Preprocessor** → **Material Props** → **Material Library** → **Select Units**. Figure 2.8 shows the resulting frame.

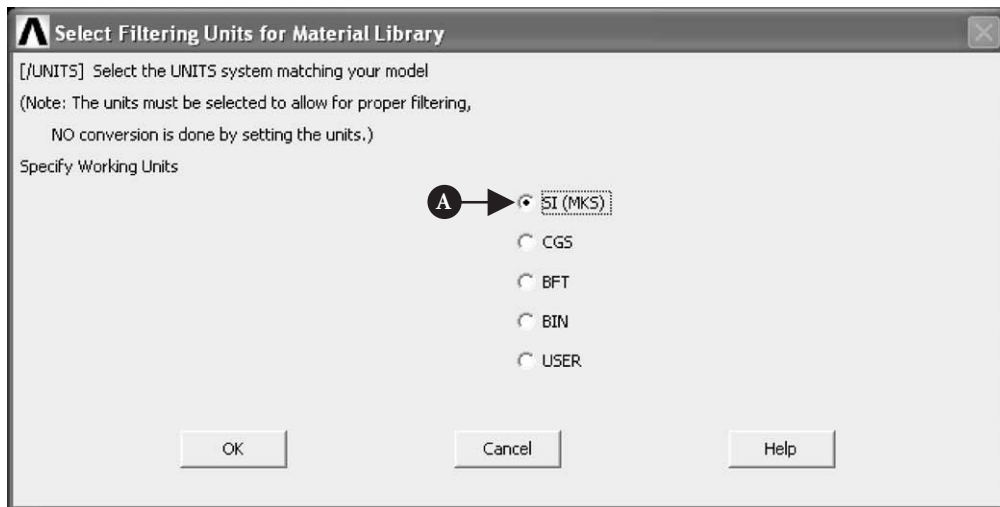


Figure 2.8 Selection of units for the problem.

Activate [A] **SI (MKS)** button to inform the ANSYS program that this system of units is proposed to be used in the analysis.

2.3.1.1 DEFINING ELEMENT TYPES AND REAL CONSTANTS

The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category. In order to define element types, one must be in PREP7. From **ANSYS Main Menu**, select **Preprocessor** or → **Element Type** → **Add/Edit/Delete**. In response, the frame shown in Figure 2.9 appears.

Click on [A] **Add** button and a new frame, shown in Figure 2.10, appears.

Select an appropriate element type for the analysis performed, e.g., [A] **Solid** and [B] **8node 183** as shown in Figure 2.10.

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. As with element types, each set of real constant has a reference number and the table of reference number versus real constant set is called the real constant table. Not all element types require real constant, and different elements of the same type may have different real constant values. **ANSYS Main Menu** command **Preprocessor** → **Modeling** → **Create** →

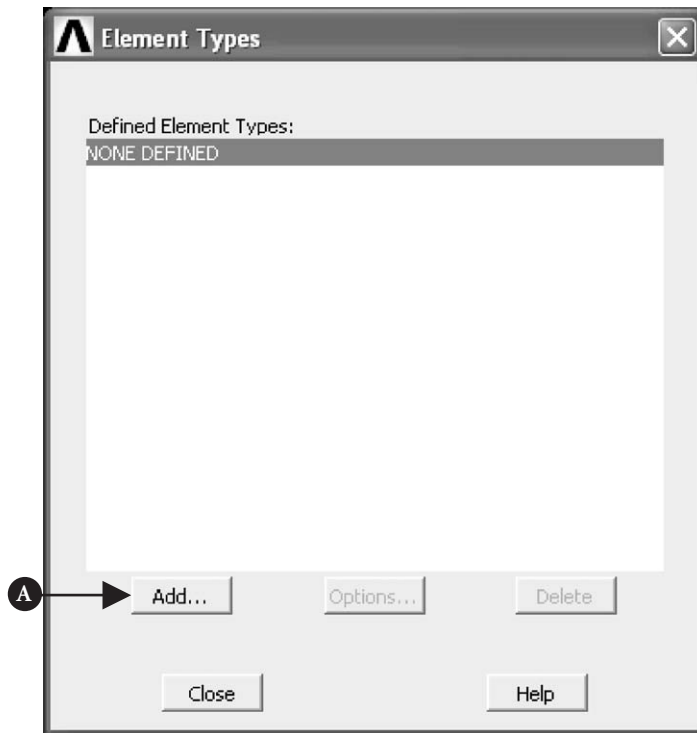


Figure 2.9 Definition of element types to be used.

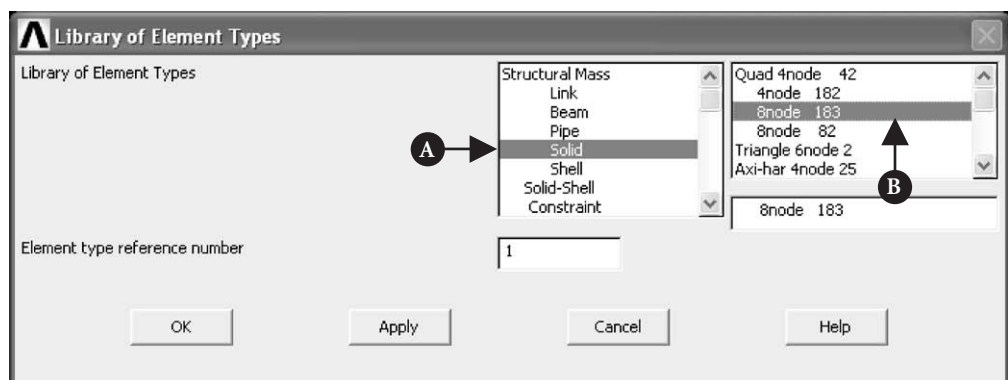


Figure 2.10 Selection of element types from the library.

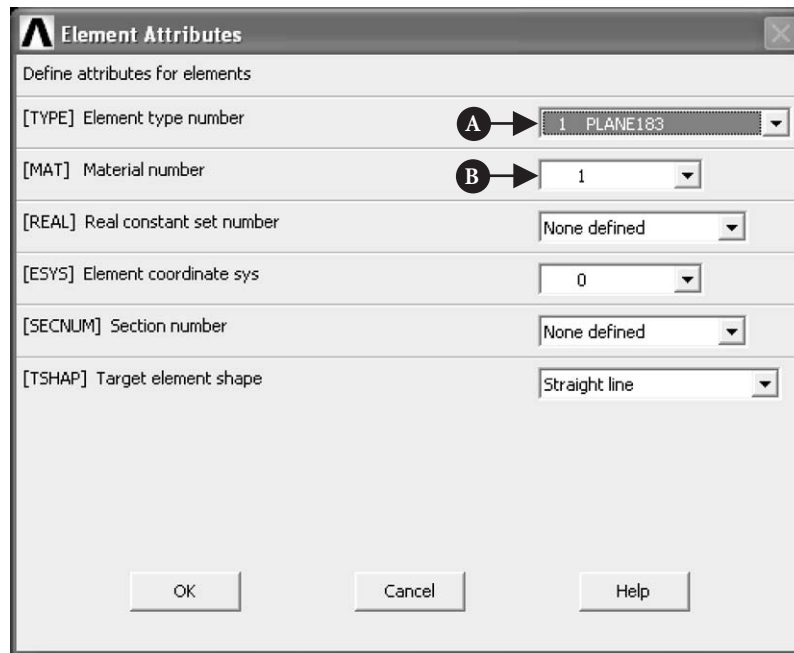


Figure 2.11 Element Attributes.

Elements → Element Attributes can be used to define element real constant. Figure 2.11 shows a frame in which one can select element type. According to Figure 2.11, an element type already selected is [A] **Plane183** for which real constant is being defined. A corresponding [B] **Material number**, allocated by ANSYS when material properties are defined (see Section 2.3.1.2) is also shown in the frame.

Other element attributes can be defined as required by the type of analysis performed. Chapter 7 contains sample problems where elements attributes are defined in accordance with the requirements of the problem.

2.3.1.2 DEFINING MATERIAL PROPERTIES

Material properties are required for most element types. Depending on the application, material properties may be linear or nonlinear, isotropic, orthotropic or anisotropic, constant temperature or temperature dependent. As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. In one analysis there may be multiple material property sets corresponding with multiple materials used in the model. Each set is identified with a unique reference number. Although material properties can be defined separately for each finite-element analysis, the ANSYS program enables storing a material property set in an archival material library file, then retrieving the set and reusing it in multiple

analyses. Each material property set has its own library file. The material library files also make it possible for several users to share commonly used material property data.

In order to create an archival material library file, the following steps should be followed:

- (i) Tell the ANSYS program what system of units is going to be used.
- (ii) Define properties of, for example, isotropic material. Use **ANSYS Main Menu** and select **Preprocessor** → **Material Props** → **Material Models**. A frame shown in Figure 2.12 appears.

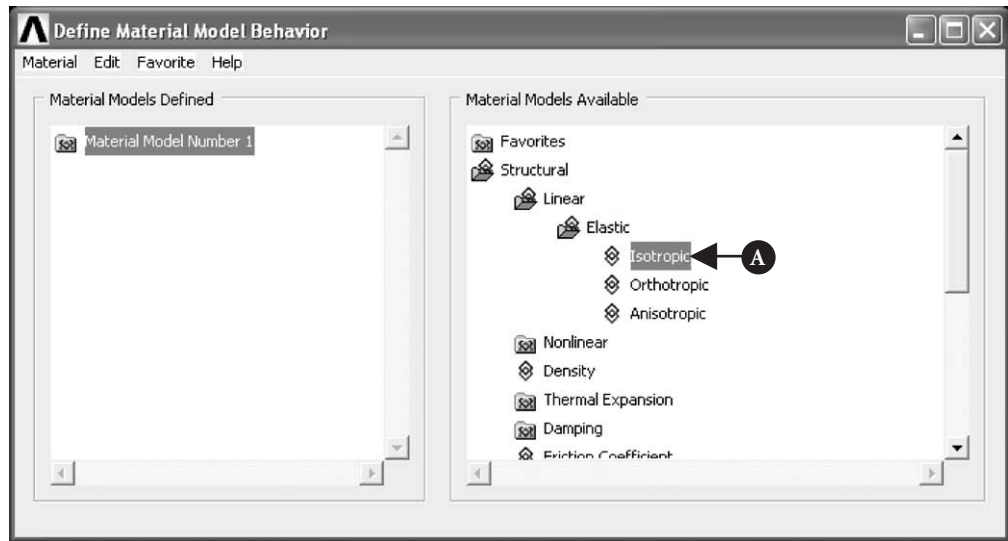


Figure 2.12 Define Material Model Behavior.

As shown in Figure 2.12, [A] **Isotropic** was chosen. Clicking twice on [A] **Isotropic** calls up another frame shown in Figure 2.13.

Enter data characterizing the material to be used in the analysis into appropriate field. For example, [A] **EX** = 2.1E+009 and [B] **PRXY** = 0.33 as shown in Figure 2.13. If the problem requires a number of different materials to be used, then the above procedure should be repeated and another material model created with appropriate material number allocated by the program.

2.3.2 CONSTRUCTION OF THE MODEL

2.3.2.1 CREATING THE MODEL GEOMETRY

Once material properties are defined, the next step in an analysis is generating a finite-element model – nodes and element adequately describing the model geometry. There

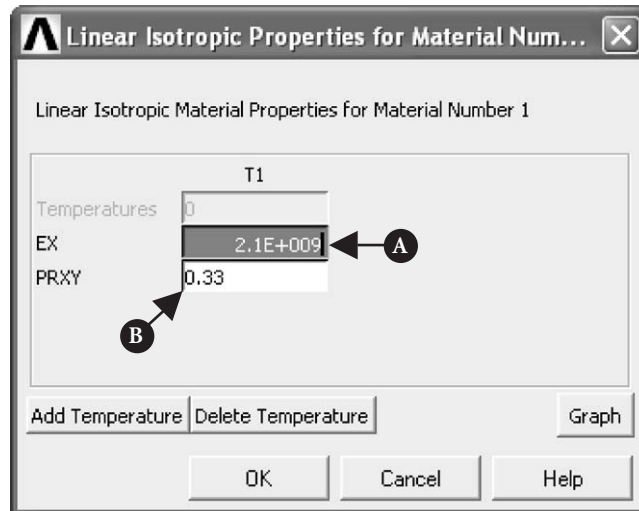


Figure 2.13 Linear isotropic material properties.

are two methods to create the finite-element model: solid modeling and direct generation. With solid modeling, the geometry of shape of the model is described, and then the ANSYS program automatically meshes the geometry with nodes and elements. The size and shape of the elements that the program creates can be controlled. With direct generation, the location of each node and the connectivity of each element is manually defined. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc., are available.

Solved example problems in this book amply illustrate, in a step-by-step manner, how to create the model geometry.

2.3.2.2 APPLYING LOADS

Loads can be applied using either **PREP7** preprocessor or the **SOLUTION** processor. Regardless of the chosen strategy, it is necessary to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite-element solution.

The analysis type to be used is based on the loading conditions and the response which is wished to calculate. For example, if natural frequencies and mode shapes are to be calculated, then a modal analysis ought to be chosen. The ANSYS program offers the following analysis types: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and substructuring. Not all analysis types are valid for all disciplines. Modal analysis, for instance, is not valid for thermal models. Analysis options allow for customization of analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton–Raphson options. In order to define the analysis type and analysis options, use **ANSYS Main Menu** and select **Main Menu: Preprocessor → Loads → Analysis Type → New Analysis**. In response to the selection, the frame shown in Figure 2.14 appears.

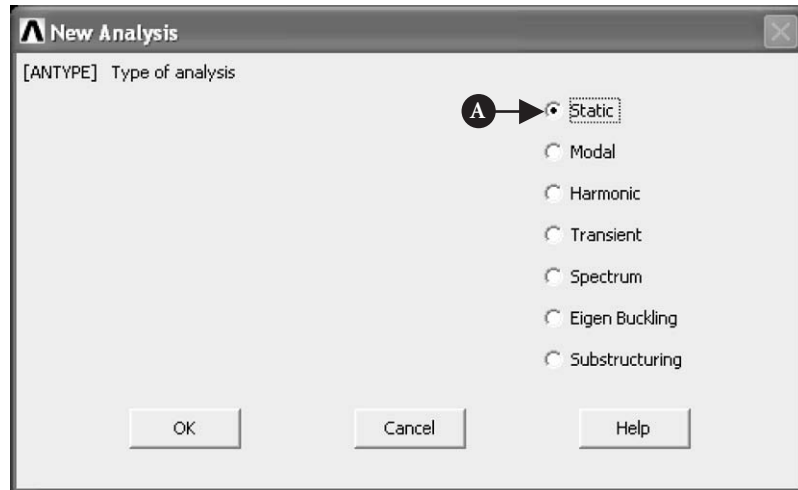


Figure 2.14 Type of analysis definition.

Select the type of analysis that is appropriate for the problem at hand by activating [A] **Static** button for example.

The word loads used here includes boundary conditions, i.e., constraints, supports, or boundary field specifications. It also includes other externally and internally applied loads. Loads in the ANSYS program are divided into six categories: DOF constraints, forces, surface loads, body loads, inertia loads, and coupled field loads. Most of these loads can be applied either on the solid model (keypoints, lines, and areas) or the finite-element model (nodes and elements).

There are two important load-related terms. A load step is simply a configuration of loads for which the solution is obtained. In a structural analysis, for instance, wind loads may be applied in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

Substeps are incremental steps taken within a load step. They are mainly used for accuracy and convergence purposes in transient and nonlinear analyses. Substeps are also known as time steps which are taken over a period of time.

Load step options are alternatives that can be changed from load step to load step, such as number of substeps, time at the end of a load step, and output controls. Depending on the type of analysis performed, load step options may or may not be required. Sample problems solved here provide practical guide to appropriate load step options as necessary.

2.4 SOLUTION STAGE

To initiate solution calculations, use **ANSYS Main Menu** selecting **Solution → Solve → Current LS**. Figure 2.15 shows resulting frame.



Figure 2.15 Start solution of current problem.

After reviewing the summary information about the model, click [A] **OK** button to start the solution. When this command is issued, the ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file and also to the database. The only difference is that only one set of results can reside in the database at one time, while a number of result sets can be written to the results file.

Once the solution has been calculated, the ANSYS postprocessors can be used to review the results.

2.5 POSTPROCESSING STAGE

Two postprocessors are available:

- (1) **POST1**: The general postprocessor is used to review results at one substep (time step) over the entire model or selected portion of the model. The command to enter POST1 requires selection from **ANSYS Main Menu General Postprocessor**. Using this postprocessor contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis can be obtained. POST1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and path operations.
- (2) **POST26**: The time history postprocessor is used to review results at specific points in the model over all time steps. The command to enter POST26 is as follows: from **ANSYS Main Menu** select **TimeHist Postprocessor**. Graph plots of results data versus time (or frequency) and tabular listings can be obtained. Other POST26 capabilities include arithmetic calculations and complex algebra.