2

CHAPTER

ANSYS OPERATIONS

2.1 GETTING STARTED

There are three basic choices (Fig. 2.1) to start ANSYS from the Windows Start Menu.

ANSYS Workbench: This is a new GUI with an emphasis on CAD connectivity, ease of use, and easy management of assembly contact.
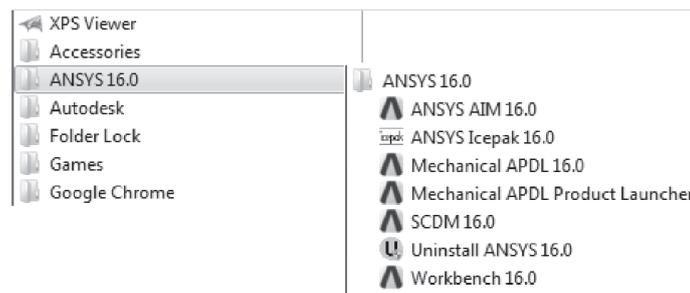

Fig. 2.1 Three basic choices to start ANSYS.

ANSYS: This starts ANSYS in the traditional GUI. The program starts immediately using the settings last changed under the next item, 'Configure ANSYS Products'.

Configure ANSYS Products: This choice brings up the ANSYS Launcher.

ANSYS has many finite-element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The process for a typical analysis involves three general tasks:

- Building the model
- Applying loads and obtaining the solution
- Reviewing the results.

2.1.1 Building the Model

Building a finite element model requires more time than any other part of the analysis. First, specify a jobname and analysis title. Then, use the preprocessor to define the element type(s), element real constant(s), material properties and the model geometry.
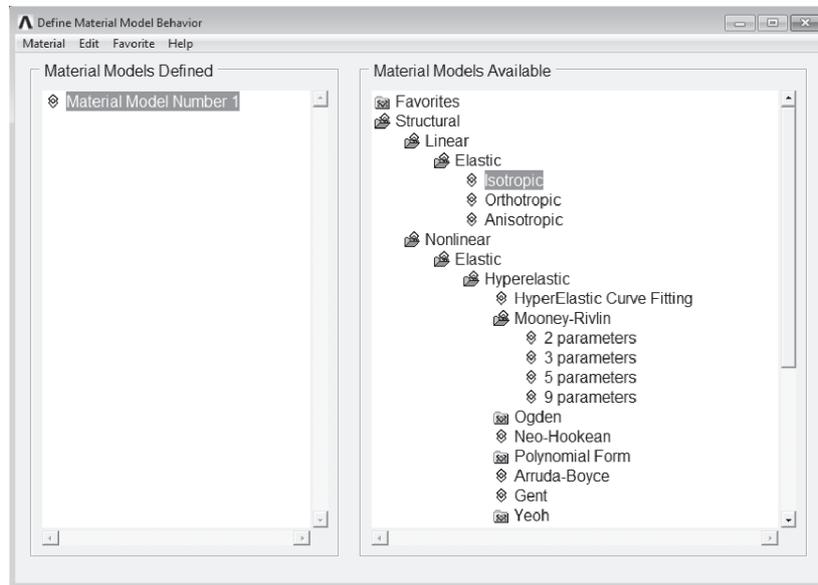
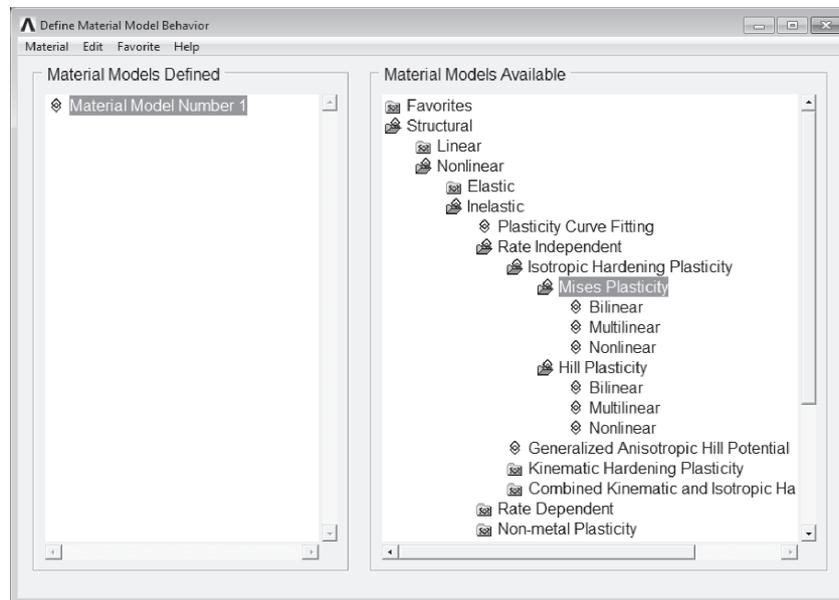**Fig. 2.2**   Material Model Interface Initial Screen.

Fig. 2.3   Material Model Interface Tree Structure.

For example, under Structural are categories Linear, Nonlinear, and others. The models are further categorized so that the analyst will eventually see a vertical list of material property sets or material models that are included under that category. Once it is decided which material property set or model will be used, then choose it by double-clicking on the item. A dialog box appears that prompts for the required input data for that particular model or property set.

Entering Material Data

Included in a data input dialog box is a table whose rows and columns can be altered depending on the requirements of the specific material property or model chosen. A typical data input dialog box is shown in Fig. 2.4.
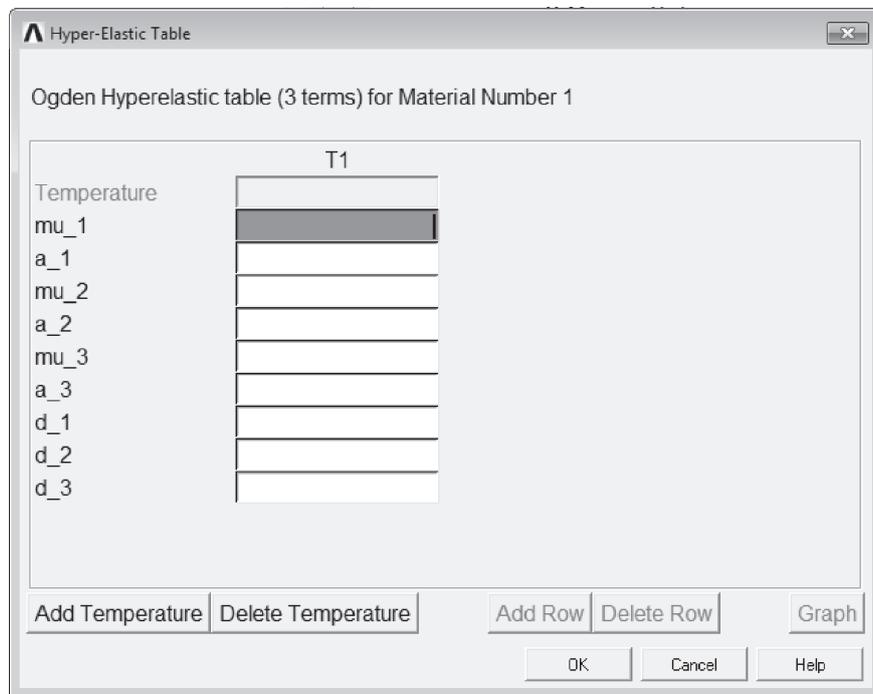

Fig. 2.4　A Data Input Dialog Box.

There are two interaction areas within a material data input dialog box: the data input table, and a series of action buttons that appear at the bottom. Depending on the material item defined, the labels in the table vary, as do the number of rows and columns that appear initially. The material item also dictates the number of rows and columns that are allowed to be added or deleted. In most cases, the columns represent temperatures, and the rows represent data values.

Temperature Dependent Data

Initially, the table is set up for temperature independent data so the temperature field is grayed out. At this point, should the analyst decide to enter data for various temperatures, s/he can quickly add columns of text fields for the data representing each temperature. The temperature dependent data can be added or deleted at any time. There is no necessity to predetermine if the data should be temperature dependent.

Adding and Deleting Columns

In order to add a column, position the text cursor in any field in the existing column, then click on the **Add Temperature** button. A new column appears to the right of the existing column, and both temperature fields become active, as shown in Fig. 2.5.
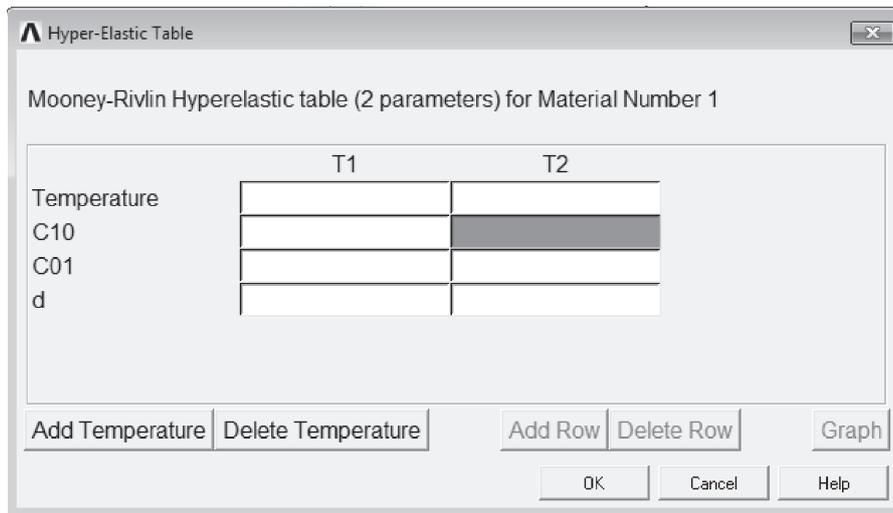

Fig. 2.5   Data Input Dialog Box - Added Column.

Then enter the two temperatures and the associated data in the rows. More temperature columns can be added, as needed, by following the same procedure. Insert columns between existing columns by clicking the text cursor in a field within a column that is to the left of where it is nesessary to insert the new column, then clicking on the **Add Temperature** button. A scroll bar appears across the bottom of the table when the number of columns exceeds the width of the dialog box.

Adding and Deleting Rows

It may be necessary to add another row of constants or other data for a specific temperature. Then add or delete rows in a similar way as is described above for adding or deleting columns. In order to add a row, click the text cursor in any field in an existing row, then click on the **Add Row** (or **Add Point**) button. A new row appears beneath the existing row, as shown in Fig. 2.6.
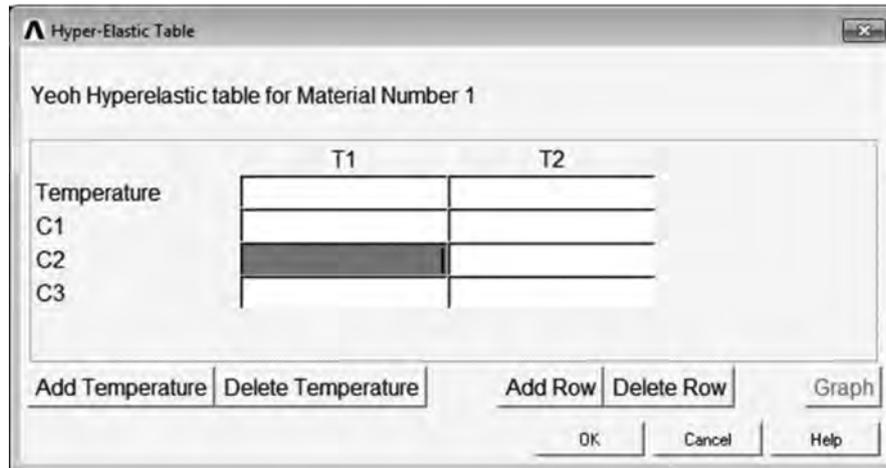
Fig. 2.6   Data Input Dialog Box – Added Row.

More rows can be added, as needed, by following the same procedure. Insert rows between existing rows by positioning the text cursor in a field in the top row, then clicking on the **Add Row** (or **Add Point**) button. A vertical scroll bar appears in the table when the number of rows exceeds the height of the dialog box.

Entering/Editing Data in Text Fields

When a data dialog box first appears, one of the text fields is selected (black highlight), meaning that the field is ready to accept and display data as the analyst types. The arrow keys can be used to move the selection status to other text fields. Also, pressing the Tab key allows to move the selection status to the text field positioned to the right of the field that is currently selected.

When the analyst starts typing within a text field, the highlight is replaced by the characters that are typed. The left and right arrow keys can be used to position the text cursor anywhere within the field should it be necessary to replace or delete characters in that field.

In order to edit data, first select the text field either by clicking on the field, or using the arrow keys to move the selection status to the particular field.

Logging/Editing Material Data

The **Material Models Defined** window displays a log of each material model specified. After OK is chosen in the data input dialog box, this window displays a folder icon, and **Material Model Number** #, followed by the properties defined for this model. Additional models can be defined with unique numbers by choosing **Material > New Model**, then typing a new number in the **Define Material ID** dialog box. On double-clicking on any material model or property, the associated data input dialog box appears where the data can be edited, if choosen.

The following analysis types can be performed in the program: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and substructuring.

Analysis options allows the analyst to customize the analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton – Raphson options.

Use **Main Menu > Preprocessor > Loads > Analysis Type > New Analysis or Main Menu > Preprocessor > Loads > Analysis Type > Restart** to define the analysis type and analysis options.

The analyst can specify either a new analysis or a restart, but a new analysis is the norm in most cases. A multi frame restart that allows to restart an analysis at any point is available for static and transient analyses. The analysis type and analysis options cannot be changed after the first solution.

After the analysis type and analysis options are defined, the next step is to apply loads. Some structural analysis types require other items to be defined first, such as master degrees of freedom and gap conditions.

2.1.2.2 Applying Loads

The word loads as used in ANSYS includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally (Fig. 2.8) and internally applied loads. Loads are divided into the following categories:

- DOF Constraints
- Forces
- Surface Loads
- Body Loads
- Inertia Loads
- Coupled -field Loads.

Fig. 2.8 External 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). Two important load-related terms need to be known are load step and substep. A *load step* is simply a configuration of loads for which a solution is to be obtained. In a structural analysis, for example, the analyst may apply wind loads 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* – steps taken over a period of time.

The program uses the concept of time in transient analyses as well as static (or steady–state) analyses. In a transient analysis, time represents actual time, in seconds, minutes, or hours. In a static or steady-state analysis, time simply acts as a counter to identify load steps and substeps.

Loads and boundary conditions can be applied in both the Preprocessor (**Main Menu** > **Preprocessor** > **Loads** > **Define Loads** > **Apply**), and the Solution processor (**Main Menu** > **Solution** > **Define Loads** > **Apply**).

1. Select the kind of constraint to be applied.
2. Select the geometric entity on which it is to be applied.
3. Enter the value and direction for it.

There is no modify command for loads and BCs. If a mistake is made simply apply it again with a new value (the old one will be replaced if it's on the same entity), or delete it and reapply it.

Although loads and boundary conditions can be applied to nodes or elements, it's generally better to apply all BCs to the geometry. When the solve command is issued, they will be automatically transferred to the underlying nodes and elements. If BCs are put on the geometry, that geometry can be remeshed without having to reapply them.

2.1.2.3 Specifying Load Step Options

Load step options are options that the analyst can change 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 s/he is doing, load step options may or may not be required. The analysis procedures in the analysis guide manuals describe the appropriate load step options as necessary.

2.1.2.4 Initiating the Solution

Use **Main Menu** > **Solution** > **Solve** > **Current LS** to initiate solution calculations. When this command is issued, the program takes model and loading information from the database and calculates the results. Results are written to the results file (`Jobname.RST`, `Jobname.RTH`, or `Jobname.RMG`) and also to the database. The only difference is that only one set of results can reside in the database at one time, while ANSYS can write all sets of results (for all substeps) to the results file.

Multiple load steps can be solved in a convenient manner using: **Main Menu** > **Solution** > **Solve** > **From LS Files.**

2.1.3 Reviewing the Results

After the solution has been calculated, use the postprocessors to review the results. Two postprocessors are available: POST1 and POST26.

Use POST1, the general postprocessor, to review results at one substep (time step) over the entire model or selected portion of the model. The GUI path for entering POST1 is **Main Menu** > **General Postproc**, valid only at the Begin level. The analyst can obtain contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis. POST1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and path operations.

- A *body load* is a volumetric or field load. Examples are temperatures and fluences in a structural analysis, heat generation rates in a thermal analysis, and current densities in a magnetic field analysis.
- *Inertia loads* are those attributable to the inertia (mass matrix) of a body, such as gravitational acceleration, angular velocity, and angular acceleration. They are mainly used in a structural analysis.
- *Coupled-field loads* are simply a special case of one of the above loads, where results from one analysis are used as loads in another analysis. For example, magnetic forces calculated in a magnetic field analysis can be applied as force loads in a structural analysis.

A *load step* is simply a configuration of loads for which a solution is obtained. In a linear static or steady-state analysis, the analyst can use different load steps to apply different sets of loads – wind load in the first load step, gravity load in the second load step, both loads and a different support condition in the third load step, and so on. In a transient analysis, multiple load steps apply different segments of the load history curve.

The program uses the set of elements which the analyst selects for the first load step. It doesn't matter which element sets are specified for the later steps for all subsequent load steps. Use **Utility Menu > Select > Entities** to select an element set.

Figure 2.10 shows a load history curve that requires three load steps : the first load step for the ramped load, the second load step for the constant portion of the load, and the third load step for load removal.
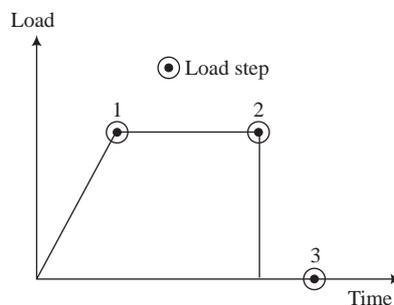

Fig. 2.10 Transient Load History Curve.

2.2.2   The Role of Time in Tracking

The program uses time as a tracking parameter in all static and transient analyses, whether they are or are not truly time-dependent. The advantage of this is that the analyst can use one consistent 'counter' or 'tracker' in all cases, eliminating the need for analysis-dependent terminology. Moreover, time always increases monotonically, and most things in nature happen over a period of time, however brief the period may be.

Obviously, in a transient analysis or in a rate-dependent static analysis (creep or viscoplasticity), time represents actual, chronological time in seconds, minutes, or hours. Assign the time at the end of each load step while specifying the load history curve. Use one of the following to assign time:

- **Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Time and Substps**
- **Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Time - Time Step**
- **Main Menu > Solution > Analysis Type > Sol'n Control**
- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Time and Substps**
- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Time - Time Step**
- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Time and Substps**
- **Main Menu > Solution > Load Step Opts > Time /Frequenc > Time - Time Step**

In a rate-independent analysis, however, time simply becomes a counter that identifies load steps and substeps. By default, the program automatically assigns time = 1.0 at the end of load step 1, time = 2.0 at the end of load step 2, and so on. Any substeps within a load step are assigned the appropriate, linearly interpolated time value.

2.2.3   Stepped Versus Ramped Loads

When more than one substep is specified in a load step, the question of whether the loads should be *stepped* or *ramped* arises (Fig. 2.11).

- If a load is *stepped*, then its full value is applied at the first substep and stays constant for the rest of the load step.
- If a load is *ramped*, then its value increases gradually at each substep, with the full value occurring at the end of the load step.
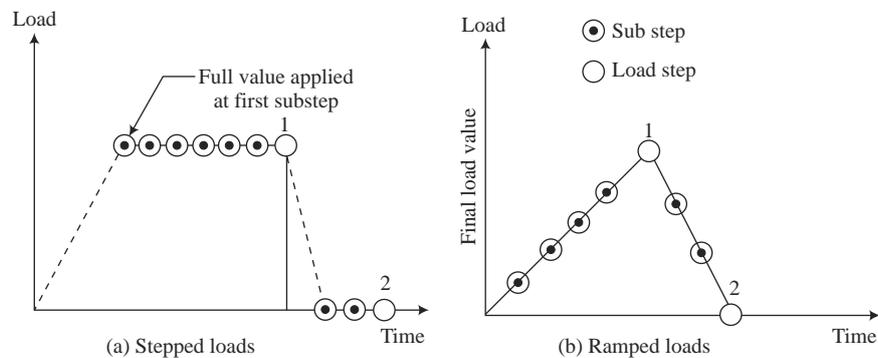

Fig. 2.11   Stepped versus ramped loads.

Use either of the following to indicate whether loads are ramped or stepped:

- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Freq & Substeps: Transient Tab**
- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Time and Substps**
- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Time & Time Step**
- **Main Menu > Solution > Load Step Opts > Time/Frequenc > Freq & Substeps**

- **Main Menu > Preprocessor > Loads > Define Loads > Apply >** *load type* **> On Nodes**
- **Utility Menu > List > Loads > DOF Constraints > On All Keypoints (or On Picked KPs)**
- **Main Menu > Solution > Define Loads > Apply >** *load type* **> On Lines**

2.2.4.4   Transferring Constraints

Use **Main Menu > Preprocessor > Loads > Define Loads > Operate > Transfer to FE > Constraints** or **Main Menu > Solution > Define Loads > Operate > Transfer to FE > Constraints** to transfer constraints that have been applied to the solid model to the corresponding finite element model.

Use **Main Menu > Preprocessor > Loads > Define Loads > Operate > Transfer to FE > All Solid Lds** or **Main Menu > Solution > Define Loads > Operate > Transfer to FE > All Solid Loads** to transfer all solid model boundary conditions.

2.2.4.5   Forces (Concentrated Loads)

Following are the examples of some of the GUI paths used for applying force loads:
- **Main Menu > Preprocessor > Loads > Define Loads > Apply >** *load type* **> On Nodes**
- **Utility Menu > List > Loads > Forces > On All Keypoints (or On Picked KPs)**
- **Main Menu > Solution > Define Loads > Apply >** *load type* **> On Line**

2.2.4.6   Surface Loads

Following are examples of some of the GUI paths used for applying surface loads:
- **Main Menu > Preprocessor > Loads > Define Loads > Apply >** *load type* **> On Nodes**
- **Utility Menu > List > Loads > Surface > On All Elements (or On Picked Elements)**
- **Main Menu > Solution > Define Loads > Apply >** *load type* **> On Lines**

The program stores surface loads specified on nodes internally in terms of elements and element faces. Therefore, if both nodal and element surface load commands are used for the same surface, only the last specification is used.

The program applies pressures on axisymmetric shell elements or beam elements on their inner or outer surfaces, as appropriate. In-plane pressure load vectors for layered shells (such as SHELL281) are applied on the nodal plane. KEYOPT(11) determines the location of the nodal plane within the shell. When using flat elements to represent doubly curved surfaces, values which should be a function of the active radius of the meridian be inaccurate.

Applying Pressure Loads on Beams

Use **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Beams** or **Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Beams** to apply pressure loads on the lateral faces and the two ends of beam elements.

Lateral pressures, which have units of force per unit length, can be applied both in the normal and tangential directions. The pressures may vary linearly along the element length, and can be specified on a portion of the element, as shown in Fig. 2.12. The analyst can also reduce the pressure down to a force at any location on a beam element by setting the JOFFST field to -1. End pressures have units of force.
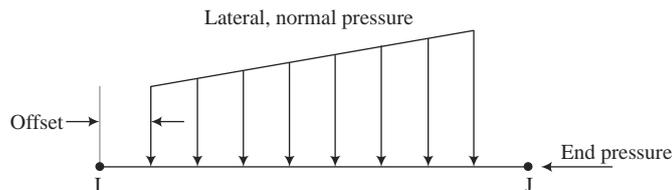

Fig. 2.12   Example of Beam Surface Loads

2.2.4.7   *Applying Body Loads*

Following are examples of some of the GUI paths used for applying body loads:

 • **Main Menu > Preprocessor > Loads > Define Loads > Apply >** *load type* **> On Nodes**
 • **Utility Menu > List > Loads > Body > On Picked Elems**
 • **Main Menu > Solution > Define Loads > Apply >** *load type* **> On Keypoints**
 • **Utility Menu > List > Loads > Body > On Picked Lines**
 • **Main Menu > Solution > Define Loads > Apply >** *load type* **> On Volumes.**

2.2.4.8   *Applying Coupled-Field Loads*

A coupled-field analysis usually involves applying results data from one analysis as loads in a second analysis. For example, the nodal temperatures calculated in a thermal analysis can be applied as body loads in a structural analysis. Similarly, magnetic forces calculated in a magnetic field analysis can be applied as nodal forces in a structural analysis. Use **Main Menu > Preprocessor > Loads > Define Loads > Apply >** *load type* **> From source** or **Main Menu > Solution > Define Loads > Apply >** *load type* **> From source** to apply such coupled-field loads.

2.2.4.9   *Axisymmetric Loads and Reactions*

For constraints, surface loads, body loads, and Y-direction accelerations, loads are defined exactly as they would be for any nonaxisymmetric model. However, for concentrated forces the procedure is a little different. For these quantities, input load values of force, moment, etc., are on a '(360°) basis'. That is, the load value is entered in terms of *total load around the circumference.* For example, if an axisymmetric axial load of 1500 pounds per inch of circumference were applied to a 10′ diameter pipe (Fig. 2.13), the total load of 47,124 lb. (1500 * 2 *n* * 5 = 47,124) would be applied to node *N*.

Axisymmetric results are interpreted in the same fashion as their corresponding input loads. That is, reaction forces, moments, etc., are reported on a total load (360°) basis.

If the Solution Controls dialog box is used to set the analysis and load step options, define each load step using the **Basic** tab. After all load step files are written, use one action command to read in the files sequentially and obtain the solution for each load step.

2.3   SOLUTION

In the solution phase of an analysis, the computer takes over and solves the simultaneous set of equations that the finite element method generates. The results of the solution are:
- Nodal degree of freedom values, which form the primary solution
- Derived values, which form the element solution.

The element solution is usually calculated at the elements' integration points. The program writes the results to the database as well as to the results file (.RST, .RTH, or .RMG files).

2.3.1   Selecting a Solver

Several methods for solving the system of simultaneous equations are available in the program: sparse direct solution, Preconditioned Conjugate Gradient (PCG) solution, Jacobi Conjugate Gradient (JCG) solution, Incomplete Cholesky Conjugate Gradient (ICCG) solution, and Quasi-Minimal Residual (QMR) solution. In addition, distributed versions of the sparse, PCG, and JCG solvers are available for use in Distributed ANSYS.

Select a solver using: **Main Menu > Preprocessor > Loads > Analysis Type > Analysis Options** or **Main Menu > Solution > Load Step Options > Sol'n Control (: Sol'n Options Tab)** or **Main Menu > Solution > Analysis Options** or **Main Menu > Solution > Unabridged Menu > Analysis Options**

2.3.2   Obtaining the Solution

Use **Main Menu > Solution > Current** LS to initiate the solution. Because the solution phase generally requires more computer resources that the other phases of an analysis, it is better suited to batch mode than interactive mode.

The solver writes output to the output file (Jobname.OUT) and the results file. If the solution is run interactively, the output 'file' is actually the screen. By using one of the following before issuing **SOLVE**, the output can be diverted to a file instead of the screen: **Utility Menu > File > Switch Output to > File** or **Output Window**

Data written to the output file consist of the following:
- Load summary information
- Mass and moments of inertia of the model
- Solution summary information
- A final closing banner that gives total CPU time and elapsed time
- Data requested by the **OUTPR** output control command or its GUI counterpart

In interactive mode, much of the output is suppressed. The results file (.RST, .RTH, or .RMG) contains all results data in binary form, which can be reviewed in the postprocessors.

2.4.1 Postprocessors Available

Two postprocessors are available for reviewing the results: POST1, the general postprocessor, and POST26, the time-history postprocessor. POST1 allows the analyst to review the results over the entire model at specific load steps and substeps. In a static structural analysis, for example, s/he can display the stress distribution for load step 3. Or, in a transient thermal analysis, s/he can display the temperature distribution at time = 100 seconds. Figure 2.14 is a typical example of a POST1 plot.
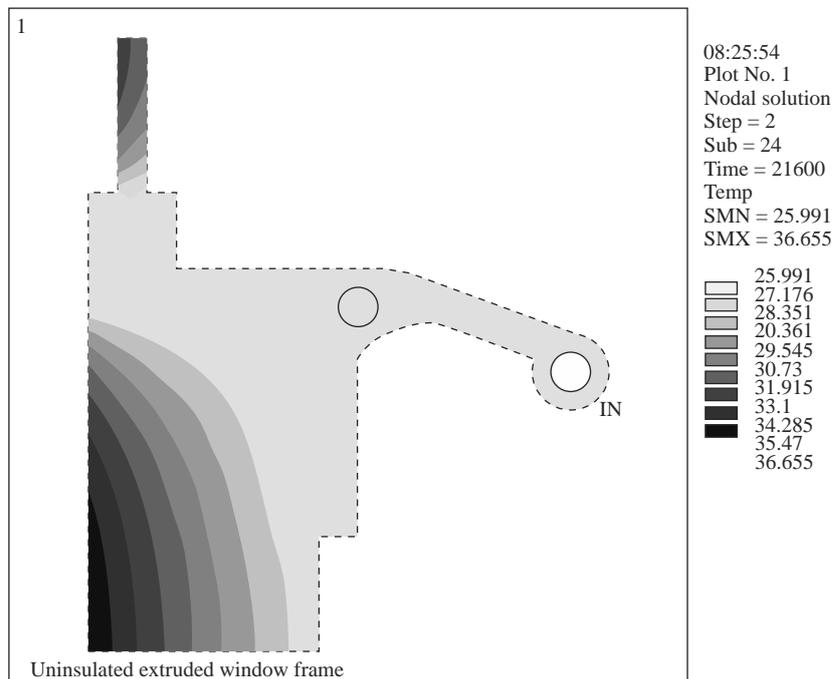

Fig. 2.14 A Typical POST1 Contour Display.

POST26 allows the analyst to review the variation of a particular result item at specific points in the model with respect to time, frequency, or some other result item. In a transient magnetic analysis, for instance, the analyst can graph the eddy current in a particular element versus time. Or, in a nonlinear structural analysis, s/he can graph the force at a particular node versus its deflection. A Typical POST26 Graph is shown in Fig. 2.15.

It is important to remember that the postprocessors in ANSYS are just tools for reviewing analysis results. The analyst still needs to use engineering judgment to interpret the results. For example, a contour display may show that the highest stress in the model is 37,800 psi. It is now up to her/him to determine whether this level of stress is acceptable for the design.
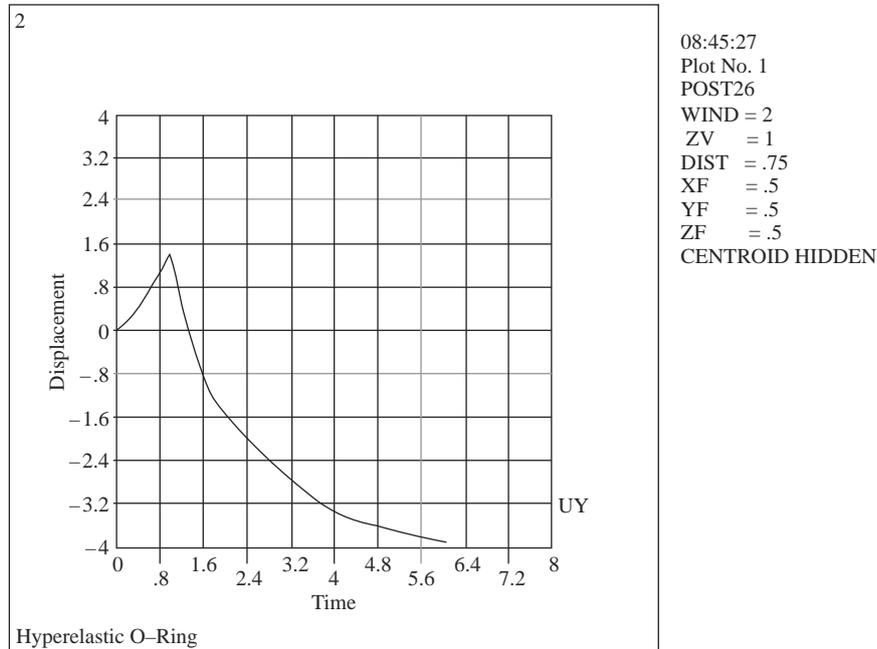
Fig. 2.15　A Typical POST26 Graph.

2.4.2　Results Files

The analyst can use **OUTRES** to direct the ANSYS solver to append selected results of an analysis to the results file at specified intervals during solution. The name of the results file depends on the analysis discipline:

- `Jobname.RST`　for a structural analysis and coupled-field analysis
- `Jobname.RTH`　for a thermal and diffusion analyses
- `Jobname.RMG`　for a magnetic field analysis.

For fluid analyses, the file extension is `.RST` or `.RTH`, depending on whether structural degrees of freedom are present.

2.4.3　Types of Data Available for Postprocessing

The solution phase calculates two types of results data – Primary data and Derived data. *Primary data* consist of the degree-of-freedom solution calculated at each node: displacements in a structural analysis, temperatures in a thermal analysis, magnetic potentials in a magnetic analysis, and so on. These are also known as nodal solution data.

Derived data are those results calculated from the primary data, such as stresses and strains in a structural analysis, thermal gradients and fluxes in a thermal analysis, magnetic fluxes in a magnetic

analysis, and the like. They are typically calculated for each element and may be reported at any of the following locations: at all nodes of each element, at all integration points of each element, or at the centroid of each element. Derived data are also known as element solution data, except when they are averaged at the nodes. In such cases, they become nodal solution data.

2.5 THE GENERAL POSTPROCESSOR (POST1)

Use POST1, the general postprocessor, to review analysis results over the entire model, or selected portions of the model, for a specifically defined combination of loads at a single time. POST1 has many capabilities, ranging from simple graphics displays and tabular listings to more complex data manipulations such as load case combinations. Issue **Main Menu > General Postproc** to enter the general postprocessor.

2.5.1 Reading Results Data into the Database

The first step in POST1 is to read data from the results file into the database. Model data must exist in the database to do so. If the database does not already contain model data, issue **Utility Menu > File > Resume Jobname.db** to read the database file, Jobname.DB. The database should contain the same model for which the solution was calculated, including the element types, nodes, elements, element real constants, material properties, and nodal coordinate systems.

2.5.2 Reviewing Results in POST1

Once the desired results data are stored in the database, the analyst can review them through graphics displays and tabular listings. In addition, s/he can map the results data onto a path.

2.5.2.1 Displaying Results Graphically

Graphics displays are perhaps the most effective way to review results. The following types of graphics can be displayed in POST1:

- Contour displays
- Vector displays
- Reaction force displays
- Deformed shape displays
- Path plots
- Particle flow traces.

2.5.2.2 Listing Results in Tabular Form

An effective way of documenting analysis results is to produce tabular listings in POST1. Listing options are available for nodal and element solution data, reaction data, element table data, and more.

Listing Nodal and Element Solution Data

Use **Main Menu > General Postproc > List Results > Nodal Solution** to list specified nodal solution data. Use **Main Menu > General Postproc > List Results > Element Solution** to list

time. In this mode, the analyst can process results data in many ways. S/he can construct graphics displays, chart representations or tabular listings, or perform math operations on the data sets. A typical time-history task would be to graph result items versus time in a transient analysis, or to graph force versus deflection in a nonlinear structural analysis.

Following is the general process for using the time-history postprocessor:

1. Start the time-history processor, either interactively or via the command line.
2. Define time-history variables. This involves not only identifying the variables, but also storing the variables.
3. Process the variables to develop calculated data or to extract or generate related variable sets.
4. Prepare output. This can be via graph plots, tabular listings or file output.

2.6.1 Entering the Time-History Postprocessor

The analyst enters the time history processor to process time or frequency related results data. Once an analysis is solved, ANSYS uses the results data to create a 'Results File.' The active results file (`*.RST, *.RTH, *.RMG,` etc.) is automatically loaded when the analyst enters postprocessing. If the current analysis contains no results file, the program queries for one. The file option can also be used to load any other results file for processing.

2.6.2 Importing Data

This feature allows the analyst to read in set(s) of data from a file into time history variable(s). This enables the user, for instance, to display and compare test results data against the corresponding ANSYS results data.

2.6.3 Exporting Data

This feature allows the analyst to write out selected time history variable(s) to an ASCII file or to APDL array/table parameter. This enables to perform several functions such as pass data on to another program for further processing or to archive data in an easily retrievable format.

2.6.4 Reviewing the Variables

Once the variables are defined, they can be reviewed via graph plots or tabular listings.

2.6.4.1 Plotting Result Graphs

The description for graph plotting, both with the variable viewer and from the command line follows:

Interactive

The 'Graph Data' button in the variable viewer allows to plot all the selected variables. A maximum of 10 variables can be plotted on a single graph. By default, the variable used for the X-axis of the graphs is TIME for static and transient analyses or FREQUENCY for harmonic analysis. The

- There is a structure with many fastener holes that analyst wants to constrain. Again, he could select them all one by one when applying the constraint, or select lines 'By Length/ Radius', type in the radius of the holes to select all of them in one shot, then 'Pick All' in the picking dialog when applying the constraint.
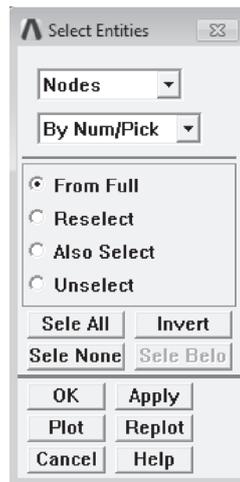

Fig. 2.17

After working with the selected set, use **Utility Menu > Select > Everything** to make the whole model active again. Select Entities Dialog Box Terminology (Fig. 2.18):

- *From Full*: Select from the entire set of entities in the model.
- *Reselect:* Select a subset from the currently selected entities.
- *Also Select:* Select in addition to (from the whole model) the set analyst has currently selected.
- *Unselect:* Remove items from the selection set.
- *Select All:* This is not the same as **Utility Menu > Select > Everything.** This selects all of whatever entity analyst has specified at the top of the dialog.
- *Invert:* Reverses the selected and unselected entities (just the entities specified at the top of the dialog).
- *OK:* This does the select operation (or brings up a picker dialog so that analyst can pick with the mouse) and then dismisses the dialog.
- *Apply:* This does the operation but keeps the dialog box. Typically use this so the dialog stays active.
- *Replot:* Replots whatever is active in the graphics window.
- *Plot:* Plots only the entity specified at the top of the dialog.
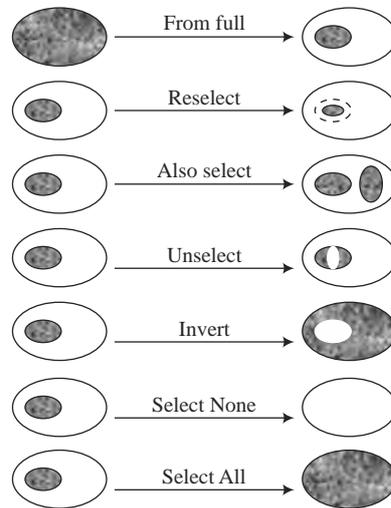
Fig. 2.18

2.7.2   Organizing the Model using Components

If a group of entities are selected and it might be necessary to use that selection set again, create a component out of it. Components are groups of entities but hold only one kind of entity at a time. Components can themselves be grouped into Assemblies, so this is the way to group different types of entities together. Use **Utility Menu > Select > Comp/Assembly > Create Component…** to create a component. The Component Manager (Fig. 2.19) makes it very easy to manage and manipulate groups and select/plot what analyst wants to see to the screen. This is found under **Utility Menu > Select > Component Manager.**
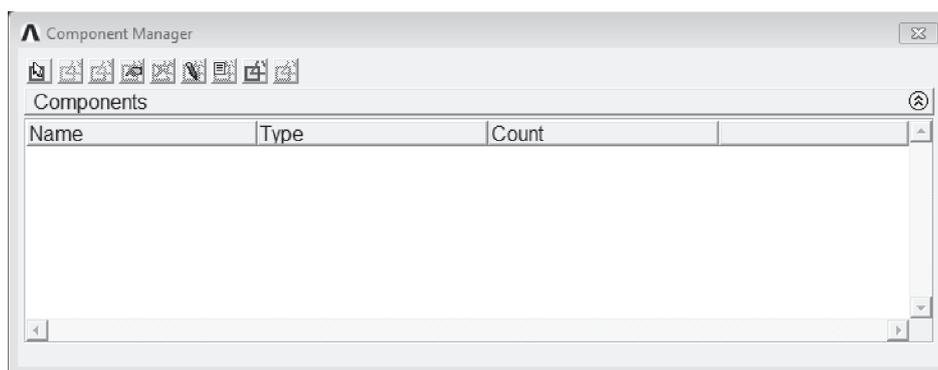

Fig. 2.19 Component Manager.

1. Use graphics *specification* functions to establish specifications (such as the viewing direction, number and color controls, etc.) for the display.
2. Use graphics *action* functions to actually produce the display.

The analyst can perform both types of graphics functions either by using menu functions in the GUI or by typing in commands directly.

2.8.3.1   GUI-Driven Graphics Functions

When running the program interactively, most users will prefer to use the GUI. As they use the GUI functions, they execute commands without actually seeing or editing them. (The program will record all underlying executed commands in your Jobname.LOG file.) Graphics specification functions can be accessed via **Utility Menu > PlotCtrls**. Graphics action functions reside under **Utility Menu > Plot.**

2.8.3.2   Immediate Mode Graphics

By default in the GUI, the model will immediately be displayed in the Graphics Window as new entities (such as areas, keypoints, nodes, elements, local coordinate systems, boundary conditions, etc.) are created. This is called *immediate mode* graphics. Anything drawn immediately in this way, however, will be destroyed if the analyst brings up a menu or dialog box on top of it. Or, if the GUI is iconified, the immediate mode graphics image will not be shown when the analyst restores the GUI icon.

An immediate image will also be automatically scaled to fit nicely within the Graphics Window — a feature called *automatic scaling*. Periodically, though, the analyst may need to issue an explicit plot function because new entities which lie 'outside' the boundaries of the scaled image are created already in the Graphics Window and are thus not captured with immediate mode graphics. The plot function will rescale and redraw the image.

To obtain a more 'permanent' image, execute one of the plot functions (such as **Utility Menu > Plot > Volumes**) or a graphics action command (such as **VPLOT**). An image generated in this way will not be destroyed by menu pop-ups or by iconifying the GUI. Also note that symbols (such as keypoint or node numbers, local coordinate systems, boundary conditions, etc.) are also shown immediately but will not be present on a 'permanent' display unless the analyst first 'turns on' the appropriate symbol using the functions under **Utility Menu > PlotCtrls** or the appropriate graphics specification command.

If the analyst prefers not to see things immediately as s/he defines them, s/he can use **Utility Menu > PlotCtrls > Erase Options > Immediate Display** to turn off immediate mode. When the program is run interactively *without* using the GUI, immediate mode is off by default.

2.8.3.3   Replotting the Current Display

The GUI path **Utility Menu > Plot > Replot** re-executes the last display action command that was executed. However, the program can execute that command only if it is valid in the current routine.

- NODE (nodes)
- ELEM (elements)
- KEYP (keypoints)
- LINE (lines)
- AREA (areas)
- VOLU (volumes)
- GRPH (graph displays)

2.8.4.3 *Choosing the Display Used for Plots*

When displaying either the ELEM or GRPH entity type, the analyst can control the type of element or graph display used for plots. Use: **Utility Menu > PlotCtrls > Multi-Plot Controls.** 'ALL' can be specified to have all windows use the selected display type, or that display type can be applied only to a specific window (default is window 1).

2.8.4.4 *Displaying Selected Entities*

Issue **Utility Menu > PlotCtrls > Multi-Plots or Utility Menu > Plot > Replot** to display the entities selected.

2.9   CREATING GRAPHS

A graph is used to review the material property curves, trace the time-history response of the system, or examine the relationship between any two items in the analysis. ANSYS graphs can be either 2-D (X-Y) or 3-D (X-Y-Z, where Z must always be *TIME*). Figure 2.20 shows two typical graphs:
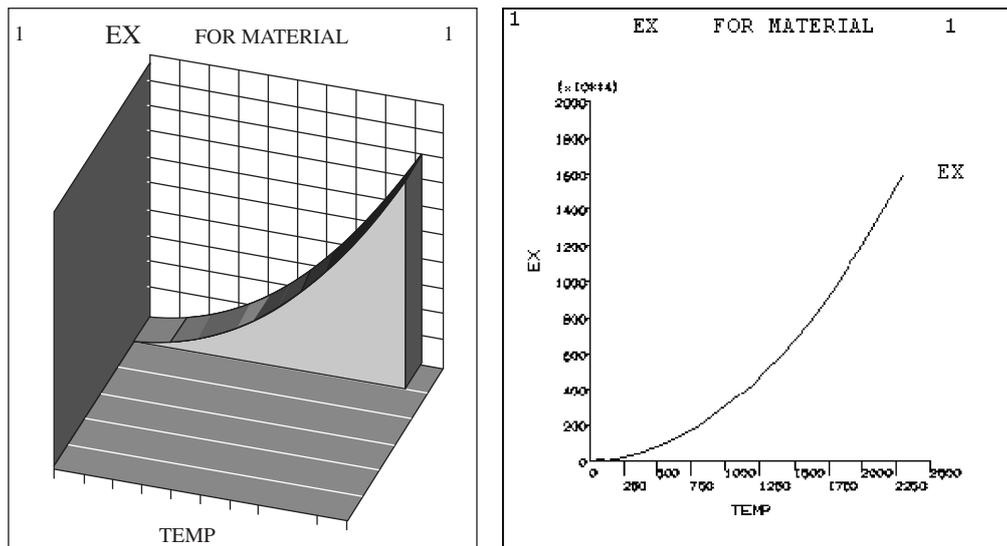

Fig. 2.20   Typical ANSYS Graphs.

The most convenient way to create and control graph displays is by using the GUI operations available under **Utility Menu > Plot** and **Utility Menu > PlotCtrls.** Alternatively, graphics action and control commands can be used.

2.9.1   Graph Display Actions

Use **Utility Menu > Plot > Materials** to display linear material properties as a function of temperature. Use **Utility Menu > Plot > Data Tables** to display nonlinear data curves family of commands. Use **Utility Menu > Plot > Array Parameters** to display column vectors of array parameters. Use **Main Menu > General Postproc > Fatigue > Store Stresses > Plot Stresses** to display a stress item associated with a particular location and event versus loading number.

2.9.2   Changing the Specifications for Graph Displays

2.9.2.1   Changing the Type, Style, and Color of the Graph Display

The appearance of the graph display can be altered as follows:

T*urning axis divisions (tick marks) on or off.* This feature can be controlled using the command **Utility Menu > PlotCtrls > Style > Graphs.**

Setting line thickness for axes, grid lines or graph curve lines. Graph items can be accentuated by increasing their line thickness, using the AXIS, GRID, and CURVE labels using **Utility Menu > Plot Ctrls > Style > Graphs.**

Turning the grid on or off (in the XY plane). A grid can be added to the graph displays, using **Utility Menu > Plot Ctrls > Style > Graphs.** A grid can be either a full grid or a partial grid.

Producing a dashed tolerance curve about the displayed curve. A range of data spread, tolerance, or uncertainty can be indicated on the graph curves using **Main Menu > TimeHist Postpro > Settings > Graph**.

Changing the color of curves (and color-filled areas under curves). The CURVE label on the **Utility Menu > PlotCtrls > Style > Colors >** *color type* allows controllingto control the color of each curve in the graph.

2.9.2.2   Labeling the Graph

Labeling the axes. The X and Y axes can be labeled using **Utility Menu > PlotCtrls > Style > Graphs**.

Labeling the curves. For POST26 plotted-variable graphs, the labels applied to the curves are established choosing: **Main Menu > TimeHist Postpro > Define Variables** or **Main Menu > TimeHist Postpro > Elec&Mag > Circuit > Define Variables.**

Adding user-defined graphics and text. Extra graphics and text can be added to the displays using the annotation functions by choosing **Utility Menu > PlotCtrls > Annotation**.

2.10   ANNOTATION

A common step in the analysis process is presenting a model and results in data with additional notations applied, such as dimensions, comments, highlights, or other text or artwork. The standard

ANSYS display can be enhanced with a variety of annotation primitives including text, dimensions, polygons, symbols, and even pie charts. The '!' and '$' characters are not available for text annotation.

2-D text and graphics annotations are formed as a 2-D overlay on the graphics screen. Because this overlay exists as an imaginary plane, when the analyst transforms the model, the constructed annotation will not move with the model. Because of this, 2-D annotation should be used primarily for finalized reports and printouts. Access 2-D annotation functions through **Utility Menu > PlotCtrls > Annotation > Create 2D Annotation.**

The following annotation primitives are available from the 2-D annotation dialog box:

- Text
- Dimensions
- Polygons
- Rectangles
- Symbols
- Arrows
- Arcs
- Lines
- Pies
- Wedges
- Circles

On choosing **Utility Menu > PlotCtrls > Annotation > Create 2D Annotation**, the text annotation dialog box appears (Fig. 2.21). Text annotation can be applied either as stroke text or as bitmap fonts. The '!' and '$' characters cann't be used in ANSYS text annotation.

Fig. 2.21   Stroke Text Annotation Dialog Box.

The fields and buttons presented in the annotation dialog box change when the annotation entity type is reset. For example, if the annotation entity is reset to arcs, the dialog box shown, changes to display the options available for annotation arcs. Regardless of which annotation entity is choosen, the annotation dialog box always displays four action buttons:

- **Undo** – Erases the last annotation entity created.
- **Refresh** – Redisplays the annotation, which is useful after move and delete operations.
- **Close** – Closes the annotation dialog box.

reference or save the image to a bitmap (*.bmp*) file. Note that although a windows bitmap file is not compressed, when it is inserted into Word it does get compressed automatically so the analyst doesn't end up with a huge bloated document.

Output a vector image: **Utility Menu > Plot Ctrls > Redirect Plots > To PSCR File….** A Postscript file is a vector file, which means that it is a 2D representation of all of the entities in the graphics window in an editable format. Because it is not a bitmap, it can be scaled to any size without losing any resolution, and is always very crisp looking. It can also be imported into a technical illustration program and manipulated very easily: change the colors, add annotations, change or resize fonts, etc. All this can be done in ANSYS but it can be quicker in an illustration package. One caution about Postscript files! Since they actually write out every entity in the model, if the model is large (say a tet mesh of a CAD model) this file can be huge. It is best suited for getting very crisp images of smallish models or wireframe displays. Microsoft Word will not display the image until it is printed.

Output a bitmap image: **Utility Menu > Plot Ctrls > Redirect Plots > To xyz File…,** where *xyz* is JPEG, TIFF, PNG, etc. These file formats produce good images with reasonably small file sizes. The size of the image file for these formats is not dependant on the size of the model like Postscript.

Controlling the Way the Model Looks

All of the visual aspects of what is seen in the graphics window are controlled from the 'Plot' and 'Plot Ctrls' pull downs from the Utility menu. Use **'Utility Menu > Plot'** to plot different types of entities to the screen. Use **'Utility Menu > Plot Ctrls'** to control the characteristics of what is to be plotted.

    'Utility Menu > Plot Ctrls > Numbering': Entity Numbers on and off.

   **'Utility Menu > Plot Ctrls > Symbols':** Turn various markers and symbols on and off (Fig. 2.23).

    'Utility Menu > Plot Ctrls > Style': Change hidden line, element edges, element shrink, etc.

    'Utility Menu > Plot Ctrls > Device Options': Change between solid shaded and wireframe display.

2.13   TYPICAL MODELING DIFFICULTIES

Certain modeling problems can be considered 'typical'. A typical analyst modeling problem is the case of keypoints, lines, areas, volumes, nodes, and elements that are identical and occupy the same space. This can lead to erroneous models. Proper use of the merge command can eliminate many instances of these problems. The merge can fail if, for example, two elements share the same space, but were defined via alternative sequences of nodes (e.g., elements in the same place, one numbered by nodes selected clockwise, the other counterclockwise).

    Another problem is failure of keypoints, or lines, or areas to be shared by higher geometric modeling entities. When this happens, the higher entities are not 'fused' or 'welded' together as intended. Consequently, the elements will not share nodes along what should have been the common boundary. The analyst must always use caution and double-check everything while developing a model.
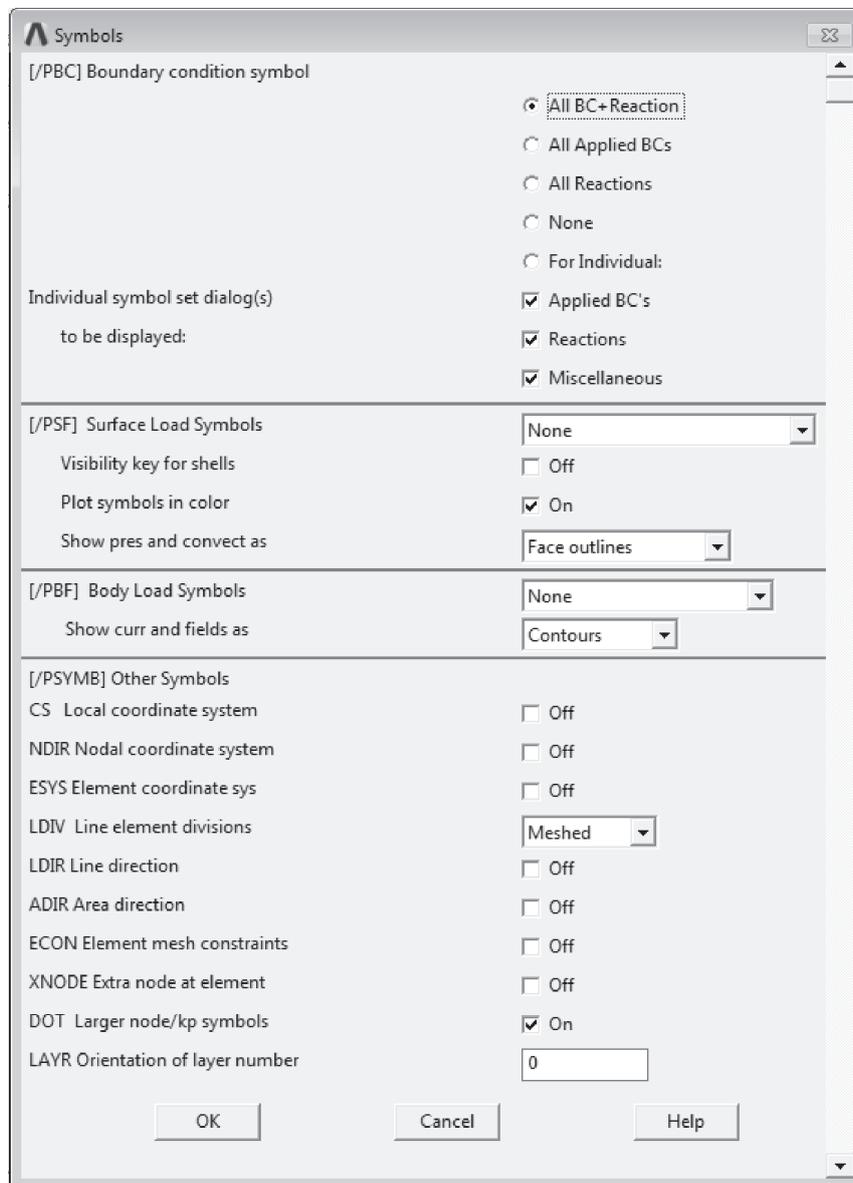
Fig. 2.23

A problem most analysts will encounter is to make a change to a model in error, long after the database was saved. The analyst will have to learn to use a text editor on the log file, to extract that portion of the log file after the last time the database was saved, or retrieved (whichever was most recent). Remove the offending command. That portion of the log file will have to be run on the model as it was the last time it was saved or retrieved. Make sure to be in the correct part of ANSYS

(usually /PREP7) when reading in the instructions with /INPUT. The same method can apply if the computer is subjected to a power failure, or if ANSYS crashes without leaving an '*ansabort.db*' file. After re-starting, take the text editor to the log file, and re-run the appropriate instructions on the model database file as it was when last saved or retrieved.

The most common of all errors in Finite Element Modeling is the incorrect application of loads and boundary conditions. This must be thought about very carefully. Most models (not all) are prevented from undergoing free body motion in 2-D or 3-D space, by eliminating at least a minimal number of degrees of freedom (2 translations plus 1 rotation in 2-D, and 3 translations plus 3 rotations in 3-D). Rotations can be prevented either by having constraints on translations at enough distinct nodes in space, or by directly constraining a rotational degree of freedom at a node. A common check on results is to see whether the sums of the reaction forces at the constrained nodes equal the sums of the applied forces and gravity loads.

2.14   MECHANICAL TOOLBAR (MTB)

The Mechanical Toolbar (MTB) is a *streamlined* GUI designed to do *mechanical* analyses quickly and easily. Main advantages of using MTB:
  – Easy to use. Tabs in the toolbar lead the analyst through the analysis steps in sequence.
  – Provides more ways to apply loads, such as total force on an area or line.
  – Handles multiple loading environments well, with analyst-assigned names and automatic 'book-keeping' of files.
  – Allows automatic generation of an HTML report of the analysis.
    The MTB does have a few limitations, mainly in view of keeping it simple and easy to use:
  – No solid modeling capabilities. All geometry is imported via IGES as well as Connection products. Analyst can also 'import' geometry from an *ANSYS .db* file.
  – Contains only commonly used graphics controls and postprocessing functions. However, the Utility Menu is available and provides access to all graphics functions.
  – No nonlinearities or Multiphysics capabilities.

Click on **Utility Menu > MenuCtrls > Mechanical Toolbar** to start the MTB (Fig. 2.24). The 'Mechanical Toolbar' window will appear, replacing the Main Menu, Toolbar and Input window. Use **Utiliy Menu > MenuCtrls** to activate them to use the Toolbar or Input window (Fig. 2.25). If a model is already there in the database, ANSYS will save it to *preMT.db,* which can be used later to 'import' the geometry portion.
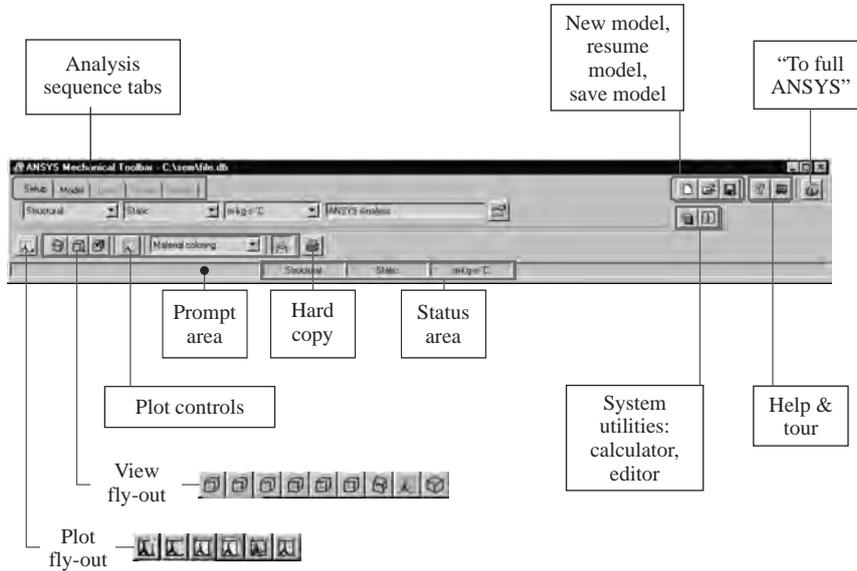

Fig. 2.24 Mechanical Tool Bar

Fig. 2.25

Use the Setup tab (Fig. 2.26) to specify analysis discipline, analysis type, units system, title and MTB properties.
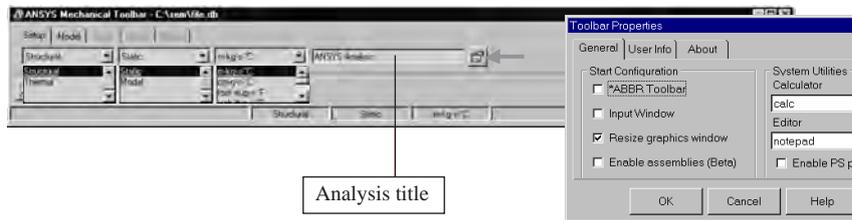

Fig. 2.26

The Model tab (Fig. 2.27) is used to import geometry, assign attributes to geometry and create the mesh. If no mesh is created, MTB will automatically mesh it during solution.
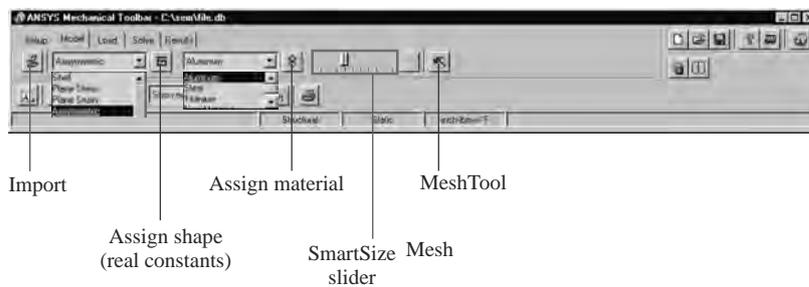

Fig. 2.27

The Load tab allows (Fig. 2.28) to specify environment name(s) and apply or delete loads.
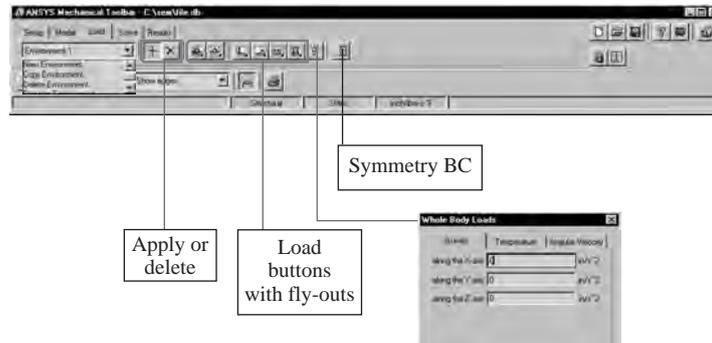

Fig. 2.28

The Solve tab (Fig. 2.29) allows to solve now or at a specified later time.
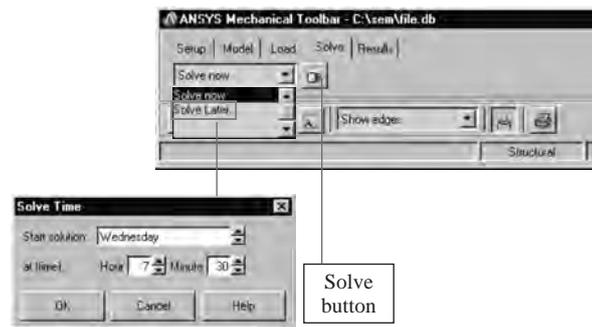

Fig. 2.29

The Results tab (Fig. 2.30) is used to review displaced shape and contour plots for the desired environment, query, animate, or list results and generate an HTML report, which can be e-mailed or posted on the web.
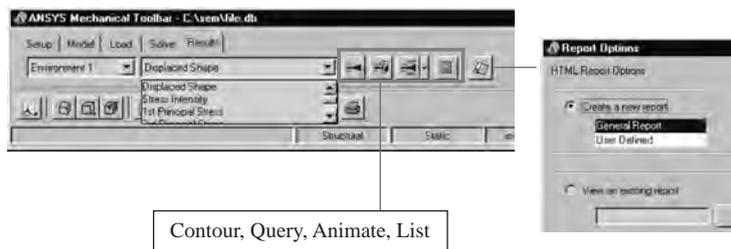

Fig. 2.30