Chapter 2

FUNDAMENTALS OF ANSYS

2.1 Useful Definitions

Before delving into the details of the procedures related to the ANSYS program, we define the following terms:

Jobname: A specific name to be used for the files created during an ANSYS session. This name can be assigned either before or after starting the ANSYS program.

Working Directory: A specific folder (directory) for ANSYS to store all of the files created during a session. It is possible to specify the *Working Directory* before or after starting ANSYS.

Interactive Mode: This is the most common mode of interaction between the user and the ANSYS program. It involves activation of a platform called *Graphical User Interface* (*GUI*), which is composed of menus, dialog boxes, push-buttons, and different windows. *Interactive Mode* is the recommended mode for beginner ANSYS users as it provides an excellent platform for learning. It is also highly effective for postprocessing.

Batch Mode: This is a method to use the ANSYS program without activating the *GUI*. It involves an *Input File* written in *ANSYS Parametric Design Language* (*APDL*), which allows the use of parameters and common programming features such as *DO* loops and *IF* statements. These capabilities make the *Batch Mode* a very powerful analysis tool. Another distinct advantage of the *Batch Mode* is realized when there is an error/mistake in the model generation. This type of problem can be fixed by modifying a small portion of the *Input File* and reading it again, saving the user a great deal of time.

Combined Mode: This is a combination of the *Interactive* and *Batch Modes* in which the user activates the *GUI* and reads the *Input File*. Typically, this method allows the user to generate the model and obtain the solution using the *Input File* while reviewing the results using the

Postprocessor within the *GUI*. This method combines the salient advantages of the *Interactive* and *Batch Modes*.

2.2   Before an ANSYS Session

The construction of solutions to engineering problems using FEA requires either the development of a computer program based on the FEA formulation or the use of a commercially available general-purpose FEA program such as ANSYS. The ANSYS program is a powerful, multi-purpose analysis tool that can be used in a wide variety of engineering disciplines. Before using ANSYS to generate an FEA model of a physical system, the following questions should be answered based on engineering judgment and observations:

- What are the objectives of this analysis?
- Should the entire physical system be modeled, or just a portion?
- How much detail should be included in the model?
- How refined should the finite element mesh be?

In answering such questions, the computational expense should be balanced against the accuracy of the results. Therefore, the ANSYS finite element program can be employed in a correct and efficient way after considering the following:

- Type of problem.
- Time dependence.
- Nonlinearity.
- Modeling idealizations/simplifications.

Each of these topics is discussed in this section.

2.2.1   Analysis Discipline

The ANSYS program is capable of simulating problems in a wide range of engineering disciplines. However, this book focuses on the following disciplines:

Structural Analysis: Deformation, stress, and strain fields, as well as reaction forces in a solid body.

Thermal Analysis: Steady-state or time-dependent temperature field and heat flux in a solid body.

2.2.1.1   *Structural Analysis*

This analysis type addresses several different structural problems, for example:

Static Analysis: The applied loads and support conditions of the solid body do not change with time. Nonlinear material and geometrical properties such as plasticity, contact, creep, etc., are available.

Modal Analysis: This option concerns natural frequencies and modal shapes of a structure.

Harmonic Analysis: The response of a structure subjected to loads only exhibiting sinusoidal behavior in time.

Transient Dynamic: The response of a structure subjected to loads with arbitrary behavior in time.

Eigenvalue Buckling: This option concerns the buckling loads and buckling modes of a structure.

2.2.1.2 Thermal Analysis

This analysis type addresses several different thermal problems, for example:

Primary Heat Transfer: Steady-state or transient conduction, convection and radiation.

Phase Change: Melting or freezing.

Thermomechanical Analysis: Thermal analysis results are employed to compute displacement, stress, and strain fields due to differential thermal expansion.

2.2.1.3 Degrees of Freedom

The ANSYS solution for each of these analysis disciplines provides nodal values of the field variable. This primary unknown is called a degree of freedom (DOF). The degrees of freedom for these disciplines are presented in Table 2.1. The analysis discipline should be chosen based on the quantities of interest.

Table 2.1 Degrees of freedom for structural and thermal analysis disciplines.

Discipline	Quantity	DOF
Structural	Displacement, stress, strain, reaction forces	Displacement
Thermal	Temperature, flux	Temperature

2.2.2   Time Dependence

The analysis with ANSYS should be time-dependent if:

- The solid body is subjected to time varying loads.
- The solid body has an initially specified temperature distribution.
- The body changes phase.

2.2.3   Nonlinearity

Most real-world physical phenomena exhibit nonlinear behavior. There are many situations in which assuming a linear behavior for the physical system might provide satisfactory results. On the other hand, there are circumstances or phenomena that might require a nonlinear solution. A nonlinear structural behavior may arise because of geometric and material nonlinearities, as well as a change in the boundary conditions and structural integrity. These nonlinearities are discussed briefly in the following subsections.

2.2.3.1   *Geometric Nonlinearity*

There are two main types of geometric nonlinearity:

> ***Large deflection and rotation:*** If the structure undergoes large displacements compared to its smallest dimension and rotations to such an extent that its original dimensions and position, as well as the loading direction, change significantly, the large deflection and rotation analysis becomes necessary. For example, a fishing rod with a low lateral stiffness under a lateral load experiences large deflections and rotations.

> ***Stress stiffening:*** When the stress in one direction affects the stiffness in another direction, stress stiffening occurs. Typically, a structure that has little or no stiffness in compression while having considerable stiffness in tension exhibits this behavior. Cables, membranes, or spinning structures exhibit stress stiffening.

2.2.3.2   *Material Nonlinearity*

A typical nonlinear stress-strain curve is given in Fig. 2.1. A linear material response is a good approximation if the material exhibits a nearly linear stress-strain curve up to a proportional limit and the loading is in a manner that does not create stresses higher than the yield stress anywhere in the body.
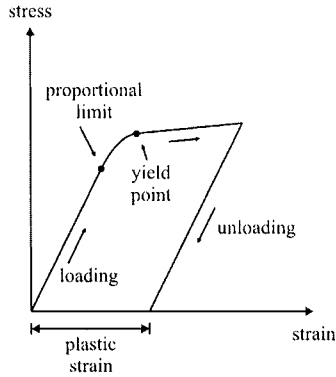
Fig. 2.1 Non-linear material response.

Nonlinear material behavior in ANSYS is characterized as:

Plasticity: Permanent, time-independent deformation.

Creep: Permanent, time-dependent deformation.

Nonlinear Elastic: Nonlinear stress-strain curve; upon unloading, the structure returns back to its original state—no permanent deformations.

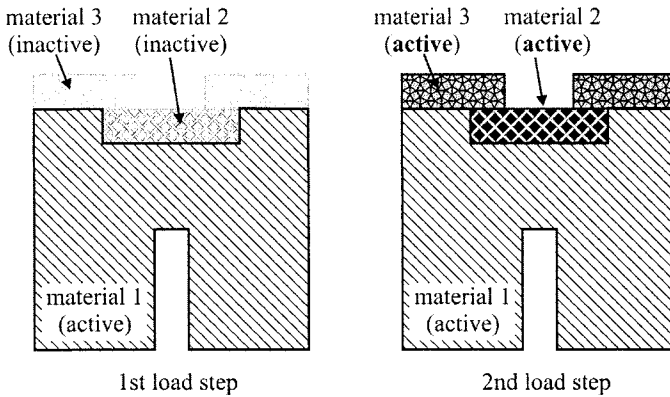*Viscoelasticity:* Time-dependent deformation under constant load. Full recovery upon unloading.

Hyperelasticity: Rubber-like materials.

2.2.3.3   Changing-status Nonlinearity

Many common structural features exhibit nonlinear behavior that is status dependent. When the status of the physical system changes, its stiffness shifts abruptly. The ANSYS program offers solutions to such phenomena through the use of nonlinear contact elements and birth and death options. This type of behavior is common in modeling manufacturing processes such as that of a shrink-fit (Fig. 2.2).

2.2.4   Practical Modeling Considerations

In order to reduce computational time, minor details that do not influence the results should not be included in the FE model. Minor details can also be ignored in order to render the geometry symmetric, which leads to a reduced FE model. However, in certain structures, "small" details such as fillets or holes may be the areas of maximum stress, which might prove to be extremely important in the analysis and design. Engineering judgment is essential to balance the possible gain in computational cost against the loss of accuracy.

material 3   material 2   material 3   material 2
(inactive)   (inactive)   (active)   (active)

material 1   material 1
(active)   (active)

1st load step   2nd load step

Fig. 2.2 Element birth and death used in a manufacturing problem.

2.2.4.1 Symmetry Conditions

If the physical system under consideration exhibits symmetry in geometry, material properties, and loading, then it is computationally advantageous to model only a representative portion. If the symmetry observations are to be included in the model generation, the physical system must exhibit symmetry in all of the following:

- Geometry.
- Material properties.
- Loading.
- Degree of freedom constraints.

Different types of symmetry are:

- Axisymmetry.
- Rotational symmetry.
- Planar or reflective symmetry.
- Repetitive or translational symmetry.

Examples for each of the symmetry types are shown in Fig. 2.3. Each of these symmetry types is discussed below.

Axisymmetry: As illustrated in Fig. 2.4, axisymmetry is the symmetry about a central axis, as exhibited by structures such as light bulbs, straight pipes, cones, circular plates, and domes.

Rotational Symmetry: A structure possesses rotational symmetry when it is made up of repeated segments arranged about a central axis. An example is a turbine rotor (see Fig. 2.5).
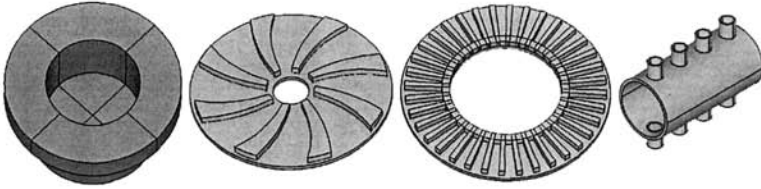
Fig. 2.3 Types of symmetry conditions (from left to right): axisymmetry, rotational, reflective/planar, and repetitive/translational.
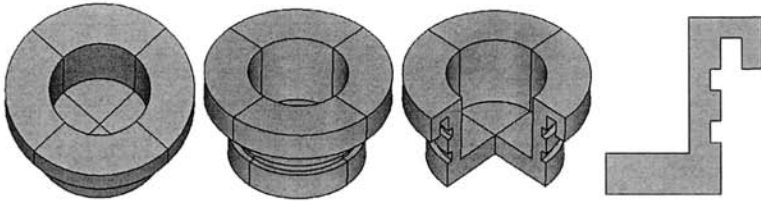
Fig. 2.4 Different views of a 3-D body with axisymmetry and its cross section (far right).
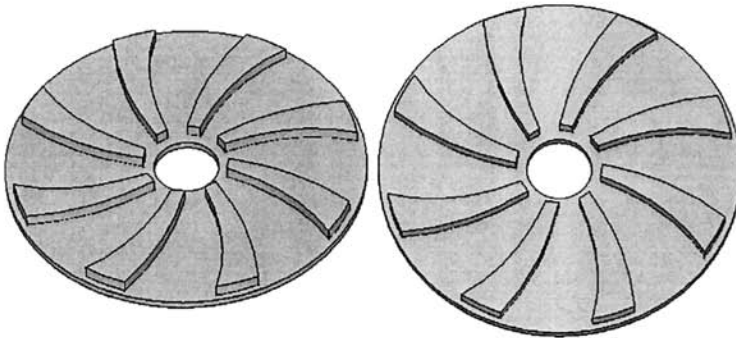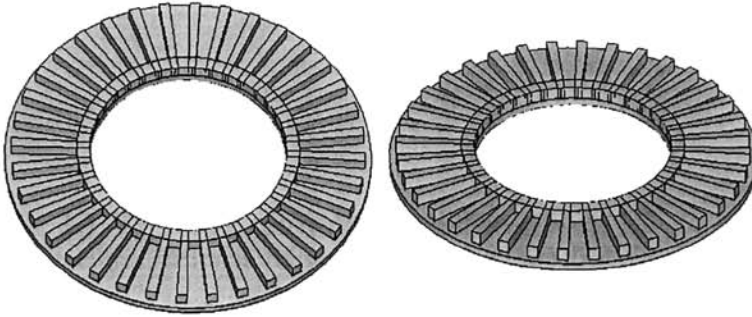

**Fig. 2.5** Different views of a 3-D body with rotational symmetry.

Planar or Reflective Symmetry When one-half of a structure is a mirror image of the other half, planar or reflective symmetry exists, as shown in Fig. 2.6. In this case, the plane of symmetry is located on the surface of the mirror.

Repetitive or Translational Symmetry Repetitive or translational symmetry exists when a structure is made up of repeated segments lined up in a row, such as a long pipe with evenly spaced cooling fins, as shown in Fig. 2.7.

Symmetry in Material Properties, Loading, Displacements Once symmetry in geometry is observed, the same symmetry plane or axis should also be valid for the material properties, loading (forces, pressure, etc.), and

**Fig. 2.6** Different views of a 3-D body with reflective/planar symmetry.
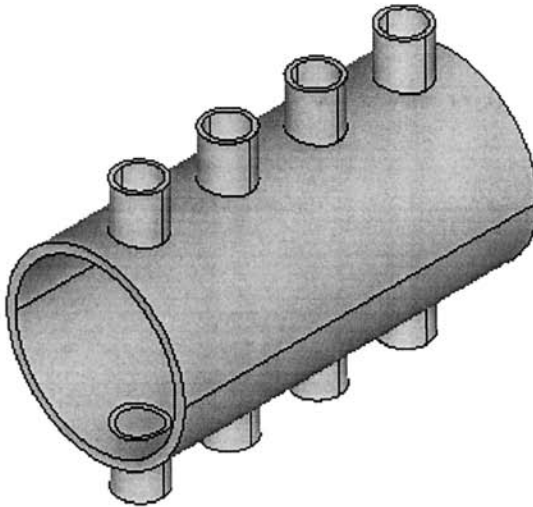
Fig. 2.7 A 3-D body with repetitive/translational symmetry.

constraints. For example, a homogeneous and isotropic square plate with a hole at the center under horizontal tensile loading (Fig. 2.8) has octant ($1/8^{th}$) symmetry in both geometry and material with respect to horizontal, vertical, and both diagonal axes. However, the loading is symmetric with respect to horizontal and vertical axes only. Therefore, a quarter of the structure is required in the construction of the solution.

If the applied loading varies in the vertical direction, as shown in Fig. 2.9, the loading becomes symmetric with respect to the vertical axis only. Although the geometry exhibits octant symmetry, half-symmetry is necessary in order to construct the solution.
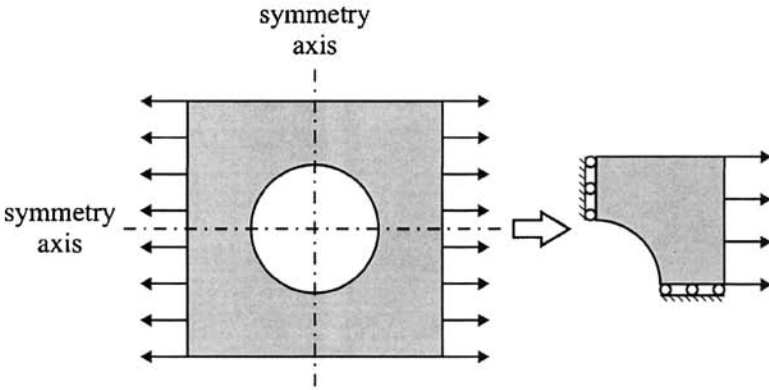
Fig. 2.8 Example of quarter-symmetry.
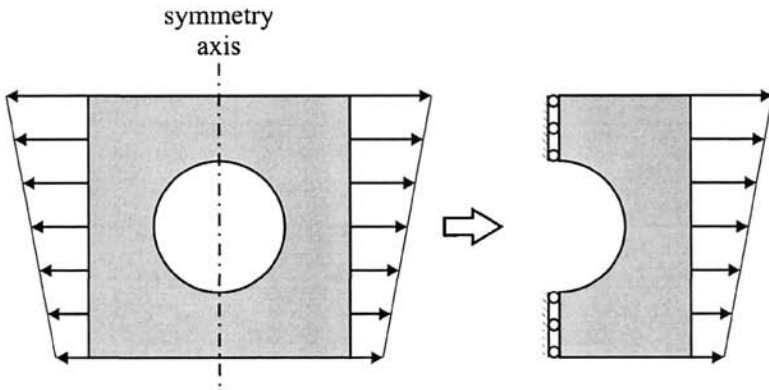
Fig. 2.9 Example of half-symmetry with respect to vertical axis.

A similar plate, this time composed of two dissimilar materials is shown in Fig. 2.10. The loading condition allows for quarter-symmetry; however, the material properties are symmetric with respect to the horizontal axis only. Therefore, it is limited to half-symmetry. If this plate is subjected to a horizontal tensile load varying in the vertical direction, as shown in Fig. 2.11, no symmetry condition is present.

Since a structure may exhibit symmetry in one or more of the afore-mentioned categories, one should try to find the smallest possible segment of the structure that would represent the entire structure. If the physical system exhibits symmetry in geometry, material properties, loading, and displacement constraints, it is computationally advantageous to use symmetry in the analysis. Typically, the use of symmetry produces better results as it leads to a finer, more detailed model than would otherwise be possible.
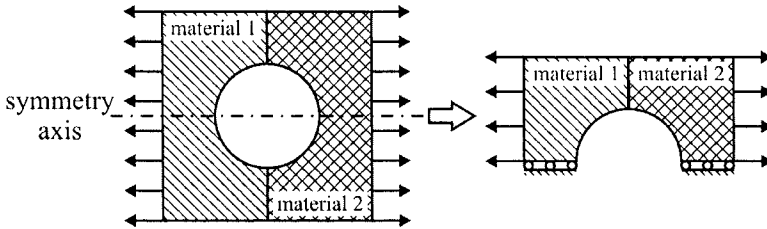
Fig. 2.10 Example of half-symmetry with respect to horizontal axis.
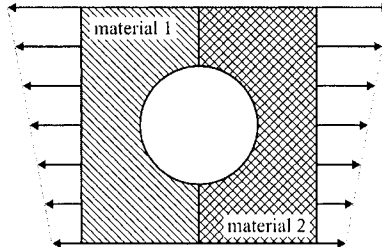

Fig. 2.11 Example of no symmetry.

A three-dimensional finite element mesh of the structure shown in Fig. 2.12 contains 18,739 tetrahedral elements with 5,014 nodes. However, the two-dimensional mesh of the cross section necessary for the axisymmetric analysis has 372 quadrilateral elements and 447 nodes. The use of symmetry in this case reduces the CPU time required for the solution while delivering the same level of accuracy in the results.

2.2.4.2 Mesh Density

In general, a large number of elements provide a better approximation of the solution. However, in some cases, an excessive number of elements may increase the round-off error. Therefore, it is important that the mesh is adequately fine or coarse in the appropriate regions. How fine or coarse the mesh should be in such regions is another important question. Unfortunately, definitive answers to the questions about mesh refinement are not available since it is completely dependent on the specific physical system considered. However, there are some techniques that might be helpful in answering these questions:

Adaptive Meshing: The generated mesh is required to meet acceptable energy error estimate criteria. The user provides the "acceptable" error level information. This type of meshing is available only for linear static structural analysis and steady-state thermal analysis.
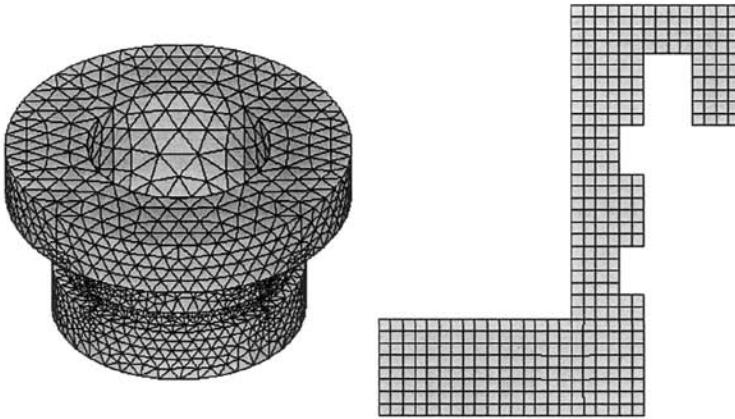
Fig. 2.12 Three-dimensional mesh of a structure (left) and 2-D mesh of the same structure (right) using axisymmetry.

Mesh Refinement Test Within ANSYS: An analysis with an initial mesh is performed first and then reanalyzed by using twice as many elements. The two solutions are compared. If the results are close to each other, the initial mesh configuration is considered to be adequate. If there are substantial differences between the two, the analysis should continue with a more-refined mesh and a subsequent comparison until convergence is established.

Submodeling: If the mesh refinement test yields nearly identical results for most regions and substantial differences in only a portion of the model, the built-in "submodeling" feature of ANSYS should be employed for localized mesh refinement. This feature is described in Chap. 11.

2.3 Organization of ANSYS Software

There are two primary levels in the ANSYS program, as shown in Fig. 2.13:

Begin Level: Gateway into and out of ANSYS and platform to utilize some global controls such as changing the *jobname*, etc.

Processor Level: This level contains the processors (preprocessor, solution, postprocessor, etc.) that are used to conduct finite element analyses.

ENTER   EXIT
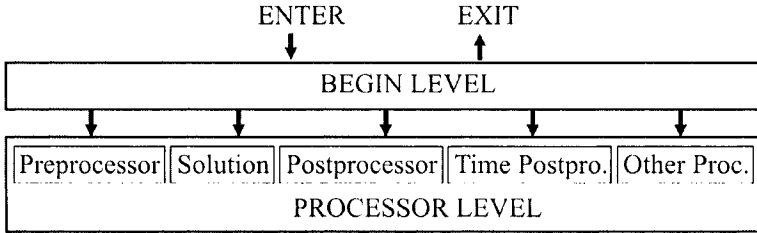
Fig. 2.13 Schematic of ANSYS levels.

The user is in the *Begin Level* upon entering the ANSYS program. One can proceed to the *Processor Level* by clicking the mouse on one of the processor selections in the ANSYS *Main Menu.*

2.4   ANSYS Analysis Approach

There are three main steps in a typical ANSYS analysis:

- Model generation:
  - Simplifications, idealizations.
  - Define materials/material properties.
  - Generate finite element model (mesh).

- Solution:
  - Specify boundary conditions.
  - Obtain the solution.

- Review results:
  - Plot/list results.
  - Check for validity.

Each of these steps corresponds to a specific processor or processors within the *Processor Level* in ANSYS. In particular, model generation is done in the *Preprocessor* and application of loads and the solution is performed in the *Solution Processor.* Finally, the results are viewed in the *General Postprocessor* and *Time History Postprocessor* for steady-state (static) and transient (time-dependent) problems, respectively. There are several other processors within the ANSYS program. These mostly concern optimization- and probabilistic-type problems. The most commonly used processors are described in the following subsections.

2.4.1 ANSYS Preprocessor

Model generation is conducted in this processor, which involves material definition, creation of a solid model, and, finally, meshing. Important tasks within this processor are:

- Specify *element type*.
- Define *real constants* (if required by the element type).
- Define *material properties*.
- Create the model geometry.
- Generate the mesh.

Although the boundary conditions can also be specified in this processor, it is usually done in the *Solution Processor*.

2.4.2 ANSYS Solution Processor

This processor is used for obtaining the solution for the finite element model that is generated within the *Preprocessor*. Important tasks within this processor are:

- Define *analysis type* and *analysis options*.
- Specify boundary conditions.
- Obtain solution.

2.4.3 ANSYS General Postprocessor

In this processor, the results at a specific time (if the *analysis type* is transient) over the entire or a portion of the model are reviewed. This includes the plotting of contours, vector displays, deformed shapes, and listings of the results in tabular format.

2.4.4 ANSYS Time History Postprocessor

This processor is used to review results at specific points in time (if the *analysis type* is transient). Similar to the General Postprocessor, it provides graphical variations and tabular listings of results data as functions of time.

2.5 ANSYS File Structure

Several files are created during a typical ANSYS analysis. Some of these files are in ASCII format while the others are binary. Brief descriptions of common file types are given below.

2.5.1   Database File

During a typical ANSYS analysis, input and output data reside in memory until they are saved in a *Database File,* which is saved in the *Working Directory*. The syntax for the name of the *Database File* is *jobname.db*. This binary file includes the element type, material properties, geometry (solid model), mesh (nodal coordinates and element connectivity), and the results if a solution is obtained. Once the *Database File* is saved, the user can resume from this file at any time. There are three distinct ways to save and resume the *Database File*:

- Use the *Utility Menu.*
- Click on *SAVE_DB* or *RESUM_DB* button on the *ANSYS Toolbar.*
- Issue the command **SAVE** or **RESUME** in the *Input Field.*

2.5.2   Log File

The *Log File* is an ASCII file, which is created (or resumed) immediately upon entering ANSYS. Every action taken by the user is stored sequentially in this file in command format (ANSYS Parametric Design Language (APDL)). The syntax for the name of the *Log File,* which is also saved in the *Working Directory*, is *jobname.log*. If *jobname.log* already exists in the *Working Directory*, ANSYS appends the newly executed actions instead of overwriting the file. The *Log File* can be utilized to:

- Understand how an analysis was performed by another user.
- Learn the command equivalents of the actions taken within ANSYS.

2.5.3   Error File

Similar to the *Log File*, the *Error File* is an ASCII file, which is created (or resumed) immediately upon entering ANSYS. This file captures all warning and error messages issued by ANSYS during a session. It is saved in the *Working Directory* with the following syntax for the name: *jobname.err*. If *jobname.err* already exists in the *Working Directory*, ANSYS appends the newly issued warning and error messages instead of overwriting the file. This file is particularly important when ANSYS issues several warning and error messages too quickly during an interactive session. The user can then consult the *Error File* to discover the exact cause(s) of each of the warnings or errors.

2.5.4   Results Files

The results of an ANSYS analysis are stored in a separate *Results File*. This file is a binary file and, depending upon the *Analysis Type*, the file's

extension takes a different form. The following syntax applies to the *Results File* name for the selected *Analysis Type*:

Structural analysis: *jobname.rst*
Thermal analysis: *jobname.rth*
Fluids analysis: *jobname.rfl*

2.6 Description of ANSYS Menus and Windows

When using the ANSYS program in *Interactive Mode*, the *Graphical User Interface (GUI)* is activated. The *GUI* has six distinct components:

> *Utility Menu:* Contains functions that are available throughout the ANSYS session, such as file controls, selecting, graphic controls, and parameters. The ANSYS *Help System* is also accessible through this menu.

> *Main Menu*: Contains the primary ANSYS functions organized by processors (*Preprocessor*, *Solution*, *General Postprocessor*, etc.).

> *Toolbar*: Contains push-buttons for executing commonly used ANSYS commands and functions. Customized buttons can be created.

> *Input Field*: Displays a text field for typing commands. All previously typed commands are stored in a pull-down menu for easy reference and access.

> *Graphics Window:* Displays the graphical representation of the models/ meshes created within ANSYS. Also, the related results are reviewed in this window.

> *Output Window:* Receives text output from the program. This window is usually positioned behind other windows and can be raised to the front when necessary.

Figure 2.14 shows a typical ANSYS *GUI* with each of the preceding components identified.
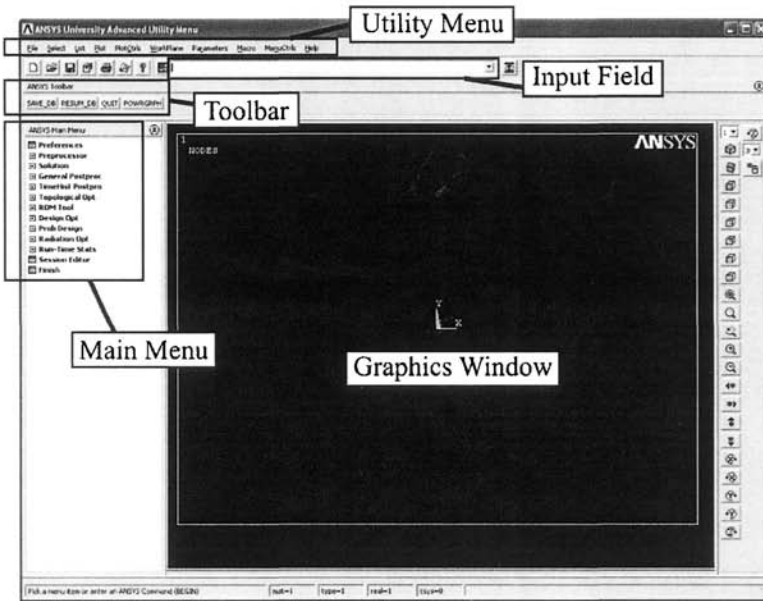
Fig. 2.14 Typical ANSYS GUI with separate components identified.

2.6.1   Utility Menu

The *Utility Menu* contains utility functions that are independent of *ANSYS Levels* (i.e., begin and processor levels), with some exceptions. The *Utility Menu* contains ten items, each of which brings up a pull-down menu of subitems. Clicking the left mouse button on these subitems will result in one of the following:

- Bring up a submenu, indicated by the icon ▶ .
- Immediately execute a function.
- Bring up a dialog box, indicated by the icon ⋯ .
- Bring up a picking menu, indicated by the icon ✛ .

Brief descriptions of each of the menu items under the *Utility Menu* are given below.

> *File* item under *Utility Menu*: Contains file- and database-related functions, such as clearing the database, reading an input file, saving the database to a file, or resuming a database from a file. This menu item can be used to exit the program.

> *Select* item under *Utility Menu*: Includes functions that allow the user to select a subset of data and to create *Components*.

List item under *Utility Menu*: This menu item allows the user to list any data stored in the ANSYS database. Also, status information about different areas of the program and contents of files in the system are available.

Plot item under *Utility Menu*: This menu item allows the user to plot ANSYS entities such as keypoints, lines, areas, volumes, nodes, and elements. If a solution is obtained, results can also be plotted through this menu item.

PlotCtrls item under *Utility Menu*: Contains functions that control the view, style, and other characteristics of graphic displays.

WorkPlane item under *Utility Menu*: Use of *WorkPlane* offers great convenience for *Solid Model* generation. This menu item enables the user to toggle the *Working Plane* on and off, and to move, rotate, and maneuver it. Coordinate system operations are also performed under this menu item.

Parameters item under *Utility Menu*: Contains functions to define, edit, and delete scalar and array parameters.

Macro item under *Utility Menu*: This menu item allows the user to execute *Macros* and data blocks. Under this menu item, the user can also manipulate the push-buttons on the *Toolbar*.

MenuCtrls item under *Utility Menu*: Allows the user to format the menus, as well as manipulate the *Toolbar*.

Help item under *Utility Menu*: Brings up the ANSYS *Help System*.

2.6.2 Main Menu

The *Main Menu* contains main ANSYS functions and processors, such as the preprocessor, solution, and postprocessor. It has a tree structure, where menus and submenus can be expanded and collapsed. Similar to the *Utility Menu*, clicking the left mouse button on the Main Menu items results in one of the following:

- Expand or collapse the submenus attached to the menu item, indicated by icons ⊞ and ⊟, respectively.
- Bring up a dialog box, indicated by the icon ▦ .
- Bring up a picking menu, indicated by the icon ⟑ .

2.6.3   Toolbar

The *Toolbar* contains a set of push-buttons that execute frequently used ANSYS functions. When the user starts ANSYS, predefined push-buttons such as *QUIT*, *SAVE_DB*, and *RESUM_DB* appear in the toolbar. The user can create customized push-buttons and delete or edit the existing ones.

2.6.4   Input Field

This field allows the user to type in commands directly as opposed to the use of menu items. The *Input Field* consists of two main regions:

- Command entry box.
- History buffer.

2.6.5   Graphics Window

All ANSYS graphics are displayed in the *Graphics Window*. Also, the user performs all of the graphical "picking" in this window.

2.6.6   Output Window

All of the text output generated as a result of command responses, warnings, and errors appear in the *Output Window*. It is positioned behind the main ANSYS window, but can be raised to the front when necessary.

2.7   Using the ANSYS Help System

Information on ANSYS procedures, commands, and concepts can be found in the ANSYS *Help System*. The importance of knowing how to use the *Help System* cannot be overemphasized. It can be accessed within the *Graphical User Interface* (*GUI*) in three ways:

- By choosing the *Help* menu item under *Utility Menu*.
- By pressing the *Help* button within dialog boxes.
- By entering the **HELP** command directly in the *Input Field*.

The *Help System* is also available as a stand-alone program outside of ANSYS. The user can bring up the desired help topic by choosing it from the system's table of contents or index, through a word search, or by choosing a hypertext link. The *Help System* is built on the HTML platform in the form of web pages. As indicated in Fig. 2.15, there are three tabs on the left of the *Help Window*: *Contents*, *Index*, and *Search*. The help pages are displayed on the right side of the *Help Window*. Selected topics regarding the *Help System* are discussed in the following subsections.
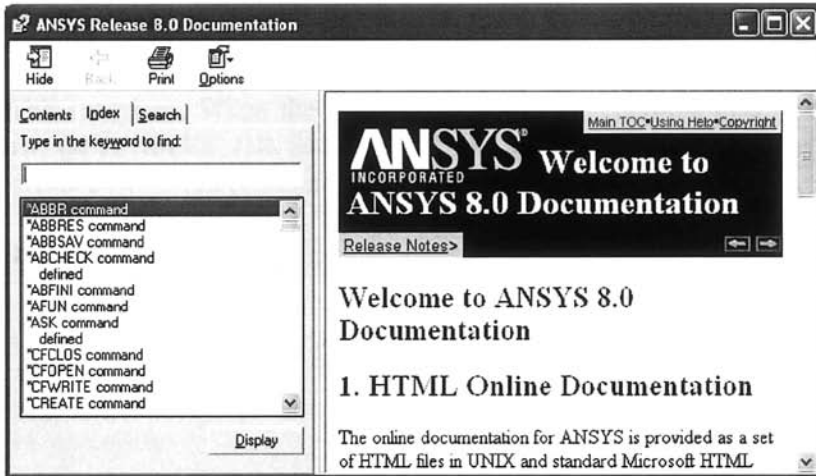
Fig. 2.15 ANSYS *Help System.*

2.7.1 Help Contents

The first tab on the left side of the *Help Window* is the *Contents Tab*, as shown in Fig. 2.16. It is a collection of several different ANSYS *Manuals* containing thousands of pages. The *Contents Tab* is organized in a tree structure for easy navigation. It is recommended that beginner ANSYS users take the time to read the relevant chapters in each *Manual*. Throughout this book, the reader is referred to several specific chapters in these *Manuals* for a thorough understanding of the topics being discussed.

2.7.2 Help Index

The *Index Tab* (Fig. 2.17) is the second tab on the left side of the *Help Window*. Every single help page contained in the ANSYS *Help System* is exhaustively listed under this tab. It is useful for finding which help pages are available for a given topic. Upon typing the topic of interest, a list of help pages appears, giving the user a chance to browse for the most-relevant help page.

2.7.3 Search in Help

The user can perform a word search of the ANSYS *Manuals* through the *Search Tab* (Fig. 2.18), which is the third tab on the left side of the *Help Window*. As a result of an inquiry, a list of help pages containing the search word appears and the user can select which pages to display.
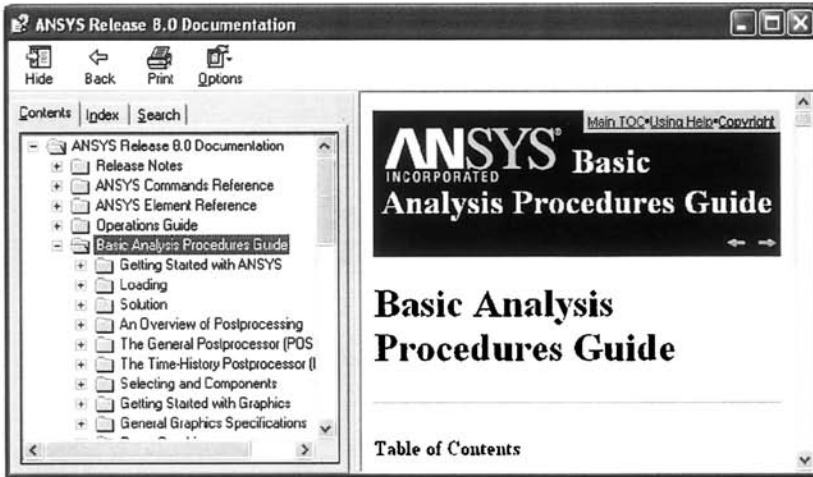
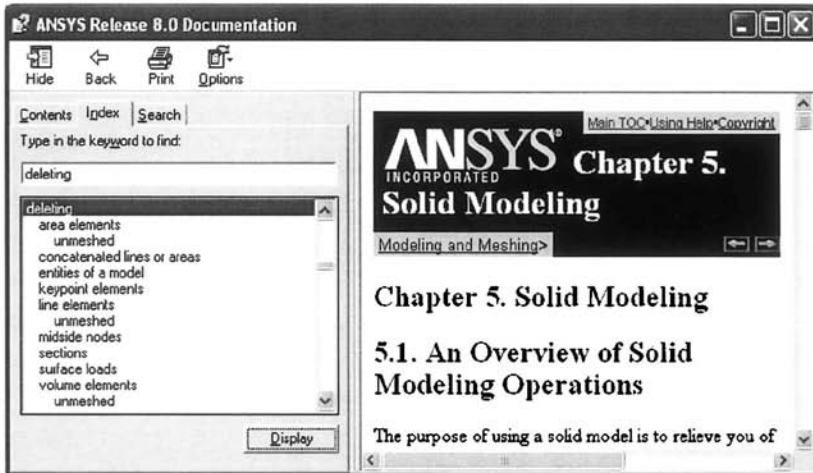Fig. 2.16 ANSYS *Help System* with *Contents* tab activated.

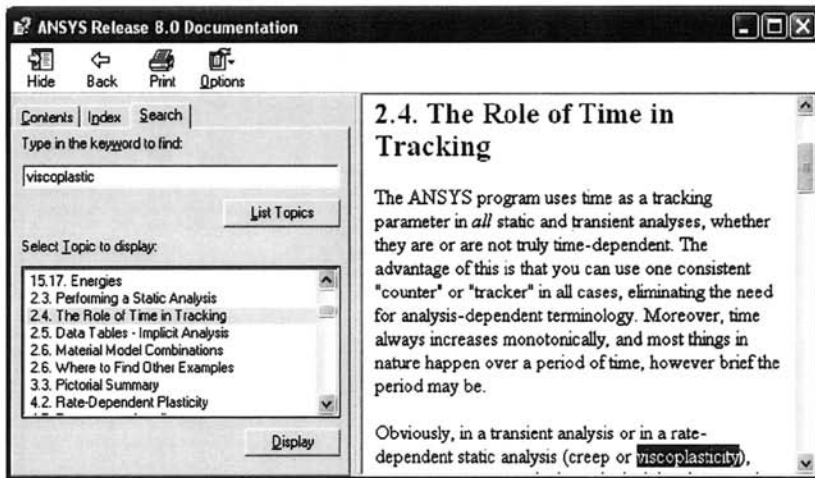Fig. 2.17 ANSYS *Help System* with *Index* tab activated.

Fig. 2.18 ANSYS *Help System* with *Search* tab activated.

2.7.4 Verification Manual

Although all of the ANSYS *Manuals* included under the *Contents Tab* are important sources of information, one particular *Manual* deserves special emphasis, the *Verification Manual*. The purpose of this *Manual* is to demonstrate the capabilities of ANSYS in solving fundamental engineering problems with analytical solutions. Another important feature of the *Verification Manual* is its suitability as an effective learning tool. There is a corresponding *Input File* for each of the verification problems included in this manual (in excess of 200). As mentioned earlier, the input files contain ANSYS commands to be executed sequentially when read from within ANSYS. Each of these commands corresponds to a specific action in the *Interactive Mode*. Once the verification problem that is the closest to the problem at hand is identified, the user can then study the corresponding *Input File* and learn the essential steps in solving the problem using ANSYS. The *Verification Manual* also serves as an excellent tool for learning to use ANSYS in *Batch Mode*.