Chapter 4
ANSYS Preprocessor

4.1 Fundamentals of Modeling

The fundamental concepts and the *Begin* and *Processor Levels* of the ANSYS finite element program are described in Chap. 2. Specifications of all the geometric and material properties, as well as the generation of solid and finite element models, are conducted at the preprocessor level.

There are two approaches for creating a finite element model: solid modeling and direct generation. The solid modeling approach utilizes *Primitives* (pre-defined geometric shapes) and operations similar to those of computer-aided design (CAD) tools, and internally generates the nodes and the elements based on user specifications. Solid modeling is the most commonly used approach because it is much more versatile and powerful. However, the user must have a strong understanding of the concept of meshing in order to utilize the solid modeling approach successfully and efficiently.

Direct generation is entirely dependent on user input for the size, shape, and connectivity of each element and coordinates of each node before it creates the nodes and elements one at a time. It requires the user to keep track of the node and element numbering, which may become tedious—sometimes practically impossible—for complex problems requiring thousands of nodes. It is, however, extremely useful for simple problems as one has full control over the model.

A combination of the two approaches is not only possible, but also advantageous in many cases. A comprehensive list of some important advantages and disadvantages is given in Table 4.1.

4.2 Modeling Operations

Within the *ANSYS Preprocessor*, a finite element model is generated by utilizing various operations, which are explained in this section.

The online version of this book (doi: 10.1007/978-1-4939-1007-6_4) contains supplementary material, which is available to authorized users

E. Madenci, I. Guven, *The Finite Element Method and Applications in Engineering Using ANSYS®*, DOI 10.1007/978-1-4899-7550-8_4

Table 4.1 Advantages and disadvantages of solid modeling and direct generation

Advantages	Disadvantages
Solid modeling	
Powerful (sometimes the only feasible way) in modeling three-dimensional solid volumes with complex geometry	If the user does not have a good understanding of meshing, ANSYS may not be able to generate the finite element mesh
User data input is rather low	For simple problems, using solid modeling may be ponderous
Common computer-aided design (CAD)-type operations such as extrusions, dragging, and rotations are utilized which are not possible when working directly with the nodes and elements	
With the basic (primitive) areas and volumes (rectangular, circular etc. areas; cubic, cylindrical, spherical etc. volumes), the Boolean operations (add, subtract, overlap etc.) can be used easily to modify (or tailor) these basic areas or volumes to obtain the desired shape	
Direct generation	
Provides the user with complete control of placement and numbering of nodes and elements	Use of direct generation is extremely tedious for solving real engineering design applications, especially when the problem can not be simplified to a two-dimensional idealization
For simple problems, the direct generation is the shortest way to generate a finite element mesh	

4.2.1 Title

This operation defines the title for the ANSYS analysis. This is an optional but recommended step in a typical ANSYS session. It helps the user to keep track of the problems by appearing in the graphics display and output. It becomes extremely useful when the user conducts a case study that involves the same model with different boundary conditions, different material properties, etc. The following menu path is used to change (or specify) the title:

Utility Menu > File > Change Title

which brings up the dialog box shown in Fig. 4.1. After entering the desired title in the *text box*, clicking on ***OK*** completes the specification of the title.

4.2.2 Elements

The nodes and elements are the essential parts of a finite element model. Before starting meshing, the *element type*(*s*) to be used must be defined (otherwise ANSYS refuses to create the mesh). The ANSYS software contains more than 100 different

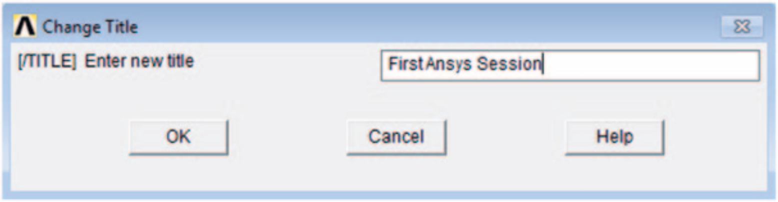


Fig. 4.1 Dialog box for specifying the title

element types in its element library. Each element type has a unique number and a prefix that identifies the element category, such as **`BEAM188`**, **`PLANE182`**, **`SOLID185`**, etc. The elements that are available in ANSYS can be classified according to many different criteria, such as dimensionality, analysis discipline, and material behavior. ANSYS classifies the elements in 23 different groups. In this section, the elements from four of these groups—specifically, structural, thermal, fluid, and FLOTRAN CFD—are considered for different analysis objectives.

1. ***Structural***: For this group of elements, the degrees of freedom at the nodes are displacements. As shown in Fig. 4.2, the structural analysis employs plane, link, beam, pipe, solid, and shell elements. All of the above "subgroups" of elements include several *element types* with different degree-of-freedom (DOF) sets.Consider the entries **`Quad 4node 182`**, **`Quad 8node 183`**, and **`Brick 8node 185`** from the ***Structural Solid*** subgroup. The first two elements types, **`Quad182`** and **`Quad183`**, are used for two-dimensional structural problems (plane stress, plane strain, or axisymmetric) whereas the third one is used for three-dimensional structural problems. The difference between **`Quad182`** and **Quad183** elements is that they have a different number of nodes per element, which implies that they are employing different interpolation functions for the variation of the degrees of freedom along the edges of the element. In this particular case, the variation of displacements along the element edges is assumed to be linear for **`Quad 4node 182`** and quadratic for **`Quad 4node 183`**, as shown in Fig. 4.3. The interpolation functions for the **`Brick 8node 185`** element are linear.
2. ***Thermal***: For this group of elements, the degrees of freedom at the nodes are temperatures. The thermal analysis employs mass, link, solid and shell subgroups. The element types in this group differ from each other with similar considerations as explained for structural discipline. Two commonly used thermal elements are shown in Fig. 4.4.
3. ***Fluid***: For this group of elements, depending on the type, the degrees of freedom appear as a pair, velocity-pressure or pressure-temperature, at the nodes. Included in this group are two- and three- dimensional acoustic, thermal-fluid coupled pipe, and contained-fluid types of elements.
4. ***FLOTRAN CFD***: This group of elements is similar to the previous one, except it is based on the method of computational finite difference.

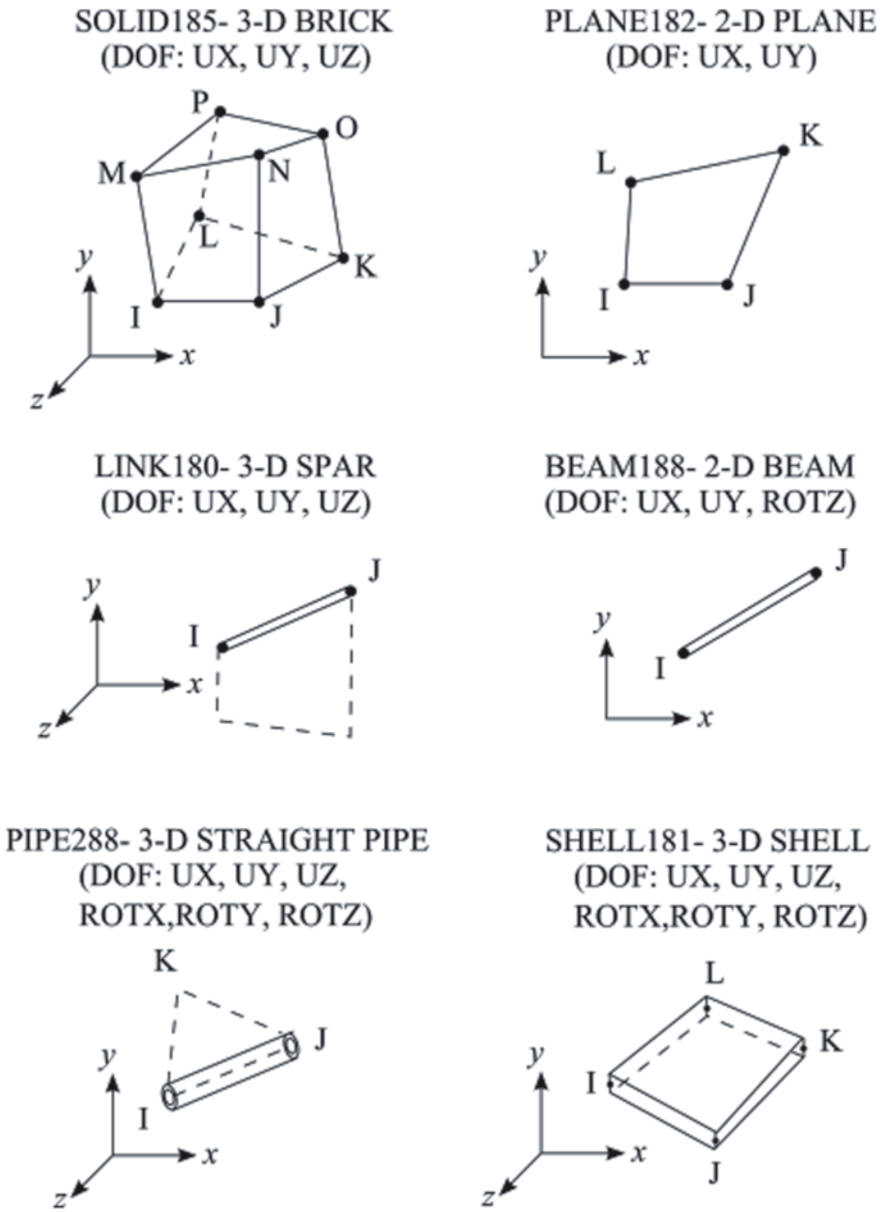


Fig. 4.2 Examples of structural elements in ANSYS

Each discipline requires the use of its own element types because the element type determines the degree-of-freedom set (displacements, temperatures, pressures, etc.) and the dimensionality of the problem (2-D or 3-D).

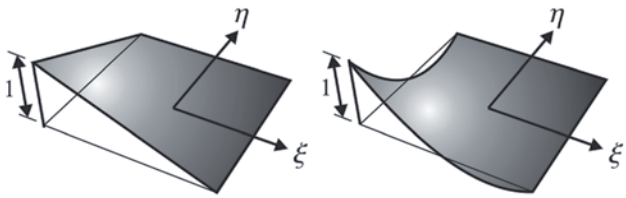
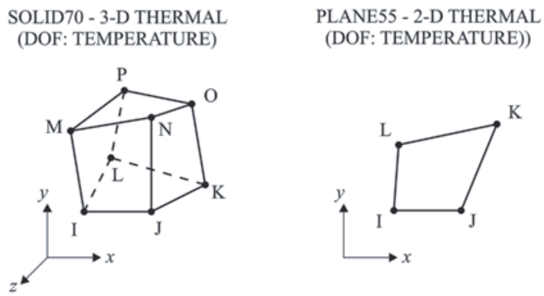


Fig. 4.3 Linear and quadratic variations of displacements within a 2-D element

Fig. 4.4 Examples of thermal elements in ANSYS



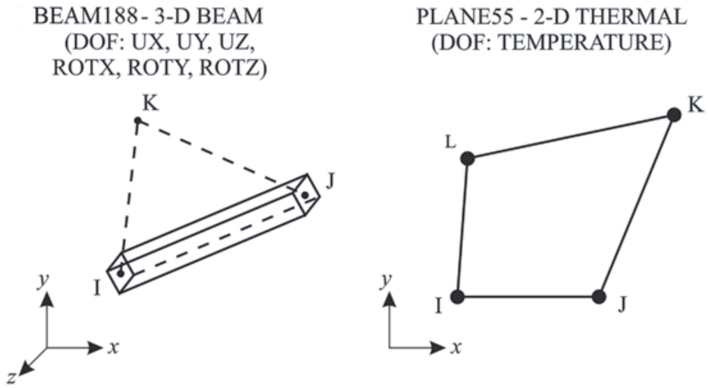


Fig. 4.5 **BEAM188** element for 3-D problems and **PLANE55** element for 2-D problems

For example, the **`BEAM188`** element, shown in Fig. 4.5, has six structural degrees of freedom (displacements and rotations in and about the *x*-, *y*-, and *z*-directions) at each of the two nodes, is a line element, and can be modeled in 3-D space. The **`PLANE55`** element, also shown in Fig. 4.5, which has a total of four thermal degrees of freedom (temperature at each node), is a 4-noded quadrilateral element, and can be used only for two-dimensional problems.

In order to specify an element type, the user must be in the *Preprocessor*. The menu path for element specification is

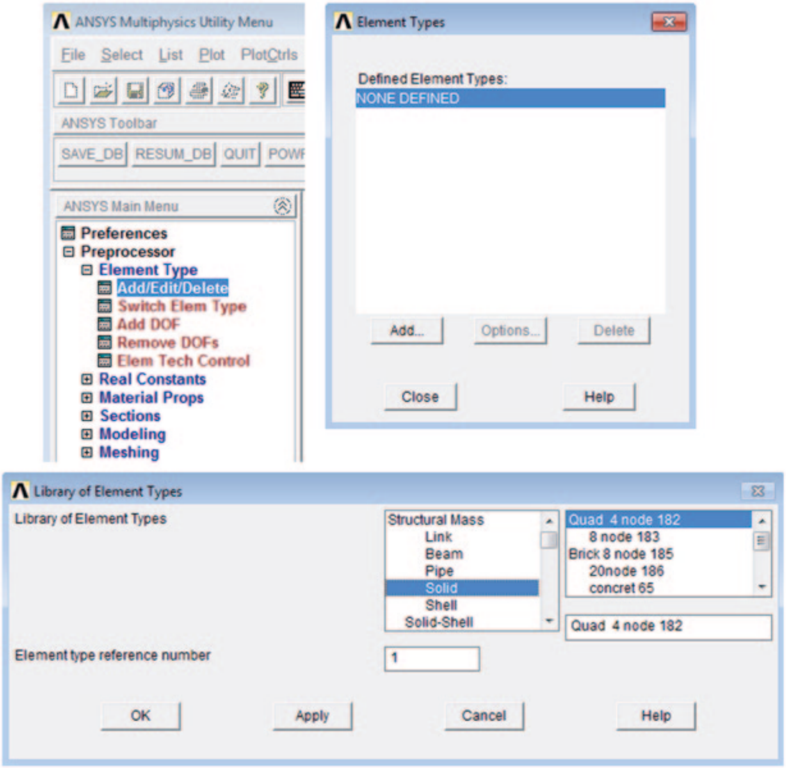


Fig. 4.6 Defining an element type in ANSYS

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

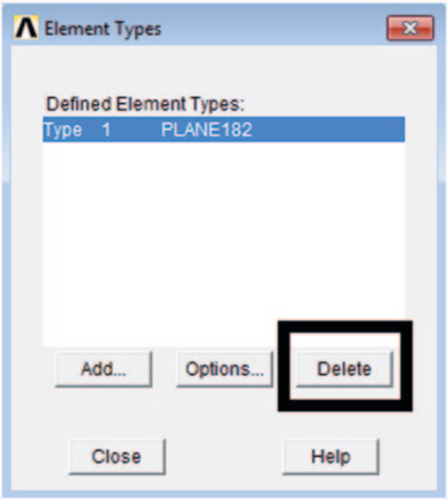
When this action is taken, a dialog box, shown in Fig. 4.6, appears with the options of ***Add***, ***Options***, ***Delete***, ***Close***, and ***Help***. Choosing ***Add*** brings up another dialog box with a list of all available elements, along with the ***Element type reference number***. The element types that are defined in a particular ANSYS analysis are assigned reference numbers. This reference number is used when creating the mesh. If the analysis requires the use of more than one element type, switching from one type to another one is achieved by referring to this number (this point is further explained when discussing *Element Attributes*).

If the user wants to delete an existing element type, it is achieved by using the same *GUI* path and choosing ***Delete***, as shown in Fig. 4.7:

Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Many element types have additional options, known as *keyoptions* (**`KEYOPT`**), and are referred to as **`KEYOPT(1)`**, **`KEYOPT(2)`**, etc. For example, as shown in Fig. 4.8, **KEYOPT(3)** for **SOLID182** (4-noded quadrilateral 2-D structural element) allows the user to specify the type of two-dimensional idealization, i.e., plane stress, plane strain, axisymmetric, or plane stress with thickness.

Fig. 4.7 Deleting an element type



Another example is shown in Fig. 4.9, in which **`KEYOPT(7)`** for **`SOLID70`** (8-noded thermal solid element for 3-D problems) permits the specification of a standard heat transfer or a nonlinear steady-state fluid flow through a porous medium.

Keyoptions are specified using the same *GUI* path and choosing ***Options*** from the *Element Types* dialog box.

4.2.3 Real Constants

As described in Chap. 1, the calculation of the element matrices requires material properties, nodal coordinates and geometrical parameters. Any data required for the calculation of the element matrix that cannot be determined from the nodal coordinates or material properties are called “real constants” in ANSYS. Typically, *real constants* are area, thickness, inner diameter, outer diameter, spring constant, damping coefficient etc. Not all element types require real constants.

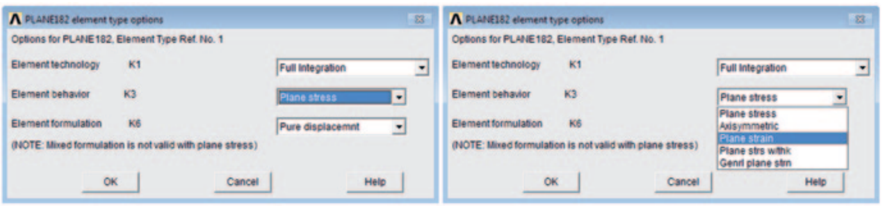


Fig. 4.8 *Keyoptions* for the **`PLANE182`** element

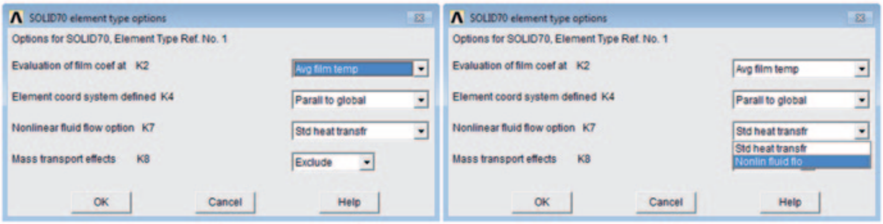


Fig. 4.9 *Keyoptions* for the **`SOLID70`** element

Real constants of a particular element type are briefly explained in the "Element Reference" of the ANSYS *Help System*. If the required real constants are not specified, ANSYS issues a warning. A good example for describing the real constants is the spring-damper element (element type **`COMBIN14`**). As shown in Fig. 4.10, the real constants for this type consist of the spring constant (***K***), damping coefficient (***CV1***), nonlinear damping coefficient (***CV2***), etc. In some cases, a complete set of real constants may not be required; in other cases, if the real constants are not specified, ANSYS may use a default value for that particular parameter. It is recommended that the "Element Reference" be consulted for the particular element type.

For each real constant set, ANSYS requires a reference number. If it is not assigned by the user, ANSYS automatically assigns a number, as shown in Fig. 4.10.

Real constants are specified using the following *GUI* path:

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

This brings up the *Real Constants* dialog box, where clicking on ***Add*** leads to another dialog box having a list of currently defined element types. Choosing the element type for which the real constants are specified (if there are no required real constants for the selected element type, a warning window pops up) and hitting ***OK*** brings up a new dialog box. The real constants for that specific element type appear; after filling in the boxes, hitting ***OK*** completes this operation.

For models having multiple element types, a distinct real constant set (that is, a different reference number) is assigned for each element type. ANSYS issues a warning message if multiple element types are referenced to the same real constant set. However, there are cases where it is necessary to specify several real constant sets for the same element type. This feature is explained further by considering a plate composed of three different sections, as shown in Fig. 4.11. Although the material properties are the same, each section has a different thickness. Modeling this plane with a plane type of element, **`PLANE182`**, requires the thickness values as the real constants. Since there are three different thicknesses, a different real constant set is defined for each of these sections; the same element type (**`PLANE182`**) in the *Real Constants* dialog box is selected. Different parts of the plane are meshed one at a time, directing ANSYS to use the real constant set corresponding to the specific part of the plate. This concept is further clarified when discussing *Element Attributes*.

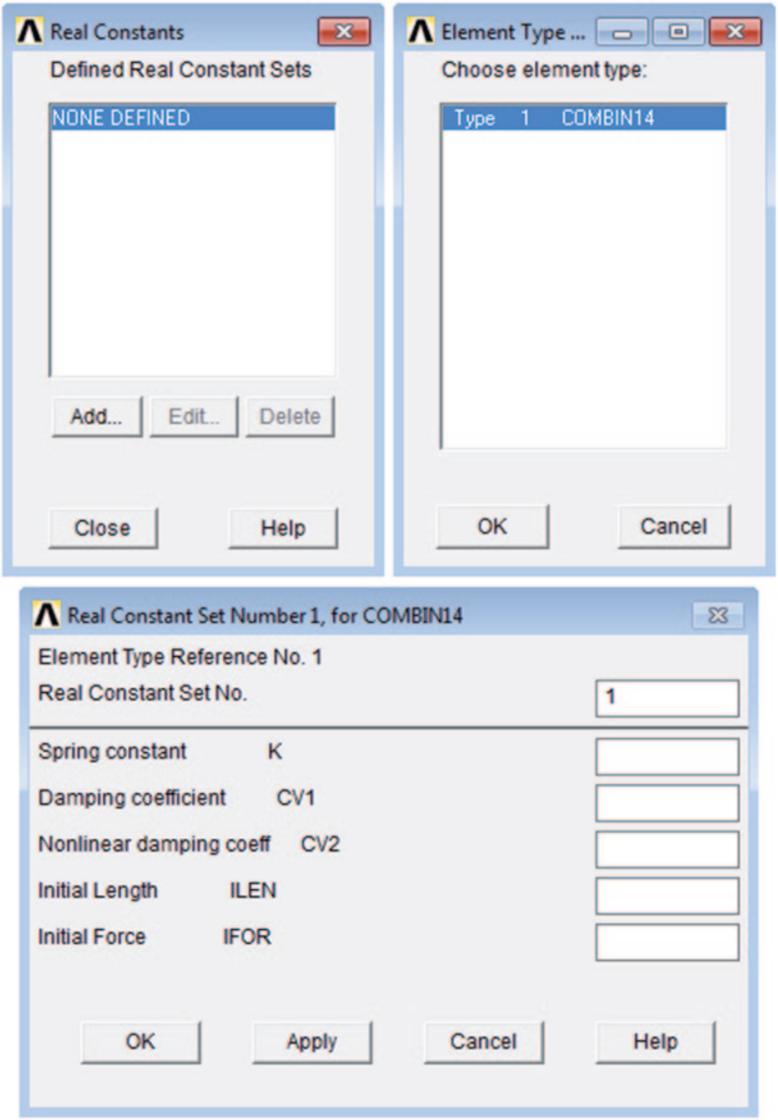
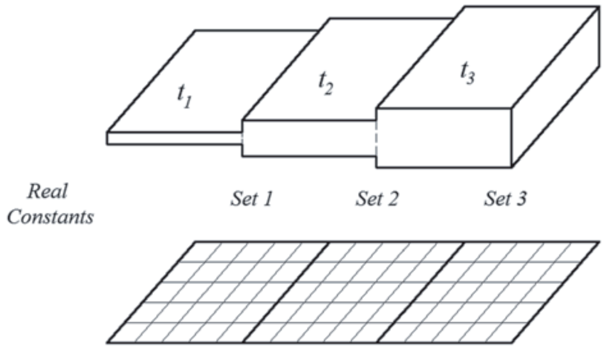


Fig. 4.10 Real constants for the `COMBIN14` element

CAUTION It is the user's responsibility to keep track of units that are used in the analysis. The user does not need to give ANSYS the system of units being used. The user should decide which system of units to use and be consistent throughout the analysis (i.e., dimensions of the input, real constants, material properties and loads). ANSYS WILL NOT CONVERT UNITS. Also, the solution quantities are given in terms of the units of the input.

Fig. 4.11 A plane with three different thicknesses; three real constant sets are required



4.2.4 Material Properties

For each element type, there are a minimum number of required material properties. This number depends on the type of analysis. The material properties may be:

- Linear or nonlinear.
- Isotropic, orthotropic, or anisotropic.
- Temperature dependent or independent.

All material properties can be input as functions of temperature. Some properties are called linear properties because typical solutions with these properties require only a single iteration. This means that the properties being used are neither time nor temperature dependent, and thus remain constant throughout the analysis.

In the presence of variable material properties, the nonlinear characteristics of the properties must be specified. For example, a material exhibiting plasticity, viscoplasticity, etc., requires the specification of a nonlinear stress-strain relation.

A complete list of linear material properties is given in Table 4.2 (properties related to electrical and magnetic analyses are not included).

Each material property set has a reference number, the same as the element types and real constants. In problems involving different materials, the user is required to specify multiple material property sets. ANSYS identifies each material by its unique reference number. The *Help System* should be consulted for the specification of nonlinear material properties.

The following menu path is used to specify constant isotropic or orthotropic material properties:

Main Menu > Preprocessor > Material Props > Material Models

This brings up the *Define Material Model Behavior* dialog box, as shown in Fig. 4.12. On the left side of this window, material models are listed based on their material reference numbers. On the right side, available material models are organized based on the analysis type (e.g., structural, thermal, etc.). Figure 4.13 shows an expanded view of the material models available under *Structural* analysis. As observed in the figure, if a linear material response is to be used, then the user double-clicks

Table 4.2 List of material properties for structural, thermal, and fluids disciplines

Label	Units	Description
EX	Force/Area	Elastic modulus, element *x*-direction
EY		Elastic modulus, element *y*-direction
EZ		Elastic modulus, element *z*-direction
ALPX	Strain/Temp	Coefficient of thermal expansion, element *x*-direction
ALPY		Coefficient of thermal expansion, element *y*-direction
ALPZ		Coefficient of thermal expansion, element *z*-direction
REFT	Temp	Reference temperature (as a property)
PRXY	None	Major Poisson's ratio, *x-y* plane
PRYZ		Major Poisson's ratio, *y-z* plane
PRXZ		Major Poisson's ratio, *x-z* plane
NUXY		Minor Poisson's ratio, *x-y* plane
NUYZ		Minor Poisson's ratio, *y-z* plane
NUXZ		Minor Poisson's ratio, *x-z* plane
GXY	Force/Area	Shear modulus, *x-y* plane
GYZ		Shear modulus, *y-z* plane
GXZ		Shear modulus, *x-z* plane
DAMP	Time	K matrix multiplier for damping
MU	None	Coefficient of friction (or, for **FLUID29** element, boundary admittance)
DENS	Mass/Vol	Mass density
C	Heat/Mass × Temp	Specific heat
ENTH	Heat/Vol	Enthalpy
KXX	Heat × Length/ (Time × Area × Temp)	Thermal conductivity, element *x*-direction
KYY		Thermal conductivity, element *y*-direction
KZZ		Thermal conductivity, element *z*-direction
HF	Heat/ (Time × Area × Temp)	Convection (or film) coefficient
EMIS	None	Emissivity
QRATE	Heat/Time	Heat generation rate (**MASS71** element only)
VISC	Force × Time/Length^2	Viscosity
SONC	Length/Time	Sonic velocity (**FLUID29** and **FLUID30**)

on the ***Linear*** option to expand. After double-clicking on the ***Elastic*** option under ***Linear***, three options are available for the user: ***isotropic***, ***orthotropic***, and ***anisotropic***. Upon double-clicking on any of these options, a new dialog box appears. Figure 4.14 (left) shows the dialog box corresponding to the ***isotropic*** option. If the material properties are temperature dependent, the ***Add Temperature*** button is used for adding columns for different temperatures, as shown in Fig. 4.14 (right).

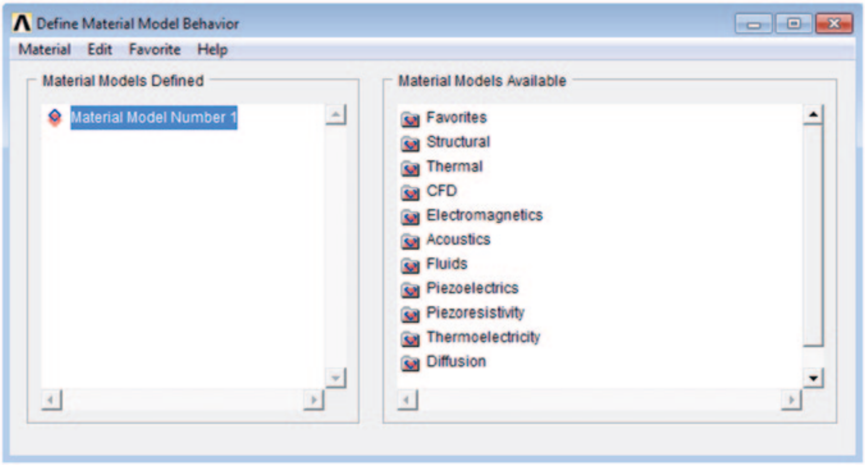


Fig. 4.12 Dialog box for defining material models

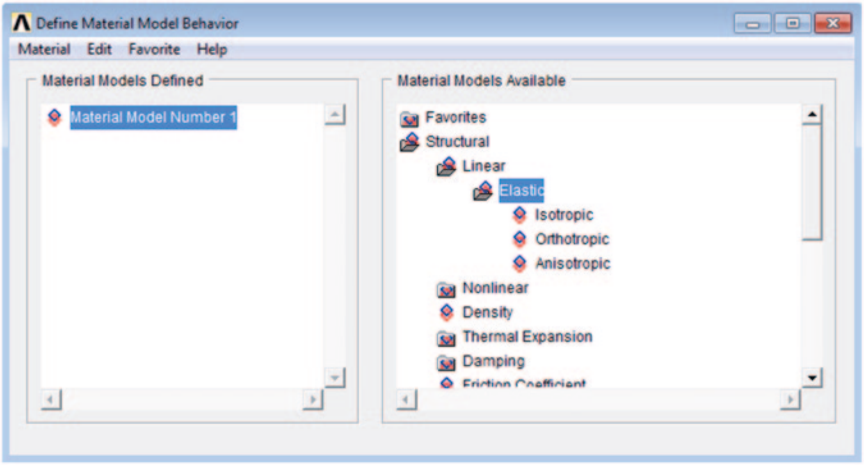


Fig. 4.13 Expanded view of the material models under the *Structural* discipline

4.2.5 Element Attributes

Every element in ANSYS is identified by the element type, real constant set, material property set, and element coordinate system. These are called *element attributes*. In order to create a mesh, the element type(s) must be specified a priori and the material properties (and real constants, depending on the element type) must be specified in order to obtain a solution. The element coordinate system is defined internally.

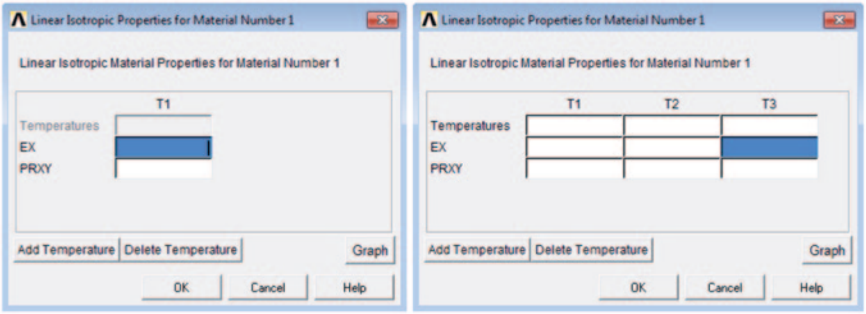


Fig. 4.14 Dialog box for isotropic properties: not temperature dependent (*left*) and temperature dependent (*right*)

4.2.6 Interaction with the Graphics Window: Picking Entities

When using ANSYS through the *GUI*, part of the interaction between the user and the software involves picking entities or locations in the *Graphics Window*. These interactions are performed using the *Pick Menus*. Figures 4.15 and 4.16 show two examples of such menus. Picking operations are performed using the left mouse button.

When picking entities through the *Pick Menu*, there are five distinct fields, as shown in Fig. 4.15:

1. *Pick/Unpick Field*: Using the radio-buttons, the user selects whether the entities are to be picked or unpicked. This feature is useful when the user picks entities other than the intended ones. Instead of using the radio-buttons, the user may use the right mouse button to toggle between the ***Pick*** and ***Unpick*** modes.
2. *Picking Style Field*: By default, the user picks entities one at a time (i.e., radio-button ***Single*** in the *Pick Menu*). However, if the number of entities to be picked is a large number, the ***Single*** picking mode may become tedious, and one of the other modes may be preferable in such situations. Available options include:

   Box**:** The user draws a rectangle in the *Graphics Window* by holding down the left mouse button; entities located inside the rectangular box are picked.

   Polygon**:** The user draws a polygon in the *Graphics Window*. Vertices of the polygon are created by single clicks on the left mouse button. The polygon is finalized when the user clicks on the first vertex created. The entities located inside the polygon are picked.

   Circle**:** When the entities follow a radial pattern, it may be more convenient to pick them through a circular region. This option permits the user to draw a circle in the *Graphics Window* by holding down the left mouse button.

3. *Information Field*: This field provides the user with useful information such as the number of currently picked entities, maximum number of entities that can be picked, and the last entity number picked.

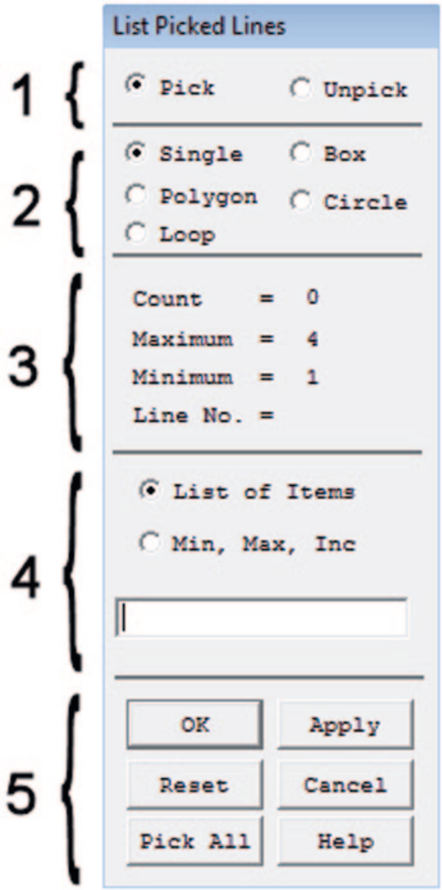


Fig. 4.15 *Pick Menu* for picking entities

4. *Text Field*: Using this option, the user may provide text input for the entities to be picked instead of picking them in the *Graphics Window*. This can be done in two different formats:

List of Items: When the radio-button next to ***List of Items*** is selected (default), the user may enter a list of the entity numbers to be picked, separated by commas, in the text field.

Min, Max, Inc: When the radio-button next to ***Min, Max, Inc*** is selected, the user may enter the entity numbers to be picked in the text field in the format Minimum, Maximum, Increment. For example, if the user enters 1, 5, 2, then ANSYS picks entities 1, 3 and 5.

5. *Action Field*:This field involves familiar actions, such as:

OK: Finishes the picking operation and closes the *Pick Menu*.
Apply: Applies the picking performed so far while keeping the *Pick Menu* active.
Reset: The picking operations performed so far are ignored and the configuration is set to the one that existed when the *Pick Menu* appeared.

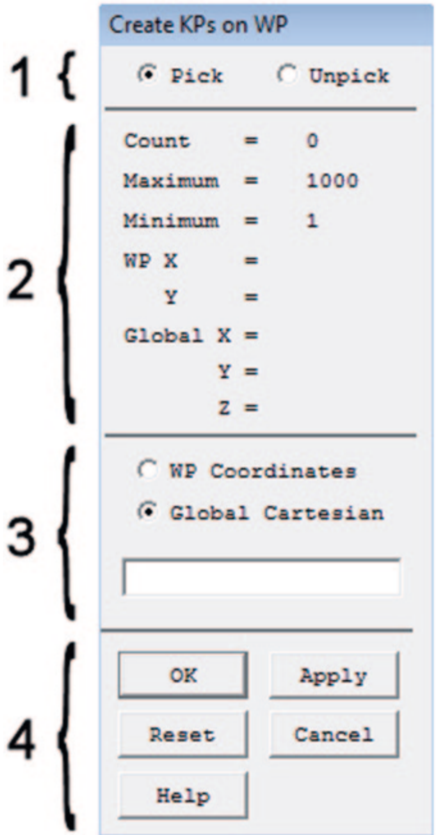


Fig. 4.16 *Pick Menu* for picking locations

Cancel: Closes the *Pick Menu* without performing picking.
Pick All: All of the items under consideration are picked and the *Pick Menu* is closed.
Help: Displays the Help Page related to the current operation.

Picking locations is similar to picking entities, except for slight differences in the *Pick Menu*. This time the menu has four fields, as shown in Fig. 4.16:

1. *Pick/Unpick Field*: This field is the same as explained above.
2. *Information Field*: Similar to the previous case, this field provides the user with useful information such as the number of currently picked locations, maximum and minimum possible picking operations, and the *Working Plane* and *Global Cartesian coordinates* of the last location picked.
3. *Text Field*: Using this option, the user can provide the coordinates of the location to be picked instead of picking them in the *Graphics Window*. This can be done in two different formats: *Working Plane* or *Global Cartesian Coordinates*. In either case, the coordinates are separated by commas.
4. *Action Field*: This field is the same as explained above, with exception of the absence of the ***Pick All*** button.

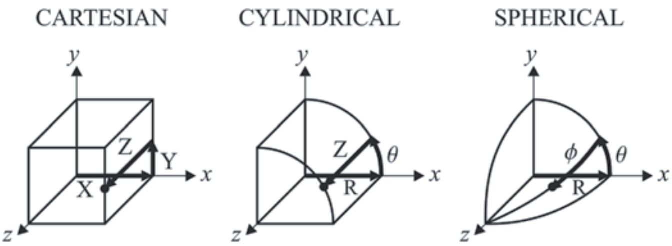
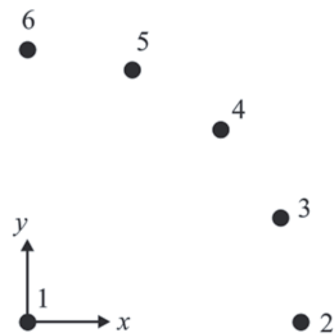


Fig. 4.17 Cartesian (*left*), cylindrical (*middle*), and spherical (*right*) coordinate systems

Fig. 4.18 Six nodes, one at the origin; the remaining five lie in a quarter-circle pattern



4.2.7 Coordinate Systems

4.2.7.1 Global Coordinate Systems

When the user starts an ANSYS session, the coordinate system (CS) is Cartesian by default. However, there are many situations where using other coordinate systems (cylindrical or spherical) is more convenient. There are four predefined coordinate systems in ANSYS: Cartesian, cylindrical, spherical, and toroidal; the first three of them are shown in Fig. 4.17.

All of these coordinate systems have the same origin (global origin) and are called global coordinate systems. Although the session starts with the Cartesian CS, the user can switch to one of the other three coordinate systems at any time. The CS currently used is referred to as the *active coordinate system* (active CS); any action referring to the coordinates is performed in the active CS. For example, either a Cartesian or cylindrical CS can be used to create the nodes at the locations shown in Fig. 4.18. The nodes around the unit circle are equally spaced.

In reference to a Cartesian CS, Nodes 1, 2, and 6 can easily be created because the coordinates are explicitly given as **(0, 0, 0)**, **(1, 0, 0)** and **(0, 1, 0)**, respectively. For nodes 3, 4, and 5, trigonometric relations can be used to calculate the *x*-, *y*-, and *z*-coordinates with a desired precision or round-off.

An alternative to the calculation of these coordinates is to change the active CS from Cartesian to cylindrical. In the cylindrical coordinate system, any reference to

Table 4.3 Nodal coordinates in Cartesian and cylindrical coordinate systems

Node	Cartesian			Cylindrical		
	x	y	z	r	θ	z
1	0	0	0	0	0	0
2	1	0	0	1	0	0
3	0.924	0.383	0	1	22.5	0
	0.9239	0.3827				
	0.92388	0.38268				
4	0.707	0.707	0	1	45	0
	0.7071	0.7071				
	0.707106	0.707106				
5	0.383	0.924	0	1	67.5	0
	0.3827	0.9239				
	0.38268	0.92388				
6	0	1	0	1	90	0

x-, y-, and z-coordinates are treated as r, θ, and z. The coordinates of the nodes 3, 4, and 5 in the cylindrical CS are specified as **(1, 22.5, 0)**, **(1, 45, 0)** and **(1, 67.5, 0)**, respectively.

By changing the active CS, unnecessary algebraic calculations and the potential loss of accuracy are avoided.

The coordinates of the nodes in the Cartesian and cylindrical coordinate systems are given in Table 4.3. The "loss of accuracy" can be observed by examining the possible x- and y-coordinates of nodes 3, 4, and 5.

The menu path to change the active CS is given as

Utility Menu > WorkPlane > Change Active CS to

Selection of one of the top three choices in the dialog box,, i.e., global Cartesian, global cylindrical, or global spherical, completes this operation. The CS that is chosen remains active and, in turn, all the coordinates are referenced to that CS, until the user changes it.

4.2.7.2 Local Coordinate Systems

The global coordinate systems all share the same origin (global origin) with a predefined orientation. There are situations where changing one type of global coordinate system to a different global coordinate system does not provide enough convenience or sometimes makes it even more complicated.

It may turn out that what the user really needs is to change the orientation of the CS and/or location of the origin.

In such cases, the user needs to define a CS by offsetting the origin or changing the orientation, or both. Such a coordinate system is called a *local coordinate system* (local CS). A *local CS* can be created by specifying either a location for the origin or

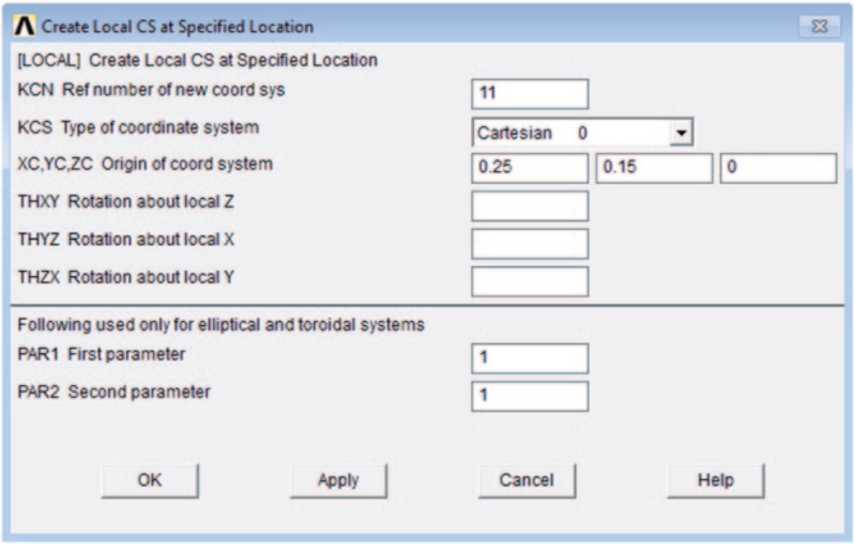


Fig. 4.19 Dialog box for creating a *Local CS* at a specified location

three keypoints or nodes. Only one CS can be active at a given time. ANSYS requires that *local coordinate systems* have reference numbers that are *greater than or equal to* 11. The menu path for creating a *local CS* at specified location is given as

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc +

This brings up a *Pick Menu*, requesting the user to enter the coordinates of the points in the *text field* inside the *Pick Menu*, or to pick the points by clicking the mouse pointer on the *Graphics Window*.

After picking the origin, clicking on ***OK*** brings up the dialog box shown in Fig. 4.19. There are several text boxes to fill out in this dialog box. First is the reference number (by default, it is 11). If a *local CS* was defined previously, as a default, ANSYS assigns the smallest available reference number that is greater than or equal to 11. If this reference number is not desired, the user enters the new reference number for this CS.Below the reference number box, there is a pull-down menu for the CS type: Cartesian, cylindrical, or spherical (toroidal is not discussed herein). The coordinates of the origin of the *local CS* with respect to the global origin should already appear in the CS type menu. Finally, rotation angles with respect to the active CS (not necessarily global Cartesian) are entered.

4.2.8 Working Plane

Within the ANSYS environment, regardless of the dimensionality of the problem (2-D or 3-D), calculations are performed in a 3-D space. If the problem is 2-D, then ANSYS uses the *x*-*y* plane, which is the $z=0$ plane.

The *Working Plane* (WP) is a 2-D plane with the origin of a 2-D coordinate system (Cartesian or polar) and a display grid. It is designed to facilitate solid model generation, where many solid model entities are created by referring to the origin of the WP.

In order to view the WP, the menu path is given as

Utility Menu > WorkPlane > Display Working Plane

A checkmark appears on the left of this menu item. Similarly, one can turn the display WP off by using the exact same menu path, resulting in the disappearance of the checkmark. By default, only the triad that is attached to the WP is shown in the *Graphics Window*. Viewing the grid is achieved by the menu path:

Utility Menu > WorkPlane > WP Settings

This brings up the *WP Settings Window*. Clicking on the ***Grid and Triad*** radio-button turns on both the grid and the triad; clicking on the ***Grid Only*** radio-button turns on only the grid. Hitting the ***Apply*** or ***OK*** button activates the new setting.

Using the two radio-buttons at the top permits a switch between the Cartesian and cylindrical (polar) CS. The WP can be placed at any point in the 3-D space with an arbitrary orientation. There can only be one working plane at a time. By default, the WP is the *x-y* plane of the global CS.A working plane can be defined by specifying either three points or nodes or keypoints.

At this point, defining a WP by three points is explained. The menu path is given as

Utility Menu > WorkPlane > Align WP with > XYZ Locations

A *Pick Menu* appears, prompting the user to enter the coordinates of the points in the *text field* or pick the points by clicking the mouse pointer on the *Graphics Window*.

The user needs three noncolinear points to define a plane. The first point is the origin of the WP. The second point defines the WP *x*-axis along the line defined between the first and itself. The third point defines the direction of the positive WP y-axis. Two examples of these operations are illustrated in Fig. 4.20. When all three points are entered, clicking on ***OK*** in the *pick menu* completes the definition of the WP by three points.

As shown in Fig. 4.21, an existing WP can be moved to a new location by providing offset distances in the *x*-, *y*-, and *z*-directions, which yields a WP parallel to its previous orientation. Also, an existing WP can be rotated in all three directions, as shown in Fig. 4.22. If the user rotates the WP about the *z*-axis (which is the direction normal to the WP—not to the global CS), then the WP remains in the same plane but the WP *x*- and *y*-axes rotate within the plane. These movements can be made by the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

This brings up the *Offset* window. This window requires the offset values in **X**, **Y**, and **Z**. In the *Offset WP* window, which is used for both translation and rotation, there are six push-buttons for translation and six push-buttons for rotation. These are used for incremental translation and rotation in and around, respectively,

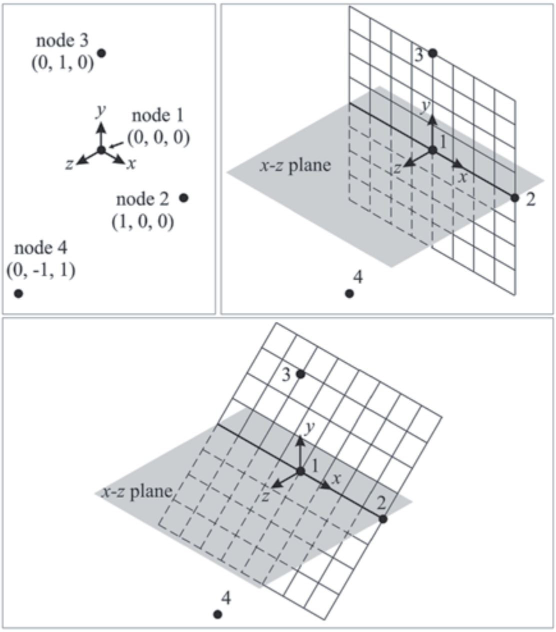


Fig. 4.20 Four nodes in 3-D space (*top left*); WP defined on the plane defined by nodes 1, 2, and 3 (*top right*) and by nodes 1, 2, and 4 (*bottom*)

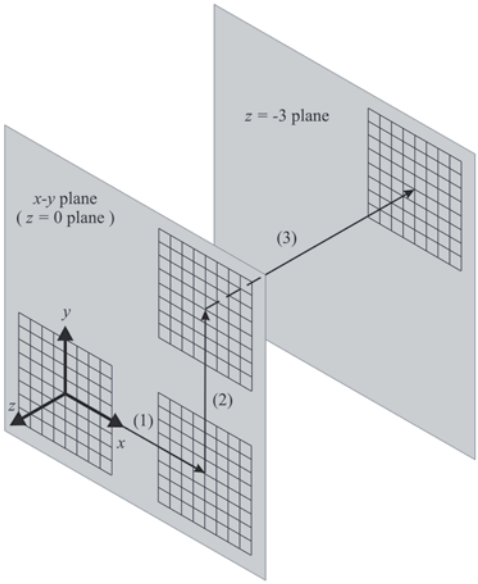


Fig. 4.21 WP first moved 3 units in the x-direction, then 3 units in the y-direction, and, finally, -3 units in the z-direction

positive and negative x-, y-, and z-axes. The increment is given by a sliding button right below the buttons (one for translation and one for rotation). If the display WP is turned on, the resulting incremental translation or rotation can be observed immediately (without having to hit ***Apply***).

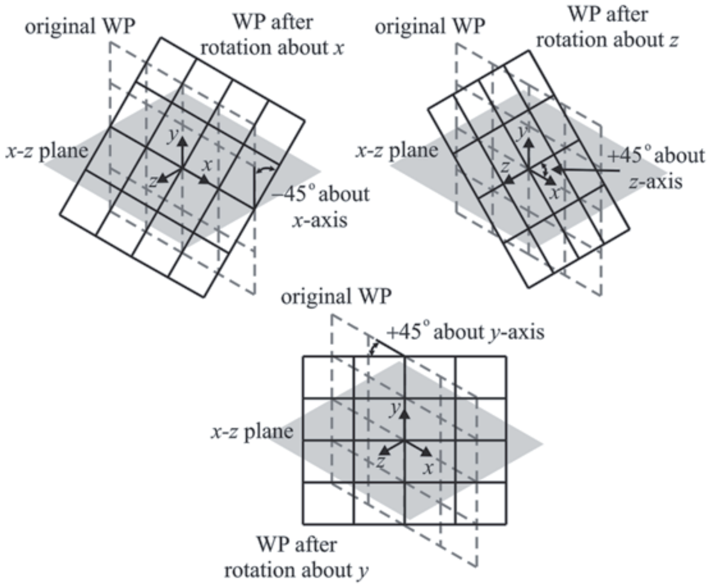


Fig. 4.22 WP rotated −45° about the x-axis (*top left*), +45° about the z-axis (*top right*), and +45° about y-axis (*bottom*)

CAUTION The **X** and **Y** refer to the WP's x- and y-axes (not global axes) and z is the direction normal to the WP (not global CS); the positive direction is established by the right-hand rule.

4.3 Solid Modeling

The geometrical representation of the physical system is referred to as the solid model. In model generation with ANSYS, the ultimate goal is to create a finite element mesh of the physical system. There are two main paths in ANSYS to generate the nodes and elements of the mesh: (1) direct generation and (2) solid modeling and meshing.

In direct generation, every single node is generated by entering their coordinates followed by generation of the elements through the connectivity information. Since most real engineering problems require a high number of nodes and elements (i.e., hundreds or thousands), direct generation is not feasible. Solid modeling is a very powerful alternative to direct generation.

Solid modeling involves the creation of geometrical entities, such as lines, areas, or volumes, that represent the actual geometry of the problem.Once completed, they can be meshed by ANSYS automatically (user still has control over the meshing through user-specified preferences for mesh density, etc.). A solid model can be created by using either *entities* or *primitives*. The *entities* refer to the *keypoints*, *lines*, *areas*, and *volumes*. The *primitives* are predefined geometrical shapes.

There is an ascending hierarchy among the entities from the keypoints to the volumes.Each entity (except keypoints) can be created by using the lower ones. When defined, each entity is automatically associated with its lower entities.If these entities are created by starting with keypoints and moving up, the approach is referred to as “bottom-up” solid modeling.

When primitives are used, lower-order entities (keypoints, lines, and areas) are automatically generated by ANSYS. Since the use of primitives involves the generation of entities without having to create lower entities, it is referred to as the “top-down” approach.Boolean or similar operations can be applied to the primitives to generate the complex geometries.

The bottom-up and top-down approaches can easily be combined since one may be more convenient at a certain stage and the other at another stage. It is not necessary to declare a preference between the two approaches throughout the analysis.

4.3.1 Bottom-up Approach: Entities

4.3.1.1 Keypoints

When the bottom-up solid model generation approach is used, the user starts by generating the keypoints. The higher entities (lines, areas, and volumes) can then be defined by using the keypoints. The keypoints necessary to create a higher-order entity for modeling different parts of the geometry should be generated a priori. When areas or volumes are generated using keypoints, the intermediate entities are generated automatically by ANSYS. The creation of keypoints on the WP and in the *active CS* is explained herein.

The following menu path is suggested to create a keypoint on WP:

Main Menu > Preprocessor > Modeling > Create > Keypoints > On Working Plane

This brings up a *Pick Menu*, where ANSYS expects the user to pick points on the WP.Once the points are picked by clicking on the left mouse key, hitting on the ***Apply*** or ***OK*** button completes this task (***OK*** closes the *Pick Menu*).When using this option, turning on the display WP with the grid visible is highly recommended.

IMPORTANT HINT When using the *Pick Menu*, picking the exact location might become a real challenge and, in turn, result in the generation of unnecessary entities. These “extra” entities might cause confusion and possible errors in the course of the solid model generation. Whenever there is a *Pick Menu*, the user can hold down (no release) the left mouse button and move the pointer on the *Graphics Window*. This action shows the mouse pointer coordinates on the *Pick Menu*. When the target coordinates are found, the user can release the button to finish the picking.

The following menu path is suggested to create keypoint(s) (KP) in the active CS:

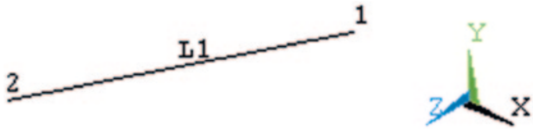


Fig. 4.23 A straight line

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

This brings up a dialog box with four input fields for the KP number and the *x*-, *y*-, and *z*-coordinates. Once this information is supplied, hitting ***OK*** creates the KP and exits from this dialog box.Alternatively, the ***Apply*** button can be clicked on and more keypoints can be created.

The coordinates defining a KP can be modified as follows:

Main Menu > Preprocessor > Modeling > Move/Modify > Keypoints > Single KP

This brings up a *Pick Menu*. First, KP is picked from the *Graphics Window*, or its number is typed in the *text field*. Then, the new location is picked or the new coordinates are typed. If a KP is modified, any mesh that is attached to that KP is automatically cleared, and any higher-order entities that are associated with that KP also are modified accordingly.

4.3.1.2 Lines

Lines are used for either creating a mesh with line elements or creating areas and volumes. A straight line, an arc, and a cubic spline can be created, as shown in Fig. 4.23 and 4.24.

Creating a True Straight Line
By using the following menu path, a straight line can be created regardless of the active CS. The only input needed is two keypoints. The menu path is given as

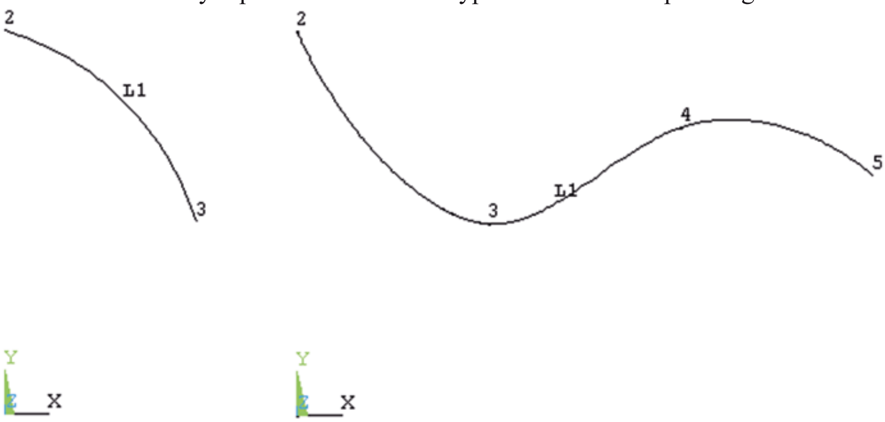


Fig. 4.24 An arc (left) and a cubic spline (right)

Main Menu > Preprocessor > Modeling > Create > Lines > Straight Line

This brings up a *Pick Menu*, requesting keypoint numbers, which can be entered through the *text field* or picked from the *Graphics Window*.Multiple lines can be generated, one at a time, without closing the *Pick Menu* (by using ***Apply*** button). The straight line (L1) is generated by keypoints (KP1) and (KP2).

Creating a Straight Line in the Active CS
This method creates a straight line in the active CS. If the active CS is a Cartesian CS, then the line is a true straight line. If the active CS is a cylindrical CS, then the line is a helical spiral. The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Lines > In Active Coord

This brings up a *Pick Menu,* requesting the keypoints. The keypoint numbers are supplied through either the *text field* or by picking them using the *Graphics Window*. Multiple lines can be generated, one at a time, without closing the *Pick Menu*. It works the same way as creating a true straight line.

Creating an Arc
Creating an arc requires three keypoints. The arc is circular, regardless of the active CS. It is generated between the first and the second keypoints. The third KP defines the plane of the arc, as well as the positive curvature side. It does not have to be at the center of the curvature. The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Lines > Arcs > By End KPs & Rad

This brings up a *Pick Menu*, requesting the two end keypoints. These keypoint numbers are supplied through either the *text field* or by picking them using the *Graphics Window*. Upon hitting ***OK*** in the *Pick Menu*, ANSYS requests the third KP, which defines the positive curvature side. After entering it the same way and hitting ***OK*** in the *Pick Menu*, a dialog box appears. The first field is the radius and the remaining 3 are the keypoints that have already been input. Entering the radius and hitting ***OK*** completes this operation.

Creating a Spline
Several keypoints (minimum 2) are needed for creating a spline.The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Lines > Splines > Spline Thru KPs

This brings up a *Pick Menu*, requesting the keypoints to be picked. When finished, hitting ***OK*** finishes the spline creation. Multiple splines can be generated, one at a time, without closing the *Pick Menu* by hitting the ***Apply*** button instead of ***OK***.

Once the lines are defined, areas can be created by using them.

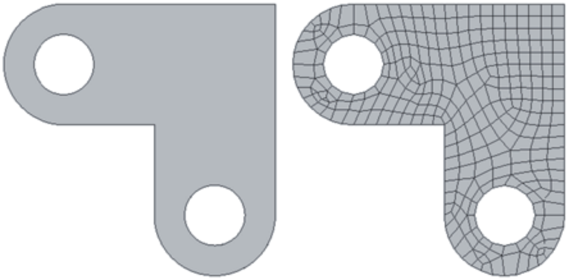


Fig. 4.25 An area in the *x-y* plane (*left*); meshed (*right*)

4.3.1.3 Areas

Areas are used to create a mesh with area elements and to create volumes.If the geometry involves a 2-D domain, the area(s) is (are) required to be flat, lying on the *x-y* plane. If the geometry involves 3-D bodies, then the areas that define the faces of the volume(s) can be flat or curved. A mesh created from a flat area and volumes created from flat and curved areas are shown in Figs. 4.25 and 4.26.

In bottom-up approach, areas can be created by using either keypoints or lines.

Creating an Area Using Keypoints

A minimum of 3 keypoints is required, and the maximum number allowed is 18.If more than 3 keypoints are used, they must lie in the same plane (co-planar), as shown in Fig. 4.27. The menu path is given as

Main Menu > Preprocessor > Modeling > Create > -Areas- Arbitrary > Through KPs

which brings up a *Pick Menu*, requesting the keypoints to be picked. When finished, clicking on ***OK*** creates the area.

CAUTION In the PC version, it is recommended that the input window be used.

Creating an Area Using Lines

In creating an area by lines, a minimum of 3 previously defined lines are required, and the maximum number of lines allowed is 10. If more than 3 lines are used, they

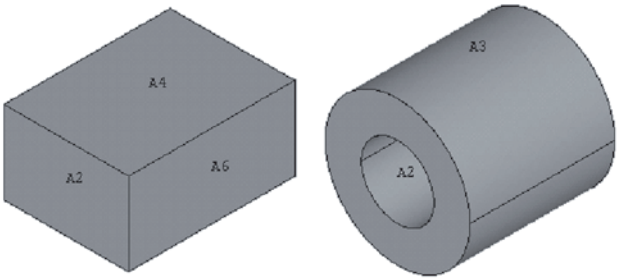


Fig. 4.26 Volume composed of flat areas (*left*) and flat and curved areas (*right*)

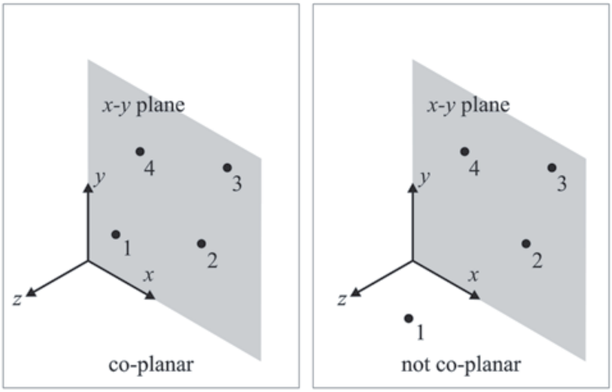
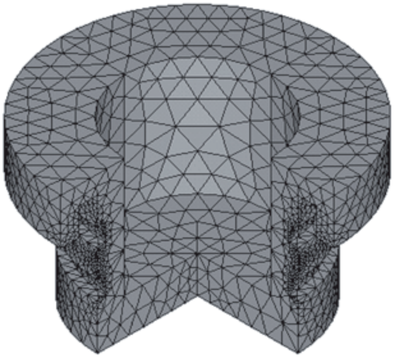


Fig. 4.27 Coplanarity of keypoints: 4 coplanar keypoints (*left*) and 4 noncoplanar keypoints (*right*)

Fig. 4.28 A meshed volume



must be co-planar.Lines must be given in a clockwise or counterclockwise order, and they must form a simply connected closed curve. The menu path is given as

Main Menu > Preprocessor > Modeling > Create > -Areas- Arbitrary > By Lines

This brings up a *Pick Menu*, requesting the lines to be picked. When finished, hitting ***OK*** creates the area.

Another commonly used method to create areas is to use primitives as part of the top-down approach; this is discussed in Sect. 4.3.2.

4.3.1.4 Volumes

Volumes are used to create a mesh with volume elements (Fig. 4.28). Volumes can be created by using either keypoints or areas. If keypoints are used, the areas and lines that are associated with the volume are automatically generated by ANSYS. Two basic methods are presented below.

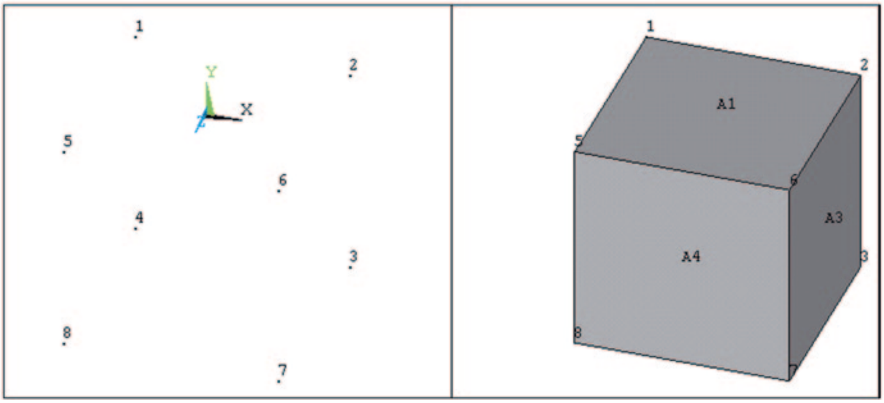


Fig. 4.29 Eight keypoints (*left*); volume created by picking keypoints in 1-2-6-5-4-3-7-8 order (*right*)

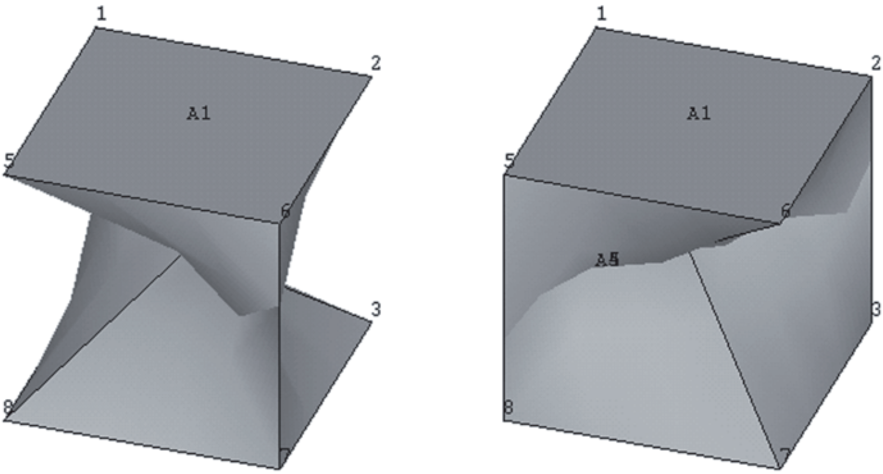


Fig. 4.30 Volumes created by picking keypoints in 1-2-6-5-4-8-7-3 order (*left*) and 1-2-6-5-7-3-4-8 order (*right*)

Creating Volumes Using Keypoints

A maximum of 8 and a minimum of 4 keypoints are required to create a volume using keypoints.Keypoints must be specified in a continuous order.If the volume has 6 faces, two of the opposite faces are required to be specified by the user, and keypoints defining both of these faces should be given in either a clockwise or counterclockwise direction.

For example, a 6-faced volume, shown in Fig. 4.29, requires 8 keypoints.The correct counterclockwise sequence of keypoints is **1-2-6-5-4-3-7-8**. Incorrect sequences, such as **1-2-6-5-4-8-7-3** or **1-2-6-5-7-3-4-8** (Fig. 4.30), have neither a clockwise nor counterclockwise sense and fail to produce the 6-faced volume. Figure 4.31 illustrates volumes with 4 and 5 faces (tetrahedron and triangular prism). The menu path is given as

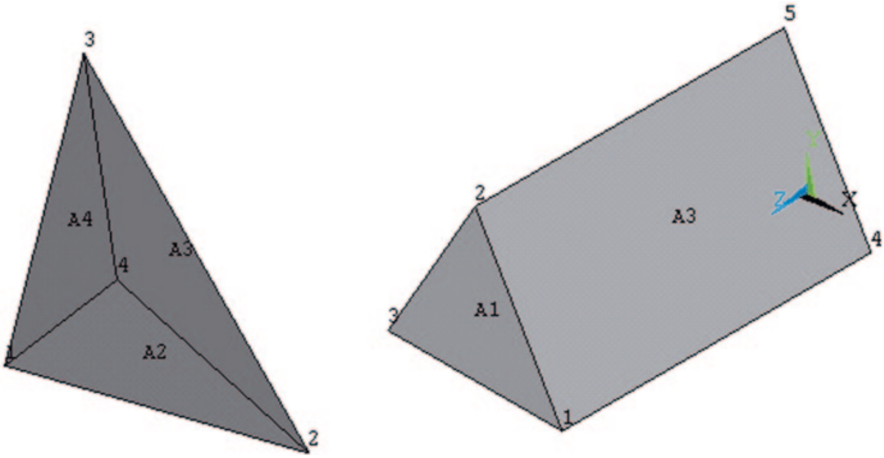


Fig. 4.31 A tetrahedron, 4 faces (*left*), and a triangular prism, 5 faces (*right*)

Main Menu > Preprocessor > Modeling > Create > -Volumes- Arbitrary > Through KPs

This brings up a *Pick Menu*, requesting the keypoints to be picked. When finished, clicking on ***OK*** creates the volume.

Creating Volumes Using Areas
At least four areas (maximum of ten) are required to create a volume through areas. Areas can be specified in any order. The surface defined by the area must be continuous. The menu path is given as

Main Menu > Preprocessor > Modeling > Create > -Volumes- Arbitrary > By Areas

which brings up a *Pick Menu*, requesting the areas. When finished, hitting ***OK*** creates the volume.

4.3.2 Top-Down Approach: Primitives

The primitives are predefined geometrical shapes that enable the user to create a solid model entity (area or volume) with the execution of a single menu item. The user is not required to create keypoints and lines prior to using primitives.

4.3.2.1 Area Primitives

Area primitives are available for the generation of rectangles, circles, and polygons. There are different ways to create each of these primitives. The basic methods are presented here.

Fig. 4.32 Circular area primitives

Rectangle by Dimension
The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Rectangle > By Dimensions

This brings up a dialog box asking for *Working Plane* ***X*** and ***Y*** coordinates of the two corners of the rectangle.After filling out the four fields in this box, clicking on ***OK*** creates the rectangle in the *Graphics Window*.

Rectangle by 2 Corners
The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Rectangle > By 2 Corners

This brings up a *Pick Menu*. There are two ways to finish this action. One way is to use the four fields in the pick menu to input WP coordinates of one corner and the dimensions of the rectangle. The other method is to use the left mouse button to click on the *Graphics Window* to define one corner. After this, as the mouse pointer is moved, ANSYS displays possible rectangles as outlines, with the dimensions quantitatively indicated. When the user finds the right dimensions, a left-click creates the rectangular area.

Solid Circular Area
The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Circle > Solid Circle

This brings up a *Pick Menu*, requesting *Working Plane* ***X*** and ***Y*** of the center of the circle and its ***radius***. They can be supplied either by filling out the fields in the *Pick Menu* or using the mouse pointer. Picking the center of the circle, moving the pointer to find the desired radius (as the mouse pointer is moved, similar to creating rectangles by dimensions, ANSYS plots the circle's outline with the radius identified), and clicking again finalizes the circle generation.

Circular Area by Dimensions
With this option, a solid circle, annulus, circular segment (wedge) or partial annulus can be generated, as shown in Fig. 4.32. The menu path is given as

Main Menu > Preprocessor > Modeling > Create > Circle > By Dimensions

This brings up a dialog box requesting the outer and inner (optional) radii and starting and ending angles of the circular sector. All four of the geometrical parameters are defined with respect to *Working Plane*.If the starting and ending angles are entered as 0 and 360, ANSYS creates a full circular solid area or annulus, depending on the radius information. Otherwise, a partial solid circle (wedge) or a partial annulus is created.If the "Optional inner radius" is left blank (or entered as 0), the area is a solid one; otherwise, it's an annulus.

Polygon
The menu path is given as

Main Menu > Preprocessor > Modeling > Create > -Areas- Polygon > By Vertices

This brings up a *Pick Menu*, requesting the vertices. All the vertices are on the *Working Plane*. Naturally, the polygon must be closed; this is achieved by picking the first point one more time after picking the last point. If the user does not pick the first point to close the polygon and hits ***OK*** in the *Pick Menu*, ANSYS automatically closes the polygon by defining a line between the last and the first point.

4.3.2.2 Volume Primitives

Volume primitives are available for generation of blocks, cylinders, prisms, spheres, or cones. There are different ways to create each of these primitives. The basic methods are presented here.

Block
A block is a rectangular prism. It is created using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Block > By Dimensions

This brings up a dialog box requesting six coordinates, the starting and ending *x*-, *y*-, and *z*-coordinates in the active coordinate system. Figure 4.33 shows the isometric view of a block created using $x_1=y_1=z_1=0$, $x_2=1$, $y_2=2$, and $z_2=3$.

Cylinder The user can create solid or hollow cylinders that encompass either the entire angular range or a part thereof. Cylinders are created using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > By Dimensions

Six parameters are requested in the dialog box:

RAD1 and ***RAD2***: Outer and inner radii of the cylinder. If ***RAD2*** is not specified (or specified as zero), then the cylinder is solid; otherwise, it's hollow.
Z1 and ***Z2***: Starting and ending z-coordinates.

THETA1 and ***THETA2***:Starting and ending angles, measured in degrees, with the active coordinate system *z*-axis defining the axis of rotation.

Figure 4.34 (left) shows an isometric view of a hollow cylinder created using ***RAD1***=1, ***RAD2***=0.5, ***Z1***=0, ***Z2***=2, ***THETA1***=0 and ***THETA2***=360.When the

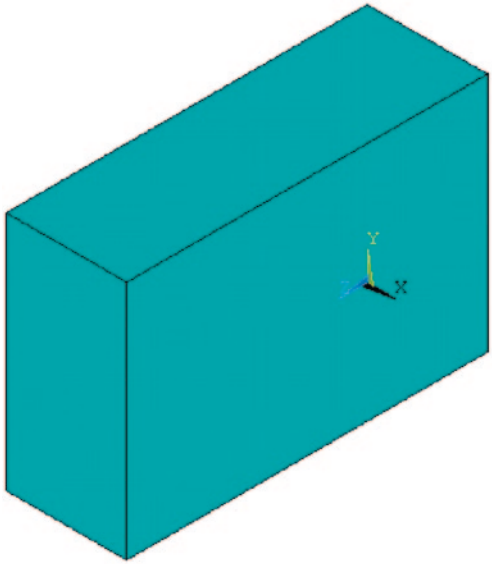


Fig. 4.33 Isometric view of a block created using $x_1=y_1=z_1=0$, $x_2=1$, $y_2=2$, and $z_2=3$

parameter ***THETA1*** is changed to 135° and the other parameters are kept the same, the partial hollow cylinder shown in Fig. 4.34 (right) is created.

Prism

Regular prisms are created using this option. A regular prism is a volume with a constant polygonal cross section in the *Working Plane z*-direction. The menu path for creating prisms is given as

Main Menu > Preprocessor > Modeling > Create > Volumes > Prism > By Side Length

This brings up a dialog box requesting the starting and ending *z*-coordinates ***Z1*** and ***Z2***, respectively; the number of sides (***NSIDES***); and the length of each side

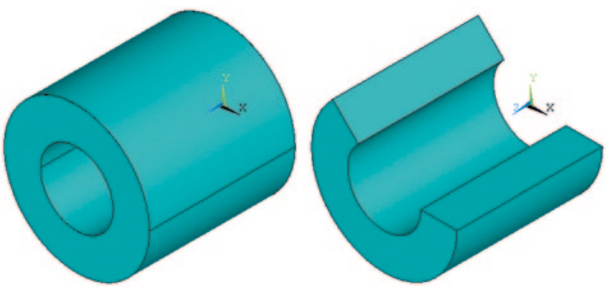


Fig. 4.34 Isometric view of a hollow cylinder (*left*) created using *RAD1*=1, *RAD2*=0.5, *Z1*=0, *Z2*=2, *THETA1*=0, and *THETA2*=360 and a partial hollow cylinder (*right*) when the parameter *THETA1* is changed to 135

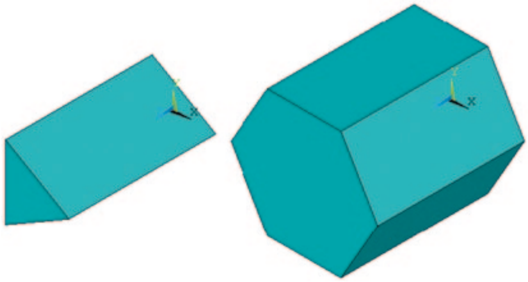


Fig. 4.35 Isometric view of a prism (*left*) created using *Z1* = 0, *Z2* = 2, *NSIDES* = 3, and *LSIDE* = 1 and the prism (*right*) when the parameter *NSIDES* is changed to 6

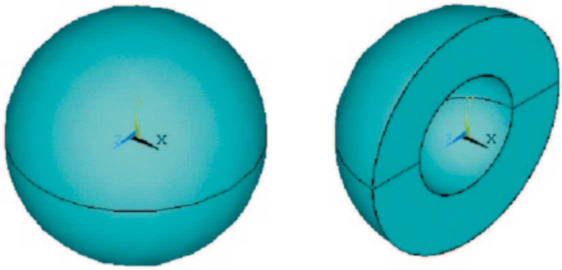


Fig. 4.36 Isometric view of a solid sphere (*left*) created using *RAD1* = 1, *RAD2* = 0, *THETA1* = 0, and *THETA2* = 360 and the partial hollow sphere (*right*) when the parameters *THETA1* and *THETA2* are changed to 90° and 270°, respectively

(***LSIDE***).The center of the polygonal area coincides with the *Working Plane* origin. Figure 4.35 (left) shows the isometric view of a prism created using ***Z1*** = 0, ***Z2*** = 2, ***NSIDES*** = 3, and ***LSIDE*** = 1. Figure 4.35 (right) shows the prism when the parameter ***NSIDES*** is changed to 6.

Sphere
The user can create solid or hollow spheres by using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Sphere > By Dimensions

Four parameters are requested in the dialog box:

RAD1 and ***RAD2***: Outer and inner radii of the cylinder. If ***RAD2*** is not specified (or is specified as zero), then the sphere is solid; otherwise, it's hollow.
THETA1 and ***THETA2***: Starting and ending angles, measured in degrees, with the *Working Plane z*-axis defining the axis of rotation.

Figure 4.36 (left) shows an isometric view of a solid sphere created using ***RAD1*** = 1, ***RAD2*** = 0, ***THETA1*** = 0, and ***THETA2*** = 360. The partial hollow sphere shown in Fig. 4.36 (right) is created when the parameters ***THETA1*** and ***THETA2*** are changed to 90° and 270°, respectively, while the other parameters are kept same.

Cone
Complete or partial cones may be created using this option by following the menu path:

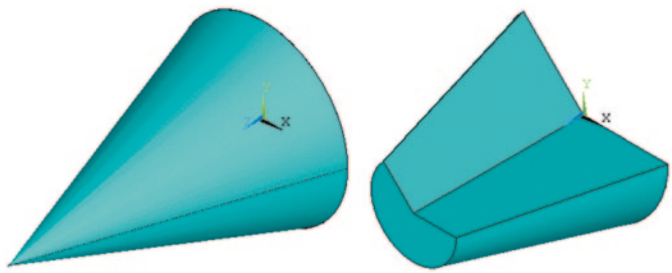


Fig. 4.37 Isometric view of a cone (*left*) created using *RBOT*=1, *RTOP*=0, *Z1*=0, *Z2*=3, *THETA1*=0, and *THETA2*=360 and the partial conical section (*right*) when parameters *RTOP*, *Z2*, and *THETA1* are changed to 0.5, 2, and 135, respectively

Main Menu > Preprocessor > Modeling > Create > Volumes > Cone > By Dimensions

In the dialog box, six parameters are requested:

RBOT and ***RTOP***: Bottom and top radii of the cone.If ***RTOP*** is not specified (or is specified as zero), then a complete cone is generated. If a nonzero ***RTOP*** is specified, then the volume generated is a conical section with parallel top and bottom sides.

Z1 and ***Z2***: Starting and ending *z*-coordinates.

THETA1 and ***THETA2***: Starting and ending angles, measured in degrees, with the *Working Plane z*-axis defining the axis of rotation.It is used for creating conical sections.

Figure 4.37 (left) shows an isometric view of a cone created using ***RBOT***=1, ***RTOP***=0, ***Z1***=0, ***Z2***=3, ***THETA1***=0, and ***THETA2***=360.The partial conical section shown in Fig. 4.37 (right) is created when the parameters ***RTOP***, ***Z2***, and ***THETA1*** are changed to 0.5, 2, and 135, respectively, while the other parameters are kept same.

4.4 Boolean Operators

Many engineering problems possess a complex geometry, making model generation a real challenge. However, the solid model entities can be subjected to certain operations that make model generation much easier. These operations, referred to as Boolean operations, utilize logical operators such as add, subtract, divide, etc. The Boolean operators are applied to generate more complex entities using simple entities (see Fig. 4.38).

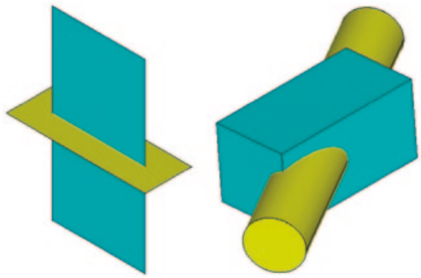


Fig. 4.38 Examples of entities that can co-exist in 3-D space

4.4.1 Adding

The areas to be added must be co-planar (lie in the same plane). As shown in Fig. 4.39, the areas (or volumes) must have either a common boundary or an overlapping region. The original areas or volumes that are added will be deleted unless otherwise enforced by the user. The addition of areas or volumes results in a single (possibly complex geometry) entity, as shown in Fig. 4.40.

Adding entities can be performed by the following menu paths:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Lines

Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas

Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Volumes

4.4.2 Subtracting

Entities can be subtracted from each other to obtain new entities.Subtracting entities can be executed through the menu paths given below:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Lines

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Volumes

This brings up a *Pick Menu*, requesting the user to pick or enter the base entity from which to subtract. The user picks the entities to be subtracted and clicks on the ***OK*** button to complete the operation.

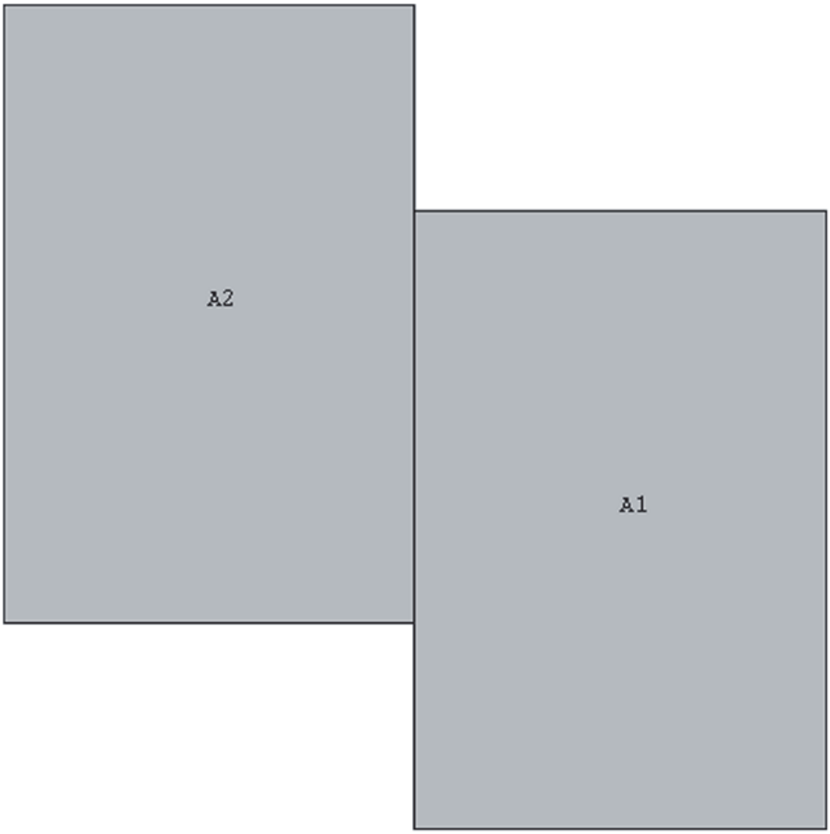
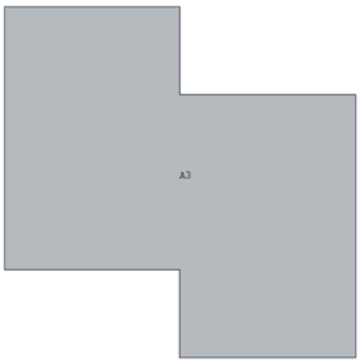


Fig. 4.39 Two areas with a common boundary

Fig. 4.40 Two areas added to produce one area



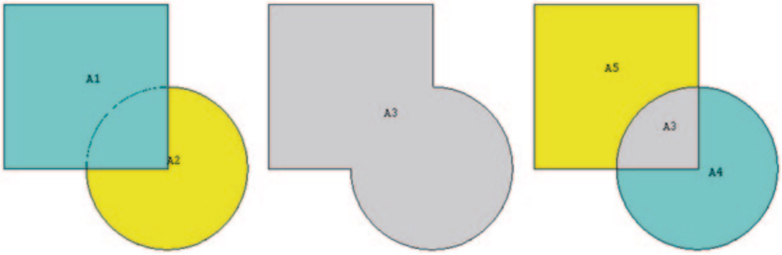


Fig. 4.41 Two areas, A1 and A2 (*left*); the result of adding A1 and A2 (*middle*); and the result of overlapping A1 and A2 (*right*)

4.4.3 Overlap

This operation joins two or more solid model entities to generate three or more entities forming a union of the entire original group of entities, as shown in Fig. 4.41.It is similar to the ***Add*** operation. The only difference between the two is that internal entities are generated in the overlapping operation.

This operation can be executed through the following menu paths:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Lines

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Volumes

This brings up a *Pick Menu* asking for the entities to be overlapped. Picking the entities followed by hitting ***OK*** completes the operation.

4.4.4 Gluing

This operation is used for connecting entities that are “touching” but not sharing any entities. If the entities are apart from or overlapping each other, gluing cannot be used. The glue operation does not produce additional entities of the same dimensionality but does create new entities that have one lower dimensionality. This operation can be executed through the following menu paths:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Lines

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes

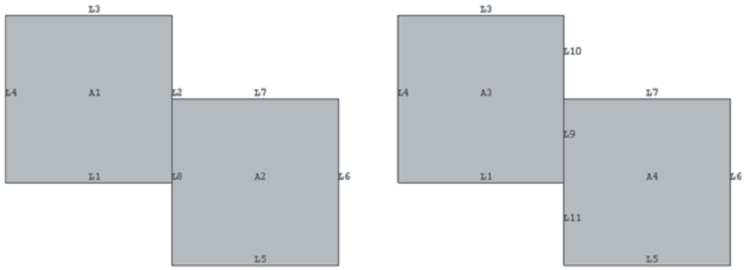


Fig. 4.42 Two areas with a common boundary plotted with line numbers (*left*); they do not share any lines as area 1 (A1) is defined by lines 1 through 4 and area 2 (A2) is defined by lines 5 through 8. After gluing (*right*), the areas share line 9

Before gluing the two areas shown in Fig. 4.42, there are two lines at the interface between Area 1 (A1) and Area 2 (A2).One of these lines is attached to A1, defined by keypoints **2** and **3**, and the other one is attached to A2, defined by keypoints **5** and **8**. Before gluing, these two areas do not "know" of each other's existence because they do not "truly" share any entities. Gluing makes sure that they share entities.After gluing, there are two lines along the right vertical side of A1, and two lines along the left side of A2. The lines along the right vertical side of A1 are defined by keypoints **3** and **8** and keypoints **8** and **2** whereas the lines along the left vertical side of A2 are defined by keypoints **2** and **8** and keypoints **5** and **2**. After gluing, the two areas share one line and two keypoints.

4.4.5 Dividing

A solid model entity can be divided into smaller parts by using other solid model entities. By default, a divided solid model entity is deleted after the operation. There is a wide range of choices for this operation. Some of the available options are presented in Fig. 4.43–4.46.

The menu paths for these operations are given as

Volume by Area

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Volume by Area

Area by Volume

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Area by Volume

Area by Area

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Area by Area

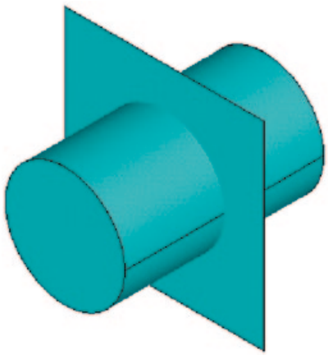


Fig. 4.43 A cylindrical volume is divided into two smaller cylindrical volumes by an area

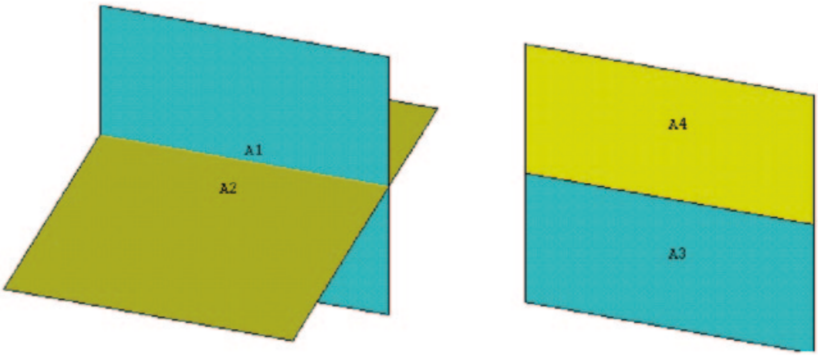


Fig. 4.44 Dividing A1 by A2 (*left*) produces A3 and A4 (*right*)

Area by Line

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Area by Line

Line by Volume

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Line by Volume

Line by Area

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Line by Area

Line by Line

Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Line by Line

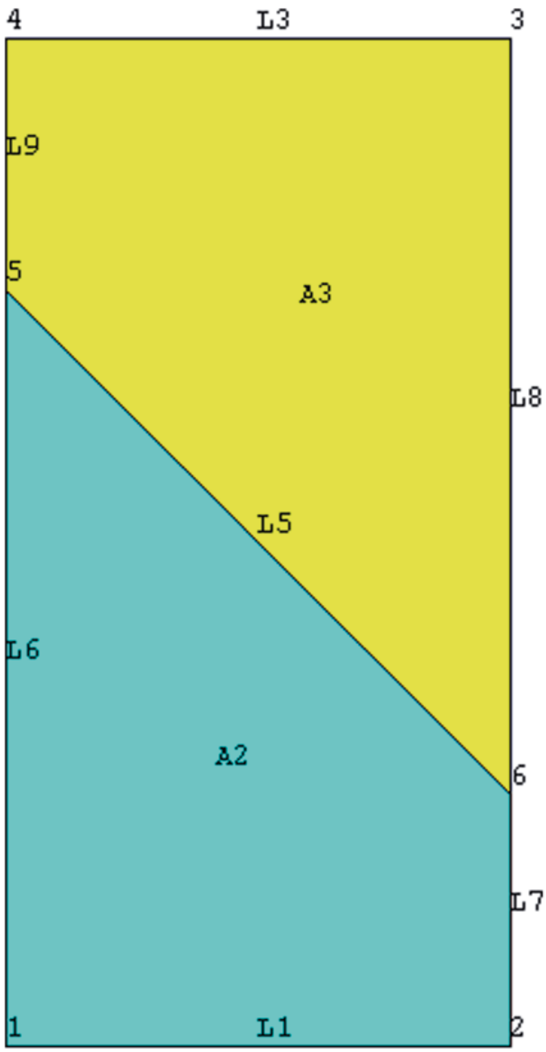


Fig. 4.45 Dividing an area by a line (requires the dividing line to be in the same plane as the area)

4.5 Additional Operations

4.5.1 Extrusion

In addition to Boolean operators, extrusion of the existing entities can be used to generate higher entities. By extruding (dragging) an entity about an axis, one can create a new solid model entity, which is one order higher than the original one (e.g., lines from keypoints, volumes from areas).

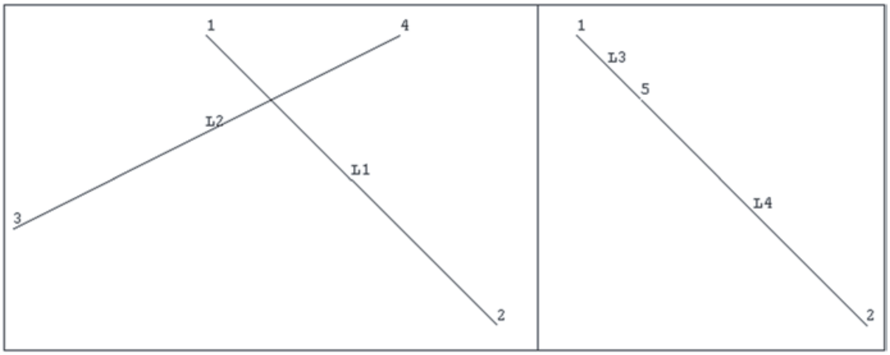


Fig. 4.46 Dividing L1 by L2 (*left*) results in L3 and L4 (*right*)

The commonly used feature of extrusion operation is described in Fig. 4.47, 4.48, 4.49, 4.50, 4.51, 4.52, 4.53 and 4.54. Following are the menu paths used for these operations:

Creating Lines by Rotating a Keypoint About an Axis

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Keypoints- About Axis

Creating Lines by Sweeping a Keypoint Along a Path

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Keypoints- Along Lines

Creating Areas by Rotating Lines About an Axis

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Lines- About Axis

Creating Areas by Sweeping Lines Along a Path

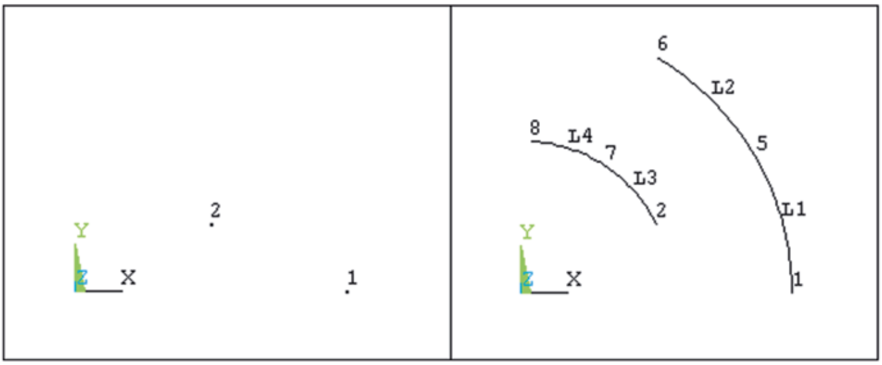


Fig. 4.47 Keypoints 1 and 2 (*left*) are rotated +60° around the *z*-axis in two increments of 30° to create the lines (*right*)

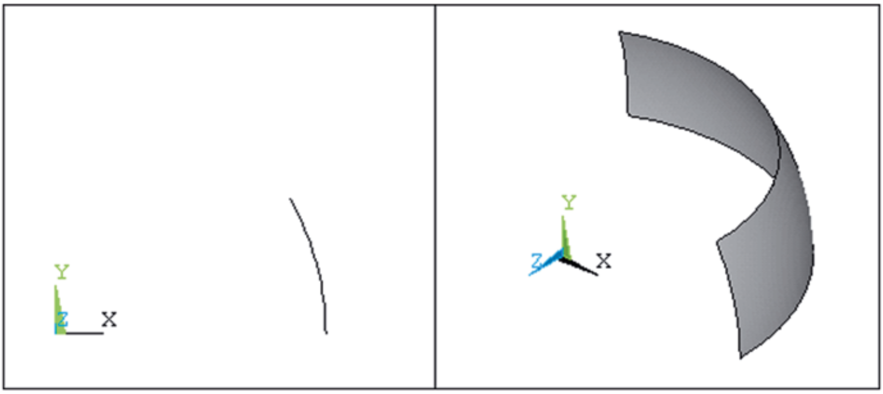


Fig. 4.48 Front view of an arc (*left*); the arc is rotated +60° about the *y*-axis in two increments of 30° to create the curved areas (shown in oblique view on the *right*)

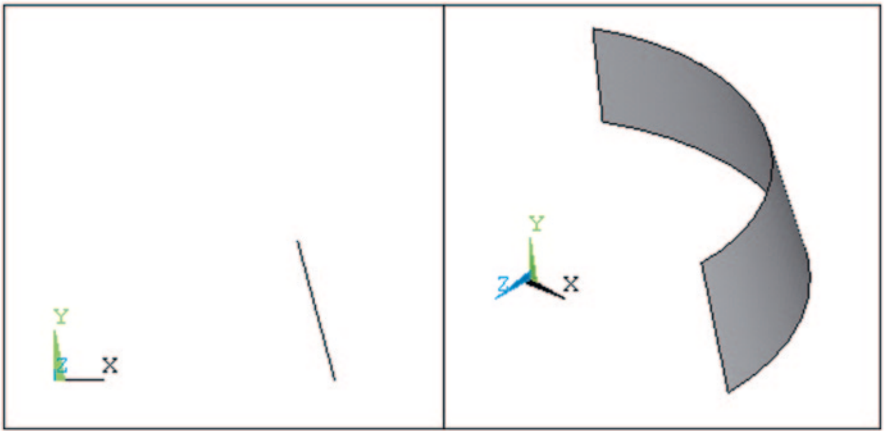


Fig. 4.49 Front view of a straight line (*left*); the line is rotated +60° about the *y*-axis in two increments of 30° to create the curved areas (shown in oblique view on the *right*)

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Lines- Along Lines

Creating Volumes by Rotating Areas About an Axis

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Areas- About Axis

Creating Volumes by Sweeping Areas Along a Path

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Areas- Along Lines

Creating Volumes by Extruding Areas

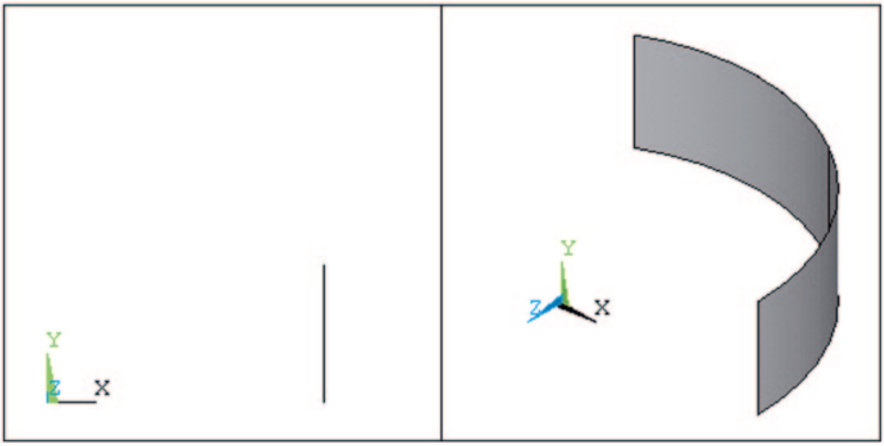


Fig. 4.50 Front view of a straight line (*left*); the line is rotated +120° about the *y*-axis in two increments of 60° to create the curved areas (shown in oblique view on the *right*)

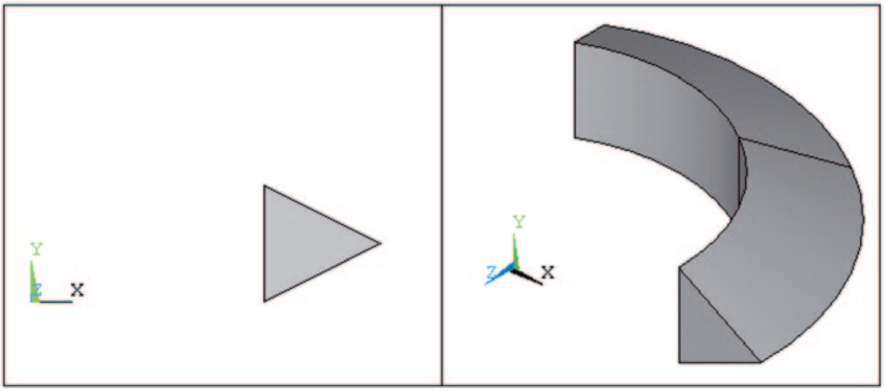


Fig. 4.51 Front view of an area (*left*); the area is rotated +120° about the *y*-axis in two increments of 60° to create the volumes (shown in oblique view on the *right*)

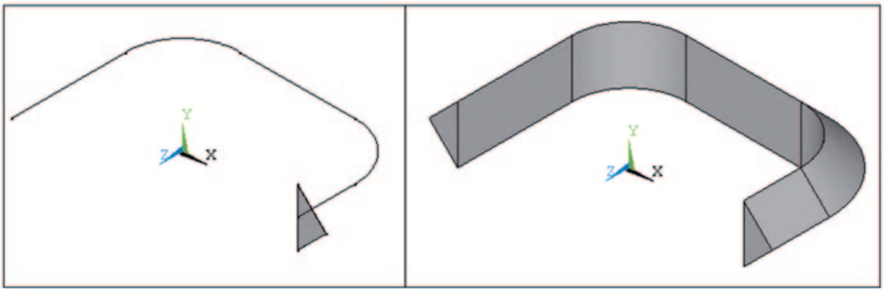


Fig. 4.52 Oblique view of a path defined by lines and an area to be swept along the path (*left*) and the volume created by sweeping the area along the path (*right*)

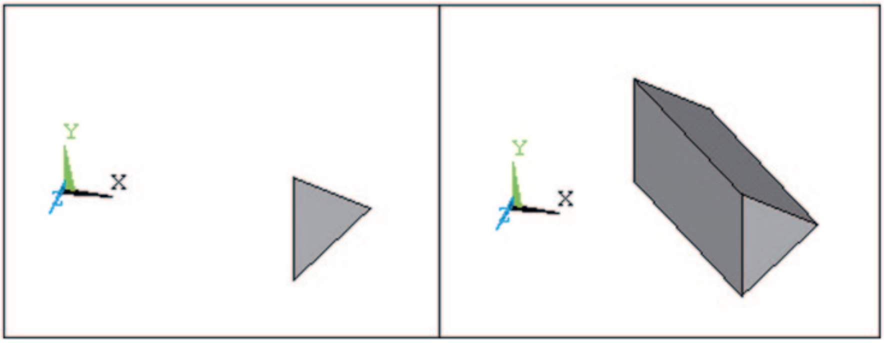


Fig. 4.53 Oblique view of an area to be extruded along its normal (*left*) and oblique view of the volume created by extrusion of the area along its normal (*right*)

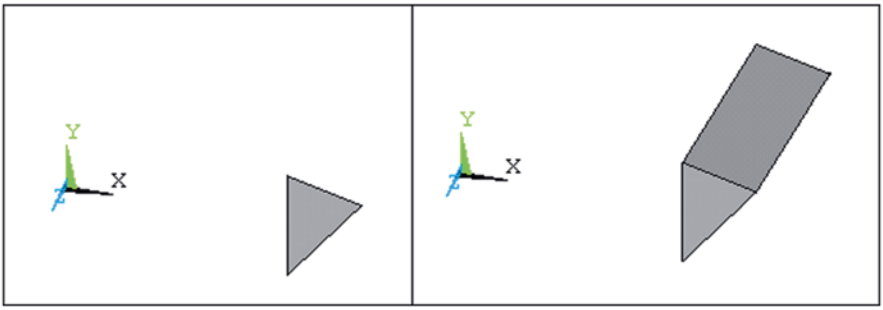


Fig. 4.54 Oblique view of an area to be offset in x, y, or z (*left*) and oblique view of the volume created by offsetting the area in the z-direction (*right*)

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Areas- Along Normal

Creating Volumes by Offsetting Areas

Main Menu > Preprocessor > Modeling > Operate > Extrude > -Areas- By XYZ Offset

4.5.2 Moving and Copying

Previously created entities can be moved or copied.Also, if a repeated symmetry or skew-symmetry exists in the geometry, the user can create a representative entity to create the geometry by copying it to a new location.Representative applications are described in Fig. 4.55, 4.56, and 4.57.

The common menu paths for moving entities are specified as

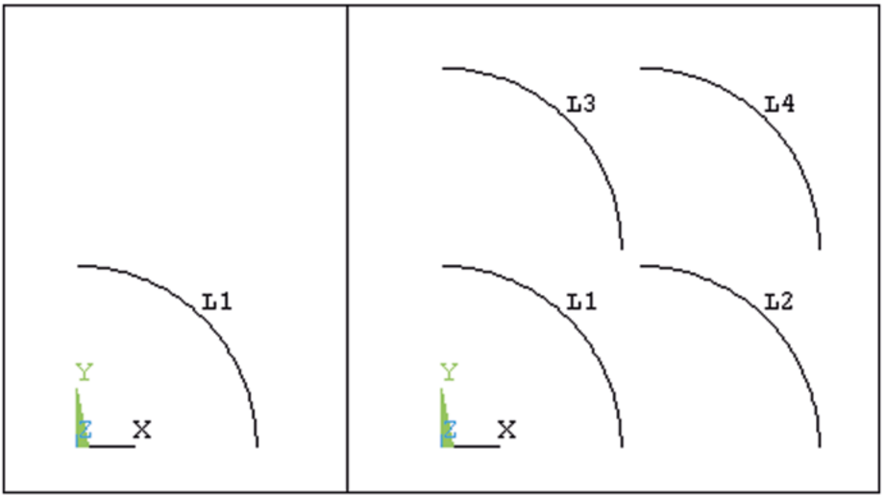


Fig. 4.55 Original line (*left*) and copies of the line created by offsets in *x* (L2), in *y* (L3), and in both *x* and *y* (L4)(*right*)

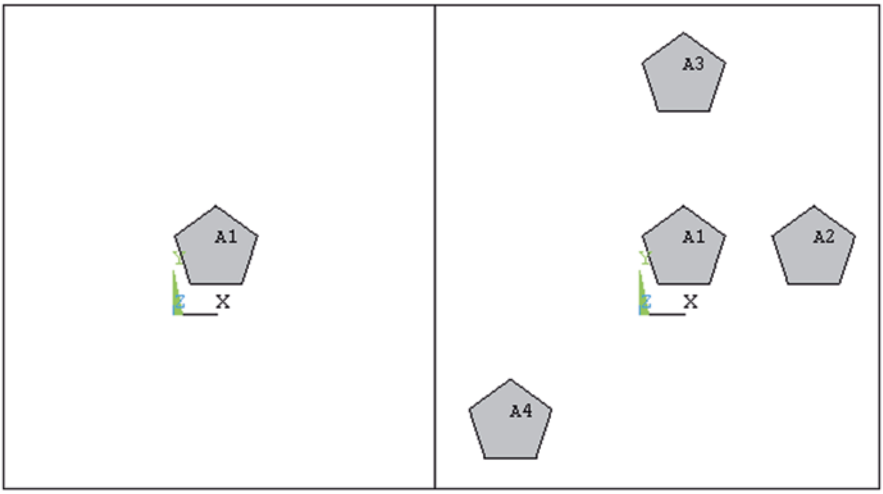


Fig. 4.56 Original area (*left*) and copies of the area created by offsets in *x* (A2), in *y* (A3), and in both *x* and *y* (A4) (*right*)

Main Menu > Preprocessor > Modeling > Move/Modify > -Keypoints- Single KP

Main Menu > Preprocessor > Modeling > Move/Modify > Lines

Main Menu > Preprocessor > Modeling > Move/Modify > -Areas- Areas

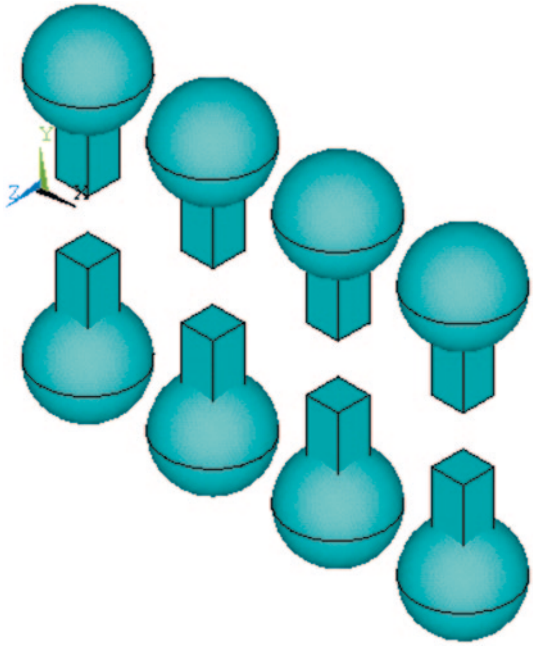


Fig. 4.57 The first volume is copied three times with 0.4 unit offset in the *x*-direction and then reflected with respect to the *x*-*z* plane

Main Menu > Preprocessor > Modeling > Move/Modify > Volumes

The common menu paths for copying entities are specified as

Main Menu > Preprocessor > Modeling > Copy > Keypoints

Main Menu > Preprocessor > Modeling > Copy > Lines

Main Menu > Preprocessor > Modeling > Copy > Areas

Main Menu > Preprocessor > Modeling > Copy > Volumes

4.5.3 Keeping/Deleting Original Entities

During the performance of Boolean-type operations, there are input entities (e.g., original areas to be added or, when dividing a line with a volume, the original line and volume) and output entities. By default, ANSYS will delete the input entities, keeping only the output entity.However, the input entities can be "kept" through the menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Settings

This brings up a dialog box requesting the user to specify certain settings. The first setting option controls whether the input entities will be kept or deleted. Answering ***Yes*** instructs ANSYS to keep the input entities; otherwise, they are deleted.

4.5.4 Listing Entities

In most cases, plotting is an effective way to quickly examine the model. However, if there are unexpected errors or if the model is not what the user intended to create, it may be difficult to identify what went wrong. In such cases, the user can examine the model in a more accurate way by listing the entities. ANSYS provides options for listing solid model entities with detailed information. All of the lists are given in a new window (which can be saved to disk or printed) where the entities are sorted by their reference numbers in ascending order. Lists for the solid model entities can be obtained by following the menu paths:

Utility Menu > List > Keypoints > Coordinates only

Utility Menu > List > Lines > Attribute format

Utility Menu > List > Areas

Utility Menu > List > Volumes

4.5.5 Deleting Entities

During solid modeling, it is very common for the user to create unintended solid modeling entities. These extra entities might make the modeling phase confusing and may potentially cause errors.In order to eliminate this possibility, the user should "clean up" the model by deleting these entities. The hierarchy of the solid model entities is important in that the entity (or entities) must not be used for the definition of any higher-order entities in order to be deleted.For example, the existence of an area automatically implies that lines and keypoints are attached to this area.None of the lines can be deleted as long as the area exists. The area must first be deleted, then the lower-order entities can be deleted. Similarly, a KP cannot be deleted as long as the line(s) to which the KP is attached exist(s). Only after the line(s) is(are) deleted, can the KP can be deleted. Solid model entities can be deleted in two different methods:

1. Delete the entity without deleting the lower-order entities that are attached to it.
2. Delete the entity and all the lower-order entities that are attached to it. In this case, if some of the lower-order entities are associated with other entities, they will not be deleted.

The following menu paths are used for these two methods:

To Delete Entities Only

Main Menu > Preprocessor > Modeling > Delete > Keypoints

Main Menu > Preprocessor > Modeling > Delete > Lines Only

Main Menu > Preprocessor > Modeling > Delete > Areas Only

Main Menu > Preprocessor > Modeling > Delete > Volumes Only

To Delete Entities and Below

Main Menu > Preprocessor > Modeling > Delete > Lines and Below

Main Menu > Preprocessor > Modeling > Delete > Areas and Below

Main Menu > Preprocessor > Modeling > Delete > Volumes and Below

4.6 Viewing a Model

ANSYS provides a very robust graphic utility to view solid model entities, nodes, elements, material properties, boundary constraints, loads, and results. The graphics-related utilities are accessible through the *Plot* and *PlotCtrls* (stands for Plot Controls) submenus under the *Utility Menu*. All of the entities can be viewed through the *Plot* submenu. However, the *PlotCtrls* submenu (as its name indicates) provides many options that enhance the use of the plot utility for many different purposes, such as plotting the numbers associated with entities, plotting the entities in different colors,[1] and viewpoint and viewing angle adjustments.

For the sake of brevity, only the most frequently used items are explained in detail here.However, many more features are discussed in the examples.

4.6.1 Plotting: Pan, Zoom, and Rotate Functions

The *Pan-Zoom-Rotate* tool is a very effective function of ANSYS for manipulating the view by panning, zooming, and rotating the model. The following menu path is used to activate this function:

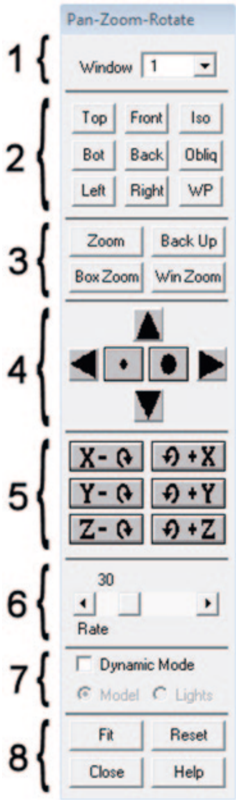
Utility Menu > PlotCtrls > Pan, Zoom, Rotate

The *Pan-Zoom-Rotate* window appears, as shown in Fig. 4.58. There are eight different fields in this window:

1. *Active Window Field*: The *Graphics Window* can be divided into "sub"-windows (up to 5) in ANSYS; only one of them can be the "active" window at any one time.This button identifies which window(s) are to be affected by the operations performed within the *Pan-Zoom-Rotate* window.
2. *Viewing Direction Field*: This group of buttons changes the viewpoint. Clicking on the ***Top*** button will redraw the model (or entities) as seen from the top. In ANSYS, "top" corresponds to the positive global Y-direction. Similarly, "front" and "right" will redraw the model as seen from the positive global Z-and

[1] Colors have not been used in the printed version of the figures. See the accompanying CD-ROM for color versions of the figures.

Fig. 4.58 The eight fields in the *Pan-Zoom-Rotate* window



X-directions, respectively. The ***Iso*** and ***Obliq*** buttons redraw the model as seen from a point that lies on a line that passes through the origin and (1, 1, 1) and (1, 2, 3), respectively. Finally, the ***WP*** button redraws the model by taking the positive Working Plane *z*-direction as the front of the model.

3. *Zoom Field*: Provides different zooming methods:

***Zoom*:** Clicking on this button, followed by a single left-click, chooses the center of the region of interest. After the first click, moving the mouse toward and away from the center will display a moving square outline of the potential target region that the user would like to zoom in. Once decided, a second left-click will zoom in to the region indicated by the outline.

***Box Zoom*:** This function works in a similar way. The user picks two corners of the zoom-in region. After picking the first corner by clicking the left mouse button, moving the mouse over the *Graphics Window* will show a moving outline of the potential zoom-in region. A second click will pick the second corner and ANSYS will redraw the zoom-in region.

***Backup*:** Clicking on this button redraws the model in the previous viewing configuration.

Win Zoom*:** This button works like the ***Box Zoom button except that after picking the first point, ANSYS locks the aspect ratio of the potential zoom-in region at the same values as the aspect ratios of the active window (the redraw of the zoom-in region will fit perfectly in the window).

4. *Pan/Zoom Field*: The arrow buttons pan the model in the indicated directions and the dots zoom in and out. A small dot indicates zooming out and a large dot indicates zooming in. The *Sliding Rate Control Bar*, explained below, dictates the rate at which pan and zoom actions operate.
5. *Rotate Field*: These six buttons rotate the model about "Screen" *x*-, *y*-, and *z*-directions. The "screen origin" is the center of the active window. The positive "Screen" *x*-direction starts from the center of the window and extends to the right.Likewise, the positive "Screen" *y*- and *z*-directions start at the center of the window and extend to, respectively, the top and out (of the monitor).
6. *Rate Control Field*: The Sliding Bar controls the rate of pan, zoom, and rotate that is performed in the active window. The range is from 1 to 100 (rate 1 pans/zooms at a smaller rate than rate 100 would).
7. *Dynamic Mode Field*: By clicking on this radio button, the user toggles on/off the option to pan and rotate dynamically. When the *Dynamic Mode* is active, the mouse pointer changes shape when it is over the *Graphics Window*. Pressing the left mouse button (without releasing) and moving around in the *Graphics Window* pans the model. Similarly, the right mouse button is used for rotating the model dynamically.
8. *Action Field*: Includes four action buttons:

***Fit*:** Fits the whole model in the active window.
***Reset*:** Restores the default orientation and size for viewing (front view).
***Close*:** Closes the *Pan-Zoom-Rotate* window.
***Help*:** Brings the help page for *Pan-Zoom-Rotate* window.

4.6.2 Plotting/Listing Entities

The following menu paths are used to plot and list the solid model (keypoints, lines, areas, and volumes) and mesh (nodes and elements) entities:

Utility Menu > Plot > Keypoints > Keypoints

Utility Menu > Plot > Lines

Utility Menu > Plot > Areas

Utility Menu > Plot > Volumes

Utility Menu > List > Volumes

Utility Menu > Plot > Nodes

Utility Menu > List > Nodes

Utility Menu > Plot > Elements

Utility Menu > List > Elements > Nodes + Attributes

The resulting plots are displayed in the *Graphics Window* and can be examined using the *Pan-Zoom-Rotate* window discussed in the previous section.

4.6.3 Numbers in the Graphics Window

Whenever an entity is being created, ANSYS either asks for a reference number or assigns the lowest available number for that type of entity. Therefore, every entity differs from the other entities of the same type by this reference number. When plotting these entities in the *Graphics Window*, by default, ANSYS will not show the entity numbers. Often times, it is important for the user to see the numbers printed when plotting entities. This can be done using the following menu path:

Utility Menu > PlotCtrls > Numbering

which brings up the *Plot Numbering Controls* dialog box, as shown in Fig. 4.59. The entity numbers for keypoints, lines, areas, volumes, and nodes can simply be turned on by placing a checkmark in the corresponding boxes. Element numbers

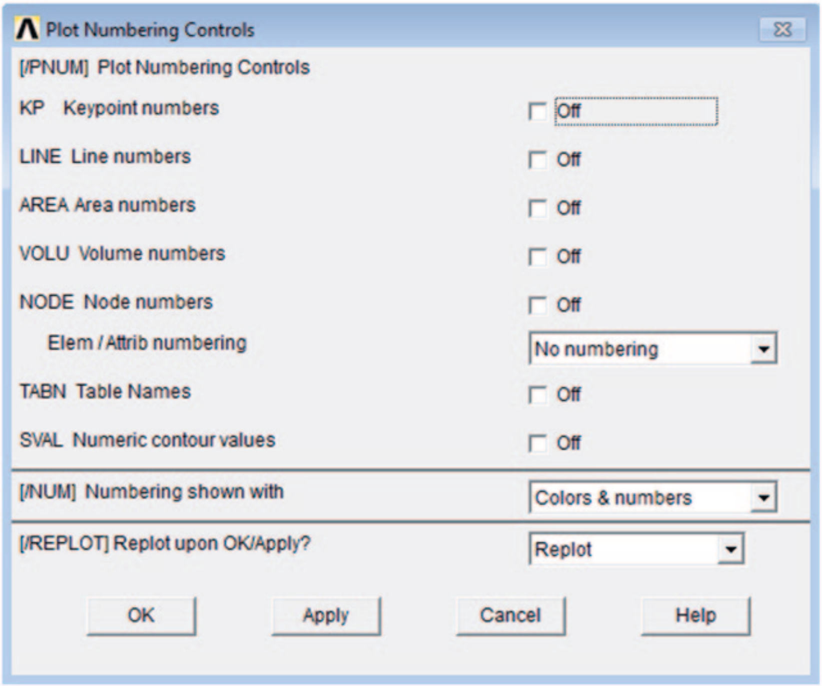


Fig. 4.59 *Plot numbering controls* dialog box

can be turned on using the ***Elem/Attrib numbering*** pull-down menu. Instead of the element numbers, the user can display element attribute numbers (element type, real constant, and material) using the same option.Also, colors may be assigned to each entity number for more convenient viewing.The ***[/NUM] Numbering shown with*** pull-down menu in this dialog box allows the user to plot numbers with or without color assignments, as well as to plot using colors only (without numbers).

4.7 Meshing

As mentioned previously (Sect. 4.3), the mesh of the geometry under consideration may be generated directly, i.e., generation of nodes and elements, one at a time. However, this may prove to be a challenging task. Almost always *Solid Modeling* constitutes a part of the finite element analysis. Thus, the sole purpose of *Solid Modeling* is to create the mesh of the geometry, as conveniently and efficiently as possible. Once the *Solid Model* is completed, the user is ready to perform meshing. Regardless of whether a *Solid Model* is generated or not, the meshing can be performed only after the specification of element type(s). ANSYS offers several convenient options to assist in meshing. These include *Automatic Meshing*, *Smart Sizing*, and *Mapped Meshing*. In the following subsections, topics related to meshing are discussed in more detail.

4.7.1 Automatic Meshing

One of the most powerful features of ANSYS is automatic mesh generation. ANSYS meshes the solid model entities upon execution of an "appropriate" single command. With automatic meshing, the user can still provide specific preferences for mesh density and shape. If no preferences are specified by the user, ANSYS uses the default preferences. The following menu paths are used for automatic mesh generation after solid model generation:

Mesh Using Line Elements
This option is used for models utilizing one-dimensional elements, such as trusses and beams. It requires existing lines. The following menu path is used to mesh lines:

Main Menu > Preprocessor > Meshing > Mesh > Lines

This brings up a *Pick Menu* asking the user to either enter the line number(s) through the *text field* or pick line(s) from the *Graphics Window*. When all the lines are input (picked), hitting ***OK*** in the *Pick Menu* generates the mesh.

Mesh Using Area Elements
This option is used for models utilizing 2-D elements, and it requires existing areas. The following menu path is used to mesh areas:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 Sided

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

Meshing can be accomplished through either the *Mapped* or *Free Meshing* methods.If free meshing is chosen, the second menu path is used, bringing up a *Pick Menu* asking the user to either enter the area number(s) through the *text field* or pick area(s) from the *Graphics Window*. When all the areas are input (picked), hitting ***OK*** in the *Pick Menu* generates the mesh. The *Mapped meshing* option is discussed in a later subsection.

Mesh Using Volume Elements
This option is used for models using 3-D elements, and it requires existing volumes.

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > 4–6 Sided

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Free

This brings up a *Pick Menu* asking the user to either enter the volume number(s) through the *text field* or pick volumes(s) from the *Graphics Window*. When all the volumes are input (picked), hitting ***OK*** in the *Pick Menu* will generate the mesh.

ANSYS allows the user to control the mesh density of the domains defined by solid model entities. The desired mesh density can be achieved by:

- Defining a target element edge size on the domain boundaries.
- Defining a default number of element edges on all lines.
- Defining the number of element edges on specific lines.
- Using *smart sizing.*
- Using *mapped meshing.*

These methods are discussed in detail in the following subsections.

4.7.1.1 Specifying Mesh Density Globally

There are two approaches for enforcing the mesh density globally. The first one involves specification of the element edge size; ANSYS attempts to generate a mesh with all elements having edge sizes as close as possible to the specified value. The second possibility is to specify a fixed number of elements along all the lines within the solid model. The following menu path is used for specifying the mesh density globally:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

This brings up the *Global Element Sizes* dialog box with two input parameters: ***SIZE*** and ***NDIV***.***SIZE*** denotes the target element edge length, and ***NDIV*** is the target number of elements along the lines. If ***SIZE*** is specified, ***NDIV*** is ignored. The following example explains these concepts. Consider a square area with sides

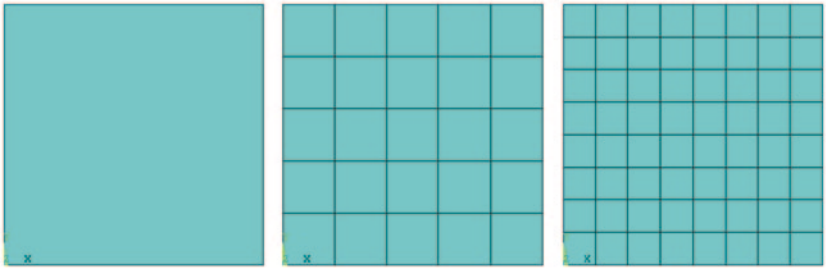


Fig. 4.60 A square area (*left*) meshed with global element size (*SIZE*) specified (*middle*) and number of line divisions (*NDIV*) specified globally (*right*)

5 units long, as shown in Fig. 4.60 (left). If the global element size is specified as 1 (***SIZE*** = 1), the mesh shown in Fig. 4.60 (middle) is generated with each element having an edge size of 1 unit. If the user chooses to the specify the number of elements along lines instead of element sizes, then ***SIZE*** is left untouched (zero), and ***NDIV*** is set to a specific value, say 8. As a result of this operation, the mesh shown in Fig. 4.60 (right) is generated, with 8 elements along each line.

Specification of mesh density globally works well when the geometry of the problem is regular, with aspect ratio close to one. When domains of irregular shapes are considered, applying the same meshing targets to lines of different sizes results in meshes with high aspect ratios, leading to potentially erroneous results. Therefore, the techniques explained in the following subsections are more desirable.

4.7.1.2 Specifying Number of Element Edges on Specific Lines

When the geometry of the problem is irregular, i.e., not basic shapes such as triangles and rectangles, specifying the number of element edges along specific lines may be a good way to avert possible meshing problems. This strategy also helps to refine the mesh around regions where it may be crucial for accuracy. Similarly, certain regions in the geometry may not be critical, and keeping the mesh around these regions may help reduce the computational cost without losing accuracy. The number of element edges on specific lines can be specified using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

which brings up the *Pick Menu* for line picking. After the user picks the lines and clicks on ***OK***, the *Element Sizes on Picked Lines* dialog box appears. The second parameter, ***NDIV***, dictates how many elements will be placed along the picked lines. The third parameter, ***SPACE***, which stands for spacing ratio, is important when a mesh graded (biased) toward a direction is desired. The default value for ***SPACE*** is 1 (no bias, uniform spacing). If it is positive, the spacing is biased from one end of

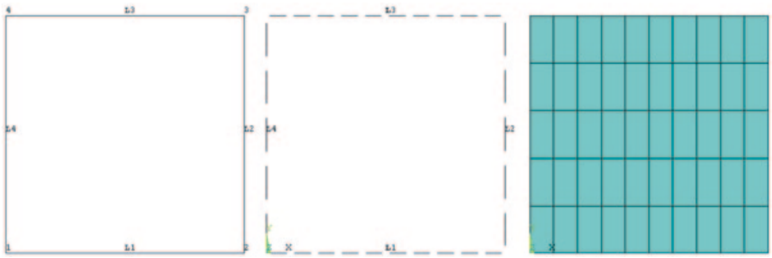


Fig. 4.61 A square area (*left*) with number of line divisions specified at specific lines (*middle*) and the resulting mesh (*right*)

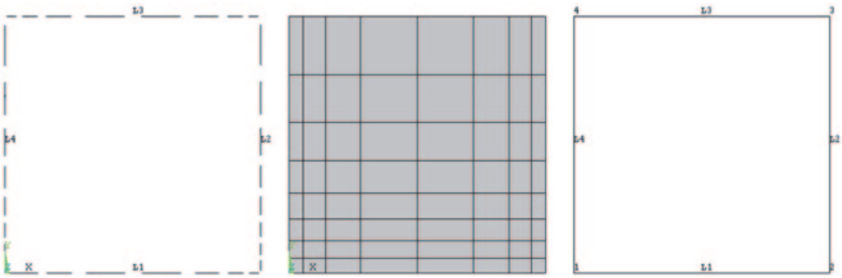


Fig. 4.62 Biased line divisions (*left*), resulting mesh (*middle*), and lines with keypoint numbers plotted (*right*)

the line to the other end.If it is negative, then the bias is from the center toward the ends.Its magnitude defines the ratio of the largest division size to the smallest.These concepts are explained in the following examples. Consider the square area used for the example in the previous subsection, shown in Fig. 4.61 (left) with line numbers. Using the menu path above, ***NDIV*** is specified to be 5 for lines 2 and 4, and 10 for lines 1 and 3. After this operation, lines are plotted with the specified divisions clearly visible [Fig. 4.61 (middle)].Meshing of the area produces the one shown in Fig. 4.61 (right). The same example is considered, this time with the specification of spacing ratios.The goal is to have a mesh graded from coarse at the center to fine at the edges in the x-direction, and from coarse at the top to fine at the bottom (in the y-direction). The same number of divisions is used for all the lines with a value of 8 (***NDIV***=8). Using the menu path given above, spacing ratios (***SPACE***) for lines 1, 2, 3, and 4 are specified as –4, 4, –4, and 0.25, respectively. The line plot after this operation is shown in Fig. 4.62 (left), and the corresponding mesh is given in Fig. 4.62 (middle). It is worth noting the reason why the parameter ***SPACE*** is different for lines 2 and 4. Figure 4.62 (right) shows the line plot with the keypoint numbers. Line 2 is defined from keypoint 2 to keypoint 3 (from bottom to top) whereas line 4 is defined from keypoint 4 to keypoint 1 (from top to bottom). When ***SPACE*** is positive and greater than 1, its value defines the ratio of the division length at the end of the line to the length of the division at the beginning of the line.

4.7.1.3 Smart Sizing

Instead of specifying the number of line divisions or element edge sizes, one can use the ANSYS “smart sizing” feature, where the mesh density is specified in a cumulative sense. In this method, the user specifies a level of refinement that ranges from **1** to **10**; the smaller the number, the finer the mesh. To use smart sizing, follow the menu path given below:

Main Menu > Preprocessor > Meshing > Size Cntrls > SmartSize > Basic

This brings up a dialog box with a pull-down menu asking the user to choose a level of refinement. Selecting the level, followed by hitting ***OK***, activates the smart sizing. Now, the user is ready to mesh the solid model entities. As this option uses meshing options involving advanced geometry, there is no easy way to explain how it works.Therefore, it is suggested that the user experiment with it, and build a knowledge base that will be helpful later on.

4.7.1.4 Mapped Meshing

Another very commonly used (by experienced ANSYS users) meshing method is *Mapped Meshing*. The mapped meshing concept is valid only in two- and three-dimensional problems (no line elements). The solid model entities (areas and volumes) meshed with this option use quadrilateral area elements or hexahedral (brick) volume elements.

The reason why mapped meshing is desirable is that it generates regular, thus computationally well-behaving, meshes. Not every area or volume can be mapped meshed. The areas or volumes to be mapped meshed must be “regular.” This regularity is governed by two properties of the solid model entity: the number of sides (lines for areas and areas for volumes) and number of divisions on opposite sides (opposite sides must have an equal number of divisions). For areas, the acceptable number of sides is 3 or 4. If the area has 3 sides (defined by 3 lines), then the number of divisions in all 3 lines must be equal and even. If 4 lines define the area, as stated before, the lines on opposite sides must have the same number of divisions. These considerations are similar for mapped meshing of volumes. The number of areas that define the volume must be either 4 (tetrahedron), 5 (prism), or 6 (hexahedron). The number of divisions on opposite sides must be equal. If 4 or 5 areas define the volume, the number of divisions on the triangular areas must be equal and even.

To use mapped meshing, follow the menu path given below:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 Sided

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 to 6 Sided

which brings up a *Pick Menu* for area picking.After the areas are picked and the ***OK*** button is pressed, the mesh is generated. Figure 4.63 (left) shows a triangular area

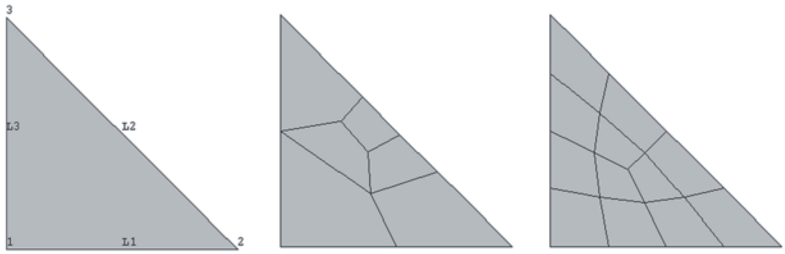


Fig. 4.63 Triangular area (*left*) and corresponding free (*middle*) and mapped (*right*) meshes

with corresponding free and mapped meshes given in Fig. 4.63 (middle) and 4.63 (right), respectively. It is clear from these figures that the mapped mesh delivers elements with controlled and desirable aspect ratios.

If the areas or volumes do not have the required number of sides that are given above, there might still be a way to "mapped mesh" these entities. For this, the user looks for sides that could be considered as a single side when combined. This way the number of sides can be reduced to the required numbers. This operation is performed through "concatenating" lines (for meshing of areas with more than 4 sides) and areas (for meshing of volumes with more than 6 sides). Line and area concatenations are performed using the following menu paths:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > Concatenate > Lines

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > Concatenate > Areas

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > Concatenate > Lines

which brings up a *Pick Menu* for lines to be concatenated. The concatenation is explained through the following example. Consider the irregular area, shown in Fig. 4.64 (top), with line numbers plotted. Free meshing of this area produces the mesh given in Fig. 4.64 (middle), with elements having large aspect ratios. As observed from Fig. 4.64 (top), the area to be meshed is enclosed by 7 lines (sides). When mapped meshing areas, the maximum number of sides is 4. Therefore, if the user wants to mapped mesh this area, line concatenations must be performed. For this purpose, lines 1, 2, and 3 are concatenated to produce a new line (line 8). Also, lines 4 and 5 are concatenated, producing line 9. With these concatenations, the number of sides defining the area is reduced from 7 to 4 and mapped meshing is possible. After specifying the number of divisions on lines 6 and 7 as 3, mapped meshing is performed by using the menu path given above, producing the mesh given in Fig. 4.64 (bottom), with elements having acceptable aspect ratios.

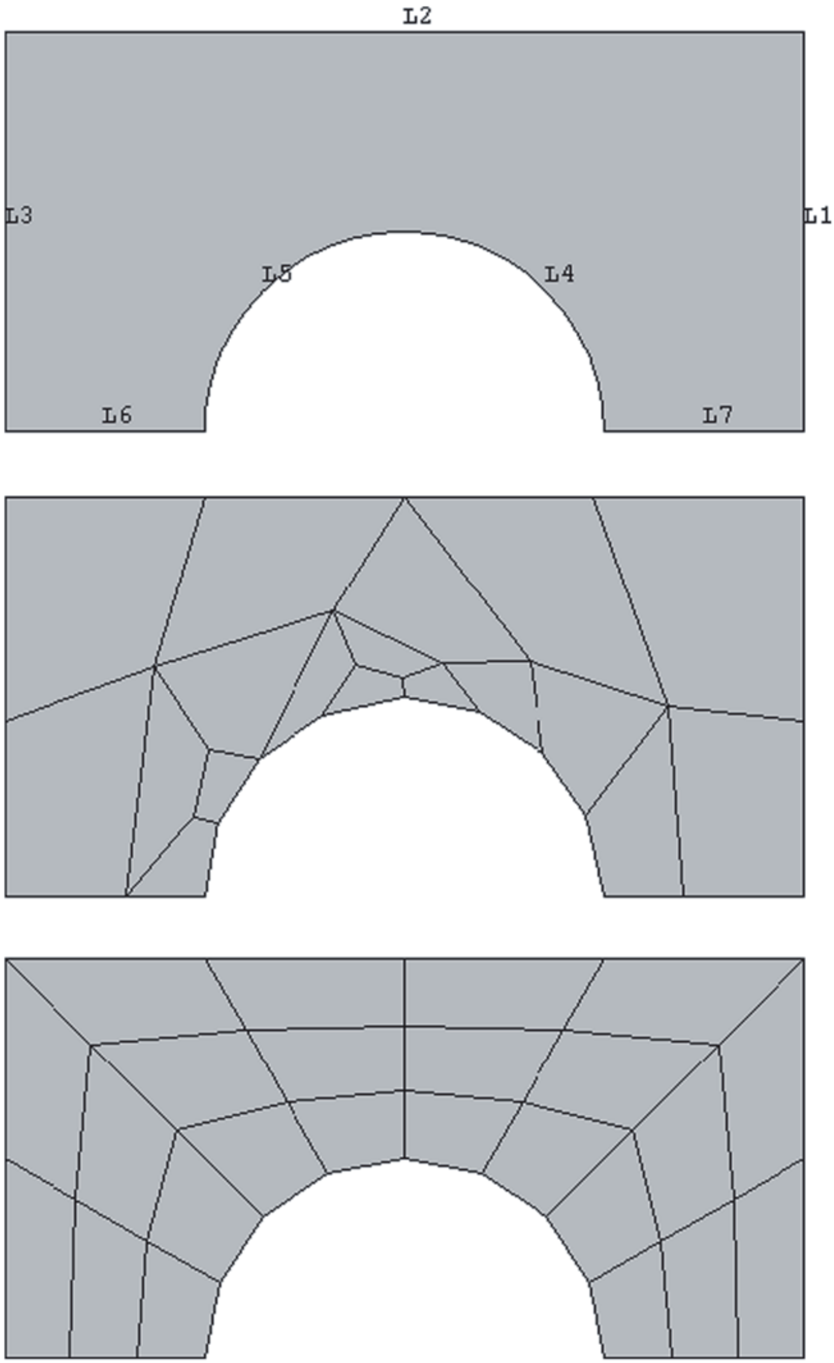


Fig. 4.64 An irregular area (*top*) meshed using free meshing (*middle*) and meshed with mapped meshing after line concatenation (*bottom*)

4.7.2 *Manipulation of the Mesh*

4.7.2.1 Changing Element Attributes

The user can change the attributes of the elements after the mesh is generated. This is achieved by using the following menu path:

Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Modify Attrib

This will bring up a *Pick Menu* asking the user to pick element(s) from the *Graphics Window*. Once the element(s) are selected, clicking on ***OK*** in the *Pick Menu* leads to a dialog box with two fields: (1) a pull-down menu containing the attributes, and (2) a new attribute reference number for the selected attribute. After selecting the attribute to be changed, followed by entering the new attribute reference number, clicking on ***OK*** finalizes the operation.

4.7.2.2 Clearing and Deleting Mesh

After a mesh is generated, there are ways to re-mesh if the user is not satisfied with the result. If direct generation was used for meshing, the user can simply “delete” elements first, and then nodes. Note that deleting elements does not automatically delete the nodes. These tasks are performed as follows:

Main Menu > Preprocessor > Modeling > Delete > Elements

Main Menu > Preprocessor > Modeling > Delete > Nodes

If the solid model approach was used, the elements and nodes cannot be “deleted” since they are “attached” to the solid model entities. However, more conveniently, the solid models that elements and nodes are attached to can be “cleared.” This deletes all elements and nodes attached to the solid model entity at once. The user can now re-mesh the solid model entities after certain changes are made in meshing controls. The menu paths for this operation are as follows:

Main Menu > Preprocessor > Meshing > Clear > Keypoints

Main Menu > Preprocessor > Meshing > Clear > Lines

Main Menu > Preprocessor > Meshing > Clear > Areas

Main Menu > Preprocessor > Meshing > Clear > Volumes

There are cases where it may be advantageous to remove the association between the solid model and the mesh. This is achieved by using the following menu path:

Main Menu > Preprocessor > Checking Ctrls > Model Checking

which brings up a dialog box with a pull-down menu.Selecting the item ***Detach*** in the pull-down menu and clicking on ***OK*** removes the association between the solid model and the mesh.

4.7.2.3 Numbering Controls

When dealing with complicated geometries, the Boolean operations explained in Sect. 4.4 are used regularly. These operations often generate new entities while removing existing ones, which creates gaps in the entity numbering. For example, when an area (say, area 1) is subtracted from another one (area 2), the resulting area is given the smallest available area number (in this case, area 3). Immediately after the creation of this area (area 3), ANSYS internally deletes the input areas (areas 1 and 2). Similarly, all the keypoints and lines associated with the new area are given new numbers while the ones associated with the old areas are removed. Similar considerations apply to nodes and elements. ANSYS provides the user with the option of "compressing" the entity numbers, which is performed as follows:

Main Menu > Preprocessor > Numbering Ctrls > Compress Numbers

which leads to a dialog box with a pull-down menu. After the entity label is selected from this menu, clicking ***OK*** finalizes this operation.

Another important concept in solid modeling and meshing is the possible existence of duplicate entities. This usually occurs when the user creates new entities by copying existing ones or by reflection about a plane. If the old and new entities occupy the same space and if the material is supposed to be continuous along the line (or plane) where duplicate items lie, then they must be merged. Although it may not be apparent in the *Graphics Window*, the existence of duplicate entities compromises the continuity of the mesh, and thus may lead to invalid solutions. The following menu path is used to merge entities:

Main Menu > Preprocessor > Numbering Ctrls > Merge Items

This brings up a dialog box in which the first field is a pull-down menu for the label of entities to be merged.

4.8 Selecting and Components

In the case of a three-dimensional finite element model, graphical picking may become a tedious and, at times, frustrating experience. In such cases, the selection tool provided by ANSYS is highly useful. This tool is efficient as it utilizes concepts from logics. The selected entity can then be saved in the ANSYS database as a "component." Thus, the next time this group of entities needs to be selected, selection of the component is sufficient. Selecting operations are discussed first, and the components next.

4.8.1 Selecting Operations

In ANSYS, entities are stored in separate sets (e.g., a set of areas, a set of volumes, etc.). Initially all of the full sets are active, until a selection operation is performed.

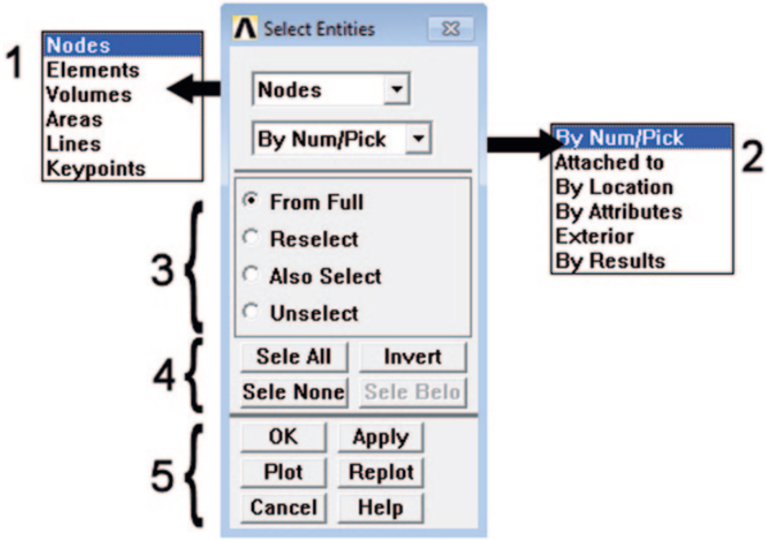


Fig. 4.65 *Select entities* dialog box (by number and picking)

When individual entities in a full set are selected (subset), they become active. Entity sets are independent of each other, i.e., selecting a group of lines does not cause any change in the selection status of the sets of keypoints or areas. Selections can be made based on several criteria, as explained below.

Selections can be performed by using the following menu path:

Utility Menu > Select > Entities

which brings up the *Select Entities* dialog box, as shown in Fig. 4.65. This dialog box has five distinct fields:

1. *Entity Field*: The entity to be selected is chosen using this pull-down menu.
2. *Criterion Field*: The entity chosen in the *Entity Field* is selected based on the criterion chosen in this pull-down menu. The following criteria are possible:

By Num/Pick: Clicking ***OK*** after choosing this criterion starts the *Pick Menu* and the entities are selected by picking.

Attached to: As discussed previously, the entities in ANSYS are associated with each other. For example, a line is composed of at least two keypoints; an area is made up of at least three lines; etc. Thus, keypoints and lines, and lines and areas, are mutually attached (the list may be extended).When this criterion is chosen, another field appears in the *Select Entities* dialog box, listing the possibilities for attachment.If ***Areas*** in the *Entity Field* and ***Attached to*** in the *Criterion Field* are chosen, the new field lists ***Lines*** and ***Volumes*** as possible attachments; choosing ***Volumes*** and clicking on ***OK*** results in the selection of volumes that are attached to the currently selected areas.

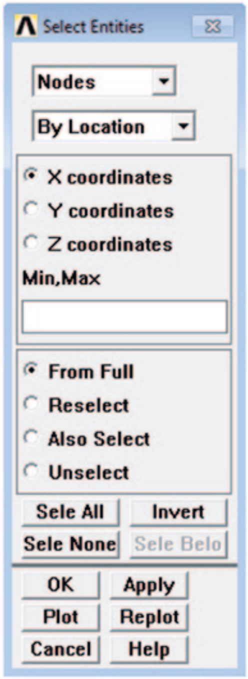


Fig. 4.66 *Select entities* dialog box (by location)

By Location: Selects entities based on their location. Upon choosing this criterion, a new field appears in the *Select Entities* dialog box with radio-buttons for x-, y-, and z-coordinates and a text field for the minimum and maximum values for the coordinate (shown in Fig. 4.66). For example, in order to select the nodes located between $y=2$ and $y=5$, the radio-button for the y-coordinate is activated and the expression "***2***, ***5***" (without the quotation marks) is entered in the text field.

By Attributes: This criterion is used for selecting entities based on their attributes (element type, material, real constant, etc.).

Exterior: Using this criterion, entities along the outer boundaries of the model are selected.

By Results: If a solution is obtained, then entities (only nodes and elements) can be selected based on result values.

3. *Domain Field*: This field determines the domain of the entity set with which the criterion is applied, as explained below:

From Full: Selection is made from the full set of entities regardless of the selection status of the particular entity set.

Reselect: This option is used to refine the selection.It is used to select entities from a previously selected subset. For example, if the goal is to select all the nodes having coordinates $x=2$ and $y=3$ (both at the same time), then the nodes with the coordinate $x=2$ are first selected ***From Full*** set and, then, using ***Reselect*** button, the nodes with coordinate $y=3$ are selected from the previously selected subset of nodes with coordinate $x=2$.

Also Select: This option is used to expand the selection. It is used to add entities to the currently selected subset, based on a different criterion.

Unselect: This option is used to deactivate (unselect) a group of entities from the selected subset.

4. Domain Action Field:

Sele All**:** Selects the full set of a specific entity.

Invert**:** Inverts the selected set; active entities become inactive and vice versa.

Sele None**:** Unselects the full set of a specific entity; the active set becomes empty.

Sele Belo**:** Following the hierarchy of entities (i.e., volume is highest and node is lowest), this option selects the lower entities attached to the selected set of entities chosen in the Entity field.

5. Action Field:

OK: Applies the selection operation and closes the Select Entities dialog box.

Apply: Applies the selection operation; the Select Entities dialog box remains open for further selections.

Plot: Plots the currently selected set of a specific entity.

Replot: Updates the plot.

Cancel: Closes the Select Entities dialog box without applying the selection operation.

Help: Displays the help pages related to selection operations.

In order to select “everything” (reset all entities to their full sets), the following menu path is used:

Utility Menu > Select > Everything

4.8.2 Components

Groups of selected entities can be saved in an ANSYS database for easy retrieval. These groups are called components, and they can only contain entities of the same kind. The main advantage of defining components is to avoid multiple selection operations every time the user needs to select the same group of entities. The following menu path is used for defining components:

Utility Menu > Select > Comp/Assembly > Create Component

which is followed by a dialog box requesting the name to be given to the component and the type of entity to include in the component. Upon clicking ***OK***, the component is created using the currently selected subset of the entity type chosen. The following menu path is used when a component has to be selected:

Utility Menu > Select > Comp/Assembly > Select Comp/Assembly

Listing and deletion of components is performed by using the following menu paths:

Utility Menu > Select > Comp/Assembly > List Comp/Assembly

Utility Menu > Select > Comp/Assembly > Delete Comp/Assembly