Chapter 5

Options Aiding Construction of Parts-I

Learning Objectives

After completing this chapter you will be able to:

- *Create holes using the HOLE dialog box.*
- *Create Round, Chamfer, and Rib.*
- *Edit features.*
- *Redefine, Reroute, and Reorder features.*
- *Suppress and delete features.*
- *Modify features.*
- *Dynamically modify a feature.*
- *Use Selections dialog box.*

OPTIONS AIDING CONSTRUCTION OF PARTS

This chapter deals with the options provided in Pro/ENGINEER that help in creating a model and editing it once the solid model is completed. In this chapter you will learn to create different types of holes that are required in most of the engineering drawings. In previous chapters you have learned to create holes using extrude cut but in this chapter you will create holes using the **Hole** dialog box. Using the **Hole** dialog box, it becomes easy to create holes as well as to modify them.

In this chapter you will also learn to create rounds, chamfers, and ribs. All the options that are discussed in this chapter increases the efficiency in creating a design using Pro/ENGINEER.

The HOLE dialog box

In Pro/ENGINEER holes are created using the **HOLE** dialog box. The **HOLE** dialog box is displayed when you choose **Insert** > **Hole** from the menu bar or **PART** > **Feature** > **Create** > **Solid** > **Hole** from the **Menu Manager**. You can create three types of holes using the **HOLE** dialog box. The first type is a straight hole, the second is a sketched hole, and the third is a standard hole.

Straight hole

Straight holes are the holes that have a circular cross-section having a constant diameter throughout the depth. They start at the placement plane and terminate at the user-defined depth or at the specified end surface. The **HOLE** dialog box with **Straight hole** radio button selected is shown in Figure 5-1. The different options available in this dialog box are discussed next.

Hole Dimension area

The **Hole Dimension** area has all the options that define the dimensions for the hole. The options in this area are discussed next.

Diameter. The **Diameter** edit box is used to enter the diameter value for the hole to be created.

Depth One. The **Depth One** option is used to specify the depth of the hole in one direction. The direction for this depth is shown by a single red colored arrow that is displayed on the plane selected for placing the hole. By default the **Variable** option is selected in this drop-down list. The other options in this drop-down list are shown in Figure 5-2. These options are similar to those discussed in the **SPEC TO** menu in Chapter 3.

Depth Two. The **Depth Two** option is used to specify the depth of the hole in the direction opposite to that of **Depth One**. This means the hole can be created on both sides of the sketching or the placement plane. The direction for this depth is shown by a yellow colored double arrow on the plane selected for placing the hole. By default, the **None** option is selected in this drop-down list. The other options in this list are shown in Figure 5-3.

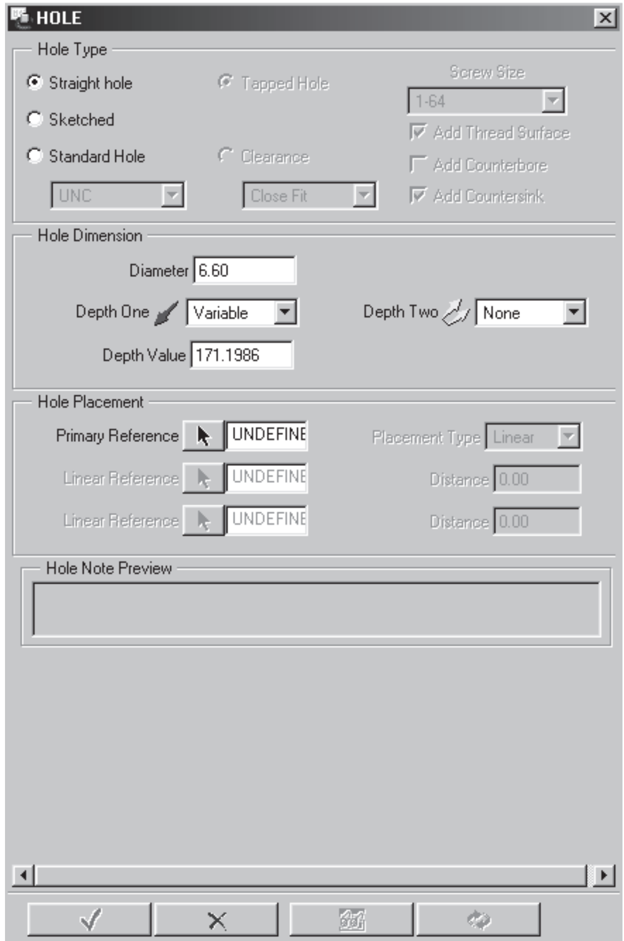


*Figure 5-1 **HOLE** dialog box with the **Straight hole** option*

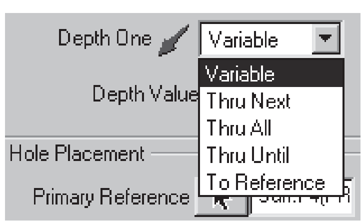


*Figure 5-2 **Depth One** drop-down list*

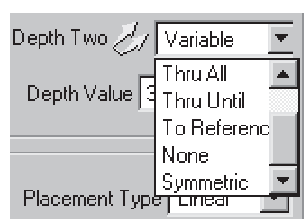


*Figure 5-3 **Depth Two** drop-down list*

Depth Value. The **Depth Value** edit box is used to enter the depth value for the hole. This edit box is displayed only when you choose the **Variable** option from the **Depth One** drop-down list or the **Depth Two** drop-down list.

Hole Placement area

In the **Hole Placement** area of the **HOLE** dialog box all the parameters that will define the placement of a hole are specified. The options in this area are discussed next.

Primary Reference. The **Primary Reference** option is used to select a reference with respect to which the hole will be placed. The primary reference can be a plane, a cylinder, a cone, an axis, or a point. The primary reference selected is used to specify the location of hole placement. When you choose the arrow button adjacent to this option, the **GET SELECT** menu is displayed. You can use the **Query Select** option from the **GET SELECT** menu to make a selection on the model.

Placement Type. The **Placement Type** drop-down list is shown in Figure 5-4. The options in this drop-down list are discussed next.

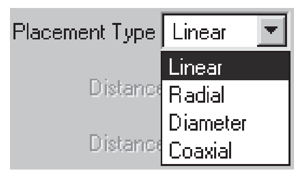


***Figure 5-4** **Placement Type** drop-down list*

Linear. When you select this option, you are prompted to specify the distances from two linear references. Generally, these linear references are the edges of the planar surface on the model, any two planar surfaces or axes, or a combination of any of these. Figure 5-5 shows the linear hole on a plane.

Radial. This option is used to create a hole that can be referenced to an axis. When you select this option, you are prompted to select an axial reference and an angular reference to place the hole. The distance from the axis is entered in the **Distance** edit box and angle is entered in the **Angle** edit box that is displayed when you select the axis and the plane for the angular reference. This option is usually used to create holes on flanges. Figure 5-6 shows a radial hole on a plane.

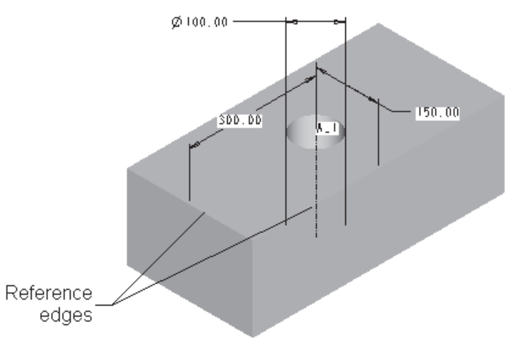


***Figure 5-5** Linear dimensioning of hole*

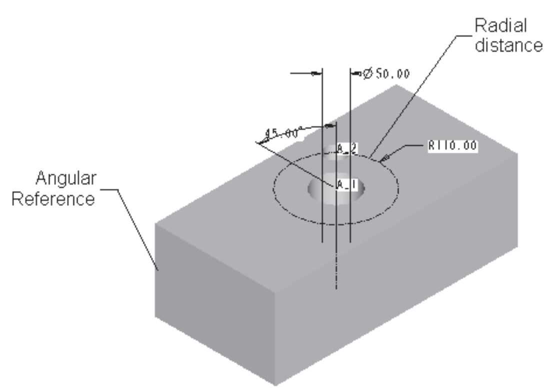


***Figure 5-6** Radial dimensioning of hole*

Diameter. This option creates a diametrically placed hole. When you select this option, you are prompted to select an axial reference and an angular reference to place the hole. Figure 5-7 shows a diameter hole on a plane.

Coaxial. This option creates a hole coaxially. When you select this option, you are prompted to select an axis. No dimensions are required to place a coaxial hole. Figure 5-8 shows a coaxial hole on a plane.

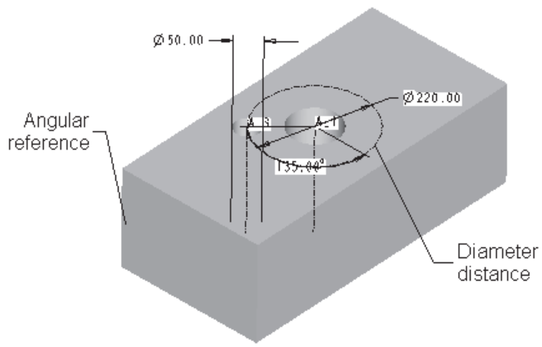


Figure 5-7 *Diameter dimensioning of hole*

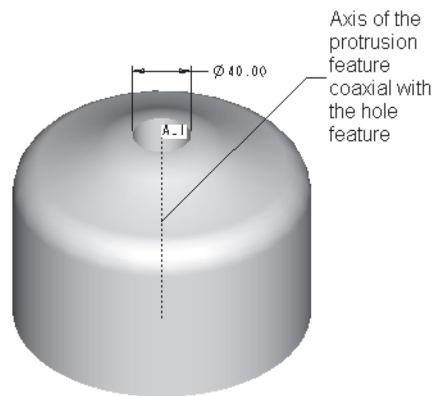
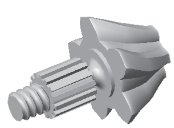


Figure 5-8 *Coaxial hole*

Sketched hole

The **Sketched** option allows you to sketch the cross-section for the hole that is revolved about a center axis. This option is used to draw custom shapes for the hole. When you choose this radio button in the **HOLE** dialog box, the system opens a new window with the sketcher environment. The cross-section for the hole is sketched using the normal sketcher options available. While drawing the sketch, a center line must be drawn that acts as the axis of revolution for the section of hole. The sketched holes can be a blind or a through hole depending upon the dimensions of the section sketch.

When you complete the sketch for the hole and choose the **Continue with the current section.** button, the window is closed and you are returned to the **HOLE** dialog box. You will be prompted to select the placement type for the sketched hole. The placement options are the same as discussed earlier.



Tip: *Remember that while placing any hole using the **HOLE** dialog box, you have to define two steps. First is the placement plane on which the hole feature will be created and the second is the dimensional references for all holes other than the Co-axial and On Point hole.*

Standard Hole

The holes created using the **Standard Hole** option are based on industry standard fastener tables. The **Standard Hole** option allows you to create two types of holes, **Tapped** holes and **Clearance** holes. In the **Tapped** holes, the cosmetic thread is included in the hole, whereas in the **Clearance** holes, the cosmetic threads are not included.

Figure 5-9 shows the **HOLE** dialog box with the **Tapped Hole** radio button and **Add Counterbore** check box selected. In the **Hole Dimension** area of the **HOLE** dialog box, the

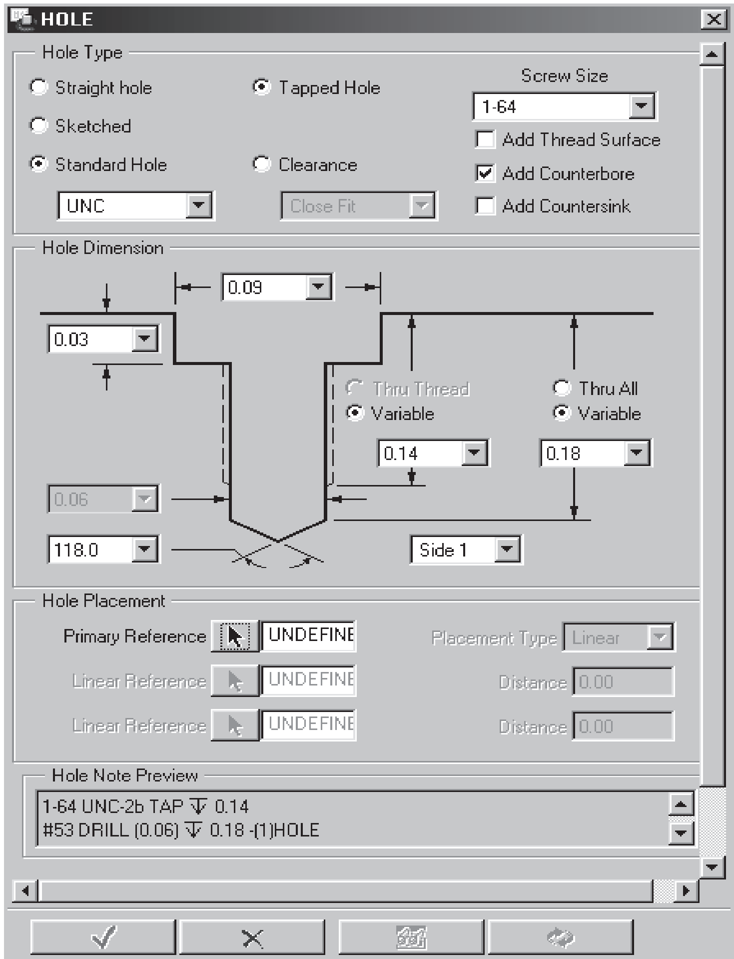


Figure 5-9 *The **HOLE** dialog box with **Standard Hole** option*

preview of the sketch for the cross-section of the counterbore hole is displayed with various dimensions. A counterbore hole is a stepped hole and has two diameters, a larger one and a smaller one. The larger diameter is called counter diameter and the smaller diameter is called drill diameter. In the preview of the sketch, the dimensions can be edited as required.

Figure 5-10 shows the **HOLE** dialog box with the **Clearance** radio button and **Add Countersink** check box selected. In the **Hole Dimension** area of the **HOLE** dialog box, the preview of the sketch for the cross-section of the countersink hole is displayed with various dimensions. A countersink hole has two diameters but the transition between the bigger diameter and the smaller diameter is in the form of a tapered cone. In the preview of the sketch, the dimensions can be edited as required.

Preview the Hole

The **Preview feature geometry** button is used to preview the hole created using

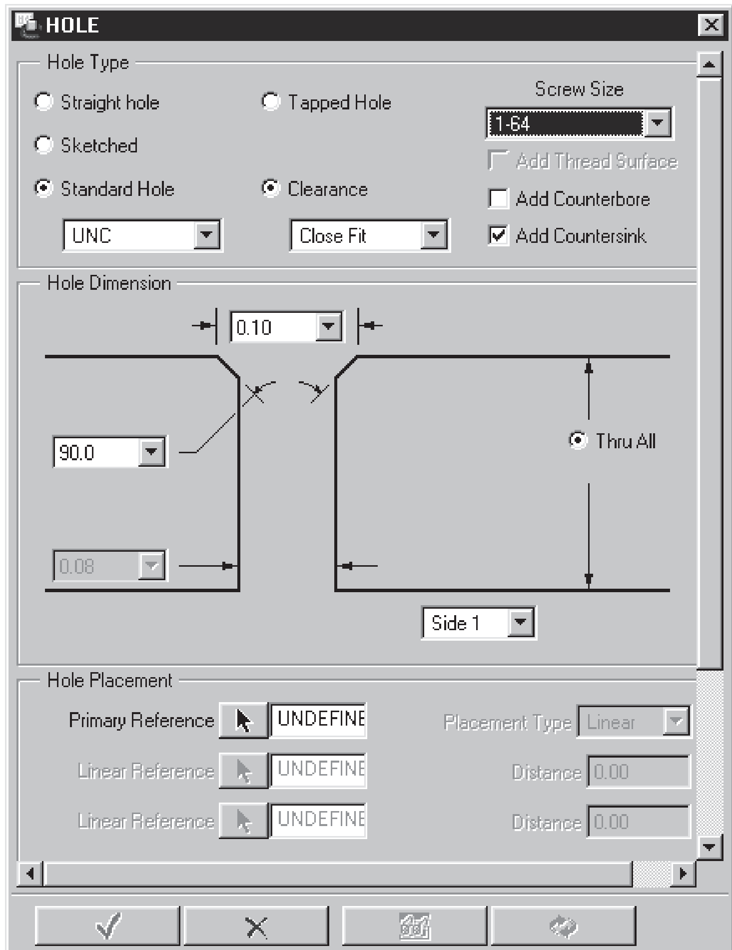


__Figure 5-10__ The __HOLE__ dialog box with __Standard Hole__ option

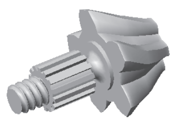
the **HOLE** dialog box before confirming its creation. This button is provide at the bottom of the **HOLE** dialog box. Changes and modifications in the hole parameters can be made easily once the hole is previewed. While previewing the hole, it is recommended to use the **Model Display** toolbar to change the model display.

Repeating the Hole feature

The **Build feature and repeat the same feature type creation.** button is used to confirm the hole feature creation and at the same time invoke the **HOLE** dialog box again to create the next hole feature. This button is generally used when you want to create more than one hole.

Note

The holes created using the __HOLE__ dialog box are parametric in nature and hence can be modified at anytime using the __Model Tree__. The method of modification using the __Model Tree__ is discussed later in this chapter.



Tip: *Rounds and chamfers are used in components to reduce the stress concentration at the sharp corners. Hence, they reduce the chances of failure of a component under a specified load condition..*

ROUNDS

The **Round** option in the **Insert** menu creates a fillet or smooth rounded transition with either a circular or a conic profile between two adjacent surfaces. This option can be invoked from the menu bar or from the **Menu Manager**. The **Round** option can either add or remove material, depending on the edge references selected. There are two types of rounds, as shown in Figure 5-11, that can be created in Pro/ENGINEER. The **ROUND TYPE** menu is displayed when you choose **PART** > **Feature** > **Create** > **Solid** > **Round** from the **Menu Manager** or **Insert** > **Round** from the menu bar.

***Figure 5-11 ROUND TYPE** menu*

Simple and Advanced Rounds

In Pro/ENGINEER, you can create two different types of rounds, simple and advanced. The type of round you create depends on your need to customize the default round geometry created. Both types of rounds need placement references.

After you specify placement references and the radius of the round, the system generates the default round geometry by using some default attributes like the round shape, cross section, and so on. You can preview the round using the **Preview** button from the **ROUND** dialog box. A simple round uses a circular cross-section and rolling shapes.

An advanced round allows you to define round sets between which the default transitions are created. These transitions can be modified later.

The options that are available to define the type of round and that help to select the geometric references are shown in Figure 5-12. These options are available in the **RND SET ATTR** (**ROUND SET ATTRIBUTE**) menu that is displayed when you choose **Simple** from the **ROUND TYPE** menu.

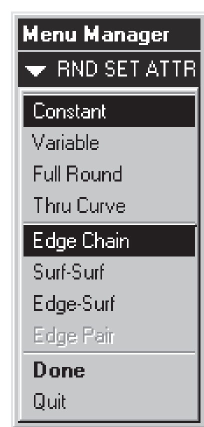


***Figure 5-12 RND SET ATTR** menu*

The options used to specify the type of round are discussed next.

Constant

The **Constant** option with the **Edge Chain** option is selected by default when the **RND SET ATTR** menu is displayed. This option creates a round by assigning the same radius value to every selected edge or surface. Figure 5-13 shows the constant round. This option is used when the selected edge require the same radii throughout.

Variable

The **Variable** option allows you to specify different radii at the end points of the selected edge or optionally at additional points along the selected edge. Additional points can be defined along the edge. But for this purpose, these points should be datum points and should lie along the selected edge. Figure 5-14 shows the variable round. This option is used when the selected edge(s) require variation in radius along it.

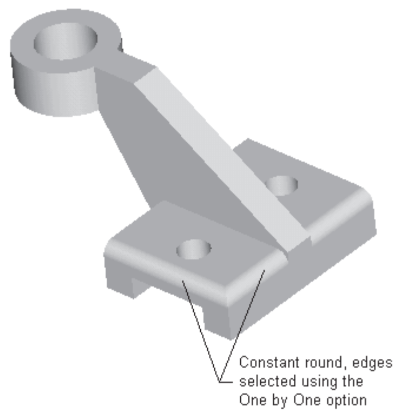


Figure 5-13 *Constant radius round using* **Edge Chain** *option*

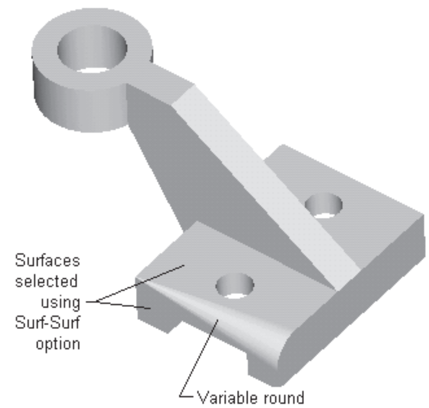


Figure 5-14 *Variable radius round using* **Surf-Surf** *option*

Full Round

The **Full Round** option creates a complete round between two selected edges or two planar or non planar surfaces. Figure 5-15 shows the round created using the **Full Round** option between the edge on the upper and the lower face of the base of the model.

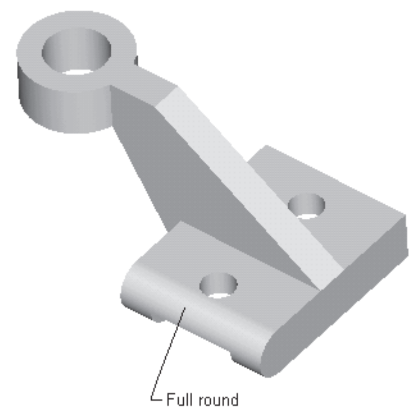


Figure 5-15 *Full round using* **Edge Pair** *option*

The following options are used to specify the references. These options are available in the **RND SET ATTR** menu.

Thru Curve

When you use the **Thru Curve** option the round is created between two surfaces in which one of the tangent edges follows a curve.

Edge Chain

The **Edge Chain** option allows you to select a chain of edges using the options from the **CHAIN** menu. This option is not available when you choose the **Full Round** type of round. The **CHAIN** menu is displayed when you choose **Done** from the **RND SET ATTR** menu. You can use the **Query Select** option from the **GET SELECT** menu to select edge(s). The round is created on the selected edge that joins the two surfaces. Figure 5-13 shows the round created using this option.

CHAIN menu

The **CHAIN** menu is displayed when you choose **Edge Chain** option from the **RND SET ATTR** menu and choose **Done**. The **CHAIN** menu is displayed with the **GET SELECT** menu as shown in Figure 5-16. The options available in the **CHAIN** menu are discussed next.

***Figure 5-16 CHAIN** menu with the **GET SELECT** menu*

One by One. The **One by One** option allows you to select individual edges or curves, one at a time. The **Query Select** option can be used to select the edges or curves. When you select the edges or curves, they are highlighted in blue one by one. After selecting all the edges or curves confirm the selection by choosing **Done Sel** from the **GET SELECT** submenu. All the selected edges will change to cyan.

Tangent Chain. The **Tangent Chain** option is used to select all the edges tangent to the selected edge. Instead of using the **One by One** option, you can use this option to select all the edges that are tangent to the selected edge.

Surf Chain. The **Surf Chain** option allows you to select the chain of edges of the selected surface. When you select a surface on the model, the **CHAIN OPT** submenu is displayed as shown in Figure 5-17. This menu has two options.

***Figure 5-17 CHAIN OPT** submenu*

Select All. When you choose this option, the system selects all the edges of the selected surface.

From-To. When you choose this option, the system displays green colored points called vertices on the selected surface edges. The system prompts you to select from and to points to specify a loop. The selected loop is highlighted in blue.

To select the loop for making the round, the **CHOOSE** submenu is displayed as shown in Figure 5-18.

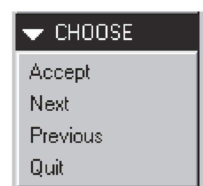


***Figure 5-18 CHOOSE** menu*

Intent Chain. The **Intent Chain** option is used to select all the tangent or non-tangent edges that form a chain with the selected edge.

Unselect. The **Unselect** option is used to unselect the selections made using the other options from the **CHAIN** menu.

Surf-Surf

The **Surf-Surf** option is used to create a fillet between two planar or non-planar surfaces by defining the radius of the fillet. When you select the **Surf-Surf** option and choose **Done** from the **RND SET ATTR** menu, you are prompted to select the first surface for round or fillet. After selecting the first surface you are prompted to select the second surface. Specify the radius of fillet in the **Message Input Window** and the round is created between the two selected surfaces and is tangent to the selected surfaces. Figure 5-14 shows the round created using this option.

Edge-Surf

The **Edge-Surf** option creates a round by defining a chain of edges and surface. The round created is tangent to the selected surface only and not to the edges. This option is not available when you choose the **Thru Curve** type of round.

Edge Pair

The **Edge Pair** option is available only when you choose to create the **Full Round** type of round. This option is used when you have to create a semicircular fillet between two surfaces by selecting its edges. Figure 5-15 shows the round created using this option.

Note

You can create advanced rounds using the different options in the same way as simple rounds are created. For advanced rounds you need to define the round sets.

CHAMFERS

Chamfers are used to bevel the selected edges and corners with the help of some specified parameters. The **Chamfer** option can be invoked from the menu bar or from the **Menu Manager**. Pro/ENGINEER creates two type of chamfers. The first is the **Edge** chamfer and the second is the **Corner** chamfer.

When you choose **Insert** > **Chamfer** from the menu bar or **PART** > **Feature** > **Create** > **Solid** > **Chamfer** from the **Menu Manager**, you have two options. These options are **Edge** and **Corner**. Figure 5-19 shows the two types of chamfers.

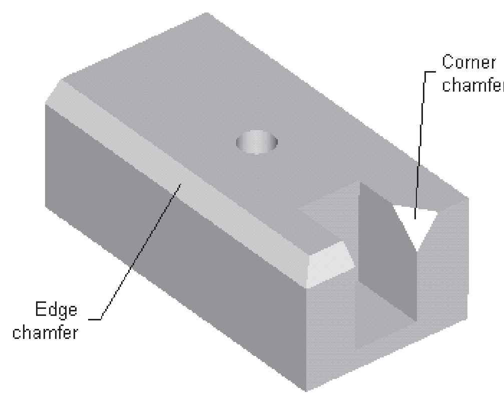


Figure 5-19 *Different types of chamfers*

Edge Chamfer

An **Edge** chamfer creates a beveled surface along the selected edge. When you choose the **Edge** option, the **SCHEME** menu is displayed as shown in Figure 5-20. The options in this menu are discussed next.

45 x d

The **45 x d** option creates a chamfer at the intersection of two perpendicular surfaces. The chamfer created is at an angle of 45-degree from both the surfaces and at a distance **d** from the edge along each surface. Once the chamfer is created, the distance d can be modified.

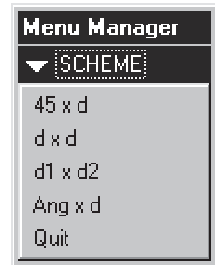


*Figure 5-20 The **SCHEME** menu*

d x d
The **d x d** option creates a chamfer such that the distance of the selected edge is equal from both the faces.

d1 x d2
The **d1 x d2** option creates a chamfer at two user-defined distances from the selected edge. When you invoke this option, you will be prompted to select the reference surface. This is the surface along which the first distance will be measured from the selected edge.

Ang x d
The **Ang x d** option creates a chamfer at a user-defined distance from the selected edge and at a user-defined angle measured from a specified surface.

Corner Chamfer
A **Corner** chamfer creates a beveled surface at the intersection of three edges. When you choose this option, the **GET SELECT** menu is displayed and you are prompted to select a corner that has to be chamfered. When you select the corner, the **PICK/ENTER** menu is displayed as shown in Figure 5-21. Use the options in the **PICK/ENTER** menu to specify the chamfer distance. The options in this menu are discussed next.

*Figure 5-21 The **PICK/ENTER** menu*

Pick Point
The **Pick Point** option is used to select a point on the highlighted edge. The point selected on the edge denotes the chamfer distance from the corner. After you have specified the point on the highlighted edge, the next edge will be highlighted.

Enter-input
When you use the **Enter-input** option, the **Message Input Window** is displayed and you are prompted to enter the chamfer distance for the highlighted edge.

After you enter the distance value for all the three edges the corner chamfer is created.

RIBS
Ribs are defined as thin wall-like structures used to bind the joints together so that they do not fail under an increased load. In Pro/ENGINEER, the section for the rib is sketched as an open section and is always extruded equally in both the directions of the sketch plane. The procedure of creating a rib is similar to that of creating a protrusion.

In Pro/ENGINEER, you can create two types of ribs: Rotational ribs and Straight ribs. Rotational ribs are constructed on cylindrical parts and straight ribs are created on planar faces. There are no separate options available for the creation of these ribs in Pro/ENGINEER.

The creation of these of ribs depends on the geometry of the base feature. Figure 5-22 shows a rotational rib and Figure 5-23 shows a straight rib.

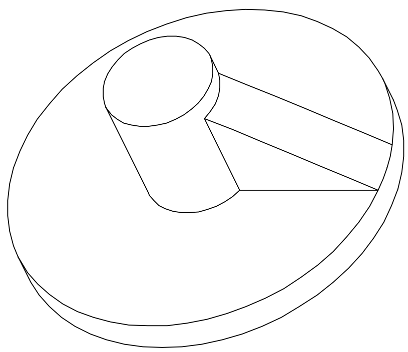


Figure 5-22 Rotational *Rib feature*

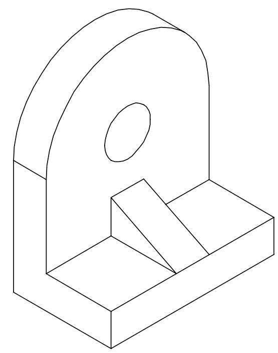


Figure 5-23 *Straight Rib feature*

Note

Since the ribs are always extruded on both sides of the sketching plane, therefore, while creating a rib you must always select an appropriate sketching plane such that it lies at the center of rib creation.

EDITING THE FEATURES OF A MODEL

Editing is one of the most important aspects of product designing. Most of the designs require editing either during their creation or after they are created. As mentioned earlier, Pro/ENGINEER is a parametric solid modeling tool. Hence, the features that together constitute a model can be individually edited. For example, Figure 5-24 shows a cylindrical part that has six counterbore holes created at some bolt circle diameter (BCD).

Now, in case you have to edit the features such that the six holes are to be converted into eight holes and the counterbore holes are to be converted into countersink holes, as shown in Figure 5-25, all you need to do is to use two operations. The first operation will convert the six holes into eight holes by using the **Modify** option in the **PART** menu. The second operation will open the **HOLE** dialog box and convert the counterbore holes into countersink holes.

Similarly, you can also edit the datums and the features referenced to these datums. Since there exists a parent-child relationship between the two, therefore, the child feature is also modified when the parent feature is modified. For example, if you have created a feature using a datum plane that is at some offset distance, the feature will be automatically repositioned when the offset value of the datum plane is changed. The following methods explain how to edit the features in Pro/ENGINEER.

Redefining Features

Redefining features allows you to make changes in the parameters that were used to create a feature. You can also modify the sketches of the sketched features by redefining it. A feature is

Figure 5-24 *Six counterbore holes*

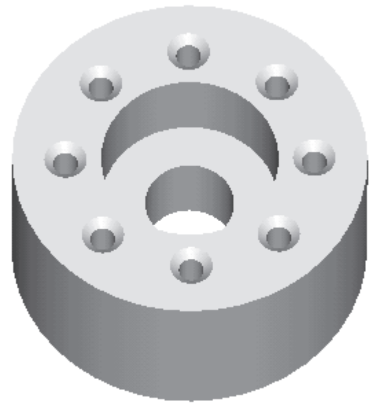


Figure 5-25 *Eight countersink holes*

redefined only when it is completed. To invoke the **Redefine** option there are three methods available in Pro/ENGINEER. These methods are listed next.

- Choose **PART** > **Feature** > **Redefine** from the **Menu Manager**. You will be prompted to select the feature that has to be redefined from the graphics screen. Select the feature using the left mouse button. The selected feature is highlighted in red.

- You can select the feature that has to be redefined from the graphics screen. The selected feature is highlighted in red. Now, hold down the right mouse button. A shortcut menu is displayed; choose **Redefine** from this menu.

- You can also use the **Model Tree** to redefine a feature. Select the feature that has to be redefined from the **Model Tree**. The selected feature is highlighted in red on the graphics screen. Now, right-click on the feature listed in the **Model Tree**. A shortcut menu is displayed; choose **Redefine** from this menu.

There are two types of features that are created in Pro/ENGINEER. Features that have elements and features that do not have elements. Features that do not have elements are ribs, etc. When you redefine a feature that does not have elements, then the **REDEFINE** menu is displayed. And when you redefine a feature that has elements, then the **Feature Creation** dialog box is displayed.

Procedure to redefine a feature with elements

Choose **PART** > **Feature** > **Redefine** and select a feature that has elements. The **Feature Creation** dialog box is displayed with all the elements and their definitions. You can select any element and choose **Define** from this dialog box. The menu corresponding to the element will be displayed and now you can redefine that element. The **Feature Creation** dialog box is discussed in detail in Chapter 3.

Procedure to redefine a feature without elements

Choose **PART** > **Feature** > **Redefine** and select a feature that do not have elements, for example, a rib. The **REDEFINE** menu is displayed. This menu has check boxes in front of each parameter. Select the parameters that have to be redefined and choose **Done**. The menu corresponding to the selected parameter is displayed and now you can redefine that parameter.

Reordering

Reordering the features is defined as the process of changing the order of features in a model. Sometimes, after creating a model it may be required to change the order in which the features of the model were created. A feature can be placed before or after another feature. For this purpose either the **Model Tree** or the **Menu Manager** is used.

If you use the **Model Tree**, you just have to drag and drop the feature you want to reorder below or above the destination feature.

If you use the **Menu Manager**, the **Reorder** option is available in **PART** > **Feature** > **Reorder**. When you choose the **Reorder** option from the **FEAT** menu, the **SELECT FEAT** submenu is displayed along with the **GET SELECT** submenu as shown in Figure 5-26 and you are prompted to select the features that you want to reorder. Select the feature to be reordered either from the **Model Tree** or from the graphics screen and then choose **Done Sel** from the **GET SELECT** submenu or use the middle mouse button to confirm the selection. In the **Message** area, the system will display the possible positions where the selected feature can be inserted. The features are numbered according to their occurrence in the **Model Tree**.

***Figure 5-26 SELECT FEAT** submenu*

The example that is discussed here will explain why and when the need for reordering of features in a model arises. Consider the model shown in Figure 5-27. It consists of a rectangular pattern of nine columns and four rows. Now, a shell feature is created on this model that removes the top and the front face as shown in Figure 5-28.

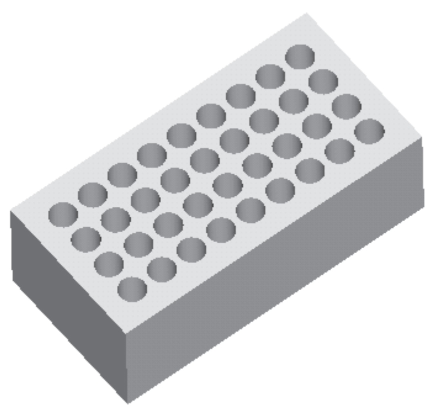


***Figure 5-27** Model with pattern*

Figure 5-28 shows that the material equal to the wall thickness of the shell feature is added around the hole features. This was not desired, and here the need for reordering the feature arises. You will reorder the features such that the shell feature is inserted before the hole feature and its pattern. When you reorder the features, all the features will be automatically adjusted in the new order as shown in Figure 5-29.

Rerouting

The **Reroute** option available in the **FEAT** menu is used to modify the references of a feature

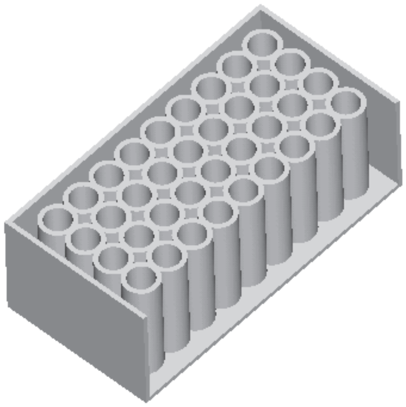


Figure 5-28 Model after creating the shell feature

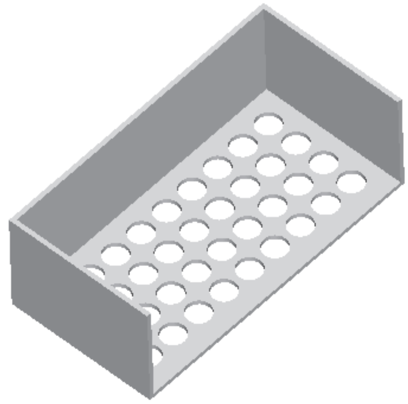


Figure 5-29 Model after reordering the features

and in turn break the parent child relation that exists between the selected feature and the other features.

Suppressing

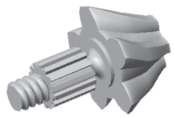
When you do not want a feature to be displayed on the graphics screen or to show up in the drawing views of a model then that feature can be suppressed. Once the feature is suppressed, it will neither be displayed in the drawing views nor on the graphics screen. Note that the feature is not deleted, only its visibility is turned off. You can anytime resume the feature by unsuppressing it using the **Model Tree** or using the **Menu Manager.** As soon as you unsuppress the feature, it will be displayed on the graphics screen as well as in the drawing views. When a model has many features then the suppressing of features decreases the regeneration time of the new feature created. To suppress a feature right-click on it in the **Model Tree** or select it on the graphics screen using the left mouse button and press and hold down the right mouse button to display the shortcut menu. Choose **Suppress** from the shortcut menu to suppress the selected feature. You can also suppress a feature using the **Menu Manager**.

Note

If the feature that is suppressed has some child features then they will also be suppressed. When you select such a feature, the system prompts you to confirm the suppression of the highlighted features on the graphics screen.

Deleting a feature

The feature that is not required can be deleted from the model. Right-click on the feature in the **Model Tree** or select it on the graphics screen and press and hold down the right mouse button to display the shortcut menu. From this menu choose the **Delete** option. The feature to be deleted is highlighted in red. If the feature to be deleted has some child features, they will also be highlighted. The system confirms before deleting the selected feature. If you confirm the deletion, the feature along with its child features will be deleted.



Tip: *By default, the suppressed features are not displayed in the* ***Model Tree****. To display the suppressed features, choose* ***View > Model Tree Setup > Item Display*** *from the menu bar. The* ***Model Tree Items*** *dialog box is displayed. Select the* ***Suppressed Objects*** *check box and choose* ***OK*** *from the* ***Model Tree Items*** *dialog box. The suppressed features will now be displayed in the* ***Model Tree****.*

The suppressed features can also be deleted by using the ***Model Tree****.*

Modifying a feature

Once a feature is created, you can still modify the feature by modifying its dimensions. This editing operation reflects the parametric nature of Pro/ENGINEER. The **Modify** option is available in the **PART** menu. When you choose the **Modify** option, the system prompts you to select the feature that is to be modified. Select the feature on the graphics screen using the left mouse button. The selected feature is highlighted in red and the dimensions appear on the feature. These dimensions are the same that were defined while sketching or creating the feature. In some cases, for example, rounds, holes, and so on, the dimensions of these features were defined as parameters. Hence, these dimensions are also displayed when you select them to modify. To modify the dimensions, select the dimension from the graphics screen. When the **Message Input Window** appears, enter the required value for the selected dimension and press ENTER. Once you modify a dimension, you also need to regenerate the feature. The **Regenerate** option is also available in the **PART** menu.

If you want to modify a pattern, it is recommended that you select the instance other than the original one. This is because if you select the original feature, the incremental dimensions are not displayed and so you cannot modify the incremental values of the pattern.

Dynamic Modification of a Feature

With this release of Pro/ENGINEER, you are allowed to dynamically modify extruded, revolved, rounded features that have a variable value assigned. For example, the depth of extrusion in case of extruded features, the angle of revolution in case of revolved features, and the radius in case of rounds. The child features, if any, will also be modified when the parent feature is modified.

To modify an extruded feature dynamically, select the feature from the **Model Tree** or from the graphics screen and hold down the right mouse button when the feature is highlighted in red. A shortcut menu is displayed. Choose the **Dynamic Modify** option from this menu. A yellow colored small box is displayed with an axis. Select the small box by holding down the left mouse button. The arrow cursor changes to a two-sided arrow and the yellow color of the box is changed to red. The two-sided arrow is displayed showing that the modification can be made on either side. Holding down the left mouse button, drag the mouse to the side on which the depth has to be modified. As you drag the mouse, the depth of extrusion is modified dynamically. You will notice that the system displays the preview of the feature before actually modifying it. After modifying the extrusion dynamically, release the left mouse button. To confirm the modification, left-click on the screen or press the middle mouse button. The feature will regenerate and is modified. Figure 5-30 shows the preview of the feature with the yellow box and the axis, and Figure 5-31 shows the feature after regeneration.

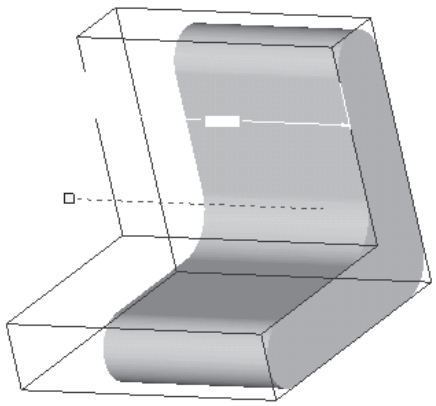


Figure 5-30 Dynamic modification of the feature

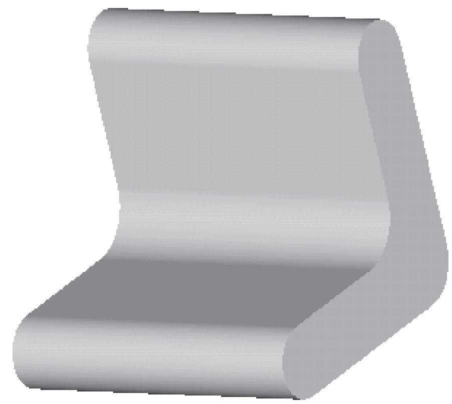


Figure 5-31 Feature after modification

Using Selections dialog box

The **Selections** dialog box is used when there are many features in a model and the feature you want to select is cluttered by other adjacent features. To display this dialog box select any feature in that portion of the model using the left mouse button and then right-click. Remember that instead of right-clicking if you hold the right mouse button down, the shortcut menu is displayed not the **Selections** dialog box. Figure 5-32 shows the **Selections** dialog box.

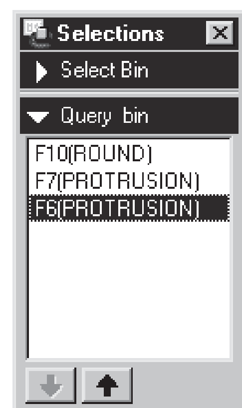


Figure 5-32 Selections *dialog box*

After the **Selections** dialog box is displayed, you can select the feature from this dialog box and perform any editing operation on it. The feature can be selected in this dialog box by either using the left mouse button or using the arrow buttons available in the dialog box. The selected feature in the dialog box is highlighted in red color on the graphics screen. To confirm the selection, use the middle mouse button.

The **Selections** dialog box is one of the methods to select a feature to edit. There are some other methods also to select a feature of a model. You can use the **Model Tree** of the model. You can also directly select the feature to be edited from the model using the left mouse button.

TUTORIALS

Tutorial 1

Create the model shown in Figure 5-33. The dimensions, the front, top, and left-side views of the model are also shown in the figure. **(Expected time: 45 min)**

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of eight features, see Figure 5-33.

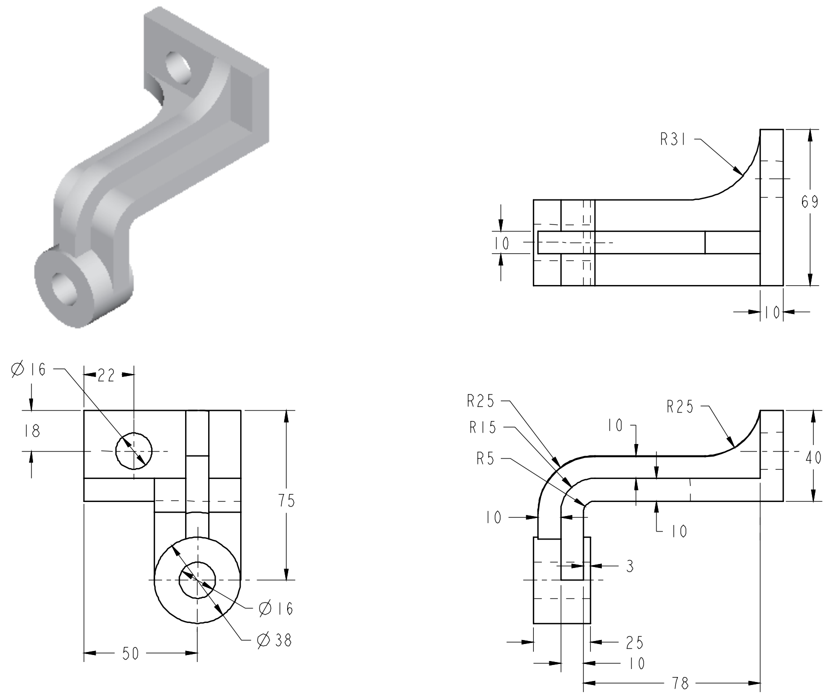


Figure 5-33 Isometric shaded view, left-side, front, and top views of the model

b. The first four features are extruded features, see Figure 5-42. First the sketch of base feature will be created on the **FRONT** datum plane, see Figure 5-34, and then it will be extruded to a depth of 10.

c. The sketch of the second feature will be created on the **TOP** datum plane and will be extruded on one side of the sketching plane. The depth of extrusion is 10, see Figure 5-37.

d. The sketch of the third feature will be created on the front planar surface of the second feature and will be extruded on one side of the sketching plane. The depth of extrusion is 10, see Figure 5-40.

e. The sketch of the cylindrical feature will be drawn on the front planar surface of the third feature and extrusion will be on both sides of the plane. The depth of extrusion on one side is 12 and on the other side is 13, see Figure 5-42.

f. The next feature is a hole that is coaxial to the cylindrical feature, see Figure 5-43. This hole will be created using the **HOLE** dialog box.

g. The next two features that will be created are rounds. The two rounds have different radii.

Hence, they will be created as two separate features.

h. The last feature is a rib. The sketch for this feature will be drawn on the **RIGHT** datum plane, see Figure 5-46.

After understanding the procedure for creating the model, you are now ready to create it. When Pro/ENGINEER session is started, the first task is to set the working directory. Since this is the first tutorial of this chapter, you need to create a folder named **c05**, if it does not exist. Choose the **New Directory** button in the **Select Working Directory** dialog box and create a directory named **c05** at **C:\ProE**.

Creating New Object File

1. Open a new part file and name it **c05tut1**. The three default datum planes will be displayed on the graphics screen. The **Model Tree** is also displayed on the graphics screen. Exit the **Model Tree** by choosing the **Model Tree on/off** button from the **Model Display** toolbar.

Creating the Base Feature

To create the sketch for the base feature, you need to first select the sketching plane for the base feature. In this model, you need to draw the base feature on the **FRONT** datum plane because direction of extrusion of this feature is perpendicular to this plane.

1. Choose **Insert > Protrusion > Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed. Choose **One Side > Done** from this menu.

2. Select the **FRONT** datum plane as the sketching plane. The **DIRECTION** submenu is displayed.

3. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

4. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

Note

Before you start sketching a section for any feature, it is recommended to turn the model display to ***No Hidden*** *and turn off the datum plane display using the* ***Datum planes on/off*** *button. This is done to improve the clarity of the graphics screen while sketching.*

When you are working on these tutorials, make extensive use of buttons available in the ***Datum Display*** *toolbar.*

5. Once you enter the sketcher environment, create the sketch of the base feature and apply constraints and dimensions as shown in Figure 5-34. Note that in the sketch, the bottom line of the rectangular section coincides with the **TOP** datum plane.

   As evident from the sketch of the base feature shown in Figure 5-34, the **RIGHT** datum plane is located at a dimension of 50 from the left edge because later in the tutorial, the rib feature will be created on this plane. The distance of 50 can be calculated from

Figure 5-33. As mentioned earlier, the sketch for a rib is by default extruded to both the sides of the sketching plane. Therefore, when you create the sketch for the rib feature on the **RIGHT** datum plane, it will be extruded on both the sides of the sketching plane.

6. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

   The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion.

7. The **Blind** option is selected by default in this menu, choose **Done**.

8. Enter the depth as **10** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

9. Choose the **Default** option from the **Saved view** drop-down list.

   The base feature is completed and now the second feature will be created.

Creating the Second Feature

The second feature is also an extruded feature and will be created on the **TOP** datum plane. Therefore, you need to define the **TOP** datum plane as the sketching plane.

1. Choose **Insert** > **Protrusion** > **Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed. Choose **One Side** > **Done** from this menu.

2. Select the **TOP** datum plane as the sketching plane. The **DIRECTION** submenu is displayed. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

3. Select the **Bottom** option from this submenu and choose the front surface of the base feature shown in Figure 5-35.

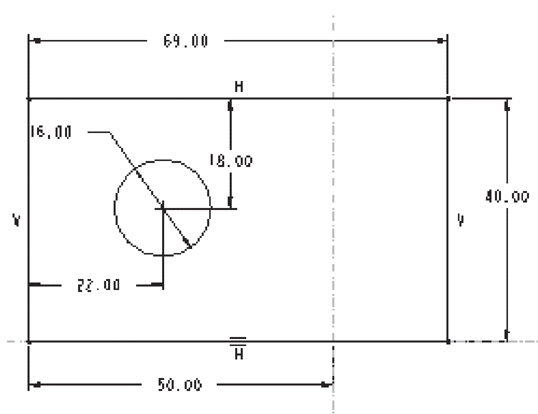


Figure 5-34 *Sketch with dimensions and constraints for the base feature*

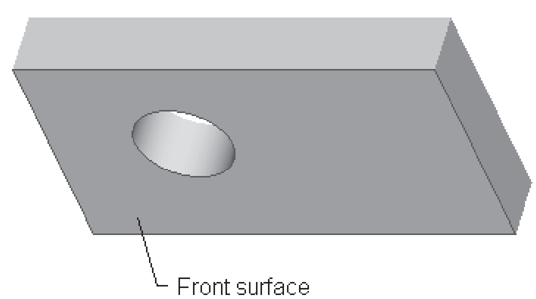


Figure 5-35 *Front planar surface selected to be at the bottom*

After entering the sketcher environment, turn the model display to **No Hidden**.

4. Create the sketch for the second feature and apply the constraints and dimensions as shown in Figure 5-36.

   You can use the **Create an entity from an edge.** button from the **Sketcher Tools** toolbar to use the edge of the base feature. The edge of the base feature is required to close the section for the second feature. Note that if you use the **Create an entity from an edge.** button, the 38 dimension in Figure 5-36 will not be required. However if you do not use the edge, you will have to draw a line to close the sketch. You will then have to align it with the bottom edge and add the constraint and dimensions as shown in Figure 5-36.

Note

*If you do not close the section loop by drawing a line or using the edge of the base feature, the **DIRECTION** menu will be displayed when you exit the sketcher environment. This menu will prompt you to specify the direction in which the material will be added. The material will be added in the direction shown by the arrow.*

5. After completing the sketch, turn the model display to **Shading** and choose the **Continue with the current. section** button. The **SPEC TO** menu is displayed.

6. The **Blind** option is selected in this menu, choose **Done**. The **Message Input Window** is displayed with a default value in it.

7. Enter a value of **10** in the **Message Input Window** and press ENTER. Choose the **OK** button from the **PROTRUSION** dialog box. The second feature is completed and the shaded default trimetric view is shown in Figure 5-37.

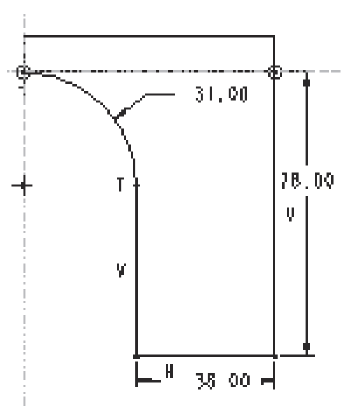


Figure 5-36 Sketch with dimensions and constraints for the second feature

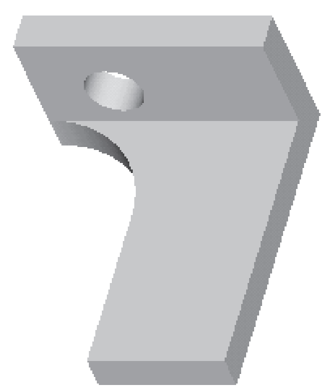


Figure 5-37 The default trimetric view of the completed second feature and the base feature

Creating the Third Feature

The sketch of the third feature will be drawn on the front planar surface of the second feature and will be extruded to the given depth.

1. Choose **Insert** > **Protrusion** > **Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed. Choose **One Side** > **Done** from this menu.

2. Select the planar surface of the second feature shown in Figure 5-38 as the sketching plane. The **DIRECTION** submenu is displayed. Choose **Okay** from this menu. The **SKET VIEW** submenu is displayed.

3. Select the **Top** option from this submenu and select the top surface of the second feature.

4. Once you enter the sketcher environment, turn the model display to **No Hidden**. Create the sketch for the third feature and apply constraints and dimensions as shown in Figure 5-39.

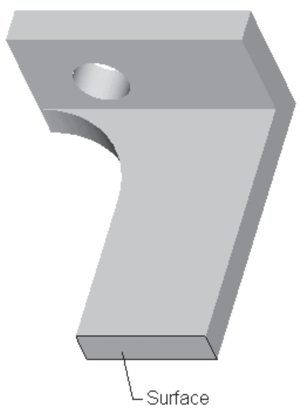


Figure 5-38 *Planar surface selected as the sketching plane for the third feature*

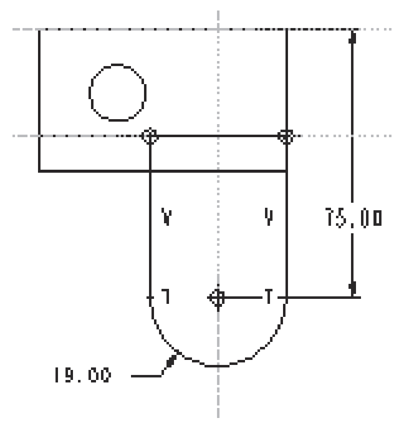


Figure 5-39 *Sketch with dimensions and constraints*

5. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed.

6. The **Blind** option is selected in the **SPEC TO** menu, choose **Done**. The **Message Input Window** is displayed with a default value in it.

7. Enter a value of **10** in the **Message Input Window** and press ENTER.

8. Choose the **OK** button from the **PROTRUSION** dialog box. Turn the model display to **Shading**. The third feature is completed. You can use the CTRL+middle mouse button to spin the model to view it from different directions.

Creating the Fourth Feature

The fourth feature of the model is an extruded feature and the sketch of this feature is drawn on the front planar surface of the third feature.

1. Choose **Insert** > **Protrusion** > **Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed. Choose **Both Sides** > **Done**.

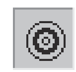
2. Select the planar surface shown in Figure 5-40 as the sketching plane. The extrusion of the cylindrical feature will be on both sides of the front planar surface. The **DIRECTION** submenu is displayed.

3. Choose **Okay** from this submenu. The **SKET VIEW** menu is displayed.

4. Select the **Top** option from this menu and choose the top planar surface of the second feature.

5. Once you enter the sketcher environment turn the model display to **No Hidden**.

6. Draw the circular section for the fourth feature as shown in Figure 5-41. Choose the **Create concentric circle.** button from the **Sketcher Tools** toolbar to draw the sketch for this feature. Select the arc using the left mouse button. As you drag the mouse the cyan colored rubber band circle changes its size. As you move the mouse cursor close to the arc, the cursor snaps to the arc. Use the left mouse button and select a point on the arc. You will notice that the equal radius constraint is applied to the sketch. Now, use the middle mouse button to abort this option.



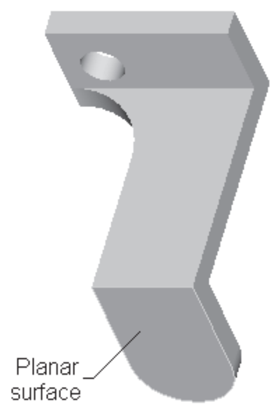


Figure 5-40 *Planar surface*

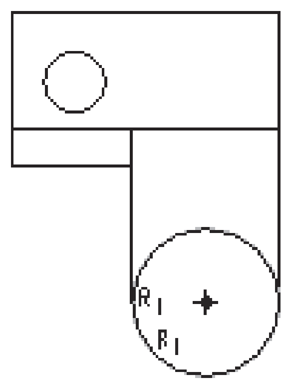


Figure 5-41 *Sketch for the fourth feature*

Note

While drawing the sketch for any feature, it is recommended to first apply the required constraints and then dimension the sketch.

7. After the sketch is completed, choose the **Continue with the current section.** button. The **SPEC FROM** menu is displayed.

   You will need to spin the model so that the direction of arrow on the graphics screen is clearly visible.

8. Select the **2 Side Blind** option from this menu and choose **Done**.

   The **2 Side Blind** option allows you to enter the depth of extrusion in two opposite directions. This option is available only when you choose the **Both Sides** option from the **ATTRIBUTES** menu. The **Both Sides** option was selected in step 1.

You are prompted to specify the depth of extrusion on the inside of the sketching plane.

9. Enter a value of 13 in the first direction shown by the red arrow and the value of 12 in the other direction. The fourth feature is completed.

10. Choose **OK** from the **PROTRUSION** dialog box.

Creating the Hole Feature

The hole feature will be created using the **HOLE** dialog box. The hole will be placed coaxial to the fourth cylindrical feature.

Note

Since the hole will be created coaxially, you will be required to select the axis of the circular feature. You may need to turn on the axis display if it is turned off. Choose the ***Datum axes on/off*** *button from the* ***Datum Display*** *toolbar to turn on the datum axis display.*

1. Choose **Insert > Hole** from the menu bar. The **HOLE** dialog box is displayed. In the **Hole Type** area, the **Straight hole** radio button is selected by default.

2. In the **Diameter** edit box of the **Hole Dimension** area, enter a value of **16**. The hole that will be created is of diameter 16.

3. From the **Depth One** drop-down list, select the **Thru All** option.

   The **None** option is selected by default in the **Depth Two** drop-down list. The **GET SELECT** menu is displayed and you are prompted to select a primary reference.

4. Select the front surface of the cylindrical feature shown in Figure 5-42 to place the hole.

   As you select the front planar surface of the cylindrical feature, the **GET SELECT** menu is displayed again. You are prompted to select the first reference for the hole placement.

5. From the **Placement Type** drop-down list, choose the **Coaxial** option. You are prompted to select the hole axis.

6. Select the axis of the cylindrical feature from the graphics screen. The hole feature is created and you can now preview it.

7. Choose the **Preview feature geometry** button from the **HOLE** dialog box. The hole created is highlighted in red.

8. Choose the **Build feature** button from the **HOLE** dialog box. The hole is created and the trimetric view of the shaded model with the hole is shown in Figure 5-43.

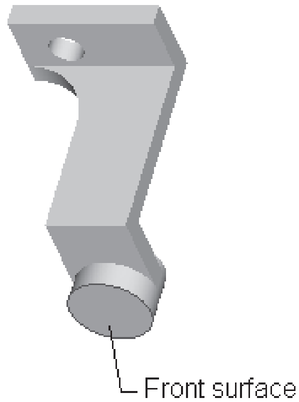


Figure 5-42 *Planar surface*

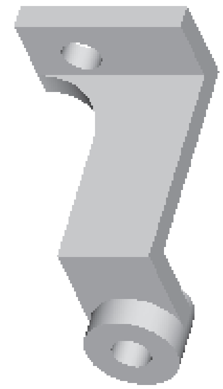


Figure 5-43 *The default trimetric view*

Creating the Two Round Features

Now, the two round features will be created. Both the rounds have different radii and will be created using the **Simple** option.

1. Choose **Insert** > **Round** from the menu bar or **PART** > **Feature** > **Create** > **Solid** > **Round** from the **Menu Manager**.

   The **ROUND TYPE** menu is displayed.

2. Select the **Simple** option and choose **Done** from the **ROUND TYPE** menu.

   The **RND SET ATTR** menu is displayed.

3. Select **Constant** > **Surf-Surf** option from the **RND SET ATTR** menu and choose **Done**.

   You will be prompted to select the surfaces to round.

4. Spin the model and select the surfaces that are shown in Figure 5-44. The **Message Input Window** is displayed with a default value.

5. Enter **5** in this window and press ENTER. The round is created and can be previewed.

6. Choose **OK** from the **ROUND** dialog box. The first round is created.

7. Using the same options, create the second round of radius 15. The surfaces that will be selected to create this round are shown in Figure 5-45. The two round features will appear as two individual features in the **Model Tree**.

Creating the Rib Feature

Some advance planning that was done while drawing the sketch for the base feature will help you to draw the rib feature now. The location for the **RIGHT** datum plane was calculated

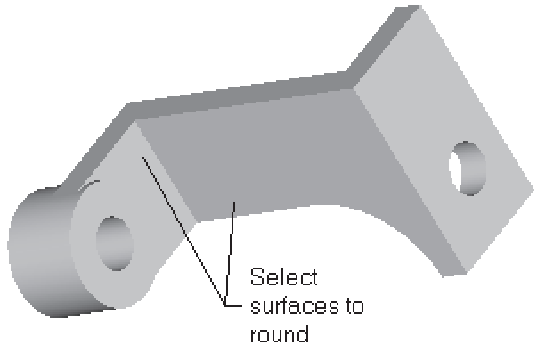


Figure 5-44 Planar surfaces for creating round of radius 5

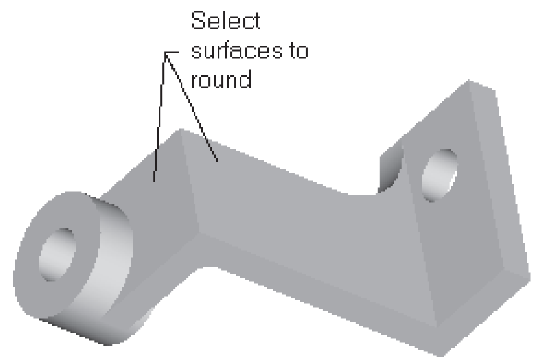


Figure 5-45 Planar surfaces for creating round of radius 15

while sketching the base feature. The section for the rib feature will be drawn on the **RIGHT** datum plane.

1. Choose **Insert > Rib** from the menu bar. You are prompted to create or select a sketching plane. As mentioned earlier, a rib feature is always sketched from the side view. Now, turn on the display of datum planes from the **Datum Display** toolbar.

2. Choose the **RIGHT** datum plane as the sketching plane from the graphics screen. The **SKET VIEW** submenu is displayed.

3. Select **Top** from this submenu and choose the **TOP** datum plane.

   When you select the **TOP** datum plane, the system takes you to the sketcher environment. The **References** dialog box is displayed and the status displayed is **Fully Placed**. Close the **References** dialog box.

   You will need to turn the model display to **No Hidden** before drawing the sketch for the rib feature.

4. Draw the open section sketch for the rib feature and apply the required constraints and dimensions as shown in Figure 5-46.

5. After the sketch is completed, choose the **Continue with the current section.** button.

   The **DIRECTION** submenu is displayed and a red arrow is displayed on the

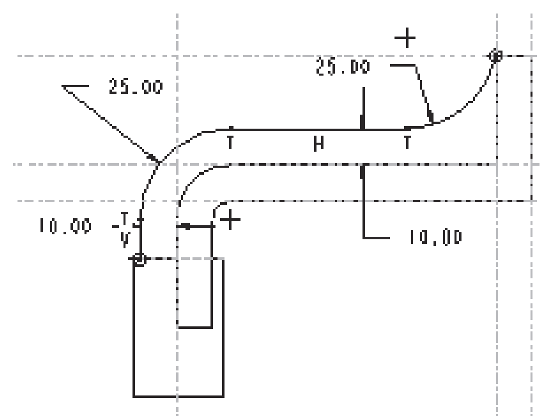


Figure 5-46 Open sketch of the rib feature with dimensions and constraints

sketch. You are prompted to specify the direction of material addition. Since, the section for the rib feature is open, therefore Pro/ENGINEER prompts you for the direction where the material should be added. By default, the red arrow points away from the section.

6. Select the **Flip** option from the **DIRECTION** submenu and choose **Okay**.

   The **Message Input Window** is displayed and you are prompted to enter the rib thickness.

7. Enter **10** in this window and press ENTER. The rib feature is created.

   All the features in the model are created and the model is now complete. The trimetric shaded view of the completed model is shown in Figure 5-47.

Saving the Model

1. Choose the **Save the active object** button from the **File** toolbar and save the model.

   The order of feature creation can be seen from the **Model Tree** shown in Figure 5-48. The feature id numbers displayed in the **Model Tree** may be different when you create the features.

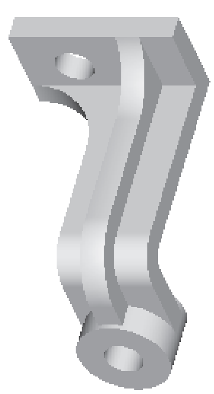


Figure 5-47 *The default trimetric view of the model*

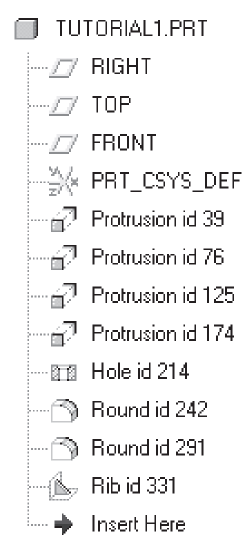


Figure 5-48 ***Model Tree*** *for Tutorial 1*

Tutorial 2

In this tutorial you will create the model shown in Figure 5-49. The dimensions of the model are shown in Figure 5-50. **(Expected time: 30 min)**

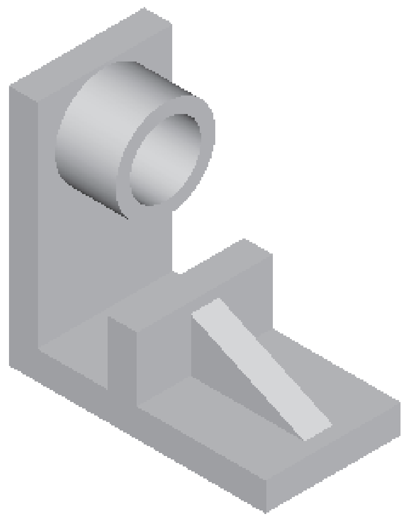


Figure 5-49 Isometric view of the solid model

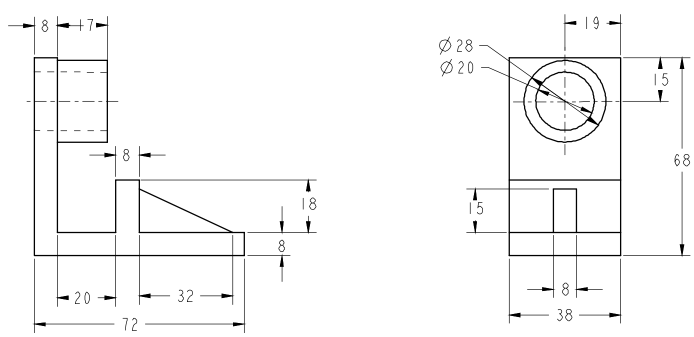


Figure 5-50 Front view and the right-side view of the model for Tutorial 2

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of four features; one at the bottom (base feature), one cylindrical feature, one hole feature on the cylindrical feature, and one rib feature.

b. Select the sketching plane for the base feature, draw the sketch using the sketcher tools, apply the constraints and dimensions, and then extrude the sketch to the given depth, see Figure 5-52.

c. Select the sketching plane for the cylindrical feature, draw the sketch using the sketcher tools, apply the dimensions and constraints, and then extrude the sketch to the given depth, see Figure 5-55.

d. Create the hole on the cylindrical feature using the **HOLE** dialog box, see Figure 5-56.

e. Create a sketching plane for the rib feature, draw the sketch using the sketcher tools, apply the dimensions and constraints, and then give the thickness to the sketch, see Figure 5-58.

After understanding the procedure for creating the model, you are now ready to draw it. The working directory is already selected in Tutorial 1 and therefore you do not need to select it again. However, if you want to change the working directory, choose **File > Set Working Directory** and then select c05 in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it as **c05tut2**. The three default datum planes and the **Model Tree** appears on the graphics screen. Exit the **Model Tree**. The datum planes and **Model Tree** will not appear if they were previously turned off.

Creating the Base Feature

1. Choose **Insert > Protrusion > Extrude**. The **ATTRIBUTES** menu is displayed. Choose **One Side > Done**.

2. Select the **FRONT** datum plane for drawing the sketch of the base feature. The **DIRECTION** submenu is displayed.

3. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

4. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

5. Once you enter the sketcher environment, create the sketch of the base feature and apply constraints and dimensions shown in Figure 5-51.

6. After the sketch is complete, choose the **Continue with the current section.** button and exit the sketcher environment.

   The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion.

7. The **Blind** option is selected in this menu, choose **Done**.

8. Enter a depth of **38** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

   The default trimetric view of the base feature is shown in Figure 5-52.

Creating the Second Feature

The second feature is a cylindrical feature that is sketched on the planar surface of the base feature that is highlighted in the Figure 5-53.

1. Choose **Insert > Protrusion > Extrude**. The **ATTRIBUTES** menu is displayed. Choose

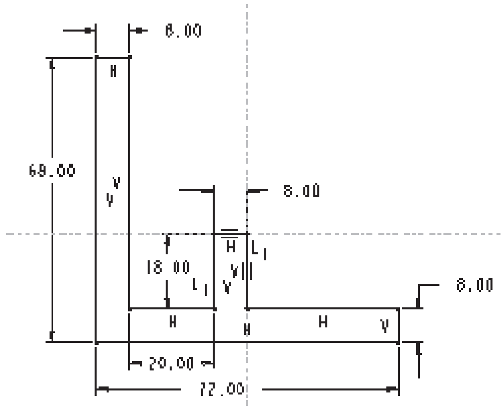


Figure 5-51 *Sketch with dimensions and constraints for the base feature*

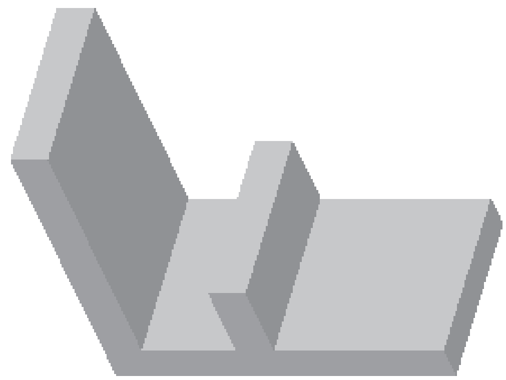


Figure 5-52 *The default trimetric view of the base feature*

One Side > **Done** from this menu.

Note

In this tutorial the base feature can also be created by extruding it on both the sides of the sketching plane. However, to make you familiar with creating datum planes on the fly using the ***Make Datum*** *option, the base feature is extruded on one side of the sketching plane.*

2. Select the planar surface of the base feature shown in Figure 5-53 as the sketching plane. The **DIRECTION** submenu is displayed. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

3. Select the **Top** option from this submenu and choose the **TOP** datum plane.

   After entering the sketcher environment, turn the model display to **No Hidden**.

4. Draw the sketch for the second feature and apply and modify the dimensions as shown in Figure 5-54.

5. Turn the model display to **Shading**. Exit the sketcher environment by choosing the **Continue with the current section.** button. The **SPEC TO** menu is displayed.

6. Select **Blind** > **Done**. Enter a value of **17** in the **Message Input Window** and press ENTER.

7. Choose the **Preview** button from the **PROTRUSION** dialog box and then choose **OK**. The trimetric view of the model with the second feature is shown in Figure 5-55.

Creating the Hole Feature

The hole feature will be created using the **HOLE** dialog box. The coaxial hole will be created on the cylindrical feature. The axis of the cylindrical feature will be used as axial reference to create the coaxial hole.

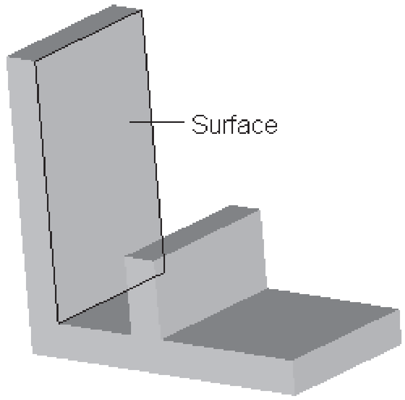


Figure 5-53 *Planar surface selected to sketch*

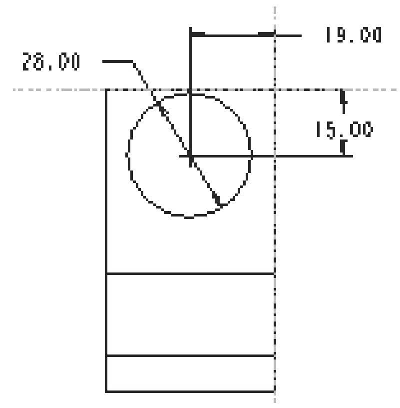


Figure 5-54 *Sketch and dimensions for the second feature*

1. Choose **Insert > Hole** from the menu bar. The **HOLE** dialog box is displayed.

   The **Straight hole** radio button in the **Hole Type** area is selected by default when the dialog box is displayed.

2. Enter a value of **20** in the **Diameter** edit box that is present in the **Hole Dimension** area of the **HOLE** dialog box.

3. From the **Depth One** drop-down list, select the **Thru All** option. In the **Depth Two** drop-down list the **None** option is selected by default.

   As you specify the depth option, the **GET SELECT** menu is displayed and you are prompted to select the primary reference to place the hole.

4. Select the front planar surface of the cylindrical feature to place the hole. The **GET SELECT** menu is displayed and you prompted to select the first reference for hole placement.

5. From the **Placement Type** drop-down list, select the **Coaxial** option. Again the **GET SELECT** menu is displayed and you are prompted to select the hole axis.

   If the datum axis is turned off, you need to turn on the datum axis display using the **Datum axes on/off** button from the **Datum Display** toolbar. This is because the axis of the cylindrical feature has to be selected to reference the hole.

6. Using the left mouse button, select the axis of the cylindrical feature.

7. Choose the **Build feature** button from the **HOLE** dialog box. The model with the hole feature is shown in Figure 5-56.

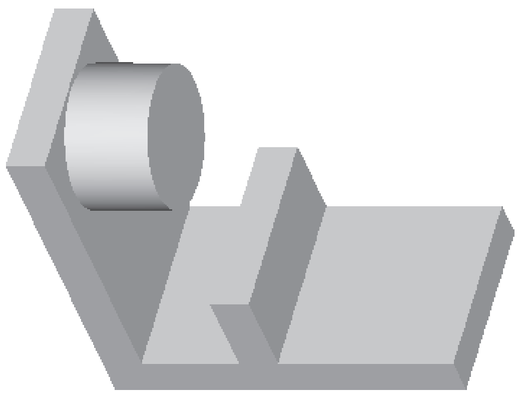


Figure 5-55 *Model with the second feature*

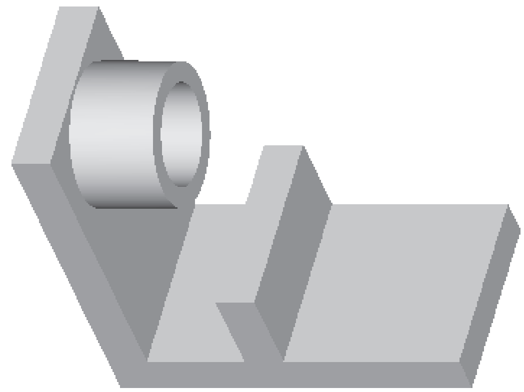


Figure 5-56 *Model with the hole feature*

Creating the Rib Feature

To create the rib feature, a datum plane is created on the fly using the **Make Datum** option. As mentioned earlier, rib features are always drawn from the side view.

1. Choose **Insert** > **Rib** from the menu bar. The **SETUP PLANE** submenu is displayed and you are prompted to select a sketching plane.

2. Choose the **Make Datum** option from the **SETUP PLANE** submenu. The **DATUM PLANE** submenu is displayed.

3. Choose the **Through** option from this menu. Select the axis of the hole and choose **Done**. A datum plane will be created that passes through the selected axis. The **SKET VIEW** submenu is displayed and you will be prompted to select a vertical or a horizontal reference for sketching.

4. Select the **Top** option from this menu and select the **TOP** datum plane. The system takes you to the sketcher environment.

5. Specify the references and close the reference dialog box. Draw the open sketch of the rib feature and apply the dimensions as shown in Figure 5-57. Exit the sketcher environment by choosing the **Continue with the current section.** button.

   The **DIRECTION** submenu is displayed. Turn the model display to **Shading**.

6. Select **Flip** and then choose **Okay** from the **DIRECTION** submenu. The **Message Input Window** appears with a default value.

7. Enter a value of **8** in this window and press ENTER. This value is the thickness of the rib. The trimetric view of the complete model with the rib feature is shown in Figure 5-58.

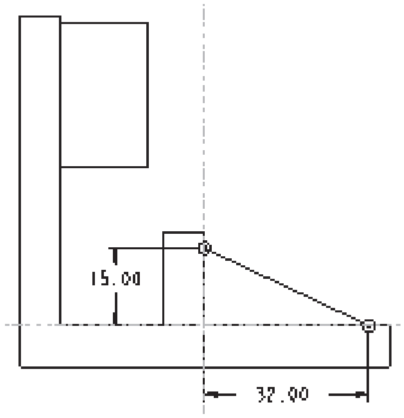


Figure 5-57 Sketch for the rib feature with the model display set to ***No Hidden***

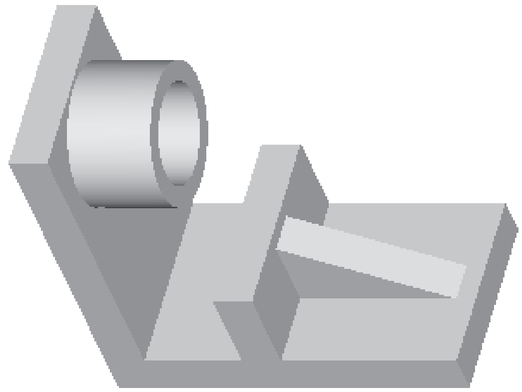


Figure 5-58 The default trimetric view of the final model

Saving the Model

1. Choose the **Save the active object** button from the **File** toolbar and save the model.

   The order of feature creation is evident from the **Model Tree** shown in Figure 5-59. The feature id numbers displayed in the **Model Tree** may be different when you create the features.

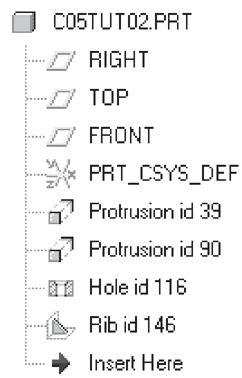


Figure 5-59 Model Tree for Tutorial 2

Tutorial 3

In this tutorial you will create the model shown in Figure 5-60. The solid model, dimensions, the front, and the right-side view are also shown in this figure. **(Expected time: 45 min)**

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of four features; one is the base feature, one cut feature, one counterbore hole on the front planar surface of the base feature, and one round feature, see Figure 5-60.

b. Select the sketching plane for the base feature, draw the sketch using the sketcher tools, apply the dimensions and constraints, and then extrude the sketch to the given depth, see Figure 5-62.

c. Create a sketching plane for the cut feature, draw the sketch using the sketcher tools,

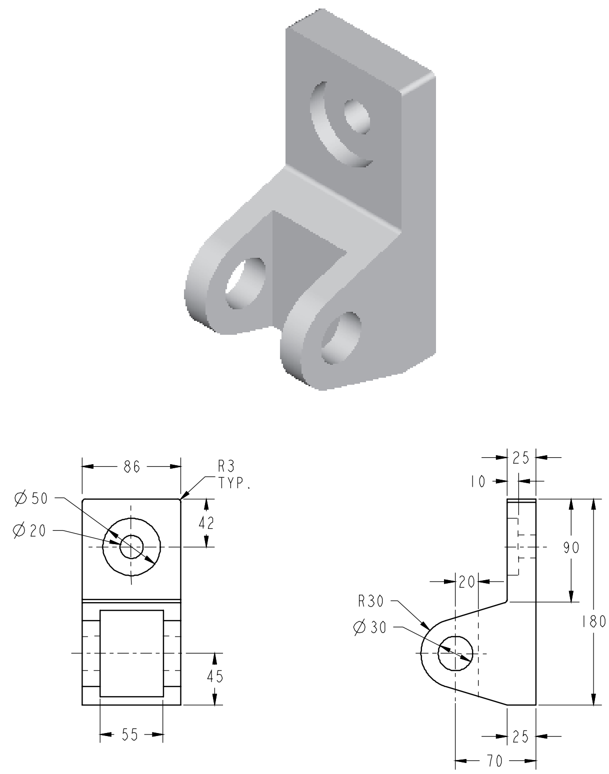


Figure 5-60 Isometric view, front view and the right-side view of the model

apply the dimensions, and then extrude the sketch to both sides of the plane, see Figure 5-64.

d. Create a concentric hole as shown in Figure 5-65

e. Create the round as shown in Figure 5-67.

Creating New Object File

1. Open a new part file and name it as **c05tut3**. The three default datum planes and the **Model Tree** appears on the graphics screen if they were not turned off previously. Exit the **Model Tree**.

Creating the Base Feature

1. Choose **Insert** > **Protrusion** > **Extrude**. The **ATTRIBUTES** menu is displayed. Choose **One Side** > **Done**.

2. Select the **RIGHT** datum plane as the sketching plane. The **DIRECTION** submenu is displayed.

3. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

4. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

5. Once you enter the sketcher environment, create the sketch of the base feature and apply the dimensions and constraints as shown in Figure 5-61.

6. Exit the sketcher environment by choosing the **Continue with the current section.** button. The **SPEC TO** menu is displayed.

7. Choose **Blind** > **Done**. The **Message Input Window** is displayed with a default value.

8. Enter a value of **86** in this window and press ENTER. Choose **OK** from the **PROTRUSION** dialog box. The base feature is completed. The default trimetric view of the base feature is shown in Figure 5-62.

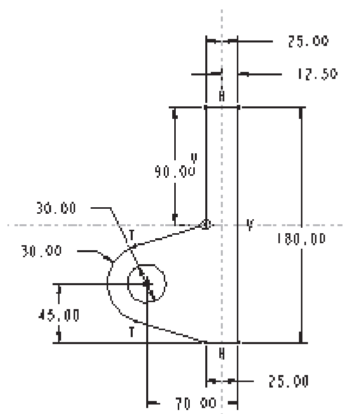


Figure 5-61 Sketch with dimensions and constraints for the base feature

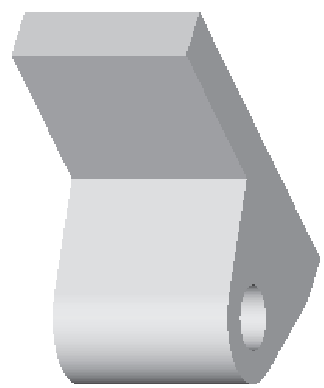


Figure 5-62 The default trimetric view of the base feature

Creating the Second Feature

The second feature is an extruded cut. This cut is created on a datum plane that passes through the center of the base feature.

1. Choose **Insert** > **Cut** > **Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed.

2. Select the **Both Sides** option from this menu and choose **Done**. The **SETUP PLANE** submenu is displayed.

3. Choose the **Make Datum** option from the **SETUP PLANE** submenu. The **DATUM PLANE** submenu is displayed.

4. Choose the **Offset** option from this submenu. Select the **RIGHT** datum plane. The **OFFSET** submenu is displayed.

5. Choose the **Enter Value** option from this submenu. The **Message Input Window** is

displayed.

6. Enter a value of **43** in this window and press ENTER. This value is half of the width of the base feature.

7. Choose **Done** from the **DATUM PLANE** submenu. The datum plane will be created and the **DIRECTION** submenu is displayed.

8. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed. Select the **Left** option from this submenu and select the **FRONT** datum plane. The system takes you to the sketcher environment.

9. Draw the sketch for the cut feature and apply and modify the dimensions as shown in Figure 5-63.

   In Figure 5-63, some dimensions appear light in color. These dimensions are weak dimensions and it is not important to convert them into strong dimensions. These dimensions are not important for the creation of this feature. However, the geometry for the cut should be similar to that shown in Figure 5-63.

10. After the sketch is completed, exit the sketcher environment. The **DIRECTION** menu is displayed.

11. Choose **Okay** from this menu. Choose **Blind** > **Done** from the **SPEC FROM** menu.

12. Enter a blind depth of **55** in the **Message Input Window** and press ENTER. The system will accept this depth symmetrical to the sketching plane.

13. Choose the **Preview** button from the **CUT** dialog box and then choose **OK**. The cut feature is completed now and the default trimetric view of the cut feature along with the base feature is shown in Figure 5-64.

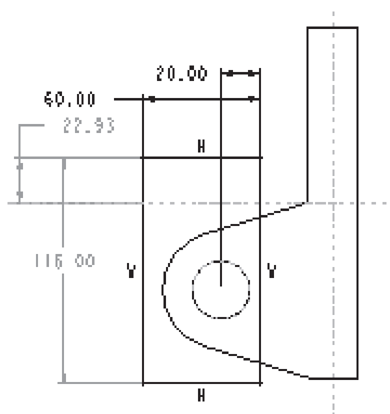


Figure 5-63 *Sketch with dimensions and constraints for the cut feature*

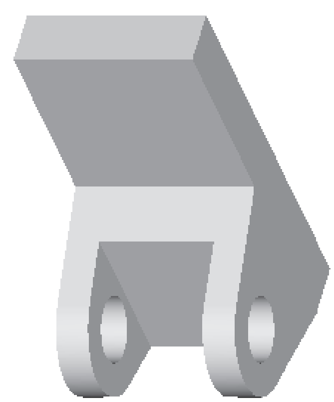


Figure 5-64 *The default trimetric view of the model*

Creating the Hole Feature

The hole is created using the **HOLE** dialog box.

1. Choose **Insert** > **Hole** from the menu bar. The **HOLE** dialog box is displayed.

2. From the **Hole Type** area select the **Standard Hole** radio button, the **Clearance** radio button, and the **Add Counterbore** check box. Clear the **Add Countersink** check box. From the **Screw Size** drop-down list, select the **1-1/4-7** option. All the other options in this area will be accepted as default.

3. In the **Hole Dimension** area, enter the dimensions for the hole as shown in Figure 5-60. The other options in this area will be accepted as default.

Note

While entering the dimensions in the ***Hole Dimension*** *area, press ENTER after entering the value for a dimension. If you do not press ENTER after entering a dimension value, the system will not accept the value.*

4. After specifying all the dimensions for the section of the hole, select the placement plane for the counterbore hole. The placement plane will be the front planar surface of the base feature. The placement type is linear.

   After specifying the placement plane, you are prompted to specify the first reference for hole placement.

5. Select the top horizontal edge of the base feature. A default value appears in the **Distance** edit box.

6. Enter a value of **42** in this edit box and press ENTER. Now, you are prompted to select the second reference.

7. Select the right vertical edge of the base feature. A default value appears in the **Distance** edit box.

8. Enter a value of **43** in this edit box and press ENTER.

9. Choose the **Build feature** button from the **HOLE** dialog box. The hole will be created as shown in Figure 5-65. The specifications of the counterbore hole are also displayed on the graphics screen.

   In Figure 5-65, the hole specifications are not shown. You can remove the text by selecting it using the left mouse button. The text is highlighted in red. If you are not able to select the text, use the CTRL+middle mouse button to spin the model such that a part of text lies outside the model and now select the text. Now, right-click on the screen and choose **Erase** from the menu. The trimetric view of the model after the hole is created is shown in the figure.

Note

When you select a feature, entity, or item from the graphics screen and right-click to invoke the shortcut menu, the right mouse button should be held down until the shortcut menu is displayed. Once the shortcut menu is displayed the right mouse button can be released.

Creating the Round Feature

The round is created using the **Simple** option.

1. Choose **Insert** > **Round** from the menu bar or **PART** > **Feature** > **Create** > **Solid** > **Round** from the **Menu Manager**.

   The **ROUND TYPE** menu is displayed.

2. The **Simple** option is selected by default. Choose **Done** from the **ROUND TYPE** menu.

   The **RND SET ATTR** menu is displayed.

3. Select the **Constant** > **Edge Chain** option from the **RND SET ATTR** menu and choose **Done**. The **CHAIN** menu is displayed.

4. Select the **One By One** option from this menu. You are prompted to select the edges to round. Select the edges that are shown in Figure 5-66. The selected edges turns blue in color.

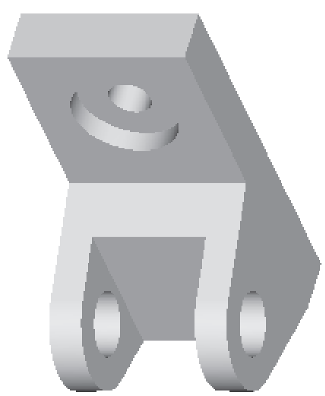


Figure 5-65 *The default trimetric view*

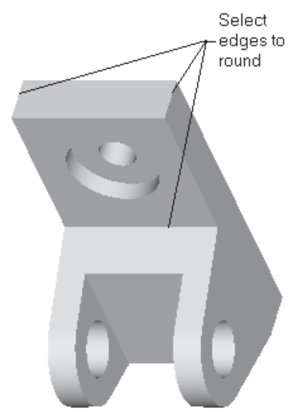


Figure 5-66 *Edges to be selected to round*

5. Choose **Done** from the **CHAIN** menu. The **Message Input Window** is displayed with a default value.

6. Enter **3** in this window and press ENTER. The round is created and can be previewed.

7. Choose **OK** from the **ROUND** dialog box. The trimetric view of the final model with round feature created is shown in Figure 5-67.

Saving the Model

1. Choose the **Save the active object** button from the **File** toolbar and save the model.

   The order of feature creation can be seen from the **Model Tree** shown in Figure 5-68.

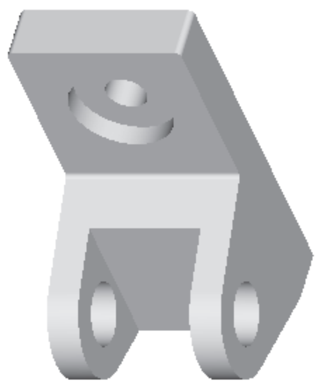


Figure 5-67 Final model for Tutorial 3

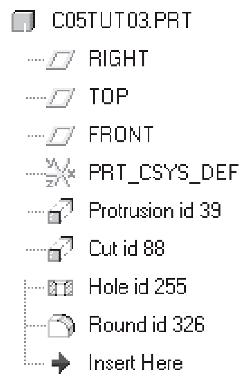


Figure 5-68 Model Tree for Tutorial 3

Self-Evaluation Test

Answer the following questions and then compare your answers to the answers given at the end of this chapter.

1. A hole created using the **HOLE** dialog box is parametric in nature. (T/F)
2. A hole cannot be created on both sides of the sketching plane or the placement plane. (T/F)
3. In Pro/ENGINEER, holes can also be sketched. (T/F)
4. The **Full Round** option in the **RND SET ATTR** menu allows you to enter the radius of round. (T/F)
5. The **Feature Creation** dialog box is used extensively in Pro/ENGINEER for editing a model. (T/F)
6. The rib feature is always created from the __________ view.
7. A __________ hole is a stepped hole and has two diameters, a larger one and a smaller one.
8. Straight holes are the holes that have circular cross-section having __________ diameter throughout the depth.

9. __________ are defined as thin wall-like structures used to bind the joints together so that they do not fail under an increased load.

10. When you redefine a feature that does not have elements then the __________ menu is displayed.

Review Questions

Answer the following questions:

1. The **Dynamic Modify** option is available only for the extruded features. (T/F)

2. While in the sketcher mode, the constraints to a sketch should be applied before the dimensions. (T/F)

3. The chamfers created in Pro/ENGINEER are parametric in nature. (T/F)

4. The **Model Tree** can be used to redefine a feature. (T/F)

5. When you redefine a rib feature, the __________ submenu is displayed.

6. Which of the following option is used to create a round on the selected edges such that the different radius values can be applied along the selected edge?

   (a) **Constant** (b) **Variable**
   (c) **Thru Curve** (d) **Full Round**

7. Which of the following menus can be used to redefine a feature?

   (a) **PART** (b) **FEAT**
   (c) **SPEC TO** (d) **GET SELECT**

8. Which of the following options of the **PART** menu can modify the dimensions of a feature after it is completed?

   (a) **Feature** (b) **Modify**
   (c) **Regenerate** (d) None

9. At the intersection of how many edges does a **Corner** chamfer create a beveled surface?

   (a) One (b) Two
   (c) Three (d) None

10. Rounds and chamfers are used in engineering components to reduce the __________ on the corners.

(a) Stress concentration (b) Tension
(c) Conduction (d) None

Exercises

Exercise 1

Create the model shown in Figure 5-69. The dimensions and front and top views of the model are shown in Figure 5-70. **(Expected time: 45 min)**

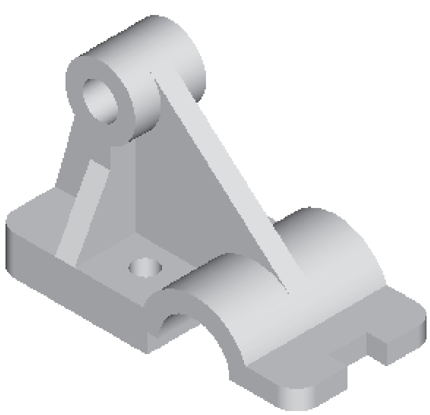


Figure 5-69 Isometric view of the model

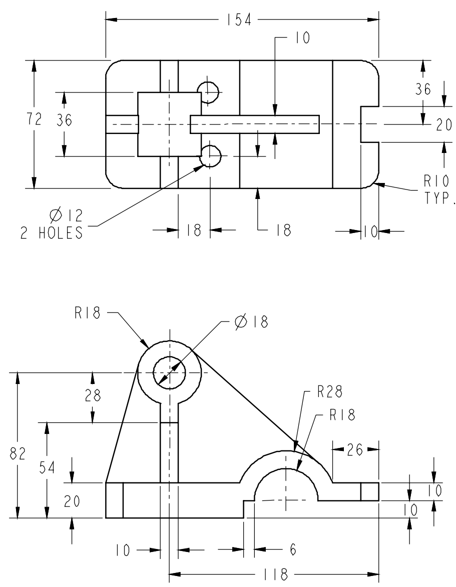


Figure 5-70 Top and front views of the model, hidden lines are suppressed for clarity

Exercise 2

Create the model shown in Figure 5-71. The dimensions, front and right-side views of the model are shown in Figure 5-72. **(Expected time: 30 min)**

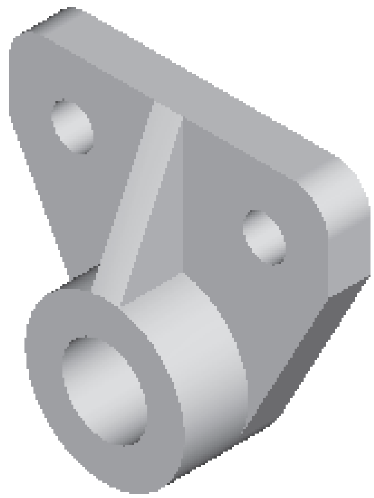


Figure 5-71 *Isometric view of the model for Exercise 2*

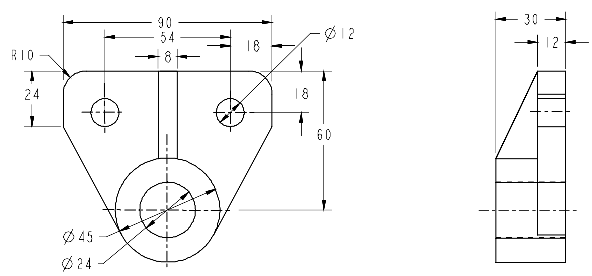


Figure 5-72 *Front view and the right-side view of the model*

Answers to the Self-Evaluation Test

1 - T, **2** - F, **3** - T, **4** - F, **5** - T, **6** - side, **7** - counterbore, **8** - constant, **9** - Ribs, **10** - **REDEFINE**.