Chapter 5

Datums

Learning Objectives

After completing this chapter, you will be able to:

- *Understand three default datum planes.*
- *Understand selection methods in Pro/ENGINEER Wildfire 4.0.*
- *Create datum planes using different constraints.*
- *Create datum planes on-the-fly.*
- *Create datum axes using different constraints.*
- *Create datum points.*
- *Create extrude and revolve cuts.*

DATUMS

Datums are imaginary features with no mass or volume. Datums are available to help you in creating models. They act as reference for sketching a feature, orienting the model, assembling components, and so on. Remember that the datums play a very important role in creating complex models in Pro/ENGINEER and therefore you must have a good understanding of datums. Datums are considered to be features but not model geometry. In Pro/ENGINEER, datums exist as datum plane, datum curve, datum point, datum coordinate system, datum graph, and so on.

Default Datum Planes

When you enter the **Part** mode or the **Assembly** mode, the three datum planes are displayed by default in the drawing area. These datum planes are known as the default datum planes and they are mutually perpendicular to each other. The only difference between the default datum planes of the **Part** mode and those of the **Assembly** mode lies in the names of the datum planes.

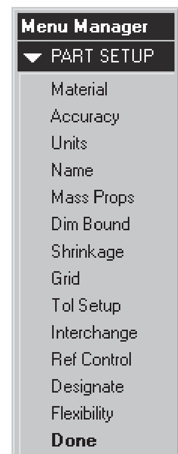


Figure 5-1 *The* ***Menu Manager***

The default datum planes in the **Part** mode are named as **FRONT**, **TOP**, and **RIGHT**. In case of the **Assembly** mode, the default datum planes are named as **ASM_FRONT**, **ASM_TOP**, and **ASM_RIGHT**. However, the names of the default datum planes can be changed if required.

To change the names, choose **Edit > Setup** from the menu bar. The **Menu Manager** with the **PART SETUP** menu will be displayed, as shown in Figure 5-1. Choose the **Name** option from this menu; you are prompted to select a feature to change the name. Select the datum plane you want to rename. When the **Message Input Window** appears at the top of the drawing window, enter the desired name in it and choose the **Accept value** button.

The **PART SETUP** menu can also be used to set the material, units, and other parameters related to the model.

Note

The default unit of a part is ***Inch lbm Second****. The length is measured in inches, mass in lbm, time in seconds, and temperature in Fahrenheit. There are other systems of units that are available and you can change the units as desired.*

NEED FOR DATUMS IN MODELING

Generally, most of the engineering components or designs consist of more than one feature. First the base feature of the model is created and then other features of the model are added. Since all features of a model cannot be drawn on a single plane, therefore, to draw rest of the features, sometimes, additional planes have to be created or selected. Also, most of the times, the three default datum planes are not enough to create a complex model having many features. For example, Figure 5-2 shows a simple model that consists of two features that require two different planes.

Tip: *Whenever you come across any solid model, first try to visualize the number of features in that model and then decide which feature is to be considered as the base feature.*

In Figure 5-2 any one of the two features that are defined on two different planes can be considered as the base feature. However, in this discussion, the feature that is selected as the base feature is shown in Figure 5-3. After creating the base feature, the next feature will be created. For the next feature, a sketching plane has to be defined. Therefore, an additional plane has been created on which you can draw the sketch for the second feature.

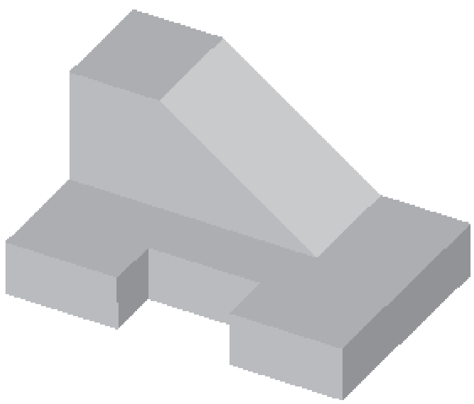


Figure 5-2 *Model having two extruded features*

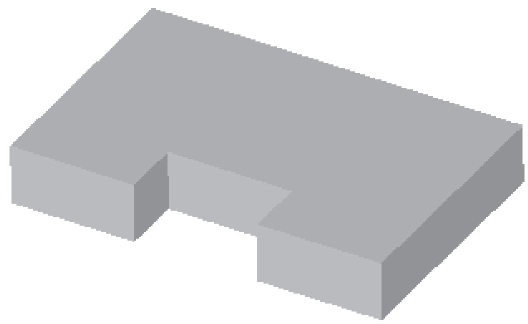


Figure 5-3 *Base feature of the model*

As shown in Figure 5-4, the plane that is used to create the base feature is highlighted by a mesh. To create the second feature, a new plane is created which is shown in Figure 5-5. The sketch of the second feature is drawn on this plane.

Note

Throughout the book, at some instances the datum planes are shown by a mesh plane as is evident from Figure 5-4. This view of the datum plane is only for explanation. In Pro/ENGINEER, when you create a datum plane, they do not appear in the form of mesh.

To create various types of datums, you need to select the references on the model. Based on the selection you make on the model, Pro/ENGINEER applies constraints to create the datum.

Selection Method in Pro/ENGINEER Wildfire 4.0

The selection method in Pro/ENGINEER Wildfire 4.0 is divided into two types:

1. Selection
2. Collection

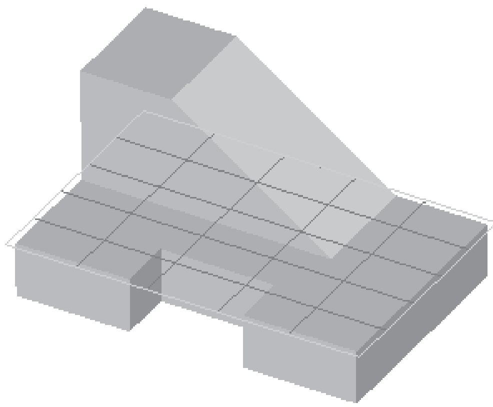


__Figure 5-4__ Plane selected for the base feature

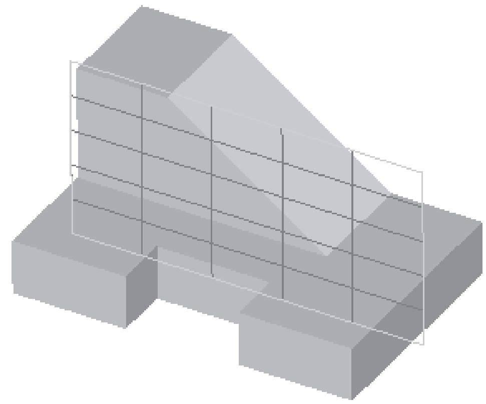


__Figure 5-5__ Plane selected for the second feature

Selection

In Pro/ENGINEER Wildfire 4.0, selection refers to the action in which you first select the entities like edge, plane, face, axis, coordinate system, and so on, and then invoke a feature creation tool. This property of a design package is also known as Object Action. To make your selection process easier, filters are available in Pro/ENGINEER Wildfire 4.0. Filters are the options that are available in the **Filter** drop-down list located at the lower right corner of the main window in the **Status Bar**. The options in the **Filter** drop-down list change with the mode that is active. The options in this drop-down list when you are in the **Part** mode are shown in Figure 5-6.

As you select the entities on the model, the number of selected entities appear to the left of the **Filter** drop-down list in the **Status Bar**. When you double-click on the number, the **Selected Items** dialog box will be displayed, as shown in Figure 5-7. All selections that you make on the model are available in this dialog box. You can use the dialog box to remove the selected entities. These selections appear highlighted in red color on the model. To remove a selection on the model, press the CTRL key and then select the entity in red.

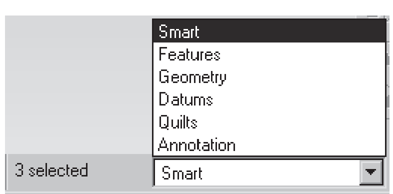


__Figure 5-6__ The __Filter__ drop-down list

__Figure 5-7__ The __Selected Items__ dialog box

By default, the **Smart** filter is selected in the **Filter** drop-down list. The second filter in the **Filter** drop-down list is **Features**. This filter is used to select the features on the model. To redefine a feature, select this option from the drop-down list. The third filter is **Geometry**. This filter allows you to select only the edges, vertices, and surfaces. The fourth filter is **Datums**. This filter allows you to select the datum features. The fifth filter is **Quilts**. This filter allows you to select surfaces. The sixth filter is **Annotation**. This filter is used to select the notes from the drawing area.

Collection

In Pro/ENGINEER Wildfire 4.0, collection refers to the action when you have invoked a feature creation tool and then select the entities to collect the references in order to create that feature. To make your selection process easy, filters can be used. The filters available in the **Filter** drop-down list depend on the feature creation tool that you have invoked.

DATUM OPTIONS

After discussing the default datum planes, which are the first feature in the **Part** mode, you must know the various other features created using the datum options. Datums are also considered as features having no geometry. Figure 5-8 shows the **Datum** toolbar and Figure 5-9 shows the method of invoking various types of datum options from the menu bar.

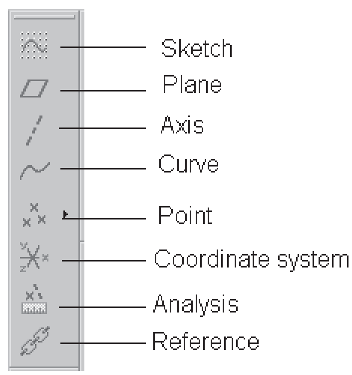


Figure 5-8 *The* ***Datum*** *toolbar*

Datum Planes

You can create datum planes, other than the three default datum planes, by using the menu bar or the **Datum** toolbar. The datum planes can be created anytime when required. The display of the datum planes can be turned on or off by using the **Plane Display** button from the **Datum Display** toolbar. Before discussing the procedure to create datum planes by using the available options, it is important to understand the use of datum planes. Some of the uses of datum planes in Pro/ENGINEER Wildfire 4.0 are listed below:

1. Datum planes are used as sketching planes to create sketches for the features of a model.

2. Datum planes are used as reference planes for sketching.

3. Datum planes are used as references for placing holes and for assembly.

4. Datum planes are used as a reference for mirroring features, copying features, creating a cross-section, and for orientation of references.

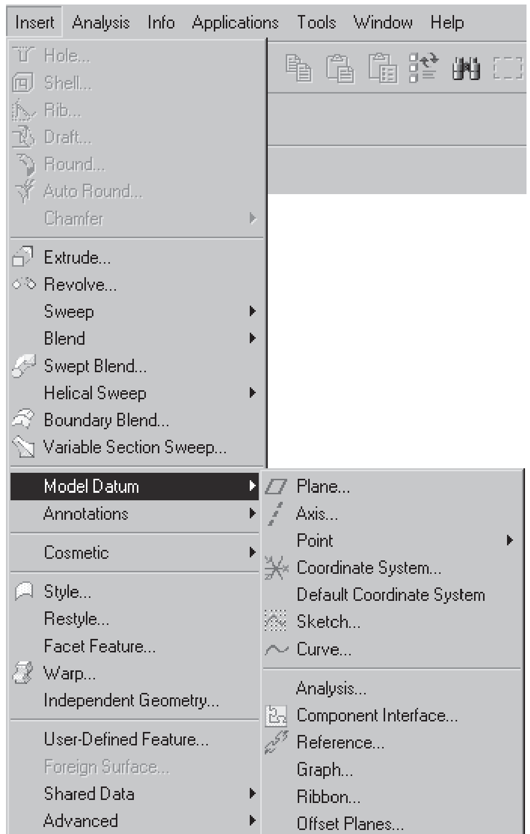


Figure 5-9 *Invoking the datum options from the* ***Insert*** *menu in the menu bar*

Pro/ENGINEER provides you with various constraints to create additional datum planes. Additional datum planes are created with the help of constraints and the filters available in the **Filter** drop-down list. Datums can also be created while you are in the sketcher environment. When you choose **Insert > Model Datum > Plane** from the menu bar or **Plane** button from the **Datum** toolbar, the **DATUM PLANE** dialog box will be displayed, as shown in Figure 5-10. Separate buttons for creating datum axis, datum curve, datum point, datum coordinate system are available in the **Datum** toolbar.

Tip: *Generally, for the base feature creation, the three default datum planes are used and as the part becomes complex or in other words, as the number of features increase, the need for additional datum planes arises.*

Figure 5-10 shows various options available in the **DATUM PLANE** dialog box to create datum planes. The options available in different tabs of the **DATUM PLANE** dialog box are discussed next.

Placement Tab

The **Placement** tab in the **DATUM PLANE** dialog box is chosen by default. Under this

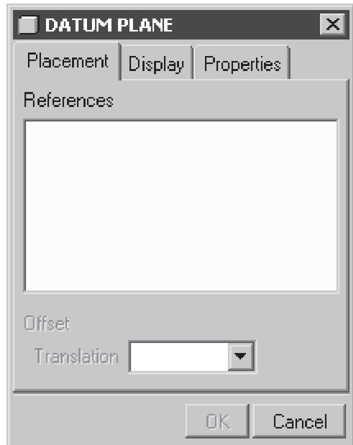


Figure 5-10 *The* ***DATUM PLANE*** *dialog box with the* ***Placement*** *tab chosen*

tab, the **References** area displays the references that are selected to create the datum plane. The constraints are displayed on the right of the references. These constraints are applied automatically based on the reference you select. To change a constraint, select the constraint in the dialog box; you will notice that a drop-down list appears in its place. From this drop-down list, select a different constraint. This drop-down list is available only for those constraints in the dialog box, which can be substituted by another constraint. The constraints that are used to create a datum plane are discussed later in the chapter.

The **Offset** area is available only when the **Offset** constraint is used to create a datum plane. The **Rotation** edit box is used to enter the rotation angle of the new datum plane. The angle can also be set dynamically on the model by using the drag handle displayed on the model.

Display Tab

The **Display** tab of the **DATUM PLANE** dialog box is shown in Figure 5-11. The **Flip** button in this dialog box is used to change the normal direction of the datum plane being created. Every datum plane has two sides, one is colored brown and the other is black. These colors are visible when the plane is rotated such that its back side comes into view. By default, the arrow on the plane points toward the brown side. The arrow direction and hence the direction of the plane can be changed by using the **Flip** button.

By default, the **Adjust Outline** check box is clear. If you select this check box, it allow you to set the size of the datum plane.

Properties Tab

The **DATUM PLANE** dialog box with the **Properties** tab chosen is shown in Figure 5-12. This tab when selected, allows you to name the datum plane you are creating. By default, the first datum plane you create is named DTM1 and then every datum plane created is successively numbered.

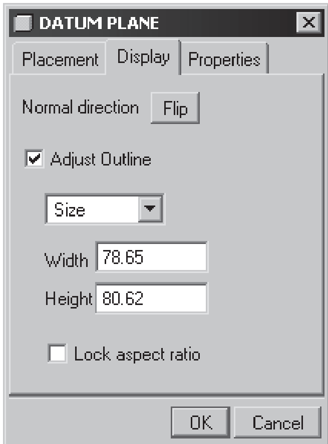


Figure 5-11 *The* ***DATUM PLANE*** *dialog box with the* ***Display*** *tab chosen*

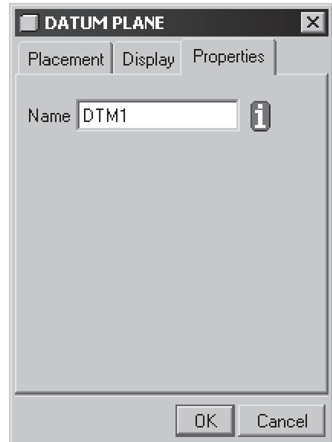


Figure 5-12 *The* ***DATUM PLANE*** *dialog box with the* ***Properties*** *tab chosen*

Creating Datum Planes

When you create a datum plane, the constraints are applied automatically based on the selection you make on the model. These constraints appear in the **DATUM PLANE** dialog box. Sometimes, while applying constraints to define a datum plane, a single constraint is enough to define the datum plane and sometimes, you may need more than one constraint to do the same. When only one constraint is used to create a datum plane, it is called stand-alone constraint. The stand-alone constraints are sufficient by themselves to constrain a datum plane definition. The constraints that are used to create a datum plane are discussed next.

Through Constraint

The **Through** constraint is used to create a datum plane through any specified axis, edge, curve, point/vertex, plane, cylinder, or coordinate system. This constraint can be used in combination with other constraints that are discussed next.

However, the **Through** constraint can also be used as a stand-alone constraint when you select a plane. Figure 5-13 shows the datum plane constraint combinations using the **Through** constraint. Datum planes can be created using any of the combinations shown in the figure. The possible combinations of datum plane creation are referred to as **Yes** and the combinations that are not possible are referred to as **No** in the figure.

While reading the table shown in Figure 5-13, first preference is given to the text written in the first column and then the text in the first row should be read. For example, if you want to make a datum plane that is passing through a cylinder and normal to a plane then look for **Through** in the first column and then for **Cylinder** in the second column. Now, look for **Normal** in the first row and for **Plane** in the second row. After finding both the combinations trace them in the respective column and row till they intersect. You will find **Yes**. This suggests that the creation of a datum plane that passes through a cylinder and is normal to a plane is possible. While reading the table shown in Figure 5-13, remember that the constraints that are not stand-alone have to be applied in pairs.

DATUM PLANE CONSTRAINT COMBINATIONS (USING THROUGH CONSTRAINT)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		Axis/Edge/ Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Through	Axis/Edge/ Curve	Yes	Yes	Yes	Yes	Yes	Yes	Yes	No
	Point/ Vertex	Yes	Yes	Yes	Yes	Yes	No	Yes	No
	Plane	No	No	No	No	No	No	No	Yes
	Cylinder	Yes	Yes	Yes	Yes	Yes	Yes	Yes	No

Figure 5-13 *Datum plane constraint combinations using the* ***Through*** *constraint*

The options in the **Filter** drop-down list shown in Figure 5-14 are used to make selection on the model. This drop-down list is located at the bottom right corner of the main window. The options in this drop-down list change depending on the operation being performed. For example, when you are creating a datum plane, the options shown in Figure 5-14 are available. Once you exit the datum plane creation, the options in this drop-down list change.

Figure 5-14 *The* ***Filter*** *drop-down list with options*

Figures 5-15 and 5-16 show the cylindrical surface and the default datum plane used to create a datum plane at an angle to the selected default datum plane and passing through the center of the cylindrical surface. When you select the cylindrical surface, it is highlighted in red, indicating that it is selected. To make the second selection, select the **Datum Plane** filter from the **Filter** drop-down list. Now, press the CTRL button and then select the datum plane. This method of selecting by pressing the CTRL button is known as collection. Click on the constraint displayed against the selected plane in the **DATUM PLANE** dialog box and select the **Offset** option from the drop-down list. Enter the angle value in the **Rotation** dimension box. Choose the **OK** button to create the datum plane.

Note

The first reference to constrain a datum plane is selected using the left mouse button. The second reference is selected using the CTRL+left mouse button.

Make sure you change the option in the ***Filter*** *drop-down list when you invoke any other tool.*

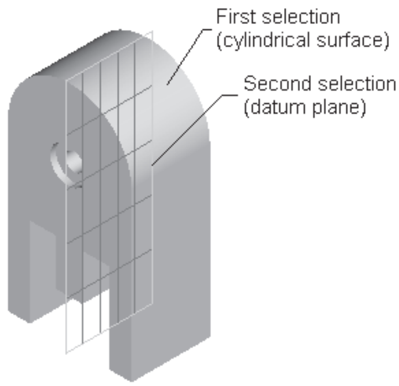


Figure 5-15 *Selecting a cylindrical surface and a default datum plane to create a datum plane*

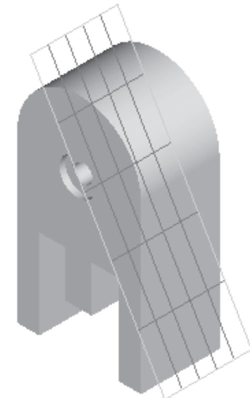


Figure 5-16 *Resulting datum plane passing through the center of a cylinder*

Normal Constraint

The **Normal** constraint is used to create a datum plane normal to any specified axis, edge, curve, or plane. This constraint is used in combination with other constraints. The **Normal** constraint cannot be used as a stand-alone constraint. Figure 5-17 shows the datum plane constraint combinations using the **Normal** constraint.

DATUM PLANE CONSTRAINT COMBINATIONS (USING NORMAL CONSTRAINT)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		Axis/Edge/ Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Normal	Axis/Edge/ Curve	Yes	Yes	Yes	No	No	No	No	No
	Plane	Yes	Yes	Yes	Yes	Yes	No	Yes	No

Figure 5-17 *Datum plane constraint combinations using the* ***Normal*** *constraint*

The possible combinations of datum plane creation are referred to as **Yes** and the combinations that are not possible are referred to as **No**.

Figure 5-18 shows a planar face and a cylindrical face of a solid model. The planar face is selected as the normal surface and the cylindrical face is selected to be tangent to the datum plane. The following steps explain the procedure to create this datum plane:

1. Invoke the **DATUM PLANE** dialog box. Select the first reference shown in Figure 5-18. The **Offset** constraint is automatically applied and will be displayed in the **References** collector.

2. Left-click on the **Offset** constraint in the collector; a drop-down list appears. Select the **Normal** constraint from this drop-down list.

3. Now, use CTRL+left mouse button to select the second reference shown in Figure 5-18. The **Through** constraint will be displayed in the **References** collector.

4. Change this constraint to **Tangent** from the drop-down list that appears when you click on the constraint in the **References** collector. The datum plane that is created is shown in Figure 5-19.

5. Choose the **OK** button from the **DATUM PLANE** dialog box.

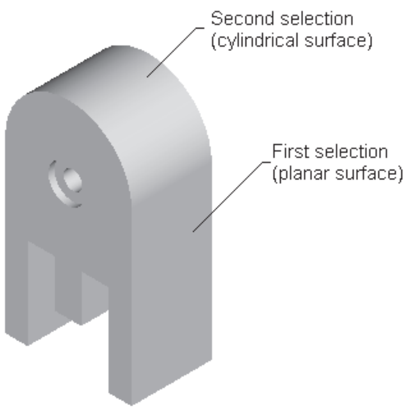


Figure 5-18 *Selecting a planar surface and a cylindrical surface to create a datum plane*

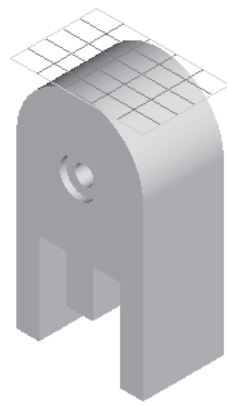


Figure 5-19 *Resulting datum plane*

Parallel Constraint

The **Parallel** constraint is used to create a datum plane parallel to any specified datum plane or planar face. This option is used in combination with other constraints. The **Parallel** constraint cannot be used as a stand-alone constraint. Figure 5-20 shows various datum plane constraint combinations using the **Parallel** constraint. The possible combinations of datum plane creation are referred to as **Yes** and the combinations that are not possible are referred to as **No** in Figure 5-20.

Figure 5-21 shows the selection of a default datum plane and an axis to create a datum plane. The resulting datum plane is parallel to the selected datum plane and passes through the axis, as shown in Figure 5-22. The steps explaining the procedure to create this datum plane are given next:

1. Invoke the **DATUM PLANE** dialog box. Select the first reference shown in Figure 5-21. The **Offset** constraint is automatically applied and will be displayed in the **References** collector.

2. Left-click on the **Offset** constraint in the collector; a drop-down list appears. Select the **Parallel** constraint from this drop-down list

DATUM PLANE CONSTRAINT COMBINATIONS (USING PARALLEL CONSTRAINT)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		Axis/Edge/ Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Parallel	Plane	Yes	Yes	Yes	No	No	No	Yes	No

Figure 5-20 *Datum plane constraint combinations using the* ***Parallel*** *constraint*

3. Now, use CTRL+left mouse button to select the second reference shown in Figure 5-21. The **Through** constraint will be displayed in the **References** collector.

4. Choose the **OK** button from the **DATUM PLANE** dialog box. The datum plane that is created is shown in Figure 5-22.

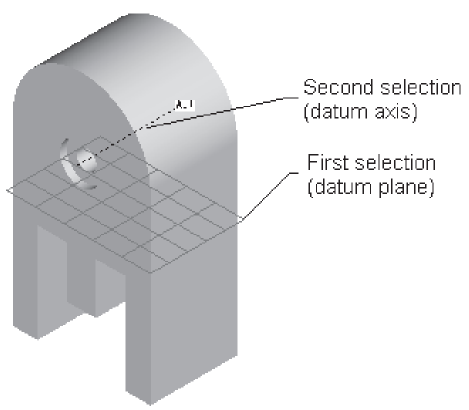


Figure 5-21 *Selecting a datum plane and an axis to create a datum plane*

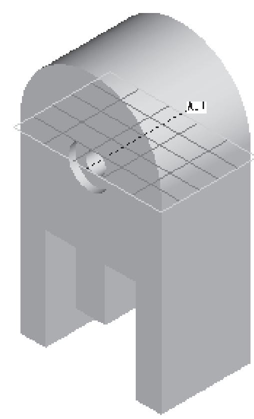


Figure 5-22 *Resulting datum plane*

Offset Constraint

The **Offset** constraint is used to create a datum plane at an offset distance to any specified plane or coordinate system. This option is used in combination with other constraints. However, the **Offset** constraint can be used as a stand-alone constraint when you select a plane to offset from.

When the **Offset** constraint is applied, the **Translation** dimension box appears in the **DATUM PLANE** dialog box. This dimension box list is used to specify an offset distance. In the case of angular planes the **Translation** dimension box changes to the **Rotation** dimension box in which you need to specify the angle. An arrow appears on the model that shows the positive direction of the offset distance or angle.

Figure 5-23 shows various datum plane constraint combinations using the **Offset** constraint. The possible combinations of datum plane creation are referred to as **Yes** and the combinations that are not possible are referred to as **No** in the Figure 5-23.

DATUM PLANE CONSTRAINT COMBINATIONS (USING OFFSET CONSTRAINT)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		Axis/Edge/ Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Offset	Plane	Yes	No	Yes	No	No	No	No	Yes
	Coord System	No	No	No	No	No	No	No	Yes

Figure 5-23 *Datum plane constraint combinations using the **Offset** constraint*

Figure 5-24 shows the selection of a default datum plane and an edge to define an offset datum plane. The resulting datum plane is at an offset to the selected datum plane and passes through the edge, as shown in Figure 5-25. The following steps explain the procedure to create this datum plane:

1. Invoke the **DATUM PLANE** dialog box. Select the first reference shown in Figure 5-24. The **Offset** constraint is automatically applied and will be displayed in the **References** collector. The **Translation** dimension box appears.

2. Now, use CTRL+left mouse button to select the second reference, which is a vertex shown in Figure 5-24. The **Through** constraint will be displayed in the **References** collector.

3. Choose the **OK** button from the **DATUM PLANE** dialog box. The datum plane that is created is shown Figure 5-25.

Note
*The **OK** button in the **DATUM PLANE** dialog box is enabled only when the datum plane you are creating is fully constrained.*

Tangent Constraint

The **Tangent** constraint creates datum planes tangent to cylindrical features. This constraint is also used with other constraints to create various types of datum planes. Figure 5-26 shows the datum plane constraint combinations using the **Tangent** constraint. The possible combinations of datum plane creation are referred to as **Yes** and the combinations that are not possible are referred to as **No** in the figure.

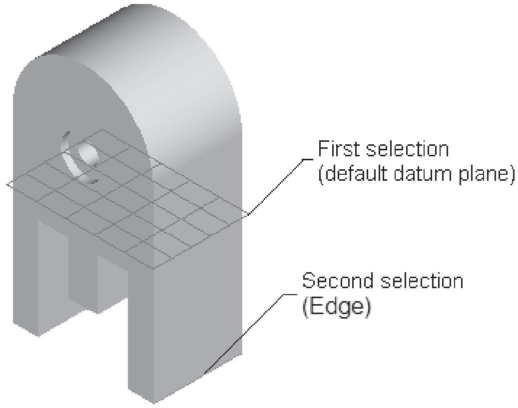


Figure 5-24 *Selecting a datum plane and an edge to create a datum plane*

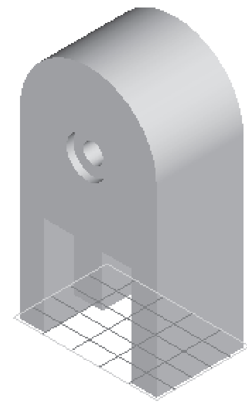


Figure 5-25 *Resulting datum plane*

DATUM PLANE CONSTRAINT COMBINATIONS (USING TANGENT CONSTRAINT)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		Axis/Edge/ Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Tangent	Cylinder	Yes	No	No	Yes	Yes	No	No	No

Figure 5-26 *Datum plane constraint combinations using the* ***Tangent*** *constraint*

Note
To remove a selected reference, select the reference in the dialog box and press DELETE.

Datum Planes Created On-The-Fly

The term **On-the-Fly** refers to the creation of a datum plane when the system prompts you to select or create a plane. At this step, if you choose the **Plane** button from the **Datum** toolbar, the datum plane created will be called datum plane on-the-fly. When you create a datum plane on-the-fly, the datum plane is neither visible in the drawing area nor will be displayed under the default display of the **Model Tree** once the feature is completed.

You can try making a datum plane on-the-fly by following these steps:

1. Choose the **Extrude** button; the **Extrude** dashboard will be displayed.

2. In the **Extrude** dashboard, choose the **Placement** tab and then choose the **Define** button to invoke the **Sketch** dialog box.

3. Now, you need to select a sketching plane. You can select an existing datum plane, a face as the sketch plane, or you can create a datum plane on-the-fly.

4. Choose the **Plane** button from the **Datum** toolbar. Select the references and constraints to create the datum plane. This datum plane when created is automatically selected as the sketching plane.

When you need to select the references to orient the sketching plane, at this step also you can create a datum plane on-the-fly. Once the datum plane on-the-fly is created, it cannot be referenced by other features.

The datum plane created on-the-fly does not appear in the **Model Tree** under the default display. But it can be seen in the **Model Tree** when you click the plus sign (+) that appears to the left of the feature you have created. The datum plane and the feature that you have created are grouped and appears as a grouped feature in the **Model Tree**. This means that the datum on-the-fly that was created during feature creation belongs to the feature that is created.

Note
Unlike the previous releases of Pro/ENGINEER, now you can create a rotational pattern without creating a datum on-the-fly.

Datum Axes

Datum axis is an imaginary axis that is created in Pro/ENGINEER to help you in creating a model. Datum axes can be created manually. They are also created automatically when any cylindrical feature is created. The display of the datum axis can be turned on or off by using the **Plane Display** button from the **Datum Display** toolbar. The uses of datum axes are given next.

1. Datum axes act as reference for feature creation.

2. They are used in creating a datum plane along with different constraint combinations.

3. They are used in placing features co-axially.

4. They are also used to create rotational patterns. You will learn to create patterns in Chapter 7.

Note
You can also create a datum axis perpendicular to the sketch plane in the sketcher environment. To create a datum axis in the sketcher environment, choose ***Sketch > Axis Point*** *from the menu bar. Then, select a point in the drawing area through which the axis will pass.*

Datum axes are named by default in Pro/ENGINEER. The default name of a datum axis is

A_(Number), where **Number** represents the number of datum axis. However, the default name of the datum axes can be changed in the same way as that of the datum planes.

When you choose **Insert > Model Datum > Axis** from the menu bar or the **Axis** button from the **Datum** toolbar, the **DATUM AXIS** dialog box appears, as shown in Figure 5-27. The options available in this dialog box are discussed next.

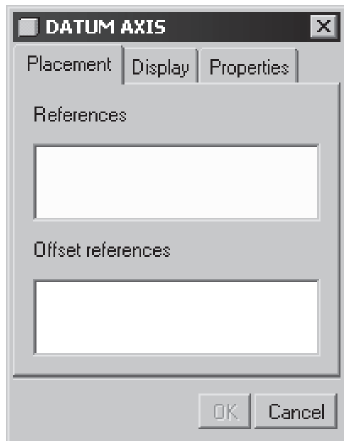


Figure 5-27 *The* ***DATUM AXIS*** *dialog box with the* ***Placement*** *tab chosen*

Placement Tab

When you invoke the **DATUM AXIS** dialog box, the **Placement** tab is chosen by default. Under this tab, the **References** collector allows you to select the references that will create the datum axis. The constraints are displayed on the right of the references. These constraints are applied automatically based on the reference you select. The constraints that are used to create a datum axis are discussed later in this chapter.

The **Offset References** collector is available only when the **Normal** constraint is used in the **References** collector to define a datum axis. To define the datum axis, you need to select two references. These references can be an edge, datum plane, face, or axis. The references can be specified dynamically on the model by dragging the handles displayed on the datum axis. The handles on the datum axis are displayed only when you select a reference. You can also select the references using the **DATUM AXIS** dialog box.

There are three drag handles that are available on the model, as shown in Figure 5-28. The middle handle is used to move the position of the axis you are creating and it appears like a white square. The other two handles are used to specify the references for dimensioning and appear like a green square.

Display Tab

The options in this tab are used to control the length of the axis. The **DATUM AXIS** dialog box with the **Display** tab chosen is shown in Figure 5-29. Select the **Adjust Outline** check box

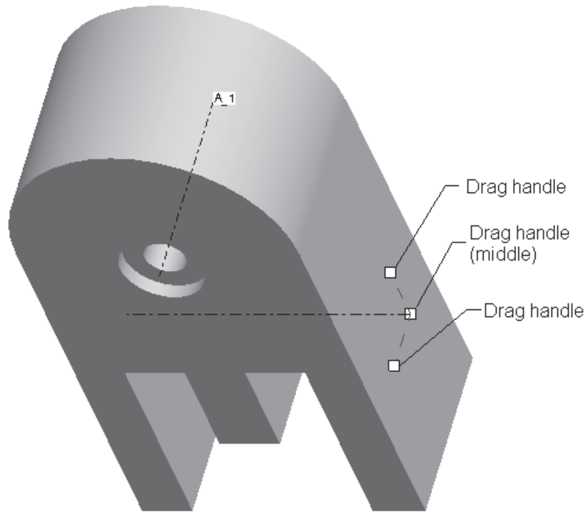


Figure 5-28 *Solid model with the datum axis and three drag handles*

to activate the **Length** edit box. By default, the **Length** edit box is in the inactive mode. You need to select the **Adjust Outline** check box to activate it. You can enter a value in this edit box or drag the handles displayed at both ends of the datum axis to modify its length.

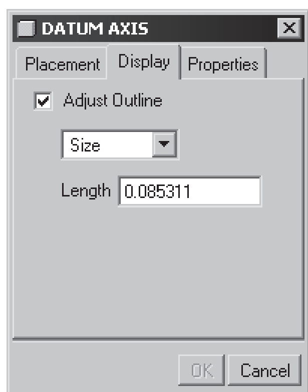


Figure 5-29 *The **DATUM AXIS** dialog box with the **Display** tab chosen*

Properties Tab

The **Properties** tab of the **DATUM AXIS** dialog box is shown in Figure 5-30. This tab, when chosen, allows you to name the datum axis you are creating. By default, the first datum axis you create is named A_1 and then additional datum axis created are successively named. As mentioned earlier, a datum axis is created using constraints which are applied automatically, and filters available in the **Filter** drop-down list located in the **Status Bar**. While creating a datum axis, the filters that are available in the **Filter** drop-down list are shown in Figure 5-31. The constraints that are used to create a datum axis are discussed next.

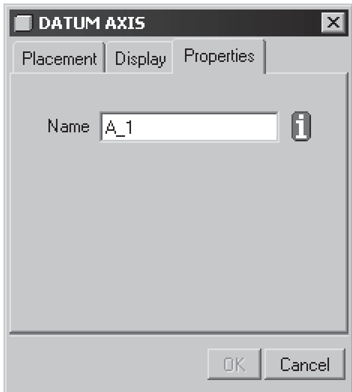


*Figure 5-30 The **DATUM AXIS** dialog box with the **Properties** tab chosen*

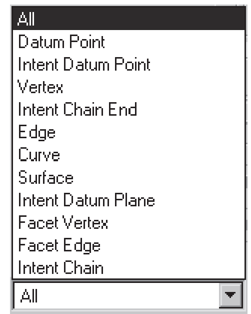


*Figure 5-31 Filters available in the **Filters** drop-down list*

Datum Axis Passing Through an Edge

The **Through** constraint is used to create a datum axis passing through any selected edge. This constraint will be displayed in the dialog box when you select an edge, as shown in Figure 5-32. In Figure 5-32, **A_1** is the datum axis created using this option.

Note

While creating a datum axis, you may need to create datum points in order to create the datum axis passing through them. Therefore, in the various cases that are discussed next you may need to create datum points. Datum points are discussed later in this chapter.

Datum Axis Normal to a Plane

The **Normal** constraint is used to create a datum axis normal to a selected face or datum plane. When you select a face or a datum plane, the **Normal** constraint will be displayed in the dialog box and you are prompted to select two references to place the axis. Notice that three drag handles appear on the model. Use the middle handle to change the location of the axis on the selected face. After you specify its placement location, you need to select two edges, axes, datums, or faces to specify the linear dimension for the placement of the datum axis. Click in the **Offset References** collector. The collector turns yellow in color. When you select the first edge for the placement dimensions of the axis, the offset value will be displayed in the **Offset References** collector. You can accept the default dimension or click on it to change its value. Similarly, select the second edge for dimensioning by pressing CTRL+left mouse button and enter the dimension value. In Figure 5-33, the preview of the datum axis with drag handles is shown. It should be noted that the references for dimensioning can also be selected dynamically using these drag handles.

Note

*When you select a reference for creating the datum axis and you are prompted to select the placement references, you need to click in the **Offset References** collector to make it yellow in color. The yellow color of this area in the **DATUM AXIS** dialog box indicates that now you can select the references to place the axis. Until the **Offset References** collector is made yellow in color, you will not be able to select the placement references.*

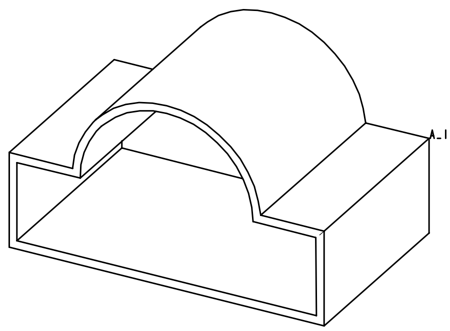


Figure 5-32 *Datum axis created along the edge*

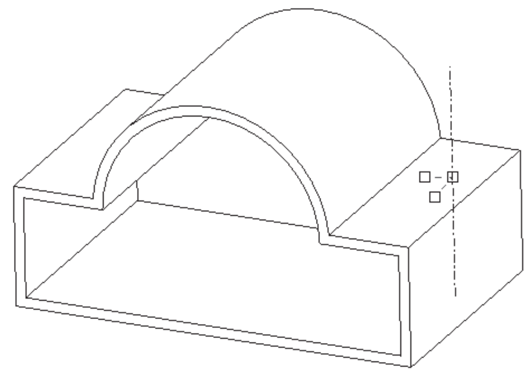


Figure 5-33 *Datum axis created normal to the plane*

Datum Axis Passing Through a Datum Point and Normal to a Plane

The **Normal** constraint creates a datum axis passing through a datum point or vertex and normal to any face or datum plane. When you select a face or a datum plane to which the datum axis will be normal, you are prompted to select two references to place the axis. Use CTRL+left mouse button to select the datum point or vertex to create an axis passing through it. Choose the **OK** button to exit the **DATUM AXIS** dialog box. In Figure 5-34, the preview of the datum axis and the datum point is shown.

Datum Axis Passing Through the Center of a Round Surface

The **Through** constraint is used to create a datum axis passing through the center of a cylindrical or round surface. To create this datum axis, select the round surface through which you need to pass the datum axis. The axis is automatically created and it passes through the center. In Figure 5-35, the preview of the datum axis is shown. The selected cylindrical surface is highlighted in this figure.

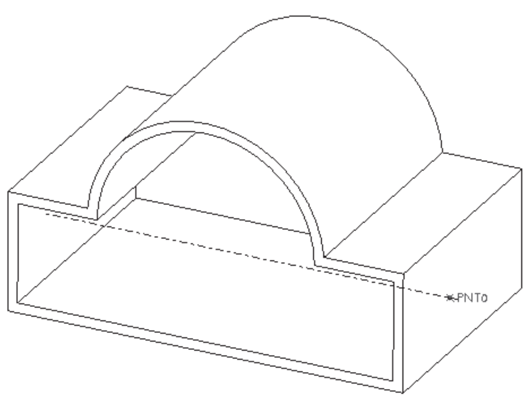


Figure 5-34 *Datum axis passing through the datum point and normal to the plane*

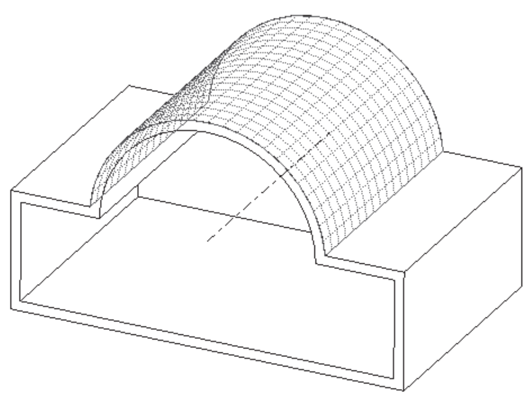


Figure 5-35 *Datum axis passing through the center of a cylinder*

Datum Axis Passing Through the Edge Formed by Two Planes

The **Through** constraint is used to create a datum axis passing through the intersection edge of two planar faces or planes. When you select the face, the **Normal** constraint is applied automatically. Click on it to open the drop-down list and select the **Through** constraint. Now, select the second face using the CTRL+left mouse button. A datum axis is created passing through the edge formed by the two faces or planes. In Figure 5-36, the preview of the datum axis is shown. The two faces that were selected are also highlighted in this figure.

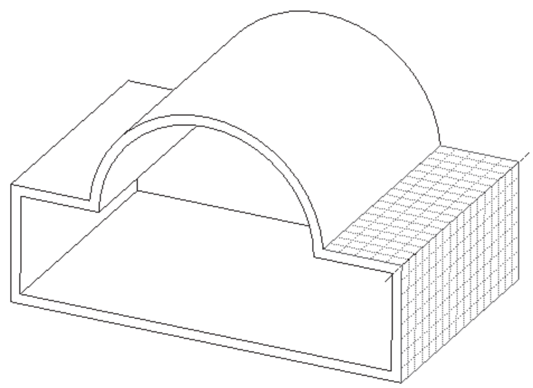


Figure 5-36 *Datum axis created on the edge where the two selected planes meet*

Datum Axis Passing Through Two Datum Points or Vertices

The **Through** constraint is used to create a datum axis between two datum points or vertices. To create this datum axis, select the first vertex and then using CTRL+left mouse button, select the second vertex. The datum axis is created along the two selected datum points or vertices. In Figure 5-37, the preview of the datum axis is shown with the two vertices highlighted.

Datum Axis Tangent to a Curve and Passing Through its Vertex

The **Tangent** and **Through** constraints create a datum axis tangent to a curve and passing through one of its vertex. Select the edge of the cylindrical surface as the first selection. Make sure you do not select the cylindrical surface. After you select a curve or an edge, use CTRL+left mouse button to select one vertex of the edge. The datum axis is created tangent to the curve and passing through its selected vertex. In Figure 5-38, the preview of the datum axis that is created using the **Tangent** and **Through** constraints is shown. The curved edge is also highlighted in this figure.

Datum Points

Datum points are imaginary points created in Pro/ENGINEER to aid in creating models, drawings, analyzing models, and so on. The uses of datum points are as follows:

1. To create datum planes and axes.

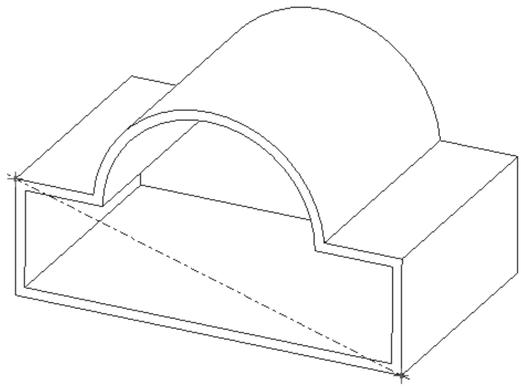


Figure 5-37 Datum axis created between the two selected vertices

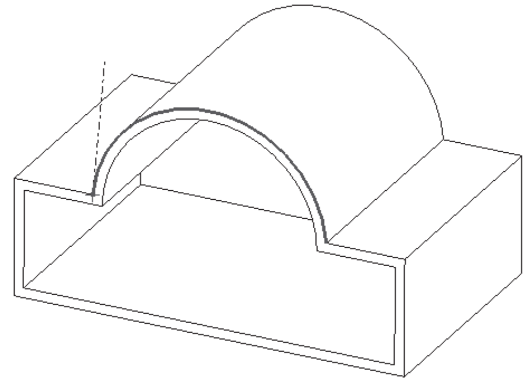


Figure 5-38 Datum axis created tangent to the selected curve

2. To associate note in the drawings and attach datum targets.

3. To create coordinate system.

4. To specify point loads for mesh generation.

5. To create pipe features.

The default name associated with a datum point in Pro/ENGINEER is **PNT(Number)** where **Number** indicates the number of datum points created in a particular model. However, you can change the default name associated with the datum points.

When you choose **Insert > Model Datum > Point** from the menu bar or the **Point** button from the **Datum** toolbar, the **DATUM POINT** dialog box is displayed with various options to create datum points, as shown in Figure 5-39. This dialog box can be used to create more than one datum point. The options in the **DATUM POINT** dialog box are discussed next.

Placement Tab

The **Placement** tab in the **DATUM POINT** dialog box has the options that change with the reference selected. However, the **References** display box shown in Figure 5-39 is always present. This area is used to select the references which aid in placing the datum point. The constraints available in the drop-down list in the **References** collector depend on the references you select from the model. This means that the references narrow down the type of constraints you can apply on the datum point that you are creating.

Tip: *While creating a datum feature, the references selected from the model help you to choose from the available constraints that you can apply to constrain the datum feature. References reduce the range of available constraints, thus reducing the time taken in selecting the appropriate constraint. For example, the lesser the number of options available for a person, the lesser the time he needs to decide among them.*

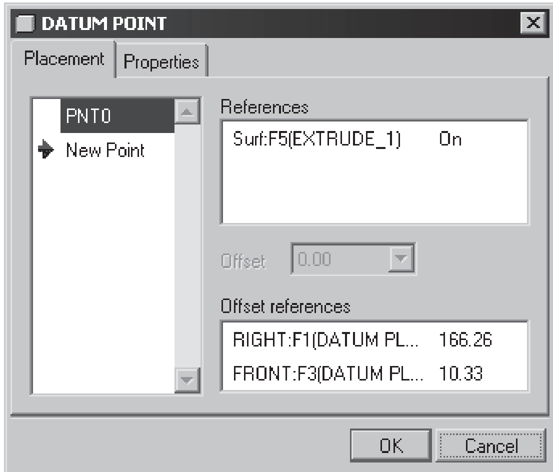


Figure 5-39 *The* ***DATUM POINT*** *dialog box with the* ***Placement*** *tab chosen*

Various cases of creating datum points are discussed next.

Datum Point on a Face or a Datum Plane

The **On** constraint is used to create datum points on a face or a datum plane. When you select a face or a datum plane to place the datum point, a yellow colored point will be displayed at the selected location on the surface with the three drag handles, as shown in Figure 5-40.

Next, click in the **Offset References** collector to make it yellow in color. You are prompted to select two references to specify the linear dimensions for the placement of the datum point. Note that the second selection to select the reference should be made by using CTRL+left mouse button. After you select the two planes or edges for the placement of dimension of the point, a default value will be displayed in the **Offset References** collector. You can accept the default value or change it to the required value. After the datum point is located at the desired position on the face or datum plane, choose the **OK** button to exit the dialog box.

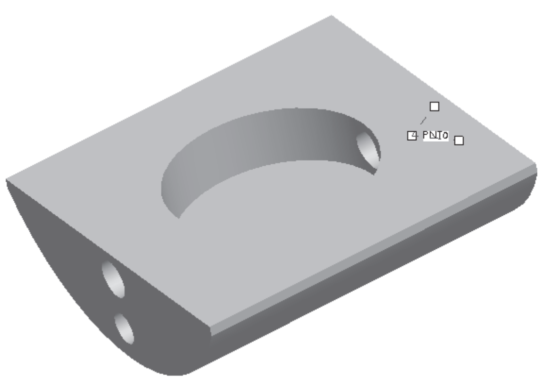


Figure 5-40 *Datum point on a face and the three drag handles*

Datum Point Offset to a Face or a Datum Plane

The **Offset** constraint creates datum points at an offset distance from a specified face or a datum plane in a specified direction. Select a face or a datum plane from where the offset distance for the placement of the datum point will be measured. The **On** constraint is applied automatically. From the drop-down list in the **References** collector, change the constraint to **Offset**. The model and the **DATUM POINT** dialog box appear, as shown in Figures 5-41 and 5-42.

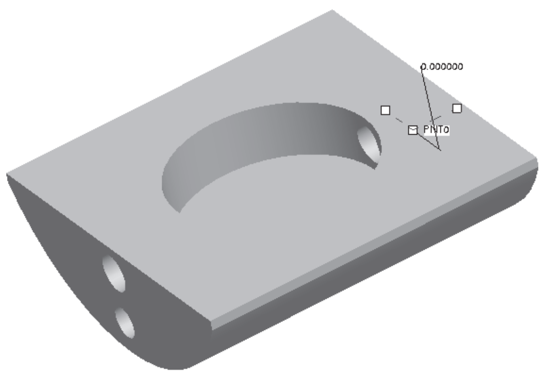


*Figure 5-41 Datum point on the top face with offset **0** and the three drag handles*

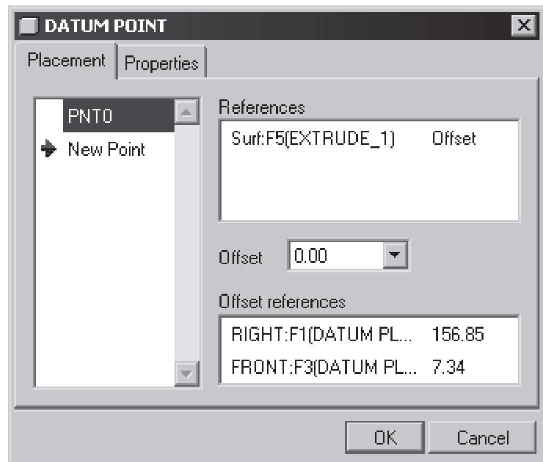


*Figure 5-42 The **DATUM POINT** dialog box with **Offset** constraint selected*

In the **Offset** edit box under the **References** collector, enter the offset value. You will be prompted to select the planes or the edges for dimensioning the point. Select two planes or edges for dimensioning and enter the distances from the highlighted references.

You can also specify the offset distance dynamically by using the middle drag handle. The two references for dimensioning the datum point can be specified by using the drag handles or selecting directly on the model.

Note

*When you use the **Offset** constraint, the middle drag handle is used to specify the offset distance and when you use the **On** constraint, it is used to specify the location of the datum point on the selected face.*

Datum Point at the Intersection of Three Surfaces

The **On** constraint is used to create a datum point at the intersection of three surfaces. To create this datum point, select the three surfaces; datum point will be created, as shown in Figure 5-43.

The datum point is created on the vertex that is common to the three surfaces. Remember that the first reference is selected using the left mouse button. The second and third references are selected using CTRL+left mouse button. After selecting the three surfaces, the **DATUM POINT** dialog box is displayed, as shown in Figure 5-44.

It might so happen that with the current selection of references, more than one intersection point exist. In such a case, use the **Next Intersection** button in the **DATUM POINT** dialog box to select the other intersection points to create the datum point on them.

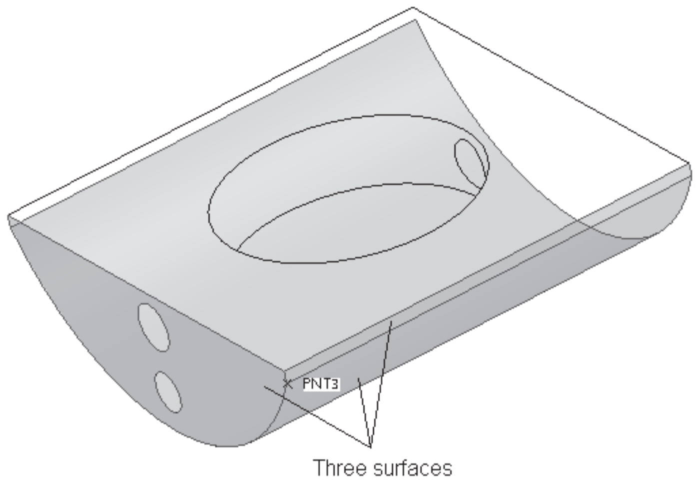


Figure 5-43 The three highlighted surfaces and the datum point at the intersection of the three surfaces

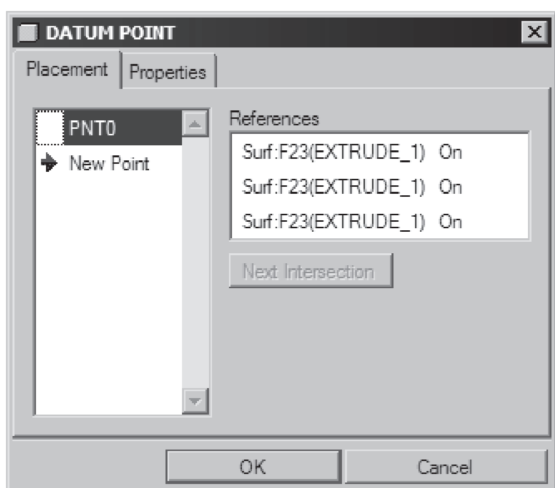


*Figure 5-44 The **DATUM POINT** dialog box*

Datum Point on a Vertex

The **On** constraint is used to create a datum point on the vertex of a face, an edge, or a datum curve. Invoke the **DATUM POINT** dialog box and then select a vertex to create the datum point. Choose the **OK** button to exit the dialog box.

Note
You can create more than one datum point by simply selecting the desired vertex.

Datum Point at the Center of a Curved Edge

The **Center** constraint creates a datum point at the center of an arc or a curved edge. Invoke the **DATUM POINT** dialog box and select the curve edge. The **On** constraint is applied by default. To modify the constraint, select the constraint in the **References** collector; the drop-down list appears to the right of the constraint. Choose the **Center** constraint from it to create the datum point at the center of the curved edge.

Datum Point on an Edge or a Curve

The **On** constraint is used to create a datum point on an edge or a curve. Invoke the **DATUM POINT** dialog box, and select an edge or a curve. The datum point is placed on the selected edge, as shown in Figure 5-45 and the **DATUM POINT** dialog box appears, as shown in Figure 5-46.

In the **DATUM POINT** dialog box, after the point is placed, the dimension type can be defined. The **Offset** dimension box displays the offset distance of the datum point from one end of the edge. The offset distance is measured as a ratio or as a real value from the end of the edge, which can be selected from the drop-down list, as shown in Figure 5-46.

In the **Offset References** area, the **Next End** button is used to flip the end of the edge from where the offset distance is measured. This button is available only when the **End of curve** radio button is selected. If you select the **Reference** radio button, you need to select a reference from which the datum point will be dimensioned.

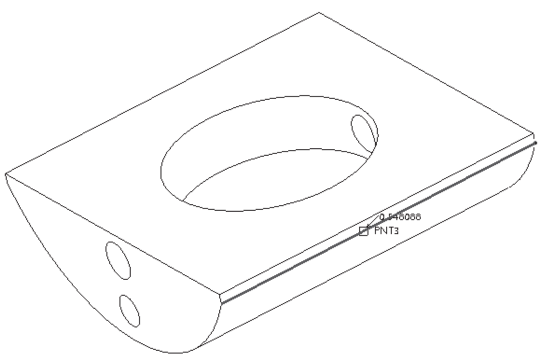


Figure 5-45 The highlighted edge and the datum point at some offset distance from one end of edge

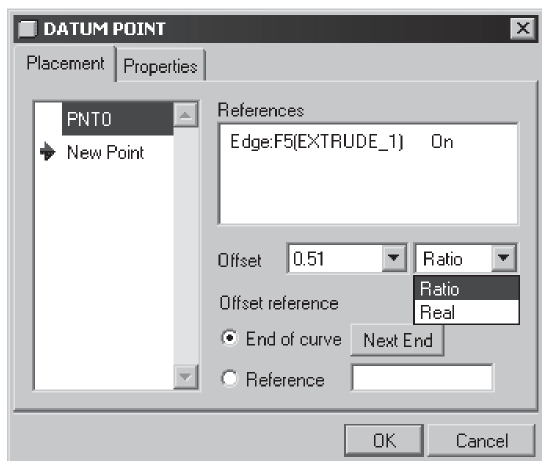
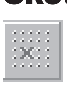


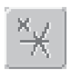
Figure 5-46 The ***DATUM POINT*** *dialog box*

Sketched Datum Point Button



When you choose the black arrow on the right of the **Point** button, a flyout will be displayed. Choose the **Sketched** button from this flyout; the **Sketch Datum Point** dialog box will be displayed. The function of this dialog box is same as the **Sketch** dialog box. Using the **Sketch Datum Point** dialog box, select the sketching plane and its orientation. When you choose the **Sketch** button, the system takes you to the sketcher environment. In the sketcher environment, you can use the construction tools and the **Point** button to sketch the datum points. The datum points are dimensioned with the references. You can also modify these dimensions. Remember that you can exit the sketcher environment only if at least one point is drawn.

To Create an Array of Datum Points from a Coordinate System



The **Offset Coordinate System** button is used to create an array of datum points at an offset distance from a coordinate system. You can change the array of the points by redefining the array. This button is available on the flyout. Note that the Datum Coordinate system must be defined before you create an array of datum points.

When you choose this button, the **Offset CSys Datum Point** dialog box will be displayed, as shown in Figure 5-47 and you are prompted to select a coordinate system. After selecting a coordinate system, you can enter the values of the coordinates for the datum points.

To add a point, click under the **Name** column in the list box; the first row gets activated. Click under the **XAxis** column; the edit box appears in which you can enter an offset value. Similarly, enter the offset values for **Y** and **ZAxis.** To add another point click in the next row. From the **Type** drop-down list, select the type of coordinate system: Cartesian, Cylindrical, or Spherical. In Figure 5-47, the coordinates of three datum points are entered in the **Offset CSys Datum Point** dialog box.

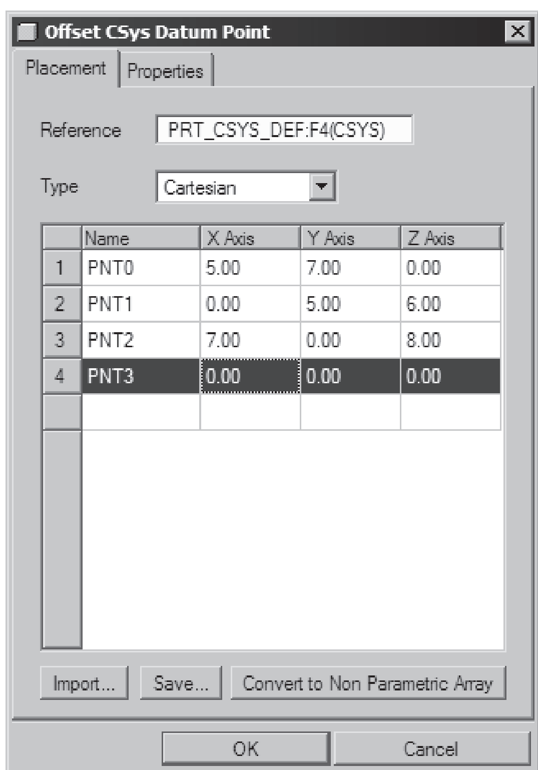


Figure 5-47 *The **Offset CSys Datum Point** dialog box*

To Create a Datum Point by Clicking

When you choose the **Field** button from the flyout, the **Field Datum Point** dialog box will be displayed, as shown in Figure 5-48. Using this dialog box, you can create a datum point anywhere on the surface of the model. You just need to specify the location using the left mouse button to place a datum point.

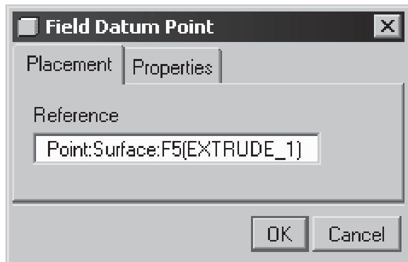


Figure 5-48 *The* ***Field Datum Point*** *dialog box*

CREATING CUTS

Cut is a material removal process and this option is available only when at least one base feature exists in the drawing area. The **Remove Material** button available on the dashboard of the feature creation tools is used to create the cuts. In this chapter, you will learn the extrude cut and the revolve cut. The procedure to create a cut on an existing feature is similar to that of adding material or protrusion.

Removing Material by Extruding a Sketch

The **Extrude Cut** is an extruded feature that is created by removing material from an existing feature. The material that is removed is defined by a sketch.

After drawing the sketch for the cut feature, you can specify the direction of material removal with respect to the sketch. For example, the yellow arrow in Figure 5-49 shows the direction of material removal. If the direction shown by the arrow is accepted, then the cut feature will be created, as shown in Figure 5-50.

However, if you choose the **Change material direction of extrude to other side of sketch** button from the **Extrude** dashboard, the arrow points in the direction shown in Figure 5-51. Entire material on the plane selected for sketching will be removed, leaving the extruded cut feature, as shown in Figure 5-52.

Note

A straight hole can also be created by drawing its cross-section, that is a circle, and then creating an extrude cut. But, Pro/ENGINEER provides predefined placement for a hole feature, which can be more desirable than dimensioning the cross-section of a cut feature. Straight holes do not require a sketch if you use the ***Hole*** *dashboard. The* ***Hole*** *dashboard is discussed in Chapter 6.*

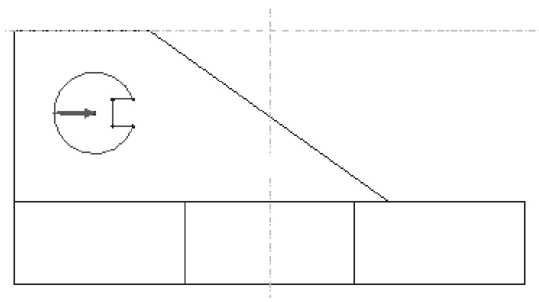


Figure 5-49 *Sketch for the extrude cut and arrow showing the direction of material removal*

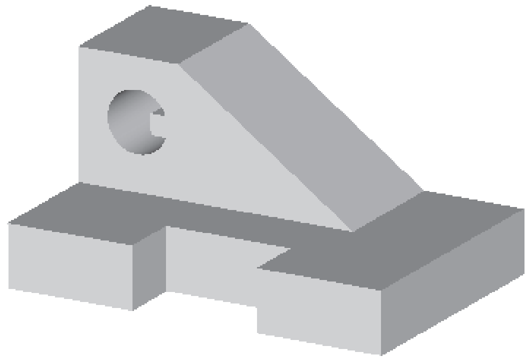


Figure 5-50 *Cut feature created on the selected plane*

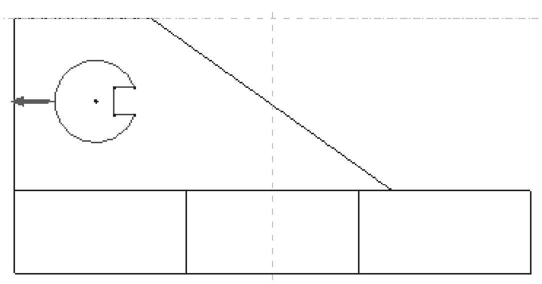


Figure 5-51 *Arrow showing the direction of material removal*

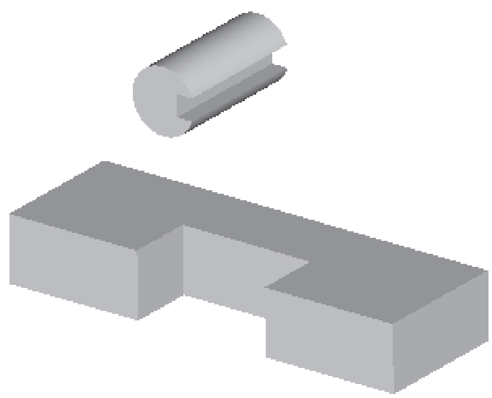


Figure 5-52 *Cut feature created in the direction shown in the adjacent figure*

Tip: *In the model shown in Figure 5-49, the sketching plane selected for creation of the extruded cut is not a datum plane but the front face of the base feature. You can also create a datum plane on the surface of an existing feature and select it as the sketching plane. But it is not recommended to create a datum plane if a planar surface of the feature can be used as a sketching plane.*

Removing Material by Revolving a Sketch

The **Revolve Cut** is a revolved feature that is created by removing material from an existing feature. The material that is removed is defined by the sketch you draw. Remember that the center line is necessary in a revolve features. Figure 5-53 shows the section drawn that is to be revolved. The front surface of the second extruded feature is selected as the sketching plane. Figure 5-54 shows the revolve cut created on the selected surface.

Note

The ***Sweep Cut*** *is explained in Chapter 8.*

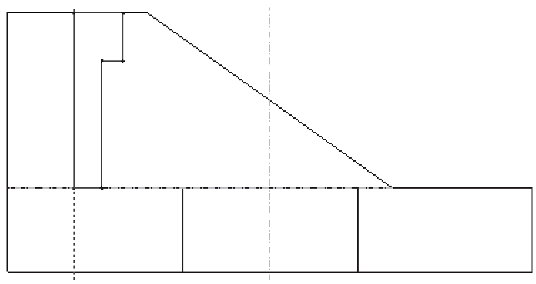


Figure 5-53 *The section for revolve cut*

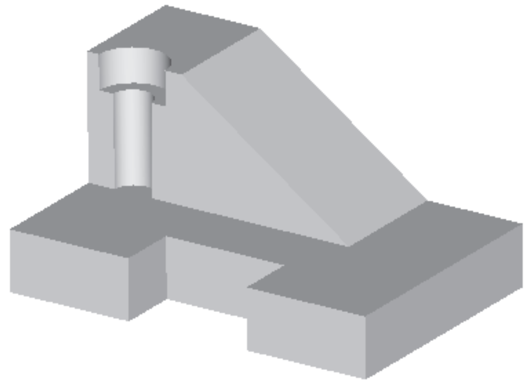


Figure 5-54 *Revolve cut created*

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 5-55. The front and the right-side views of the solid model are shown in Figure 5-56. **(Expected time: 30 min)**

Figure 5-55 *Model for Tutorial 1*

The following steps are required to complete this tutorial:

a. Examine the model and determine the number of features in it, refer to Figure 5-55.
b. Create the base feature, refer to Figures 5-57 and 5-58.
c. Create the second extrude feature that is at the bottom of the base feature, refer to Figures 5-59 through 5-62.
d. Create the third feature on an offset plane, refer to Figures 5-63 through 5-68.
e. Create the circular cut feature, refer to Figures 5-69 through 5-72.

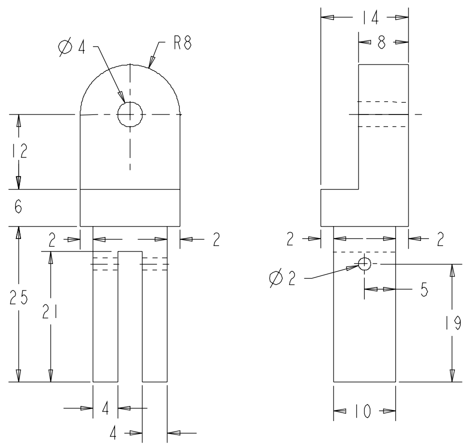


***Figure 5-56** Front and side views of the model*

Setting the Working Directory

When Pro/ENGINEER session is started, the first task is to set the working directory. A working directory is a directory on your system where you can save the work done in the current session of Pro/ENGINEER. You can set any existing directory on your system as the working directory.

1. Choose the **Set Working Directory** option from the **File** menu; the **Select Working Directory** dialog box is displayed. Select *C:\ProE-WF-4.0* folder.

2. Choose the **Organize** tab from the **Select Working Directory** dialog box to display the flyout. Next, choose the **New Folder** option from the flyout; the **New Folder** dialog box is displayed.

3. Enter *c05* in the **New Directory** edit box and choose **OK** from the dialog box. You have created a folder named *c05* in *C:\ProE-WF-4.0*.

4. Next, choose **OK** from the **Select Working Directory** dialog box. The working directory is set to *C:\ProE-WF-4.0\c05*. A message **Successfully changed to *C:\ProE-WF-4.0\c05* directory** is displayed in the message area.

Starting a New Object File

1. Open a new part file and name it as *c05tut1*. The three default datum planes are displayed in the drawing area. The **Model Tree** also appears on the left in the Navigator. Close the **Model Tree** by clicking on the sash on its right edge.

Selecting the Sketching Plane for the Base Feature

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

Note

The model can be created by selecting any plane as the sketching plane for the base feature. But when the base feature is created, the orientation of the base feature will not be proper. Hence, the final model will be oriented in the wrong direction. You will have to be careful while defining the sketching plane for the base feature. The desired orientation of the model is shown in Figure 5-55.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed above the graphics window.

2. Choose the **Placement** tab and from the slide-down panel, choose the **Define** button; the **Sketch** dialog box is displayed.

3. Select the **FRONT** datum plane as the sketching plane.

   A yellow arrow is displayed on the **FRONT** datum plane, pointing in the direction of viewing the sketch.

4. Select the **TOP** datum plane from the drawing area and then select the **Top** option from the **Orientation** drop-down list.

   The **TOP** datum plane is selected in order to orient the sketching plane.

5. Choose the **Sketch** button from the **Sketch** dialog box. Now, you enter the sketcher environment.

Creating and Dimensioning the Sketch for the Base Feature

The base feature can be created by drawing the sketch and then extruding it to the given distance.

1. Draw the sketch using various sketcher tools and add the required constraints and dimensions shown in Figure 5-57. When you initially draw the sketch, it is dimensioned automatically and some weak dimensions are assigned to it.

   Tip: *It is recommended that you use the **Modify** button to modify the weak dimensions. In the **Modify Dimensions** dialog box that appears, clear the **Regenerate** check box and then modify the dimensions using the thumbwheel or the dimension edit box. This way the sketch will not regenerate as you edit dimensions.*

2. Modify the dimension values to those shown in Figure 5-57.

3. After the sketch is completed, choose the **Done** button. Now, you are out of the sketcher environment.

4. Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Choose the **Default Orientation** option from the flyout; the model orients in its default orientation, that is, trimetric view, as shown in Figure 5-58. The yellow colored arrow is displayed on the model, indicating the direction of extrusion.

5. In the dimension box present on the **Extrude** dashboard, enter the value **8**, the model appears, as shown in Figure 5-58.

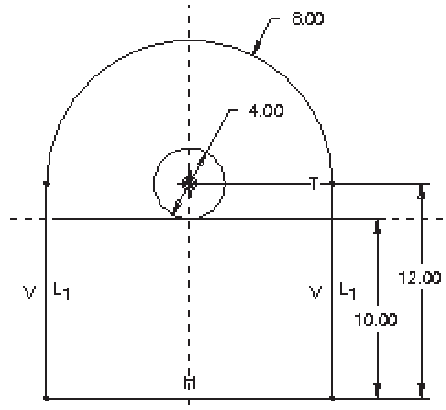


Figure 5-57 Sketch for the base feature with dimensions and constraints

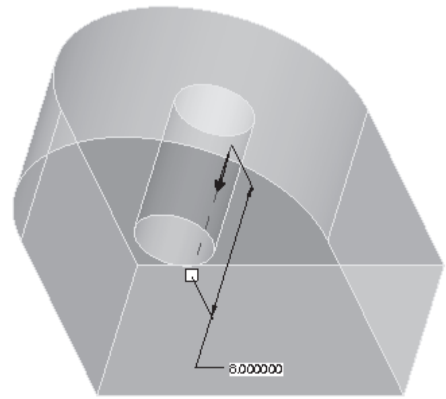


Figure 5-58 Default orientation of the model and the arrow showing the direction of feature creation

6. Choose the **Build feature** button from the **Extrude** dashboard. The base feature is completed. You can use the middle mouse button to spin the model in order to view it from various directions.

Note

*When you choose the **Default Orientation** option from the **Named View List** button, the orientation of the model is trimetric and not isometric. If you want the isometric view of the model to be displayed whenever you choose the **Default** option, then you have to use the **Environment** dialog box. The **Environment** dialog box is displayed when you choose **Tools > Environment** from the menu bar. From the dialog box, in the **Standard Orient** drop-down list, choose the **Isometric** option. Now, the default orientation will be set to isometric.*

Tip: *It is recommended to check the orientation of the base feature of a model when it is completed. To check whether the plane you specified for sketching was correct or not, choose the **Named View List** button from the **View** toolbar a flyout is displayed. Next, choose the **FRONT** option from the flyout; the base feature will reorient in the drawing area such that you can view the front view of the base feature.*

Selecting the Sketching Plane for the Second Feature

The second feature is an extrude feature and will be created on the previous plane that was used to create the base feature.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed above the drawing area.

2. Choose the **Placement** tab and from the slide-down panel, choose the **Define** button; the **Sketch** dialog box is displayed.

3. Choose the **Use Previous** button from the **Sketch Plane** area in the **Sketch** dialog box; the sketcher environment is activated automatically.

   When you choose the **Use Previous** button, the system selects the previous sketching plane that was used to create the base feature. This option is selected because the base feature and the second feature are on the same plane, but have different depths of extrusion. If they had the same depth of extrusion, you could have drawn them on the same plane as a single feature. The **TOP** datum plane and its orientation is set automatically.

Drawing the Sketch for the Second Feature

The second feature has a rectangular section that will be extruded to a depth of 14. To improve the clarity of the edges of the base feature, choose the **No hidden** button from the **Model Display** toolbar before you start sketching the second feature.

1. Draw the sketch, as shown in Figure 5-59.

2. The sketch is automatically constrained and some weak dimensions are assigned to it. Add the required constraints and modify the weak dimensions, as shown in Figure 5-59.

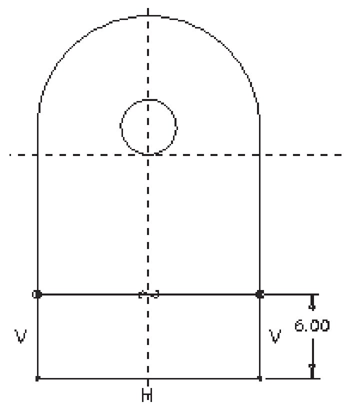


Figure 5-59 *Sketch for the second feature*

Tip: *You can use the **Use** button from the **Sketcher Tools** toolbar to use the bottom edge of the base feature. The edge of the base feature is required to complete the sketch for the second feature. Or else, draw an aligned line on the edge.*

3. Choose the **Done** button to exit the sketcher environment. The **Extrude** dashboard is enabled above the drawing area. Choose the **Shading** button from the **Model Display** toolbar to view the shaded model.

4. Use the middle mouse button to orient the model, as shown in Figure 5-60. This orientation of the model gives you a better view of the sketch in the three-dimensional

(3D) space. The yellow colored arrow is also displayed on the model, indicating the direction of extrusion.

5. Enter the value **14** in the dimension box present on the **Extrude** dashboard and press ENTER. The second extruded feature is completed and its preview is displayed in the drawing area.

6. Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Choose the **Default Orientation** option from the flyout; the model orients in its default orientation, that is, trimetric view.

7. Now, choose the **Build feature** button from the **Extrude** dashboard to confirm the feature creation. The model orients on the screen, as shown in Figure 5-61. You can also use the middle mouse button to spin the model.

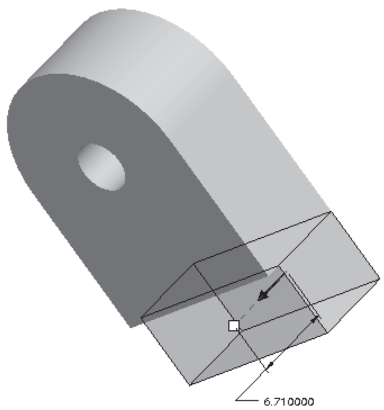


Figure 5-60 *Arrow showing the direction of feature depth*

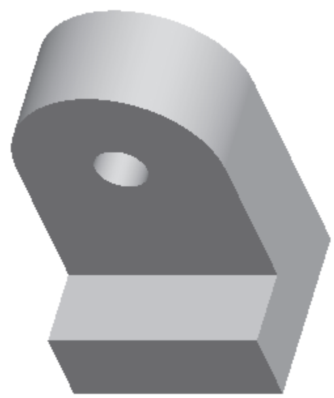


Figure 5-61 *Second extruded feature with the base feature*

Creating the Datum Plane for the Third Feature

A new datum plane is required to create the next feature. The datum plane will be created at an offset distance of 2 from the front face of the second feature. You need to turn on the display of datum planes if their display is off.

1. Choose the **Plane** button from the **Datum** toolbar; the **DATUM PLANE** dialog box is displayed.

   Make sure that before opening the dialog box, no selections are made. This can be confirmed from the **Status Bar** that is available below the drawing area. If there is a selection, you can remove it by left-clicking in the empty space of the drawing area.

2. Using the left mouse button, select the front face of the second feature highlighted in Figure 5-62. The boundary of the selected front face is highlighted in red color.

   In the **DATUM PLANE** dialog box, under the **References** collector, the **Offset** constraint is displayed. This means that by the smart selection property of Pro/ENGINEER Wildfire 4.0, the **Offset** constraint is applied automatically to the datum plane you are going to create.

Now, you need to specify the offset distance. If you enter a positive value, the datum plane will be created in the direction shown by the arrow and if you enter a negative value then the datum plane will be created in the direction opposite to that shown by the arrow.

3. In the **Translation** dimension box, under the **Offset** area of **DATUM PLANE** dialog box, enter the value **-2** and press ENTER.

   The negative value is entered because the datum plane has to be created in the direction opposite to that shown by the yellow arrow.

4. Choose the **OK** button. The datum plane named **DTM1** is created, as shown in Figure 5-63.

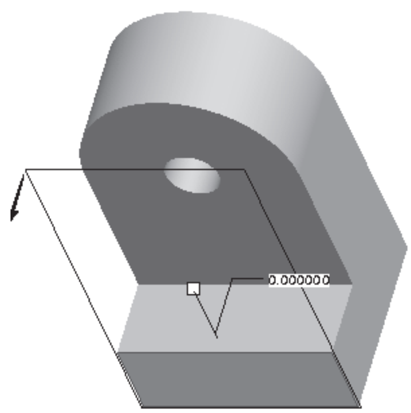


Figure 5-62 The selected face and the arrow showing the positive direction of datum plane

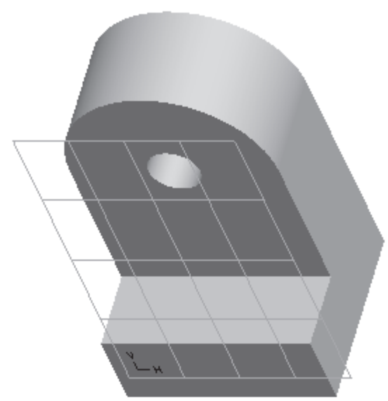


Figure 5-63 The highlighted datum plane

Creating the Third Feature on DTM1

The datum plane **DTM1** is created and can be seen in the **Model Tree** as well as in the drawing area. The sketch of the next feature that will be extruded has to be created on the datum plane **DTM1**.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed above the drawing area.

2. Choose the **Placement** tab and from the slide-down panel, choose the **Define** button; the **Sketch** dialog box is displayed.

3. Select **DTM1** as the sketching plane for the third feature. A yellow arrow is displayed on the selected datum plane, as shown in Figure 5-64. This arrow shows the direction of viewing the sketching plane.

   The **TOP** datum plane and its orientation is selected by default.

4. Choose the **Sketch** button from the **Sketch** dialog box. The system takes you to the sketcher environment. Choose the **No hidden** button from the **Model Display** toolbar.

5. Sketch the section for the third feature of the model and add constraints and dimensions to the sketch, as shown in Figure 5-65.

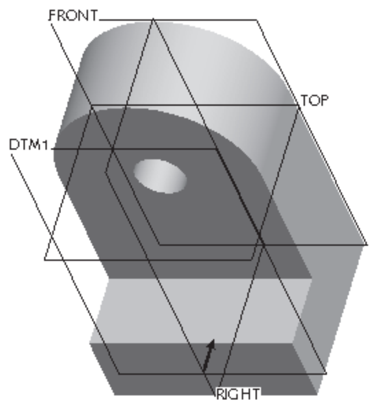


***Figure 5-64** Arrow on **DTM1** showing the direction of viewing the sketching plane*

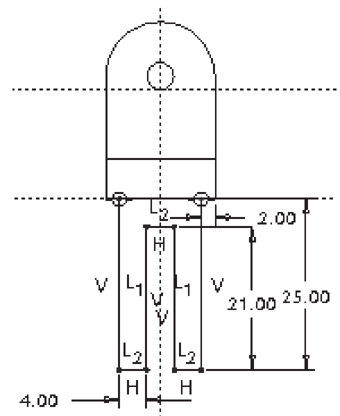


***Figure 5-65** Sketch with dimensions and constraints of the third feature*

6. Choose the **Done** button to exit the sketcher environment.

   The **Extrude** dashboard is enabled above the drawing area. Turn the model display to **Shading**. Use the middle mouse button to orient the model, as shown in Figure 5-66. This orientation gives a better view of the model.

   Notice that the yellow arrow is pointing toward the direction of feature creation. But, you need to extrude the sketch in the opposite direction.

8. Choose the **Change depth direction of extrude to other side of sketch** button from the **Extrude** dashboard to change the direction of arrow. 

9. Enter a value of **10** in the dimension box present on the **Extrude** dashboard. The preview of the third feature is displayed in the drawing area.

10. Now, choose the **Build feature** button from the **Extrude** dashboard to confirm the feature creation.

    Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Choose the **Default Orientation** option from the flyout; the model orients in its default orientation, that is, trimetric view, as shown in Figure 5-67.

Selecting the Sketching Plane for the Cut Feature

A through cut will be created on the outer right face of the third feature. The circular section for the cut will be sketched and using the **Remove Material** button in the

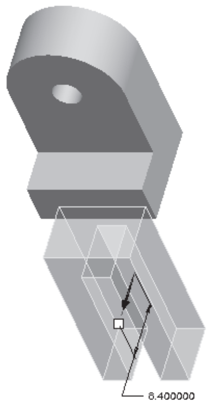


Figure 5-66 Arrow showing the direction of material addition

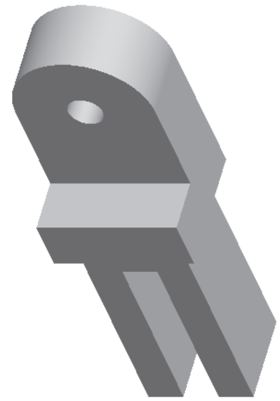


Figure 5-67 Model after creating the third feature

Extrude dashboard, the circular cut will be created. The sketching plane for the cut feature is shown in Figure 5-68.

Note

*The circular cut feature can also be created using the **Hole** tool, which will be discussed in Chapter 6.*

1. Choose the **Extrude** button from the **Base Features** toolbar. The **Extrude** dashboard is displayed above the drawing area.
2. Choose the **Remove Material** button from the **Extrude** dashboard.
3. Choose the **Placement** tab, and from the slide-down panel choose the **Define** button; the **Sketch** dialog box is displayed.
4. Select the face shown in Figure 5-68 for sketching.
5. Using the left mouse button, select the **TOP** default datum plane and then select the **Top** option from the **Orientation** drop-down list.
6. Choose the **Sketch** button; the system takes you to the sketcher environment.

Sketching the Cut Feature

1. Turn the model display to **No hidden**. Draw the sketch of the cut feature and add dimensions to it, as shown in Figure 5-69.
2. Choose the **Done** button and turn the model display to **Shading**.
3. Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Choose the **Default Orientation** option from the flyout; the model orients in its default orientation, that is, trimetric view, as shown in Figure 5-70. Two yellow arrows also appear on the model. One arrow indicates the direction of feature creation and the other arrow

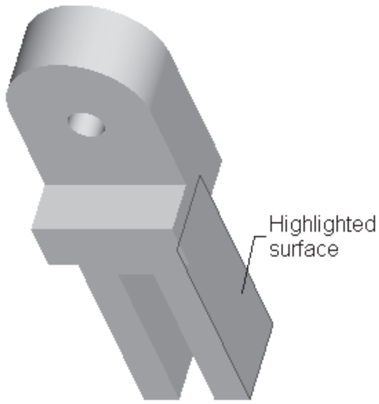


Figure 5-68 Sketching plane for the hole feature

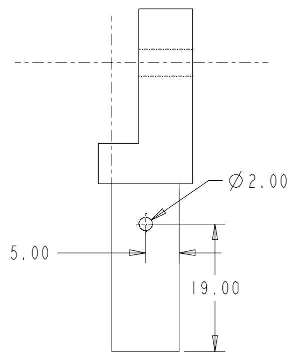


Figure 5-69 Sketch and dimensions for cut

indicates the direction with respect to the sketch from where the material will be removed. Figure 5-70 shows the direction where the arrows are pointing.

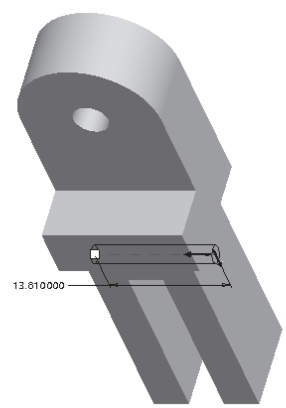


Figure 5-70 The two arrows on the cut feature

4. Choose the **Options** tab in the **Extrude** dashboard; the **Depth** slide-down panel is displayed.

5. From the **Side 1** drop-down list, choose the **Through All** option. The cut feature is completed and can now be previewed in the drawing area.

6. Choose the **Build feature** button from the **Extrude** dashboard. Turn the model display to shaded by choosing the **Shading** button from the **Model Display** toolbar. The trimetric view of the model is shown in Figure 5-71.

Saving the Model

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

   The order of feature creation can be seen from the **Model Tree** shown in Figure 5-72.

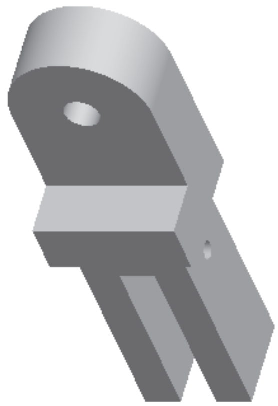


Figure 5-71 *Completed model for Tutorial 1*

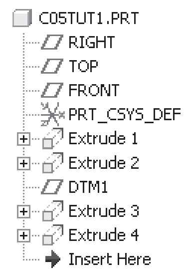


Figure 5-72 ***Model Tree*** *for Tutorial 1*

Tutorial 2

In this tutorial, you will create the model shown in Figure 5-73. This figure also shows the front, top, and right-side views of the solid model. **(Expected time: 45 min)**

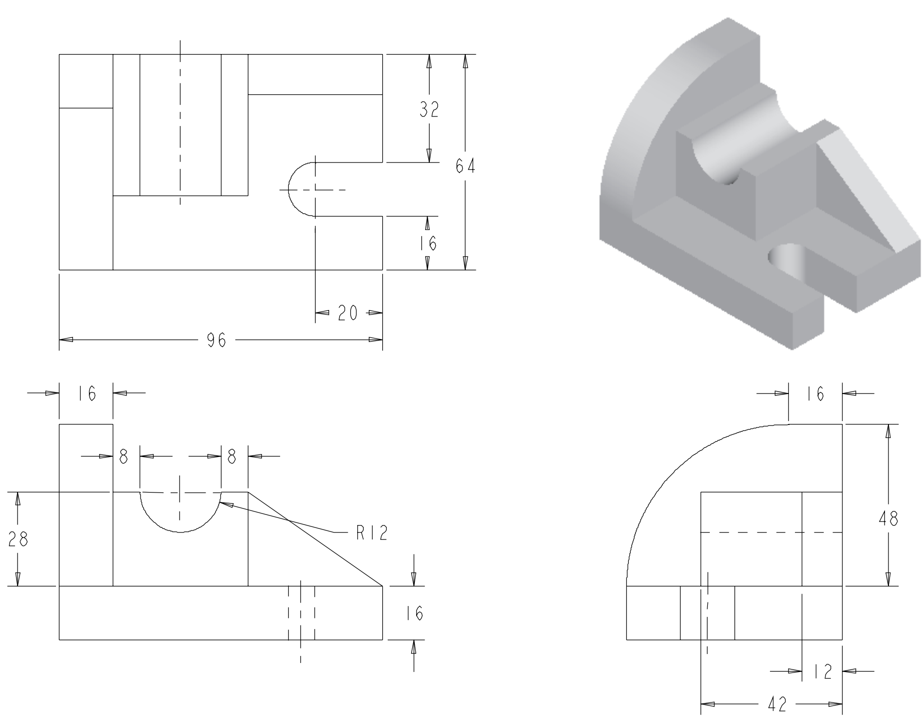


Figure 5-73 *Top, front, right-side, and isometric views of the model*

The following steps are required to complete this tutorial:

a. Examine the model and determine the number of features in it, refer to Figure 5-73.
b. Create the base feature, refer to Figures 5-74 and 5-75.
c. Create the second feature on the left face of the base feature, refer to Figures 5-76 through 5-80.
d. Create the third and fourth features on the same plane, but with different extrusion depths, refer to Figures 5-81 through 5-88.

After understanding the procedure for creating the model, you are now ready to create it. Set the working directory, if required.

Starting a New Object File

1. Start a new part file and name it as *c05tut2*. The three default datum planes are displayed in the drawing area.

Selecting the Sketching Plane for 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 **TOP** datum plane. This is because the direction of extrusion of the base feature is perpendicular to the **TOP** datum plane.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed above the drawing area.

2. Choose the **Placement** tab and from the slide-down panel choose the **Define** button; the **Sketch** dialog box is displayed.

3. Select the **TOP** datum plane as the sketching plane; a yellow arrow is displayed on the **TOP** datum plane, pointing in the direction of viewing the sketch. The **RIGHT** datum plane and its orientation is selected automatically.

4. Choose the **Sketch** button in the **Sketch** dialog box to enter the sketcher environment.

Creating and Dimensioning the Sketch for the Base Feature

The sketch for the base feature consists of a rectangular shape with a slot, as shown in Figure 5-74. When this sketch is extruded, it will create the base feature with the slot, as shown in Figure 5-75.

1. Draw the sketch using various sketcher tools and add the required constraints and dimensions to it. Modify these dimensions, as shown in Figure 5-74.

2. Choose the **Done** button to exit the sketcher environment.

3. Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Choose the **Default Orientation** option from the flyout; the model orients in its default orientation, that is, trimetric view.

The default view of the model is displayed, but it does not fit on the screen. You may need to zoom for displaying the model properly in the drawing area. This gives you a better view of the sketch in the 3D space. A yellow arrow is also displayed on the model, indicating the direction of extrusion.

4. Enter a value of **16** in the dimension box present on the **Extrude** dashboard. The preview of the base feature is displayed in the drawing area.

5. Choose the **Build feature** button from the **Extrude** dashboard. The base feature is completed and is shown in Figure 5-75. You can use the middle mouse button to spin the object to view it from various directions.

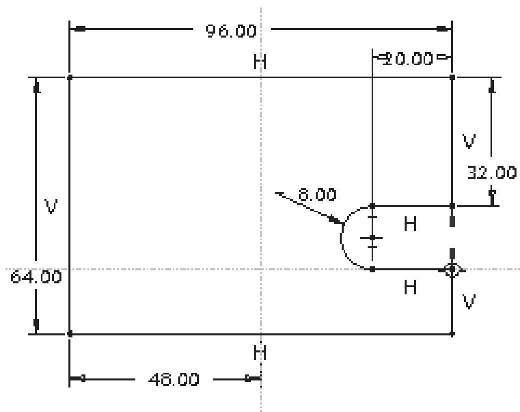


Figure 5-74 *Sketch with dimensions and constraints for the base feature*

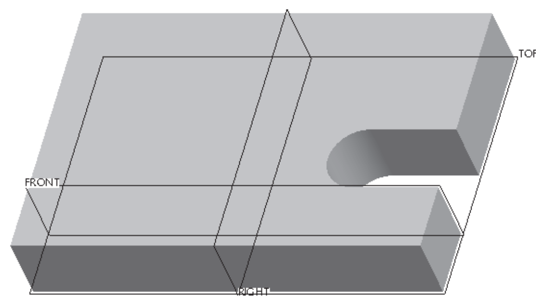


Figure 5-75 *Base feature of the model*

Selecting the Sketching Plane for the Second Feature

The next feature is an extruded feature and will be created on the left face of the base feature. Therefore, you need to select the left face of the base feature as the sketching plane and then draw the sketch.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.

2. Choose the **Placement** tab and from the slide-down panel choose the **Define** button; the **Sketch** dialog box is displayed.

3. Use the middle mouse button to spin the model, as shown in Figure 5-76.

4. Now, select the left face of the base feature as the sketching plane. A yellow arrow that points in the direction of viewing the sketching plane appears on the left face of the base feature.

5. Choose the **Flip** button to flip the yellow arrow and to change the direction of viewing the sketching plane, refer to Figure 5-76.

6. Select the top face of the base feature shown in Figure 5-77 and choose the **Top** option from the **Orientation** drop-down list.

   By selecting the top surface of the base feature, the model will be oriented in such a way that the highlighted planar surface will be at the top while sketching.

7. Choose the **Sketch** button from the **Sketch** dialog box to enter the sketcher environment.

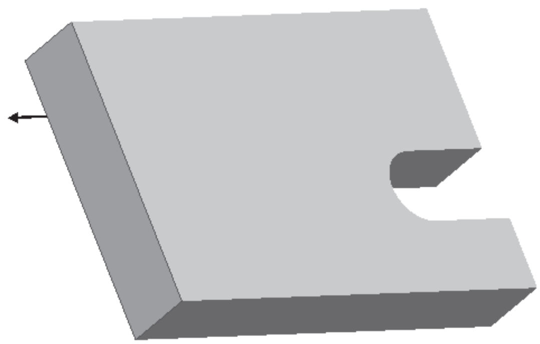


Figure 5-76 *Arrow showing the direction of viewing the model*

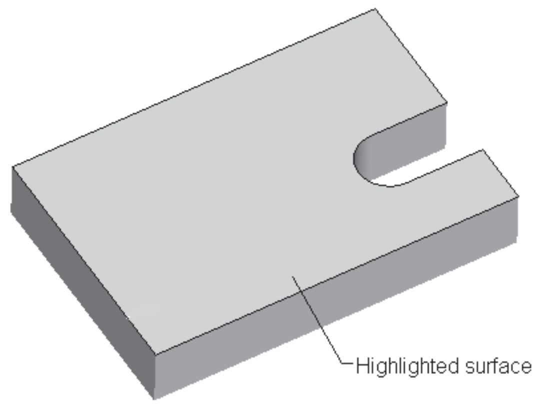


Figure 5-77 *Surface selected to be at the top*

Creating and Dimensioning the Sketch for the Second Feature

The sketch for the second feature consists of two lines and an arc. The bottom edge of the sketch coincides with the top edge of the base feature. This sketch is extruded to a depth of 16. Before drawing the sketch, turn the model display to **No hidden**.

1. Draw the sketch using various sketcher tools, as shown in Figure 5-78. The sketch is dimensioned automatically and some weak dimensions are assigned to it.

2. Apply the constraints and modify the weak dimensions to the dimensions shown in Figure 5-78.

3. After the sketch is complete, turn the model display to **Shading** and choose the **Done** button.

   The **Extrude** dashboard is displayed above the drawing area. Use the middle mouse button to orient the model, as shown in Figure 5-79. A yellow arrow is also displayed on the model, indicating the direction of extrusion.

4. Enter the value **16** in the dimension box present in the **Extrude** dashboard and press ENTER. The second feature is completed and its preview is displayed on the screen.

5. Choose the **Build feature** button from the **Extrude** dashboard. The second feature is completed and is shown in Figure 5-80. You can use the middle mouse button to spin the model to view it from various directions.

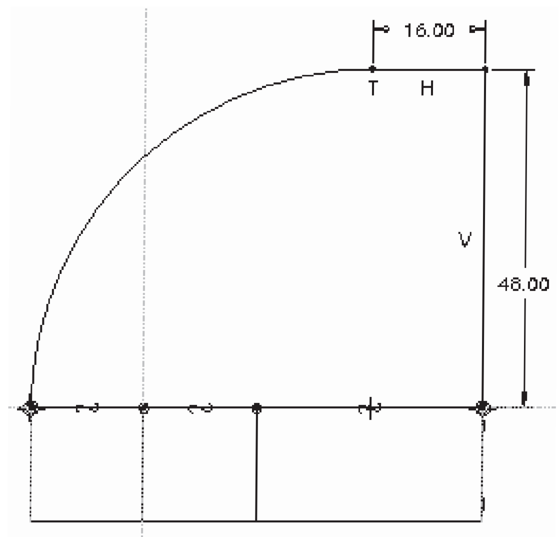


Figure 5-78 *Sketch with dimensions and constraints for the second feature*

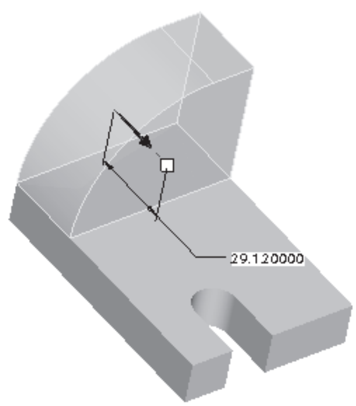


Figure 5-79 *Arrow showing the direction of material addition*

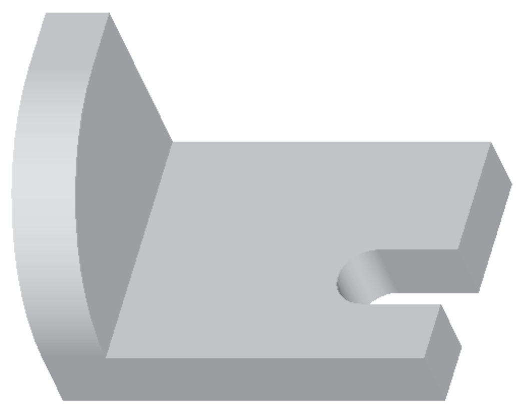


Figure 5-80 *Model with the second extruded feature*

Selecting the Sketching Plane for the Third Feature

The third feature is an extruded feature and will be created on the back planar surface of the base feature. Therefore, you need to define the back face of the base feature as the sketching plane and then draw the sketch.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed above the drawing area.

2. Choose the **Placement** tab and from the slide-down panel choose the **Define** button; the **Sketch** dialog box is displayed.

3. Use the middle mouse button to spin the model and then select the face of the base feature shown in Figure 5-81 as the sketching plane. A yellow arrow is displayed on the selected face.

4. Choose the **Flip** button to flip the yellow arrow, as shown in Figure 5-81. The arrow points in the direction of viewing the sketching plane.

5. Select the top face shown in Figure 5-82 and then select the **Top** option from the **Orientation** drop-down list. Now, the top face of the base feature is selected to be at the top while drawing the sketch.

6. Choose the **Sketch** button to enter the sketcher environment.

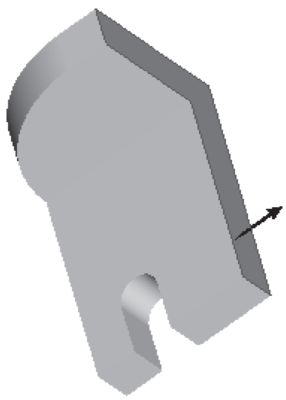


Figure 5-81 *Arrow showing the direction of viewing the model*

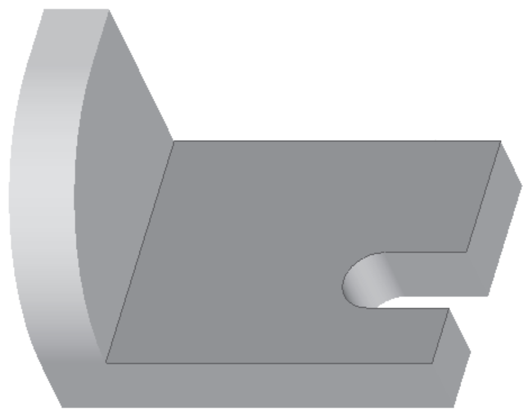


Figure 5-82 *Face selected to be at the top*

Creating and Dimensioning the Sketch for the Third Feature

The sketch for the base feature consists of a rectangular section with a semicircular cut at the top. When this section is extruded, a feature with a semicircular slot will be created. Turn the display to **No hidden**.

1. Draw the sketch using various sketcher tools, as shown in Figure 5-83.

2. The sketch is dimensioned automatically and some weak dimensions are assigned to it. Add the required constraints and modify the weak dimensions to the dimensions shown in Figure 5-83.

3. Choose the **Done** button to exit the sketcher environment. The **Extrude** dashboard is enabled above the drawing area. Turn the display to **Shading**.

4. Use the middle mouse button to orient the model, as shown in Figure 5-84. This orientation of the model gives a better view of the sketch in 3D space. A yellow arrow is displayed on the model, indicating the direction of extrusion.

5. Enter the value **42** in the dimension box present in the **Extrude** dashboard and press ENTER. This feature is completed and its preview can be seen in the drawing area.

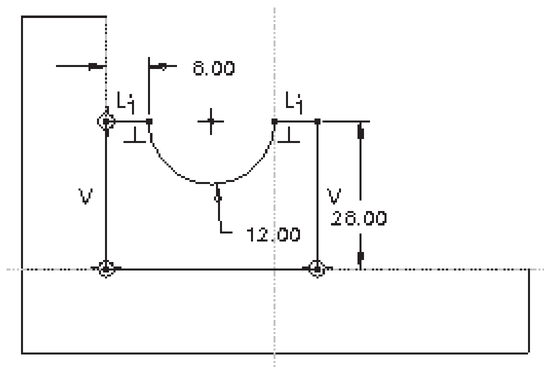


Figure 5-83 Sketch with dimensions and constraints

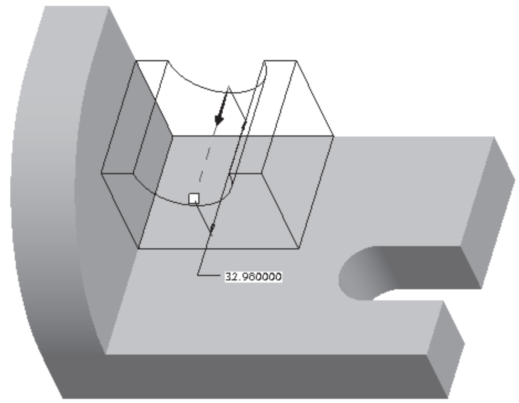


Figure 5-84 Arrow showing the direction of material addition

6. Choose the **Build feature** button from the **Extrude** dashboard to confirm the feature creation. Choose the **Named View List** button from the **View** toolbar; a flyout is displayed. Choose the **Default Orientation** option from the flyout; the default view of the model is displayed, as shown in Figure 5-85.

Figure 5-85 Model with the third extruded feature

Selecting the Sketching Plane for the Last Feature

The last feature and the third feature are created on the same plane. But the depth of extrusion is different for both of them. This is the reason they are considered as separate features. Therefore, for sketching this feature, you can use the sketching plane that was used for creating the third feature.

1. Choose the **Extrude** button and then invoke the **Sketch** dialog box.

2. Choose the **Use Previous** button from the **Sketch Plane** area in the **Sketch** dialog box; the sketcher environment is activated automatically.

When you choose the **Use Previous** button, the system selects the previous sketching plane that was used to create the base feature.

3. Choose the **Sketch** button to enter the sketcher environment.

Creating and Dimensioning the Sketch for the Last Feature

The sketch of this feature consists of three lines. The bottom edge of the sketch is aligned with the top edge of the base feature and the left edge is aligned with the right edge of the third feature. When this sketch is extruded, it creates a rib shape.

1. Draw the section sketch using various sketcher tools, as shown in Figure 5-86. The sketch is dimensioned automatically and some weak dimensions are assigned to it. You can add the required constraints to align the lines and points in the sketch with other features, as shown in Figure 5-86. In the figure, the constraint symbol displayed on the line indicates that the edges of the adjacent features are used to close the section.

2. After the sketch is complete, choose the **Done** button. The **Extrude** dashboard is enabled. Use the middle mouse button to orient the model, as shown in Figure 5-87.

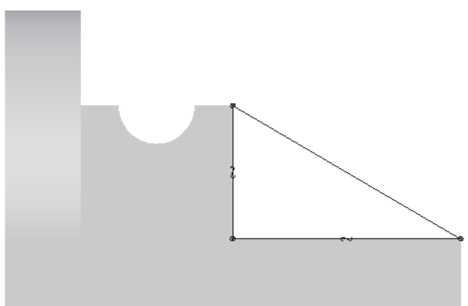


Figure 5-86 Sketch and constraints for the last feature

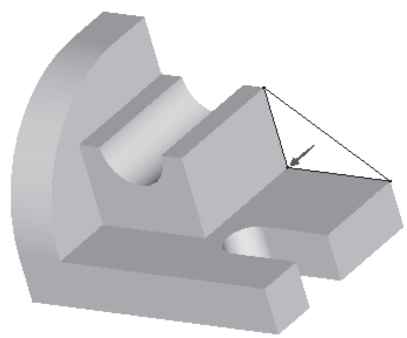


Figure 5-87 Arrow showing the direction of material addition

3. Enter the value **12** in the dimension box present in the **Extrude** dashboard. The feature is completed and its preview can be seen in the drawing area.

4. Choose the **Build feature** button in the **Extrude** dashboard.

The default view of the model is displayed, when you choose the **Default Orientation** option from the flyout. To invoke the flyout, choose the **Named View List** button from the **View** toolbar. The default view of the model is shown in Figure 5-88.

Saving the Model

1. Choose the **Save** button from the **File** toolbar and save the model. The order of feature creation can be seen from the **Model Tree** shown in Figure 5-89.

Figure 5-88 *Completed model for Tutorial 2*

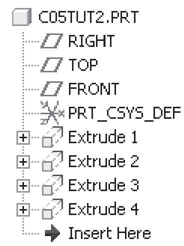


Figure 5-89 ***Model Tree*** *for Tutorial 2*

Tutorial 3

In this tutorial, you will create the model shown in Figure 5-90. This figure also shows the front, top, and right-side views of the solid model. **(Expected time: 45 min)**

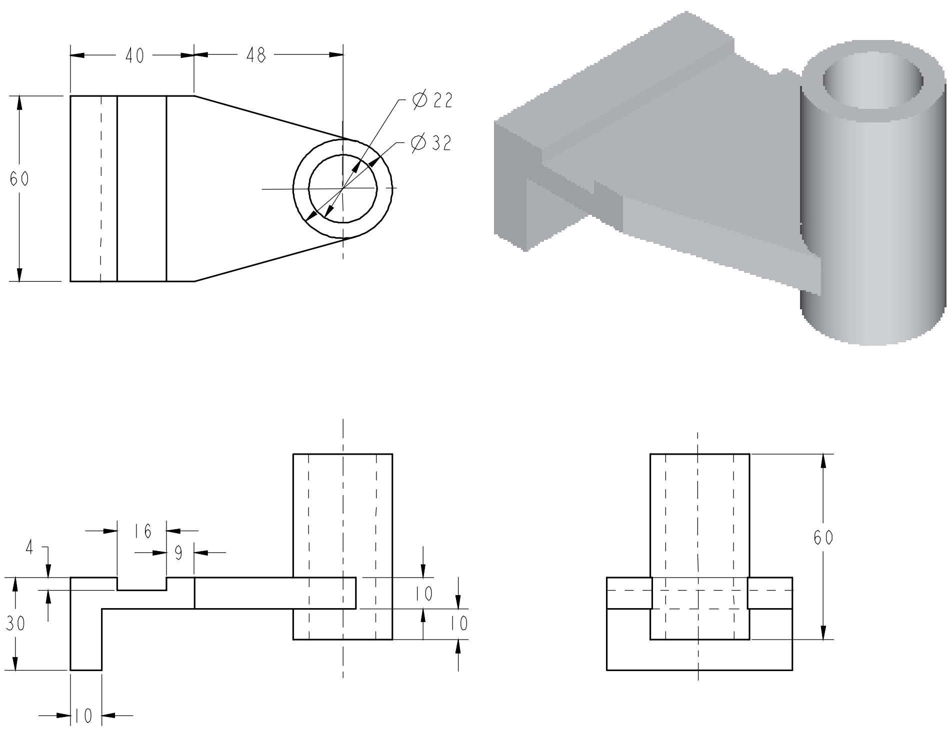


Figure 5-90 *Top, front, right-side, and isometric views of the model*

The following steps are required to complete this tutorial:

a. Examine the model and then determine the number of features in it, refer to Figure 5-90.
b. Create the base feature, refer to Figures 5-91 and 5-92.
c. Create the second feature, refer to Figures 5-93 through 5-96.
d. Create the hollow cylindrical feature on an offset datum plane, refer to Figures 5-97 through 5-102.

Starting a New Object File

1. Set the working directory if required and open a new part file with the name *c05tut3*. The three default datum planes are displayed in the drawing area if the **Plane Display** button is activated.

Selecting the Sketching Plane for the Base Feature

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

1. Choose the **Extrude** button from the **Base Features** toolbar.

2. Choose the **Placement** tab and from the slide-down panel choose the **Define** button; the **Sketch** dialog box is displayed.

3. Select the **FRONT** datum plane as the sketching plane.

   A yellow arrow is displayed on the **FRONT** datum plane and it points in the direction of viewing the sketch plane. The **RIGHT** datum plane and its orientation are selected automatically.

4. Choose the **Sketch** button to enter the sketcher environment.

Creating and Dimensioning the Sketch for the Base Feature

From the model, the section to be extruded for the base feature is evident. The section sketch is shown in Figure 5-91. When this sketch is extruded, it will create the base feature.

1. Draw the sketch using various sketcher tools, as shown in Figure 5-91.

2. The sketch is dimensioned automatically and some weak dimensions are assigned to it. Add the required constraints and modify the weak dimensions, as shown in Figure 5-91.

3. Choose the **Done** button; the **Extrude** dashboard is displayed.

4. Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Choose the **Default Orientation** option from the flyout; the model orients in its default orientation, that is, trimetric view and a yellow arrow is also displayed on it, indicating the direction of extrusion.

5. Enter the value **60** in the dimension box that is present on the **Extrude** dashboard and press ENTER.

6. Choose the **Build feature** button from the **Extrude** dashboard.

   The base feature is completed, as shown in Figure 5-92. You can use the middle mouse button to spin the model to view it from various directions.

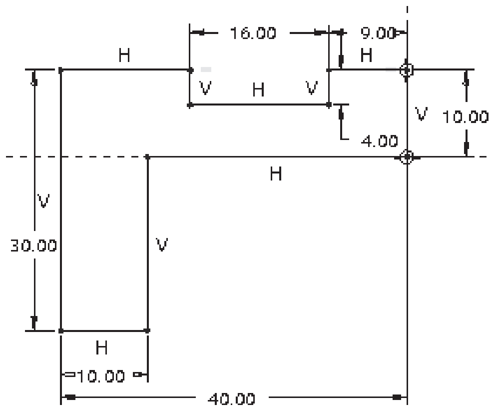


Figure 5-91 *Sketch with dimensions and constraints for the base feature*

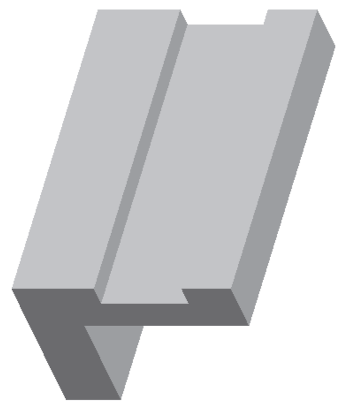


Figure 5-92 *Base feature of the model*

Selecting the Sketching Plane for the Second Feature

The next feature is an extruded feature. The sketching plane for this feature is the top face of the base feature.

1. Choose the **Extrude** button from the **Base Features** toolbar.

2. Choose the **Placement** tab; the slide-down panel is displayed. Choose the **Define** button from the slide-down panel; the **Sketch** dialog box is displayed.

3. Select the top face of the base feature shown in Figure 5-93 as the sketching plane. A yellow arrow that points in the direction of viewing the sketch is displayed on the top face.

4. Choose the **Flip** button to flip the yellow arrow to point in the direction shown in Figure 5-93.

5. Select the **RIGHT** datum plane and then choose the **Right** option from the **Orientation** drop-down list.

6. Choose the **Sketch** button to enter the sketcher environment.

Creating and Dimensioning the Sketch for the Second Feature

The next feature to be created is an extrude feature. The section for the extrude feature is shown in Figure 5-94. Before drawing the sketch, turn the model display to **No hidden**.

1. Draw the sketch using various sketcher tools. In the sketch, draw a center line passing through the center of the arc, as shown in Figure 5-94. This center line helps in dimensioning the sketch. Add the required constraints and dimensions to the sketch, as shown in Figure 5-94. Before exiting the sketcher environment, turn the model display to **Shading**.

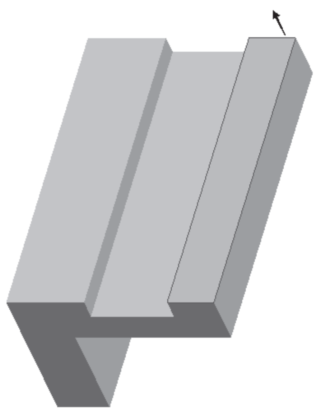


Figure 5-93 *Arrow pointing from the sketching plane in the direction of feature creation*

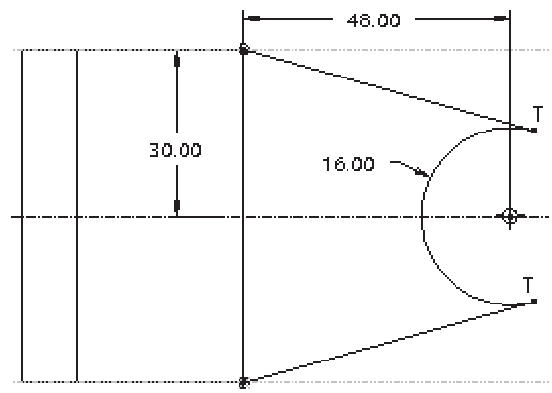


Figure 5-94 *Sketch with dimensions and constraints*

2. Choose the **Done** button. The **Extrude** dashboard is enabled. Use the middle mouse button to orient the model, as shown in Figure 5-95.

3. Enter the value **10** in the dimension box present on the **Extrude** dashboard and press ENTER.

4. Now, choose the **Build feature** button to confirm the feature creation. The default trimetric view of the extruded feature is shown in Figure 5-96.

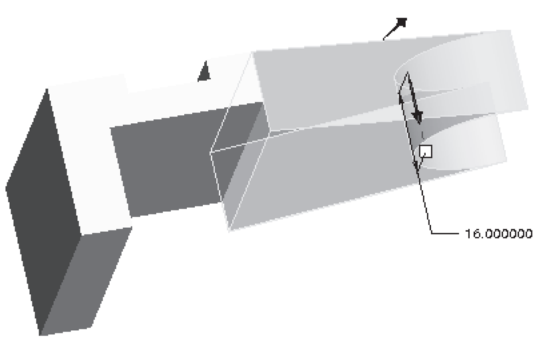


Figure 5-95 *Arrow showing the direction of material addition*

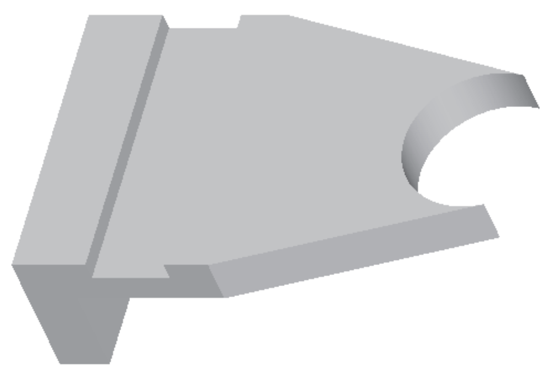


Figure 5-96 *Model with the second extruded feature*

Creating a Datum Plane for the Last Feature

To create the hollow cylindrical feature, you require a datum plane. This datum plane will be created at an offset distance of 10 from the bottom face of the second feature shown in Figure 5-97.

Note

The other method to create this feature is to select the top planar surface of the second feature as the sketching plane and extrude the sketch on both sides of the sketching plane. The depth of extrusion will be different at both the sides. If you use this method to create this cylindrical feature, you do not need to create a datum plane.

1. Choose the **Plane** button from the **Datum** toolbar; the **DATUM PLANE** dialog box is displayed.

2. Spin the model using the middle mouse button and then using the left mouse button, select the bottom face of the second feature.

   As you select the face of the second feature, the **Offset** constraint is displayed in the **References** collector of the dialog box.

3. In the **Translation** dimension box, enter the value **10**.

4. Choose the **OK** button from the **DATUM PLANE** dialog box.

   Datum plane **DTM1** is created, as shown in Figure 5-98 and will be selected as the sketching plane for creating the sketch.

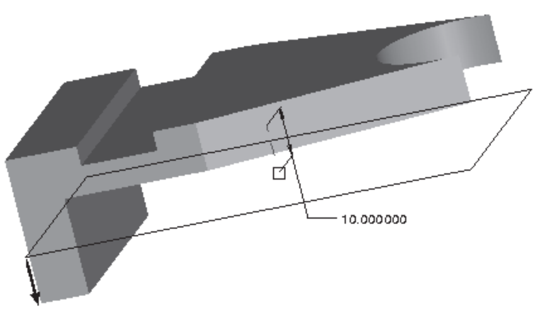


Figure 5-97 *Creating the datum plane*

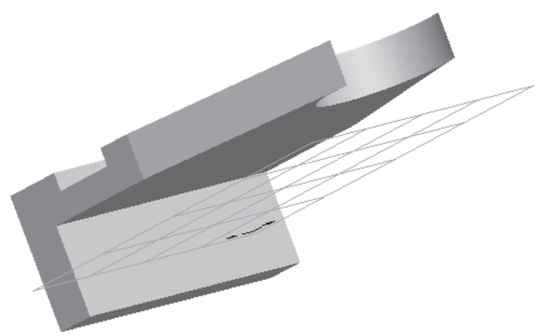


Figure 5-98 *Model after creating the datum plane*

Selecting the Sketching Plane for the Last Feature

The plane **DTM1** will be selected as the sketching plane and the depth of extrusion will be defined from this plane.

1. Choose the **Extrude** button from the **Base Features** toolbar.

2. Choose the **Placement** tab; the slide-down panel is displayed. Choose the **Define** button from the slide-down panel; the **Sketch** dialog box is displayed.

3. Select **DTM1** as the sketching plane. The yellow arrow appears on the datum plane.

4. Choose the **Flip** button to reverse the direction of viewing the sketch. Now, the arrow points in the direction shown in Figure 5-99.

5. Select the **FRONT** datum plane and choose the **Bottom** option from the **Orientation** drop-down list.

6. Choose the **Sketch** button to enter the sketcher environment.

Creating and Dimensioning the Sketch for the Last Feature

The sketch for the hollow cylindrical feature will be drawn on the datum plane **DTM1**. The sketch for the hollow cylindrical feature consists of two concentric circles. Before drawing the sketch, turn the model display to **No hidden**.

1. Draw the sketch using various sketcher tools and add the required constraints and dimensions to it, as shown in Figure 5-100.

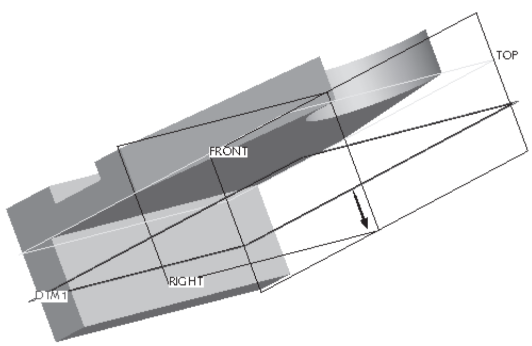


Figure 5-99 Direction of viewing the sketching plane

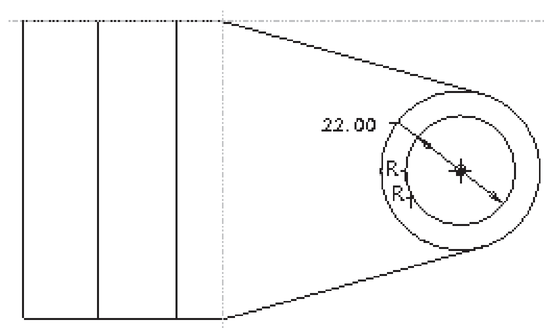


Figure 5-100 Sketch with dimensions and constraints

2. Choose the **Done** button; the **Extrude** dashboard is displayed. Now, turn the model display to **Shading**.

   Use the middle mouse button to orient the model, as shown in Figure 5-101. This view gives you a better view of the sketch in the 3D space.

3. Enter the value **60** in the dimension box present on the **Extrude** dashboard.

4. Choose the **Build feature** button to confirm the feature creation. The default trimetric view of the complete model is shown in Figure 5-102.

Saving the Model

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

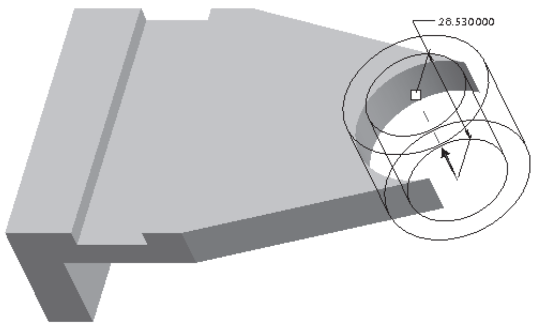


Figure 5-101 Preview of the feature

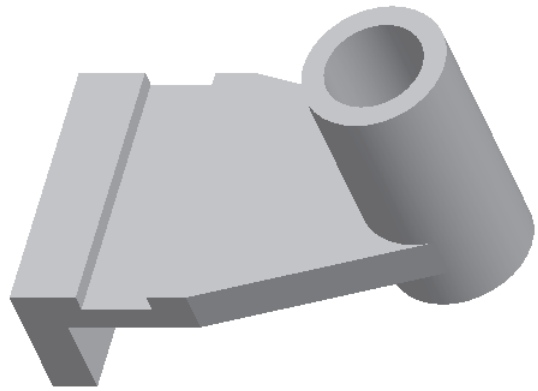


Figure 5-102 Final model of Tutorial 3

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. You can change the default names assigned to the datum planes. (T/F)

2. Datum points are also used to associate note in the drawings and attach datum targets. (T/F)

3. The constraint combination of creating an offset plane through cylinder is a valid combination. (T/F)

4. Generally, all features in a model are created on a single sketching plane. (T/F)

5. A sketching plane can be selected on the existing face of a feature. (T/F)

6. While creating a datum feature, the __________ selected from the model helps you choose from the available constraints that can be applied to constraint the datum feature.

7. Datum axis is an __________ axis that is created in Pro/ENGINEER.

8. The trimetric view of the model is displayed when you choose the __________ option from the **Named View List** button.

9. When you create a new object file, __________ default datum planes are displayed in the drawing area.

10. The default datum planes in the **Part** mode are named as __________, __________, and __________.

Review Questions

Answer the following questions:

1. Which one of the following is not a type of datum available in Pro/ENGINEER?

   (a) Axis (b) Plane
   (c) Circle (d) Curve

2. How many methods are available in Pro/ENGINEER to create a datum plane?

   (a) One (b) Two
   (c) Three (d) Four

3. Which one of the following dialog boxes is displayed when you choose the **Point** button from the **Datum** toolbar while extruding a section sketch?

   (a) **DATUM POINT** dialog box (b) **DATUM PLANE** dialog box
   (c) **DATUM AXIS** dialog box (d) None

4. Which one of the following combinations can be used to spin a model in the drawing area?

   (a) CTRL+ALT (b) middle mouse button
   (c) left mouse button (d) CTRL+right mouse button

5. Which one of the following menus in the menu bar is used to set the default orientation of the model to isometric?

   (a) **File** (b) **Insert**
   (c) **Tool** (d) None

6. Datum planes are considered as feature geometry and have mass and volume. (T/F)

7. To set the default orientation of the model to isometric, you have to use the **Environment** dialog box. (T/F)

8. Generally, the sketching plane for the base feature of any model is decided after viewing the isometric view or the drawing views of the model. (T/F)

9. Datum planes are used as a reference for mirroring features, copying features, for creating a cross-section, as well as for orientation of references. (T/F)

10. Unlike the datum planes constraint option, all datum axes constraint options are stand-alone. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 5-103. The dimensions, front view, and right-side view of the model are shown in Figure 5-104. **(Expected time: 45 min)**

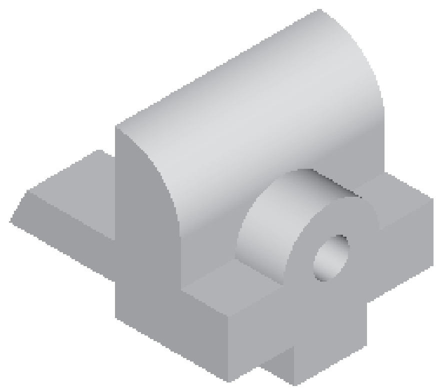


Figure 5-103 *Isometric view of the model*

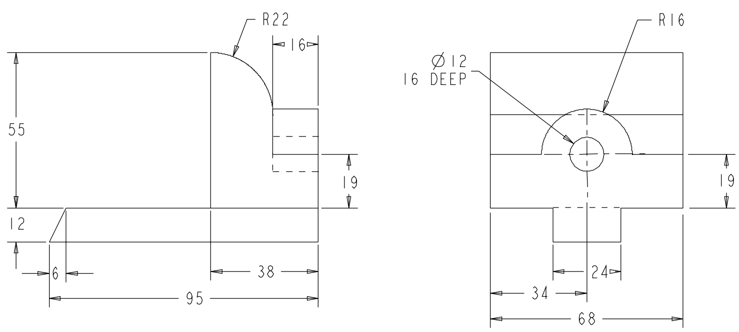


Figure 5-104 *Front and right-side views of the model*

Exercise 2

Create the model shown in Figure 5-105. The dimensions, and the front, top, right-side, and isometric views of the model are also shown in the figure. **(Expected time: 45 min)**

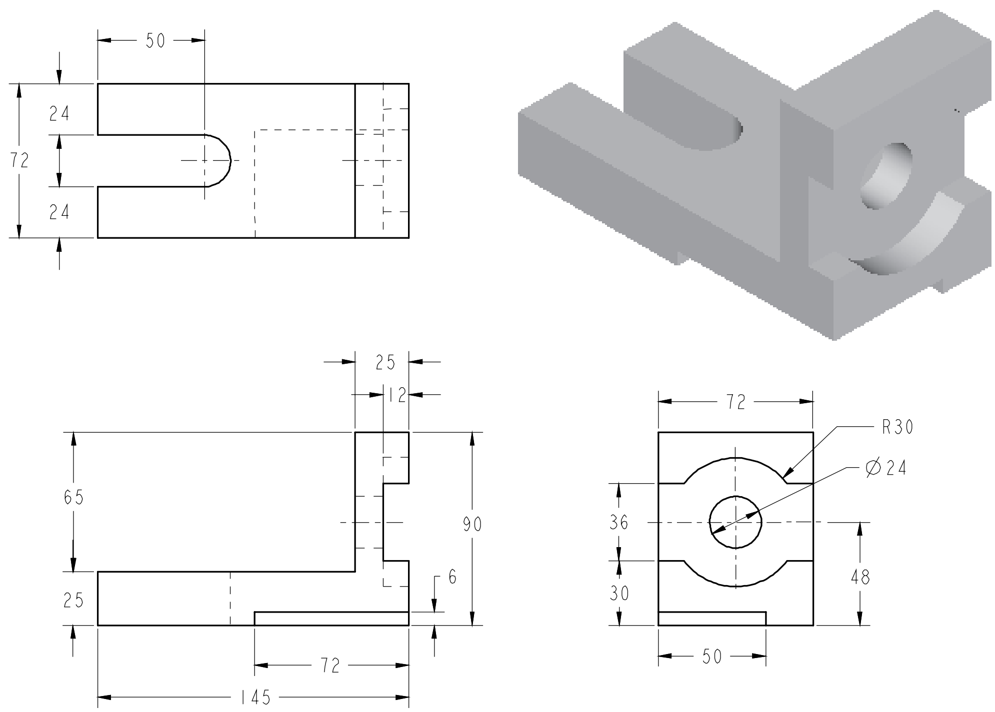


Figure 5-105 *Top, front, right-side, and isometric views of the model*

Exercise 3

Create the model shown in Figure 5-106. The top, front section, and two auxiliary views of the model are shown in Figure 5-107. **(Expected time: 1 hr)**

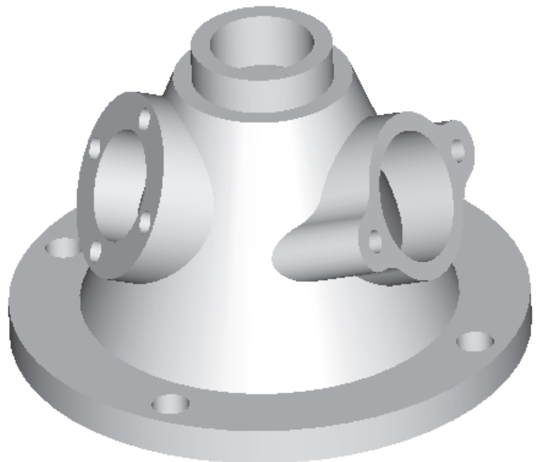


Figure 5-106 *The 3D view of the model*

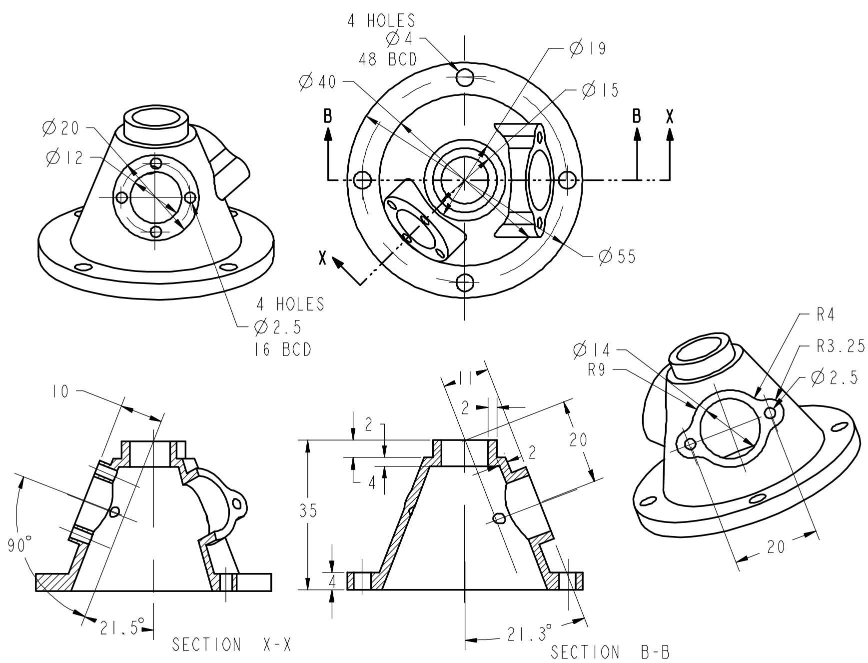


Figure 5-107 *Top, front section, and two auxiliary views of the model*

Hint to create the two side features in Exercise 3:

<u>Feature on the right</u>

1. Create an axis passing through the top face of the base feature and through the **RIGHT** datum plane.

2. Create a datum plane passing through the axis (created in the previous step) and at an angle of 21.5 degrees from the **RIGHT** datum plane.

3. Now, create an offset plane at a distance of 11 from the datum plane (created in the previous step). Select this datum plane as the sketching plane.

<u>Feature on the left</u>

1. Create a datum plane passing through the axis of revolution of the base feature and at an angle of 45 degrees from the **FRONT** datum plane.

2. Create a datum axis passing through the top face of the base feature and through the datum plane (created in the previous step).

3. Create a datum plane passing through datum axis (created in the previous step) and at an angle of 21.5 degrees from the datum plane (created in step1).

4. Create a datum plane offset to a distance of 10 from the datum plane (created in the previous step).

Exercise 4

Create the model shown in Figure 5-108. The dimensions, the left-side view, auxiliary view, and front view of the model are also shown in the Figure 5-109. **(Expected time: 45 min)**

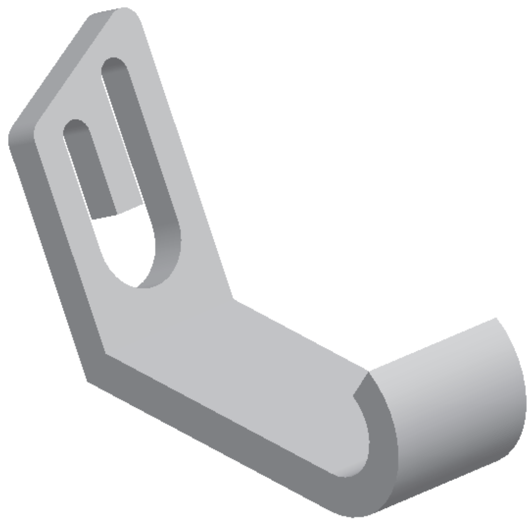


Figure 5-108 Isometric view of the model

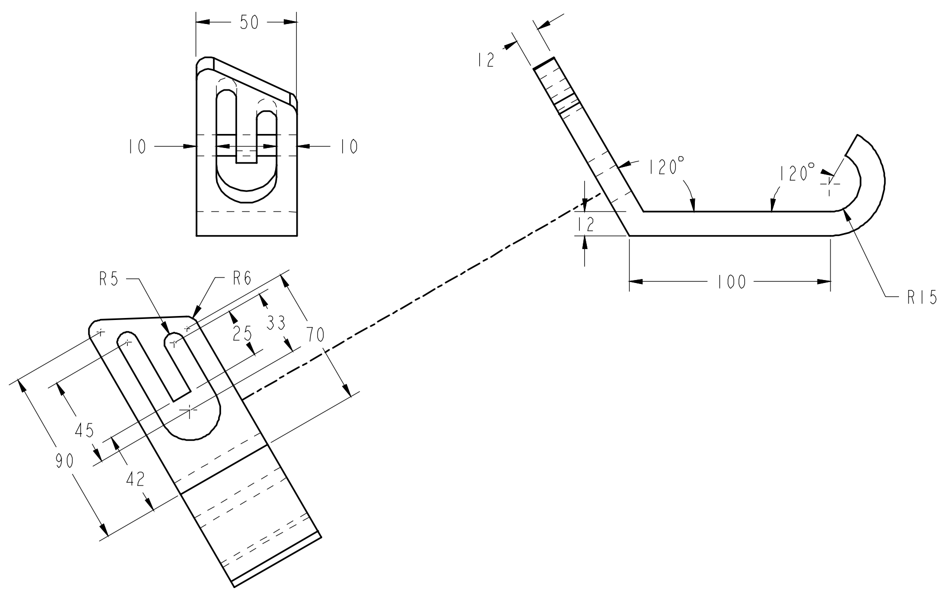


Figure 5-109 Left-side view, auxiliary view, and front view of the model

Answers to Self-Evaluation Test

1. T, **2.** T, **3.** T, **4.** F, **5.** T, **6.** geometry, **7.** imaginary, **8. Default Orientation**, **9.** three, **10. TOP**, **FRONT**, **RIGHT**.