

Chapter 4

Datums



Learning Objectives

After completing this chapter, you will be able to:

- Understand the three default datum planes.
- Understand selection methods in Pro/ENGINEER Wildfire 3.0.
- Create the datum planes using different constraints available.
- Create embedded datum planes.
- Create datum axes using the different constraints available.
- Create the datum points.
- Create extrude and revolve cuts.

DATUMS

Datums are imaginary features with no mass or volume and 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 a 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 by default displayed 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.

The default datum planes in the **Part** mode are named **FRONT**, **TOP**, and **RIGHT**. In case of the **Assembly** mode, the default datum planes are named **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**; the **PART SETUP** menu will be displayed, as shown in Figure 4-1. 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, enter the desired name in it and choose the **Accept value** button.

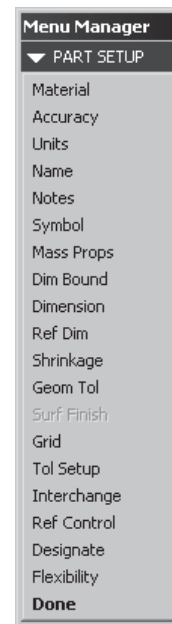


Figure 4-1 The **PART SETUP** menu

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



Note

*The default units of a part are **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 the 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 4-2 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 it and then decide which feature is to be considered as the base feature.

In Figure 4-2, 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 4-3. 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.

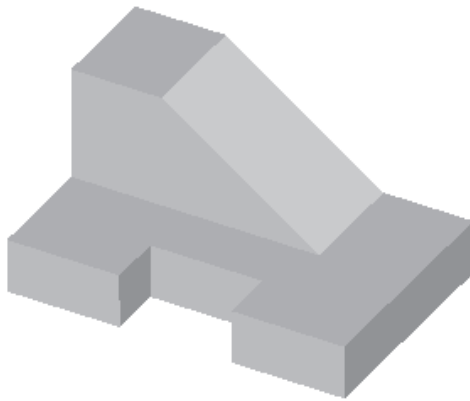


Figure 4-2 Model having two extruded features

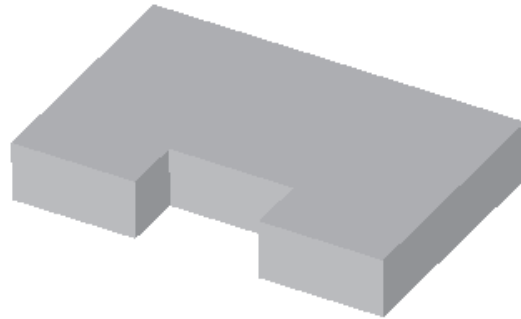


Figure 4-3 Base feature of the model

As shown in Figure 4-4, 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 4-5. 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 evident from Figure 4-4. 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 3.0

The selection method in Pro/ENGINEER Wildfire 3.0 is divided into two types:

1. Selection
2. Collection

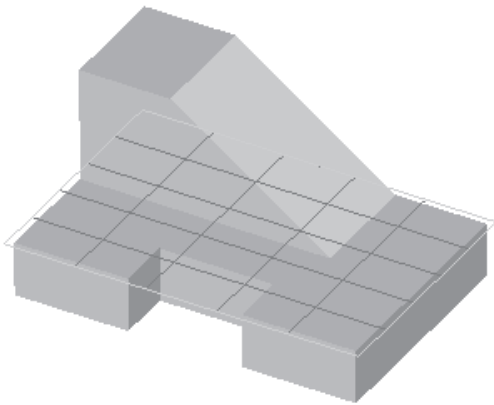


Figure 4-4 Plane selected for the base feature

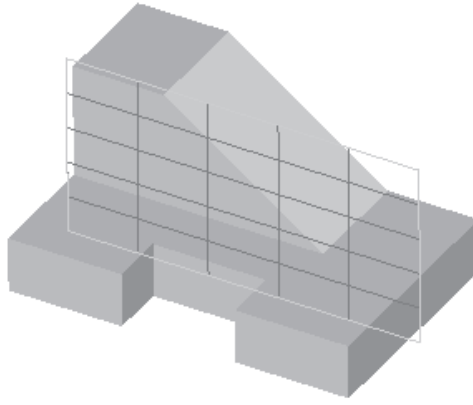


Figure 4-5 Plane selected for the second feature

Selection

In Pro/ENGINEER Wildfire 3.0, 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 3.0. 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 4-6.

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 4-7. All selections that you made 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.

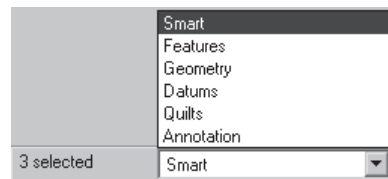


Figure 4-6 The Filter drop-down list

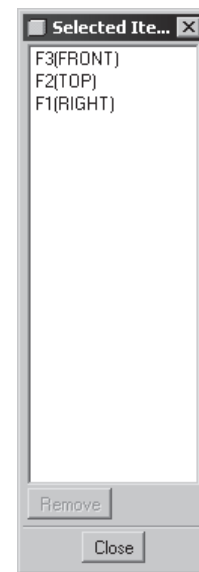


Figure 4-7 The Selected Items dialog box

By default, in the **Filter** drop-down list, the **Smart** filter is selected. 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 3.0, collection refers to the action when you invoke 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 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 4-8 shows the **Datum** toolbar and Figure 4-9 shows the method of invoking various types of datum features from the menu bar.

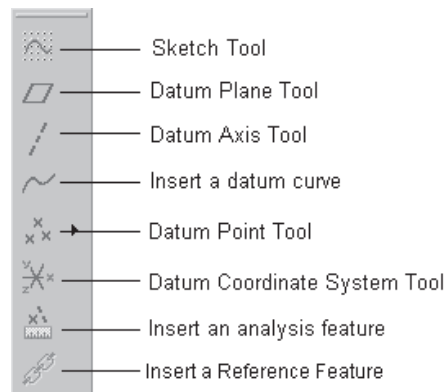


Figure 4-8 The **Datum** toolbar

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 at anytime when required. The display of the datum planes can be turned on or off by using the **Datum planes on/off** 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 the datum planes in Pro/ENGINEER Wildfire 3.0 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 also as references in assembly.

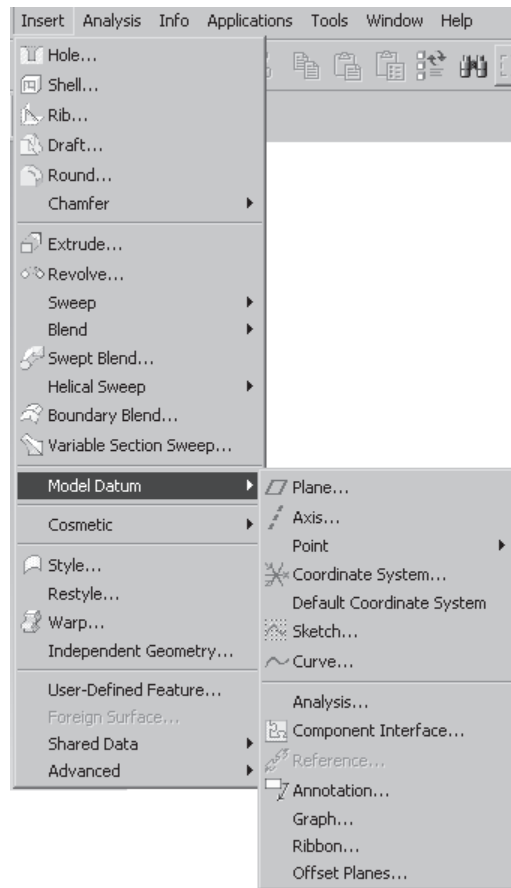


Figure 4-9 Invoking the datum options from the **Insert** menu in the menu bar

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

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 **Datum Plane Tool** button from the **Datum** toolbar, the **DATUM PLANE** dialog box will be displayed, as shown in Figure 4-10. Note that in this dialog box, some references have already been selected. 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 4-10 The **DATUM PLANE** dialog box with the **Placement** tab chosen*

Figure 4-10 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 selected by default. Under this 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 4-11. 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 the view. By default the arrow on the plane points toward the brown side. The arrow direction and so the direction of the plane can be changed by using the **Flip** button.

The **Adjust Outline** check box, when selected, allows you to set the size for the datum plane.

Properties Tab

The **Properties** tab of the **DATUM PLANE** dialog box is shown in Figure 4-12. This tab, when chosen, 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.

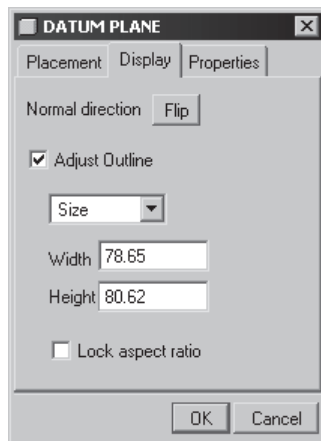


Figure 4-11 The **DATUM PLANE** dialog box with **the Display** tab chosen

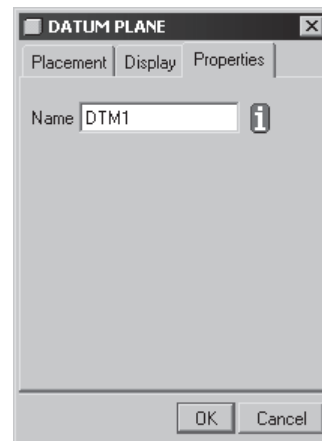


Figure 4-12 The **DATUM PLANE** dialog box with **the Properties** tab chosen

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 need more than one constraint to do the same. When only one constraint is used to create a datum plane, it is called a stand-alone constraint. The stand-alone constraints, by themselves are sufficient to define a datum plane. 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 4-13 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 4-13, the 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 create a datum plane that passes through a cylinder and normal to a plane then look for

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 4-13 Datum plane constraint combinations using the **Through** constraint

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 4-13, remember that the constraints that are not stand-alone have to be applied in pairs.

The options in the drop-down list shown in Figure 4-14 are used to make a 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 4-14 are available. Once you exit the datum plane creation, the options in this drop-down list change.

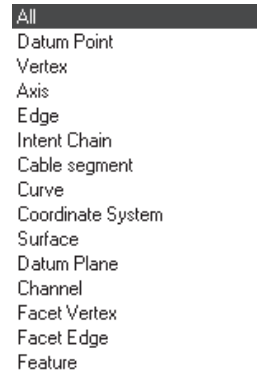


Figure 4-14 The **Filter** drop-down list with options to make selection

Figure 4-15 shows that 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, as shown in Figure 4-16. 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.

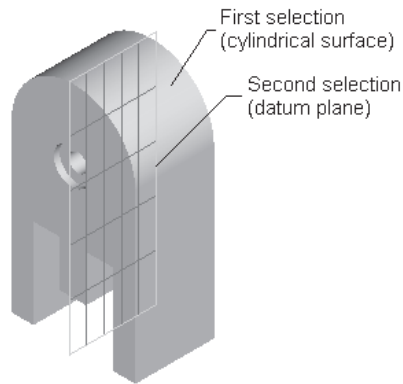


Figure 4-15 Selecting a cylindrical surface and a default datum plane to create a datum plane

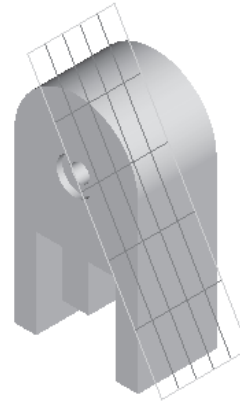


Figure 4-16 Resulting datum plane passing through the center of a cylinder and at an angle



Note

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

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

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 4-17 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	No
	Plane	Yes	Yes	Yes	Yes	Yes	No	Yes	No	No

Figure 4-17 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 4-18 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 4-18. The **Offset** constraint is automatically applied and 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 4-18. 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 4-19.

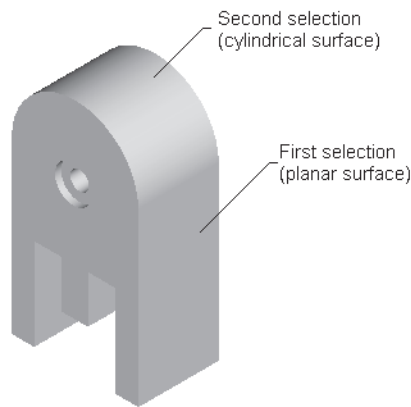


Figure 4-18 Selecting a planar surface and a cylindrical surface to create a datum plane

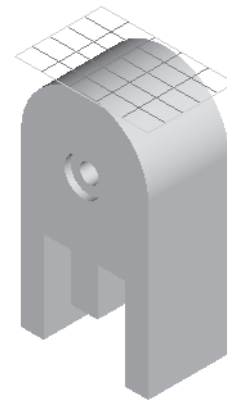


Figure 4-19 Resulting datum plane

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

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 4-20 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 the figure.

Figure 4-21 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 4-22. The following steps explain the procedure to create this datum plane:

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 4-20 Datum plane constraint combinations using the **Parallel** constraint*

1. Invoke the **DATUM PLANE** dialog box. Select the first reference shown in Figure 4-21. 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.
3. Now, use CTRL+left mouse button to select the second reference shown in Figure 4-21. 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 4-22.

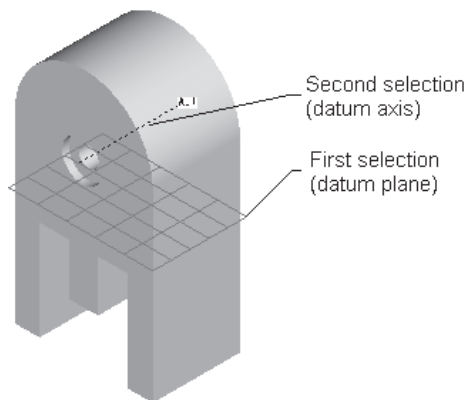


Figure 4-21 Selecting a datum plane and an axis to create a datum plane

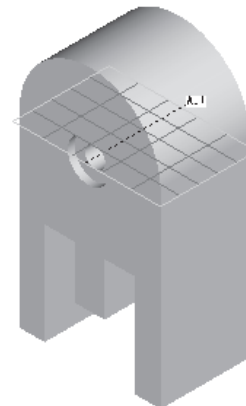


Figure 4-22 Resulting datum plane

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.

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

Figure 4-23 Datum plane constraint combinations using the **Offset** constraint

When the **Offset** constraint is applied, the **Translation** dimension box appears. 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 4-23 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.

Figure 4-24 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 4-25. 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 4-24. The **Offset** constraint is automatically applied and displayed in the **References** collector. The **Translation** dimension box appears.
2. Now, use CTRL+left mouse button to select the second reference, which is an edge shown in Figure 4-24. 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 4-25.



Note

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

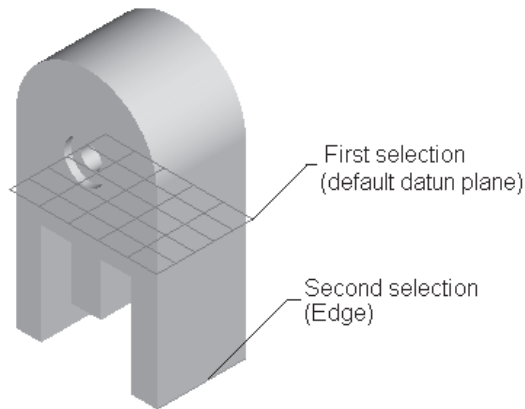


Figure 4-24 Selecting a datum plane and an axis to create a datum plane

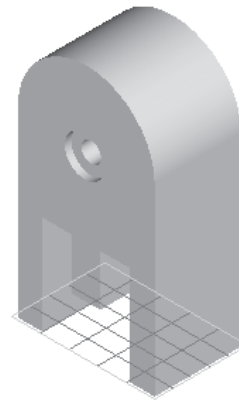


Figure 4-25 Resulting datum plane

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 4-26 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.

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 4-26 Datum plane constraint combinations using the **Tangent** constraint



Note

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

Embedded Datum Planes

The term **Embedded Datum Planes** 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 **Datum Plane Tool** button from the **Datum** toolbar, the datum plane created will be called an embedded datum plane.

You can try creating an embedded datum plane by following these steps:

1. Choose the **Extrude Tool** button; the **Extrude** dashboard will be displayed.
2. In the **Extrude** dashboard, choose the **Placement** tab and then click on 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 an embedded datum plane.
4. Choose the **Datum Plane Tool** 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 an embedded datum plane. Once the embedded datum plane is created, it cannot be referenced by other features.

The embedded datum plane 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 appear as a grouped feature in the **Model Tree**. This means that the embedded datum plane that was created during feature creation belongs to the feature that is created.

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 a datum axis can be turned on or off by using the **Datum axes on/off** button from the **Datum Display** toolbar. The uses of the datum axes are discussed 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 coaxial features.
4. They are also used to create rotational patterns. You will learn to create patterns in Chapter 6.



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 **Datum Axis Tool** button from the **Datum** toolbar, the **DATUM AXIS** dialog box appears, as shown in Figure 4-27. The options in this dialog box are discussed next.

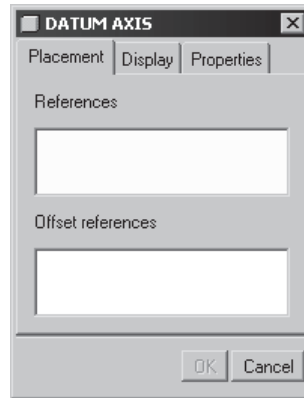


Figure 4-27 The **DATUM AXIS** dialog box with the **Placement** tab chosen

Placement Tab

When you invoke the **DATUM AXIS** dialog box, the **Placement** tab is selected 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 available on the model, as shown in Figure 4-28. These handles appear like a white square. The middle handle is used to move the position of the axis you are creating. The other two handles are used to specify the references for dimensioning.

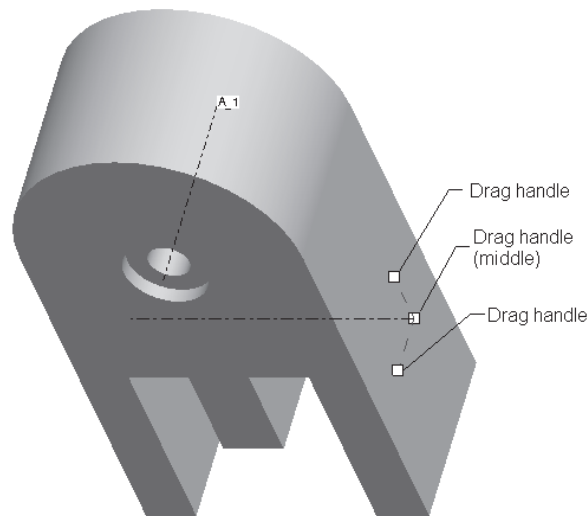


Figure 4-28 Solid model with the datum axis and three drag handles

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 is shown in Figure 4-29. Select the **Adjust Outline** check box to activate the **Length** edit box. 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.

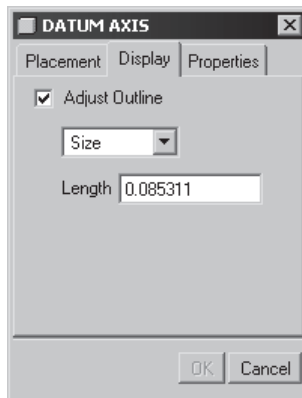


Figure 4-29 The DATUM AXIS dialog box with the Display tab chosen

Properties Tab

The **Properties** tab of the **DATUM AXIS** dialog box is shown in Figure 4-30. This tab, when selected, 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 axes created are successively named.

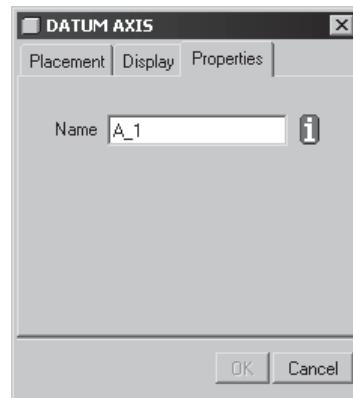


Figure 4-30 The **DATUM AXIS** dialog box with the **Placement** tab chosen

As mentioned earlier, a datum axis is created using constraints, which are applied automatically, and also filters that are available in the **Filter** drop-down list located in the **Status** bar. While creating a datum axis, the filters that are available are shown in Figure 4-31. The constraints that are used to create a datum axis are discussed next.

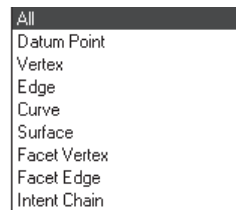


Figure 4-31 Filters available in the **Filters** drop-down list

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 4-32. In Figure 4-32, **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 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 4-33, the preview of the datum axis with the drag handles is shown. It should be noted that the references for dimensioning can also be selected dynamically using these drag handles.

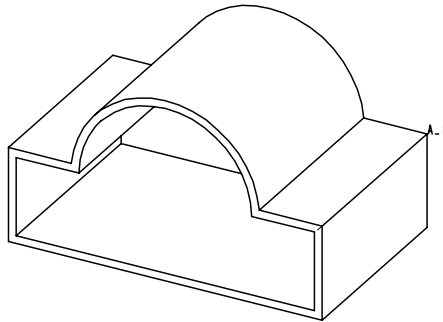


Figure 4-32 Datum axis created along the edge

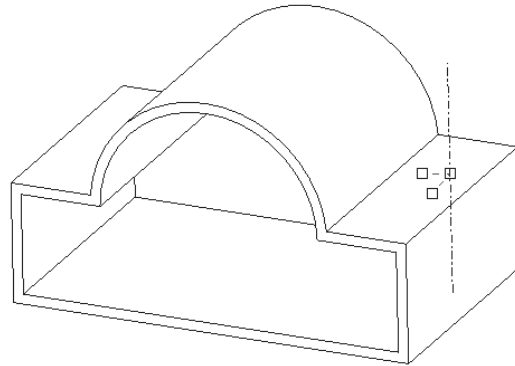


Figure 4-33 Datum axis created normal to the plane



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.

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 4-34, 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 4-35, the preview of the datum axis is shown. The selected cylindrical surface is highlighted in this figure.

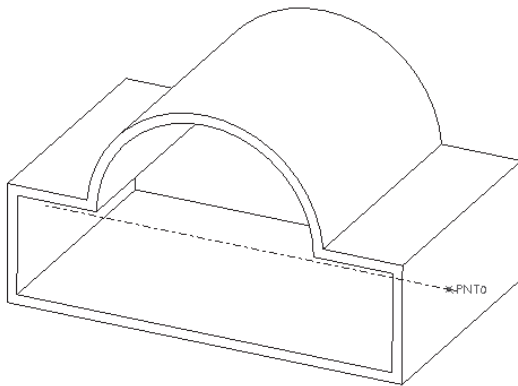


Figure 4-34 Datum axis passing through the datum point and normal to the plane

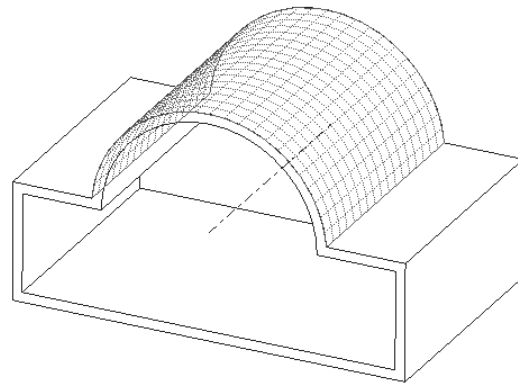


Figure 4-35 Datum axis passing through the center of a cylinder

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 4-36, the preview of the datum axis is shown. The two faces that were selected are also highlighted in this figure.

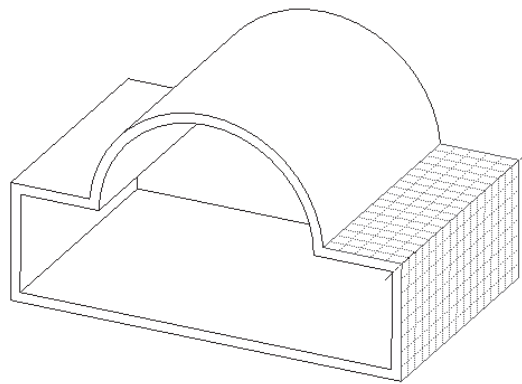


Figure 4-36 Datum axis created on the edge where the two selected planes meet

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 4-37, 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 vertices. 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 4-38, 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.

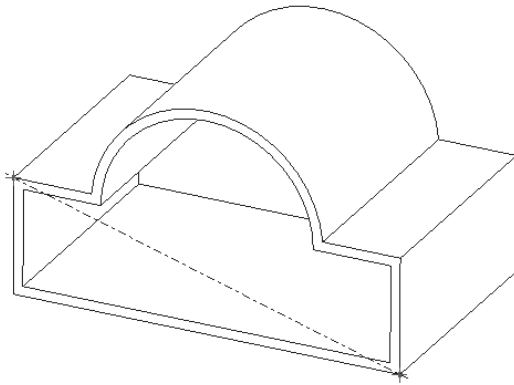


Figure 4-37 Datum axis created between the two selected vertices

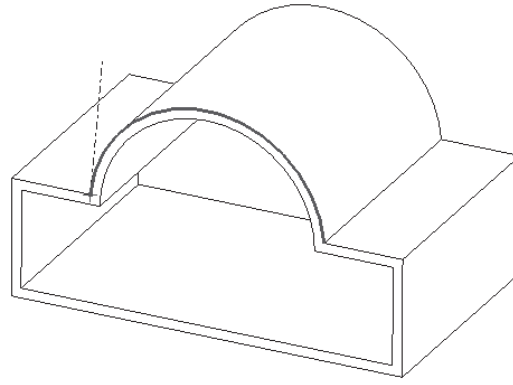


Figure 4-38 Datum axis created tangent to the selected curve

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.
2. To associate note in the drawings and attach datum targets.
3. To create a 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 **Datum Point Tool** button from the **Datum** toolbar; the **DATUM POINT** dialog box appears, along with various

options to create datum points, as shown in Figure 4-39. This dialog box can be used to create more than one datum point. The options in the **DATUM POINT** dialog box are explained 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 4-39 is always present. This area is used to select the references which aid in placing the datum point. The constraints 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.

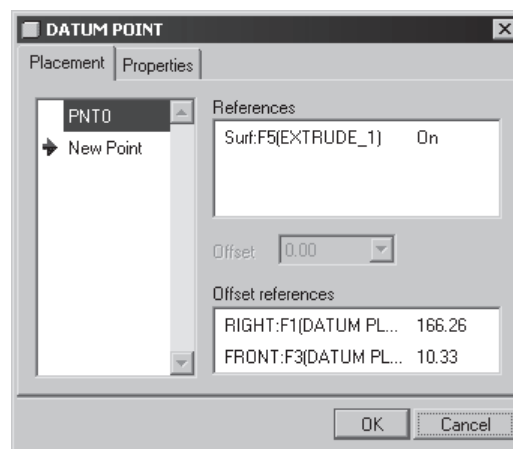


Figure 4-39 The **DATUM POINT** dialog box with the **Placement** tab chosen



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.

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 4-40.

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 the dimension of the point, a default value will be displayed in the **Offset References** collector. You can accept

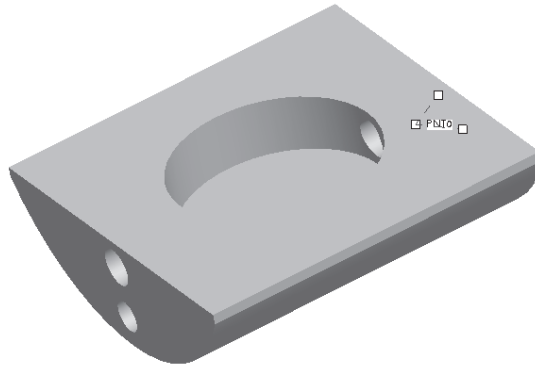


Figure 4-40 Datum point on a face and the three drag handles

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.

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 4-41 and 4-42.

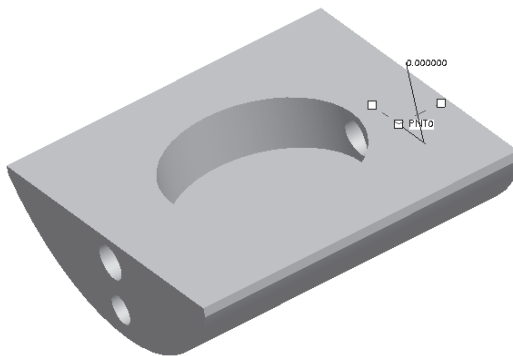


Figure 4-41 Datum point on the top face with offset 0 and the three drag handles

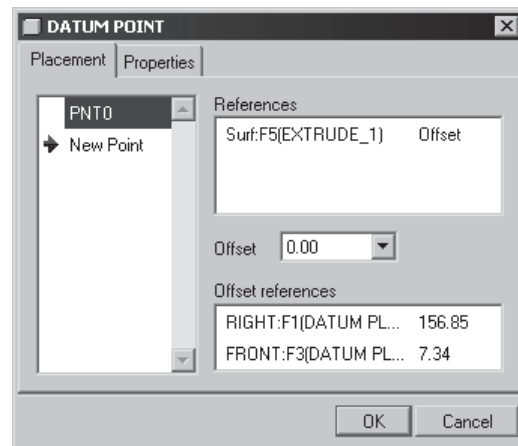


Figure 4-42 The DATUM POINT dialog box with the Offset constraint selected

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 selected 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 and the datum point is created, as shown in Figure 4-43.

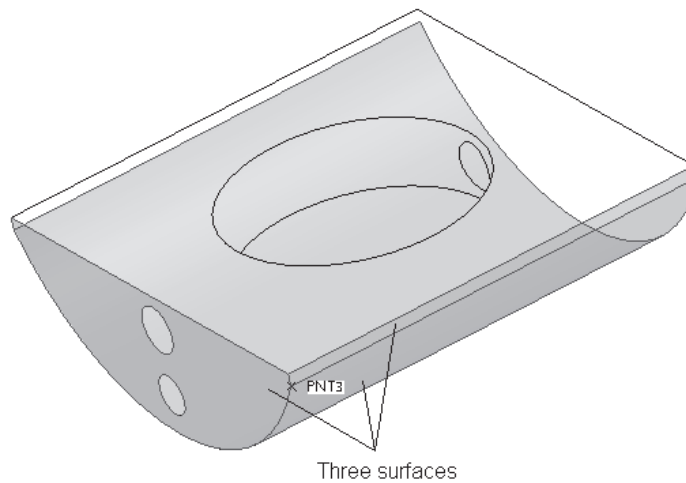


Figure 4-43 The three highlighted surfaces and the datum point at the intersection of the three surfaces

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 you select the three surfaces, the **DATUM POINT** dialog box appears, as shown in Figure 4-44.

It might so happen that with the current selection of references, more than one intersection point exists. 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.

Datum Point on a Vertex

The **On** constraint is used to create a datum point on the vertex of a face, edge, or 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.

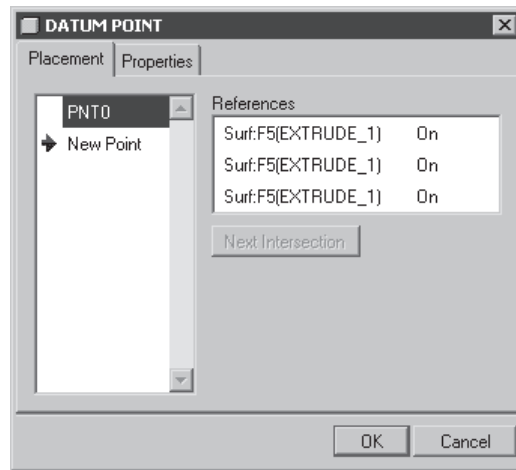


Figure 4-44 The **DATUM POINT** dialog box



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 4-45 and the **DATUM POINT** dialog box appears, as shown in Figure 4-46.

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 4-46.

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.

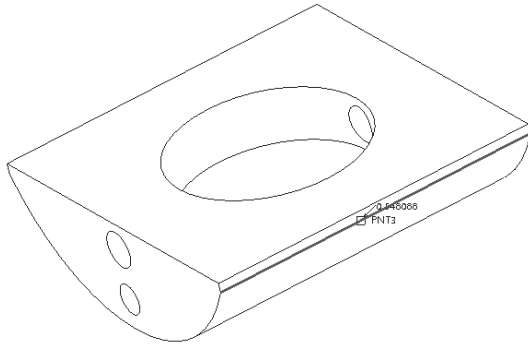


Figure 4-45 The highlighted edge and the datum point at some offset distance from one end of the edge

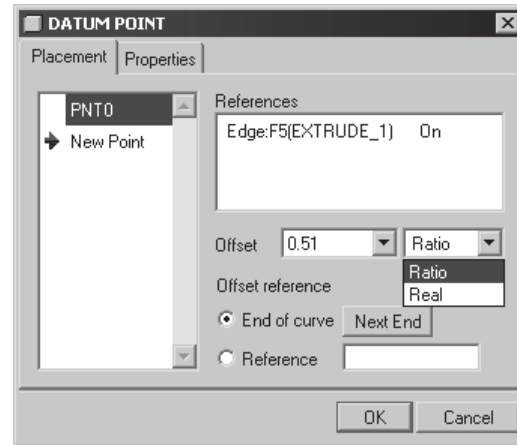


Figure 4-46 The **DATUM POINT** dialog box

Sketched Datum Point Tool Button



When you choose the black arrow on the right of the **Datum Point Tool** button, a flyout will be displayed. Choose the **Sketched Datum Point Tool** button from this flyout; the **Sketch Datum Point** dialog box will be displayed. The function of this dialog box is the 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 **Create Points** 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 Datum Point Tool** 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 in 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 4-47 and you will be 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; and edit box appears in which you can enter an offset value. Similarly, enter the offset values for Y and Z axes by entering the value in the **YAxis** and **ZAxis** columns, respectively. 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 the Figure 4-47, the dialog box is shown with the coordinates of three datum points entered.

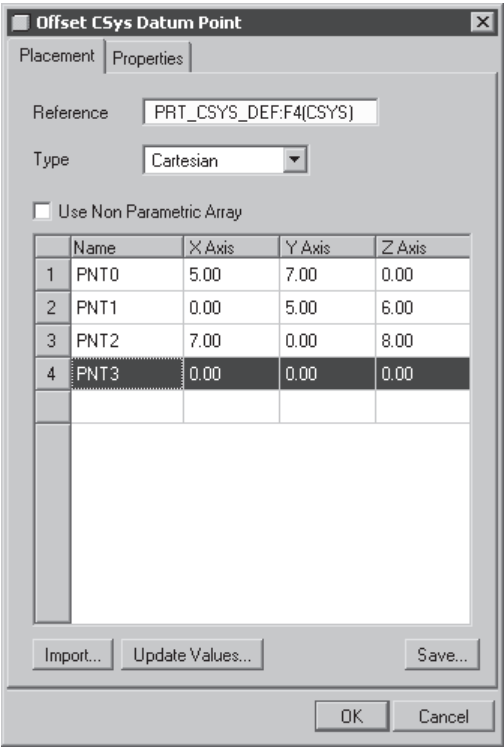


Figure 4-47 The Offset CSys Datum Point dialog box

To Create a Datum Point by Clicking



When you choose the **Field Datum Point Tool** button from the flyout, the **Field Datum Point** dialog box will be displayed, as shown in Figure 4-48. 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.

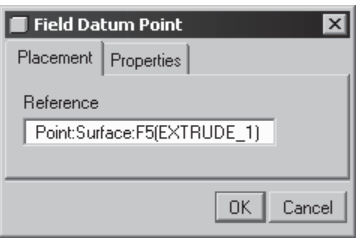


Figure 4-48 The Field Datum Point dialog box

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 about 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 4-49 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 4-50.

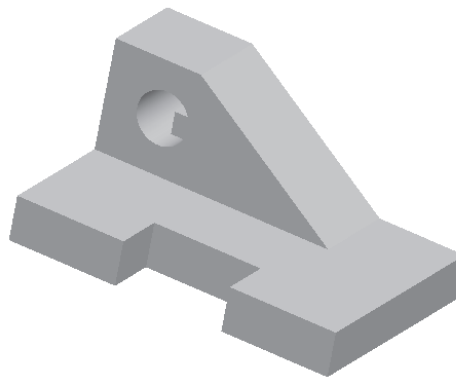
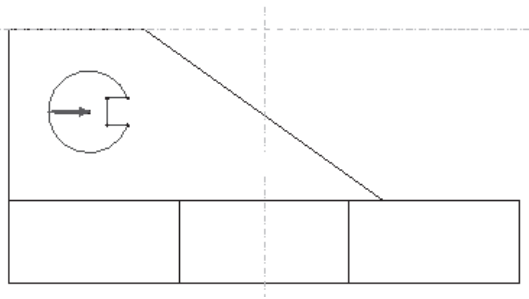


Figure 4-49 Sketch for the extrude cut and arrow showing the direction of material removal **Figure 4-50** Cut feature created on the selected plane

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 4-51. All material on the plane selected for sketching will be removed leaving the extruded cut feature, as shown in Figure 4-52.



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 5.



Tip: In the model shown in Figure 4-49, the sketching plane selected for the 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.

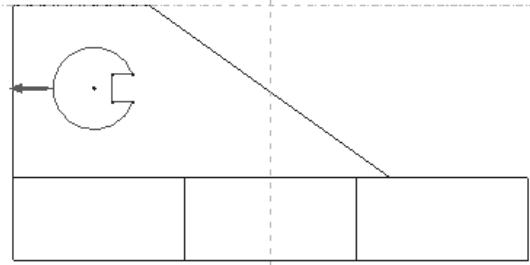


Figure 4-51 Arrow showing the direction of material removal

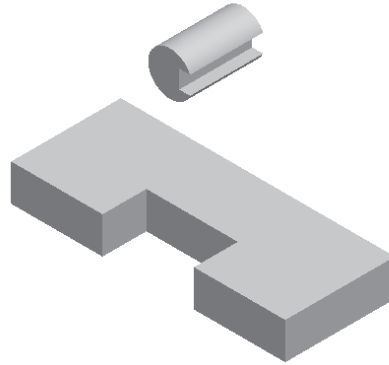


Figure 4-52 Cut feature created in the direction shown in the adjacent figure

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 4-53 shows the section drawn that is to be revolved. The front surface of the second extruded feature is selected as the sketching plane. Figure 4-54 shows the revolve cut created on the selected surface.

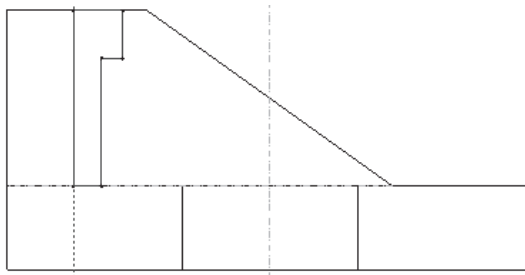


Figure 4-53 The section for revolve cut

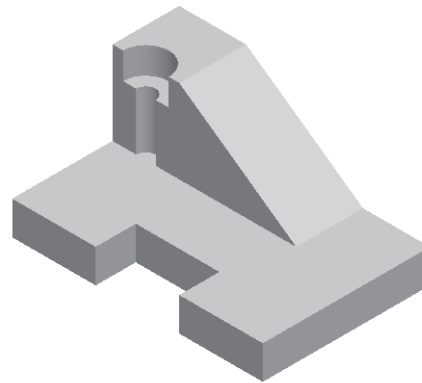


Figure 4-54 Revolve cut created



Note

The **Sweep Cut** is explained in Chapter 7.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 4-55. The front and the right-side view of the solid model are shown in Figure 4-56. **(Expected time: 30 min)**



Figure 4-55 Model for Tutorial 1

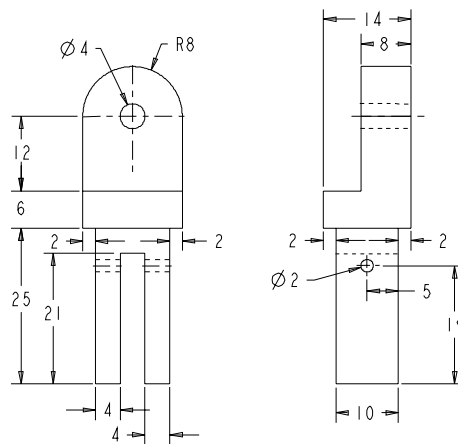


Figure 4-56 Front and side views of the model

The following steps are required to complete this tutorial:

- Examine the model and determine the number of features in it, refer to Figure 4-55.
- Create the base feature, refer to Figures 4-57 and 4-58.
- Create the second extrude feature that is at the bottom of the base feature, refer to Figures 4-59 through 4-62.
- Create the third feature on an offset plane, refer to Figures 4-63 through 4-68.
- Create the circular cut feature, refer to Figures 4-69 through 4-72.

When the Pro/ENGINEER session is started, the first task is to set the working directory. Since this is the first tutorial of this chapter, you need to create a folder named *c04*, if it does not exist. Choose the **New Directory** button in the **Select Working Directory** dialog box and create a folder named *c04* in *C:\ProE-WF-3.0*.

Starting a New Object File

1. Open a new part file and name it as **c04tut1**. 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, its orientation will be incorrect. Therefore, the final model will be oriented incorrectly. 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 4-55.

1. Choose the **Extrude Tool** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Placement** tab and from the slide-up 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; you will 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 4-57. 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 the values of dimensions, geometry of splines, or text entities** 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 the dimensions.

2. Modify the dimension values to those shown in Figure 4-57.
3. After the sketch is completed, choose the **Continue with the current section** button; you will exit the sketcher environment.
4. Choose the **Default Orientation** option from the **Saved view list** button of the **View** toolbar. 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 a value of 8; the model appears, as shown in Figure 4-58.

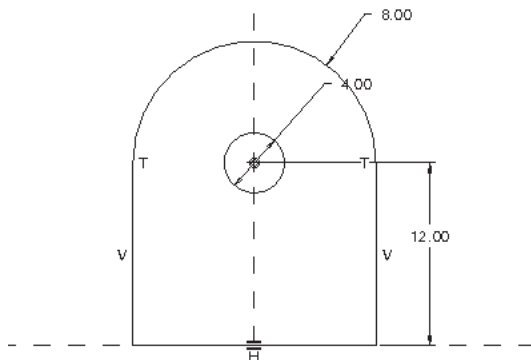


Figure 4-57 Sketch for the base feature with dimensions and constraints

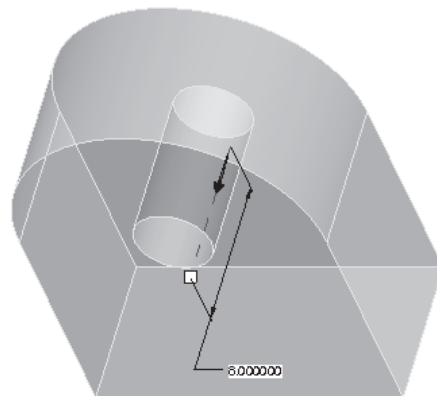


Figure 4-58 Default orientation of the model and the arrow showing the direction of feature creation



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 **Saved view list** button from the **View** toolbar. Choose the **FRONT** option from the drop-down list; the base feature will reorient in the drawing area such that you can view the front view of the base feature.

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 **Saved 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, select the **Isometric** option; the default orientation will be set to isometric.

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 Tool** button from the **Base Features** toolbar. The **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Placement** tab and from the slide-up 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 yellow arrow is displayed in the drawing area, as shown in Figure 4-59.

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.

In Figure 4-59, the model is oriented in its default orientation. The model can be oriented in its default position by choosing the **Default** option from the **Saved view list** button.

4. Choose the **Sketch** button from the **Sketch** dialog box; you enter the sketcher environment.

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 4-60.
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 4-60.
3. Choose the **Continue with the current section** button to exit the sketcher environment. The **Extrude** dashboard is enabled below the drawing area. Choose the **Shading** button from the **Model Display** toolbar to view the shaded model.



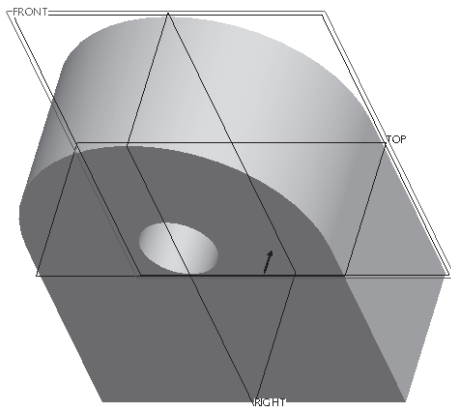


Figure 4-59 Arrow showing the direction of viewing the sketching plane

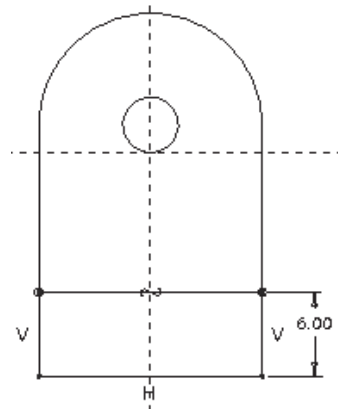


Figure 4-60 Sketch for the second feature



Tip: You can use the **Create an entity from an edge** 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.

4. Use the middle mouse button to orient the model, as shown in Figure 4-61. 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 a value of **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 **Saved view list** button from the **View** toolbar. From the drop-down list, select the **Default Orientation** option.
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 4-62. You can also use the middle mouse button to spin the model.

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

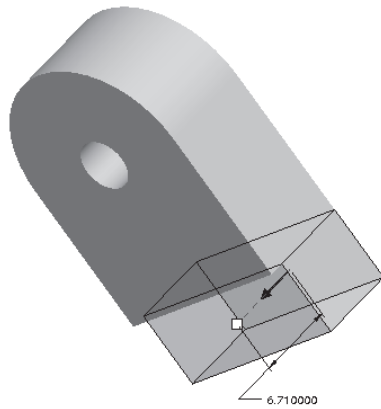


Figure 4-61 Arrow showing the direction of feature depth

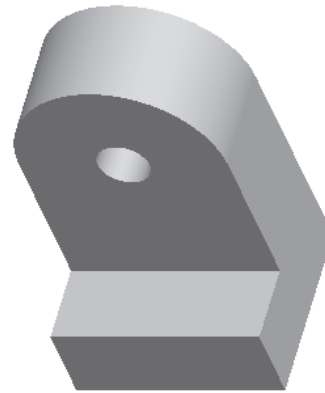


Figure 4-62 Second extruded feature with the base feature

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

- Using the left mouse button, select the front face of the second feature highlighted in Figure 4-63. The boundary of the selected front face is highlighted red in 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 3.0, 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.

- In the **Translation** dimension box, under the **Offset** area of the **DATUM PLANE** dialog box, enter a value of **-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. The datum plane named **DTM1** is created, as shown in Figure 4-64.

- Choose the **OK** button.

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 needs to be extruded should be created on the datum plane **DTM1**.

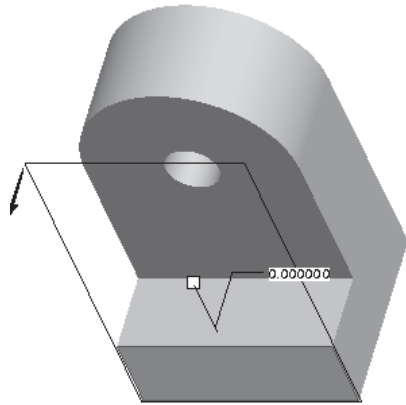


Figure 4-63 The selected face and the arrow showing the positive direction of datum plane

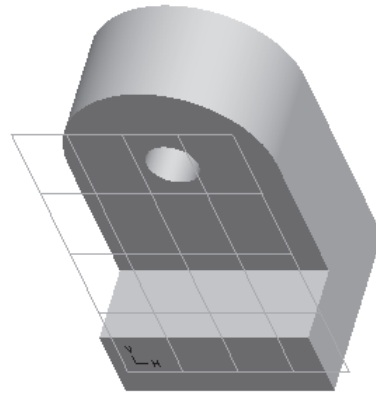


Figure 4-64 The highlighted datum plane

1. Choose the **Extrude Tool** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Placement** tab and from the slide-up 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 4-65. 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 4-66.
6. Choose the **Continue with current section** button to exit the sketcher environment.

The **Extrude** dashboard is enabled below the drawing area. Turn the model display to **Shading**. Use the middle mouse button to orient the model, as shown in Figure 4-67. This orientation gives a better view of the model.

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

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



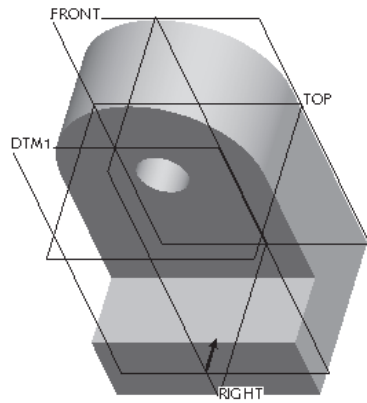


Figure 4-65 Arrow on **DTM1** showing the direction of viewing the sketching plane

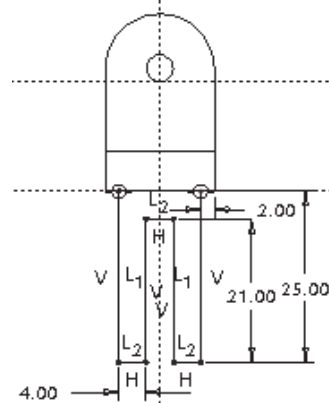


Figure 4-66 Sketch with dimensions and constraints of the third feature

8. 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.
9. Now, choose the **Build feature** button from the **Extrude** dashboard to confirm the feature creation.

The default view that is displayed when you choose the **Default Orientation** option from the **Saved view list** button is shown in Figure 4-68.

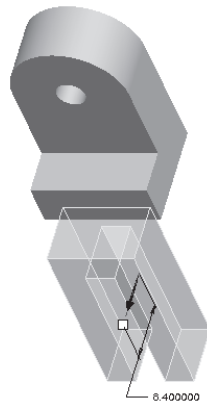


Figure 4-67 Arrow showing the direction of material addition

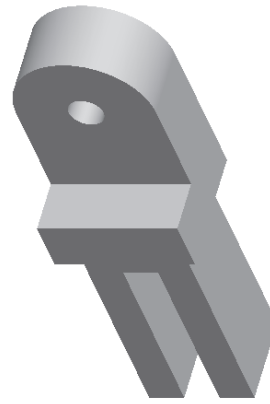


Figure 4-68 Model after creating the third

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 **Extrude** dashboard, the circular cut will be created. The sketching plane for the cut feature is shown in Figure 4-69.

**Note**

The circular cut feature can also be created using the *Hole* tool, which will be discussed in Chapter 5.

1. Choose the **Extrude Tool** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Remove Material** button from the **Extrude** dashboard.
3. Choose the **Placement** tab and from the slide-up panel, choose the **Define** button; the **Sketch** dialog box is displayed.
4. Select the face shown in Figure 4-69 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; you enter 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 4-70.

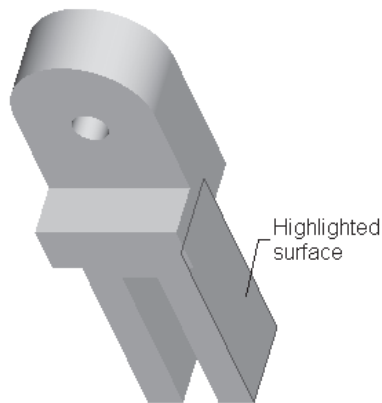


Figure 4-69 Sketching plane for hole feature

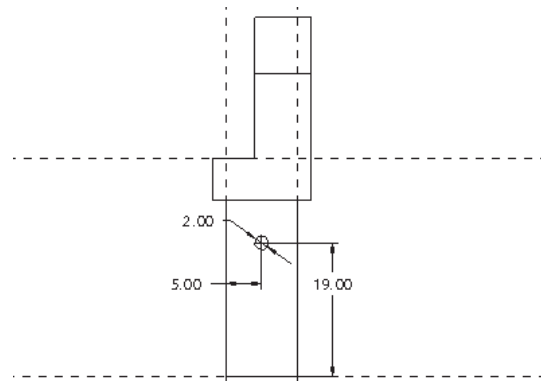


Figure 4-70 Sketch and dimensions for cut

2. Choose the **Continue with current section** button and turn the model display to **Shading**.
3. Choose the **Saved view list** button from the **View** toolbar and from the drop-down list, select the **Default Orientation** option. The model is displayed, as shown in Figure 4-71. Two yellow arrows also appear on the model. One arrow shows the direction of feature creation and the other arrow shows the direction with respect to the sketch from where the material will be removed. The arrows should point, as shown in Figure 4-71.

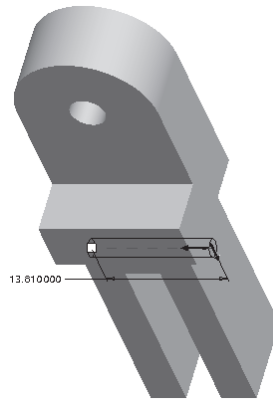


Figure 4-71 The two arrows on the cut feature

4. Choose the **Options** tab in the **Extrude** dashboard; the **Depth** slide-up 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 4-72.

Saving the Model

1. Choose the **Save the active object** button from the **File** toolbar and save the model. The order of feature creation can be seen from the **Model Tree** shown in Figure 4-73.

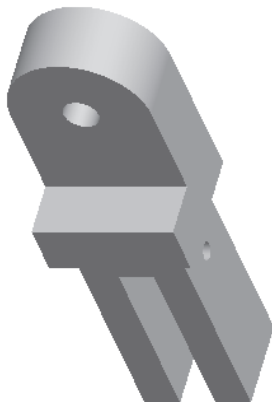


Figure 4-72 Completed model for Tutorial 1

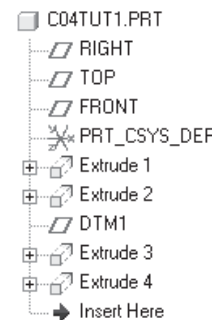


Figure 4-73 The Model Tree for Tutorial 1

Tutorial 2

In this tutorial, you will create the model shown in Figure 4-74. This figure also shows the front, top, and right-side views of the solid model. **(Expected time: 45 min)**

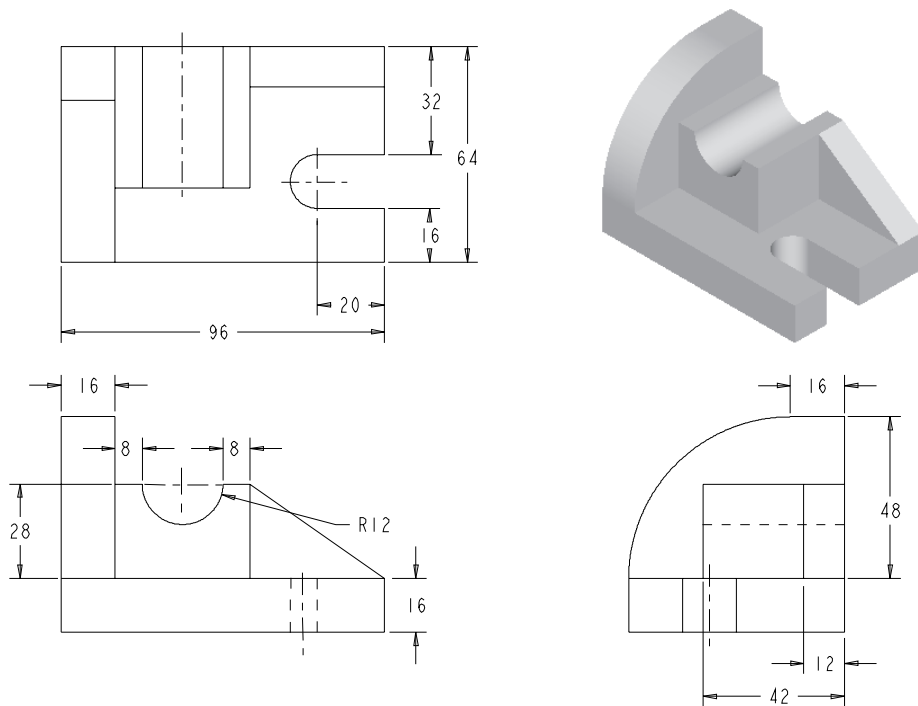


Figure 4-74 Top, front, right-side, and isometric views of the model

The following steps are required to complete this tutorial:

- Examine the model and determine the number of features in it, refer to Figure 4-74.
- Create the base feature, refer to Figures 4-75 and 4-76.
- Create the second feature on the left face of the base feature, refer to Figures 4-77 through 4-81.
- Create the third and fourth features on the same plane, but with different extrusion depths, refer to Figures 4-82 through 4-89.

Set the working directory if required.

Starting a New Object File

- Start a new part file and name it as **c04tut2**. 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 Tool** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Placement** tab and from the slide-up 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 4-75. When this sketch is extruded, it will create the base feature with the slot, as shown in Figure 4-76.

1. Draw the sketch using various sketcher tools and add the required constraints and dimensions to it. Modify these dimensions, as shown in Figure 4-75.
2. Choose the **Continue with the current section** button to exit the sketcher environment.
3. Choose the **Default Orientation** option from the **Saved view list** button of the **View** toolbar.

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 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 4-76. You can use the middle mouse button to spin the object to view it from various directions.

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.

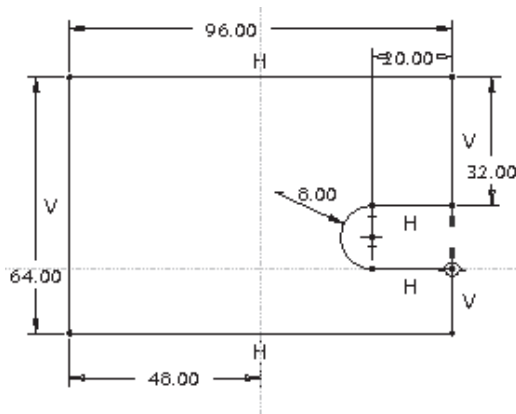


Figure 4-75 Sketch with dimensions and constraints for the base feature

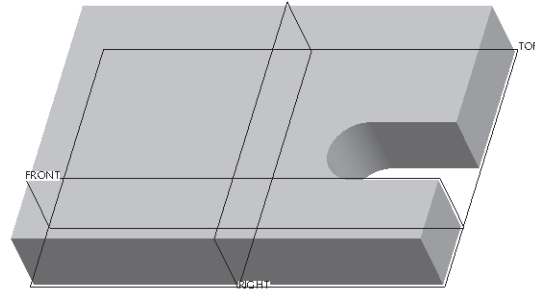


Figure 4-76 Base feature of the model

1. Choose the **Extrude Tool** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Placement** tab and from the slide-up panel, choose the **Define** button; the **Sketch** dialog box is displayed.
3. Using the middle mouse button, spin the model, as shown in Figure 4-77.
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, as shown in Figure 4-77.
6. Select the top face of the base feature shown in Figure 4-78 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.

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. Close the **References** dialog box that is displayed on the top right corner of the screen.

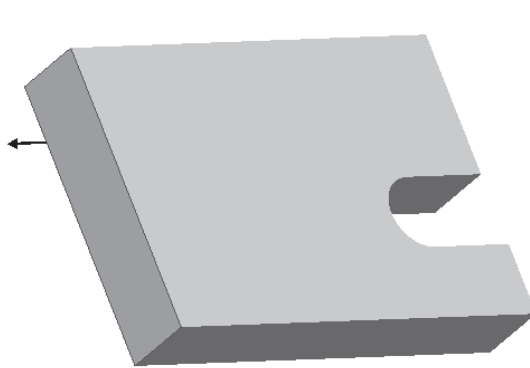


Figure 4-77 Arrow showing the direction of viewing the model

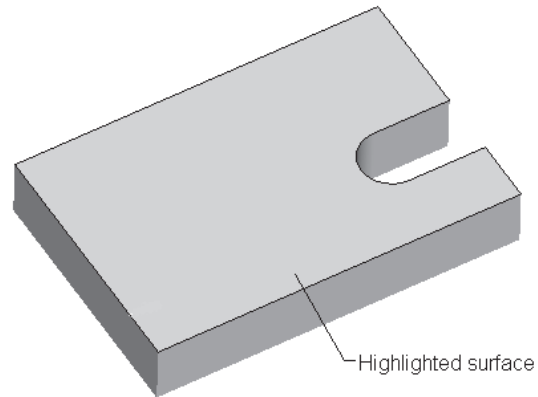


Figure 4-78 Surface selected to be at the top

2. Draw the sketch using various sketcher tools, as shown in Figure 4-79. The sketch is dimensioned automatically and some weak dimensions are assigned to it.
3. Apply the constraints and modify the weak dimensions to the dimensions shown in Figure 4-79.

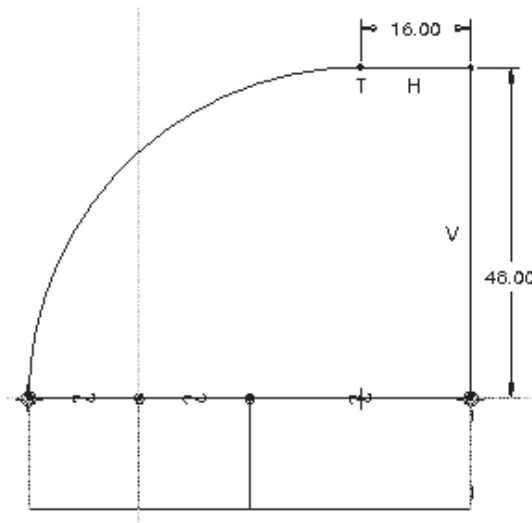


Figure 4-79 Sketch with dimensions and constraints for the second feature

4. After the sketch is complete, turn the model display to **Shading** and choose the **Continue with the current section** button.

The **Extrude** dashboard is displayed below the drawing area. Use the middle mouse button to orient the model, as shown in Figure 4-80. A yellow arrow is also displayed on the model, indicating the direction of extrusion.

5. Type a value of **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.
6. Choose the **Build feature** button from the **Extrude** dashboard. The second feature is completed and is shown in Figure 4-81. You can use the middle mouse button to spin the model to view it from various directions.

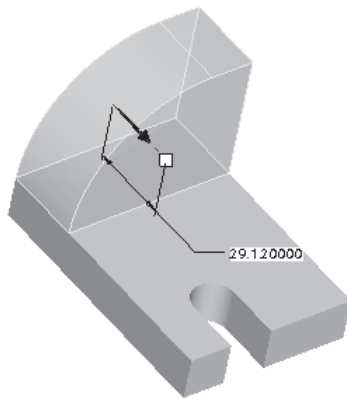


Figure 4-80 Arrow showing the direction of material addition

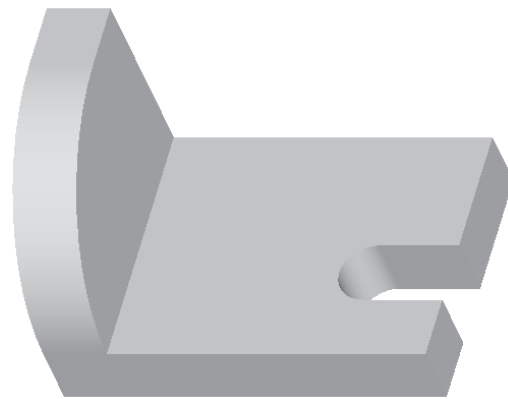


Figure 4-81 Model with the second extruded feature

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 Tool** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed below the drawing area.
2. Choose the **Placement** tab and from the slide-up panel, choose the **Define** button; the **Sketch** dialog box is displayed.
3. Using the middle mouse button, spin the model and select the face of the base feature shown in Figure 4-82 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 4-82. The arrow points in the direction of viewing the sketching plane.
5. Select the top face shown in Figure 4-83 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.

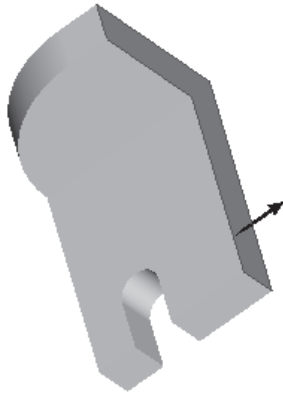


Figure 4-82 Arrow showing the direction of viewing the model

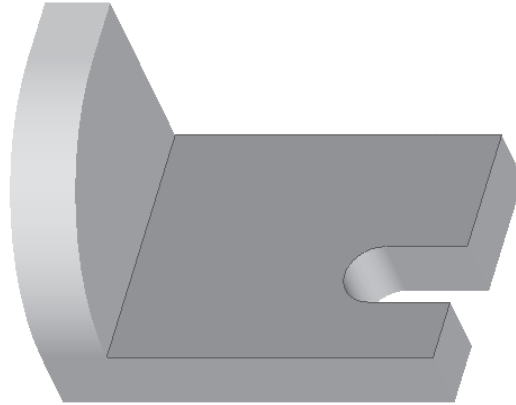


Figure 4-83 Face selected to be at the top

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

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 4-84.
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 4-84.
3. Choose the **Continue with the current section** button to exit the sketcher environment. The **Extrude** dashboard is enabled below the drawing area. Turn the display to **Shading**.
4. Use the middle mouse button to orient the model, as shown in Figure 4-85. 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. Type a value of **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.
6. Choose the **Build feature** button from the **Extrude** dashboard to confirm the feature creation. The default view that is displayed when you choose the **Default Orientation** option from the **Saved view list** button is shown in Figure 4-86.

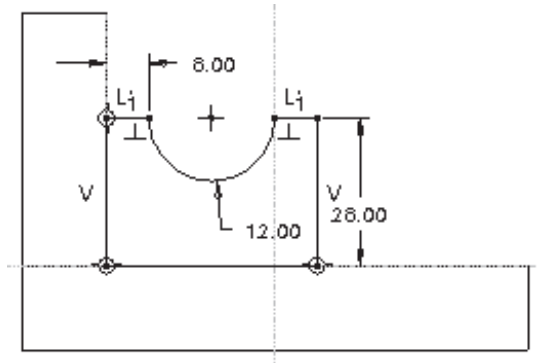


Figure 4-84 Sketch with dimensions and constraints

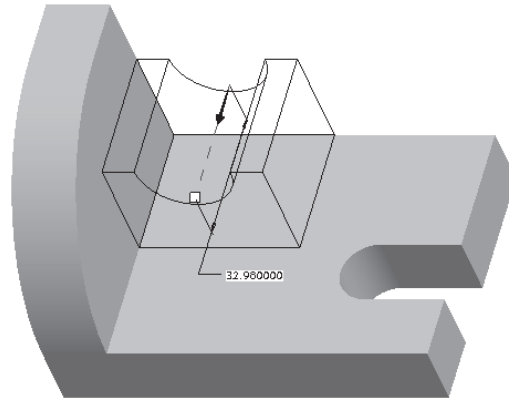


Figure 4-85 Arrow showing the direction of material addition

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 Tool** 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 yellow arrow is displayed in the drawing area, as shown in Figure 4-87.

By specifying the **Use Previous** button, you are using the previous sketching plane that was defined for the previous feature. A yellow arrow is displayed on the face and points in the direction of viewing the sketch, as shown in Figure 4-87.

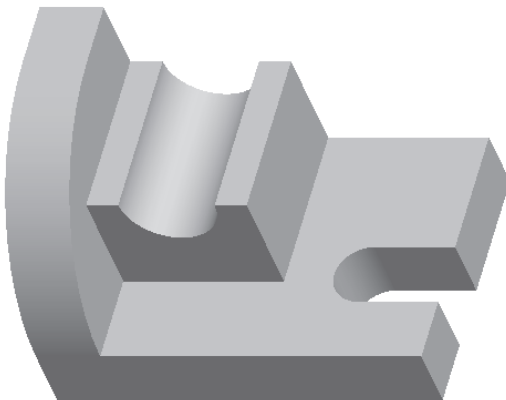


Figure 4-86 Model with the second extruded feature

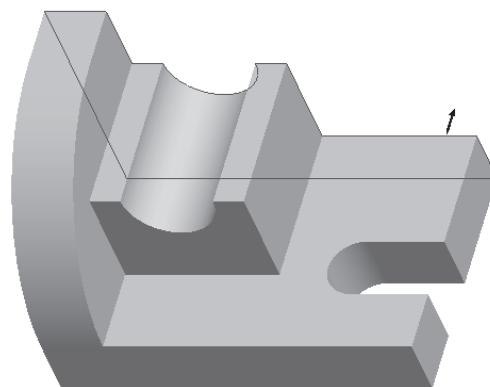


Figure 4-87 Arrow pointing in the direction of viewing the sketch plane

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 4-88. 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 4-88. 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 **Continue with the current section** button. The **Extrude** dashboard is enabled. Using the middle mouse button, orient the model, as shown in Figure 4-89.

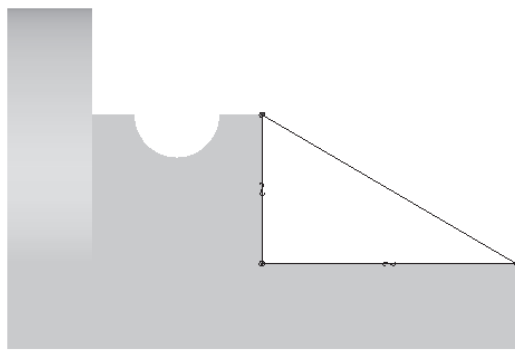


Figure 4-88 Sketch and constraints for the last feature

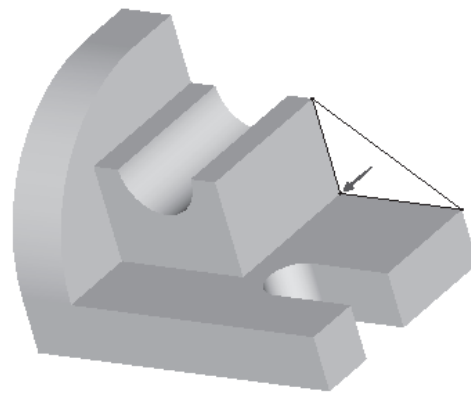


Figure 4-89 Arrow showing the direction of material addition

3. Enter a value of **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, when you choose the **Default Orientation** option from the **Saved view list** button from the **View** toolbar, is shown in Figure 4-90.

Saving the Model

1. Choose the **Save the active object** button from the **File** toolbar and save the model. The order of feature creation can be seen from the **Model Tree** shown in Figure 4-91.



Figure 4-90 Completed model for Tutorial 2

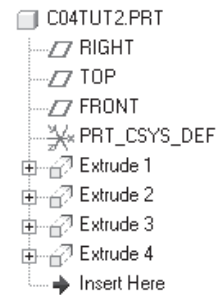


Figure 4-91 The *Model Tree* for Tutorial 2

Tutorial 3

In this tutorial, you will create the model shown in Figure 4-92. This figure also shows the front, top, and right-side views of the solid model.
(Expected time: 45 min)

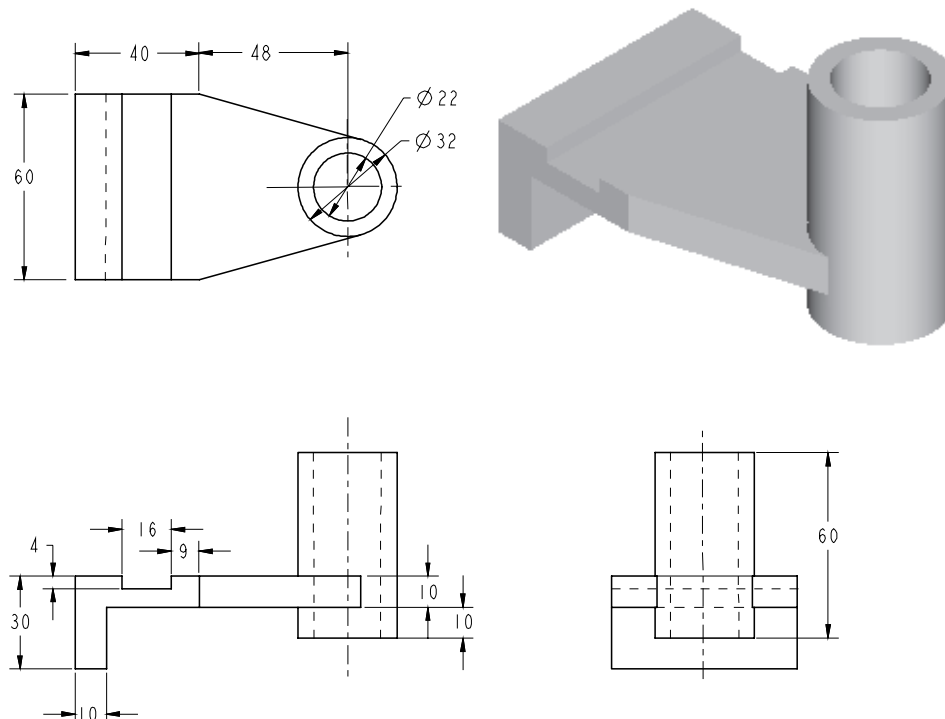


Figure 4-92 Top, front, right-side, and isometric views of the model

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 4-92.
- b. Create the base feature, refer to Figures 4-93 and 4-94.
- c. Create the second feature, refer to Figures 4-95 through 4-98.
- d. Create the hollow cylindrical feature on an offset datum plane, refer to Figures 4-99 through 4-104.

Starting a New Object File

1. Set the working directory, if required, and open a new part file with the name **c04tut3**. The three default datum planes are displayed in the drawing area, if the **Datum planes on/off** button is turned on.

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 Tool** button from the **Base Features** toolbar.
2. Choose the **Placement** tab and from the slide-up 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 by 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 4-93. When this sketch is extruded, it will create the base feature.

1. Draw the sketch using various sketcher tools, as shown in Figure 4-93.
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 4-93.
3. Choose the **Continue with the current section** button; the **Extrude** dashboard is displayed.
4. Choose the **Default Orientation** option from the **Saved view list** button of the **View** toolbar. The model is reoriented and a yellow arrow is also displayed on it, indicating the direction of extrusion.

5. Type a value of **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 4-94. You can use the middle mouse button to spin the model to view it from various directions.

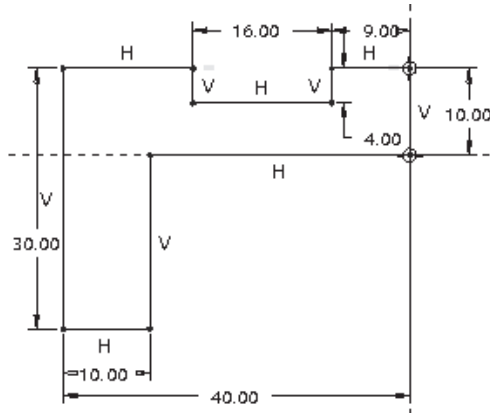


Figure 4-93 Sketch with dimensions and constraints for the base feature

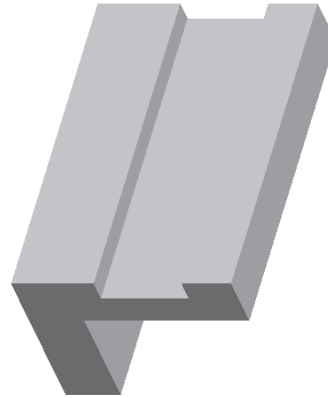


Figure 4-94 Base feature of the model

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 Tool** button from the **Base Features** toolbar.
2. Choose the **Placement** tab and from the slide-up panel, choose the **Define** button; the **Sketch** dialog box is displayed.
3. Select the top face of the base feature shown in Figure 4-95 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 4-95.
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 4-96. Before drawing the sketch, turn the model display to **No Hidden**.

1. Close the **References** dialog box that is displayed on the top right corner of the screen.
2. 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 4-96. This center line helps in dimensioning the sketch. Add the required constraints and dimensions to the sketch, as shown in Figure 4-96. Before exiting the sketcher environment, turn the model display to **Shading**.

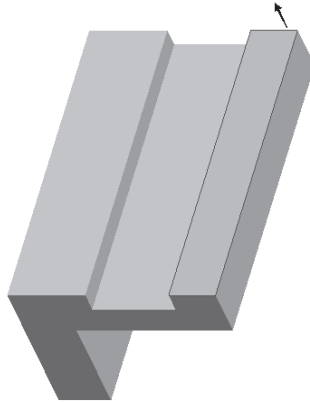


Figure 4-95 Arrow pointing from the sketching plane in the direction of feature creation

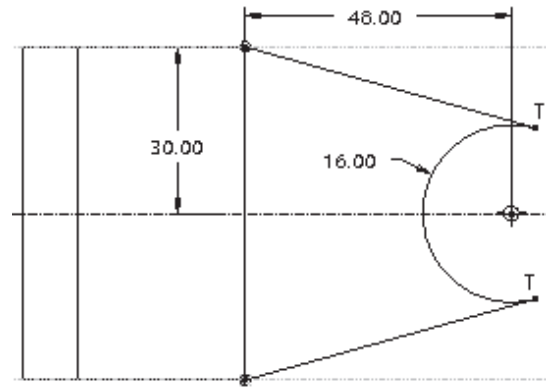


Figure 4-96 Sketch with dimensions and constraints

3. Choose the **Continue with the current section** button. The **Extrude** dashboard is enabled. Use the middle mouse button to orient the model, as shown in Figure 4-97.
4. Type a value of **10** in the dimension box present on the **Extrude** dashboard and press ENTER.
5. Now, choose the **Build feature** button to confirm the feature creation. The default trimetric view of the extruded feature is shown in Figure 4-98.

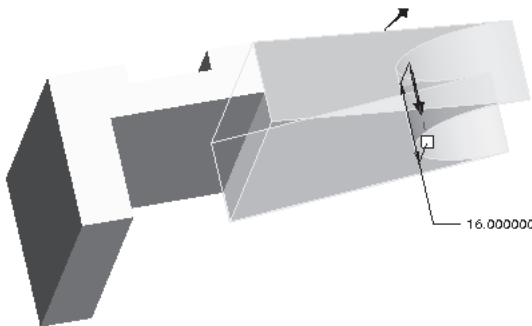


Figure 4-97 Arrow showing the direction of material addition

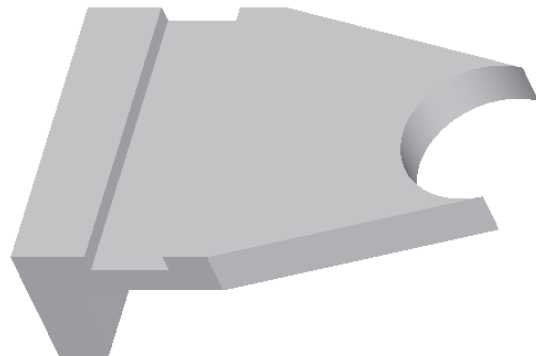


Figure 4-98 Model with the second extruded feature

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 4-99.



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 **Datum Plane Tool** 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 a value of **10**.
4. Choose the **OK** button from the **DATUM PLANE** dialog box.

Datum plane **DTM1** is created, as shown in Figure 4-100 and will be selected as the sketching plane for creating the sketch.

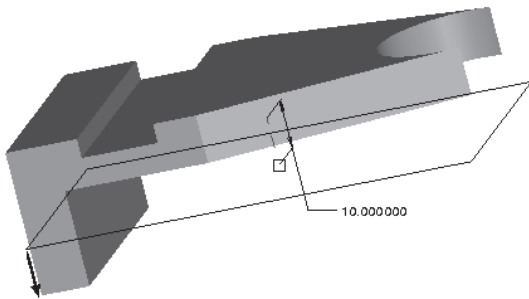


Figure 4-99 Creating the datum plane

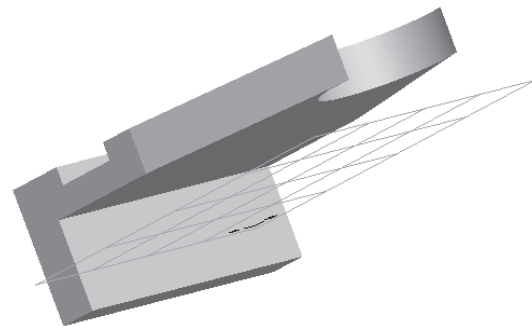


Figure 4-100 Model after creating the datum plane

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 Tool** button from the **Base Features** toolbar.

2. Choose the **Placement** tab and from the slide-up panel choose the **Define** button. 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 4-101.
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. Close the **References** dialog box that is displayed on the top right corner of the screen.
2. Draw the sketch using various sketcher tools and add the required constraints and dimensions to it, as shown in Figure 4-102.

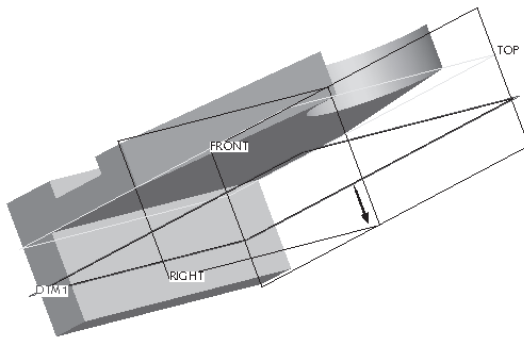


Figure 4-101 Direction of viewing the sketching plane

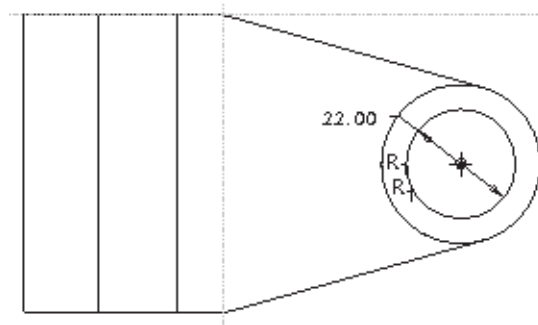


Figure 4-102 Sketch with dimensions and constraints

3. Choose the **Continue with the current section** 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 4-103. This view gives you a better view of the sketch in the 3D space.

4. Enter a value of **60** in the dimension box present on the **Extrude** dashboard.

5. Choose the **Build feature** button to confirm the feature creation. The default trimetric view of the complete model is shown in Figure 4-104.

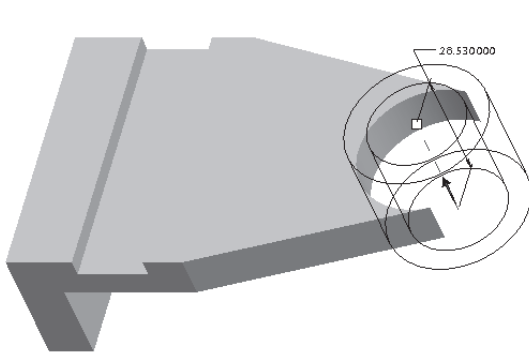


Figure 4-103 Preview of the feature

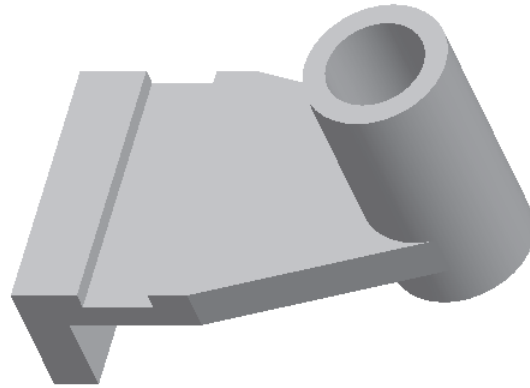


Figure 4-104 Final model of Tutorial 3

Saving the Model

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

Self-Evaluation Test

Answer the following questions and then compare your answers with 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 a 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 an existing face of a feature. (T/F)
6. While creating a datum feature, the _____ selected from the model help you to choose from the available constraints that you can apply 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 **Saved 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 **Datum Point Tool** 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 the orientation of references. (T/F)
10. Unlike the datum planes constraint options, all datum axes constraint options are stand-alone. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 4-105. The dimensions, front view, and right-side view of the model are shown in Figure 4-106. **(Expected time: 45 min)**

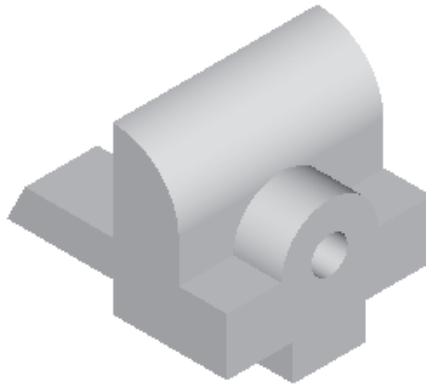


Figure 4-105 Isometric view of the model

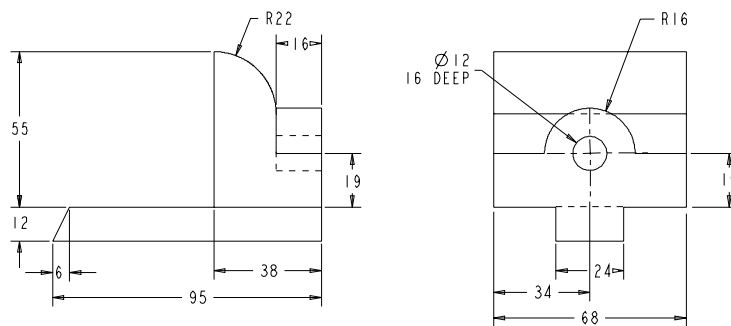


Figure 4-106 Front and right-side views of the model

Exercise 2

Create the model shown in Figure 4-107. The dimensions, and the front, top, right-side, and isometric views of the model are also shown in the figure. **(Expected time: 45 min)**

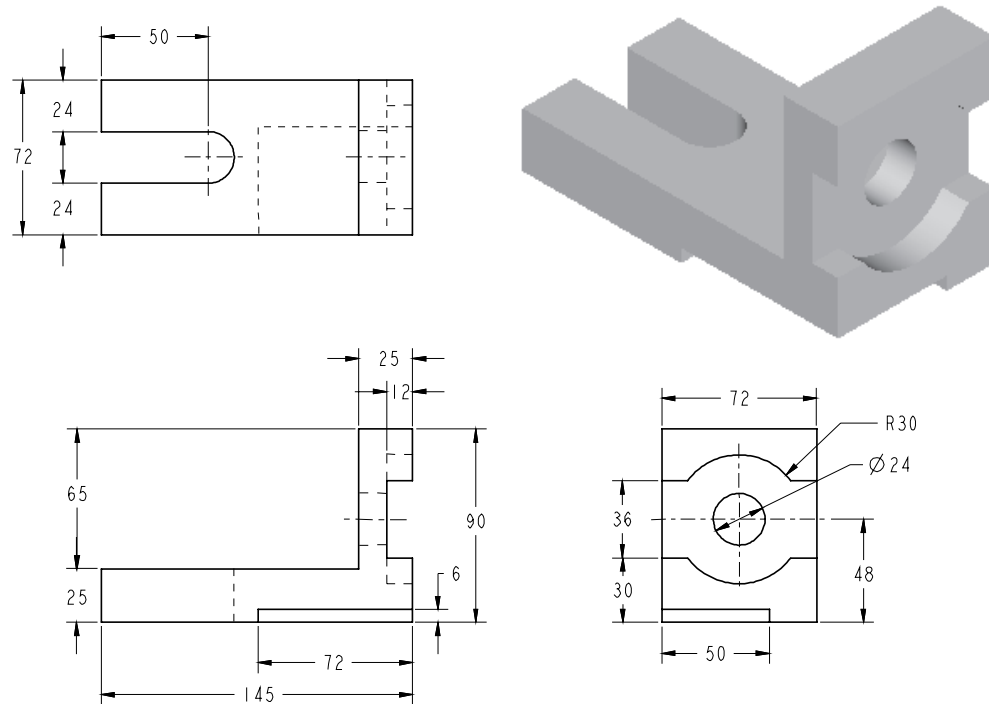


Figure 4-107 Top, front, right-side, and isometric views of the model

Exercise 3

Create the model shown in Figure 4-108. The top, front section, and two auxiliary views of the model are shown in Figure 4-109. (Estimated time: 1 hr)

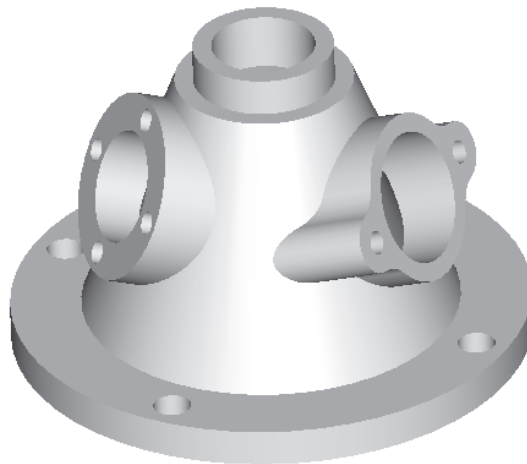


Figure 4-108 The 3D view of the model

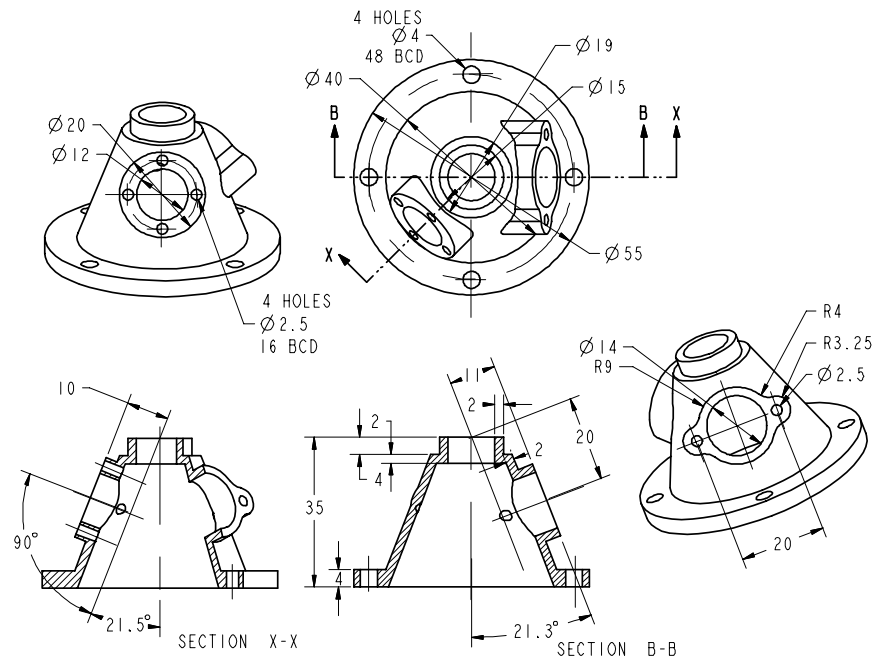


Figure 4-109 Top, front section, and two auxiliary views of the model

Hint to create the two side features in Exercise 3:

Feature on the right

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 21.3 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.

Feature on the left

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 the 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 4-110. The dimensions, the left-side view, auxiliary view, front view, and isometric view of the model are also shown in Figure 4-111.

(Expected time: 45 min)

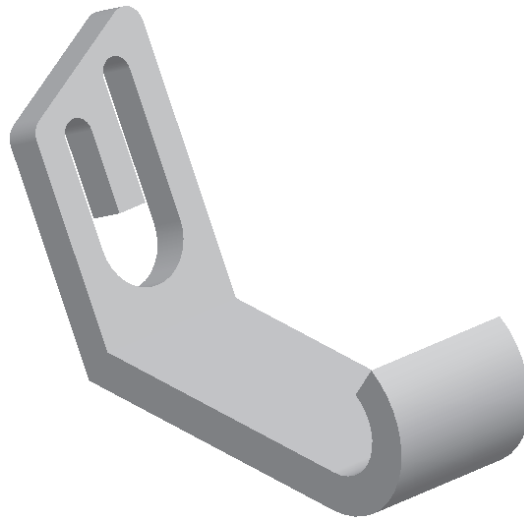


Figure 4-110 Isometric view of the model

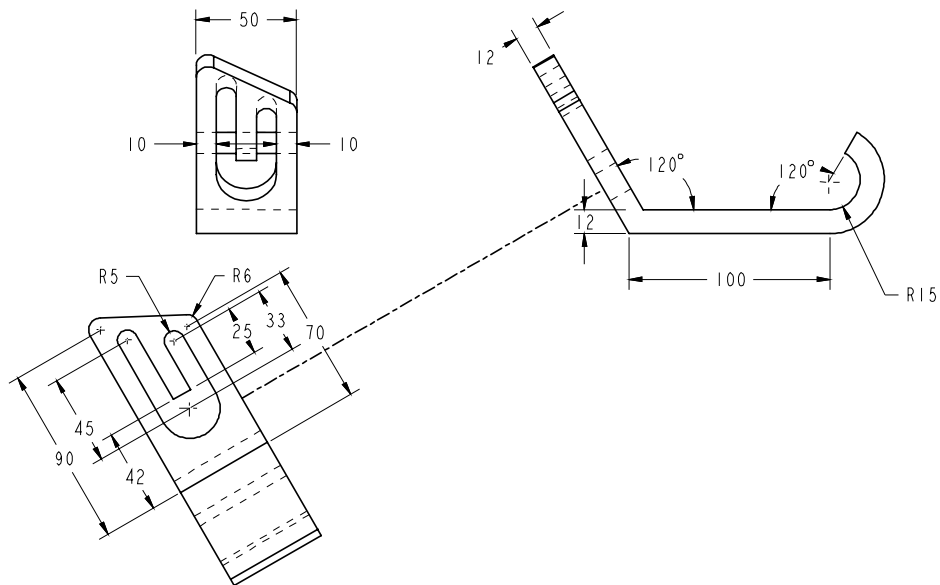


Figure 4-111 Left-side view, auxiliary view, front view, and isometric view of the model

Answers to Self-Evaluation Test

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