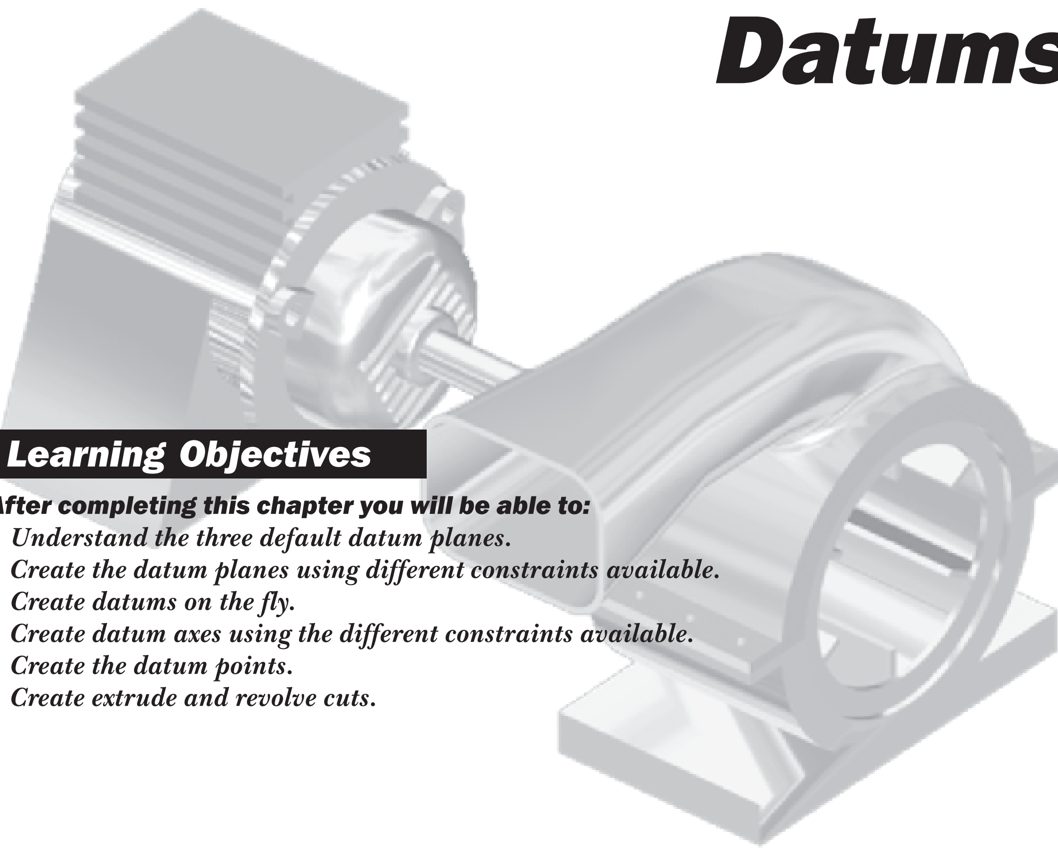
Chapter 4

Datums



Learning Objectives

After completing this chapter you will be able to:

- *Understand the three default datum planes.*
- *Create the datum planes using different constraints available.*
- *Create datums on the fly.*
- *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 a model, they act as reference for sketching of a feature, orientation of a model, assembling components, and so on. Remember that the datums play a very important role in creating complex models in Pro/ENGINEER and therefore you must have a good understanding of datums. Datums are considered to be features but not model geometry. In Pro/ENGINEER, datums exist as datum plane, datum curve, datum point, datum coordinate system, datum graph, and so on.

Default Datum Planes

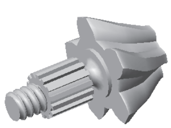
When you enter the Part mode or the Assembly mode, the three datum planes are by default displayed on the graphics screen. These datum planes are mutually perpendicular to each other. These are known as the default datum planes. The only difference between the default datum planes of the Part mode and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

The default datum planes in the Part mode are named as **FRONT**, **TOP**, and **RIGHT**. In case of Assembly mode, the default datum planes are named as **ASM_FRONT**, **ASM_TOP**, and **ASM_RIGHT**. However, the names of the default datum planes can be changed as required. To change the names, choose **PART > Set Up > Name**. You will be 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 this window.

There are two sides of a datum plane, colored yellow and red. Generally, the protrusion takes place toward the yellow side of the datum plane and the cut takes place towards the red side. This is the reason, while extruding a section, by default, the red arrow always points out of the screen. You can change the colors of the datum planes according to your convenience. The colors of the datum planes help in identifying the direction of feature orientation.

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 the other features of the model are created. Since all the 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 4-1 shows a simple model that consists of two features that require two different planes.



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

In Figure 4-1 any of the two features that are defined on two different planes can be considered as the base feature. However, in this discussion the base feature that is decided to be created is shown in Figure 4-2. After creating the base feature, the next feature has to be created. For the next feature, a sketching plane has to be defined. Therefore, an additional plane will be created on which you can draw the sketch for the second feature.

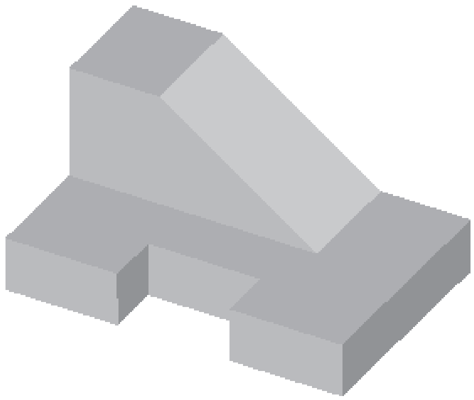


Figure 4-1 *Model having two extruded features*

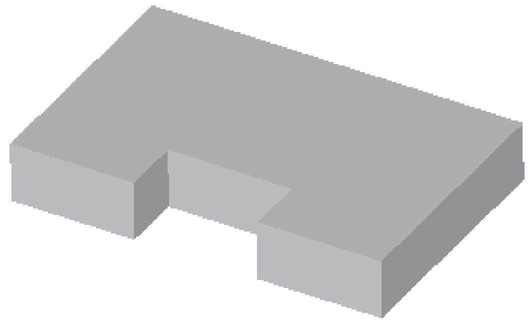


Figure 4-2 *Base feature of the model*

As shown in Figure 4-3, the plane that is used for the creation of the base feature is highlighted by a mesh. To create the second feature, a new plane is created that is shown in Figure 4-4. The sketch of the second feature is drawn on this plane and this is the reason, the front planar surface is coplanar with the datum plane.

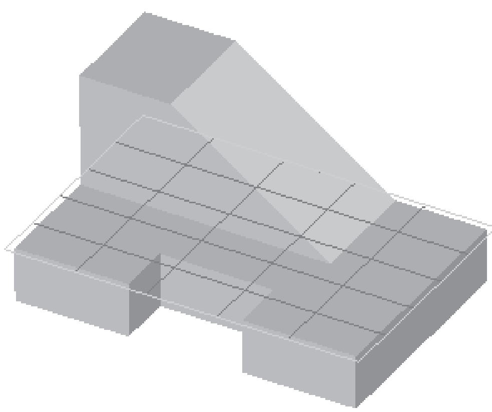


Figure 4-3 *Plane selected for the base feature*

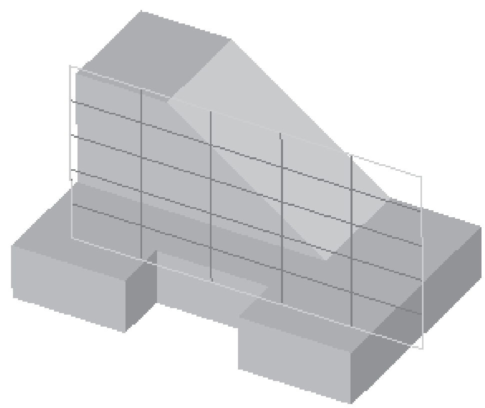


Figure 4-4 *Plane selected for the second feature*

DATUM OPTIONS

After discussing the default datum planes that are the first feature in the Part mode, you must know the different features created using the datum options. Datums are also considered as features that have no geometry. Figure 4-5 shows the **Datum** toolbar. Figure 4-6 shows the method of invoking different types of datum features from the menu bar.

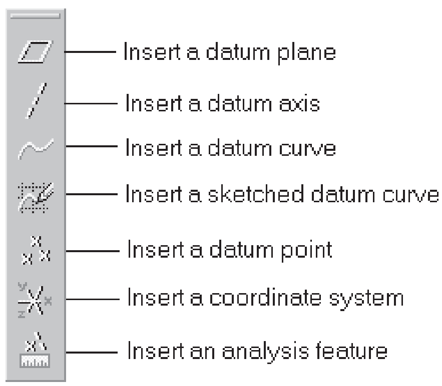


Figure 4-5 ***Datum*** *toolbar*

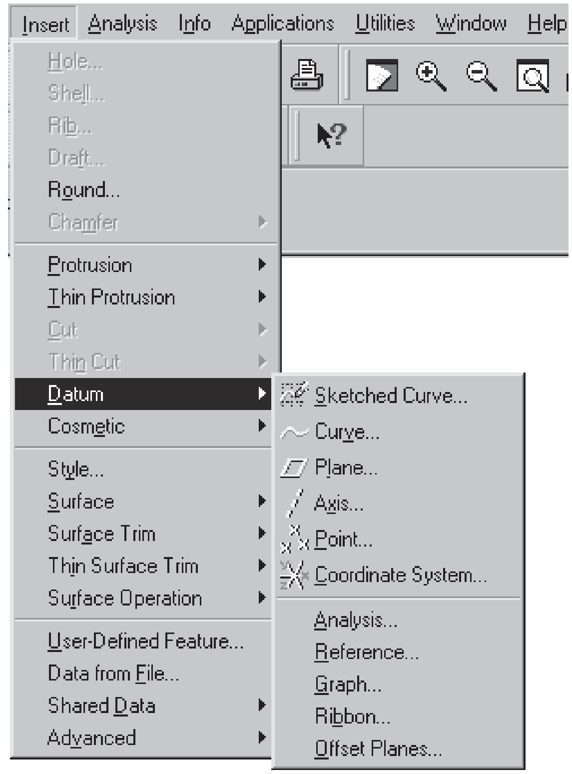


Figure 4-6 Invoking the datum options from the ***Insert*** *menu in the menu bar*

Datum Planes

We can create datum planes other than the three default datum planes 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 the datum planes using the different options, it is important for you to understand the use of datum planes. Some of the uses of datum planes are listed below:

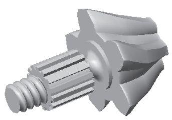
1. Datum planes are used as sketching planes to create sketches for the different features of a model.

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

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

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

Pro/ENGINEER provides you with different options to create datum planes other than the default datum planes. With this release of Pro/ENGINEER, datums can even be created while you are in the sketcher environment. When you choose **Insert > Datum > Plane** from the menu bar or **Insert a datum plane.** button from the **Datum** toolbar, the **DATUM PLANE** submenu appears with the different options to create datum planes. Similarly, separate buttons for creating datum axis, datum curve, datum point, datum coordinate system, and so on 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 increases, the need for additional datum planes arises.*

Figure 4-7 shows the different options available in the **DATUM PLANE** submenu to create datum planes. Some of the options are standalone and some require more than one constraint to define a datum plane, that is, they are applied in pairs. The standalone options are constraints that are sufficient by themselves to constrain a datum plane definition.

***Figure 4-7 DATUM PLANE** submenu*

Through Option

The **Through** option is used to create a datum plane through any specified axis, edge, curve, point/vertex, plane, cylinder, or coordinate system. This option can be used in combination with other different sub-options that are available in the **DATUM PLANE** submenu. However, the combinations of options that you can use as standalone are **Through > AxisEdgeCurv**, **Through > Plane**, and **Through > Cylinder**. Figure 4-8 shows the datum plane constraint combinations using the **Through** option. 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 4-8, first preference is given to the text written in first column and then the text in the first row should be read. For example, if you want to make a datum plane that is passing through a cylinder and normal to a plane then look for **Through** in the first column and then for **Cylinder** in second column. Now, look for **Normal** in the first row and for **Plane** in the second row. After finding both the combination 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 4-8, remember that the constraints that are not standalone have to be applied in pairs. When the constraint applied is sufficient to constrain a datum plane, the message, **Datum Plane is fully constrained. Select "Done", "Quit" or "Restart"** is displayed in the **Message Area**.

Figure 4-9 shows that the cylindrical surface and the default datum planes are 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 4-10.

DATUM PLANE CONSTRAINT COMBINATIONS (USING THROUGH OPTION)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		AxisEdge Curv	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Through	AxisEdge Curv	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
	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	Yes

*Figure 4-8 Datum plane constraint combinations using the **Through** option*

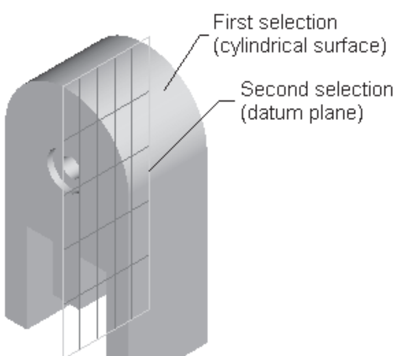


Figure 4-9 Selecting a cylindrical surface and a default datum plane to create a datum plane

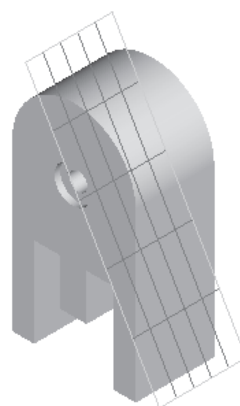


Figure 4-10 Resultant datum plane passing through the center of a cylinder and at an angle

Normal Option

The **Normal** option is used to create a datum plane normal to any specified axis, edge, curve, or plane. This option is used in combination with other different sub-options that are available in the **DATUM PLANE** submenu. The **Normal** option in combination with any of the sub-options cannot be used as standalone. Figure 4-11 shows the datum plane constraint combinations using the **Normal** option. The possible combinations of datum plane creation are referred to as **Yes** and the combinations that are not possible are referred to as **No**.

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

Figure 4-11 *Datum plane constraints combinations using the* ***Normal*** *option*

Figure 4-12 shows a planar surface and a cylindrical surface. The planar surface is selected as the normal surface and the cylindrical surface is selected to be tangent to the datum plane. The datum plane that is created is shown Figure 4-13.

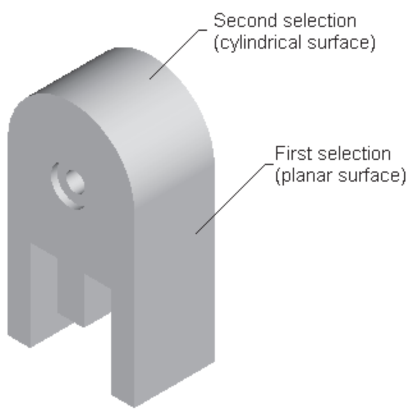


Figure 4-12 *Selecting a planar surface and a cylindrical surface to create a datum plane*

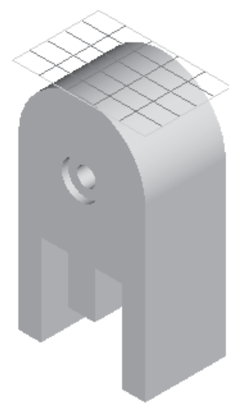


Figure 4-13 *Resultant datum plane*

Parallel Option

The **Parallel** option is used to create a datum plane parallel to any specified datum plane or planar surface. This option is used in combination with other different sub-options in the **DATUM PLANE** submenu. The **Parallel** option in combination with the **Plane** sub-option cannot be used as standalone. Figure 4-14 shows different datum plane constraint combinations using the **Parallel** option. 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 PARALLEL OPTION)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		AxisEdge Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Parallel	**Plane**	**Yes**	**Yes**	**Yes**	**No**	**No**	**No**	**Yes**	**No**

Figure 4-14 *Datum plane constraint combinations using the **Parallel** option*

Figure 4-15 shows the selection of a default datum plane and an axis to create a datum plane. The resultant datum plane is parallel to the selected datum plane and passes through the axis as shown in Figure 4-16.

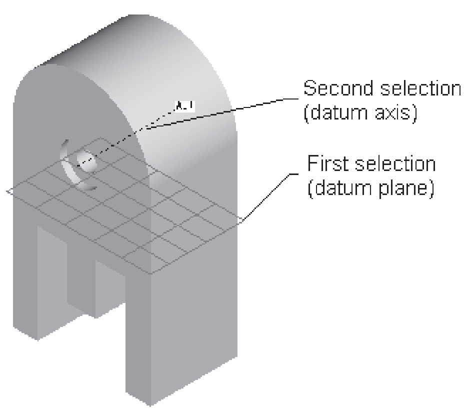


Figure 4-15 *Selecting a datum plane and an axis to create a datum plane*

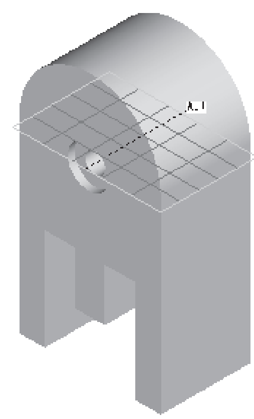


Figure 4-16 *Resultant datum plane*

Offset Option

The **Offset** option 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 different sub-options available in this submenu. However, the **Offset** > **Plane** option can be used as standalone. This option is used to create a datum plane at some specified parameters. The parameters that are required to specify the offset distance are discussed below:

Thru Point

The **Thru Point** option is used to specify a point on the model through which the datum plane will pass.

Enter Value

The **Enter Value** option is used to specify an offset distance and in the case of angular planes you have to specify the angle. These values are entered in the **Message Input Window** that appears. An arrow appears on the model that shows the positive direction of the offset distance or angle.

Figure 4-17 shows different datum plane constraint combinations using the **Offset** option. 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 OFFSET OPTION)		**Through**			**Normal**	**Parallel**	**Angle**	**Tangent**	**Standalone Constraints**
		AxisEdge Curve	**Point/ Vertex**	**Cylinder**	**Plane**	**Plane**	**Plane**	**Cylinder**	
Offset	**Plane**	**Yes**+	**Yes**+	**No**	**No**	**No**	**No**	**No**	**Yes**
	Coord System	**No**	**No**	**No**	**No**	**No**	**No**	**No**	**Yes**

+ Possible only when you select **Thru Point** from **OFFSET** submenu

Figure 4-17 *Datum plane constraint combinations using the **Offset** option*

Figure 4-18 shows the selection of a default datum plane and a vertex to define an offset datum plane. The resultant datum plane is at an offset to the selected datum plane and passes through the vertex as shown in Figure 4-19.

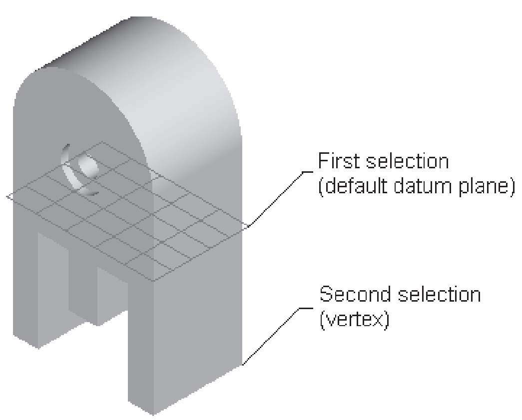


Figure 4-18 *Selecting a datum plane and an axis to create a datum plane*

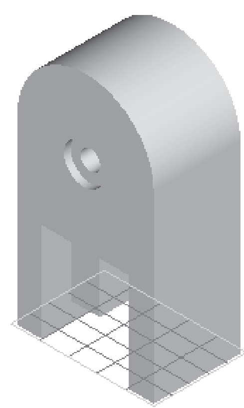


Figure 4-19 *Resultant datum plane*

Angle Option

The **Angle** option is used to create datum planes through any specified plane. This option is used with other sub-options in the **DATUM PLANE** submenu to create different types of datum planes. The **Angle** > **Plane** combination of options cannot be used as standalone. The value for angle is entered in the **Message Input Window** that appears when all the constraints are defined. Figure 4-20 shows the datum plane constraint combinations using the **Angle** option. 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 ANGLE OPTION)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		AxisEdge Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Angle	Plane	Yes	No	Yes	No	No	No	No	No

*Figure 4-20 Datum plane constraint combinations using the **Angle** option*

Figure 4-21 shows the selection of a planar surface, an edge and a vertex to create a datum plane that is shown in Figure 4-22. The datum plane created is at an angle to the selected planar surface and passes through the selected edge and vertex. The vertex is selected by choosing the **Thru Point** option from the **OFFSET** submenu that is displayed when you choose **Done** from the **DATUM PLANE** submenu.

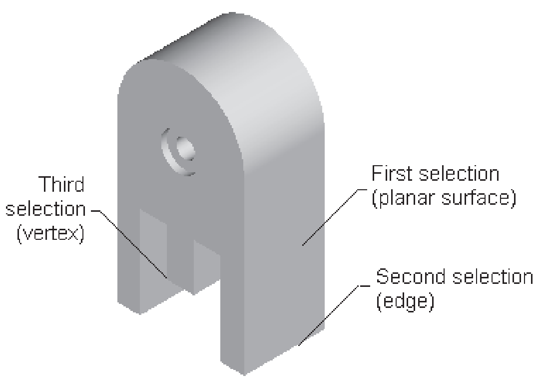


Figure 4-21 Selecting a datum plane and an axis to create a datum plane

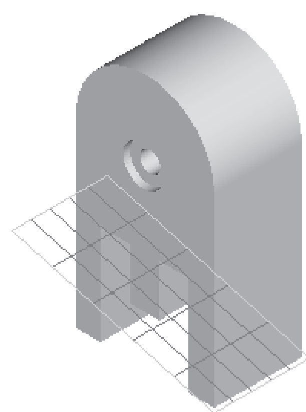


Figure 4-22 Resultant datum plane

Tangent Option

The **Tangent** option creates datum planes tangent to cylindrical features. This option is also used with the other options in the **DATUM PLANE** submenu to create different types of datum planes. Figure 4-23 shows the datum plane constraint combinations using the **Tangent** option. 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 OPTION)		Through			Normal	Parallel	Angle	Tangent	Standalone Constraints
		AxisEdge Curve	Point/ Vertex	Cylinder	Plane	Plane	Plane	Cylinder	
Tangent	Cylinder	Yes	No	No	Yes	Yes	No	No	No

*Figure 4-23 Datum plane constraints combination using the **Tangent** option*

BlendSection Option

The **BlendSection** option is used to create the datum planes by selecting the features. This option works as standalone.

Datum Planes Created "On The Fly"

The term "**On the Fly**" refers to the creation of a datum plane when the system prompts you to select or create a plane. At this step, the **SETUP PLANE** submenu is displayed. When you choose the **Make Datum** option from this submenu, the **DATUM PLANE** submenu is displayed. You can select the options from this submenu to create a datum plane. When you create a datum plane using the **Make Datum** option, the datum plane is neither visible on the graphics screen nor is displayed on the **Model Tree** once the feature is completed. This option of creating datum planes is referred to as "creating datum planes on the fly". This option is provided by Pro/ENGINEER in order to avoid cluttering of datum planes in a complex model.

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 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 features co-axially.

4. They are also used to create radial patterns. You will learn to create patterns in Chapter 6.

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 that 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 > Datum > Axis** from the menu bar or **Insert a datum axis.** button from the **Datum** toolbar, the **DATUM AXIS** submenu appears with different options to create datum axes as shown in Figure 4-24. The options in the **DATUM AXIS** submenu are explained next. These options are explained using an extruded model.

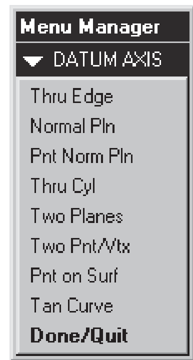


***Figure 4-24 DATUM AXIS** submenu*

Thru Edge Option

The **Thru Edge** option is used to create a datum axis through any selected edge. The selected edge must be straight for the creation of a datum axis. In Figure 4-25, **A_1** is the datum axis created using this option.

Note

Unlike the datum planes constraint options, all the datum axes constraint options are standalone.

*While creating a datum axis, some options in the **DATUM AXIS** submenu require datum points to be selected while constraining the datum axis. Therefore, while using options like **Pnt Norm Pln** and **Pnt on Surf** from the **DATUM AXIS** submenu, you need to create datum points.*

Normal Pln Option

The **Normal Pln** option is used to create a datum axis normal to any selected planar surface or datum plane. When you select a planar surface or a datum plane, you are prompted to select its placement location on the datum plane. After you select its placement location, you are prompted to select two edges, axes, datums, or planar surfaces to specify the linear dimension for the placement of the datum axis. When you select the first edge for the placement dimensions of the axis, the **Message Input Window** is displayed. A default value is displayed in the window. You can accept the default dimension or change it to the required value. Similarly, select the second edge for dimensioning and enter the dimension value in the window that appears. In Figure 4-26, **A_2** is the datum axis created using this option.

Pnt Norm Pln Option

The **Pnt Norm Pln** option creates a datum axis passing through a datum point and normal to any planar surface or datum plane. When you choose this option to create a datum axis, you

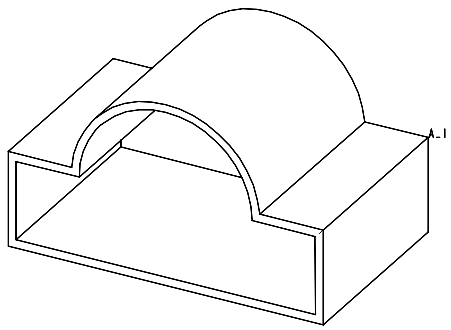


Figure 4-25 *Datum axis created along the edge*

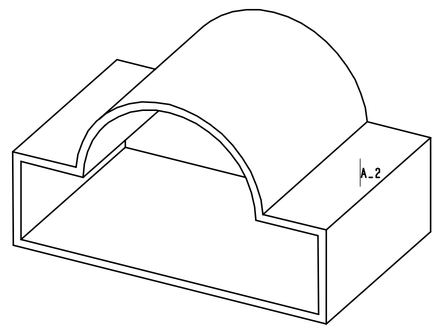


Figure 4-26 *Datum axis created normal to the plane*

are prompted to select a datum plane or a planar surface. Select a plane to which the datum axis will be normal. Now, you are prompted to select a datum point. Select a datum point to create an axis passing through the datum point. In Figure 4-27, **A_3** is the datum axis created using this option.

Thru Cyl Option

The **Thru Cyl** option is used to create a datum axis through a cylindrical or a round surface. When you choose this option from the **DATUM AXIS** submenu, you are prompted to select a revolved surface. The axis is automatically created around the revolved surface through an imaginary axis. In Figure 4-28, **A_4** is the datum axis that is created using this option.

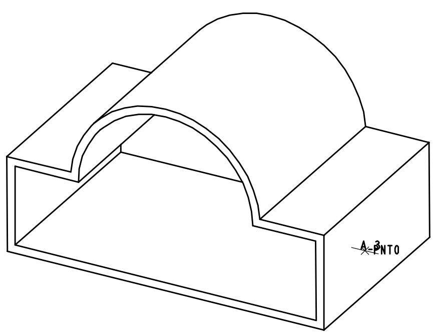


Figure 4-27 *Datum axis passing through the datum point and normal to the plane*

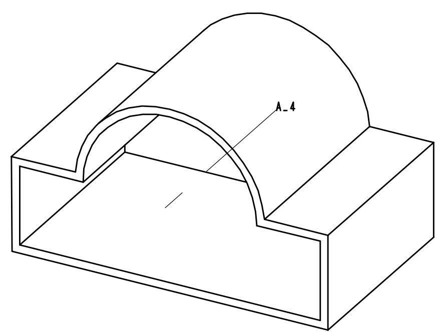


Figure 4-28 *Datum axis passing through a cylinder*

Two Planes Option

The **Two Planes** option is used to create a datum axis passing through the edge where two planes meet or at the intersection edge of two planar surfaces or datum planes. When you choose this option, you are prompted to select a planar surface or a datum plane. In Figure 4-29, **A_5** is the datum axis that is created using this option.

Two Pnt/Vtx Option

The **Two Pnt/Vtx** option is used to create a datum axis between two datum points or edge vertices. When you choose this option, you are prompted to select datum points or edge vertices. The datum axis is created along the two selected datum points or edge vertices. In Figure 4-30, **A_6** is the datum axis that is created using this option.

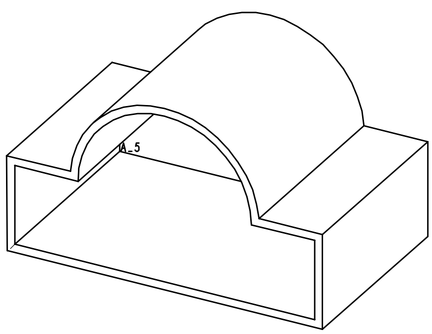


Figure 4-29 Datum axis created on the edge where the two selected planes meet

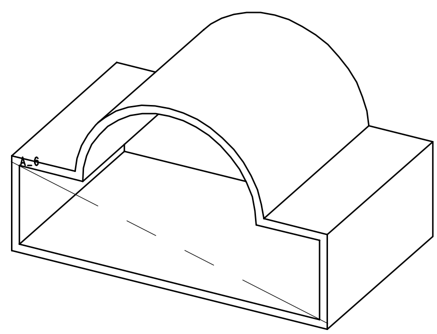


Figure 4-30 Datum axis created between the two selected vertices

Pnt on Surf Option

The **Pnt on Surf** option is used to create a datum axis passing through any selected datum point on a surface. When you choose this option, you are prompted to select a placement point. The datum axis is created normal to the surface on which the datum point is selected and passes through the datum point. In Figure 4-31, **A_7** is the datum axis that is created using this option.

Tan Curve Option

The **Tan Curve** option creates a datum axis tangent to a curve and passing through one of its vertex. When you choose this option, you are prompted to select an edge or a curve. After you select a curve or an edge you are prompted to select one vertex of the edge. The datum axis is created tangent to the curve and passes through its selected vertex. In Figure 4-32, **A_8** is the datum axis that is created using this option.

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 discussed next.

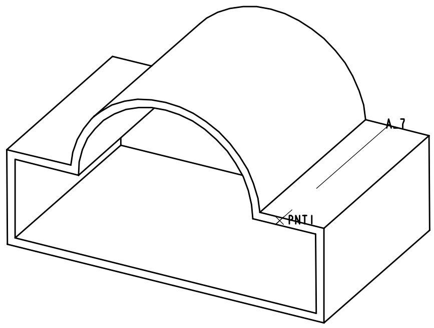


Figure 4-31 *Datum axis passing through the datum point*

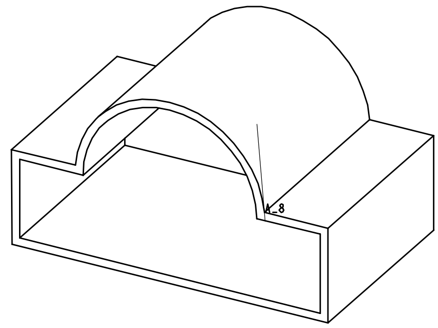


Figure 4-32 *Datum axis created tangent to the selected curve*

1. To create datum planes and axes.
2. To associate note in the drawings and attach datum targets.
3. To create coordinate system.
4. To specify point loads for mesh generation.
5. To create pipe features.

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

When you choose **Insert > Datum > Point** from the menu bar or **Insert a datum point.** button from the **Datum** toolbar, the **DATUM POINT** submenu appears with different options to create datum points as shown in Figure 4-33. The options in the **DATUM POINT** submenu are explained next.

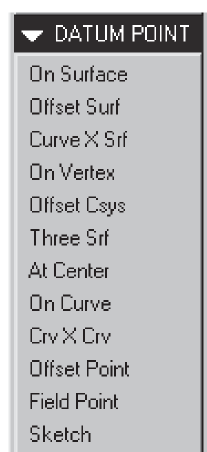


Figure 4-33 DATUM POINT *submenu*

On Surface Option

The **On Surface** option is used to create datum points on a planar surface. When you choose this option from the **DATUM POINT** submenu, you are prompted to select the desired location for the placement of the datum point. When you select a planar surface or a datum plane to place the datum point, a red colored point is displayed at the selected point on the surface. Confirm the selection using the middle mouse button. Next, you are prompted to select two planes or edges to specify the linear dimensions for the placement of the datum point. After you select the two planes or edges for the placement dimension of the point, the **Message Input Window** is displayed with the first selection highlighted and you are prompted to specify the distance from the highlighted references. A default value is displayed in the window. You can accept the default value or change it to the required value and then press ENTER. The second selection will be highlighted and you will be prompted to enter the distance from the highlighted references. Enter the dimension value in the window that appears. Press the middle mouse button, the datum point is created.

Offset Surf Option

The **Offset Surf** option creates datum points at an offset distance from a specified planar surface or a datum plane in a specified direction. When you choose this option to create a datum point, you are prompted to select the desired location for the datum point. Select a planar surface or a datum plane from where the offset distance for the placement of the datum point will be measured. Use the middle mouse button to confirm the selection. 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 will now be prompted to enter the offset distance in the specified direction shown by the arrow. Enter the value in the **Message Input Window** that appears and press ENTER. Press the middle mouse button and the datum point is created. If you enter a negative value in the **Message Input Window** then the datum point will be created in the direction opposite to that shown by the arrow.

Curve X Srf Option

The **Curve X Srf** option is used to create a datum point at the intersection of a curve and a surface. When you choose this option to create a datum point, you will be prompted to select a curve, edge, or axis. After selecting the curve, edge, or axis, you are prompted to select surfaces that intersect the edge. Select a surface or a datum plane. Press the middle mouse button and the datum point is created.

On Vertex Option

The **On Vertex** option is used to create a datum point on the vertex of a part, edge, surface feature edge, or a datum curve. When you choose this option, you are prompted to select vertices where you want to place the datum points. Select the vertices and press the middle mouse button to create the datum points.

Offset Csys Option

The **Offset Csys** option 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.

When you choose this option, you are prompted to select a coordinate system. After selecting a coordinate system, the **SET CSYS TYP** (Set Coordinate System Type) submenu is displayed and you are prompted to select the type of coordinate system; Cartesian, Cylindrical, or Spherical. After selecting the type of coordinate system, the **POINT ARRAY** submenu is displayed and you are prompted to enter the points. Choose the **Enter Points** option from the

POINT ARRAY submenu. The **Message Input Window** is displayed and you are prompted to enter a parameter, this parameter will define the location of the datum point and depends on the type of coordinate system selected from the **SET CSYS TYP** submenu.

Three Srfs Option

The **Three Srfs** option is used to create datum points at the intersection of three surfaces. When you choose this option, you are prompted to select the first surface. After selecting the first surface, you are prompted to select the second surface. Select the second surface. You will then be prompted to select the third surface. Select the third surface on the model. The datum point is placed at the intersection of the three surfaces selected and appears in green color. Confirm the selection by using the middle mouse button. Using this option, you can also select datum planes to create datum points.

At Center Option

The **At Center** option creates a datum point at the center of an arc or a circle. When you choose this option, you are prompted to select an edge or a curve, at the center of which the datum point will be created. Confirm the selection by using the middle mouse button. The datum point will be created.

On Curve Option

The **On Curve** option is used to create a datum point on an edge or a curve. When you choose this option, the **DTM PNT MODE** submenu and the **PNT DIM MODE** submenu are displayed and you are prompted to specify the dimension type for the datum point. Choose the options from the **PNT DIM MODE** submenu to select the type of dimensioning.

Crv X Crv Option

The **Crv X Crv** option is used to create a datum point on a datum curve at the point that is at the minimum distance from another datum curve.

When you choose this option, you are prompted to select a curve where the point should be placed. After selecting the datum curve, you are prompted to select a second curve close to the placement of the point. The datum point will be created on the first curve at a point that is closest to the second curve.

Note
Datum curves will be discussed in Chapter 7.

Offset Point Option

The **Offset Point** option is used to create datum points at an offset distance from a point or a vertex. When you choose the **Offset Point** option from the **DATUM POINT** submenu, the **OFFSET DIR** submenu is displayed as shown in Figure 4-34. The options in this submenu are discussed next.

***Figure 4-34 OFFSET DIR** submenu*

Entity/Edge

When you choose the **Entity/Edge** option, you are prompted to select an axis, a straight edge, or a straight curve. After you select any one of the above mentioned entities, you are prompted to select vertices, points, or coordinate systems to offset from. After selecting a vertex, a point, or a coordinate system, press the middle mouse button. The **Message Input Window** is displayed and you are prompted to specify the offset distance in the direction shown by the red arrow. The datum point will be placed at the specified distance from the selected entity. In case you want to create more then one datum points, you need to select more then one vertices, points, or coordinate systems to place the datum points. The system will prompt you to enter the offset distance from each point selected. After specifying the offset distance all the points, press the middle mouse button to create the datum points.

The procedure to create datum points using the other options in the **OFFSET DIR** submenu is the same as discussed in the **Entity/Edge** option.

Plane Norm

The **Plane Norm** option places one or more datum points normal to the plane selected and at the specified offset distance.

2 Points

The **2 Points** option creates one or more datum points in a direction along a straight line that is defined by the two selected points.

Coord Sys

The **Coord Sys** option creates one or more datum points aligned with the three directions of the selected coordinate system.

Field Point

When you choose the **Field Point** option, the **FIELD PNT** submenu is displayed as shown in Figure 4-35. The options in this submenu are discussed next.

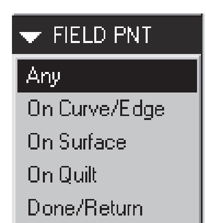


***Figure 4-35 FIELD PNT** submenu*

Any

The **Any** option is used to create a datum point anywhere on the model. You just need to use the left mouse button to place a datum point.

On Curve/Edge

The **Curve/Edge** option is used to create a datum point on any edge or curve of the model.

On Surface

The **On Surface** option creates a datum point on the selected surface.

On Quilt
The **On Quilt** option creates a datum point on a quilt.

Sketch
The **Sketch** option allows you to sketch a datum point. When you choose this option, you are prompted to select a sketching plane. After selecting the sketching plane and the horizontal and vertical references for sketching, the system takes you to the sketcher environment. Using the sketcher options, draw the datum point and regenerate the sketch. The datum point is created where it is placed in the sketch using dimensions.

CREATING CUTS
The **Cut** is a material removal process and this option is available only when at least a base feature exists on the graphics screen. The **Cut** option can be invoked from the menu bar or from the **Menu Manager**. Figure 4-36 shows the method of invoking the **CUT** option from the menu bar. In the cascading menu, the types of cut that can be created in Pro/ENGINEER are given. The procedure to create a cut on an existing feature is similar to that of adding material or protrusion. The method to invoke the **Cut** option from the **Menu Manager** is, **PART > Feature > Create > Solid > Cut**.

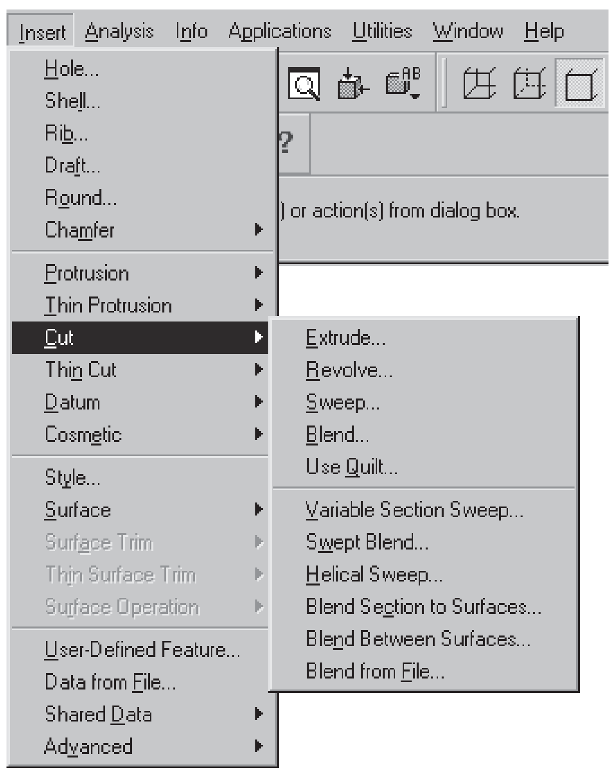


*Figure 4-36 Invoking the **Cut** option from the menu bar*

Extrude Cut

The **Extrude Cut** is used to create an extruded feature by removing material from an existing feature. The material that is removed is defined by the sketch you draw.

After drawing the sketch for the cut feature, you are prompted to specify the direction of material removal with respect to the sketch. For example, the red arrow in Figure 4-37 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 4-38.

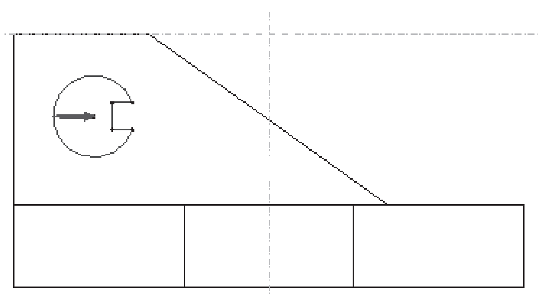


Figure 4-37 *Sketch for the extruded cut and arrow showing the direction of material removal*

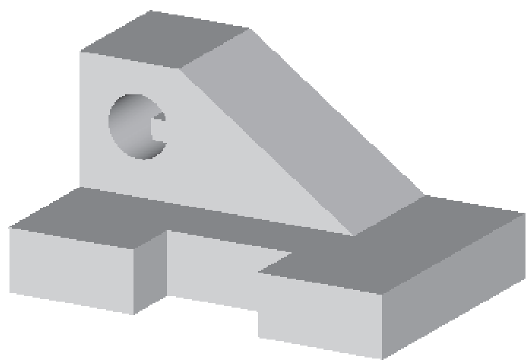


Figure 4-38 *Cut feature created on the selected plane*

Tip: *In the model shown in Figure 4-38, the sketching plane selected for creation of the extruded cut is not a datum plane but the planar surface 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 in cases where a planar surface of the feature can be used as a sketching plane.*

However, if you choose **Flip** from the **DIRECTION** menu, the arrow points in the direction shown in Figure 4-39. All the material on the plane selected for sketching will be removed leaving the extruded cut feature as shown in Figure 4-40.

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 that 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** dialog box. The **HOLE** dialog box is discussed in Chapter 5.*

Revolve Cut

The **Revolve Cut** is used to create a revolved feature by removing material from an existing feature. The material that is removed is defined by the sketch you draw. Remember that the centre line is necessary in the revolve features. Figure 4-41 shows the section drawn to be revolved. The front surface of the second extruded feature is selected as the sketching plane.

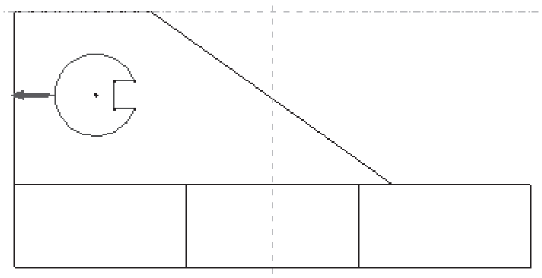


__Figure 4-39__ Arrow showing the direction of material removal

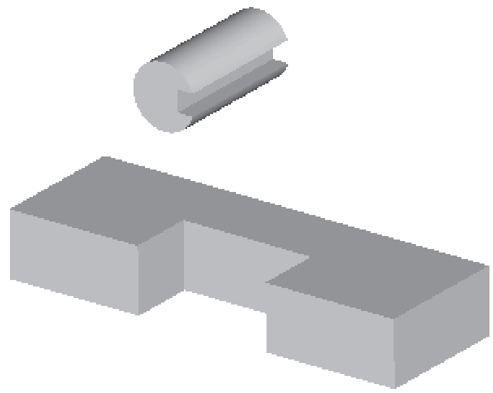


__Figure 4-40__ Cut feature created in the direction shown in the adjacent figure

Figure 4-42 shows the revolve cut created on the selected surface.

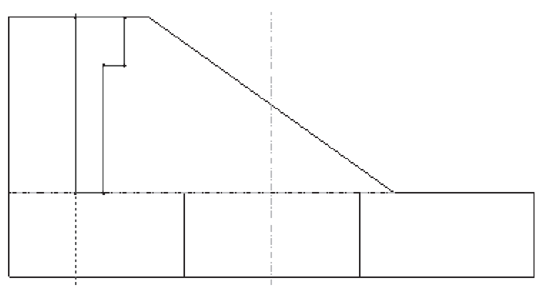


__Figure 4-41__ The section for revolve cut

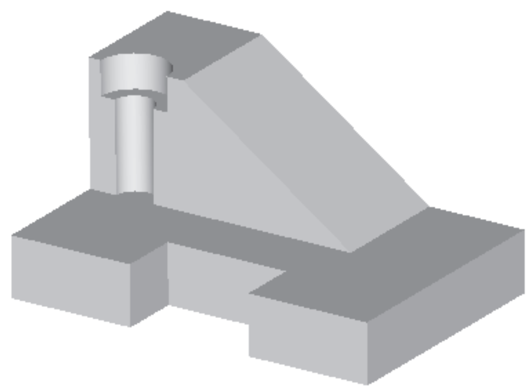


__Figure 4-42__ Revolve cut created

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

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 4-43. The front view and the right-side view with dimensions of the solid model is shown in Figure 4-44.

(Expected time: 25 min)

The following steps outline the procedure for creating this model:

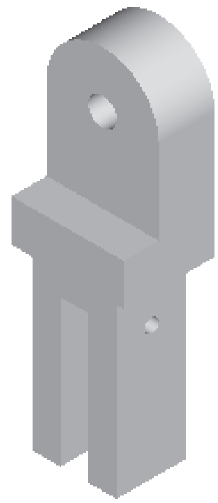


Figure 4-43 *Model for Tutorial 1*

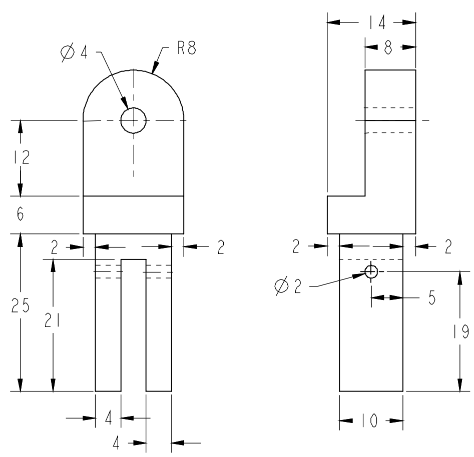


Figure 4-44 *Front and side views of the model*

a. First examine the model and then determine the number of features in it, see Figure 4-43. The model is composed of four features: two at the top, one at the bottom, and one hole on the right surface. Also, from the model it is evident that the two features at the top of the model can be created on the same plane.

b. Select the sketching plane for the base feature, draw the sketch using the sketching tools, apply the dimensions and constraints, and then extrude the sketch to the given distance, see Figure 4-46.

c. Select the sketching plane for the feature that is at the bottom of the base feature. This feature will be sketched on the same plane that was used by the base feature. Draw the sketch using the sketching tools, apply the dimensions and constraints, and then extrude the sketch to the given distance, see Figure 4-50.

d. The third feature that is at the bottom of the second feature will be created on a datum plane that is at an offset distance of 2 from the front planar surface of the second feature. Draw the sketch, apply the dimensions and constraints, and then extrude it to the given distance, see Figure 4-56.

e. Similarly, select a sketching plane for the cut feature. The cut has a circular section. Draw the sketch for this feature and create the cut feature as shown in Figure 4-60.

After understanding the procedure for creating the model, you are now ready to create it. When Pro/ENGINEER session is started, the first task is to set the working directory. Since this is the first tutorial of this chapter, you need to create a folder named **c04**, if it does not exist. Choose the **New Directory** button in the **Select Working Directory** dialog box and create a directory named **c04** at **C:\ProE**.

Creating New Object File

1. Open a new part file and name it as **c04tut1**. The three default datum planes are displayed on the graphics screen. The **Model Tree** also appears on the left of the graphics screen. Exit the **Model Tree** by choosing the **Model Tree on/off** button from the **Model Display** toolbar.

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 **FRONT** datum plane because from the isometric view of this model, it is evident that the direction of extrusion for this feature is perpendicular to the **FRONT** datum plane.

Note

The model can be created by selecting any plane as the sketching plane for the base feature. But when the base feature is created, the orientation of the base feature will not be proper. Hence, the final model will be oriented wrongly. 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 4-43.

1. Invoke the **Extrude** option from the menu bar by selecting **Insert > Protrusion > Extrude**. The **ATTRIBUTES** menu is displayed on the screen.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default. Choose **Done**.

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

   A red arrow is displayed on the **FRONT** datum plane pointing in the direction of feature creation and you are prompted to specify the direction of feature creation.

4. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

5. Select **Top** from this menu and select the **TOP** datum plane from the graphics screen.

   The **TOP** datum plane is selected in order to orient the sketching plane. As you select the **TOP** datum plane, the system takes you to the sketcher environment.

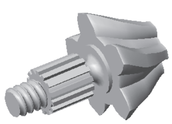
Specifying References

In the sketcher environment, the **References** dialog box is displayed at the top right corner of the screen. The status displayed in the **Reference status** area is **Fully Placed**. Close the **References** dialog box by choosing the **Close** button from the dialog box.

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 section sketch using various sketcher tools and add the required constraints and dimensions shown in Figure 4-45. Since the **Intent Manager** is on by default, the sketch is dimensioned automatically and some weak dimensions are assigned to it.



> **Tip**: *It is recommended to 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 dimensions.*

2. Modify the dimension values to the values shown in Figure 4-45.

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

4. Choose the **Default** option from the **Saved view list** button of the **View** toolbar.

   The default view is displayed. This gives you a better view of the sketch in the 3D space. The red colored arrow is also displayed on the model, indicating the direction of extrusion.

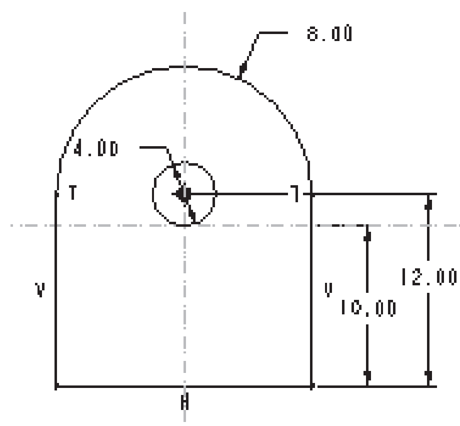


Figure 4-45 *Sketch for the base feature with dimensions and constraints*

5. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**.

   The **Message Input Window** is displayed with a default value in it.

6. Enter a value of **8** in the **Message Input Window** and press ENTER.

7. Choose the **Preview** button from the **PROTRUSION** dialog box and then choose **OK** to complete the feature and to exit the **PROTRUSION** dialog box.

   The base feature is completed and is shown in Figure 4-46. You can use CTRL+middle mouse button to spin the model to view it from different directions.

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 on the graphic screen such that you can view the front view of the base feature, similar to that shown in Figure 4-44.*

Note

*When you choose the **Default** 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 the **Environment** option from the **Utilities** menu in the menu bar. From the dialog box in the **Default Orient** drop-down list, choose the **Isometric** option. Now, 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 drawn on the previous plane that was used to draw the base feature.

1. Invoke the **Extrude** option by selecting **Insert** > **Protrusion** > **Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default, choose **Done**.

   You will be prompted to select the sketching plane.

3. Choose the **Use Prev** option from the **SETUP SK PLN** menu. The red arrow is displayed on the graphics screen as shown in Figure 4-47.

   When you choose the **Use Prev** option, 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 same depth of extrusion, you could have drawn them on the same plane as a single feature.

In Figure 4-47, 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 **Okay** from the **DIRECTION** submenu.

The system takes you to the sketcher environment.

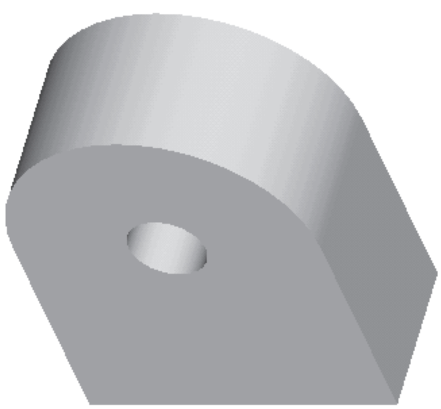


Figure 4-46 *Base feature of the model without datums*

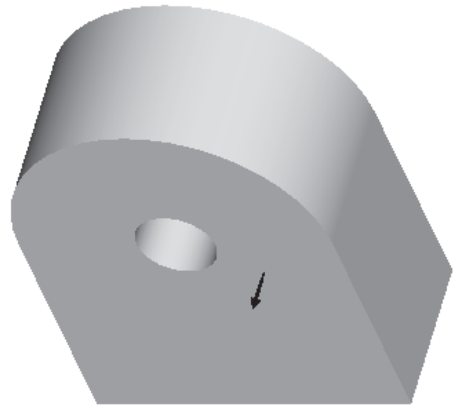
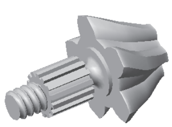


Figure 4-47 *Arrow showing the direction of feature creation*

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 4-48.



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

Since the **Intent Manager** is on by default, therefore, the sketch is automatically dimensioned and some weak dimensions are assigned to it. Add the required constraints and modify the weak dimensions as shown in Figure 4-48.

2. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed. Choose the **Shading** button from the **Model Display** toolbar to view the shaded model.

3. Use the CTRL+middle mouse button to orient the model as shown in Figure 4-49. This orientation of the model gives you a better view of the sketch in the 3D space. The red colored arrow is also displayed on the model, indicating the direction of extrusion.

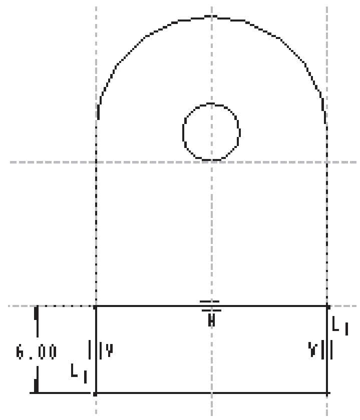


Figure 4-48 Sketch for the second feature

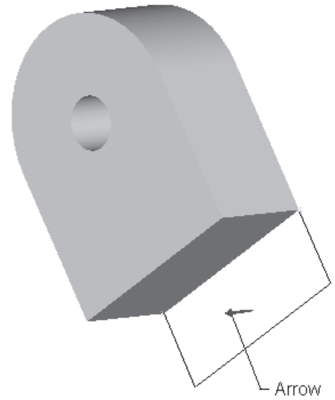


Figure 4-49 Arrow showing the direction of feature depth

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

5. Enter a value of **14** in the **Message Input Window** and press ENTER.

   The second extruded feature is completed and it can now be previewed.

6. Choose the **Preview** button from the **PROTRUSION** dialog box.

7. Choose the **Saved view list** button from the **View** toolbar. From the drop-down list choose the **Default** option.

8. Now, choose the **OK** button from the **PROTRUSION** dialog box to confirm the feature creation and exit the dialog box. The model orients on the screen as shown in Figure 4-50. You can also use the CTRL+middle mouse button to spin the model.

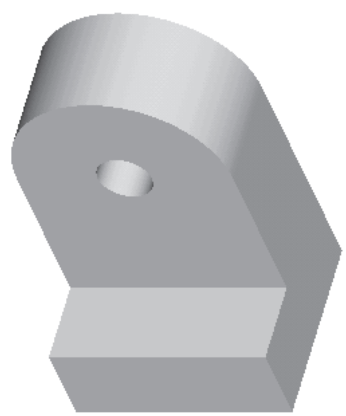
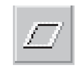


Figure 4-50 Second extruded feature with the base feature

Creating a Datum Plane for the Third Feature

A datum plane is required to create the next feature. The datum plane will be at an offset distance of 2 from the front planar surface of the second feature.



1. Choose the **Insert a datum plane.** button from the **Datum** toolbar. The **DATUM PLANE** submenu is displayed.

2. Choose the **Offset** option from the **DATUM PLANE** submenu. As you choose the **Offset** option, the **Plane** and the **Coord Sys** options are highlighted and the system prompts you to select either a plane or a coordinate system.

3. Using the left mouse button, select the front planar surface of the second feature highlighted in Figure 4-51. The boundary of the selected front planar surface is highlighted red in color.

   The **OFFSET** submenu is displayed and you are prompted to select a location or enter a value for the datum plane to pass through.

4. From the **OFFSET** submenu, choose the **Enter Value** option.

   A green arrow is displayed on the selected planar surface and the **Message Input Window** is displayed with a default value. You are prompted to enter an offset value in the direction shown by the arrow. 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.

5. Enter a value of **-2** in the **Message Input Window** and press ENTER. Choose **Done** from the **DATUM PLANE** submenu.

   The negative value is entered because the datum plane has to be created in the direction opposite to that shown by the green arrow. The datum plane named **DTM1** is created as shown in Figure 4-52.

Note

Throughout the book at some instances the datum planes are shown by a mesh plane as evident from Figure 4-52. 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.

Creating the Third Feature on DTM1

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

1. Invoke the **Extrude** option from the menu bar. The **ATTRIBUTES** menu is displayed.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default, choose **Done**. The **SETUP PLANE** submenu is displayed.

3. Select **DTM1** as the sketching plane for the third feature. A red arrow is displayed on the selected datum plane.

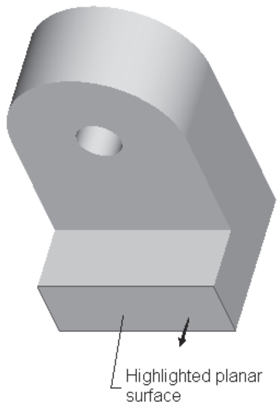


Figure 4-51 *Planar surface*

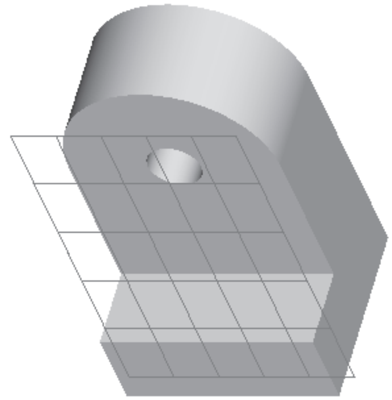


Figure 4-52 *The highlighted datum plane*

4. Choose **Flip** from the **DIRECTION** submenu. The arrow displayed on the model now points in the direction as shown in Figure 4-53.

   The red arrow shows the direction of feature creation.

5. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed and you are prompted to select a horizontal or vertical reference.

6. Select the **Top** option from the **SKET VIEW** submenu and choose the **TOP** default datum plane. The system takes you to the sketcher environment. Choose the **No Hidden** button from the **Model Display** toolbar.

7. Sketch the section for the third feature of the model and add constraints and dimensions to the sketch as shown in Figure 4-54.

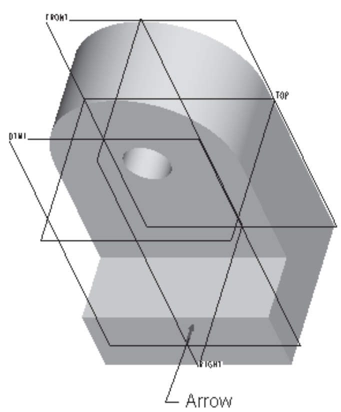


Figure 4-53 *Arrow on* ***DTM1*** *showing the direction of feature creation*

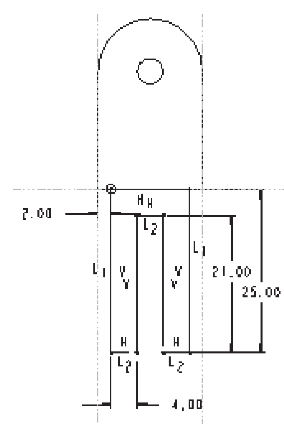
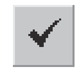


Figure 4-54 *Sketch with dimensions and constraints of the third feature*

8. Choose the **Continue with current section.** button.



The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion. The red color arrow is also displayed on the model. Use the CTRL+middle mouse button to orient the model as shown in Figure 4-55. This orientation gives a better view of the model.

9. The **Blind** option in the **SPEC TO** menu is selected by default, choose **Done**.

The **Message Input Window** is displayed with a default value in it.

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

The third extruded feature is completed and it can now be previewed.

11. Choose the **Preview** button from the **PROTRUSION** dialog box. To view the shaded image of the model, choose the **Shading** button from the **Model Display** toolbar.

12. Choose the **OK** button from the **PROTRUSION** dialog box to confirm the feature creation and to exit the dialog box.

The default view that is displayed when you choose the **Default** option from the **Saved view list** button is shown in Figure 4-56.

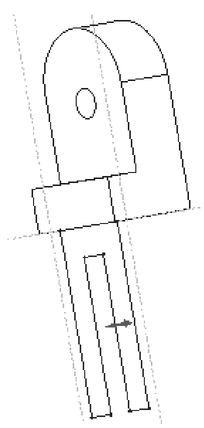


Figure 4-55 *Arrow showing the direction of material addition*

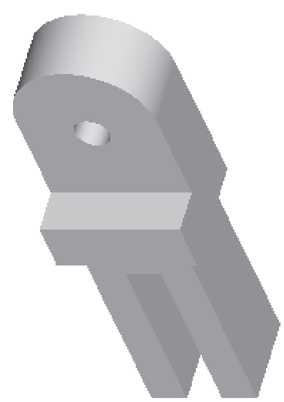


Figure 4-56 *Model after creating the third feature*

Selecting the Sketching Plane for the Cut Feature

A through cut will be created on the outer right surface of the third feature. The circular section for the cut will be sketched and using the **Cut** option from the **SOLID** menu, the circular cut will be created. The sketching plane will be the surface of the third feature that is shown in Figure 4-57.

Note

*The circular cut feature can also be created using the **HOLE** dialog box that will be discussed in Chapter 5.*

1. Choose **Insert** > **Cut** > **Extrude** from the menu bar or choose **PART** > **Feature** > **Create** > **Solid** > **Cut** > **Extrude** > **Solid** > **Done** from the **Menu Manager**. The **ATTRIBUTES** menu is displayed.

2. Choose **One Side** > **Done** from the **ATTRIBUTES** menu.

   You will be prompted to select or create a sketching plane.

3. Select the planar surface shown in Figure 4-57 and specify the direction of feature creation.

   The **SKET VIEW** submenu is displayed and you are prompted to select a horizontal or vertical reference.

4. Using the left mouse button select the **Top** option from the **SKET VIEW** submenu and select the **TOP** default datum plane.

   The system takes you to the sketcher environment.

Sketching the Cut Feature

1. Turn the model display to **No Hidden**. Draw the sketch of the cut feature and add dimensions to it as shown in Figure 4-58. The hole dimension shown in Figure 4-58 appears as a radial dimension but it is a diameter dimension.

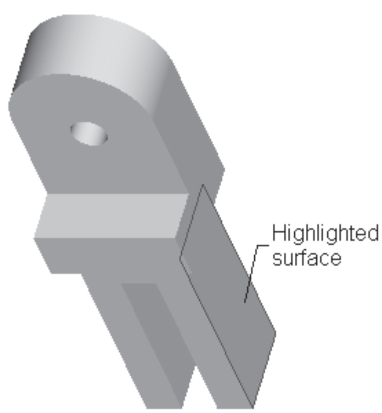


Figure 4-57 Sketching plane for hole feature

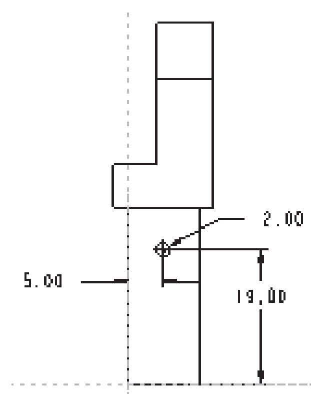


Figure 4-58 Sketch and dimensions for hole

2. Choose the **Continue with current section.** button.

   You will be prompted to select the direction of material removal. The direction is shown by the red arrow. The arrow should point as shown in Figure 4-59.

3. Choose **Okay** from the **DIRECTION** menu.

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

4. Select the **Thru All** option from the **SPEC TO** menu and then choose **Done**.

   The cut feature is completed and it will now be previewed.

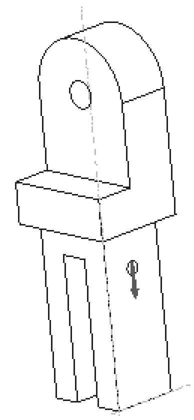


Figure 4-59 *Arrow showing the direction of material removal*

5. Choose the **Preview** button from the **CUT** dialog box and then choose **OK**. Turn the model display to shaded by choosing the **Shading** button from the **Model Display** toolbar.

   The default view, when you choose the **Default** option from the **Saved view list** button from the **View** toolbar, is shown in Figure 4-60.

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 4-61.

Note

*The feature id numbers that are suffixed to the feature name in the **Model Tree** may be different when you create the features.*

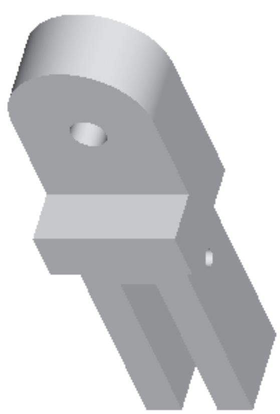


Figure 4-60 *Completed model for Tutorial 1*

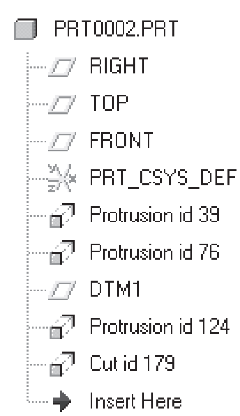


Figure 4-61 ***Model Tree*** *for Tutorial 1*

Tutorial 2

In this tutorial you will create the model shown in Figure 4-62. This figure also shows the front view, the top view, and the right-side view of the solid model. **(Expected time: 45 min)**

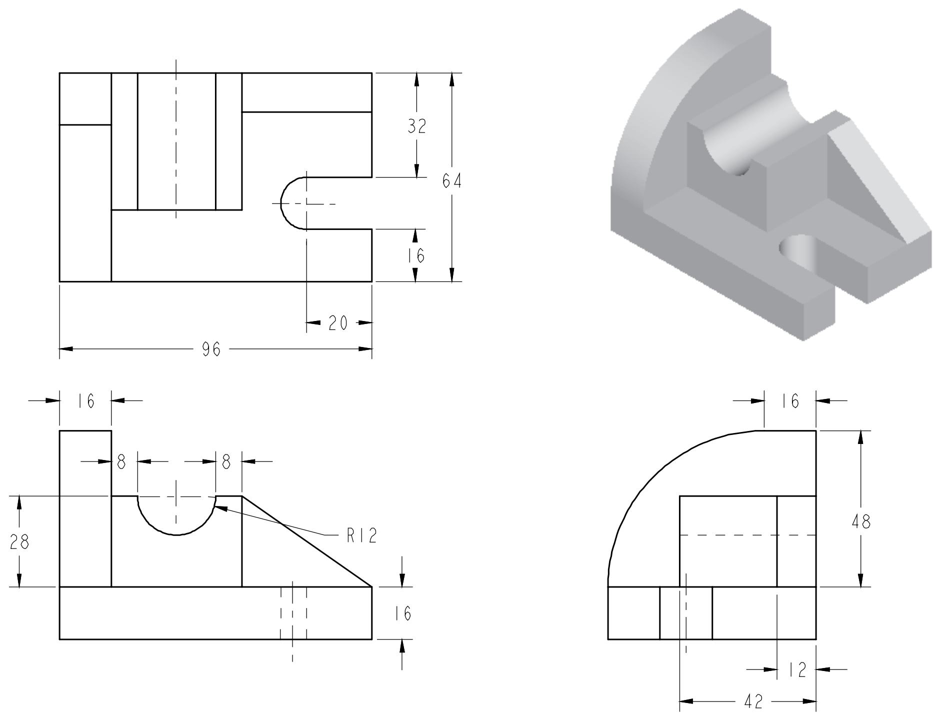


Figure 4-62 *Top, front, right-side, and isometric views of the model*

The following steps outline the procedure for creating this model:

a. First, examine the model and then determine the number of features in it, see Figure 4-62. The model is composed of four features: one at the bottom (base feature), one on the left, and two at the back. Also, from the model it is evident that the two features on the back face of the model can be created on the same plane.

b. Select the sketch plane for the base feature, draw the sketch using the sketching tools, apply the dimensions and constraints, and then extrude the sketch to the given depth, see Figure 4-64.

c. Select the sketch plane for the feature on the left face of the base feature, draw the sketch using the sketching tools, apply dimensions and constraints, and then extrude the sketch to the given depth, see Figure 4-69.

d. From the model it is evident that the third and fourth features are created on the same plane, but their depths of extrusion are different and hence they will be created as two separate extruded features. See Figure 4-74 and Figure 4-78.

   After understanding the procedure for creating the model, you are now ready to create it. The working directory is already selected in Tutorial 1 and therefore you do not need to select it again. However, if you want to select the working directory, choose **File > Set Working Directory** and then select c04 in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it as **c04tut2**. The three default datum planes are displayed on the graphics screen if the **Datum planes on/off** button is turned on. The **Model Tree** is not displayed on the graphics screen as its display was turned off in the previous tutorial.

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 because the direction of extrusion of the base feature is perpendicular to the **TOP** datum plane.

1. Invoke the **Extrude** option from the menu bar by choosing **Insert > Protrusion > Extrude**. The **ATTRIBUTES** menu is displayed.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default. Choose **Done**. The **One Side** option will extrude the sketch to one side of the sketching plane. You are now prompted to select or create a sketching plane.

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

   A red arrow is displayed on the **TOP** datum plane pointing in the direction of feature creation and you are prompted to specify the direction of feature creation.

4. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

5. Choose the **Right** option from this submenu and select the **RIGHT** datum plane from the graphics screen.

   The **RIGHT** datum plane is selected in order to orient the model. As you choose the **RIGHT** datum plane, the system takes you to the sketcher environment.

Specifying References

In the sketcher environment, the **References** dialog box is displayed at the top right corner of the screen. The status displayed in the **Reference status** area is **Fully Placed**. Exit the **References** dialog box by choosing the **Close** button from the dialog box.

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 4-63. When this sketch is extruded, it will create the base feature with the slot as shown in Figure 4-64.

1. Draw the section sketch using various sketcher tools and add the required constraints and dimensions to it as shown in Figure 4-63.

   Since the **Intent Manager** is on by default, the sketch is automatically dimensioned and some weak dimensions are assigned to it. Modify these dimensions as shown in Figure 4-63.

2. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion.

3. Choose the **Default** option from the **Saved view list** button of the **View** toolbar.

   The default view is displayed. This gives you a better view of the sketch in the 3D space. A red arrow is also displayed on the model, indicating the direction of extrusion.

4. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**.

   The **Message Input Window** is displayed with a default value in it.

5. Enter a value of **16** in the **Message Input Window** and press ENTER.

6. Choose the **Preview** button from the **PROTRUSION** dialog box and then choose **OK**.

   The base feature is completed and is shown in Figure 4-64. You can use the CTRL+middle mouse button to spin the object to view it from different directions.

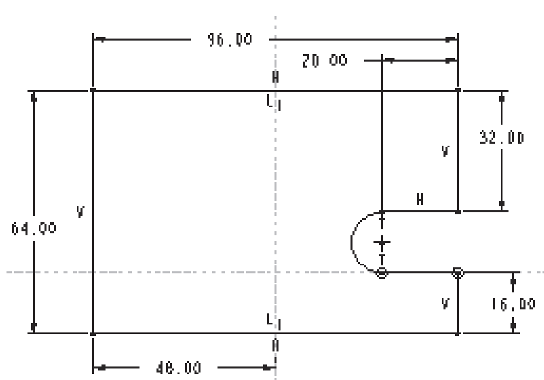


__Figure 4-63__ Sketch with dimensions and constraints for the base feature

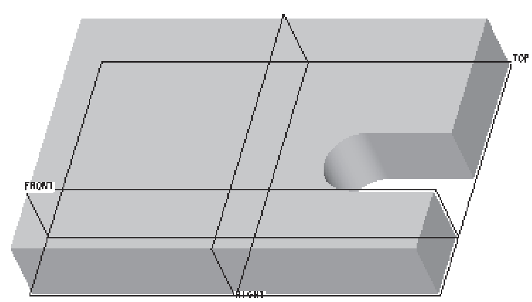


__Figure 4-64__ Base feature of the model

Selecting the Sketching Plane for the Second Feature

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

1. Invoke the **Extrude** option by selecting **Insert** > **Protrusion** > **Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default. Choose **Done**. You will now be prompted to select or create a sketching plane.

3. Using the CTRL+middle mouse button, spin the model as shown in Figure 4-65.

4. Now, select the left planar surface of the base feature as the sketching plane.
   A red arrow that points in the direction of feature creation is displayed on the planar surface.

5. Use the **Flip** option to flip the red arrow to point in the direction of extrusion as shown in Figure 4-65.

6. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

7. Select **Top** from this menu and choose the top planar surface of the model shown in Figure 4-66.

   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. After you select the surface, the system takes you to the sketcher environment.

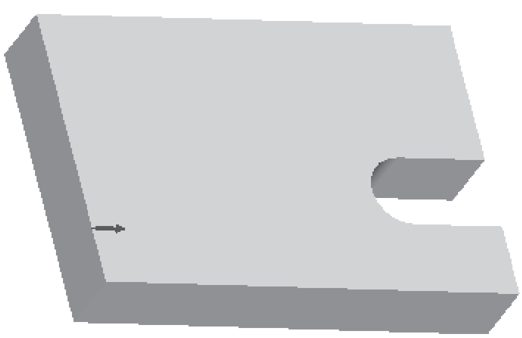


Figure 4-65 Arrow showing the direction of feature creation from the sketching plane

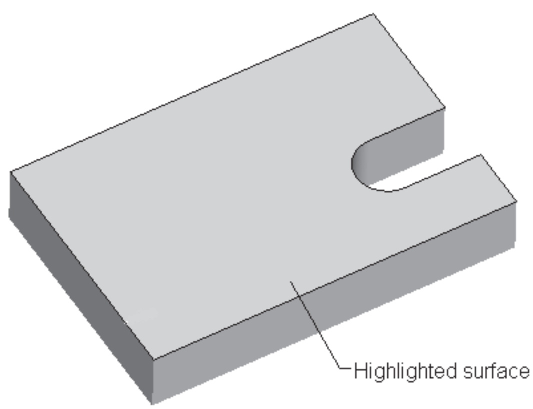


Figure 4-66 Surface selected to be at the top

Creating and Dimensioning the Sketch for the Second Feature

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

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

2. Draw the sketch using various sketcher tools as shown in Figure 4-67.

   Since the **Intent Manager** is on by default, 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 4-67.

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

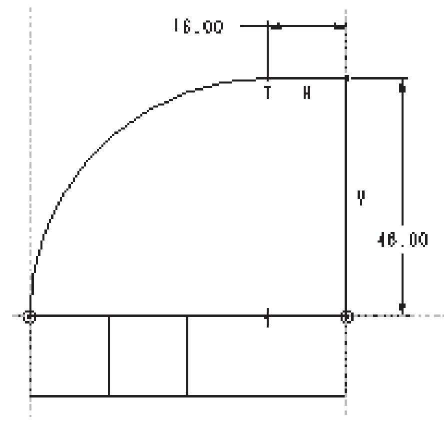


Figure 4-67 *Sketch with dimensions and constraints for the second feature*

   The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion. Use the CTRL+middle mouse button to orient the model as shown in Figure 4-68. A red arrow is also displayed on the model, indicating the direction of extrusion.

5. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**. The **Message Input Window** is displayed with a default value in it.

6. Enter a value of **16** in the **Message Input Window** and press ENTER. The second feature is completed and it can now be previewed.

7. Choose the **Preview** button from the **PROTRUSION** dialog box and then choose **OK**.

   The second feature is completed and is shown in Figure 4-69. You can use the CTRL+middle mouse button to spin the model to view it from different directions.

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. Invoke the **Extrude** option from the menu bar. The **ATTRIBUTES** menu is displayed.

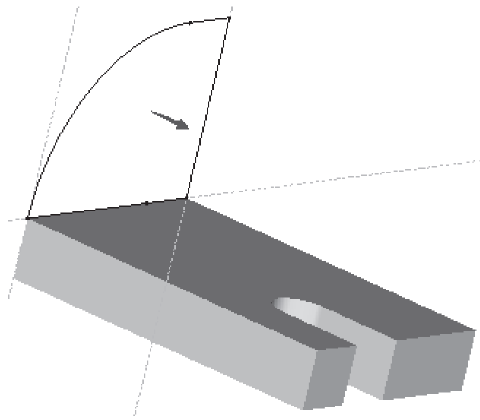


Figure 4-68 Arrow showing the direction of material addition

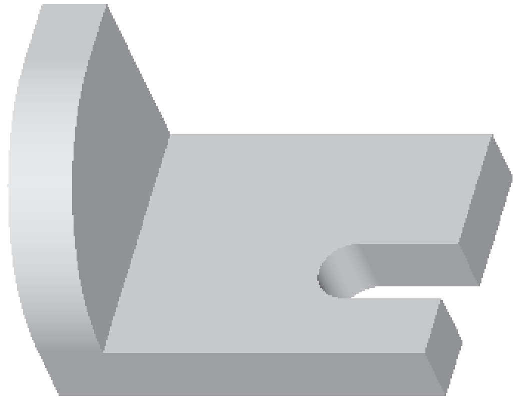


Figure 4-69 Model with the second extruded feature

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default. Choose **Done**.

   The **One Side** option will extrude the sketch to one side of the sketching plane. You are now prompted to select or create a sketching plane.

3. Using the CTRL+middle mouse button, spin the model and select the planar surface of the base feature shown in Figure 4-70 as the sketching plane.

   A red arrow is displayed on the planar surface and it points in the direction of feature creation.

4. Use the **Flip** option to flip the red arrow in the direction of extrusion as shown in Figure 4-70. The sketch will be extruded in the direction of the arrow.

5. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

6. Select **Top** from this menu and choose the top planar surface shown in Figure 4-71.

   The top planar surface of the base feature is selected to be at the top while drawing the sketch. As you choose the planar surface from the base feature, the system takes you to 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.

1. Draw the section sketch using various sketcher tools as shown in Figure 4-72.

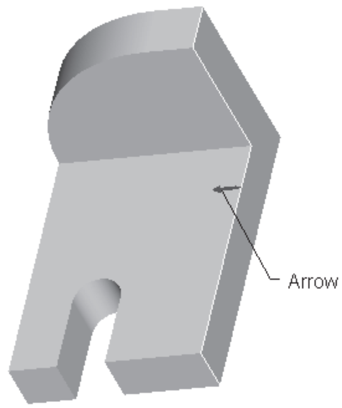


Figure 4-70 *Arrow showing the direction of feature creation from the sketching plane*

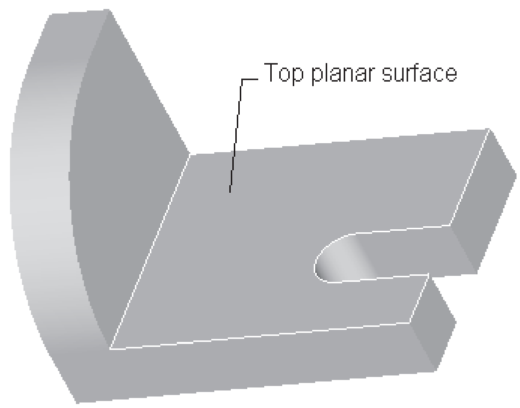


Figure 4-71 *Surface selected to be at the top*

2. Since the **Intent Manager** is on by default, 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 4-72.

3. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion.

4. Use the CTRL+middle mouse button to orient the model as shown in Figure 4-73. This orientation of the model gives a better view of the sketch in 3D space. A red arrow is also displayed on the model, indicating the direction of extrusion.

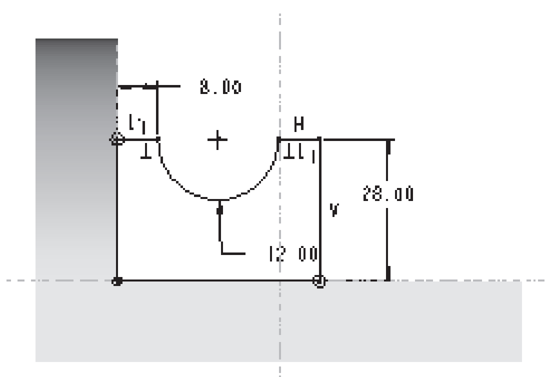


Figure 4-72 *Sketch with dimensions and constraints*

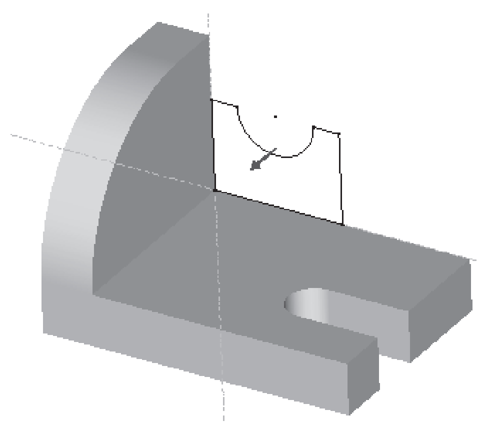


Figure 4-73 *Arrow showing the direction of material addition*

5. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**.

The **Message Input Window** is displayed with a default value in it.

6. Enter a value of **42** in the **Message Input Window** and press ENTER. This feature is completed and it can now be previewed.

7. Choose the **Preview** button from the **PROTRUSION** dialog box.

8. Now, choose the **OK** button from the **PROTRUSION** dialog box to confirm the feature creation and to exit the dialog box. The default view that is displayed when you choose the **Default** option from the **Saved view list** button is shown in Figure 4-74.

Selecting the Sketching Plane for the Last Feature

As mentioned earlier, this feature and the third feature are 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. Invoke the **Extrude** option from the menu bar. The **ATTRIBUTES** menu is displayed.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default. Choose **Done**. You are now prompted to select or create a sketching plane.

3. Choose the **Use Prev** option from the **SETUP SK PLN** menu.

   By specifying the **Use Prev** option, you are using the previous sketching plane that was defined for the previous feature. A red arrow is displayed on the planar surface and points in the direction of feature creation.

4. Flip the red arrow by choosing the **Flip** option to make the arrow point in the direction shown in Figure 4-75.

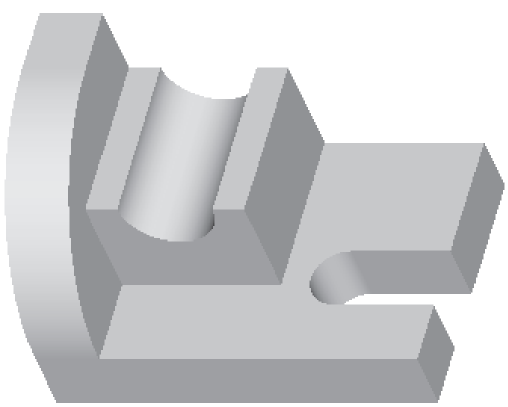


Figure 4-74 *Model with the second extruded feature*

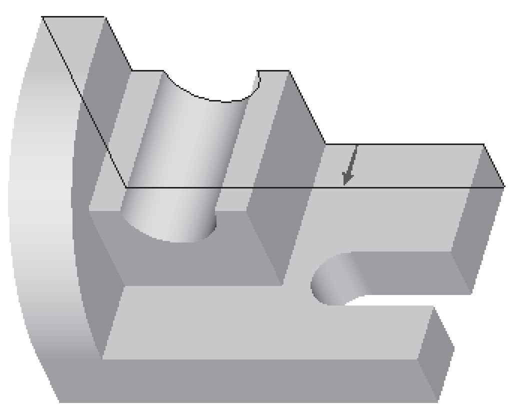


Figure 4-75 *Arrow showing the direction of feature creation*

5. Choose **Okay** from the **DIRECTION** submenu.

   As you choose the **Okay** option, the system takes you to the sketcher environment.

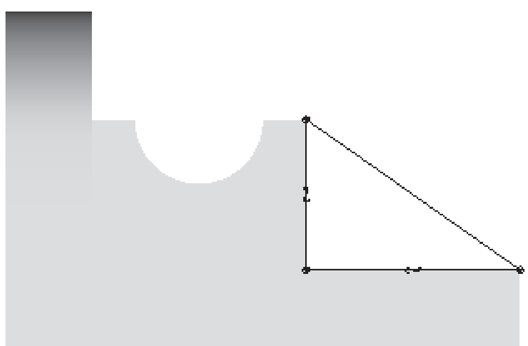
Creating and Dimensioning the Sketch for the Last Feature

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 4-76.

   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 the other features as shown in Figure 4-76. 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 **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion. Using the CTRL+middle mouse button orient the model as shown in Figure 4-77.



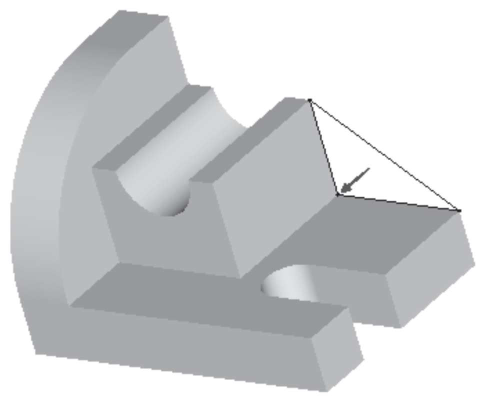


Figure 4-76 *Sketch and constraints for the last feature*

Figure 4-77 *Arrow showing the direction of material addition*

3. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**.

   The **Message Input Window** is displayed with the default value.

4. Enter a value of **12** in the **Message Input Window** and press ENTER. The feature is completed and it can now be previewed.

5. Choose the **Preview** button from the **PROTRUSION** dialog box.

6. Choose **OK** from the **PROTRUSION** dialog box to exit it.

   The default view of the model, when you choose the **Default** option from the **Saved view list** button from the **View** toolbar, is shown in Figure 4-78.

Note
*The last feature of this tutorial can also be created using the **Rib** option. The **Rib** option will be discussed in Chapter 5.*

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 4-79.

Figure 4-78 *Completed model for Tutorial 2*

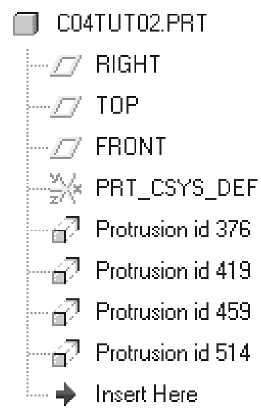


***Figure 4-79 Model Tree** for Tutorial 2*

Tutorial 3

In this tutorial you will create the model shown in Figure 4-80. This figure also shows the front view, the top view, and the right-side view of the solid model. **(Expected time: 45 min)**

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it, see Figure 4-80. The model is composed of three features: one on the left (base feature), one in the middle, and one hollow cylindrical feature with a hole.

b. Select the sketch plane for the base feature, draw the sketch using the sketching tools, apply the dimensions and constraints, and then extrude the sketch to the given depth, see Figure 4-82.

c. Select the sketch plane for the middle feature, draw the sketch using the sketching tools,

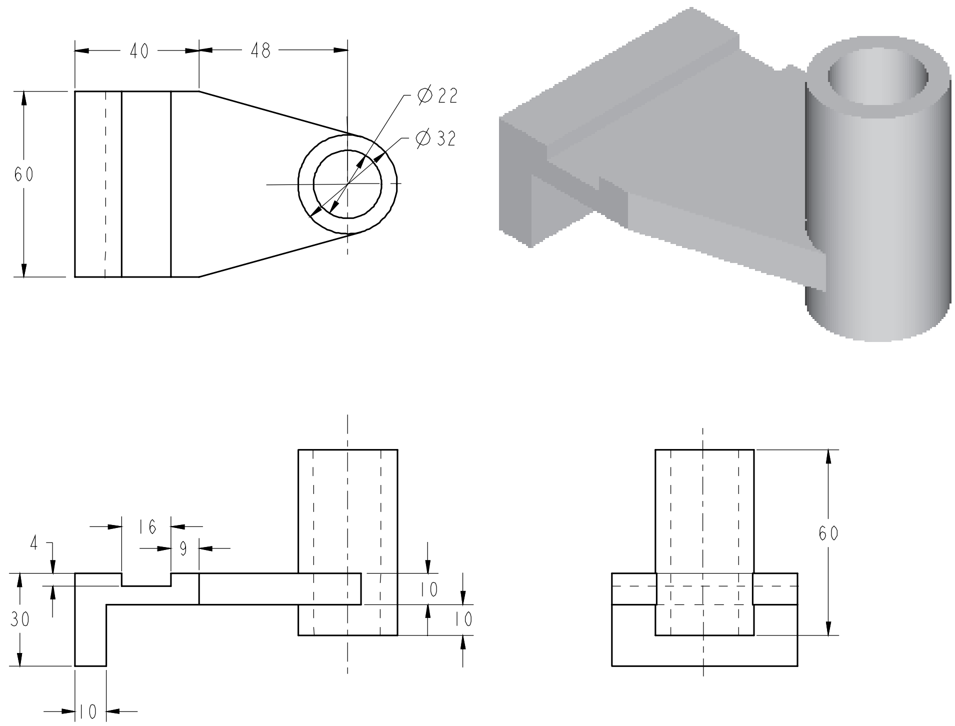


Figure 4-80 *Top, front, right-side, and isometric views of the model*

apply the dimensions and constraints, and then extrude the sketch to the given depth, see Figure 4-87.

d. To create the hollow cylindrical feature, create a datum plane on the fly. Then draw the sketch, apply the dimensions and constraints, and then extrude the sketch to the given depth, see Figure 4-92.

The working directory is already selected in Tutorial 2 and therefore you do not need to select it again. However, if you want to select the c04 directory, choose **File > Set Working Directory** and then select c04 in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it as **c04tut3**. The three default datum planes are displayed on the graphics screen if the **Datum planes on/off** button is turned on. The **Model Tree** is not displayed on the graphics screen as its display was turned off in the first tutorial.

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. Invoke the **Extrude** option from the menu bar by choosing **Insert** > **Protrusion** > **Extrude**. The **ATTRIBUTES** menu is displayed.

2. The **One Side** option in the **ATTRIBUTES** menu is selected by default. Choose **Done**.

   The **One Side** option will extrude the sketch to one side of the sketching plane. You will now be prompted to select or create a sketching plane.

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

   A red arrow is displayed on the **FRONT** datum plane and it points in the direction of feature creation.

4. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

5. Select **Right** from this submenu and choose the **RIGHT** datum plane from the graphics screen.

   The **RIGHT** datum plane is selected in order to orient the sketching plane. As you choose the **RIGHT** datum plane, the system takes you to the sketcher environment.

Specifying References

In the sketcher environment, the **References** dialog box is displayed. The status displayed in the **Reference status** area is **Fully Placed**. Close this dialog box.

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 4-81. When this sketch is extruded, it will create the base feature.

1. Draw the section sketch using various sketcher tools as shown in Figure 4-81.

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 4-81.

3. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion.

4. Choose the **Default** option from the **Saved view list** button of the **View** toolbar.

   The default view is displayed which gives you a better view of the sketch in the 3D space.

A red arrow is also displayed on the model, indicating the direction of extrusion.

5. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**.

   The **Message Input Window** is displayed with a default value in it.

6. Enter a value of **60** in the **Message Input Window** and press ENTER.

7. Choose the **Preview** button from the **PROTRUSION** dialog box and then choose **OK**.

   The base feature is completed as shown in Figure 4-82. You can use the CTRL+middle mouse button to spin the model to view it from different directions.

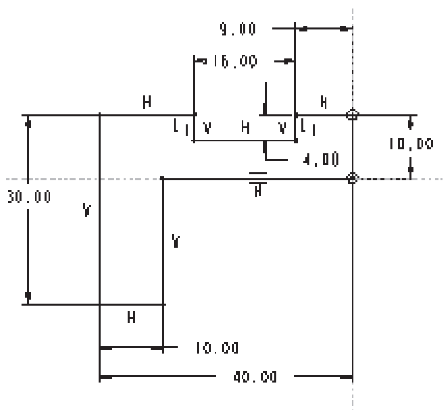


Figure 4-81 *Sketch with dimensions and constraints for the base feature*

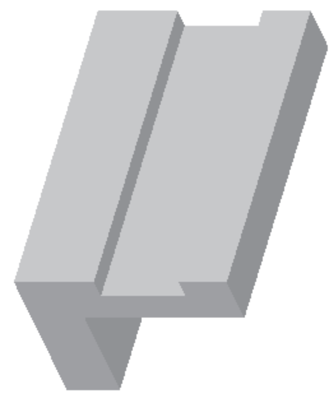


Figure 4-82 *Base feature of the model*

Selecting the Sketching Plane for the Second Feature

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

1. Invoke the **Extrude** option from the menu bar. The **ATTRIBUTES** menu is displayed.

2. Select the **One Side** option from the **ATTRIBUTES** menu and choose **Done**.

   You are now prompted to select or create a sketching plane.

3. Select the top planar surface of the base feature shown in Figure 4-83 as the sketching plane. A red arrow that points in the direction of feature creation is displayed on the planar surface.

4. Use the **Flip** option to flip the red arrow to point in the direction shown in Figure 4-83.

5. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

6. Select **Right** from this submenu and choose the planar surface highlighted on the model shown in Figure 4-84.

   By selecting the right planar surface of the base feature, the model will be oriented in such a way that the highlighted planar surface will be at the right while drawing the sketch. When you select the surface, the system takes you to the sketcher environment.

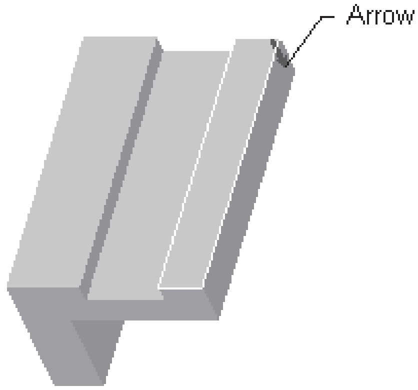


Figure 4-83 *Arrow pointing from the sketching plane in the direction of feature creation*

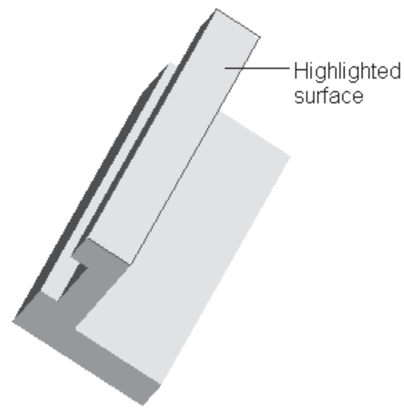


Figure 4-84 *Planar surface of the base feature*

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 4-85. 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 section sketch using various sketcher tools. In the sketch, draw a center line passing through the centre of the arc as shown in Figure 4-85. This center line helps in dimensioning the sketch. Add the required constraints and dimensions to the sketch as shown in Figure 4-85.

   Before exiting the sketcher environment, turn the model display to **Shading**.

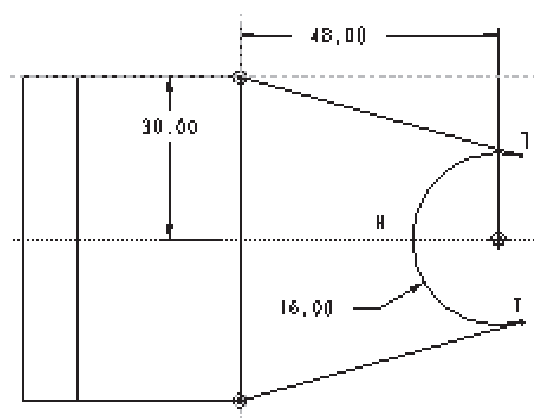


Figure 4-85 *Sketch with dimensions and constraints*

3. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion. Use the CTRL+middle mouse button to orient the model as shown in Figure 4-86.

4. The **Blind** option in the **SPEC TO** menu is selected by default. Choose **Done**.

   The **Message Input Window** is displayed with a default value in it.

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

6. Choose the **Preview** button from the **PROTRUSION** dialog box.

7. Now, choose the **OK** button from the **PROTRUSION** dialog box to confirm the feature creation and to exit the dialog box. The default trimetric view of the extruded feature is shown in Figure 4-87.

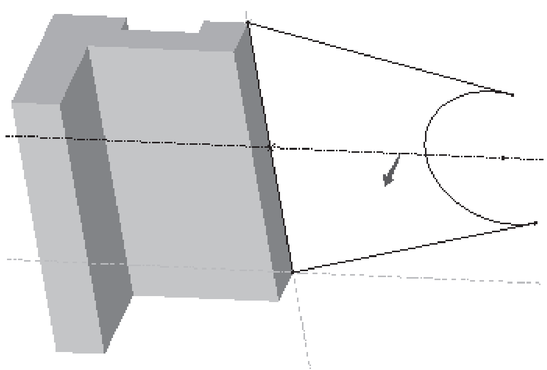


Figure 4-86 *Arrow showing the direction of material addition*

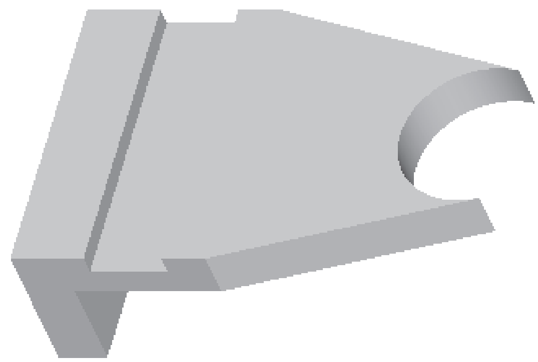


Figure 4-87 *Model with the second extruded feature*

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 surface of the second feature shown in Figure 4-88. The datum plane will be created on the fly.

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 at 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 then you do not need to create a datum plane.

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

2. Select the **One Side** option from the **ATTRIBUTES** menu and choose **Done**.

   You will now be prompted to select or create a sketching plane. You will create a sketching plane.

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

4. Choose the **Offset** option from the **DATUM PLANE** submenu.

   When you choose the **Offset** option, the **Plane** and the **Coord Sys** options in the submenu are highlighted and the system prompts you to select either a coordinate system or a plane.

5. Spin the model using the CTRL+middle mouse button and then using the left mouse button, select the planar surface shown in Figure 4-88.

   As you select the planar surface from the second feature, the **OFFSET** submenu is displayed and you are prompted to select a location or enter a value for the datum plane to pass through.

6. From the **OFFSET** submenu, choose the **Enter Value** option.

   A green colored arrow is displayed on the selected planar surface as shown in Figure 4-88 and the **Message Input Window** is displayed with a default value. You are prompted to enter a value in the direction shown by the arrow. The arrow shows the direction of datum plane creation.

7. Enter a value of **10** in the **Message Input Window** and press ENTER. Choose **Done** from the **DATUM PLANE** submenu.

   Datum plane **DTM1** is created as shown in Figure 4-89 and will be selected as the sketching plane for creating the sketch. A red arrow is attached to **DTM1** that shows the direction where the feature will be created with respect to the datum plane.

8. Select **Flip** and then choose **Okay** from the **DIRECTION** submenu. The arrow should point in the direction shown in Figure 4-89.

   The **SKET VIEW** submenu is displayed. Using this menu you will specify how the **FRONT** datum plane should be oriented while drawing the sketch.

9. Select the **Bottom** option from the **SKET VIEW** submenu and then choose the **FRONT** datum plane from the graphics screen. The system takes you to 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.

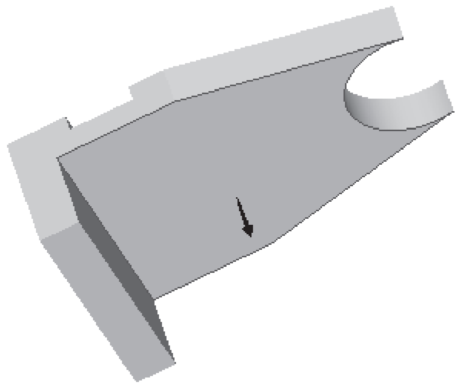


Figure 4-88 Arrow showing the direction of feature creation

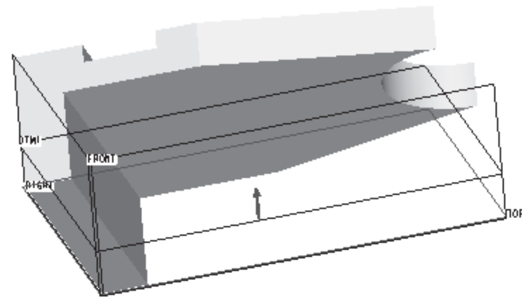


Figure 4-89 Arrow showing the direction of feature creation

2. Draw the section sketch using various sketcher options and add the required constraints and dimensions to it as shown in Figure 4-90.

3. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed and you are prompted to specify the depth of extrusion. Now, turn the model display to **Shading**.

   Use the CTRL+middle mouse button to orient the model as shown in Figure 4-91. This view gives you a better view of the sketch in the 3D space.

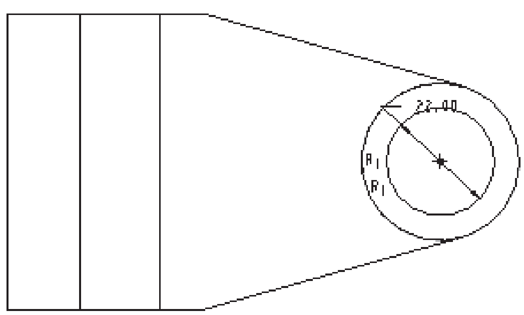


Figure 4-90 Sketch with dimensions and constraints

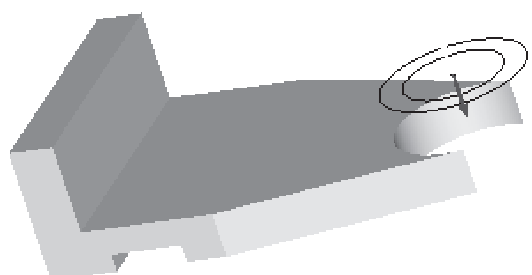


Figure 4-91 Arrow showing the direction of material addition

4. The **Blind** option is selected in the **SPEC TO** menu, choose **Done**.

   The **Message Input Window** is displayed with a default value in it. A red arrow is also displayed on the model.

5. Enter a value of **60** in the **Message Input Window** and press ENTER.

6. Choose the **Preview** button from the **PROTRUSION** dialog box.

7. Now, choose the **OK** button from the **PROTRUSION** dialog box to confirm the feature creation and to exit the dialog box. The default trimetric view of the complete model is shown in Figure 4-92.

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 4-93. The feature id numbers that are displayed in the **Model Tree** may be different when you create the features.

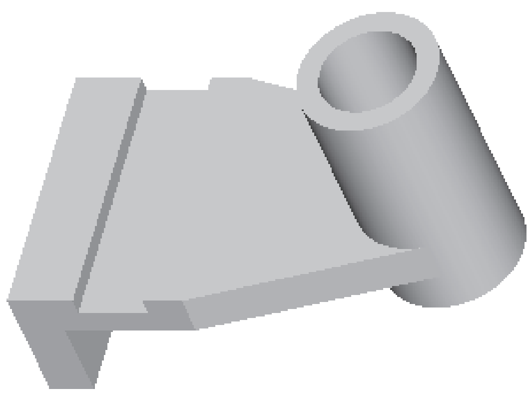


Figure 4-92 *Complete model for Tutorial 3*

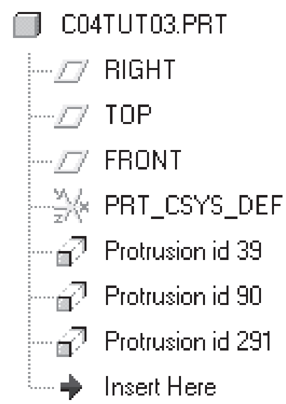


Figure 4-93 ***Model Tree*** *for Tutorial 3*

Self-Evaluation Test

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

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

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

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

4. In Pro/ENGINEER, all the features in a model can be created on a single sketching plane. (T/F)

5. A sketching plane can be selected on an existing planar surface of a feature. (T/F)

6. When you create a datum plane using the __________ option from the **SETUP PLANE** menu, the datum plane created is neither visible on the graphics screen nor is visible in the **Model Tree** after the feature is completed.

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

8. The __________ 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 on the graphics screen.

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 datums 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 menus is displayed when you choose the **Continue with the current section.** button from the **Sketcher Tools** toolbar while extruding a section sketch?

   (a) **DIRECTION** (b) **SPEC TO**
   (c) **SETUP PLN** (d) None

4. Which one of the following combinations of keys and mouse buttons can be used to spin a model on the graphics screen?

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

5. Which one of the following menus is displayed when you invoke the **Extrude** option from the menu bar?

   (a) **SKET PLN** (b) **DIRECTION**
   (c) **ATTRIBUTES** (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, and as well as for orientation of references. (T/F)

10. Unlike the datum planes constraint options, all the datum axes constraint options are standalone. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 4-94. The dimensions, front view, and right side view of the model are shown in Figure 4-95. **(Expected time: 45 min)**

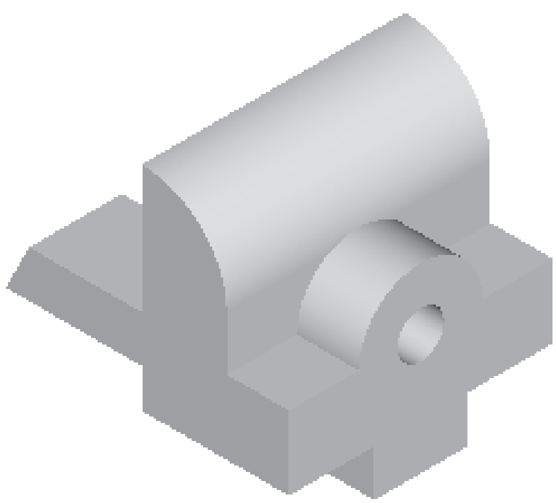


Figure 4-94 *Isometric view of the model*

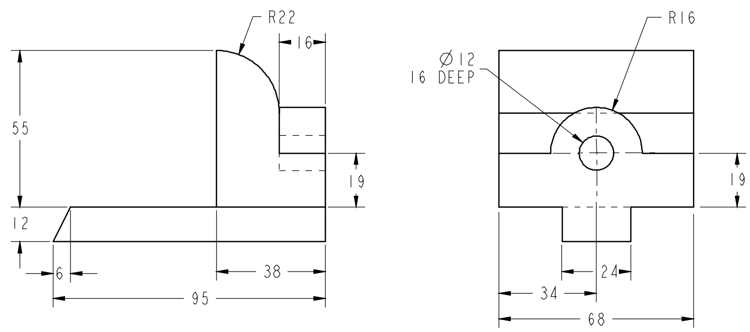


Figure 4-95 *Front view and right-side view of the model*

Exercise 2

Create the model shown in Figure 4-96. The dimensions, the front, top, right-side, and isometric views of the model are also shown in the figure. **(Expected time: 45 min)**

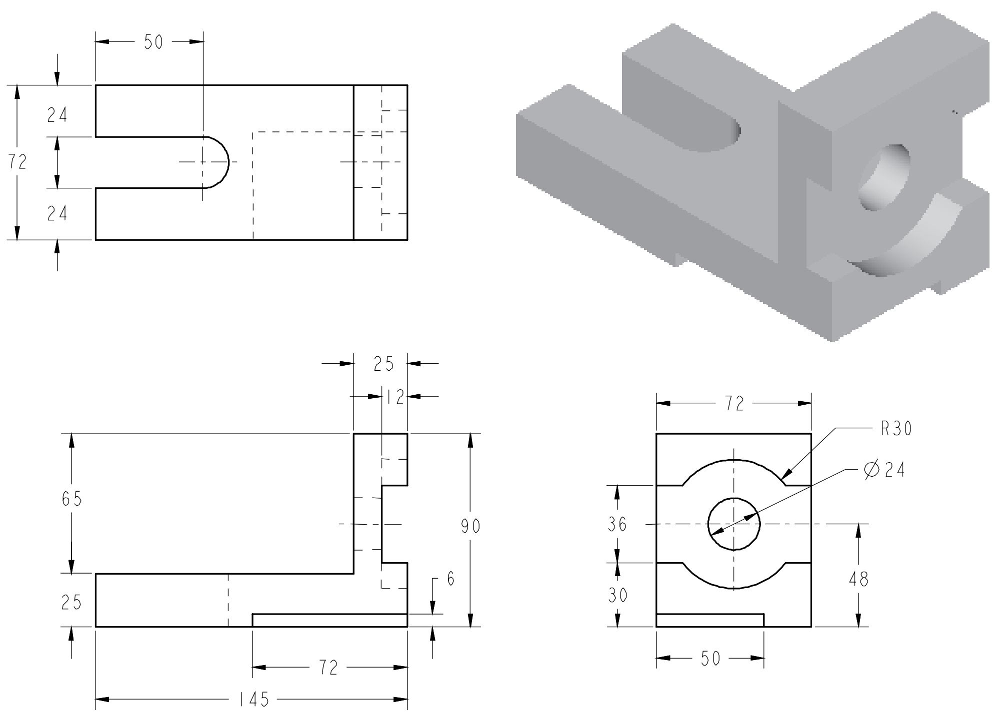


Figure 4-96 *Top, front, right-side, and isometric views of the model*

Exercise 3

Create the model shown in Figure 4-97. The dimensions, the top view, front section view, and right section view of the model are shown in Figure 4-98. **(Expected time: 1 hr)**

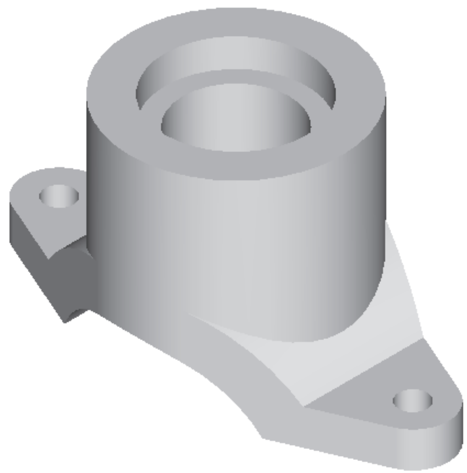


Figure 4-97 *Isometric view of the model*

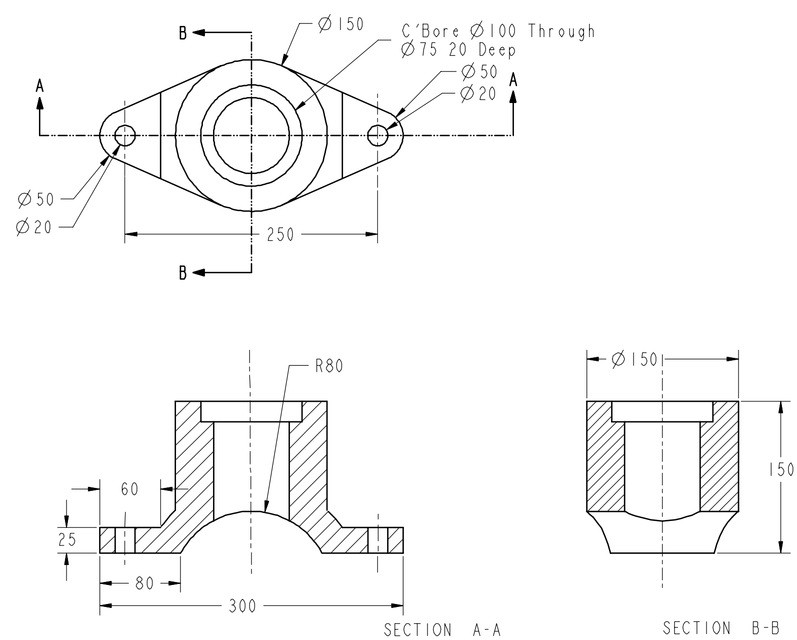


Figure 4-98 *Top view, front section view, and right section view of the model*

Exercise 4

Create the model shown in Figure 4-99. The dimensions, the top view, front section view, and the two auxiliary views of the model are shown in Figure 4-100. **(Estimated time: 1 hr)**

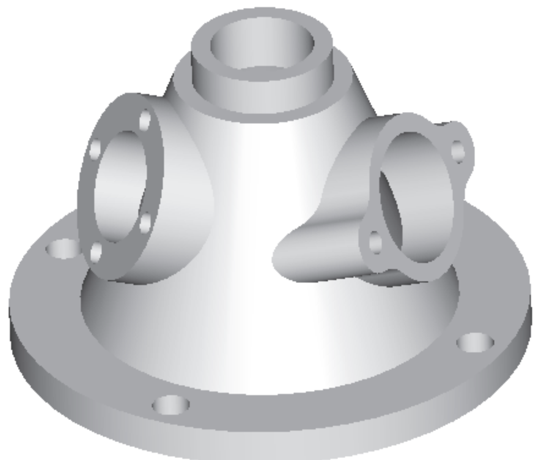


Figure 4-99 *The 3D view of the model*

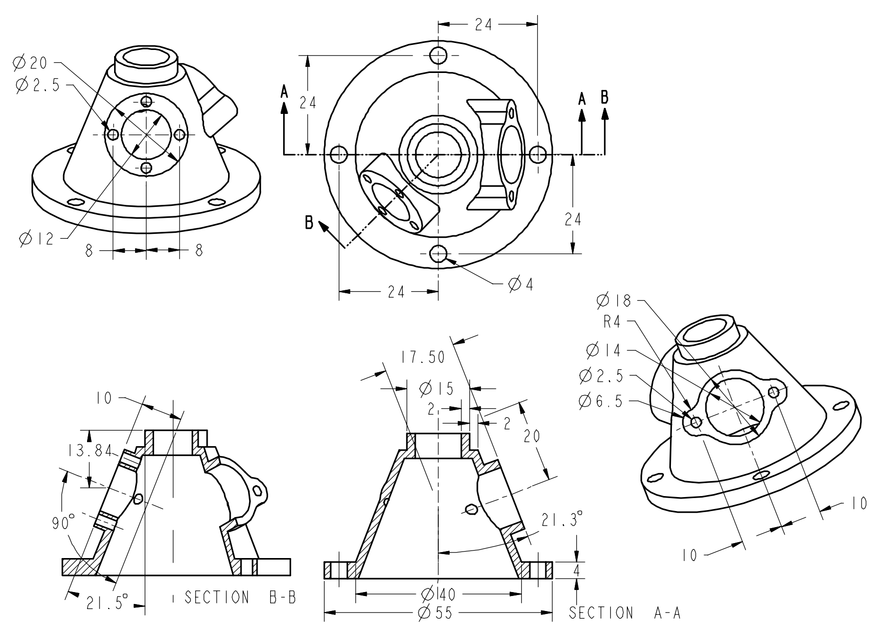


Figure 4-100 *Top view, front section view, and the two auxiliary views of the model*

Exercise 5

Create the model shown in Figure 4-101. The dimensions, the left-side view, auxiliary view, front view and isometric view of the model are also shown in the figure.

(Expected time: 45 min)

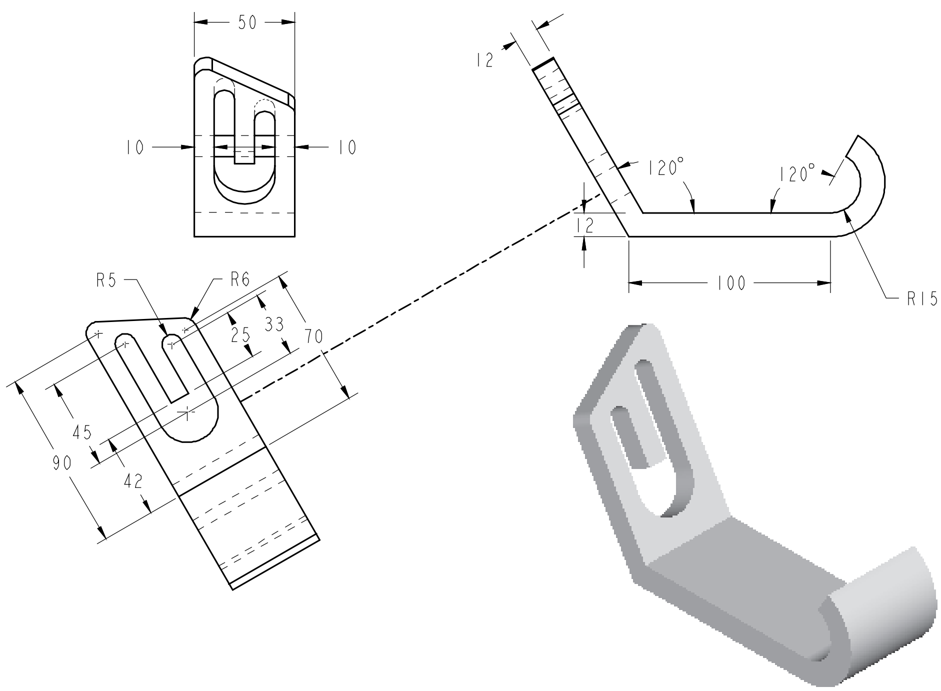


Figure 4-101 *Left-side view, auxiliary view, front view, and isometric view of the model*

Answers to the Self-Evaluation Test

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