

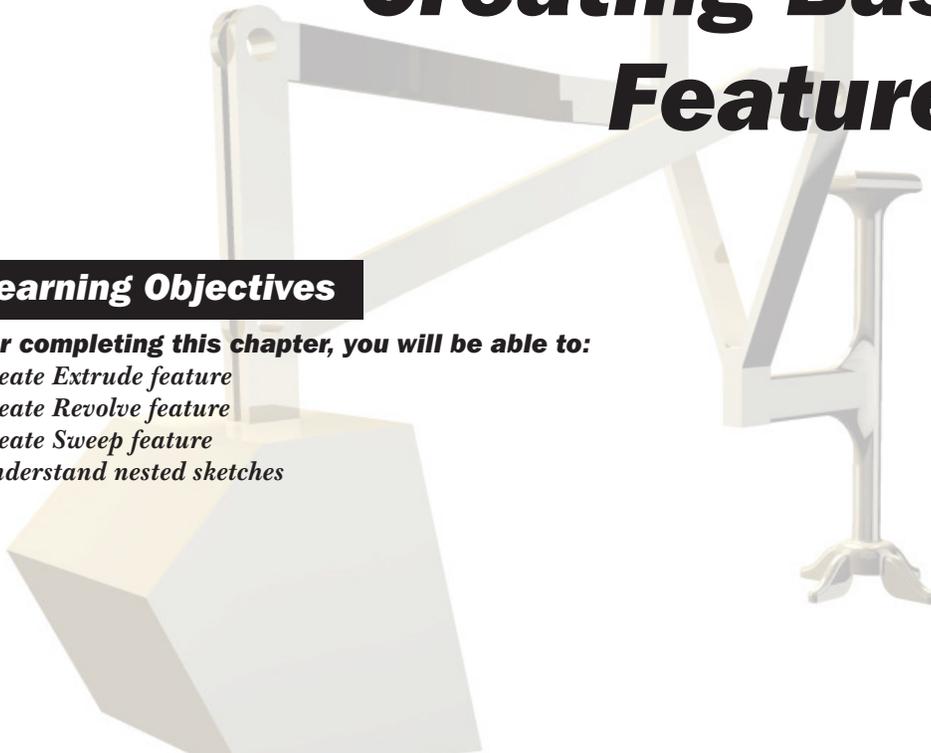
Chapter 3

Creating Base Features

Learning Objectives

After completing this chapter, you will be able to:

- *Create Extrude feature*
- *Create Revolve feature*
- *Create Sweep feature*
- *Understand nested sketches*



INTRODUCTION

When you create a model in the **Part** mode, the base feature is the first solid feature to be created. To create a base feature, you need to enter the **Part** mode. While creating the base feature of a model, the designer must be careful in specifying the attributes, so as to create correct features on it. This helps a designer to save time and effort. The base feature is created using the datum planes. In the **Part** mode, there are three datum planes available in the **Model Tree** by default. The datum planes that are available in the Creo Direct are discussed next.

THE DEFAULT DATUM PLANES

Generally, the three default datum planes act as the first feature in the **Part** mode. These datum planes are used to create the base feature. Also, they are used to draw a 2D sketch and then convert it into a 3D model by using the feature creation tools.



Note

*Although it is said that the three datum planes act as the first feature in the **Part** mode, in the **Model Tree**, the **RIGHT**, **TOP**, and **FRONT** datum planes appear as three separate features. If you delete any one of them, only that datum plane is deleted. By default, the **TOP** plane is selected as sketching plane.*

These three default datum planes are mutually perpendicular to each other. They are not referenced to each other and are individual features. When a solid model is created, these datum planes adjust their size as per the size of the model. You can create as many datum planes as required. The creation of additional datum planes is discussed in Chapter 4.

SELECTION PROCESSES

Before proceeding to the base feature creation, you need to learn about the selection processes which can be used for base feature creation. You can select an entity by using the options available in the **Select** drop-down list of the **Selection** group in the **Home** tab, as shown in Figure 3-1. There are two options available in the **Select** drop-down list. These options are discussed next.



*Figure 3-1 The **Select** drop-down list*

Select (with default)

This option is selected by default in the **Select** drop-down list. As a result, on selecting an entity, a live toolbar is displayed with the corresponding option. For example, if you select a closed sketch, the options displayed in the live toolbar will be extrude, revolve, and sweep.

Select (no default)

If this option is selected, the live toolbar will not be displayed on selecting an entity. In this case, you need to choose a tool from the **Ribbon** to perform the desired operation.

CREATING BASE FEATURES

In Creo Direct, you can create a base feature by using the **Extrude**, **Revolve**, and **Sweep** tools. These tools are available in the **Shape** group of the **Home** tab. You can also invoke these tools by using the **Select (with default)** tool available in the **Selection** group of the **Home** tab. The procedure to create base features is discussed next.

Creating Extrude Features

To extrude a sketch, choose the **Select (with default)** tool available in the **Selection** group of the **Home** tab; you will be prompted to select an entity to operate on. Select a sketch; the live toolbar will be displayed with the **Extrude** tool selected in it, refer to Figure 3-2. Also, a linear dragger with extrude depth value will be displayed, refer to Figure 3-3. Now, you can specify the extrude depth by using the dragger or by entering extrude depth value in the edit box displayed on double-clicking on the value displayed in the graphic area. Specify an extrude depth and then press the middle mouse button; the extrude feature will be created. Figure 3-4 shows the extrude feature created. The options available in the live toolbar are discussed next.

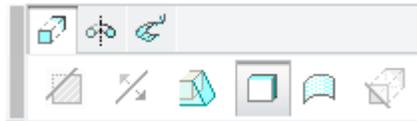


Figure 3-2 The live toolbar

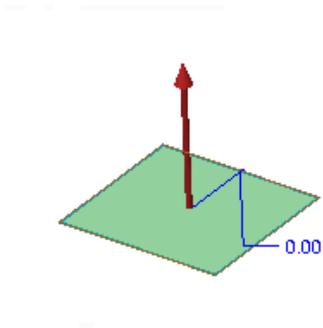


Figure 3-3 Linear dragger with extrude depth value

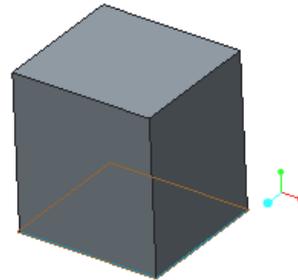


Figure 3-4 The extrude feature created

Solid Geometry



The **Solid Geometry** option is selected by default and is used to create a solid by adding material to the section. Note that if you select a sketch for extrusion using this option, then the sketch should be a single closed loop.

Surface Geometry



The **Surface Geometry** option is used to create a surface by adding material to the section. Using this option, you can create surface models. In this case, the sketch that is drawn for the surface model need not be a closed loop.

Remove Material



The **Remove Material** option is available in the **Extrude** live toolbar only after the base feature is created. This is because this option is used to remove the material from an existing feature. Therefore, when you create a base feature, this button is not available. The use of this option is explained in later chapters.

Capped Ends



The **Capped Ends** option is used to create the surface feature that is closed at both the ends. When you choose this option, the surface model will be created with its end faces capped.

Material Side



The **Material Side** option is available only when you choose the **Remove Material** option. This option is used while creating the second feature or removing the material. It toggles the direction of material removal.

Extrude Taper



The **Extrude Taper** option is used to add a taper to the entity being extruded. If you select this option, an angular dragger will be displayed, as shown in Figure 3-5. Now, you can specify the taper value by using this dragger or by entering a new value in the edit box that is displayed on double-clicking on the taper value displayed in the graphic area. Note that you can also provide taper in opposite direction by providing a negative value for taper or by dragging the dragger in opposite direction. Figure 3-6 shows the extruded feature with tapered surfaces created.

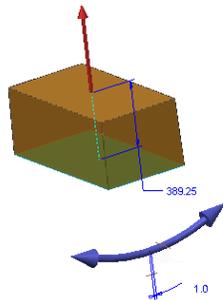


Figure 3-5 The angular dragger with soft dimensions

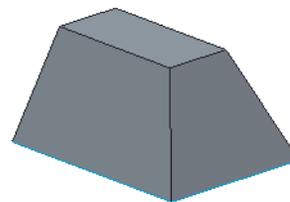


Figure 3-6 Extruded feature with tapered surfaces



Complete Sketch

This button is available in the **Sketch** group of the **Home** tab. It is used to exit the sketch environment.

Figures 3-7, 3-8, and 3-9 show the sections and their respective extruded features.

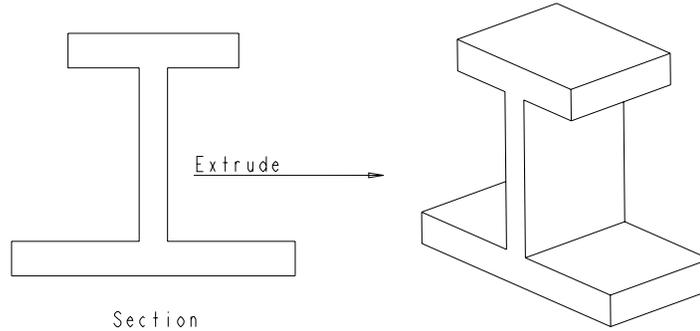
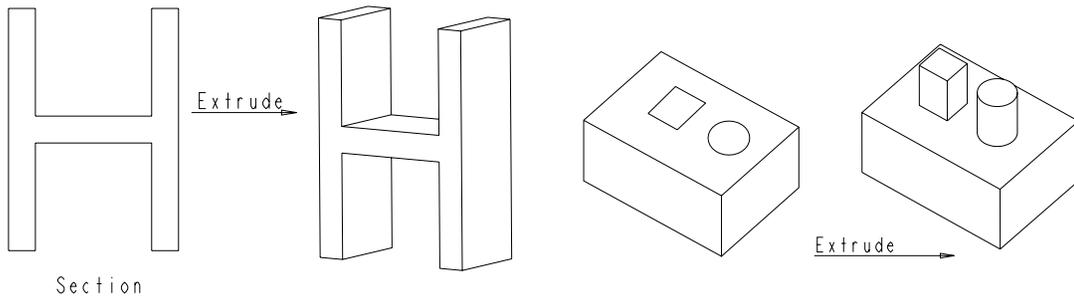


Figure 3-7 The I-section extruded to a certain depth



*Figure 3-8 Model created using the **Extrude** tool*

*Figure 3-9 Model created using the **Extrude** tool*



Note

You can also invoke the **Extrude** tool from the **Shape** group of the **Home** tab. This method is generally used when the **Select (no default)** tool is selected in the **Select** drop-down of the **Select** group. The procedure to create the extrude feature is same as explained above.

Creating Revolve Features

Some of the points to be kept in mind while creating a revolve feature are given next.

1. If the revolve feature is a solid feature, then the section drawn should be a closed one so that the revolved feature can be created as solid.
2. You need to draw a centerline to complete the sketch of the revolved feature, or select an axis of the coordinate system, an existing straight curve, or an edge of an existing feature as the axis of revolution.

3. The section should be drawn on one side of the centerline.

To revolve a sketch, choose the **Select (with default)** tool available in the **Selection** group of the **Home** tab; you will be prompted to select the entity to operate on. Select the sketch to be revolved; an angular dragger with default values will be displayed. By default, the **Extrude** tool is chosen in the live toolbar. Choose the **Revolve** tool; the options related to revolve will be displayed in the live toolbar, as shown in Figure 3-10. Now, specify the centerline by selecting an edge, line, or an axis; the angular dragger will be displayed, as shown in Figure 3-11. Now, you can specify the angular value by using the dragger or by entering a value in the edit box displayed on double-clicking on the value displayed along the dragger. Figure 3-12 shows the revolved feature after specifying the values. The options available in the live toolbar are similar to the **Extrude**. Figures 3-13 and 3-14 show two examples of revolving a sketch.

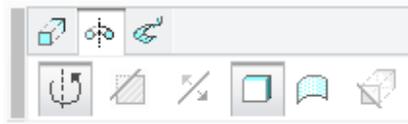


Figure 3-10 The live toolbar

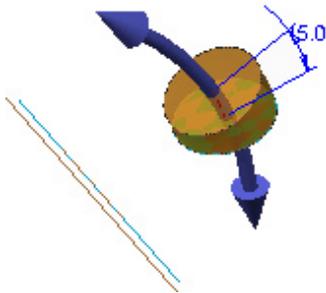


Figure 3-11 Angular dragger displayed after specifying the centerline



Figure 3-12 The revolve feature created

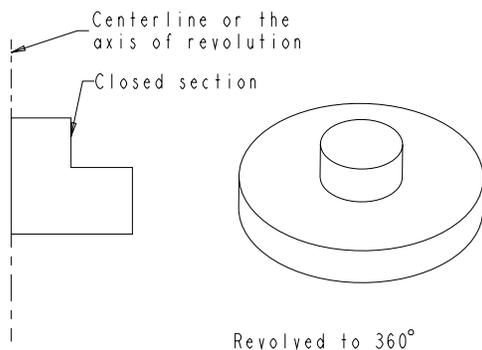


Figure 3-13 Example 1

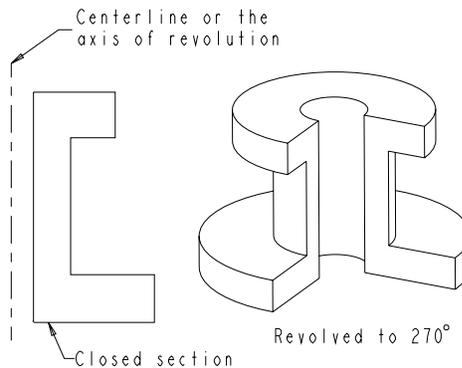


Figure 3-14 Example 2

**Note**

You can also invoke the **Revolve** tool from the **Shape** group of the **Home** tab. This method is generally used when the **Select (no default)** tool is selected in the **Select** drop-down of the **Select** group. The procedure to create the revolve feature is same as explained above.

Creating Sweep Features

To sweep a sketch, choose the **Select (with default)** tool available in the **Selection** group of the **Home** tab; you will be prompted to select the entity to operate on. Select a sketch to sweep; a dragger with default values will be displayed. By default, the **Extrude** tool is chosen. Choose the **Sweep** tool from the live toolbar; the options related to sweep will be displayed in the live toolbar, as shown in Figure 3-15 and you will be prompted to select an entity to define the trajectory. Select the spline curve as a trajectory, refer to Figure 3-16; the preview of the sweep feature will be displayed. Note that the trajectory should intersect the plane of the section. Now, you can specify the sweep value by using the dragger or by entering a value in the edit box displayed on double-clicking on the original value. Next, click anywhere in the empty area of the screen; the sweep feature will be created. Figures 3-16 and 3-17 show the sketch and sweep features created.

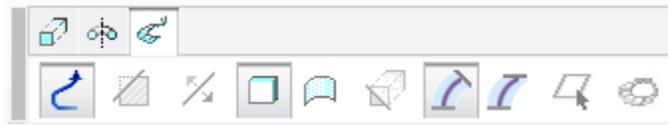


Figure 3-15 The live toolbar

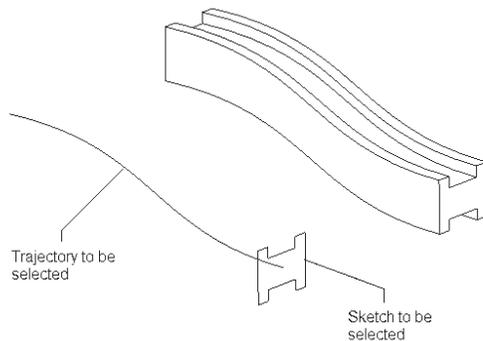


Figure 3-16 Section swept along the sketched trajectory

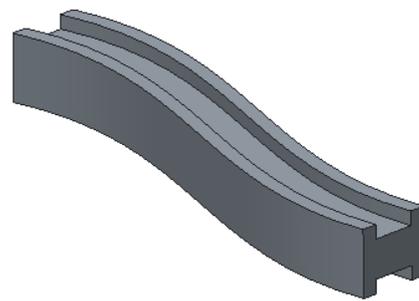


Figure 3-17 Shaded image of the sweep feature

The options available in the live toolbar are discussed next.

Normal to Trajectory



This option is used to make the sweep section normal to the trajectory of sweep. Figure 3-18 shows the preview displayed after selecting the **Normal to Trajectory** option.

Constant Normal



This option is used to make the sweep section parallel to the plane in which the sketch is created. Figure 3-19 shows the preview of sweep displayed after selecting the **Constant Normal** option.

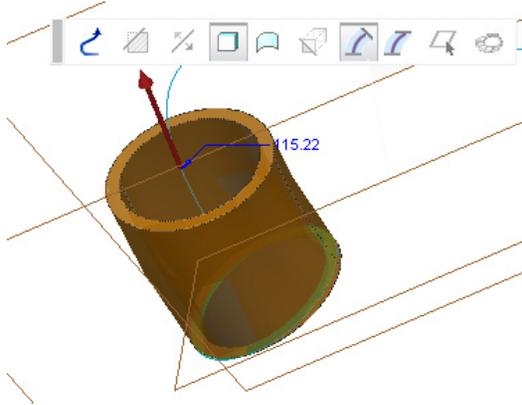


Figure 3-18 Sweep section normal to the trajectory

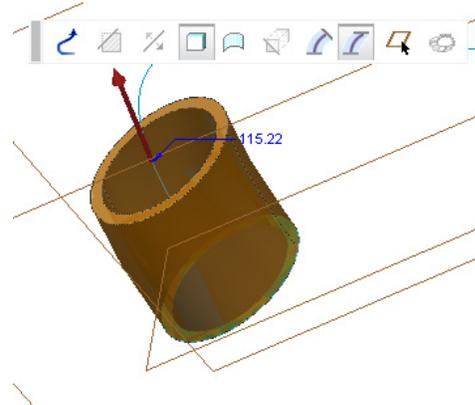


Figure 3-19 Sweep section parallel to the plane

Constant Normal Direction



This option will become active when you select the **Constant Normal** option. This option is used to select a new reference plane with which you need to make sweep feature parallel. Figures 3-20 and 3-21 show the previews of the sweep feature with two different planes selected as reference.

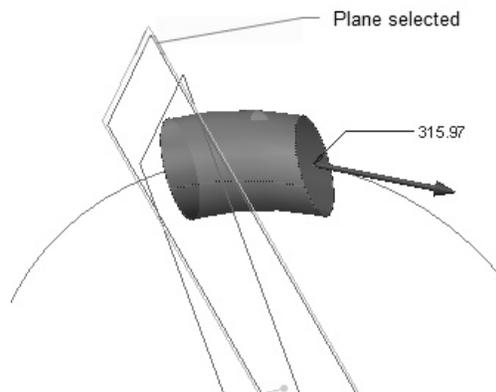


Figure 3-20 Section swept along the sketched trajectory

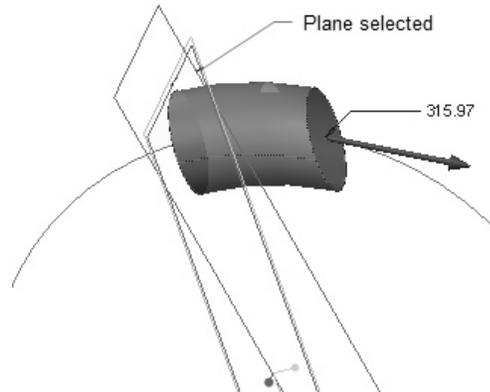


Figure 3-21 Section swept along the sketched trajectory

Full



This option is used to sweep the sketch to the entire length of a closed sketch. Figures 3-22 and 3-23 show the previews of sweep feature before and after using the **Full** option. Alternatively, you can use drag handle to drag the length of sweep.

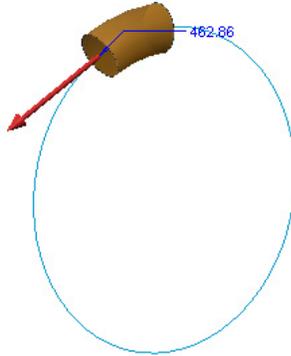


Figure 3-22 Section swept along the sketched trajectory



Figure 3-23 Section swept along the sketched trajectory



Note

You can also invoke the **Sweep** tool from the **Shape** group of the **Home** tab. This method is generally used when the **Select (no default)** tool is selected in the **Select** drop-down of the **Selection** group. The procedure to create the sweep feature is same as explained above.

Rest of the options available in the live toolbar are similar as discussed while creating extrude features.



Tip: The following points should be remembered while creating a sweep feature:

1. Similar to other sketched features, the trajectory of the sweep feature is also sketched after selecting a sketching plane.
2. At bends in a trajectory, the radius of the bend should be proportionate to the cross-section to be swept to avoid overlapping. If the section size is large and the radius of the curve or bend is small, overlapping takes place and the sweep feature will not be created. Therefore, make sure that the ratio of the size of the section to the size of the trajectory is appropriate.

NESTING OF SKETCHES

In the sketch of a single feature, if you draw multiple closed loops one inside the other, it is called nesting of sketches. These sketches are drawn in the sketcher environment. Figure 3-24 shows a nested sketch in which two circles are created on the base profile. Figure 3-25 shows the resultant extruded feature. The advantages of nesting the sketches is that the number of features used to create a model is reduced. For example, in Figure 3-25, the model has two features. The base feature is the base plate and the second feature are the holes in the model. But, when you nest the two sketches to create the model, you used only one feature to create the model. Also, there is a disadvantage of nested sketch. In nesting, since the two features on the model are combined into one feature, therefore there is no flexibility in editing the features of a model.

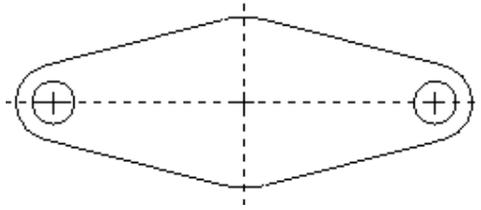


Figure 3-24 Nested sketch

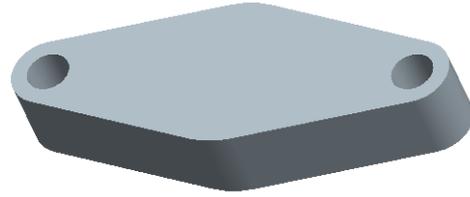


Figure 3-25 Solid model of the sketch



Note

After understanding the advantages and disadvantages of nesting the sketches, it is recommended to divide a model into separate features. Draw all features as individual features so that the created model is flexible. However, it depends on the need of the designer and the approach of creating the model.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 3-26. The dimensions of the model are shown in Figure 3-27. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- Set the working directory and create a new object file in the **Part** mode.
- First examine the model and then determine the type of protrusion for the model. Select the sketching plane for the model and orient it parallel to the screen.
- Draw the sketch by using the sketching tools, refer to Figures 3-28.
- Create the model by extruding the sketch, refer to Figure 3-30.

Setting the Working Directory

After starting Creo Direct session, the first task is to set the working directory. A working directory is a directory on your system where you can save the work done in the current session of Creo Direct. You can set any existing directory on your system as the working directory.

- Choose the **Manage Session > Select Working Directory** from the **File** menu; the **Select Working Directory** dialog box is displayed. Browse to the *C:\CreoDirect-2.0* folder.
- Choose the **Organize** button from the **Select Working Directory** dialog box; a flyout is displayed. Choose the **New Folder** option from the flyout; the **New Folder** dialog box is displayed.

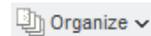




Figure 3-26 The isometric view of the model

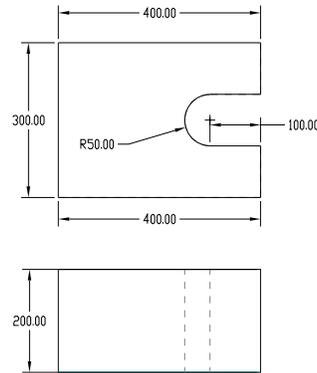


Figure 3-27 The front and right views of the model with dimensions

3. Enter **c03** in the **New Directory** edit box and choose **OK** from the **New Folder** dialog box. Now, you have created a folder named *c03* in *C:\CreoDirect-2.0*.
4. Next, choose **OK** from the **Select Working Directory** dialog box; the working directory is set to *C:\CreoDirect-2.0\c03*.

Starting a New Object File

Solid models are created in the **Part** mode of Creo Direct. The file extension for the files created in this mode is *.prt*.

1. Choose the **New** option from the **File** menu; the **New** dialog box is displayed. The **Part** button is chosen by default in the **Type** area of the **New** dialog box.
2. Enter the file name as *c03tut1* in the **Name** edit box and choose the **OK** button. The TOP plane is displayed in the drawing area, by default. Also, the **Model Tree** appears on the left of the drawing area in the Navigator.
3. Close the **Model Tree** by choosing the **Toggle the display of the navigation area** button available at the right edge of the Navigator. Now, the drawing area is increased. 

Drawing the Sketch

You need to draw the sketch of the solid model that will be extruded later to create the 3D model.

1. Choose the **Sketch Plan View** button from the **In-graphics** toolbar to make the sketching plane parallel to the screen.

2. Choose the **Line/Arc Chain** tool from the **Line** drop-down of the **Sketch** group in the **Home** tab; you are prompted to specify the first point of line. 
3. Specify **0** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **0** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. Move the cursor horizontally towards the left direction and specify the length of line as **400**.
6. Move the cursor in the vertical upward direction and specify the length of line as **300**.
7. Move the cursor horizontally towards the right direction and specify the length of line as **400**.
8. Move the cursor in the vertical downward direction and specify the length of line as **100**.
9. Move the cursor horizontally towards the left direction and specify the length of line as **100**.
10. Move the cursor horizontally towards the left direction and press S. Next, move the cursor in the vertical downward direction and specify the arc length and angle for the arc as **100** and **270**, respectively.
11. Next, move the cursor horizontally towards the right direction and specify the length of line as **100**.
12. Move the cursor in vertical downward direction and specify the length of line as **100**. Figure 3-28 shows the sketch created.

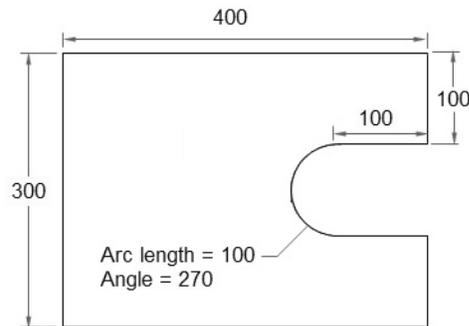


Figure 3-28 Sketch created

Extruding the Sketch

1. Choose the **Extrude** tool available in the **Shape** group of the **Home** tab; you are prompted to select a sketch region to extrude.
2. Select the sketch, refer to Figure 3-28; the preview of the extrude with default extrude value is displayed, as shown in Figure 3-29. Also, the live toolbar is displayed. You can orient the model by using the middle mouse button.

3. Double-click on the extrude value; a edit box is displayed.
4. Specify the value of extrude as **200** in the edit box and press ENTER. Next, click anywhere in the empty area of the screen; the extrude feature is created, as shown in Figure 3-30.

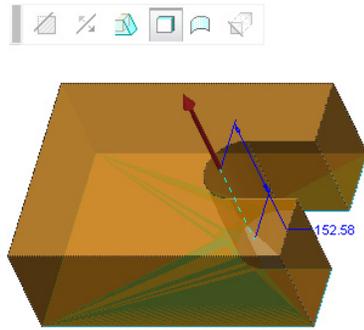


Figure 3-29 Preview of the extrude displayed with the live toolbar

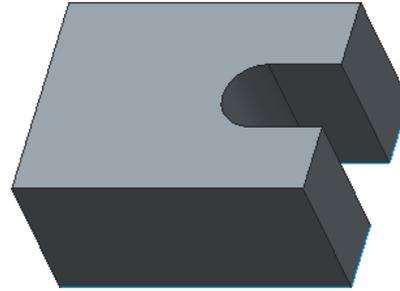


Figure 3-30 The extrude feature created

Saving the Model and Closing the File

1. Choose the **Save** option from the **File** menu or choose the **Save** button from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name of the object file that you have specified earlier in the **Model Name** edit box of the dialog box.
2. Choose the **OK** button from the **Save Object** dialog box to save the file.
3. Choose **File > Close** from the File menu to close the file.



Note

*You can also change the current name of the file before saving it by choosing the **Rename** option from the **Manage File** flyout in the **File** menu. On doing so, the **Rename** dialog box will be displayed. Enter the required name of the file in the **New Name** edit box and then choose the **OK** button to accept the new name.*

Tutorial 2

In this tutorial, you will create the model shown in Figure 3-31. The dimensions of the model are shown in Figure 3-32. **(Expected time: 30 min)**

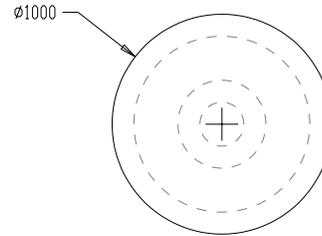
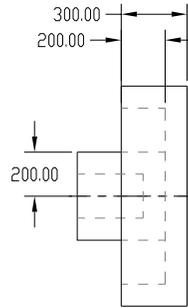
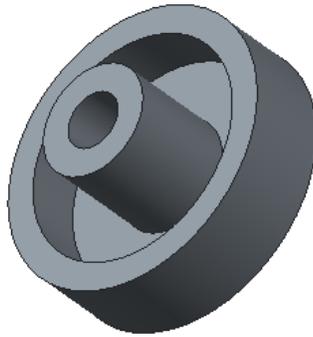


Figure 3-31 The isometric view of the solid model

Figure 3-32 Different views of the solid model

The following steps are required to complete this tutorial:

- Create a new object file in the **Part** mode.
- First examine the model and then determine the type of protrusion required for the model. Next, select the sketching plane for the model.
- Draw the sketch for the revolved feature and a centerline to revolve it, refer to Figure 3-33.
- Revolve the sketch, as shown in Figure 3-35.
- Save the model and exit the environment.

Setting the Working Directory

The working directory was selected in Tutorial 1; therefore, there is no need to select it again. But if you are starting a new session of Creo Direct, you need to set the working directory again by following the steps given below.

- Open the Navigator by choosing the **Toggle the display of the navigation area** button at the bottom left corner of the main window of Creo Direct; the Navigator slides out. In the Navigator, choose the **Folder Browser** tab; the **Folder Tree** is displayed at the bottom. Click on the black arrow on the right of the **Folder Tree**; the **Folder Tree** expands.
- Click on the node adjacent to the *CreoDirect-2.0* folder in the Navigator; the contents of the *CreoDirect-2.0* folder are displayed.
- Now, right-click on the *c03* folder to display a shortcut menu. Choose the **Set Working Directory** option from the shortcut menu; the working directory is set to *c03*.

4. Close the **Model Tree** by choosing the **Toggle the display of the navigation area** button available at the right edge of the Navigator.



Starting a New Object File

1. Open a new object file in the **Part** mode and then name the file as *c03tut2*. The default datum plane is displayed in the drawing area.
2. If the datum planes are not displayed by default then turn on the display of datum planes by using the **Plane Display** button from the **Datum Display Filters** drop-down in the **In-graphics** toolbar.

Drawing the Sketch

Now, you need to draw the sketch of the revolved feature. The sketch to be drawn is the cross-section of the revolved feature, which will be revolved about the centerline.

1. Choose the **Sketch Plan View** button from the **In-graphics** toolbar to make the sketching plane parallel to the screen.
2. Choose the **Line/Arc Chain** tool from the **Line** drop-down of the **Sketch** group in the **Home** tab; you are prompted to specify the first point of line. 
3. Specify **0** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **0** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. Move the cursor in vertical upward direction and specify the length of line as **500**.
6. Move the cursor horizontally towards the left direction and specify the length of line as **300**.
7. Move the cursor in the vertical downward direction and specify the length of line as **100**.
8. Move the cursor horizontally towards the right direction and specify the length of line as **200**.
9. Move the cursor in the vertical downward direction and specify the length of line as **200**.
10. Move the cursor horizontally towards the left direction and specify the length of line as **400**.
11. Move the cursor in the vertical downward direction and specify the length of line as **100**.

12. Move the cursor horizontally towards the right direction and specify the length of line as **300**. Next, move the cursor in vertical downward direction and specify the length of line as **100**.
13. Move the cursor horizontally towards the right direction and specify the length of line as **200**.

Figure 3-33 shows the sketch created for creating the revolved feature.

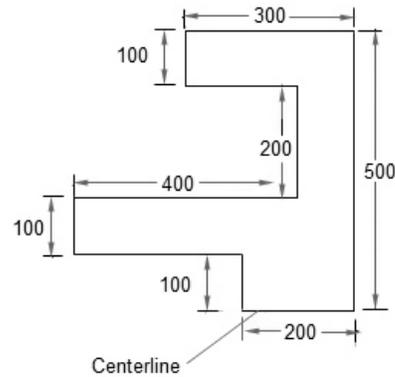


Figure 3-33 Sketch created

Revolving the Sketch

1. Choose the **Revolve** tool available in the **Shape** group of the **Home** tab; you are prompted to select a sketch region to revolve.
2. Select the sketch, refer to Figure 3-33; you are prompted to select the axis or straight edge for axis of rotation.
3. Select the edge of sketch as centerline, refer to Figure 3-33; the preview of the revolve feature is displayed with default revolve value, as shown in Figure 3-34. Also, the live toolbar is displayed. You can use the middle mouse button to orient the model.
4. Double-click on the angle value; a edit box is displayed.
5. Specify the angle for revolve as **360** in the edit box and press ENTER. Next, click anywhere in the empty area of the screen; the revolve feature is created, as shown in Figure 3-35.

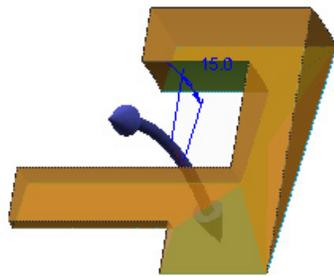


Figure 3-34 Preview of the revolve feature displayed

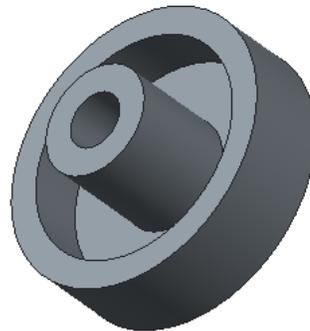


Figure 3-35 Revolve feature created

Saving the Model

1. Choose the **Save** option from the **File** menu or choose the **Save** button from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name of the object file specified earlier.
2. Choose the **OK** button to save the file.

Closing the Current Window

The given model is completed and is also saved. Now, you can close the current window.

1. Choose **File > Close** from the menu bar.



Note

*If you need to view the **Model Tree** of the model that you have created, then you need to open it. To do so, choose the **Toggle the display of the navigation area** button on the bottom left corner of the screen; the **Model Tree** is displayed. To close the **Model Tree**, choose the button again.*

Tutorial 3

In this tutorial, you will create the model shown in Figure 3-36. The dimensions of the sketch are shown in Figure 3-37. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- a. Create a new object file in the **Part** mode.
- b. First examine the model and then determine the type of protrusion for the model.
- c. Draw the sketch by using the sketching tools, refer to Figure 3-38.
- d. Extrude the model, refer to Figure 3-40.
- e. Save the model and close the file.



Figure 3-36 The isometric view of the model

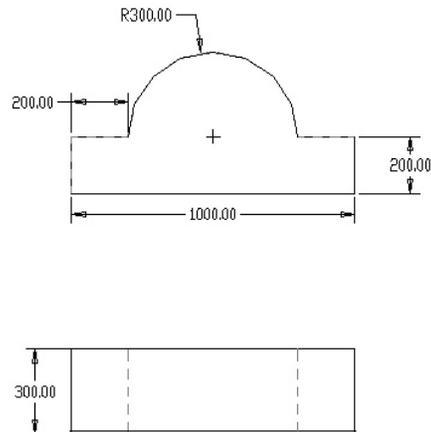


Figure 3-37 Front and bottom views of the model

The working directory was selected in Tutorial 1, and therefore, you do not need to select the working directory again. But if a new session of Creo Direct is started, then set the working directory using the Navigator.

Starting a New Object File

1. Open a new object file in the **Part** mode and then name it as *c03tut3*.

If the default datum plane is not turned off in the previous tutorial, they will appear in the drawing area. If the datum plane is not displayed, then turn them on by using the **Plane Display** button.

Drawing the Sketch

In this section, you need to draw the sketch for the extruded model using the sketcher tools.

1. Choose the **Sketch Plan View** button from the **In-graphics** toolbar to make the sketching plane parallel to the screen.
2. Choose the **Line/Arc Chain** tool; you are prompted to specify the first point of line.
3. Specify **0** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **0** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. Move the cursor horizontally towards the left direction and specify the length of line as **1000**.
6. Move the cursor in the vertical upward direction and specify the length of line as **200**.
7. Move the cursor horizontally towards the right direction and specify the length of line as **200**.
8. Move the cursor in the vertical upward direction and press S.
9. Next, move the cursor horizontally towards the right direction and specify the length and angle of arc as **600** and **0**, respectively.
10. Move the cursor horizontally towards the right direction and specify the length of line as **200**.
11. Move the cursor in the vertical downward direction and specify the length of line as **200**; refer to Figure 3-38.

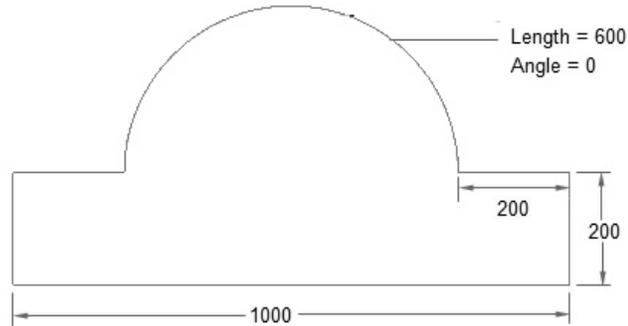


Figure 3-38 Sketch of the thin extruded model

Extruding the Sketch

1. Choose the **Extrude** tool available in the **Shape** group of the **Home** tab; you are prompted to select the sketch region to extrude.
2. Select the sketch, refer to Figure 3-38; the preview of the extrude feature is displayed, as shown in Figure 3-39. Also, the live toolbar is displayed
3. Double-click on the extrude value; a edit box is displayed.
4. Specify the value as **300** in the edit box and press ENTER. Next, click anywhere in the empty area of the screen; the extrude feature is created, as shown in Figure 3-40.

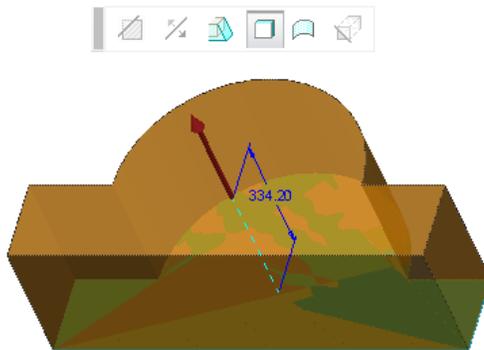


Figure 3-39 Preview of the extrude feature displayed



Figure 3-40 The extrude feature created

Saving the Model

1. Choose the **Save** option from the **File** menu or choose the **Save** button from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name of the object file specified earlier.
2. Choose the **OK** button to save the file.

Closing the Current Window

The given model is completed and is also saved. Now, you need to close the current window.

1. Choose **File > Close** from the menu bar.

Tutorial 4

In this tutorial, you will create the model shown in Figure 3-41. The dimensions of the sketch are shown in Figure 3-42. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- a. Create a new object file in the **Part** mode.
- b. First examine the model and then determine the type of protrusion for the model.
- c. Draw the sketch by using the sketching tools, refer to Figure 3-43.
- d. Create the Sweep feature, refer to Figures 3-44 and 3-45.
- e. Save the model and close the file.

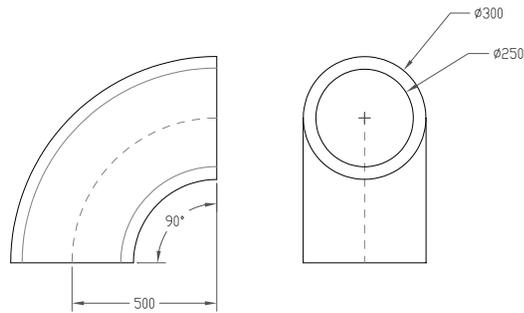


Figure 3-41 The isometric view of the model

Figure 3-42 Drawing views of the model with dimensions

The working directory was selected in Tutorial 1, and therefore, you do not need to select the working directory again. But if a new session of Creo Direct is started, then set the working directory using the Navigator as discussed earlier.

Starting a New Object File

1. Open a new object file in the **Part** mode and then name it as *c03tut4*.

If the default datum plane is not turned off in the previous tutorial, they will appear in the drawing area. If the datum plane is not displayed, turn them on by using the **Plane Display** button.

Drawing the Sketch

1. Choose the **Sketch Plan View** button from the **In-graphics** toolbar to make the sketching plane parallel to the screen.
2. Choose the **Center & Ends** tool from the **Arc** drop-down of the **Sketch** group in the **Home** tab; you are prompted to specify the first point for creating an arc.
3. Specify **500** in the X edit box of the dynamic input.
4. Next, press the down key; the Y edit box is activated. Specify **0** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
5. Move the cursor horizontally towards the left direction and specify the radius and angle for the first point as **500** and **180**, respectively.
6. Move the cursor in the vertical upward direction and specify the angle as **90** to specify the second point.
7. Next, select the **Select (no default)** option from the **Select** drop-down list of the **Selection** group in the **Home** tab to exit the sketcher environment.
8. Choose the **Center and Point** tool from the **Circle** drop-down of the **Sketch** group in the **Home** tab; you are prompted to define the plane for sketching.
9. Select the **FRONT** option from the **Planes** drop-down list of the **Model Tree**; the Front plane is selected for sketching. Next, choose the **Sketch Plan View** button from the **In-graphics** toolbar to make the sketching plane parallel to the screen.
10. Specify **0** in the X edit box of the dynamic input.
11. Next, press the down key; the Y edit box is activated. Specify **0** in the Y edit box and press ENTER; the start point of the line is specified and one end of the line is attached to the cursor.
12. Next, specify the radius of circle as **150** in the dynamic input.

13. Similarly, create the another circle with start point as **0, 0** and radius value of **125**.
14. Exit the sketcher environment and choose the **Standard Orientation** button from the **In-graphics** toolbar. Figure 3-43 shows the sketch created for creating the sweep feature.

Creating the Sweep feature

1. Choose the **Sweep** tool available in the **Shape** group of the **Home** tab; you are prompted to select the sketch region to sweep.
2. Select the region between two circles to define the sketch region, refer to Figure 3-43; you are prompted to select the line or spline to define the trajectory.
3. Select the arc to define the trajectory, refer to Figure 3-43; the preview of sweep with the default sweep value is displayed, as shown in Figure 3-44. Also, the live toolbar and dragger are displayed.
4. Drag the dragger to the full length of trajectory.
5. Next, click anywhere in the empty area of the screen; the sweep feature is created, as shown in Figure 3-45.

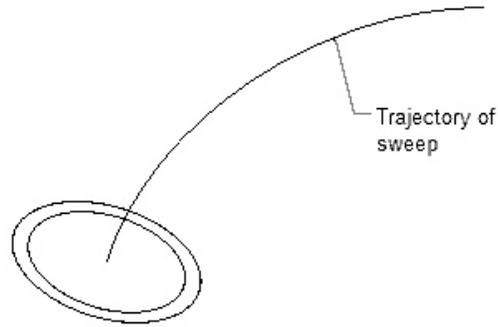


Figure 3-43 Sketch created

Saving the Model

1. Choose the **Save** option from the **File** menu or choose the **Save** button from the **Quick Access Toolbar**; the **Save Object** dialog box is displayed with the name of the object file specified earlier.
2. Choose the **OK** button to save the file.

Closing the Current Window

The given model is completed and is also saved. Now, you need to close the current window.

1. Choose **File > Close** from the menu bar.

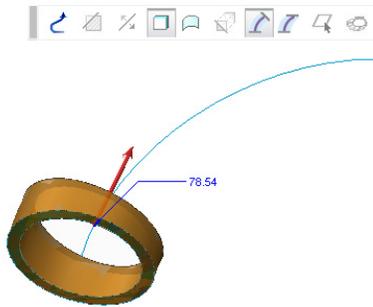


Figure 3-44 Preview of the sweep created



Figure 3-45 Sweep feature created

Self-Evaluation Test

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

1. All features created in the **Part** mode are called the base features. (T/F)
2. You can extrude a sketch in Creo Direct to both sides of the sketching plane symmetrically. (T/F)
3. You can create a sweep feature using a closed section and a closed trajectory. (T/F)
4. The _____ button is chosen by default when the live toolbar is displayed.
5. The _____ button in the live toolbar is used to create a thin revolved model.
6. After you exit the sketcher environment, the _____ appear on the model to dynamically modify the extrusion depth or the angle of revolution.
7. The **Revolve** tool revolves the sketched section about a _____ through the specified angle.
8. The **Sweep** option extrudes a section along a _____.
9. The cross-section of a swept feature remains _____ throughout the sweep.

Review Questions

Answer the following questions:

- Which of the following groups in the **Home** tab contains the **Extrude** tool?
 - Edit**
 - Engineering**
 - Shape**
 - Selection**
- How many datum planes are displayed by default when you enter the **Part** mode?
 - 4
 - 3
 - 1
 - None of these
- The revolved section should have a centerline. (T/F)
- A revolved section can be drawn on both sides of a centerline. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 3-46. The dimensions of the model are shown in Figure 3-47. **(Expected time: 20 min)**

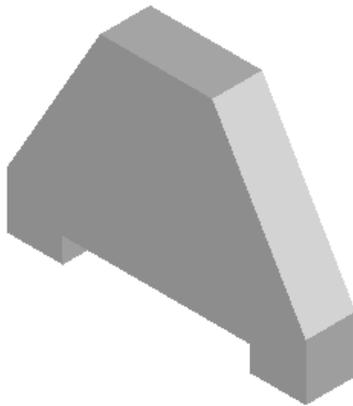


Figure 3-46 The isometric view of the model

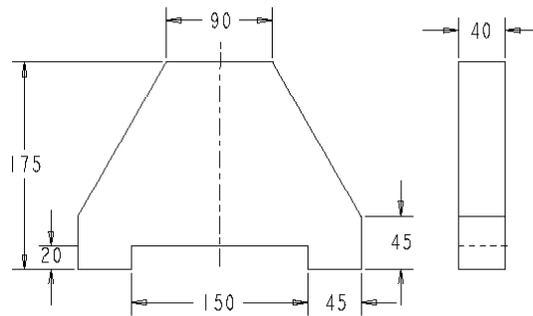


Figure 3-47 The front and right views of the model

Exercise 2

Create the model shown in Figure 3-48. The dimensions of the model are shown in Figure 3-49. **(Expected time: 30 min)**

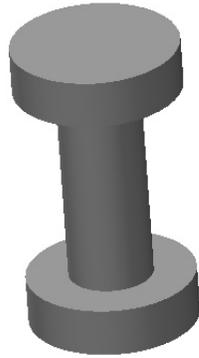


Figure 3-48 The isometric view of the model

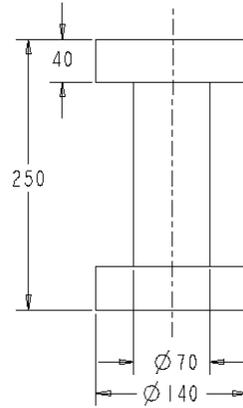


Figure 3-49 The front view of the model

Exercise 3

Create the model shown in Figure 3-50. The dimensions of the model are shown in Figure 3-51. **(Expected time: 30 min)**



Figure 3-50 The isometric view of the model

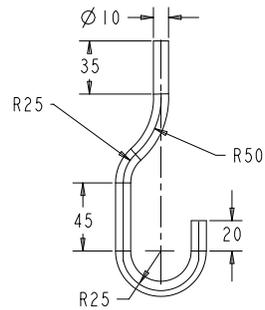


Figure 3-51 The front view of the model

Exercise 4

Create the model shown in Figure 3-52. The dimensions of the model are shown in Figure 3-53.
(Expected time: 30 min)

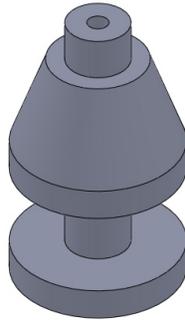


Figure 3-52 The isometric view of the model

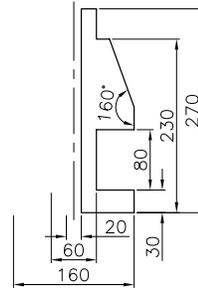


Figure 3-53 The front view of the model

Exercise 5

Create the model shown in Figure 3-54. The dimensions of the model are shown in Figure 3-55.
(Expected time: 30 min)

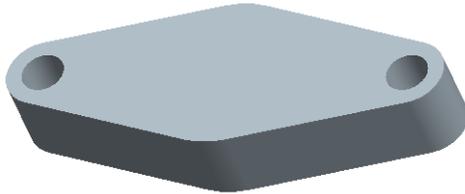


Figure 3-54 The isometric view of the model

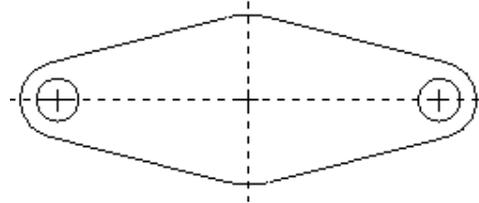


Figure 3-55 The front view of the model

Answers to Self-Evaluation Test

1. F, 2. T, 3. T, 4. Extrude as solid, 5. Thicken Sketch, 6. handles, 7. geometric centerline. 8. trajectory, 9. constant