Chapter 6

Options Aiding Construction of Parts-II

Learning Objectives

After completing this chapter you will be able to:

- *Create Linear pattern.*
- *Create Rotational pattern.*
- *Create Reference pattern.*
- *Use the Copy option.*
- *Use the Move option.*
- *Use the Mirror option.*
- *Use the Mirror Geom option.*

In this chapter you will learn about the different methods of duplicating the existing features. In Pro/ENGINEER, you can duplicate a feature by using the following methods:

- **Pattern**
- **Copy**
- **Mirror**

PATTERN

Patterns are one or two dimensional incremental array of features created from a single feature called the parent feature or the leader. When a pattern is created, the leader also becomes a part of the pattern. When you pattern a feature, Pro/ENGINEER prompts you to specify the total number of features to be created including the one that is being patterned and the increment in the dimensions if required.

Uses of Patterns

Patterns are very helpful in solid modeling and they speed up the model creation. The uses of patterns in solid modeling are discussed next.

1. Patterns create multiple copies of a feature, and hence save time of creating the features individually.

2. All the instances in a pattern, including the parent feature, act as a single feature. Therefore, they can easily be suppressed or mirrored.

3. All the instances in a pattern are related parametrically. Hence, you can modify the number of instances in a pattern, the spacing between the instances, and other parameters.

4. If the dimensions of the parent feature are modified then the dimensions of the child features are also modified.

Pattern Types

In Pro/ENGINEER, two types of patterns can be created. They are dimensional and reference patterns.

In the dimensional patterns, the existing dimensions of the parent feature are used to create a pattern. This pattern can be created in one direction or two directions. When you choose the option to create the pattern in the second direction, all instances that are created in the first direction are also created in the second direction. You need to specify the increment value for the instances. You can enter a positive or a negative value of the increment. Once you have specified the increment value in a direction, the system creates the specified number of instances (including the parent feature) in that direction.

In the reference pattern, an existing pattern is referenced to create a new pattern. In this type of pattern, the parent feature of the new pattern should be referenced to the parent feature of the existing pattern. Figure 6-1 shows a rib feature that is referenced to the parent hole feature. In this figure, the parent rib feature is created on a plane that was created on the fly using the

Make Datum option while creating the rib feature. This plane passes through the axis of the parent hole feature. Hence, a relationship is built between the parent rib feature and the parent hole feature. Now, when you select the rib feature to pattern, the pattern shown in Figure 6-2 is created without specifying the increment in dimensions.

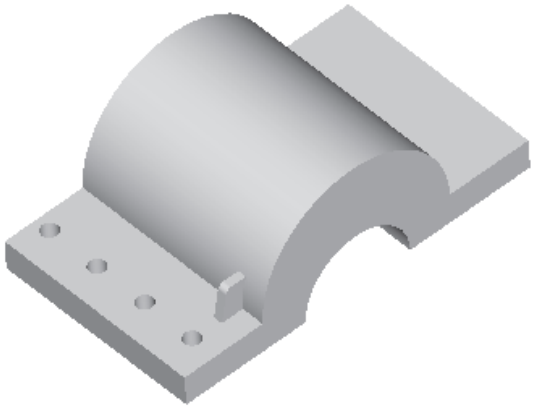


Figure 6-1 *Rib feature referenced to the hole feature*

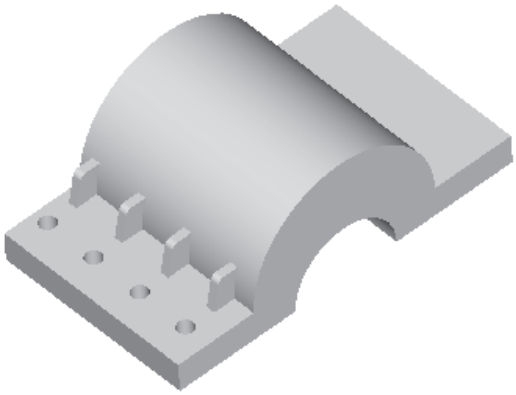


Figure 6-2 *Rib feature is Reference patterned*

Note

If you create the datum plane passing through the axis of the hole using the ***Insert a datum plane.*** *button and then create the rib feature on it, the rib feature will not be patterned with reference to the holes. This is because the rib does not have any direct relation with the hole. The rib has relation with the datum plane and the datum plane has a relation with the hole. However, since the datum plane has relation with the hole, you can pattern the datum planes with reference to the holes using the* ***Reference Pattern*** *option.*

To create a pattern whose parent feature is referenced to the parent feature of an existing pattern, the **PRO PAT TYPE** submenu shown in Figure 6-3 is used. This submenu is automatically displayed when you select features that can be patterned with reference to an existing pattern.

Figure 6-3 PRO PAT TYPE *submenu*

Pattern Options

Using the **Pattern** option from the **FEAT** menu you can create the dimensional patterns. Choose **PART > Feature > Pattern**; the **SELECT FEAT** submenu is displayed and the system prompts you to select a feature to be patterned. When you select a feature to be patterned the **PAT OPTIONS** submenu is displayed as shown in Figure 6-4. The options to create a pattern are discussed next.

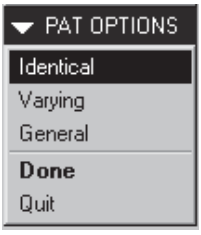


Figure 6-4 PAT OPTIONS *submenu*

Identical Pattern

The **Identical** option is used to create an identical pattern. You need to select at least one incremental dimension to pattern the feature. Depending upon the incremental dimension selected, the resultant pattern will be linear or rotational patterns. A linear

pattern is created when the driving dimension is linear and a rotational pattern is created when the driving dimension is angular. You can enter a positive or a negative value as the increment in a pattern dimension. All the instances of a pattern that are created by using this option are identical in size and geometry. This is the reason the patterns created by using this option are known as the **Identical** patterns. Figure 6-5 shows a hole feature on the base feature and Figure 6-6 shows holes patterned linearly. Similarly, Figure 6-7 shows a hole feature on the base feature and Figure 6-8 shows the rotational pattern of the hole feature.

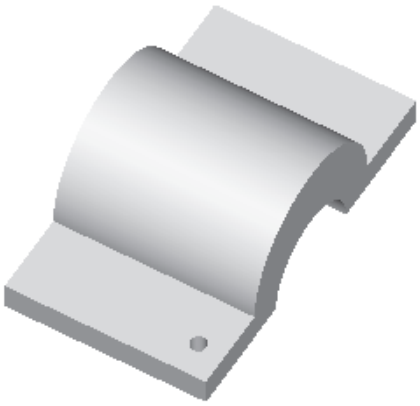


Figure 6-5 *Hole on the base feature*

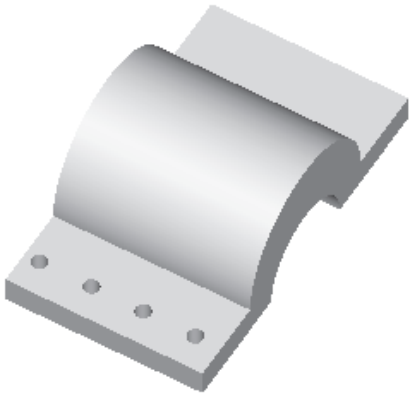


Figure 6-6 *Hole patterned on the base feature*

Figure 6-7 *Hole on the base feature*

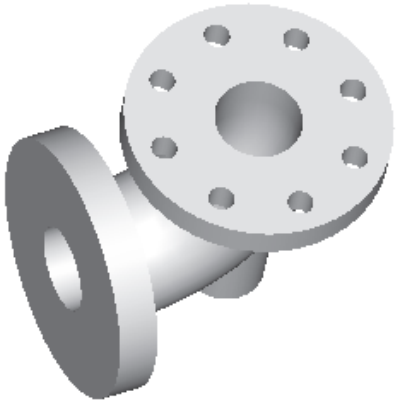


Figure 6-8 *Rotational pattern of hole feature*

As evident from Figures 6-6 and Figure 6-8, all the instances in the identical patterns are placed on the same placement surface and no feature intersects the edges of the placement surface, any other instance, or any other feature other than the placement surface. Note that it not possible to pattern the hole feature shown in Figure 6-5 on the right flap by using the identical patterns. However, you can use the **General** option to pattern the hole on the right flap of the model shown in Figure 6-5.

Varying Pattern

The **Varying** type of pattern is used when the instances vary in size. In this type of pattern the instances can be placed on different surfaces and can also intersect with the edges of the

placement surface. The feature shown in Figure 6-9 is patterned using the **Varying** option and is shown in Figure 6-10. In Figure 6-10, the length and the diameter of the rod varies in all the instances.

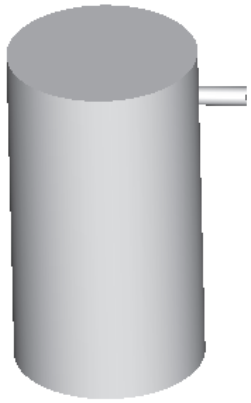


Figure 6-9 *A rod on the base feature*

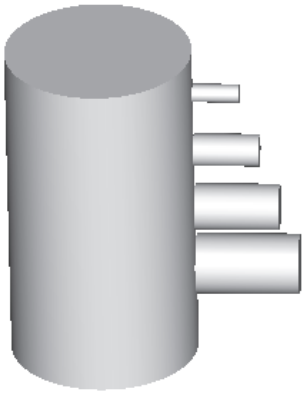


Figure 6-10 *Varying pattern of the rod*

General Pattern

The most complex patterns can be created using the **General** pattern. This is used to create patterns in which the instances touch each other and intersect with other instances or the edges of the surface. This option of creating patterns is also used when instances intersect with the base feature and the intersection is not visible. The hole shown in Figure 6-11 is patterned using the **General** option and is shown in Figure 6-12.

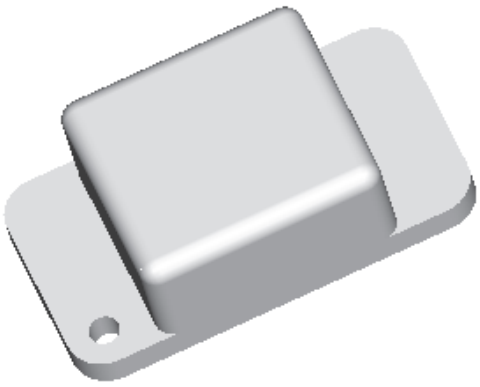


Figure 6-11 *Hole on the base feature*

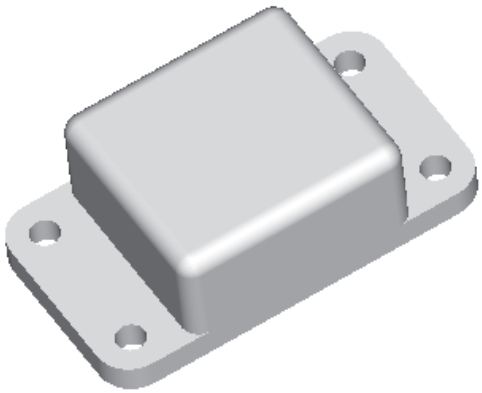


Figure 6-12 *General pattern*

Deleting a Pattern

You can delete a pattern by selecting it from the graphics screen or from the **Model Tree**. Chose the **Del Pattern** option from the **FEAT** menu in the **Menu Manager**. You will be prompted to select the feature to delete pattern. Select one of the features created using the pattern. When you select the feature, the pattern is deleted. However, note that the parent feature is not deleted when you delete the pattern by using the **Del Pattern** option even if you selected the parent feature for deleting the pattern.

COPY

The **Copy** option facilitates the model and speeds up its creation by copying and mirroring the selected features. This option reduces the time required in the model creation and also helps in maintaining the symmetry of the model. The **Copy** option is available in the **FEAT** menu in the **Menu Manager**.

When you choose the **Copy** option from the **FEAT** menu, the **COPY FEATURE** submenu appears with different options as shown in Figure 6-13. The different options of this submenu are explained next.

***Figure 6-13 COPY FEATURE** submenu*

New Refs

The **New Refs** option is used to copy a feature by varying the dimension values and by selecting new references. The references can be edges, axes, or placement planes. You can copy a feature using the **New Refs** option by two methods. You can keep the same dimensional or placement reference for the copied feature as that of the original feature by using the **Same** option in the **WHICH REF** menu shown in Figure 6-14. This means that you can use the same edge or surface as the references for the copied part. The other possibility is that you can use new references for the copied feature. This can be achieved by using the **Alternate** option in the **WHICH REF** menu. This provides you with a greater flexibility to copy the features.

Using the **GP VAR DIMS** menu shown in Figure 6-15, you can specify the dimensions that are to be varied while copying a feature with new references.

***Figure 6-14 WHICH REF** menu*

***Figure 6-15 GP VAR DIMS** menu*

Same Refs

When you use the **Same Refs** option, you have the flexibility to vary the dimensions of the copied features. You need to select the dimensions you want to vary while copying a feature. The dimensional and placement references of the copied feature are the same as for the source feature.

Mirror

The **Mirror** option is used to copy a feature by mirroring it about a specified datum plane or a planar surface. When you invoke this option, you are prompted to select the features to be mirrored. Once you select the features to be mirrored, you are prompted to select a plane or create a datum about which the features will be mirrored. As soon as you select a datum plane or a planar surface, the selected feature will be mirrored. As mentioned earlier, this option not only reduces the model completion time, it also helps in maintaining the symmetry in the features of a model. Figure 6-16 shows a rib feature that is to be mirrored and the datum plane about which the rib feature will be mirrored. Figure 6-17 shows the model after mirroring the rib feature and turning off the visibility of the model.

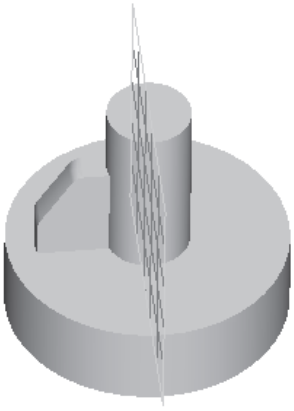


Figure 6-16 *Rib feature and the mirror plane*

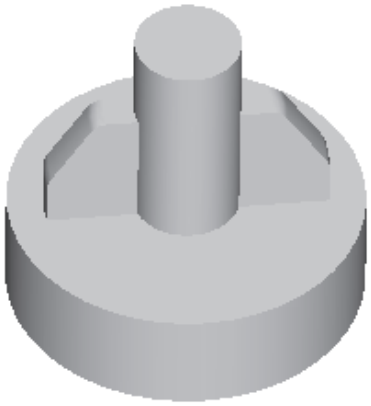
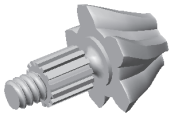


Figure 6-17 *Mirrored rib feature*



Tip: *To mirror a feature at an angle of 90 degrees to the parent feature, create a datum plane at an angle of 45 degrees to the parent feature and then use this datum plane to mirror the feature..*

Move

The **Move** option is used to copy features by translating or rotating them. When you invoke this option, you will be prompted to select the features to be translated. After selecting the features, you are prompted to define the movements by combination of translation and rotation. The **MOVE FEATURE** submenu is displayed as shown in Figure 6-18. The selected feature can be translated or rotated using the options in this submenu.

Figure 6-18 MOVE FEATURE *submenu*

Rotating Features

You can select a feature from the graphics screen or from the **Model Tree** and then rotate it about an axis, edge, datum curve, or coordinate system. Using this option you can select multiple features to copy.

Translating Features

You can select a feature from the graphics screen or from the **Model Tree** to copy it by translating

it. You need to specify a perpendicular plane and the direction in which the feature will be copied. You can also select multiple features to copy.

Select

The **Select** option provides the flexibility of choosing the features to be copied. When you use this option, you need to select the features either from the graphics screen or from the **Model Tree**. You can select any number of features to copy.

All Feat

The **All Feat** option is available only when you copy a feature using the **Mirror** or the **Move** option. When you use this option, all the features created are copied. Remember that you need to specify a coordinate system while using this option.

FromDifModel

The **FromDifModel** option is used to copy a feature from a different model. This option is available only when you are using the **New Refs** option of the **COPY FEATURE** submenu. This is due to change in the references required to copy from one model to another. Therefore, all the references will be new.

FromDifVers

You can copy features from a different version of the current model using the **FromDifVers** option. This option is available only when you are using the **New Refs** and the **Same Refs** options of the **COPY FEATURE** submenu.

Independent

The **Independent** option specifies that the dimensions of the copied features are independent of the dimensions of the parent feature. The features that you copy from a different model or different versions are independent by default. The dimensions of such copied features have no relation with the original feature. This is the reason the **Dependent** option is not available while using the **FromDifModel** or the **FromDifVers** options.

Dependent

The **Dependent** option specifies that the dimensions of the copied features are dependent on the dimensions of the parent feature. Therefore, if you make any modification in the section of the parent feature, the changes are automatically reflected in the copied feature.

MIRRORING A GEOMETRY

Pro/ENGINEER allows you to mirror an existing model about a datum plane. All the features in the model are mirrored about a datum plane. This option is different from the **Copy > Mirror** option. When you use the **Copy > Mirror** option, you can select features to be copied. But, while using the **MirrorGeom** option from the **FEAT** menu, the whole model is mirrored about a specified datum plane and the mirrored portion is automatically merged with the original portion. All the features in the mirrored model are related to the parent model. Any modification in the parent model reflects that modification in the mirrored model. Figure 6-19

shows a model and a datum plane for mirroring the model. Figure 6-20 shows the resultant mirrored model. Notice the merging in the resultant model.

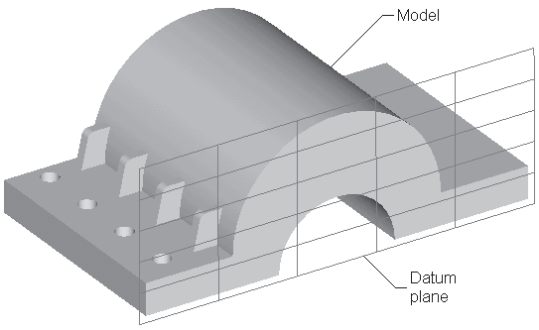


Figure 6-19 *Model and a plane*

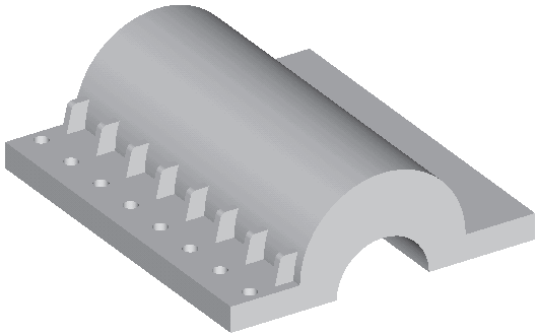


Figure 6-20 *Resultant mirrored model*

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 6-21. This figure also shows the top view, front view, and the right-side view of the model. **(Expected time: 30 min)**

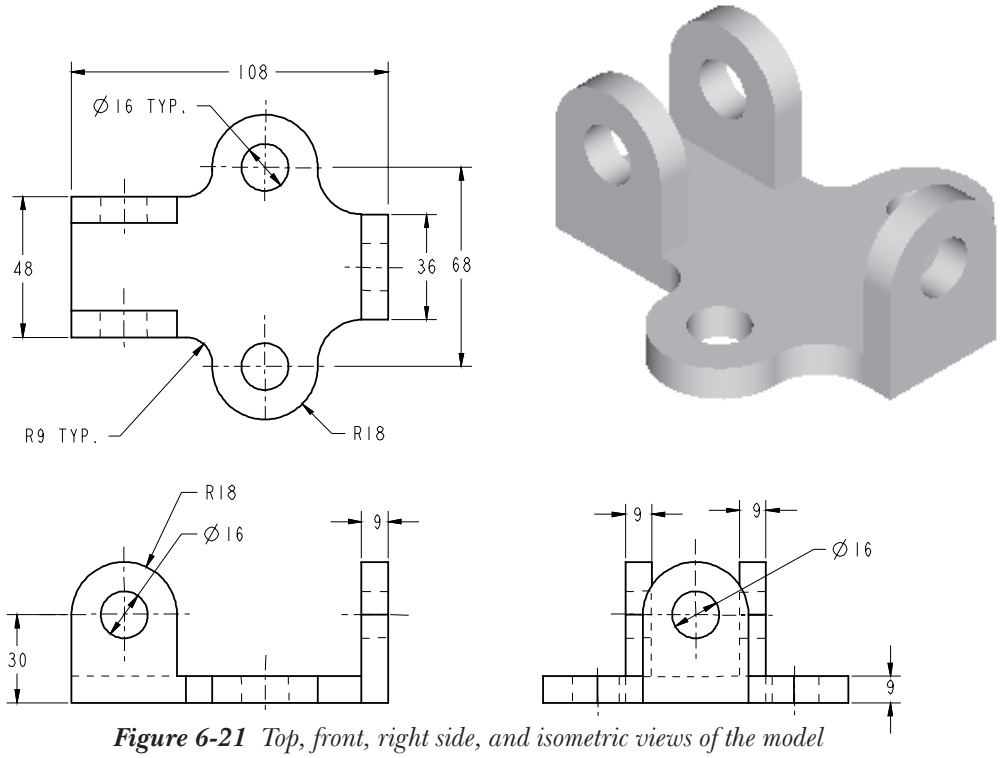


Figure 6-21 *Top, front, right side, and isometric views of the model*

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of four features, see Figure 6-21.

b. The base feature is an extruded feature, see Figure 6-23. First the sketch of the base feature will be created on the **TOP** datum plane, see Figure 6-22, and then it will be extruded to a depth of 9.

c. The second feature is also an extruded feature, see Figure 6-25. The sketch of the second feature will be created on the right planar surface of the base feature, see Figure 6-24, and then it will be extruded to a depth of 9.

d. The third feature is the same as the second feature, and therefore, a copy of it will be created by defining new references, see Figure 6-27.

e. The fourth feature is the same as the third feature and therefore, a mirror copy of it will be created as shown in Figure 6-28.

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 **c06**, if it does not exist. Choose the **New Directory** button in the **Select Working Directory** dialog box and create a directory named **c06** at **C:\ProE**.

Creating New Object File

1. Open a new part file and name it **c06tut1**. The three default datum planes will be displayed on the graphics screen. The **Model Tree** is also displayed on the graphics screen. Exit the **Model Tree** by choosing the **Model Tree on/off** button from the **Model Display** toolbar.

Creating the Base Feature

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

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

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

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

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

5. Once you enter the sketcher environment, create the sketch of the base feature and apply constraints and dimensions as shown in Figure 6-22. Note that in the sketch, the bottom half of the sketch is mirrored to create the top half of the sketch. This is evident from the constraints of symmetry applied to the sketch in Figure 6-22. Also, as evident from the sketch of the base feature shown in Figure 6-22, the **RIGHT** datum plane is located at a dimension of 18 from the left edge because later in the tutorial, this datum plane will be used as a reference to copy a feature.

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

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

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

8. Enter the value of the depth as **9** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

   The base feature is completed and now the second feature will be created. The default trimetric view of the base feature is shown in Figure 6-23.

Note

The two holes are integrated in the base feature. These holes are sketched while drawing the sketch for the base feature. Hence, the base feature is created as one single feature that includes the two holes. The other method is to create the two holes separately on the base feature using the ***HOLE*** *dialog box. In the method to create the features separately, the total features created will be three.*

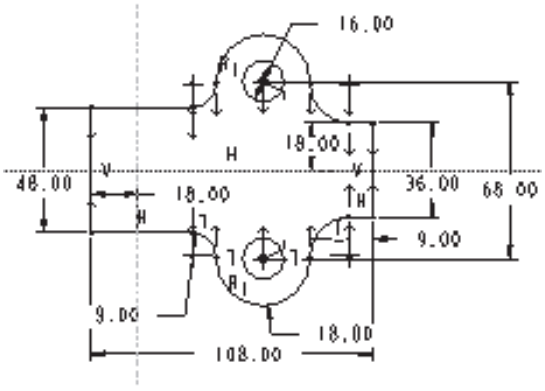


Figure 6-22 *Sketch of the base feature*

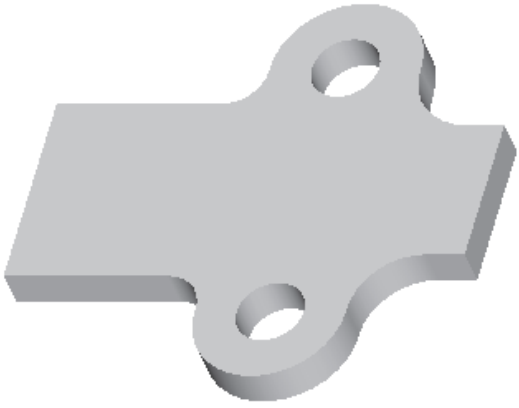


Figure 6-23 *The default trimetric view of the base feature*

Creating the Second Feature

The second feature is also an extruded feature and is created on the right planar surface of the base feature. Therefore, you need to select the right planar surface as the sketching plane.

Note that the sketch of this feature has to be created keeping in mind some important steps. The feature may not get copied later on if you do not use the following steps in creating the sketch of this feature.

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

2. Select the right planar surface of the base feature as the sketching plane. The **DIRECTION** submenu is displayed. Choose **Flip** and then choose the **Okay** option from this submenu. The **SKET VIEW** submenu is displayed.

3. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen. After entering the sketcher environment, turn the model display to **No Hidden**.

4. Choose the **Create lines.** button from the **Right Toolchest** and draw the right vertical line starting from the point shown in Figure 6-24. Notice that the endpoint of the right vertical line is not aligned with the edge on the top face of the base feature. Next, draw the horizontal line in continuation with the first line as shown in Figure 6-24. Notice that since the endpoint of the right vertical line is not aligned with the edge on the top face of the base feature, the horizontal line is also not aligned with that edge. Remember that if the horizontal line is aligned with the top face, the resultant feature may not get copied.

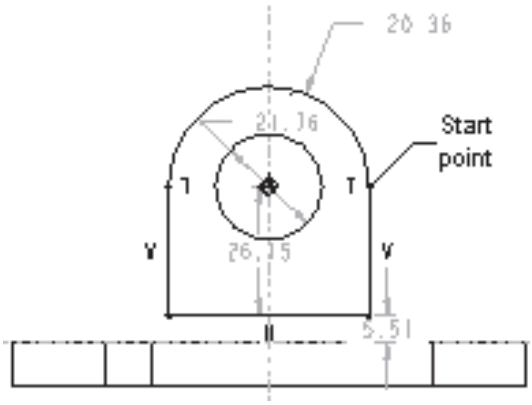


Figure 6-24 *Sketch of the second feature*

5. Next, draw the left vertical line in continuation with the horizontal line and then complete the sketch by drawing the arc and the circle, see Figure 6-24. Since the **Intent Manager** is on, some weak dimensions will be applied to the sketch.

6. Choose the **Impose sketcher constraints on the section.** button to display the **Constraints** dialog box. Choose the **Create same points, points on entity or collinear constraint** button and select the bottom left vertex of the sketch as the first point to apply this constraint. Now, select the edge on the top face of the base feature as the second entity to apply the constraint. You will notice that the sketch will extend such that the horizontal line is now aligned with the top face.

7. Since the **Create same points, points on entity or collinear constraint** button is still chosen, you will be prompted to select two entities or vertices to align. Select the vertex at the bottom right corner of the sketch and then select the edge on the top face of the base feature to apply the constraint. The **Resolve Sketch** dialog box will be displayed and you will be informed that the highlighted constraints conflict. Select the **Horizontal** constraint at the third number and then choose the **Delete** button to delete this constraint.

8. Add the dimensions and then modify them as shown in Figure 6-25. After completing the sketch, turn the model display to **Shading** and choose the **Continue with the current section** button. The **SPEC TO** menu is displayed.

9. The **Blind** option is selected by default in this menu, choose **Done**. The **Message Input Window** is displayed with a default value in it. Enter a value of **9** in the **Message Input Window** and press ENTER.

10. Choose **OK** from the **Protrusion** dialog box. The default shaded trimetric view of the model after creating the second feature is shown in Figure 6-26.

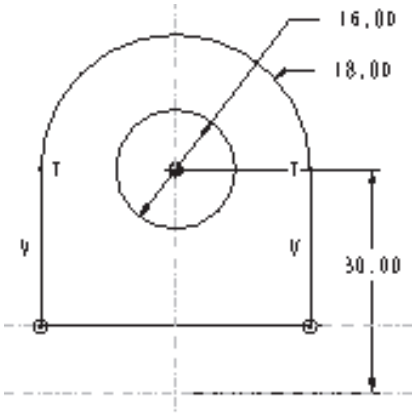


Figure 6-25 *Sketch of the second feature*

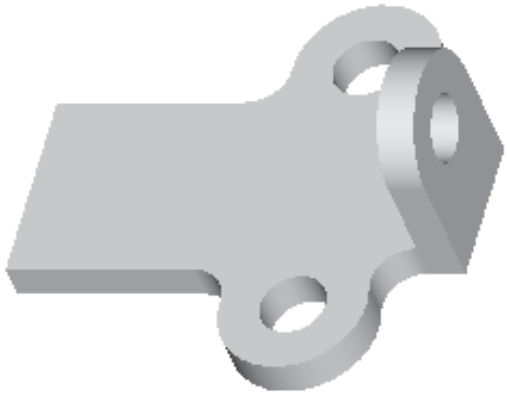


Figure 6-26 *Model after creating the second feature*

Creating the Third Feature

You can create the third feature using two methods. The first method is to draw the sketch of the third feature by defining a sketching plane and then extruding it to the given depth. The second method is to copy the second feature by defining new references. This is because the third feature is similar in geometry and dimensions to the second feature of the model. In this tutorial you will use the second method to create the third feature.

1. Choose **PART > Feature > Copy** from the **Menu Manager**. The **COPY FEATURE** submenu is displayed.

2. Choose **New Refs > Select > Dependent** from the **COPY FEATURE** submenu and choose **Done**. The **SELECT FEAT** submenu is displayed and you are prompted to select the feature to be copied.

3. Select the second feature from the graphics screen. The selected feature turns red in color.

4. Choose **Done** from the **SELECT FEAT** submenu. The **GP VAR DIMS** menu is displayed. Since you do not need to vary any dimension of the source feature, you can proceed further without selecting any dimension.

5. Choose **Done** from the **GP VAR DIMS** menu. The **WHICH REF** menu is displayed and

you are prompted to select a sketching plane reference corresponding to the highlighted surface.

6. Select the surface shown in Figure 6-27 from the graphics screen as the surface on which the copied feature will be placed. The TOP datum plane is highlighted and you are prompted to select a horizontal sketcher reference corresponding to the highlighted surface.

7. Choose the **Same** option from the **WHICH REF** menu. The FRONT datum plane is highlighted and you are prompted to select section dimensioning reference corresponding to highlighted surface.

8. Select the **RIGHT** datum plane. The top surface of the base feature is highlighted and you are prompted to select section dimensioning reference corresponding to the highlighted surface.

9. Choose the **Same** option from the **WHICH REF** menu. A red arrow is displayed and you are prompted to select the upward direction of the horizontal plane for protrusion.

10. Choose **Okay** from the **DIRECTION** submenu and then choose **Done** from the **GRP PLACE** menu. The feature is copied to the required location on the model as shown in Figure 6-28.

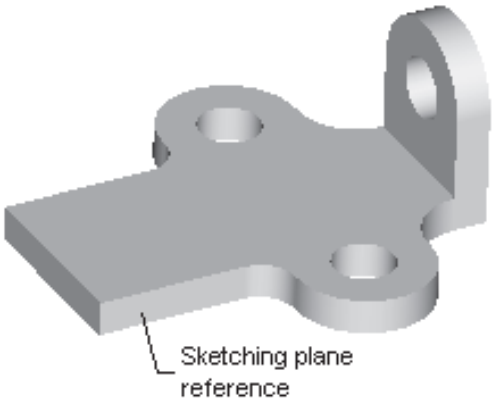


Figure 6-27 *Sketching plane reference for third feature*

Figure 6-28 *The copied third feature with new references*

Creating the Fourth Feature

The fourth feature can be created by sketching and extruding it to the given depth. You can also create this feature by placing a mirrored copy of the third feature at the required location. In this tutorial you will use the second method because it consumes less time.

1. Choose **PART > Feature > Copy** from the **Menu Manager**. The **COPY FEATURE** submenu is displayed. Choose **Mirror > Select > Dependent > Done** from the **COPY FEATURE** submenu. The **SELECT FEAT** submenu is displayed and you are prompted to select the features to be mirrored.

2. Select the third feature from the graphics screen and choose **Done** from the **SELECT FEAT** submenu. You are prompted to select a plane or create a datum to mirror about. Select the **FRONT** datum plane as the mirror plane. The third feature is mirrored about the **FRONT** datum plane. The trimetric view of the final model is shown in Figure 6-29.

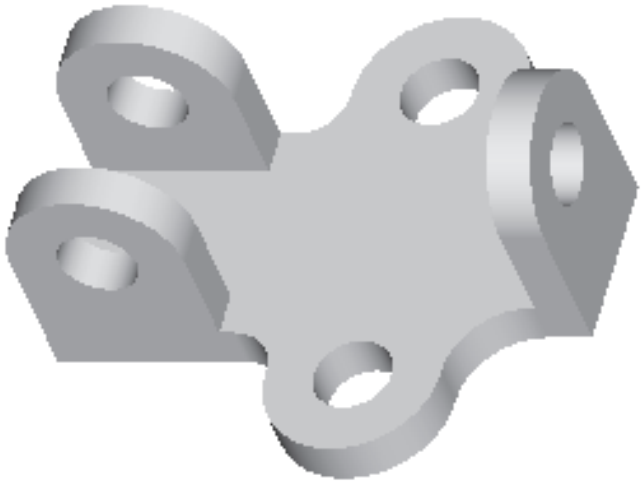


Figure 6-29 *Default trimetric view of the model*

3. 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 6-30. Note that the feature id in your model may be different from the one shown in this figure.

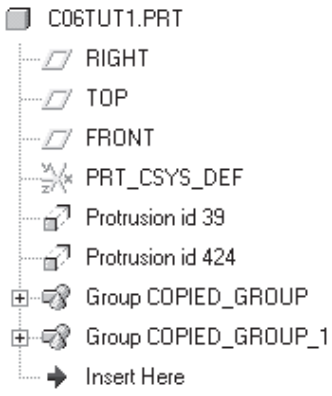


Figure 6-30 Model Tree *for Tutorial 1*

Tutorial 2

In this tutorial you will create the model shown in Figure 6-31. The dimensions of the model are given in the top view and front view of the model shown in Figure 6-32.

(Expected time: 45 min)

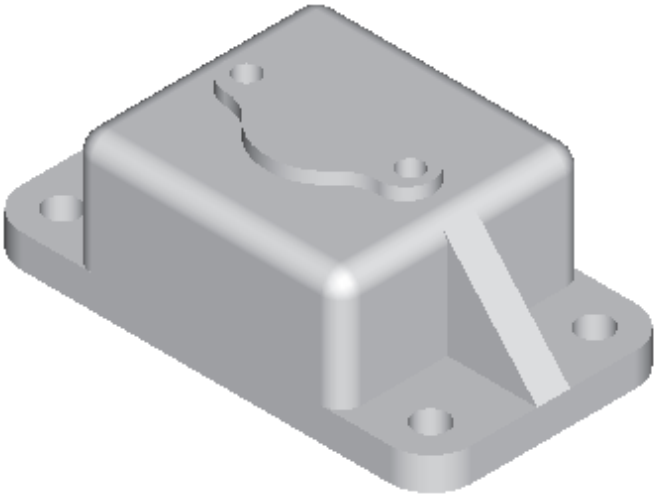


Figure 6-31 *Solid model for Tutorial 2*

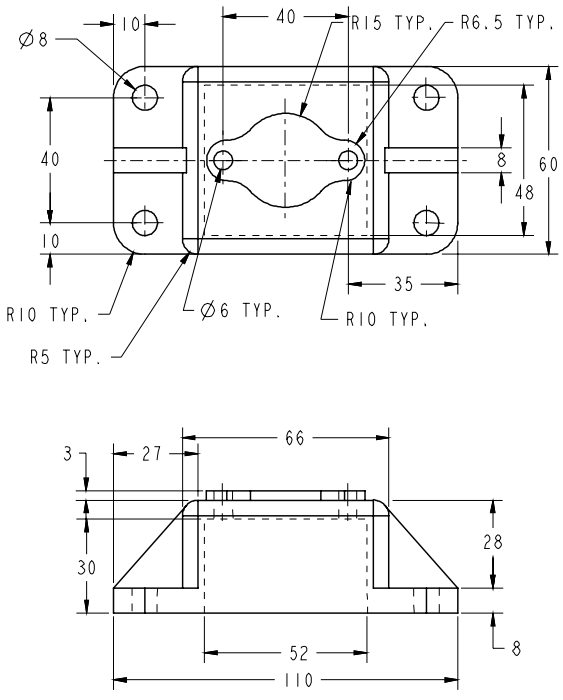


Figure 6-32 *Top view and front view of the model*

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of 10 features, see Figure 6-31.

b. The base feature is an extruded feature, see Figure 6-34. First the sketch of the base feature will be created on the **TOP** datum plane, see Figure 6-33, and then it will be extruded to a depth of 9.

c. The second feature is a round of radius 5 on the top portion of the base feature, see Figure 6-36.

d. The third feature is also a round feature of radius 10 on the bottom portion of the base feature, see Figure 6-37.

e. The fourth feature is a hole feature, see Figure 6-38, and then this hole will be patterned, see Figure 6-39. After you pattern the hole feature, the hole and the pattern features will be combined in a single feature.

f. The fifth feature is a rib feature, see Figure 6-41. The sketch of the rib feature will be created on the **FRONT** datum plane, see Figure 6-40, and then the given thickness will be applied to it.

g. The sixth feature is the mirror copy of the rib feature, see Figure 6-42.

h. The seventh feature is an extruded feature, see Figure 6-44. The sketch for this feature will be created on the top planar surface of the base feature and then it will be extruded to the given distance, see Figure 6-43.

i. The eighth feature is a cut feature, see Figure 6-46. The sketch for this feature will be created on the bottom planar surface of the base feature and then it will be extruded to the given distance, see Figure 6-45.

j. The ninth feature is a hole feature created on the seventh feature, see Figure 6-47.

k. The tenth feature is the copy of the hole feature, see Figure 6-48.

After understanding the procedure for creating the model, you are now ready to draw it. The working directory is already selected in Tutorial 1 and therefore you do not need to select it again. However, if you want to change the working directory, choose **File > Set Working Directory** and then select **c06** in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it **c06tut2**. The three default datum planes and the **Model Tree** is displayed on the graphics screen. Exit the **Model Tree**. The **Model Tree** will not appear if it has been previously turned off.

Creating the Base Feature

To create the sketch for the base feature, you first need to select the sketching plane for the base feature. In this model, you need to draw the base feature on the **FRONT** datum plane because the direction of extrusion of this feature is perpendicular to the **FRONT** datum plane. The base feature will also be created symmetric to the **FRONT** datum plane, and therefore the sketch will be created equally on both the sides of the **FRONT** datum plane.

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

   The **Both Sides** option is selected in order to create the base feature symmetrical to the **FRONT** datum plane. Later in the tutorial, the default datum planes will be used as mirror planes for mirroring features. Hence, you do not need to create datum planes.

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

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

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

5. Once you enter the sketcher environment, create the sketch of the base feature and apply constraints and dimensions as shown in Figure 6-33. Here in the sketch, you should note that since the base feature is symmetrical, therefore, a center line is drawn and then symmetrical constraint is applied to the vertices in the sketch. This is evident from the constraints of symmetry applied to the sketch in Figure 6-33. These constraints appear as arrow symbols in the sketch.

   As evident from the sketch of the base feature shown in Figure 6-33, the **TOP** datum plane is aligned with the bottom line segment.

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

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

7. Select the **Blind** option from this menu and choose **Done**.

8. Enter a depth of **60** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

   The base feature is completed and now the second feature will be created. The default trimetric view of the base feature is shown in Figure 6-34.

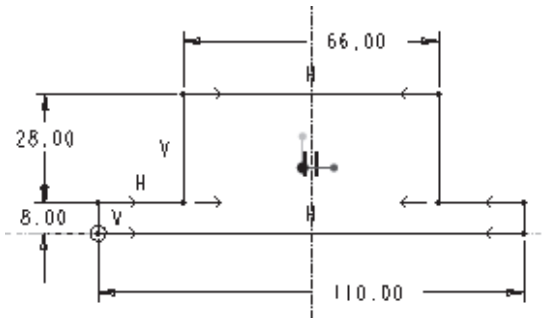


Figure 6-33 Sketch with dimensions and constraints for the base feature

Figure 6-34 Default trimetric view of the base feature

Creating the Second Feature

The second feature is a round feature of radius 5

1. Choose **Insert > Round** from the menu bar. The **ROUND TYPE** menu is displayed. The **Simple** option is selected by default; choose **Done** from this menu.

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

2. The **Constant > Edge Chain** option is selected by default. Choose **Done** from this menu. The **CHAIN** menu is displayed.

3. The **Tangnt Chain** option is selected by default and you are prompted to select an edge.

4. Select the edges shown in Figure 6-35 to round.

5. After selecting the edges, choose the **Done** option from the **CHAIN** menu. The **Message Input Window** is displayed and you are prompted to specify the radius of round.

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

7. Choose **OK** from the **ROUND** dialog box.

   The round feature is completed. The default trimetric view of the round feature is shown in Figure 6-36.

Creating the Third Feature

The third feature is also a round feature. Use the same options as discussed above to create this round feature. The only difference is that to create this round you will select the four vertical edges of the bottom portion of the base feature to round and specify a round radius of 10 in the **Message Input Window**. Figure 6-37 shows the round feature of radius 10.

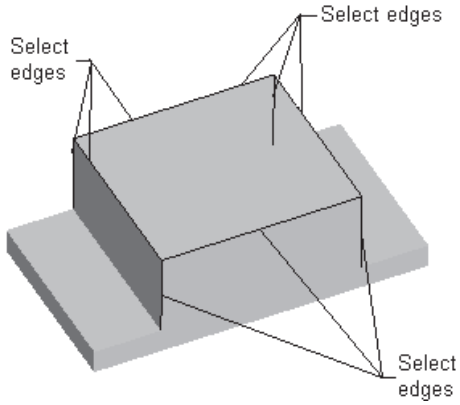


Figure 6-35 *Edges to be selected to round*

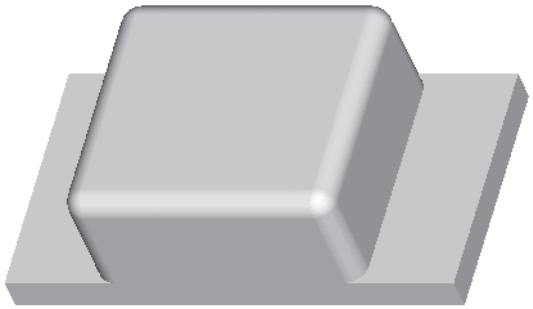


Figure 6-36 *The default trimetric view of the base feature with round*

Creating the Fourth Feature

The fourth feature is a through hole and will be created using the **HOLE** dialog box.

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

2. Create the hole as shown in Figure 6-38 by specifying the placement parameters. The placement parameters can be referred from Figure 6-32.

Figure 6-37 *The round feature of radius 10*

Figure 6-38 *The hole feature on the base feature*

Creating a Pattern of the Hole Feature

As evident from Figure 6-31, you need to create four instances of the hole. The first instance is created by using the **HOLE** dialog box and you can use the **Pattern** option to create the remaining three instances. You will create a rectangular pattern of the hole feature. You can also create all the holes using the **HOLE** dialog box and specify the placement parameters for each of them. But to save time, you will create a pattern of the hole.

1. Choose **PART > Feature > Pattern** from the **Menu Manager**. You are prompted to select the feature to be patterned.

2. Select the hole that is on the base feature. The **PAT OPTIONS** submenu is displayed.

3. Select the **General** option from the **PAT OPTIONS** submenu and choose **Done**. The placement dimensions are displayed on the hole feature.

   You cannot use the **Identical** option to create the rectangular pattern of the hole feature. This is because when you use the **Identical** option to pattern, the pattern cannot intersect the base feature on which the hole is created. If the top portion of the base feature is created as a separate feature, then the hole can be patterned using the **Identical** option.

4. Select the dimension value **10** from the graphics screen. Since the two dimensions that are displayed on the graphics screen are both 10, therefore, select the dimension 10 that is along the shorter side of the base feature. After you select the dimension in the first direction, the **Message Input Window** is displayed.

5. Enter a value of **40** in the **Message Input Window** and press ENTER. You are prompted to select a dimension in the same direction or choose Done.

6. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and you are prompted to enter the total number of instances in that direction including the original.

7. Enter **2** in the **Message Input Window** and press ENTER. Now, you are prompted to enter dimension increment in the second direction.

8. Select the dimension value **10** that is along the longer side of the base feature. After you select the dimension in the second direction, the **Message Input Window** is displayed.

9. Enter a value of **90** in the **Message Input Window** and press ENTER. You are prompted to select another dimension in the same direction.

10. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and you are prompted to enter the total number of instances in that direction including the original.

11. Enter **2** in the **Message Input Window** and press ENTER. The rectangular pattern of holes will be displayed as shown in Figure 6-39.

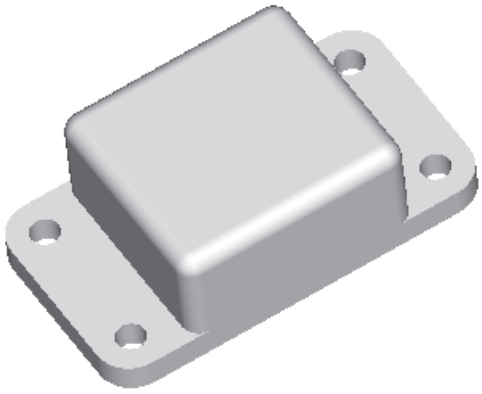


Figure 6-39 *Model after creating the hole pattern*

You can use CTRL+middle mouse button to spin the model and to display the model as shown in Figure 6-39.

Note

If you have multiple holes to be created on a model, it is recommended to create their pattern, if possible. This is because when you assemble bolts in these holes in the Assembly mode, it becomes very easy to assemble them using the reference pattern.

Creating the Rib Feature

The sketch of the rib feature will be created on the **FRONT** datum plane and the given thickness will be applied to it. This is the fifth feature of the model.

1. Choose **Insert > Rib** from the menu bar. You are prompted to specify the sketching plane.

2. Select the **FRONT** datum plane. The **SKET VIEW** submenu is displayed.

3. Select the **Top** option from the **SKET VIEW** submenu and choose the **TOP** datum plane from the graphics screen to proceed to the sketcher environment.

4. Draw the sketch of the rib feature as shown in Figure 6-40.

Note

As evident from Figure 6-40, the top end of the inclined line in the sketch is aligned with the curve and the tangent constraint is applied to the line and the curve. Similarly, the bottom end of the inclined line is also aligned with the two edges. This is the reason there are no dimensions in the sketch and the sketch for the rib feature is fully constrained.

5. Exit the sketcher environment and choose **Flip** from the **DIRECTION** menu. Specify the rib thickness as 8 in the **Message Input Window** to create the rib.

   The rib feature is shown in Figure 6-41. You can use CTRL+middle mouse button to spin the model in the orientation as shown in Figure 6-41.

Figure 6-40 *Fully constrained sketch for the rib*

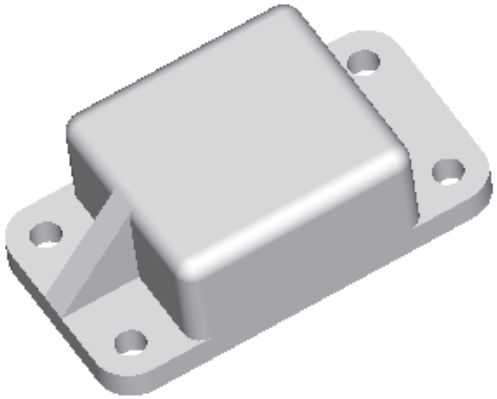


Figure 6-41 *Model after creating the rib feature*

Creating a Copy of the Rib Feature

A copy of the rib feature will be created as shown in Figure 6-42. This copy feature is the sixth feature of the model. The other method to create the same is to create the sketch of the rib feature on a sketching plane. But this method will consume a lot of time. Therefore, the rib feature will be copied about the **RIGHT** datum plane.

1. Choose **PART > Feature > Copy** from the **Menu Manager**. The **COPY FEATURE** submenu is displayed.

2. Choose **Mirror > Select > Independent > Done** from the **COPY FEATURE** submenu. The **SELECT FEAT** submenu is displayed and you are prompted to select the features to be mirrored.

3. Select the rib feature from the graphics screen and choose **Done** from the **SELECT FEAT** submenu. You are prompted to select a plane or create a datum plane to mirror about.

4. Select the **RIGHT** datum plane as the mirror plane. The selected feature is mirrored about the **RIGHT** datum plane as shown in Figure 6-42.

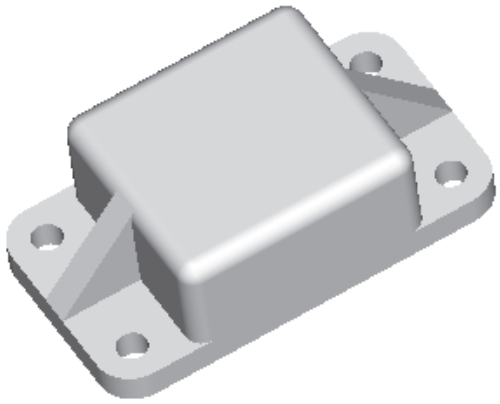


Figure 6-42 *Model after mirroring the rib*

Creating the Protrusion Feature

The seventh feature is an extruded feature that will be created on the top planar surface of the base feature.

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

   You are prompted to select the sketching plane.

2. Choose the top planar surface of the base feature as the sketching plane. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.

3. Select the **Right** option from the **SKET VIEW** submenu and choose the **RIGHT** datum plane.

4. After you enter the sketcher environment, draw the sketch of the extruded feature as shown in Figure 6-43.

   Here in the sketch, you should note that the tangent and equal radii constraints are applied. Also the center of the top arc and the bottom arc coincides with the intersection of the two datum planes.

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

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

6. Select the **Blind** option from this menu and choose **Done**.

7. Enter a depth of **3** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

   The extruded feature is completed and the default trimetric view is shown in Figure 6-44.

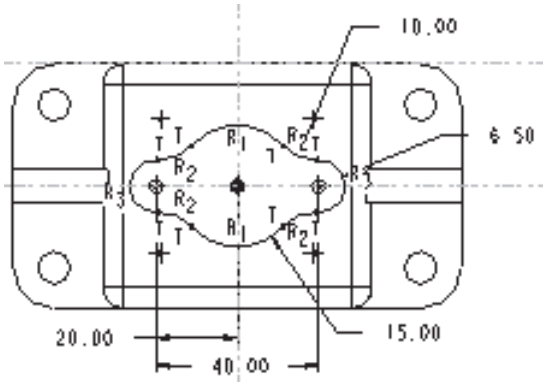


Figure 6-43 *Sketch with dimensions and constraints for the extruded feature*

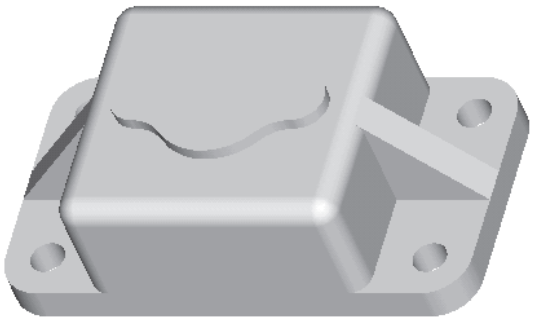


Figure 6-44 *The default trimetric view after creating the extruded feature*

Creating the Cut Feature

You need to create an extruded cut on the bottom planar surface of the base feature and this is the eighth feature of the model.

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

   You are prompted to select a sketching plane.

2. Choose the bottom planar surface of the base feature as the sketching plane. The **DIRECTION** submenu is displayed.

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

4. Select the **Right** option from the **SKET VIEW** submenu and choose the **RIGHT** datum plane from the graphics screen.

5. After you enter the sketcher environment, draw the sketch and dimension it as shown in Figure 6-45.

6. After completing the sketch, choose the **Continue with the current section** button. The **DIRECTION** menu is displayed and you are prompted to specify the direction of material removal shown by the red arrow.

7. Choose **Okay** from the **DIRECTION** menu. The **SPEC TO** menu is displayed and the **Blind** option is selected by default.

8. Choose **Done** from the **SPEC TO** menu. The **Message Input Window** is displayed with a default value.

9. Enter a value of **30** in the **Message Input Window** and press ENTER.

10. Choose **OK** from the **CUT** dialog box. You can spin the model using the CTRL+middle mouse button. The cut feature is shown in Figure 6-46.

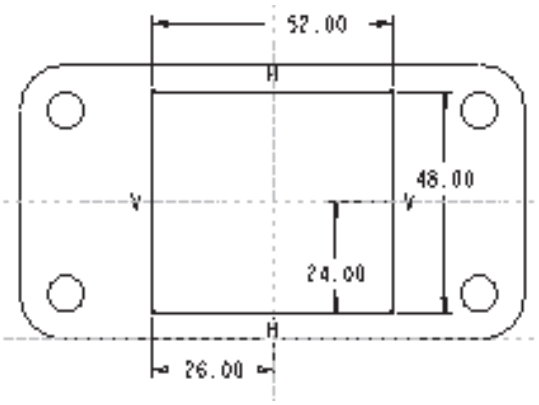


Figure 6-45 *Sketch for the cut feature with dimensions*

Figure 6-46 *Model after creating the cut feature*

Creating the Hole

Next, a through hole will be created on the top extruded feature using the **HOLE** dialog box.

1. Choose **Insert > Hole** from the menu bar. The **HOLE** dialog box is displayed. The **Straight hole** radio button is selected by default. Specify the placement parameters of the hole as given in Figure 6-32 and create the hole. The model after creating the hole is shown in Figure 6-47.

Creating a Copy of the Hole

You need to create a mirror copy of the hole as shown in Figure 6-46 and this will be the tenth feature of the model. The hole feature will be mirrored about the **RIGHT** datum plane.

1. Choose **PART > Feature > Copy** from the **Menu Manager**. The **COPY FEATURE** submenu is displayed.

2. Choose **Mirror > Select > Independent > Done** from the **COPY FEATURE** submenu.

The **SELECT FEAT** submenu is displayed and you are prompted to select the features to be mirrored.

3. Select the previous hole feature from the graphics screen and choose **Done** from the **SELECT FEAT** submenu. You are prompted to select a plane or create a datum plane to mirror about.

4. Select the **RIGHT** datum plane as the mirror plane. The hole feature is mirrored about the **RIGHT** datum plane as shown in Figure 6-48.

Figure 6-47 *Hole feature on the extruded feature*

Figure 6-48 *The copied hole feature*

Note

*The default trimetric view of the model is displayed when you choose the **Default** option from the **Saved view list** button. If you want to change the default view to isometric, then you need 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.*

Saving the Model

You have to save the model because you may need it later.

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 6-49. The feature id numbers displayed in the **Model Tree** may be different when you create the features.

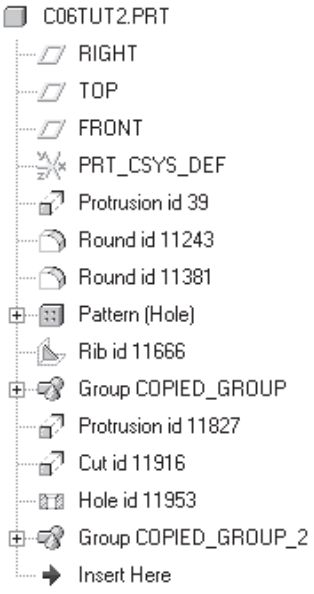


Figure 6-49 ***Model Tree*** *for Tutorial 2*

Tutorial 3

In this tutorial you will create the model shown in Figure 6-50. This figure also shows the top view, front view, and isometric view of the model. **(Expected time: 30 min)**

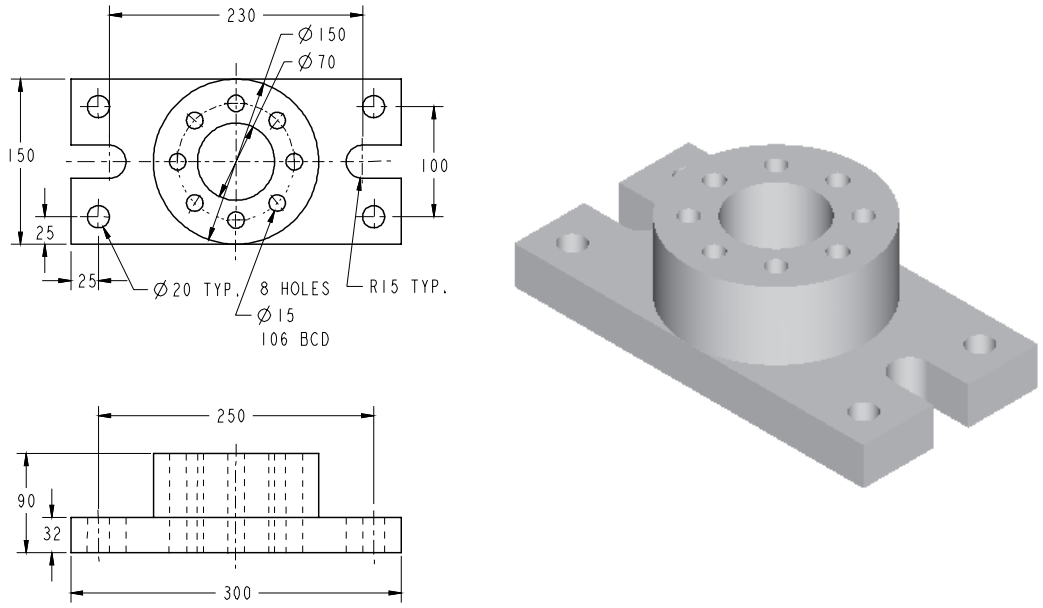


Figure 6-50 *Top, front, and isometric views of the model*

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of five features, see Figure 6-50.

b. The base feature is an extruded feature, see Figure 6-52. First the sketch of the base feature will be created on the **TOP** datum plane, see Figure 6-51, and then it will be extruded to a depth of 32.

c. The second feature is a cylindrical extruded feature, see Figure 6-54. The sketch of the second feature will be created on the top planar surface of the base feature, see Figure 6-53, and then it will be extruded to a depth of 58.

d. The third feature is a hole feature and this hole is coaxial with the cylindrical feature, see Figure 6-55.

e. The fourth feature is a hole feature and will be created on the top planar surface of the base feature, see Figure 6-56. A rectangular pattern of this hole will be created, see Figure 6-57.

f. The fifth feature is also a hole feature and will be created on the top planar surface of the cylindrical feature, see Figure 6-58. A rotational pattern of this hole will be created, see Figure 6-59.

After understanding the procedure for creating the model, you are now ready to draw it. The working directory is already selected in Tutorial 1 and therefore you do not need to select it again. However, if you want to change the working directory, choose **File > Set Working Directory** and then select **c06** in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it **c06tut3**. The three default datum planes and the **Model Tree** are displayed on the graphics screen. Exit the **Model Tree**. The **Model Tree** will not appear if it has been previously turned off.

Creating the Base Feature

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

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

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

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

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

5. Once you enter the sketcher environment, create the sketch of the base feature and apply constraints and dimensions as shown in Figure 6-51.

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

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

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

8. Enter a depth of **32** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

The base feature is completed and now the second feature will be created. The default trimetric view of the base feature is shown in Figure 6-52.

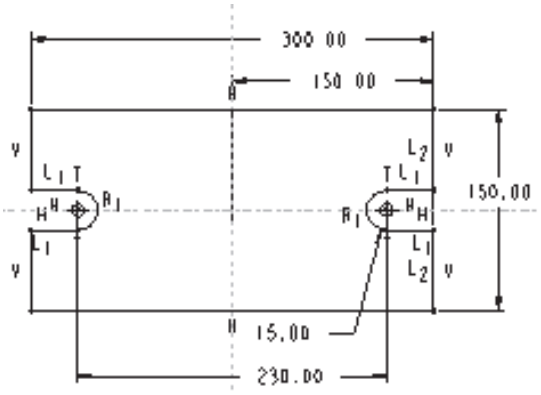


Figure 6-51 *Sketch with dimensions and constraints for the base feature*

Figure 6-52 *Default trimetric view of the base feature*

Creating the Second Feature

The second feature is also an extruded feature and will be created on the top planar surface of the base feature. Therefore, you need to define the top planar surface as the sketching plane for the second feature.

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

2. Select the top planar surface of the base feature as the sketching plane. The **DIRECTION** submenu is displayed.

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

4. Select the **Right** option from the **SKET VIEW** submenu and choose the **RIGHT** datum plane from the graphics screen.

5. Create the sketch for the second feature and dimension it as shown in Figure 6-53.

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

7. 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.

8. Enter a value of **58** in the **Message Input Window** and press ENTER.

9. Choose the **Preview** button from the **PROTRUSION** dialog box and then choose **OK**. The second feature is completed and the default trimetric view is shown in Figure 6-54.

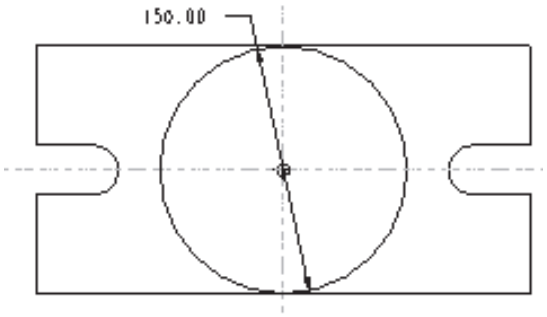


Figure 6-53 Sketch of the cylindrical feature with diameter of the cylinder

Figure 6-54 Default trimetric view of the cylindrical feature

Creating the Third Feature

The third feature is a through hole that is coaxial to the cylindrical feature. The hole feature will be created using the **HOLE** dialog box.

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

2. Create the hole as shown in Figure 6-55 by specifying the placement parameters.

Creating the Fourth Feature

The fourth feature is a through hole and will be created on the top planar surface of the base feature. The hole will be created using the **HOLE** dialog box.

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

   Create the hole as shown in Figure 6-56 by specifying the placement parameters.

Patterning the Hole Feature

You need to create a rectangular pattern of the hole feature that is created on the base feature. You can also create the individual holes, but this will consume a lot of time and the number of features in the **Model Tree** will increase. Hence, you need to create a rectangular pattern of the hole feature.

1. Choose **PART** > **Feature** > **Pattern** from the **Menu Manager**. You are prompted to select a feature to be patterned.

2. Select the hole feature from the graphics screen. The **PAT OPTIONS** submenu is displayed with the **Identical** option selected by default.

Figure 6-55 *Coaxial hole on the cylindrical feature*

Figure 6-56 *Hole on the base feature*

Here, you can use the **Identical** option because the feature on which the pattern is created is not intersecting the pattern.

3. Choose **Done** from the **PAT OPTIONS** submenu. The reference placement dimensions of the hole are displayed on the graphics screen and you are prompted to select pattern dimensions for first increment type. The prompt means that you select the dimension in any one direction where the pattern will be created. The pattern to be created is a rectangular pattern, and hence you can select the dimension in any direction.

4. Select the dimension value **25** from the graphics screen. Since the two dimensions that are displayed on the graphics screen are both 25, hence select the dimension 25 that is along the shorter side of the base feature. After you select the dimension in the first direction, the **Message Input Window** is displayed.

5. Enter a value of **100** in the **Message Input Window** and press ENTER. You are prompted to select another dimension in the same direction or choose **Done**.

6. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and you are prompted to enter the total number of instances in that direction including the original.

7. Enter **2** in the **Message Input Window** and press ENTER. Now, you are prompted to enter dimension increment in the second direction.

8. Select the dimension value **25** that is along the longer side of the base feature. This selected dimension is in the second direction.

9. Enter a value of **250** in the **Message Input Window** and press ENTER. You are prompted to select another dimension in the same direction.

10. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and you are prompted to enter the total number of instances in that direction including the original.

11. Enter **2** in the **Message Input Window** and press ENTER. The rectangular pattern of holes will be displayed as shown in Figure 6-57.

Creating a Hole on the Cylindrical Feature

The hole on the cylindrical feature will created diametrically using the **HOLE** dialog box.

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

2. Enter a value of **15** in the **Diameter** edit box in the **Hole Dimension** area.

3. From the **Depth One** drop-down list, select the **Thru All** option. Now, you are prompted to select a placement plane.

4. Select the top planar surface of the cylindrical feature as the placement plane.

5. From the **Placement Type** drop-down list in the **Hole Placement** area, select the **Diameter** option. You are prompted to select the axis for the hole.

6. Select the axis of the cylindrical feature from the graphics screen. Enter a value of **106** in the **Diameter** edit box in the **Hole Placement** area and press ENTER. You are prompted to select a plane that intersects the placement plane for angular dimension.

7. Select the **FRONT** datum plane from the graphics screen. Enter a value of **90** in the **Angle** edit box.

8. Choose the **Build feature** button from the **HOLE** dialog box. The hole is created as shown in Figure 6-58.

Figure 6-57 *Rectangular pattern of the hole feature*

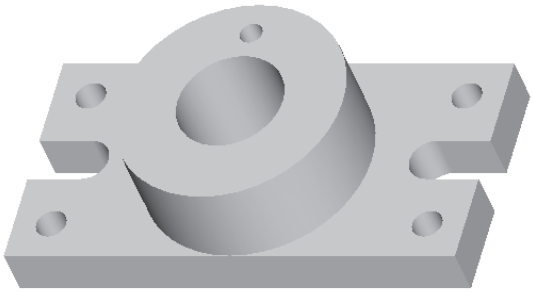


Figure 6-58 *Diametrical hole on the cylindrical feature*

Creating the Rotational Pattern of the Hole Feature

Creating the remaining holes individually on the cylindrical feature will consume a lot of

time. Therefore, you need to create a rotational pattern of the hole feature.

1. Choose **PART > Feature > Pattern** from the **Menu Manager**. You are prompted to select the feature to be patterned.

2. Select the hole feature from the graphics screen. The **PAT OPTIONS** submenu is displayed. The **Identical** option is selected by default. Choose **Done**.

3. The dimensions of the hole feature are displayed on the graphics screen. Select the angular dimension **90**. The **Message Input Window** is displayed.

4. Enter **45** in the **Message Input Window** and press ENTER. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and prompts you to specify the number of instances of the hole feature in the pattern.

5. Enter **8** in the **Message Input Window** and press ENTER. Choose **Done** from the **EXIT** submenu. The rotational pattern is created as shown in Figure 6-59.

Saving the Model

You have to save the model because you may need it later.

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 6-60. The feature id numbers displayed in the **Model Tree** may be different when you create the features.

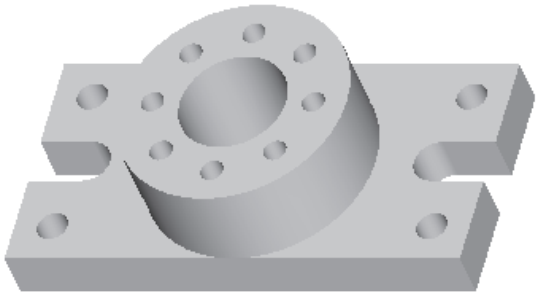


Figure 6-59 *The complete model*

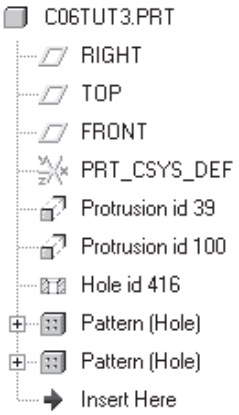


Figure 6-60 Model Tree *for Tutorial 3*

Self-Evaluation Test

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

1. When a pattern is created, the leader or the parent feature also becomes a part of the pattern. (T/F)

2. Once a pattern is created, all the instances in the pattern including the parent feature acts as a single feature. (T/F)

3. In Reference pattern, an existing pattern is referenced to create a new pattern. (T/F)

4. Using the **Pattern** option from the **FEAT** menu you can create only linear patterns. (T/F)

5. If you select a pattern feature from the **Model Tree** and right click to display the shortcut menu and then choose the **Delete** option, the whole pattern is deleted including the leader. (T/F)

6. The _____ option is used to delete a pattern leaving the leader feature.

7. The _____ option is used to mirror an entire geometry about a plane.

8. The _____ option from the **PAT OPTIONS** submenu is used to create a pattern in which the dimensions of the instances can be varied.

9. The first feature in a pattern is called _____.

10. The _____ menu is used to select the dimensions that you want to vary from the leader while copying.

Review Questions

Answer the following questions:

1. Which of the following options in the **PAT OPTIONS** submenu is used to create patterns in which the instances touch each other, and intersect with other instances or the edges of the surface?

   (a) **Identical** (b) **Varying**
   (c) **General** (d) None

2. Which of the following datums is required to create a rotational pattern?

   (a) Graph (b) Curve
   (c) Axis (d) Point

3. Which of the following options in the **PAT OPTIONS** submenu cannot be used to create a pattern that intersects an edge of a feature on which the pattern has to be created?

   (a) **Identical** (b) **Varying**
   (c) **General** (d) None

4. Which of the following options in the **PAT OPTIONS** submenu is used to create a pattern that has all instances of different size?

   (a) **Identical** (b) **Varying**
   (c) **General** (d) None

5. Which of the following options in the **COPY FEATURE** submenu creates a parent-child relationship between the copied feature and the source feature?

   (a) **FromDifModel** (b) **FromDifVers**
   (c) **Independent** (d) **Dependent**

6. You can mirror features using the datum planes or planar surfaces. (T/F)

7. To create a rotational pattern, you should specify an angular increment. (T/F)

8. The option **New Refs** is in the **COPY FEATURE** submenu. (T/F)

9. When you copy a feature using the **New Refs** option, then it is not related with the original feature. (T/F)

10. The features copied from different models or from different versions are always independent. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 6-61. The top view, front view, right-side view, detailed, and sectioned views of the model are shown in Figure 6-62. **(Expected time: 1 Hr)**

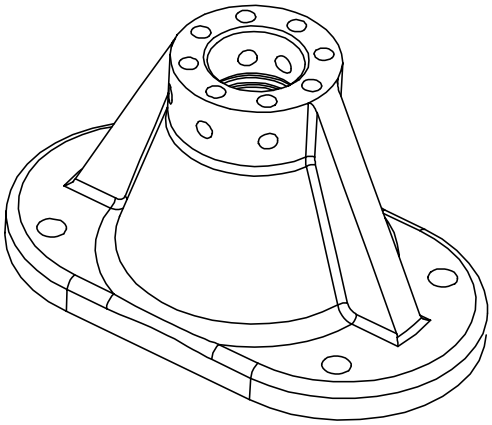


Figure 6-61 *Solid model for Exercise 1*

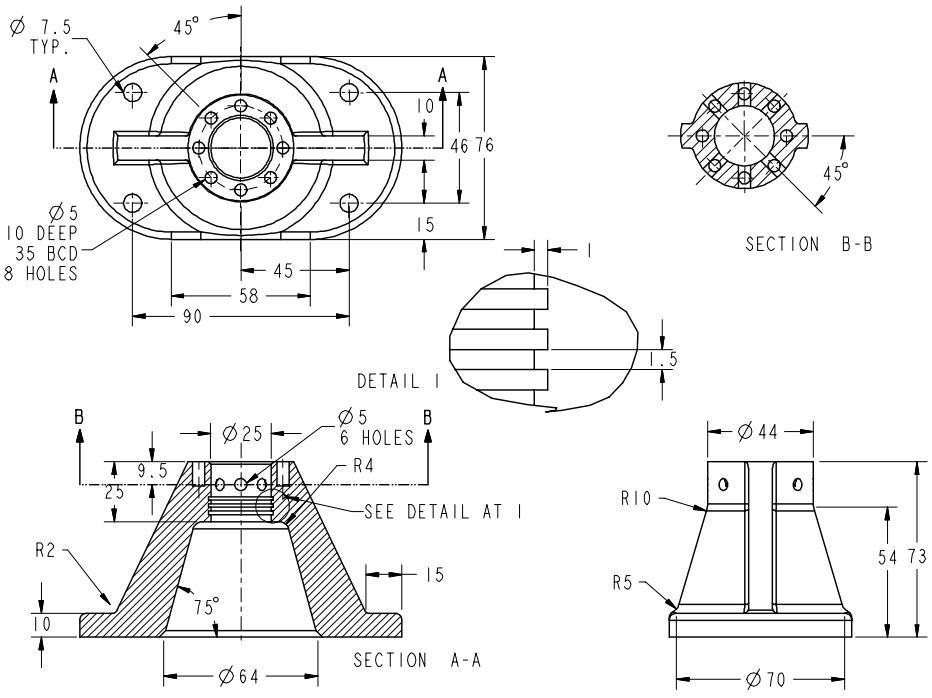


Figure 6-62 *Top, front, right-side, detailed, and sectioned views of the model*

Exercise 2

Create the model shown in Figure 6-63. The figure also shows the top view, front view, and the right-side view are of the model. **(Expected time: 30 min)**

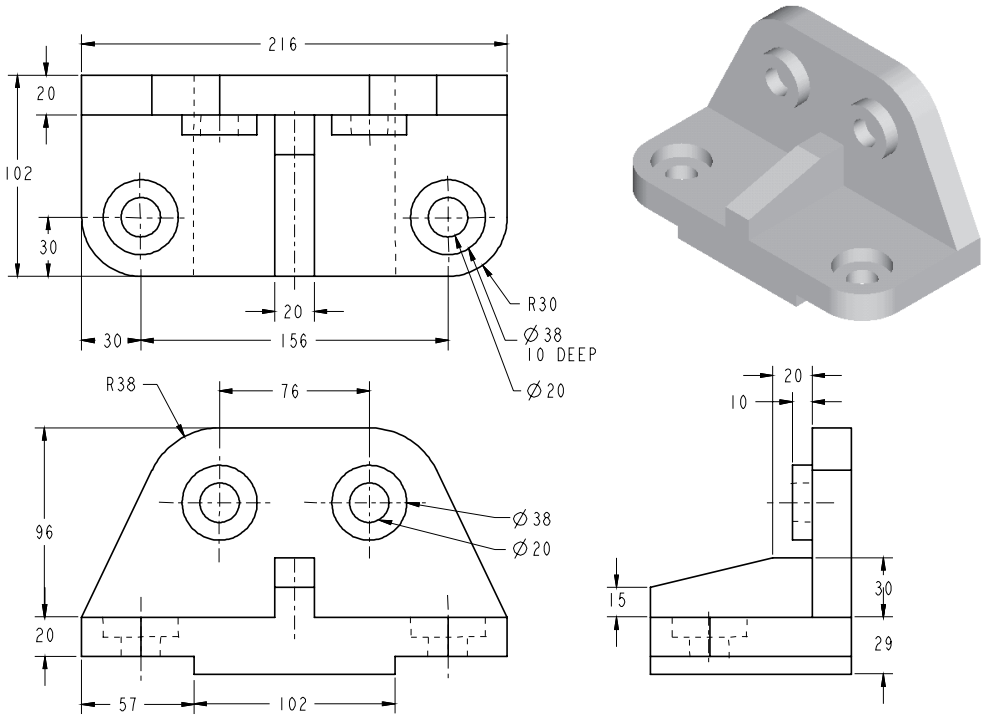


Figure 6-63 *Top, front, right-side, and isometric views of the model*

Exercise 3

Create the model shown in Figure 6-64. The top view, front view, and the isometric view is shown in the figure. **(Expected time: 30 min)**

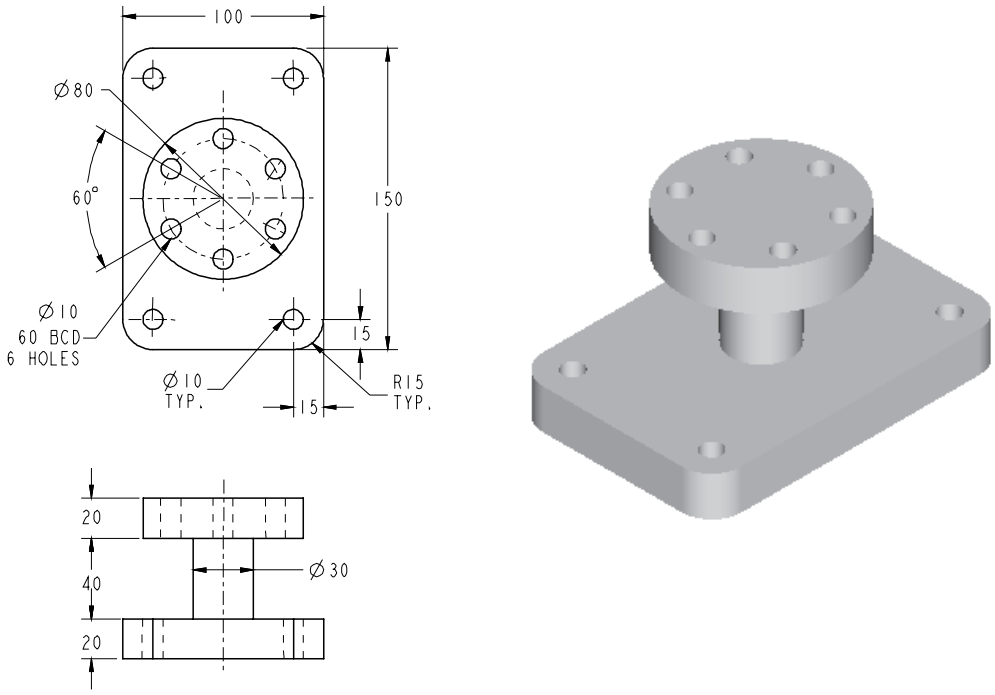


Figure 6-64 *Top view, front view, and the isometric view of the model*

Exercise 4

Create the model shown in Figure 6-65. The front and sectioned right-side views are shown in the Figure 6-66. **(Expected time: 45 min)**

Note

To create the three protrusion features on the outer cylindrical surface, you will have to create a datum on the fly using the ***Make Datum*** *option to orient the sketch view. Also, when you enter the sketcher environment, you need to select the central axis of the model for defining the references using the* ***References*** *dialog box. As you select the central axis, the message in the* ***Reference status*** *area of the* ***References*** *dialog box will be* ***Fully Placed****. This suggests that you do not require any datum plane to define the references. Use this central axis to dimension the sketch. If you do not select the central axis to define the references using the* ***References*** *dialog box, the resultant feature may not get patterned.*

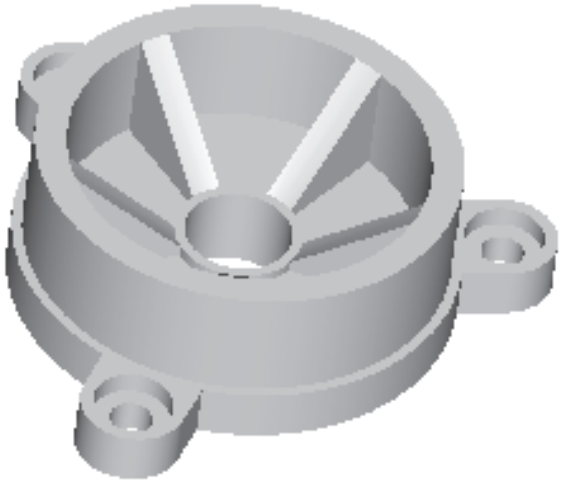


Figure 6-65 *Solid model for Exercise 4*

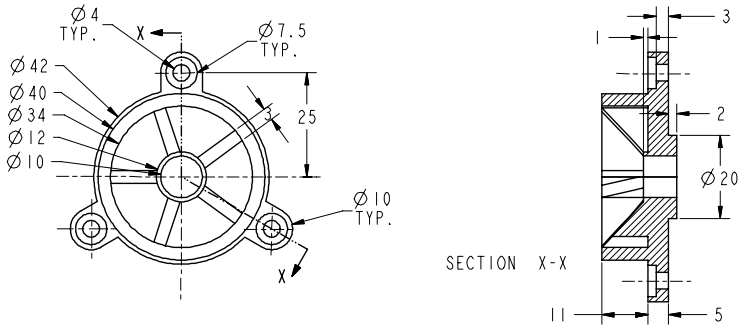


Figure 6-66 *Top view and the total aligned sectioned view of the model*

Note

You will learn about the aligned section views and total aligned section views in Chapter 10.

Answers to the Self-Evaluation Test

1 - T, **2** - T, **3** - T, **4** - F, **5** - T, **6** - **Del Pattern**, **7- Mirror Geom**, **8** - **Varying**, **9** - leader, **10** - **GP VAR DIMS**.