



# **Chapter 8**

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## ***Editing Features and Advanced Modeling Tools-III***

### **Learning Objectives**

**After completing this chapter, you will be able to:**

- *Edit hole features.*
- *Edit the positioning of groove features.*
- *Edit the positioning of slot features.*
- *Edit the parameters of features.*
- *Edit the parameters of features with rollback.*
- *Reorder features.*
- *Create boss features.*
- *Create pocket features.*
- *Create pad features.*
- *Create draft features.*

## EDITING FEATURES

Editing is one of the most important aspects of the product design cycle. Almost all designs require editing during or after their creation. As discussed earlier, NX is a feature-based parametric software. Therefore, the design created in NX is a combination of individual features integrated together to form a solid model. All these features can be edited individually. The following sections explain the editing operations that can be performed in NX:

### Editing a Hole Feature

After creating a hole, you may need to edit its parameters. The parameters that can be edited in NX include the diameters, depth, and the positioning values of the hole. To modify the parameters of a simple hole, double-click on it; the **Edit Parameters** dialog box will be displayed if the hole is created by the **Pre-NX5 Hole** tool, as shown in Figure 8-1. Also, the parameters of the hole will be displayed on the model, as shown in Figure 8-2.

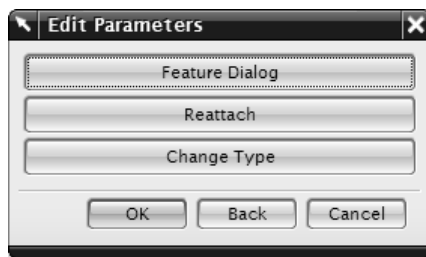


Figure 8-1 The **Edit Parameters** dialog box

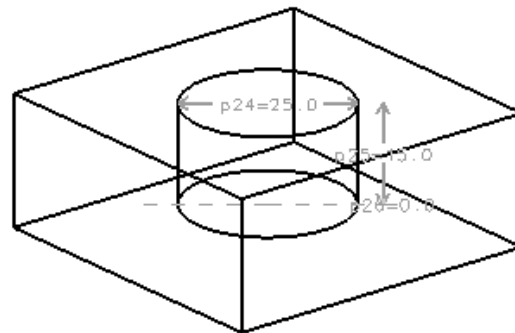


Figure 8-2 Parameters of a simple hole



#### Note

*In case, the hole is created by the **Hole** tool, the **Hole** dialog box will be redisplayed. In this dialog box, you can redefine the parameters of the hole.*

### Feature Dialog Button

To modify the parameters of the hole, choose the **Feature Dialog** button from the **Edit Parameters** dialog box; the **Diameter**, **Depth**, and **Tip Angle** edit boxes will be displayed. If the hole selected is a counterbore or a countersink hole, the counter related values will also be displayed. Enter the new values in the respective edit boxes and choose the **OK** button; the original options of the **Edit Parameters** dialog box will be restored. Choose the **OK** button from this dialog box; the changes made in the parameter values of the hole will be reflected in the model.

### Reattach Button

To change the placement face of the hole, choose the **Reattach** button from the **Edit Parameters** dialog box; the **Reattach** dialog box will be displayed, as shown in Figure 8-3. By default, the **Specify Target Placement Face** button is chosen from the **Selection Steps** area and you will be prompted to select the target face. Select a new placement face to reattach the hole. On doing so, the **Redefine Positioning Dimensions** button will be automatically chosen from the **Selection Steps** area of the **Reattach** dialog box. To retain the same

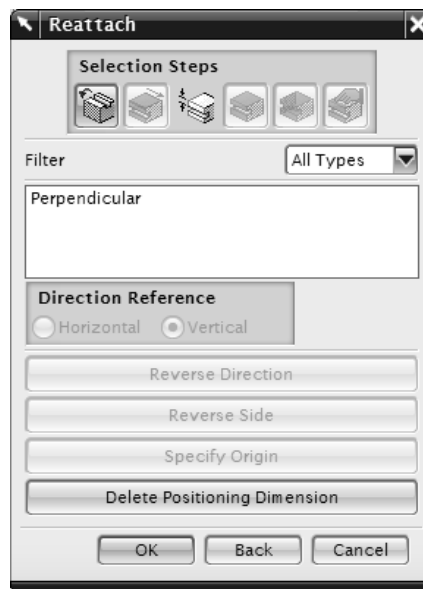


Figure 8-3 The *Reattach* dialog box

positional values for the hole on the new placement face and create it, choose the **OK** button. To specify new positional values for the hole, select the dimension to be modified; a dialog box will be displayed and it will be named based on the type of dimension used to place the hole. Also, you will be prompted to select the target object. Select an edge or datum to define the new reference for the dimension in the new placement face; you will be prompted to select the reference from the tool (hole). Select the bottom circular edge from the hole; the **Select Arc Position** dialog box will be displayed. To define the positional value between the selected edge/datum and the center point of the hole, choose the **Arc Center** button from the **Select Arc Position** dialog box. To define the positional value between the selected edge/datum and any of the quadrant points of the hole, choose the **End Point** button. To define the positional value between the selected edge/datum and the tangent point on the bottom edge of the hole, choose the **Tangent Point** button. After specifying the dimension, choose the **OK** button from the **Select Arc Position** dialog box. Also, choose the **OK** button from the **Reattach** dialog box; the changes will be reflected in the model.



#### Note

You can also delete the hole dimensions using the *Reattach* dialog box. To do so, choose the **Delete Positioning Dimension** button from the *Reattach* dialog box and then select the dimension to be deleted.

## Editing the Positioning of a Hole Feature

NX allows you to edit the positioning of the hole features. To do so, choose the **Part Navigator** button from the resource bar; the **Part Navigator** cascading menu will be displayed. Right-click on the hole feature name and choose the **Edit Positioning** option from the shortcut menu; the **Edit Positioning** dialog box will be displayed, as shown in Figure 8-4. You can use the options in this dialog box to add, edit, or delete a dimension.



Figure 8-4 The *Edit Positioning* dialog box

### Editing the Positioning Dimensions Using the Edit Dimension Value Button

The **Edit Dimension Value** button is used to edit the existing dimension values that are defined to position the hole feature. To do so, choose the **Edit Dimension Value** button from the **Edit Positioning** dialog box; the **Edit Positioning** dialog box will be modified and you will be prompted to select the positioning dimension to edit. Select the dimension that needs to be modified; the **Edit Expression** dialog box will be displayed. Enter the modified value in the edit box and choose the **OK** button; again the **Edit Positioning** dialog box will be displayed and you will be prompted to select the next positioning dimension to edit. Select another dimension that needs to be modified and enter the modified value. Choose the **OK** button twice; the position of the hole feature will be modified.

### Adding the Positioning Dimensions Using the Add Dimension Button

The **Add Dimension** button is used to add a positioning dimensions to an already positioned feature. To add the new positioning dimensions, choose the **Add Dimension** button; the **Positioning** dialog box will be displayed. The tools in this dialog box can be used to create a new positioning dimensions, as discussed in the previous chapter.

### Deleting the Positioning Dimensions Using the Delete Dimension Button

The **Delete Dimension** button is used to delete the positioning dimensions. To delete the positioning dimensions, choose the **Delete Dimension** button; the **Remove Positioning** dialog box will be displayed, and you will be prompted to select the positioning dimension to be deleted. Select the positioning dimensions to be deleted and choose the **OK** button; the selected dimensions will be deleted.

### Editing the Positioning of a Groove Feature

As mentioned earlier, you can edit the positioning of a groove feature. To do so, choose the **Part Navigator** button from the **Resource Bar** to invoke the **Part Navigator** cascading menu. Right-click on the groove feature displayed in the **Part Navigator** and choose the **Edit Positioning** option from the shortcut menu; the **Edit Positioning** dialog box will be displayed. Also, the edges selected for applying the positioning dimension will be displayed in the dashed line format. Next, choose the **Edit Dimension Value** button from the dialog box; the **Edit Expression** dialog box will be displayed. Enter the new positioning value in the edit box and choose the **OK** button; the **Edit Positioning** dialog box will be redisplayed. Choose the **OK** button from this dialog box to reflect the changes made in the positioning value.

## Editing the Positioning of a Slot Feature

To edit the positioning of a slot feature, invoke the **Part Navigator** cascading menu. Right-click on the slot feature displayed on the tree and choose the **Edit Positioning** option from the shortcut menu; the **Edit Positioning** dialog box will be displayed. Next, choose the **Edit Dimension Value** button from the dialog box; the **Edit Expression** dialog box will be displayed. Enter the new positioning value in the edit box and choose the **OK** button; the **Edit Positioning** dialog box will be redisplayed. Choose the **OK** button from this dialog box to reflect the changes made in the positioning value.

## Editing the Parameters of Features

Similar to editing the parameters of holes, NX also allows you to edit parameters of other features such as extruded features, revolved features, and so on. To edit parameters of these features, right-click on the feature name in the **Part Navigator** and choose **Edit Parameters** from the shortcut menu. Depending on the feature selected, the related dialog box will be displayed. You can select the options from the dialog box to modify the parameters of the selected feature.

## Editing the Parameters of Features with Rollback

In NX, you can edit features such as the extrude features, edge blend features, face blend features, and so on with the rollback. This editing operation is similar to editing parameters, except that this option temporarily suppresses all the features created after the feature to be edited. To do so, right click on the required feature and select the **Edit with Rollback** option from the shortcut menu displayed; a dialog box corresponding to the selected feature will be displayed in which you can modify the parameters. Once the editing is complete, the suppressed features are automatically restored.

## Editing Sketches of the Sketch-based Features

NX also allows you to edit the sketches of the sketch-based features. To do so, right-click on the sketch in the **Part Navigator** and choose **Edit Sketch** from the shortcut menu displayed; the sketching environment will be invoked and the model will be oriented such that the selected sketch is normal to the view. Also, all dimensions of the sketch will be displayed. You can modify the dimensions of the sketch or remove the existing entities and add new entities to the sketch. However, you need to make sure that the sketch is closed after adding or removing the sketched entities.

## Reordering Features

This option allows you to change the order of the features, in which they are created. The feature can be reordered before or after the specified reference feature. To reorder a feature, right-click on the feature name in the **Part Navigator** and choose **Reorder Before** or **Reorder After** from the shortcut menu; the cascading menu containing the names of the reference features will be displayed. Select the feature after or before which you need to reorder a selected feature.

## ADVANCED MODELING TOOLS

As discussed in the previous chapter, the advanced modeling tools are mostly used to place different types of standard and user-defined features on the model. Each advanced modeling tool has its specific use in designing a real-world component. These advanced feature tools reduce the time taken in creating a design.

### Creating Boss Features

**Toolbar:** Feature > Boss (*Customize to add*)



The **Boss** tool is used to add the material to the model in a circular cross-section that has been defined by the user. The boss feature can be placed on a planar surface or a datum plane. Note that the boss feature cannot be the first feature of the model. This is because you need to associate it to an existing target body.

The boss feature is a placed feature and so, it does not require a sketch. You need to specify the diameter, height, and taper angle. To create the boss feature, choose the **Boss** button from the **Feature** toolbar; the **Boss** dialog box will be displayed, as shown in Figure 8-5, and you will be prompted to select the planar placement face.



Figure 8-5 The **Boss** dialog box

Select the planar face or the datum plane; the preview of the boss will be displayed on the screen. Enter the values of the diameter, height, and taper, if required, in the respective edit boxes and choose the **OK** button; the **Positioning** dialog box will be displayed, as shown in Figure 8-6. Also, you will be prompted to select the positioning method. Using the buttons in this dialog box, you can position the boss about the placement face. Figure 8-7 shows a boss feature of diameter 50, height 30, and taper angle 2 degree.

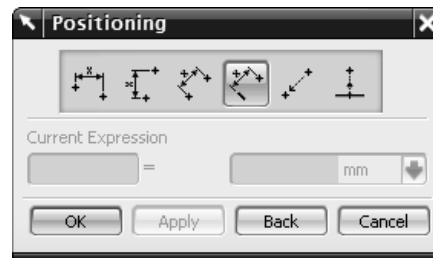


Figure 8-6 The **Positioning** dialog box

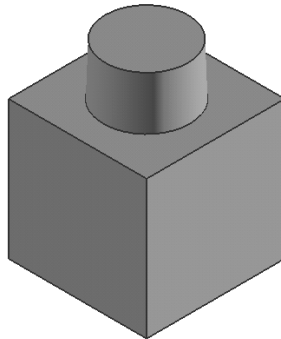


Figure 8-7 The resulting boss feature

**Note**

If you try to create the boss feature as the first feature, after selecting the placement plane, you will be prompted to select a target body.

## Creating Pocket Features

**Toolbar:** Feature > Pocket (Customize to add)



The **Pocket** tool is used to remove the material from the model in the cylindrical or rectangular cross-section. To create a pocket, choose the **Pocket** button from the **Feature** toolbar; the **Pocket** dialog box will be displayed, as shown in Figure 8-8. In NX, you can create three types of pockets, cylindrical, rectangular, and general. The procedure to create different types of pockets is discussed next.



Figure 8-8 The Pocket dialog box

## Creating Cylindrical Pockets

The cylindrical pocket has a circular cutout of a specific depth. The bottom edge of the pocket feature can be blended using the floor radius. You can also define a taper angle for the pocket. To create the cylindrical pocket, choose the **Pocket** button from the **Feature** toolbar; the **Pocket** dialog box will be displayed. Choose the **Cylindrical** button from the **Pocket** dialog box; the **Cylindrical Pocket** dialog box will be displayed and you will be prompted to select a planar placement face. Select a face or a datum plane to specify the placement plane of the pocket feature. On doing so, the **Cylindrical Pocket** dialog box will display additional options, as shown in Figure 8-9, and you will be prompted to enter the pocket parameters.

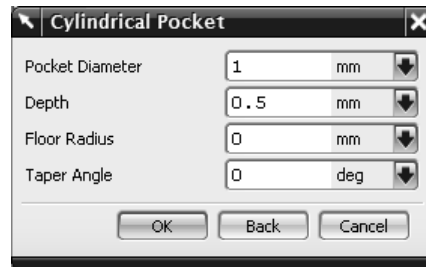


Figure 8-9 The Cylindrical Pocket dialog box

Enter the values of the diameter, depth, floor radius, and taper angle in the **Cylindrical Pocket** dialog box and choose the **OK** button; the **Positioning** dialog box will be displayed, along with the preview of the pocket. You can position the cylindrical pocket on the placement face using the options in this dialog box. Figure 8-10 shows a cylindrical pocket feature of diameter 40, depth 40, floor radius 10, and taper angle 5 degree.

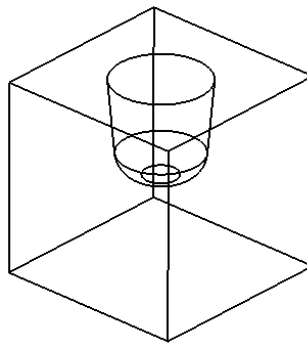


Figure 8-10 A cylindrical pocket feature

### Creating Rectangular Pockets

This pocket type has a rectangular cutout of a specific depth, length, width, with or without radii in the corners and on the floors, and with or without straight or tapered sides. To create a rectangular pocket, choose the **Pocket** button from the **Feature** toolbar; the **Pocket** dialog box will be displayed. Choose the **Rectangular** button from the **Pocket** dialog box; the **Rectangular Pocket** dialog box will be displayed, as shown in Figure 8-11, and you will be prompted to select a planar placement face.

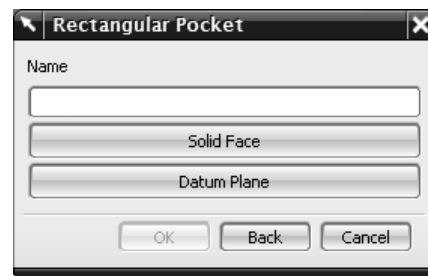
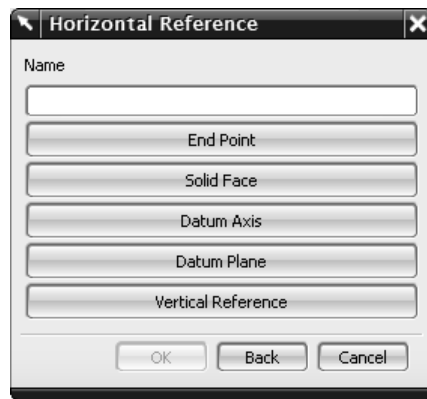


Figure 8-11 The Rectangular Pocket dialog box

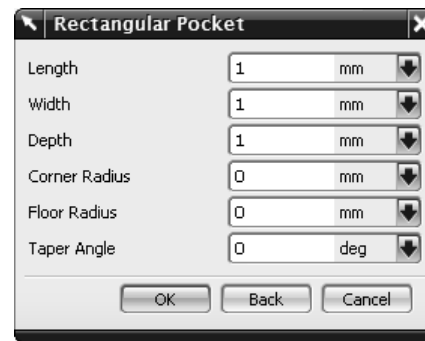
Select a face or datum plane to start creating the pocket feature; the **Horizontal Reference** dialog box will be displayed, as shown in Figure 8-12, and you will be prompted to select a horizontal reference. The length of the pocket will be parallel to the horizontal reference. You can select a linear edge to specify the horizontal reference. Alternatively, you can use the options in the **Horizontal Reference**



dialog box to select the horizontal reference. On selecting the horizontal reference, the **Rectangular Pocket** dialog box will be displayed, as shown in Figure 8-13, and you will be prompted to enter the pocket parameters. Enter the values of the length, width, depth, corner radius, floor radius, and taper angle in the **Rectangular Pocket** dialog box. Note that the corner radius must be greater or equal to the floor radius.

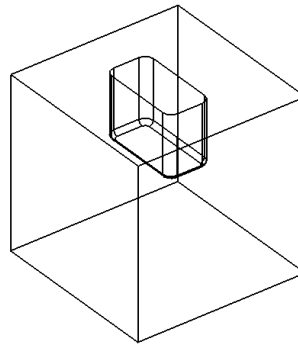


**Figure 8-12** The *Horizontal Reference* dialog box



**Figure 8-13** The *Rectangular Pocket* dialog box

After entering the values in the edit boxes, choose the **OK** button; the **Positioning** dialog box will be displayed, along with the preview of the pocket. Using the buttons in this dialog box, you can position the rectangular pocket about the placement face. Figure 8-14 shows a rectangular pocket feature of length 50, width 30, depth 35, corner radius 5, floor radius 3, and taper angle 2 degree.



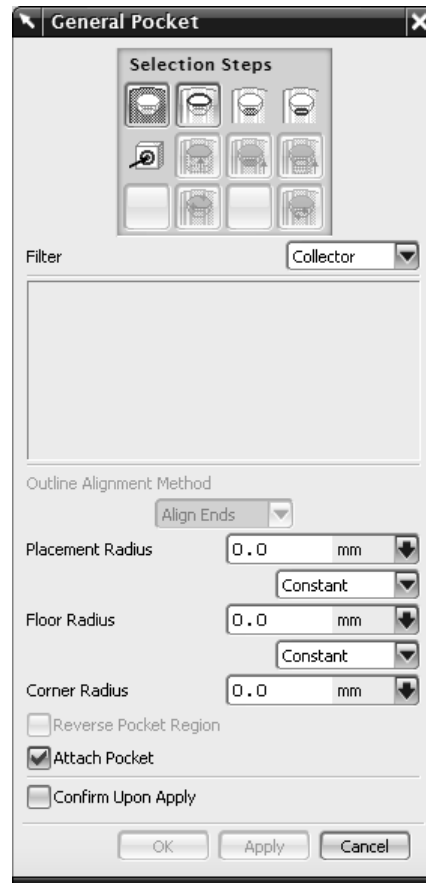
**Figure 8-14** A rectangular pocket feature

### Creating General Pockets

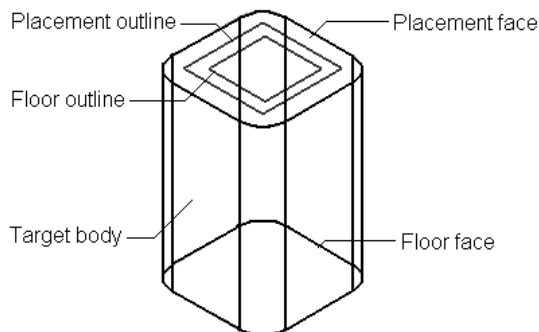
The **General** pocket tool allows you to create pocket features with more flexibility than the **Cylindrical** and **Rectangular** pocket options. In case of a general pocket, the placement face can be non-planar. Before invoking this tool, you need to draw the sketch of the top and bottom faces of the pocket feature. You do not need to draw the sketches on the faces where you want to place the feature. You can draw both the sketches on the same plane also. To create a general pocket, choose the **Pocket** button from the **Feature** toolbar; the **Pocket**

dialog box will be displayed. Choose the **General** button from the **Pocket** dialog box; the **General Pocket** dialog box will be displayed, as shown in Figure 8-15.

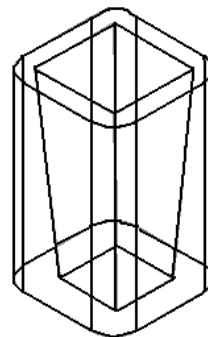
By default, the **Placement Face** button is chosen from the **Selection Steps** area and you will be prompted to select the placement faces of the pocket. Select the top face of the model as the placement face, as shown in Figure 8-16. Choose the **Placement Outline** button from the **Selection Steps** area; you will be prompted to select the placement outline curves. Select the sketch or curves to define the outer boundary of the pocket at the top face, as shown in Figure 8-16. Next, choose the **Floor face** button from the **Selection Steps** area; you will be prompted to select the floor faces of the pocket. Select the plane or face on which the bottom face of the pocket will be placed, as shown in Figure 8-16. Choose the **Floor outline** button from the **Selection Steps** area; you will be prompted to select the floor outline curves. Select the sketch or curves to define the bottom face of the pocket, as shown in Figure 8-16. Choose the **Target body** button from the **Selection Steps** area; you will be prompted to select the optional target body. Select the solid body to create the pocket feature, as shown in Figure 8-16. Enter the values of the placement radius, floor radius, and corner radius in the respective edit boxes. The placement radius is the radius between the placement face and the sides of the pocket. The floor radius is the radius between the floor face and the sides of the pocket. The corner radius is the radius placed on the corners. Choose the **Apply** button and then the **Cancel** button from the **General Pocket** dialog box. Figure 8-17 shows a general pocket feature created using the selections made in Figure 8-16.



**Figure 8-15** The *General Pocket* dialog box



**Figure 8-16** Objects to be selected



**Figure 8-17** The general pocket feature

## Creating Pad Features

**Menu:** Insert > Design Feature > Pad  
**Toolbar:** Feature > Pad (*Customize to add*)



The **Pad** tool is used to add material to the model in the rectangular or user-defined cross-sections. To create pad features, choose the **Pad** button from the **Feature** toolbar; the **Pad** dialog box will be displayed, as shown in Figure 8-18.



Figure 8-18 The **Pad** dialog box

In NX, you can create two types of pad features, rectangular and general. The procedure to create different types of pads is the same as for the pocket features and is discussed next.

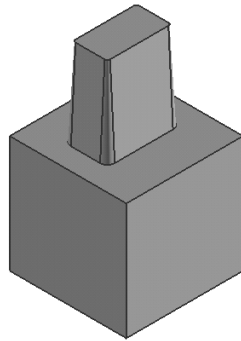
### Creating Rectangular Pads

This type of pad has a rectangular cross-section of a specific length, width, and height. You can also specify the corner radius and the taper angle for the pad feature. To create rectangular pad, choose the **Pad** button from the **Feature** toolbar; the **Pad** dialog box will be displayed. Choose the **Rectangular** button; the **Rectangular Pad** dialog box will be displayed and you will be prompted to select a planar placement face. You can select the datum plane or planar face as the placement plane. Select the face or datum plane to start the pad feature; the **Horizontal Reference** dialog box will be displayed and you will be prompted to select the horizontal reference. The length of the pad will be parallel to the horizontal reference. The options in the **Horizontal Reference** dialog box can be used to select the horizontal reference. You can also directly select an edge of the model to define the horizontal reference. On doing so, the **Rectangular Pad** dialog box will be displayed, as shown in Figure 8-19, and you will be prompted to enter the pad parameters.

You need to enter the values of the length, width, height, corner radius, and taper angle in their respective edit boxes. After entering the values in the edit boxes, choose the **OK** button; the **Positioning** dialog box will be displayed, along with the preview of the rectangular pad. Using the options in this dialog box, you can position the rectangular pad about the placement face. Figure 8-20 shows the rectangular pad feature of length 50, width 30, height 60, corner radius 5, and taper angle 3 degree.



Figure 8-19 The **Rectangular Pad** dialog box



**Figure 8-20** The resulting rectangular pad feature

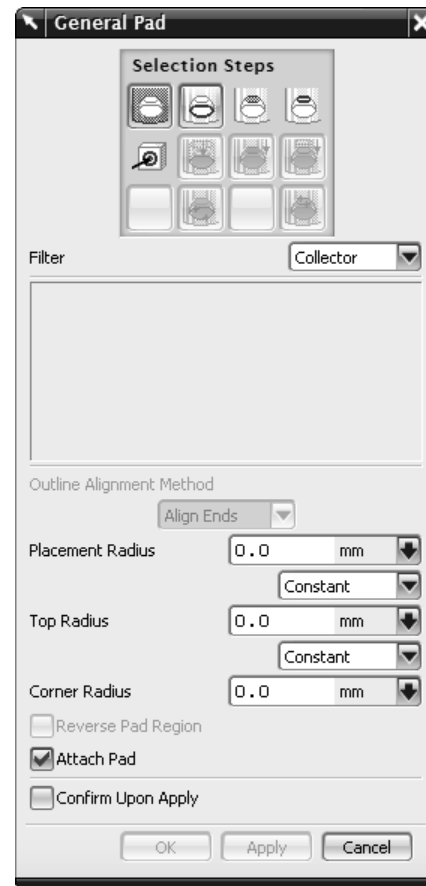


#### Note

While creating a feature, you can choose **Analysis > Distance** from the menu bar to measure the distance between any two entities.

### Creating General Pads

Using the **General** pad option, you can add the material with a greater flexibility than with the **Rectangular** pad option. But in this case, the placement faces can be nonplanar. Similar to creating general pocket features, you can draw the sketches of the general pad feature on a single plane. To create the general pad, choose the **Pad** button from the **Feature** toolbar; the **Pad** dialog box will be displayed. Choose the **General** button from the **Pad** dialog box; the **General Pad** dialog box will be displayed, as shown in Figure 8-21. By default, the **Placement Face** button from the **Selection Steps** area will be chosen and you will be prompted to select the placement face of the pad. Select a face of the model as the placement face, as shown in Figure 8-22. Choose the **Placement Outline** button from the **Selection Steps** area; you will be prompted to select the placement outline curves. Select the sketch or curves that define the shape of the general pad at the top face, as shown in Figure 8-22. Choose the **Top face** button from the **Selection Steps** area; you will be prompted to select the top faces. Select the top face of the model, as shown in Figure 8-22. Choose the **Top Outline** button from the **Selection Steps** area; you will be prompted to select the top outline curves. Select the sketch or curves that will define the shape of the pad feature at the bottom, as shown in



**Figure 8-21** The **General Pad** dialog box

Figure 8-22. Choose the **Target body** button from the **Selection Steps** area; you will be prompted to select the optional target body. Now, select the model to create the pad feature, as shown in Figure 8-22. Then, choose **Apply** and then the **Cancel** button from the **General Pad** dialog box. The resultant general pad feature is shown in Figure 8-23.

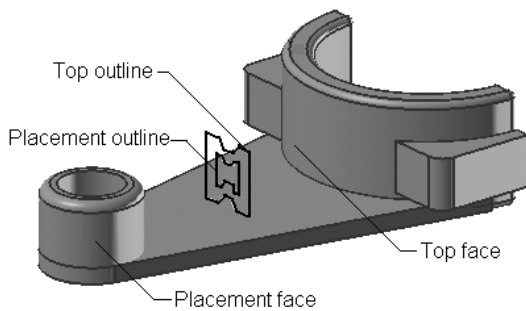


Figure 8-22 Objects to be selected

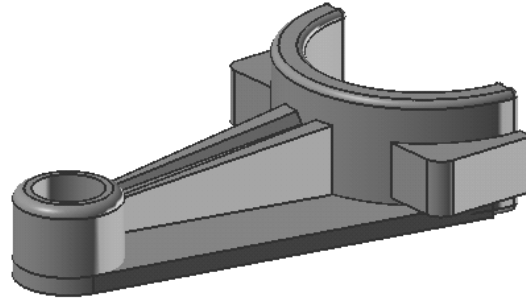


Figure 8-23 The resulting general pad feature



#### Note

The outline curves drawn should intersect the placement and top faces.

## Creating Drafts

**Menu:** Insert > Detail Feature > Draft  
**Toolbar:** Feature Operation > Draft



This tool is used to create a draft on an existing model. To create the draft, choose the **Draft** button from the **Feature Operation** toolbar; the **Draft** dialog box will be displayed, as shown in Figure 8-24. In NX, you can create four types of drafts: draft from plane, draft from edges, draft tangent to faces, and draft to parting edges. The drop-down list in the **Type** rollout contains all the four options to create the draft. The procedure for creating different types of drafts is discussed next.

### Creating the Draft Using the From Plane Option

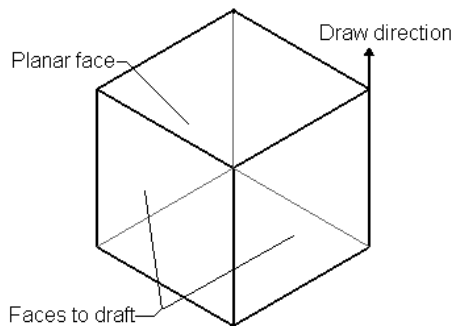
This draft type is used to create a draft by selecting the stationary plane and faces of the model. To create the draft using this option, choose the **Draft** button from the **Feature Operation** toolbar; the **Draft** dialog box will be displayed.

By default, the **From Plane** option will be selected from the drop-down list in the **Type** rollout and the **Inferred Vector** button is chosen from the **Draw**

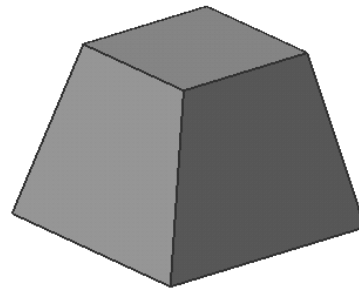


Figure 8-24 The Draft dialog box

**Direction** rollout. As a result, you will be prompted to specify the draw direction. Select the edge for the draw direction, as shown in Figure 8-25. Alternatively, select the draw direction using the **Inferred Vector** drop-down list. You can flip the draw direction by choosing the **Reverse Direction** button from the **Draw Direction** rollout. Next, you will be prompted to select the planar face. Select the face of the model, as shown in Figure 8-25. Next, you will be prompted to select the faces to draft; select the faces, as shown in Figure 8-25. Enter the angle value in the **Angle 1** edit box. Choose the **OK** button to create the draft. The resulting model, after creating the draft feature, is shown in Figure 8-26.



**Figure 8-25** Objects to be selected



**Figure 8-26** The draft created using the stationary plane

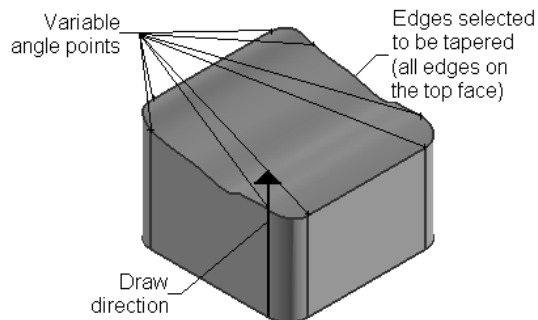
### Creating the Draft Using the From Edges Option

This option is used to create a draft by selecting the edges of the model. In such cases, you need to specify the variable points and angles. This option is useful when the edges of the faces to be tapered are non linear. To create this type of draft, select the **From Edges** option from the drop-down list in the **Type** rollout. You will be prompted to specify the draw direction. Select the edge of the model to specify the draw direction, as shown in Figure 8-27. Alternatively, you can use the options in the **Inferred Vector** drop-down list to specify the draw direction. You can flip the draw direction by choosing the **Reverse Direction** button from the **Draw Direction** rollout. Next, you will be prompted to select the stationary edges; select the edges, as shown in Figure 8-27.

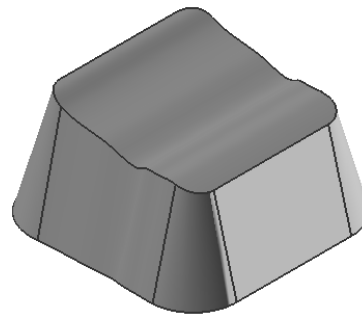
Now, you need to select the points on the stationary edges to specify different draft angles. You can enter different angles for different selected points. To do so, expand the **Variable Draft Points** rollout and choose the **Inferred Point** button; you will be prompted to select the points. You can select any number of points on the edges to specify different angle values. Next, select the variable angle points, as shown in Figure 8-27. Enter different angle values in the **Pt A** edit boxes, displayed after selecting the points. Choose the **OK** button to create the draft. The model, after creating the draft feature, is shown in Figure 8-28.

### Creating the Draft Using the Tangent to Faces Option

This option is used to create a draft that is tangent to the selected faces. In such cases, you need to specify the draw direction and the tangent face. To create this type of draft, select the **Tangent to Faces** option from the drop-down list in the **Type** rollout; you will be prompted to specify the draw direction. Select the edge of the model, as shown in Figure 8-29. Alternatively, you can use the options in the **Inferred Vector** drop-down list to select the

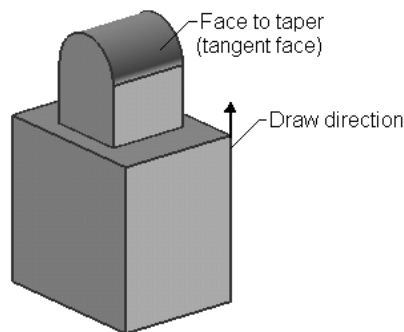


**Figure 8-27** Objects to be selected

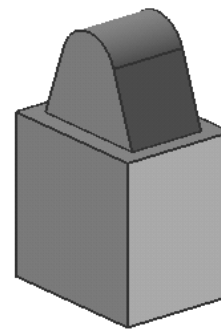


**Figure 8-28** The draft created using the stationary edges

draw direction. Next, you will be prompted to select the tangent faces; select the face, as shown in Figure 8-29. Enter the angle values in the **Angle 1** edit box. Choose the **Apply** button and then the **Cancel** button to create the draft. The resulting model, after creating the taper feature, is shown in Figure 8-30.



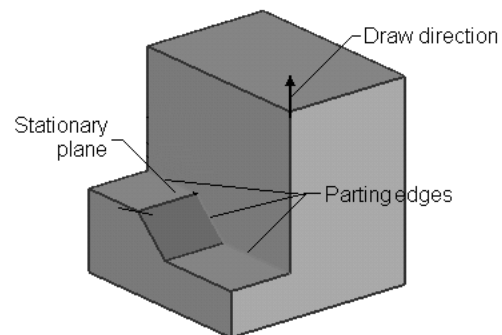
**Figure 8-29** Objects to be selected



**Figure 8-30** The draft created using the tangent faces

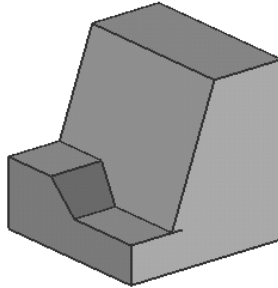
### Creating the Draft Using the To Parting Edges Option

This draft type is used to create the draft along the selected set of edges by specifying the angle, draw direction, stationary plane, and the parting edges. To create this type of draft, select the **To Parting Edges** option from the drop-down list in the **Type** rollout; you will be prompted to specify the draw direction. Select the edge of the model, as shown in Figure 8-31. You can also use the options in the **Inferred Vector** drop-down list to specify the draw direction. Then, you will be prompted to select the planar face; select it, as shown in Figure 8-31. Next, you will be prompted to select the parting edges; select the parting edges, as shown in Figure 8-31. Enter the angle



**Figure 8-31** Objects to be selected

values in the **Angle 1** edit box. Choose the **Apply** button and then the **Cancel** button to create the draft. The resulting model, after creating the draft feature, is shown in Figure 8-32.



**Figure 8-32** The resulting draft feature created using the parting edges



**Note**

The **Draft All Instances** check box in the **Settings** rollout allows you to specify whether to draft only the specified instance or all the instances in the array.

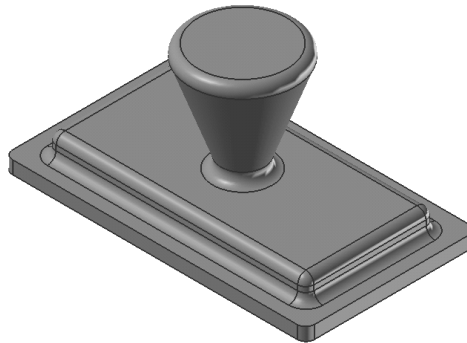
## TUTORIALS

### Tutorial 1

In this tutorial, you will create the model shown in Figure 8-33. The dimensions of this model are given in Figure 8-34. After creating the solid model, save it with the name

|NX 6|c08|c08tut1.prt

(Expected time: 30 min)



**Figure 8-33** Solid model for Tutorial 1

It is recommended that you first outline the steps to create this model. The following steps are required to complete this tutorial:

- Draw the sketch for the base feature and extrude it.
- Create the rectangular pad using the **Pad** tool.
- Create the boss feature using the **Boss** tool.
- Create the draft on the boss feature.
- Create the edge blend on the rectangular pad and the boss feature.
- Save the file.



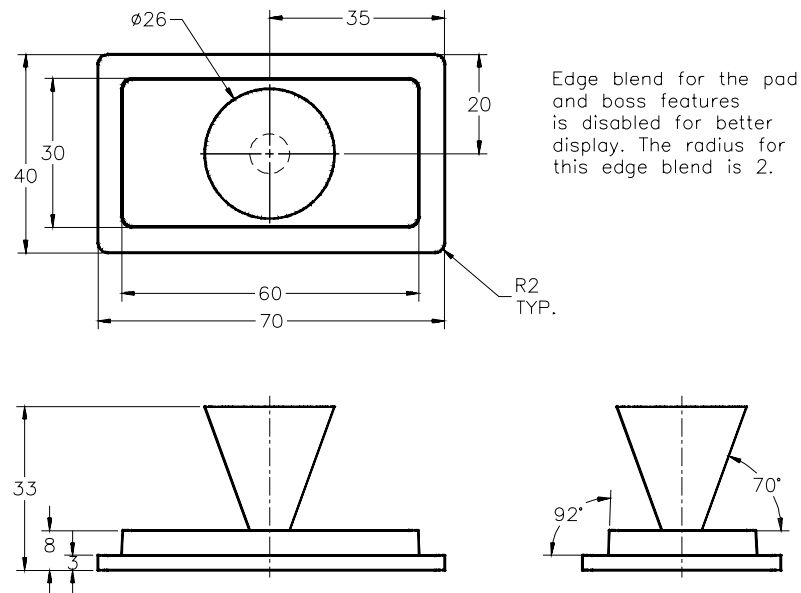


Figure 8-34 Views and dimensions for Tutorial 1

### Creating the Base Feature of the Model

1. Start a new file using the Model template, save the file in the *C:\NX 6\c08* folder with the name *c08tut1.prt*. Create three fixed datum planes.
2. Select the XC-YC plane as the sketching plane and create the sketch for the base feature, as shown in Figure 8-35.
3. Exit the **Sketcher** environment and extrude the sketch with **0** and **3** values in the **Start** and **End** edit boxes, respectively. Turn off the display of the sketch and the datum planes. The resulting base feature is shown in Figure 8-36.

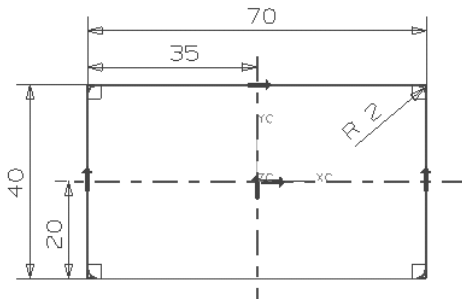


Figure 8-35 Sketch for the base feature

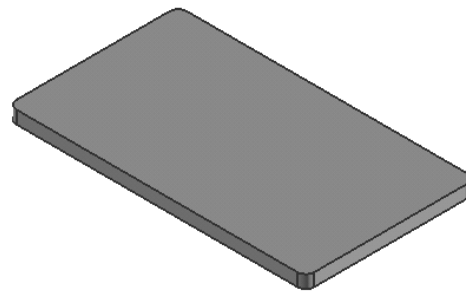


Figure 8-36 Base feature of the model

### Creating the Rectangular Pad Feature

1. Choose the **Pad** tool from the **Feature** toolbar; the **Pad** dialog box is displayed and you are prompted to select the pad type.



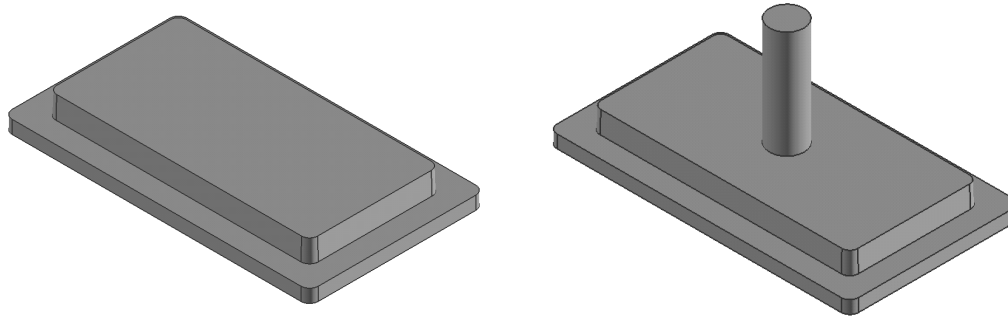
2. Choose the **Rectangular** button; the **Rectangular Pad** dialog box is displayed and you are prompted to select the planar placement face.
3. Select the top face of the base feature as the placement face; the **Horizontal Reference** dialog box is displayed and you are prompted to select the horizontal reference.
4. Select the edge of the base feature that measures 70; the **Rectangular Pad** dialog box is displayed. Enter the values **60, 30, 5, 2, and 2** in the **Length, Width, Height, Corner Radius, and Taper Angle** edit boxes, respectively.
5. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pad. Next, you need to position the pad on the base feature.
6. Choose the **Perpendicular** button from the **Positioning** dialog box and select the 40 mm dimension edge of the base feature and 30 mm dimension edge of the pad. Enter the value **5** in the **Create Expression** dialog box.
7. Choose the **OK** button; the **Positioning** dialog box is displayed. Choose the **Perpendicular** button from the **Positioning** dialog box. Next, select the edge of dimension 70 mm of the base feature and edge of dimension 60 mm of the pad and enter **5** as the value in the **Create Expression** dialog box.
8. Choose the **OK** button twice and the **Cancel** button once. The resulting rectangular pad feature is shown in Figure 8-37.

### Creating the Boss Feature

The third feature is a boss feature and will be created using the following steps:

1. Choose the **Boss** tool from the **Feature** toolbar; the **Boss** dialog box is displayed and you are prompted to select the planar placement face.
2. Select the top face of the second feature as the placement face; the preview of the boss is displayed. Enter the values **8, 25, and 0** in the **Diameter, Height, and Taper Angle** edit boxes, respectively.
3. Choose the **OK** button; the **Positioning** dialog box is displayed. Next, you need to position the boss on the second feature.
4. Choose the **Perpendicular** button from the **Positioning** dialog box and select the 30 mm dimension edge of the second feature. Next, enter **15** as the value in the **Current Expression** edit box.
5. Choose the **Perpendicular** button from the **Positioning** dialog box and select the 60 mm dimension edge of the second feature. Enter **30** as the value in the **Current Expression** edit box and choose the **OK** button. The boss feature created is shown in Figure 8-38.



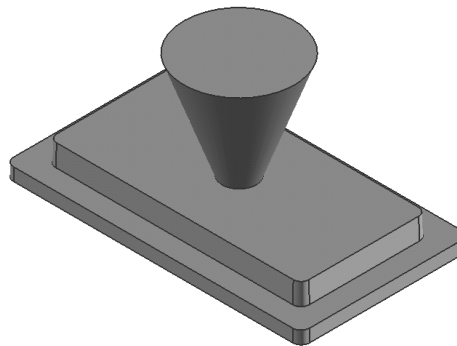


**Figure 8-37** Model after creating the pad feature    **Figure 8-38** Model after creating the boss feature

### Creating the Draft Feature

Next, you need to create a draft on the boss feature. Use the following steps to create the draft feature.

1. Choose the **Draft** tool from the **Feature Operation** toolbar; the **Draft** dialog box is displayed. By default, the **From Plane** option is selected from the drop-down list in the **Type** rollout and you are prompted to specify the draw direction.
2. Select the **- ZC Axis** option from the **Inferred Vector** drop-down list.
3. Next, you are prompted to select the planar face; select the top face of the rectangular pad feature.
4. Next, you are prompted to select faces for drafting; select the cylindrical face of the boss feature.
5. Enter **20** as the value in the **Angle 1** edit box and choose the **OK** button. The draft feature is created, as shown in Figure 8-39.

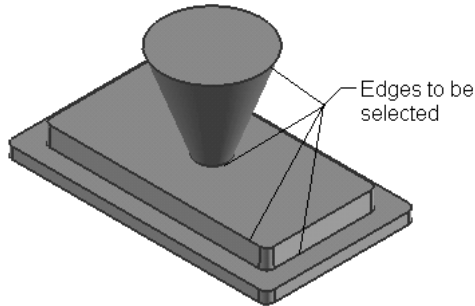


**Figure 8-39** The model after creating the draft feature

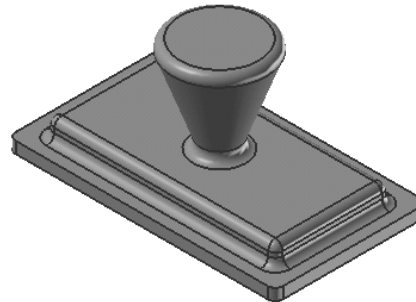
### Creating the Edge Blend Feature

You can create the edge blend feature by using the following steps:

1. Choose the **Edge Blend** button from the **Feature Operation** toolbar; the **Edge Blend** dialog box is displayed.
2. Select the edges of the model, as shown in Figure 8-40, and enter the value **2** in the **Radius 1** edit box. Choose the **OK** button from the **Edge Blend** dialog box; the final model for tutorial 1 is shown in Figure 8-41.



*Figure 8-40 Edges to be selected*



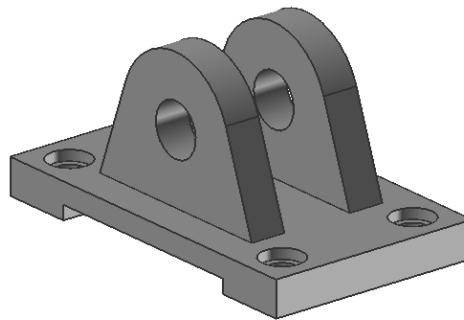
*Figure 8-41 The final model*

3. Choose **File > Close > Save and Close** from the menu bar to save and close the part file.

## Tutorial 2

In this tutorial, you will create the model shown in Figure 8-42. The dimensions of this model are given in Figure 8-43. After creating the solid model, save it with the name  
 \NX 6\c08\c08tut2.prt

**(Expected time: 30 min)**



*Figure 8-42 The solid model for Tutorial 2*

The following steps are required to complete this tutorial:

- a. Draw the sketch for the base feature and extrude the same.
- b. Create the rectangular pocket using the **Pocket** tool.
- c. Create the rectangular pad using the **Pad** tool.
- d. Create the edge blend on the pad feature using the **Edge Blend** tool.
- e. Create the draft on the edge blend surface of the rectangular pad using the **Draft** tool.

- f. Create the cylindrical pocket in the rectangular pad feature using the **Pocket** tool.
- g. Mirror the features with respect to the datum plane using the **Mirror Feature** tool.
- h. Create a counterbore hole using the **Hole** tool.
- i. Create a rectangular array of the hole feature using the **Instance Feature** tool.
- j. Save the file.

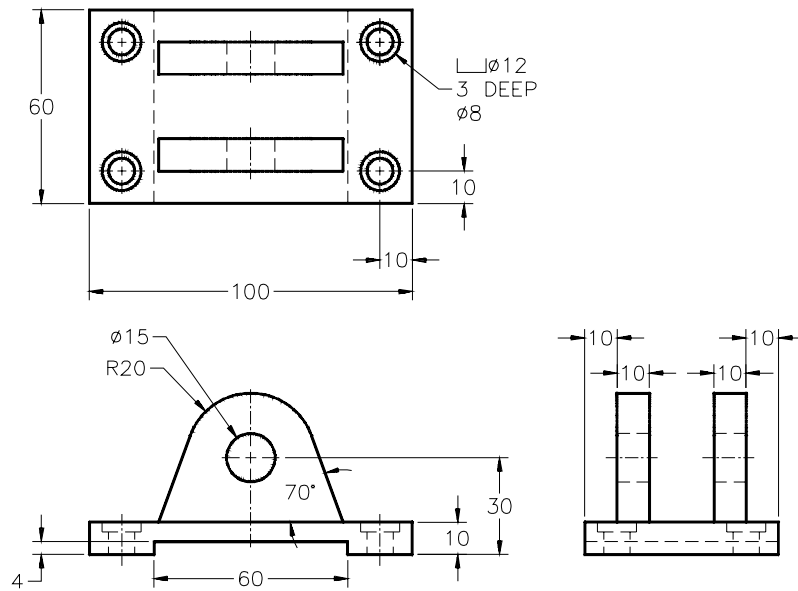


Figure 8-43 Views and dimensions for Tutorial 2

### Creating the Base Feature of the Model

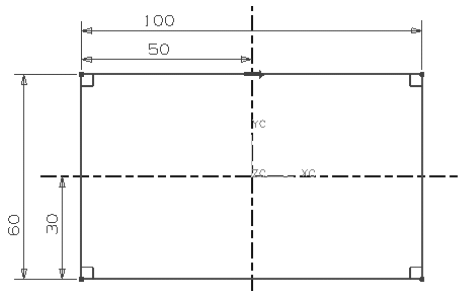
1. Start a new file using the Model template, save the file in the *C:\NX 6\c08* folder with the name *c08tut2.prt*. Create three fixed datum planes.
2. Select the XC-YC plane as the sketching plane and create the sketch for the base feature, as shown in Figure 8-44.
3. Exit the **Sketcher** environment and extrude the sketch with the value **10** in the **End** edit box.
4. Turn off the display of all entities except the base feature. The resulting model is shown in Figure 8-45.

### Creating the Rectangular Pocket Feature

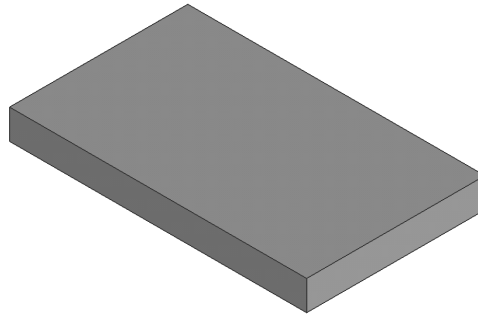
Next, you need to create a rectangular pocket feature using the following steps:

1. Choose the **Pocket** tool from the **Feature** toolbar; the **Pocket** dialog box is displayed and you are prompted to select the pocket type.



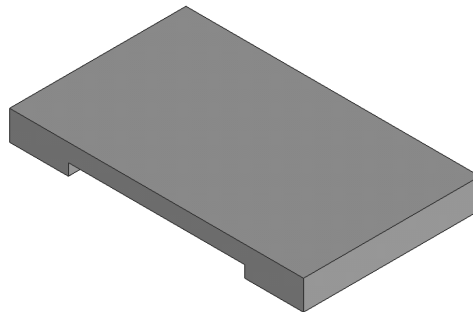


**Figure 8-44** Sketch for the base feature



**Figure 8-45** Base feature of the model


2. Choose the **Rectangular** button; the **Rectangular Pocket** dialog box is displayed and you are prompted to select the planar placement face.
3. Select the front face of the model that has the length 100 mm; the **Horizontal Reference** dialog box is displayed and you are prompted to select the horizontal reference.
4. Select the top edge of the same face; the **Rectangular Pocket** dialog box is displayed. Enter the values **60**, **4**, and **60** in the **Length**, **Width**, and **Height** edit boxes, respectively.
5. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pocket. Next, you need to position the pocket on the placement face.
6. Choose the **Perpendicular** button from the **Positioning** dialog box and select the edge of dimension 10 mm for the base feature and the vertical center line of the pocket. Enter **50** as the value in the **Create Expression** dialog box and choose the **OK** button.
7. Choose the **Perpendicular** button from the **Positioning** dialog box and select the bottom edge of the base feature and the horizontal center line of the pocket; enter **2** as the value in the **Create Expression** dialog box.
8. Choose the **OK** button twice and the **Cancel** button once. The rectangular pocket feature is created, as shown in Figure 8-46.

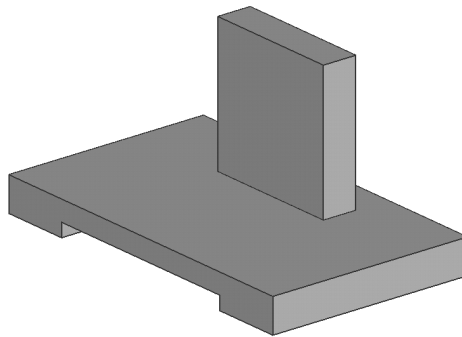


**Figure 8-46** The model after creating the rectangular pocket feature

### Creating the Rectangular Pad Feature

The next feature is a rectangular pad feature and will be created using the steps discussed next.

1. Choose the **Pad** tool from the **Feature** toolbar; the **Pad** dialog box is displayed and you are prompted to select the pad type. 
2. Choose the **Rectangular** button; the rectangular dialog box is displayed and you are prompted to select the planar placement face.
3. Select the top face of the base feature as the placement face; the **Horizontal Reference** dialog box is displayed and you are prompted to select the horizontal reference.
4. Select the edge of the base feature that has 100 mm dimension; the **Rectangular Pad** dialog box is displayed. Enter the values **40**, **10**, and **40** in the **Length**, **Width**, and **Height** edit boxes, respectively.
5. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pad. Next, you need to position the pad on the base feature.
6. Choose the **Perpendicular** button from the **Positioning** dialog box; select the 60 mm dimension edge of the base feature and 10 mm dimension edge of the pad and enter 30 as the value in the **Create Expression** dialog box. Choose the **OK** button; the **Positioning** dialog box is displayed.
7. Choose the **Perpendicular** button from the **Positioning** dialog box. Select the top edge of the front face of the base feature and the bottom edge of the front face of the new pad feature that measures 40 mm. Enter 40 as the value in the **Create Expression** dialog box.
8. Choose the **OK** button twice and then the **Cancel** button once. The rectangular pad feature is created, as shown in Figure 8-47.



*Figure 8-47 The model after creating the rectangular pad feature*

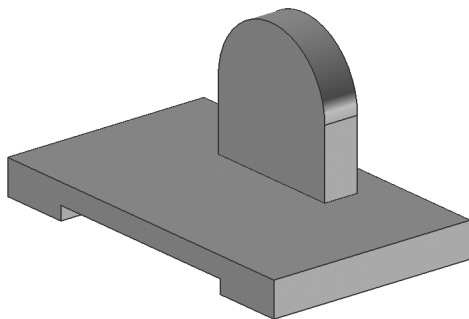
### Creating the Edge Blend Feature

1. Choose the **Edge Blend** button from the **Feature Operation** toolbar; the **Edge Blend** dialog box is displayed.
2. From the top face of the pad feature, select the two edges that measure 10 mm. Enter value **20** in the **Radius 1** edit box.
3. Choose the **OK** button from the **Edge Blend** dialog box. The rectangular pad, after creating the edge blend feature, is shown in Figure 8-48.

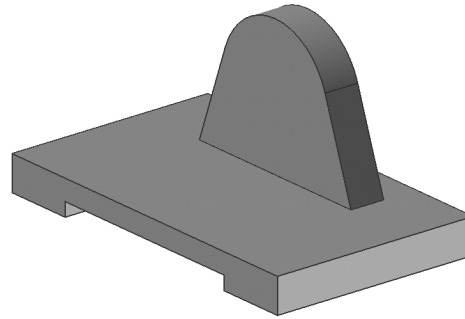
### Creating the Draft Feature

Next, you need to create the draft feature on the side faces of the pad feature. To do so, you need to follow the steps given next.

1. Choose the **Draft** tool from the **Feature Operation** toolbar; the **Draft** dialog box is displayed. By default, the **From Plane** option is selected from the drop-down list in the **Type** rollout.
2. Select the **Tangent to Faces** option from the drop-down list in the **Type** rollout; you are prompted to specify the draw direction. Select the **ZC Axis** option from the **Inferred Vector** drop-down list.
3. Next, you are prompted to select the tangent faces; select the blended face from the rectangular pad feature. The two side faces that are tangent to the blended face will also be selected.
4. Enter the angle value **20** in the **Angle** edit box. Choose the **Apply** button and then the **Cancel** button. The resulting model is shown in Figure 8-49.



**Figure 8-48** The model after creating the edge blend feature



**Figure 8-49** The model after creating the draft feature

### Creating the Cylindrical Pocket Feature

The hole in the pad feature will be created using the cylindrical pocket feature.

1. Choose the **Pocket** tool from the **Feature** toolbar; the **Pocket** dialog box is displayed and you are prompted to select the pocket type.

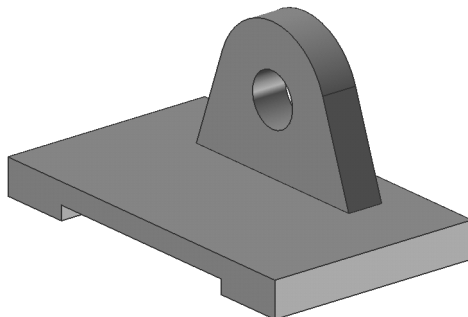




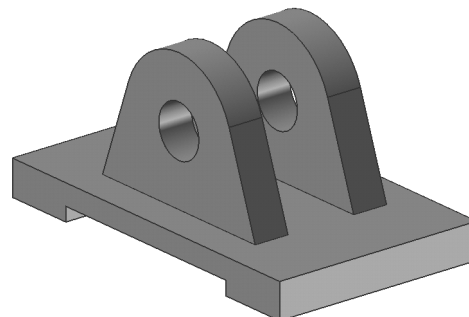
2. Choose the **Cylindrical** button; the **Cylindrical Pocket** dialog box is displayed and you are prompted to select the planar placement face.
3. Select the front face of the rectangular pad feature; the **Cylindrical Pocket** dialog box is displayed and you are prompted to enter the pocket parameters.
4. Enter the values 15 for the pocket diameter and 10 for the depth in the respective edit boxes. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pocket.
5. Choose the **Point onto Point** button from the **Positioning** dialog box, and from the pad feature, select the curved edge that was created because of the edge blend; the **Set Arc Position** dialog box is displayed. Choose the **Arc Center** button.
6. Next, select the cylindrical pocket edge from the preview of the pocket and choose the **Arc Center** button from the **Set Arc Position** dialog box. The cylindrical pocket is created, as shown in Figure 8-50.

### Mirroring Features

1. Choose the **Mirror Feature** tool from the **Feature Operation** toolbar; the **Mirror Feature** dialog box is displayed. You are prompted to select the features to be mirrored.
2. Expand the **Related Features** sub-rollout. Next, select the Rectangular Pad feature and all features below it from the list in this sub-rollout. The selected features are highlighted in the drawing window.
3. Choose the **Plane** button from the **Mirror Plane** rollout; you are prompted to select the plane to mirror about. Press the SHIFT+CTRL+K keys to invoke the **Show** tool and turn on the display of the XC-ZC plane. When you exit the **Show** tool, the **Mirror Feature** dialog box is redisplayed.
4. Select the XC-ZC plane as the mirror plane. Choose the **Apply** button and then the **Cancel** button. The resulting model is shown in Figure 8-51.



**Figure 8-50** The model after creating the cylindrical pocket feature



**Figure 8-51** The model after creating the mirror feature

### Creating the Counterbore Hole Feature on the Base Feature

1. Choose the **Hole** button from the **Feature Operation** toolbar; the **Hole** dialog box is displayed and you are prompted to select the placement face.
2. By default, the **General Hole** option is selected in the drop-down list of the **Type** rollout. Select the **Counterbored** option from the **Form** drop-down list of the **Form and Directions** rollout.
3. Select the top face of the base feature as the placement face; the **Sketching** environment will be activated.
4. Place the point and apply the dimensions to it, as shown in Figure 8-52.
5. Exit the sketching environment; the preview of the hole is displayed with the default dimensions.
6. Enter **12** as the value in the **C-Bore Diameter** edit box, **3** in the **C-Bore Depth** edit box, **8** in the **Diameter** edit box, and **10** in the **Depth** edit box of the **Dimensions** sub-rollout.
7. Press ENTER; the preview of the resultant model is displayed. Accept the default settings and choose the **OK** button. The counterbore hole is created, as shown in Figure 8-53.

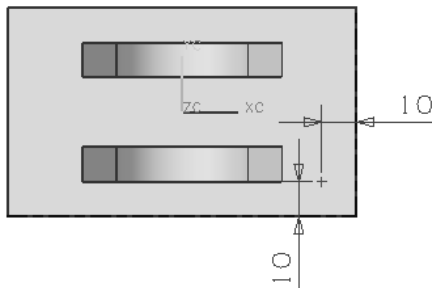


Figure 8-52 Dimensions applied to the point

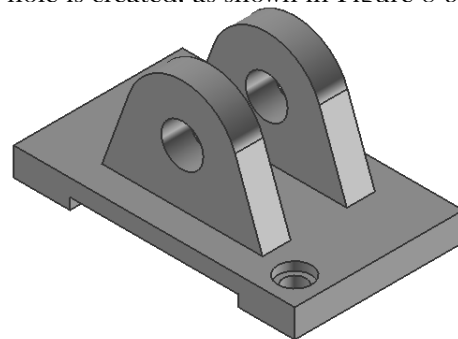
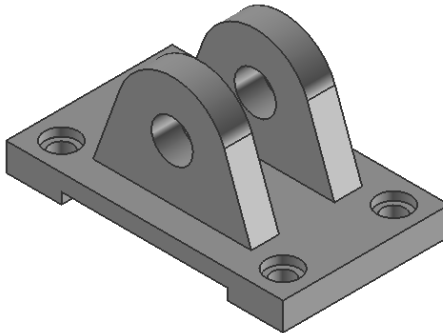


Figure 8-53 The model after creating the hole feature

### Creating the Rectangular Array of the Hole Feature

1. Choose the **Instance Feature** tool from the **Feature Operation** toolbar; the **Instance** dialog box is displayed.
2. Choose the **Rectangular Array** button and select the counterbore hole from the **Filter** list. Choose the **OK** button; the **Enter Parameters** dialog box is displayed.
3. Enter **2** as the value in the **Number Along XC** and **Number Along YC** edit boxes. Enter **-80** as the value in the **XC Offset** edit box and **40** in the **YC Offset** edit box.
4. Choose the **OK** button from the **Enter parameters** dialog box and then choose the **Yes** button from the **Create instances** dialog box. Exit the **Instance** dialog box. The final model is shown in Figure 8-54.



*Figure 8-54 The final model*

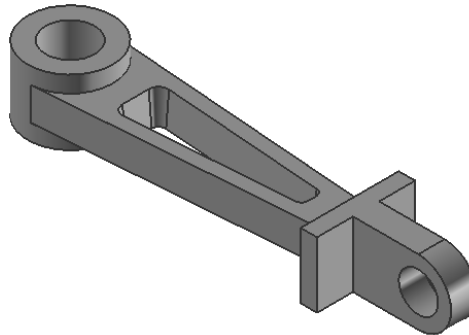
5. Choose **File > Close > Save and Close** from the menu bar to save and close the part file.

### Tutorial 3

In this tutorial, you will create the model shown in Figure 8-55. The dimensions of this model are shown in Figure 8-56. After creating the solid model, save it with the name

`|NX 6|c08|c08tut3.prt`

**(Expected time: 30 min)**



*Figure 8-55 The solid model for Tutorial 3*

The following steps are required to complete this tutorial:

- Draw the sketch for the base feature and extrude it.
- Draw the sketch for the second feature and extrude it.
- Create the cylindrical pocket in the second feature using the **Pocket** tool.
- Create the cut extrude feature.
- Create the rectangular pad using the **Pad** tool.
- Create the rectangular pad using the **Pad** tool.
- Create the edge blend using the **Edge blend** tool.
- Create the cylindrical pocket in the rectangular pad using the **Pocket** tool.
- Save the file.

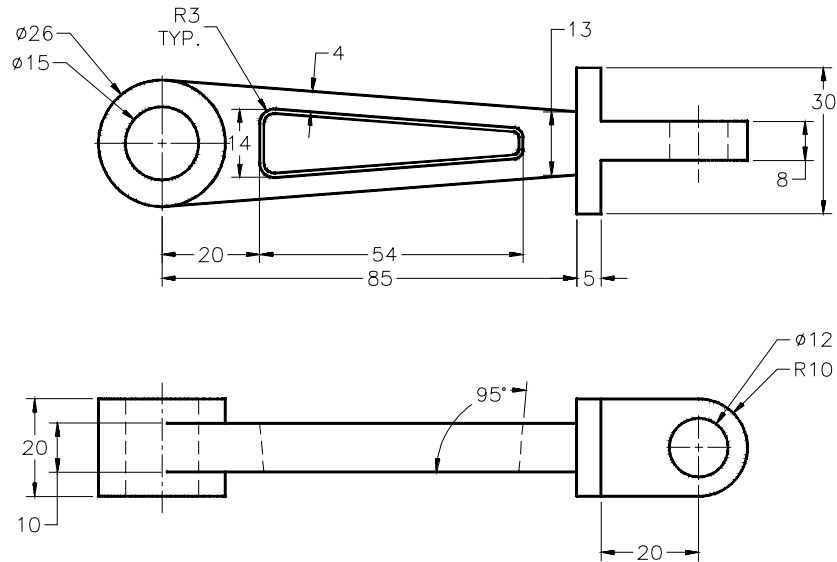


Figure 8-56 Views and dimensions of the model for Tutorial 3

### Creating the Base Feature of the Model

1. Start a new file using the Model template, save the file in the C:\NX 6\c08 folder with the name *c08tut3.prt*. Create three fixed datum planes.
2. Select the XC-YC plane as the sketching plane and create the sketch for the base feature, as shown in Figure 8-57.
3. Exit the **Sketcher** environment and extrude the sketch using the **Symmetric Value** option. Enter **5** as the value in the **Distance** edit box.
4. Turn off the display of all entities except the base feature. The resulting base feature is shown in Figure 8-58.

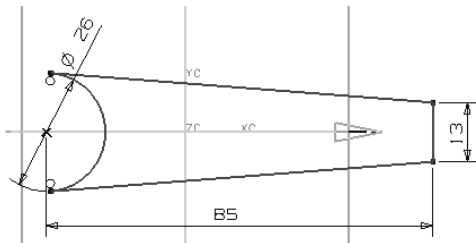


Figure 8-57 Sketch for the base feature

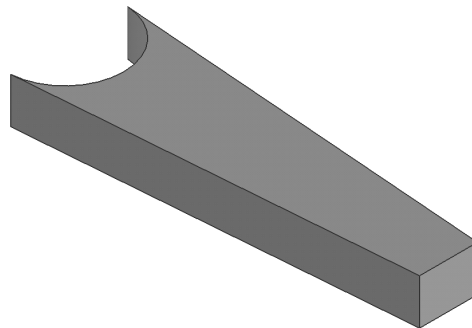
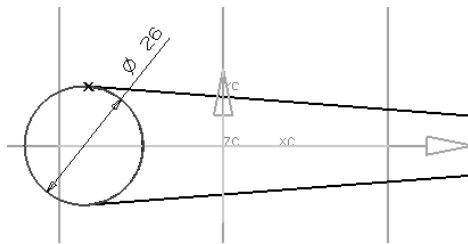


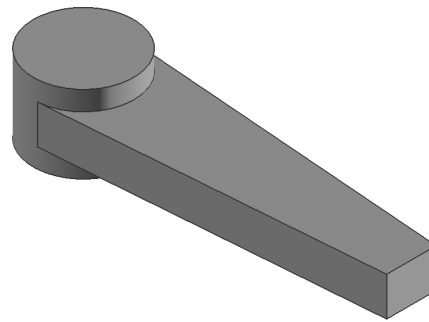
Figure 8-58 Base feature of the model

### Creating the Second Feature

1. Draw the sketch on the XC-YC plane, as shown in Figure 8-59.
2. Exit the **Sketcher** environment and invoke the **Extrude** dialog box.
3. Select the **Symmetric Value** option from the **Start** drop-down list. Enter **10** as the value in the **Distance** edit box. Select the **Unite** option from the **Boolean** drop-down list.
4. Choose the **OK** button to create the second feature. The resulting model is shown in Figure 8-60.



*Figure 8-59 Sketch for the second feature*



*Figure 8-60 The model after creating the second feature*

### Creating the Cylindrical Pocket Feature

Create the cylindrical pocket feature using the following steps:

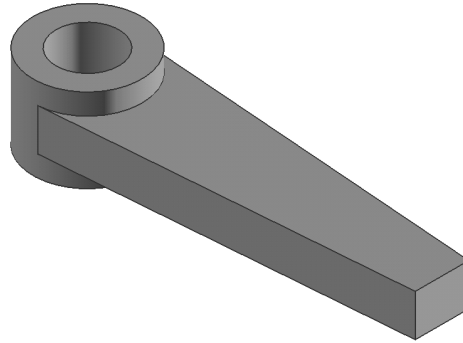
1. Choose the **Pocket** tool from the **Feature** toolbar; the **Pocket** dialog box is displayed and you are prompted to select the pocket type.
2. Choose the **Cylindrical** button; the **Cylindrical Pocket** dialog box is displayed and you are prompted to select the planar placement face.
3. Select the top face of the second feature; the **Cylindrical Pocket** dialog box is displayed and you are prompted to enter the pocket parameters.
4. Enter the pocket diameter value as 15 and the depth value as 20 in the respective edit boxes. Choose the **OK** button.



Next, you need to position the pocket on the face of the second feature.

5. Choose the **Point onto Point** button from the **Positioning** dialog box and select the edge of the second feature; the **Set Arc Position** dialog box is displayed. Choose the **Arc Center** button.
6. Next, select the cylindrical pocket edge from the preview of the pocket and choose

the **Arc Center** button from the **Set Arc Position** dialog box. The cylindrical pocket feature is created, as shown in Figure 8-61.

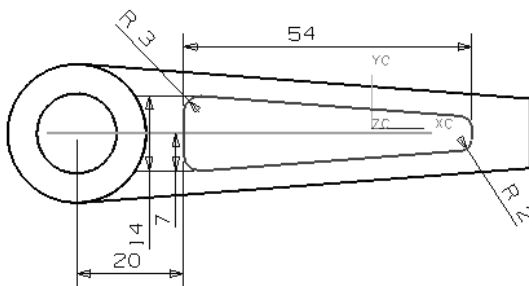


*Figure 8-61 Model after creating the pocket feature*

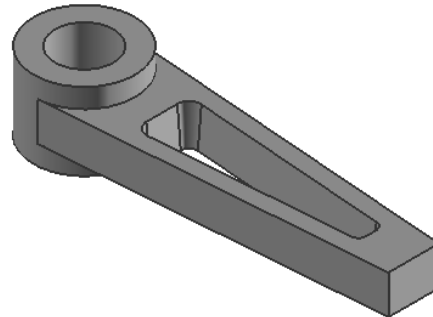
### Creating the Cut Extrude Feature

Next, you need to create the cut extrude feature. The sketch for the feature will be created on the top face of the base feature.

1. Select the **top** surface of the base feature as the sketching plane and create the sketch for the cut extrude feature, as shown in Figure 8-62.
2. Exit the **Sketcher** environment and subtract the sketch from the base feature with 5-degree as the draft angle.
3. Turn off the display of all entities except the model. The model, after creating the cut extrude feature, is shown in Figure 8-63.



*Figure 8-62 Sketch for the cut extrude feature*



*Figure 8-63 The model after creating the cut extrude feature*

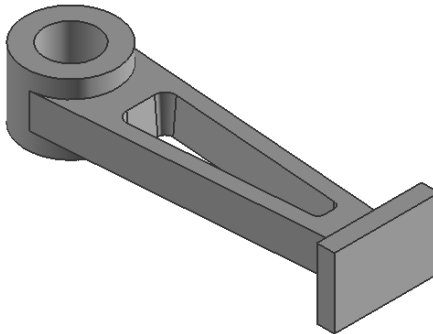
### Creating the Rectangular Pad Feature

Next, you need to create the rectangular pad feature on the right face of the base feature.

1. Choose the **Pad** button from the **Feature** toolbar; the **Pad** dialog box is displayed and you are prompted to select the pad type.



2. Choose the **Rectangular** button; the **Rectangular Pad** dialog box is displayed and you are prompted to select the planar placement face.
3. Select the right face of the base feature as the placement face; the **Horizontal Reference** dialog box is displayed and you are prompted to select the horizontal reference.
4. Select the edge of the base feature that measures 13 mm; the **Rectangular Pad** dialog box is displayed. Enter the length value as **30**, width as **20**, and depth as **5** in their respective edit boxes.
5. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pad.
6. Choose the **Perpendicular** button from the **Positioning** dialog box and select any one of the edges of the right face that measures 13 mm and the center line parallel to this edge from the preview of the rectangular pad.
7. Enter **5** as the value in the **Create Expression** dialog box and choose the **OK** button; the **Positioning** dialog box is displayed.
8. Choose the **Perpendicular** button from **Positioning** dialog box, and from the right face of the base feature, select any of the edges of the right face that measures 10 mm. Also, select the center line parallel to this edge from the preview of the rectangular pad.
9. Enter **7** as the value in the **Create Expression** dialog box.
10. Choose the **OK** button twice and then the **Cancel** button once. The rectangular pad feature is created, as shown in Figure 8-64.



*Figure 8-64 The model after creating the pad feature*

### Creating the Second Rectangular Pad Feature

The next feature is also a rectangular pad feature and will be created on the right face of the previously created rectangular pad feature.

1. Choose the **Pad** tool from the **Feature** toolbar; the **Pad** dialog box is displayed and you are prompted to select the pad type.



2. Choose the **Rectangular** button; the **Rectangular Pad** dialog box is displayed and you are prompted to select the planar placement face.
3. Select the right face of the previous rectangular pad feature; the **Horizontal Reference** dialog box is displayed and you are prompted to select the horizontal reference.
4. Select the edge of the rectangular pad feature that measures 20 mm; the **Rectangular Pad** dialog box is displayed. Enter the value for length as **20**, width as **8**, and depth as **30** in their respective edit boxes.
5. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pad.
6. Choose the **Perpendicular** button from the **Positioning** dialog box and from the previous rectangular pad feature, select the edge that measures 20 mm. Also, select the center line parallel to this edge from the preview of the rectangular pad.
7. Enter **15** as the value in the **Create Expression** dialog box and choose the **OK** button; the **Positioning** dialog box is displayed.
8. Choose the **Perpendicular** button from the **Positioning** dialog box and select the edge that measures 30 mm from the previous pad feature. Also, select the center line parallel to this edge from the preview of the rectangular pad. Enter **10** as the value in the **Create Expression** dialog box.
9. Choose the **OK** button twice and then the **Cancel** button once. The rectangular pad feature is created, as shown in Figure 8-65.

### Creating the Edge Blend Feature

1. Invoke the **Edge Blend** tool from the **Feature Operation** toolbar; the **Edge Blend** dialog box is displayed.
2. Select the upper and lower edges of the right face of the second pad feature. Both these edges measure 8 mm. Enter **10** as the value in the **Radius 1** edit box.
3. Choose the **OK** button from the **Edge Blend** dialog box. The rectangular pad, after creating the edge blend feature, is shown in Figure 8-66.

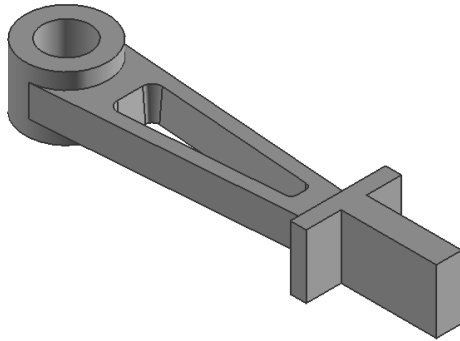
### Creating the Cylindrical Pocket Feature

To complete this model, you need to add a cylindrical pocket feature to the second rectangular pad feature.

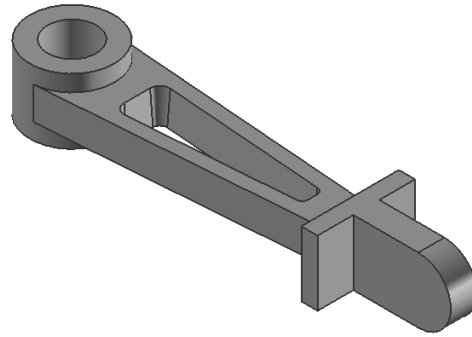
1. Choose the **Pocket** tool from the **Feature** toolbar; the **Pocket** dialog box is displayed and you are prompted to select the pocket type.
2. Choose the **Cylindrical** button; the **Cylindrical Pocket** dialog box is displayed and you are prompted to select the planar placement face.





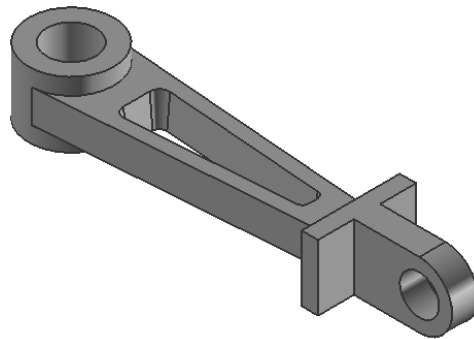


**Figure 8-65** Model after creating the pad feature



**Figure 8-66** Model after creating the edge blend feature

3. Select the front face of the previous rectangular pad feature; the **Cylindrical Pocket** dialog box is displayed and you are prompted to enter the pocket parameters. Enter the value of the diameter as **12** and depth as **10** in their respective edit boxes.
4. Choose the **OK** button; the **Positioning** dialog box is displayed, along with the preview of the pocket.
5. Choose the **Point onto Point** button from the **Positioning** dialog box and select the edge of the blend feature; the **Set Arc Position** dialog box is displayed. Choose the **Arc Center** button.
6. Select the cylindrical pocket edge from the preview of the pocket and choose the **Arc Center** button from the **Set Arc Position** dialog box; the cylindrical pocket is placed on the rectangular pad feature. The final model is shown in Figure 8-67.
7. Choose **File > Close > Save and Close** from the menu bar to save and close the part file.



**Figure 8-67** The final model for Tutorial 3

### Self-Evaluation Test

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

1. The **Reordering Features** option allows you to change the order in which the features are created. (T/F)
2. Boss can be placed on a nonplanar surface. (T/F)
3. In NX, you can create three types of pockets. (T/F)
4. The pad feature is defined as the process of adding material to the model. (T/F)
5. In the **Tangent to Faces** draft type, there is no need to select a stationary plane. (T/F)
6. General pockets can be created on both \_\_\_\_\_ and \_\_\_\_\_ faces.
7. In NX, you can create \_\_\_\_\_ types of drafts.
8. In a draft tool, the \_\_\_\_\_ check box allows you to select whether to taper only the specified instance or all instances in the array features.
9. The **Tangent to Faces** type is used to create a draft, which is \_\_\_\_\_ to the selected faces.
10. The length of the rectangular pad will be \_\_\_\_\_ to the horizontal reference.

### Review Questions

Answer the following questions:

1. Which one of the following cross-sections does the boss feature have?
 

(a) <b>Circular</b>	(b) <b>Rectangle</b>
(c) <b>Square</b>	(d) <b>None</b>
2. Which tool is used to create the cutout feature?
 

(a) <b>Pad</b>	(b) <b>Pocket</b>
(c) <b>Boss</b>	(d) <b>None</b>
3. Which of the following pocket types needs a sketch?
 

(a) <b>Cylindrical pocket</b>	(b) <b>General pocket</b>
(c) <b>Rectangular pocket</b>	(d) <b>None</b>

4. Which draft type is used to create a draft on the nonuniform edges?
- (a) **From Plane** (b) **From Edges**  
(c) **Tangent to Faces** (d) None
5. Which one of the following is the radius between the placement face and the side faces of the pocket?
- (a) Placement radius (b) Corner radius  
(c) Floor radius (d) None
6. Which tool is used to create the cylindrical extrusion feature?
- (a) **Boss** (b) **Pad**  
(c) **Pocket** (d) None
7. The **Inferred Vector** drop-down list from the **Draft** dialog box is used to specify the draw direction of the taper. (T/F)
8. A datum plane can be selected as the placement face for the boss feature. (T/F)
9. A boss is a feature that has a uniform diameter throughout its length. (T/F)
10. A draft feature can be placed on the surface body. (T/F)

## Exercises

### Exercise 1

Create the model shown in Figure 8-68. The drawing views and dimensions of the model are given in Figure 8-69. After creating the model, save it in the name specified below:

|NX 6|c08|c08exr1.prt

(Expected time: 30 min)

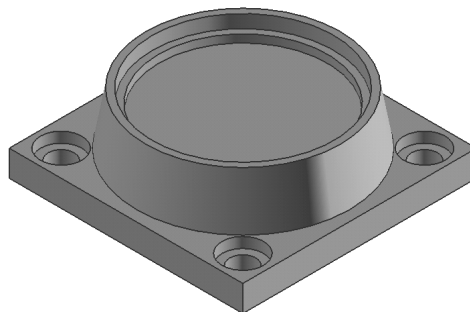
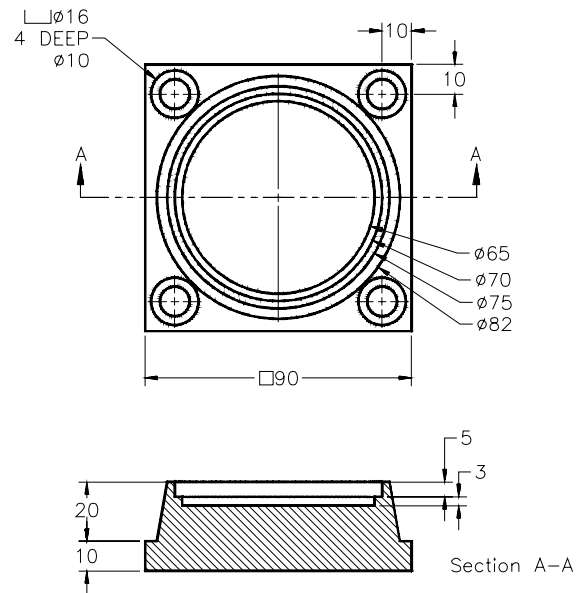


Figure 8-68 The solid model for Exercise 1



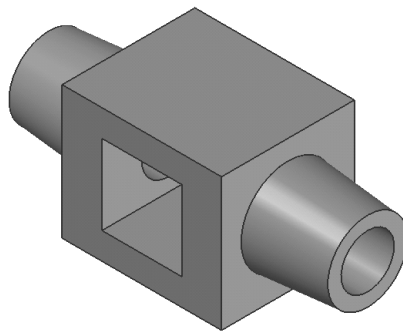
**Figure 8-69** Views and dimensions for Exercise 1

## Exercise 2

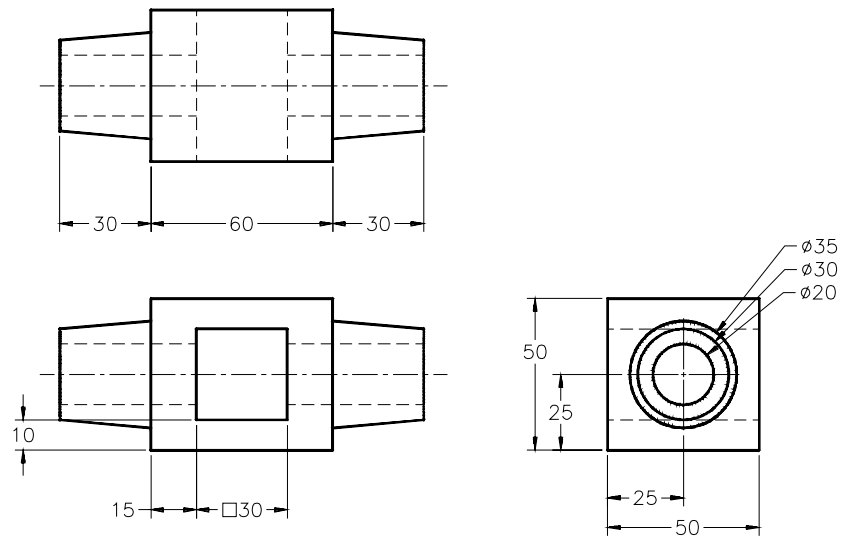
Create the model shown in Figure 8-70. The drawing views and dimensions of the model are given in Figure 8-71. After creating the model, save it in the name specified below:

|NX 6|c08|c08exr2.prt

(Expected time: 30 min)



**Figure 8-70** The solid model for Exercise 2



**Figure 8-71** Views and dimensions for Exercise 2

**Answers to Self-Evaluation Test**

1. T, 2. F, 3. T, 4. T, 5. T, 6. planar, nonplanar, 7. four, 8. **Draft All Instances**, 9. tangent, 10. parallel.