



Chapter 8

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

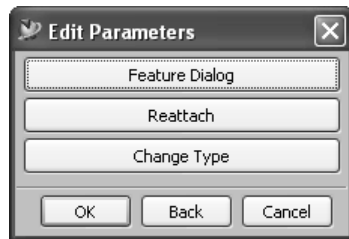


Figure 8-1 The **Edit Parameters** dialog box

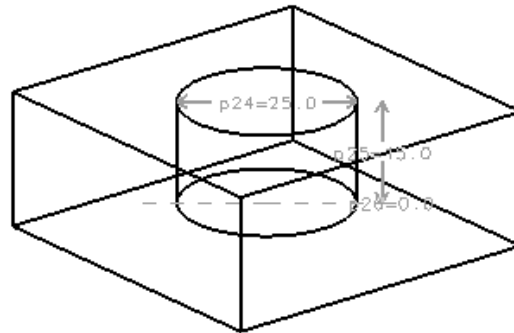


Figure 8-2 Parameters of a simple hole

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 a 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 positional values for the hole on the new placement face and create it, choose the **OK** button.

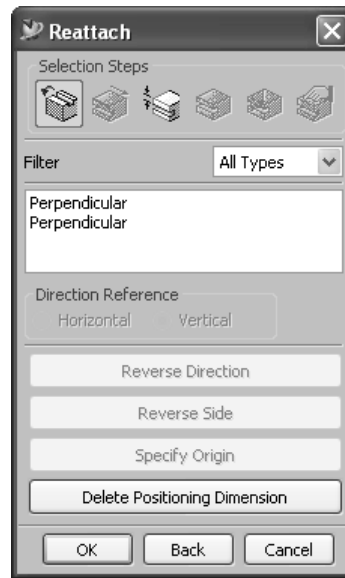


Figure 8-3 The *Reattach* dialog box

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

After selecting a dimension, you can also delete it by choosing the **Delete Positioning Dimension** button from the **Reattach** dialog box.

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** cascade menu will be displayed. Right-click on the hole feature name and select 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.

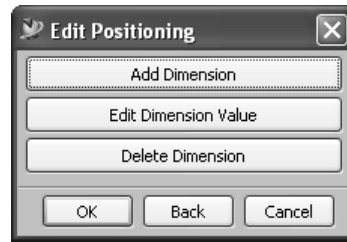


Figure 8-4 The *Edit Positioning* dialog box

Editing a Positioning Dimension using the Edit Dimension Value Button

The **Edit Dimension Value** button is used to edit an existing dimension value that was 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; the **Edit Positioning** dialog box will be displayed. Choose the **OK** button from this dialog box. Again, choose the **OK** button from the **Edit Positioning** dialog box, which will display the different options.

Adding a Positioning Dimension using the Add Dimension Button

The **Add Dimension** button is used to add a positioning dimension to an already positioned feature. To add a new positioning dimension, 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 dimension, as discussed in the previous chapter.

Deleting a Positioning Dimension Using the Delete Dimension Button

The **Delete Dimension** button is used to delete a positioning dimension. To delete a positioning dimension, 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 dimension and choose the **OK** button from the same dialog box; the selected dimension 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** cascade menu. Right-click on the groove feature displayed on the tree and select 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 from the same; the **Edit Positioning** dialog box will be redisplayed. Choose the **OK** button from the same 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** cascade menu. Right-click

on the slot feature displayed on the tree and select 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 from the same; the **Edit Positioning** dialog box will be redisplayed. Choose the **OK** button from the same dialog box to reflect the changes made in the positioning value.

Editing the Parameters of Features

Similar to editing 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 or icon options will be displayed. You can select the options from the dialog box or buttons from the icon options to modify the parameters of the selected feature.

Editing the Parameters of Features with the 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 a feature to be edited is selected. 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, click on the + sign on the left of the sketch-based feature in the **Part Navigator** to expand the tree view. Next, turn on the display of the sketch and right-click on it. Choose **Edit** from the shortcut menu; the sketching environment will be invoked and the model will be oriented such that the selected sketch is normal to the view. All dimensions applied to the sketch will be displayed. You can modify the sketch dimensions or remove the 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 orders 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 shortcut menu containing the names of the reference features will be displayed. Select the feature after or before which you need to reorder the selected feature.

Reorder Before

The feature will be reordered before the reference feature.

Reorder After

The feature will be reordered after the reference 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

Menu: Insert > Design Feature > Boss
Toolbar: Form Feature > Boss



Boss is defined as the process of adding material 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 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 **Form 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 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-degrees.

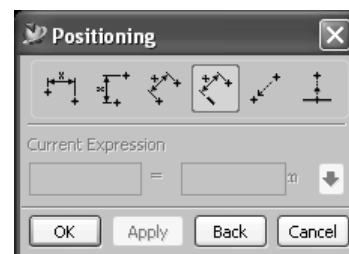


Figure 8-6 The **Positioning** dialog box

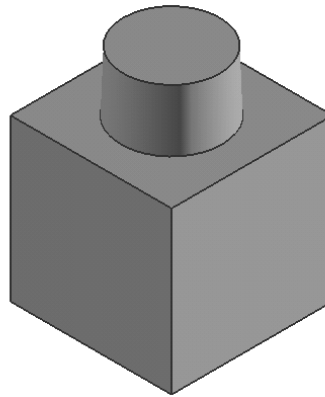


Figure 8-7 The resulting boss feature

**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

Pocket is defined as the process of removing the material in the cylindrical or rectangular cross-section from the model. In NX, you can create three types of pockets: cylindrical, rectangular, and general. The **Pocket** tool from the **Form Feature** toolbar is used to create all the three pocket types. The procedure for creating different types of pockets is discussed next.

Creating Cylindrical Pockets

Menu:	Insert > Design Feature > Pocket
Toolbar:	Form Feature > Pocket



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 **Form Feature** toolbar; the **Pocket** dialog box will be displayed, as shown in

Figure 8-8.

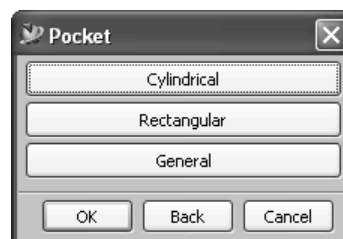


Figure 8-8 The **Pocket** dialog box

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.

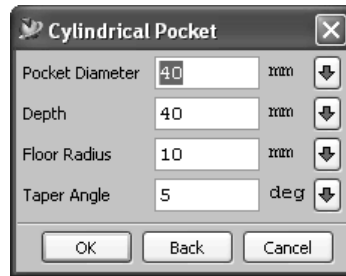


Figure 8-9 The Cylindrical Pocket dialog box

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-degrees.

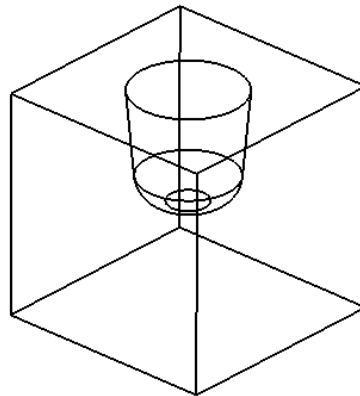


Figure 8-10 A cylindrical pocket feature

Creating Rectangular Pockets

Menu: Insert > Design Feature > Pocket
Toolbar: Form Feature > Pocket



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 the rectangular pocket, choose the **Pocket** button from the **Form 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.

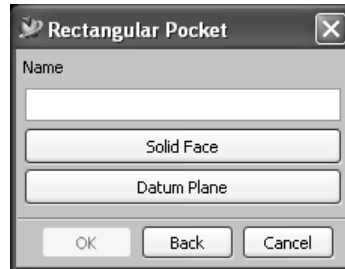


Figure 8-11 The **Rectangular Pocket** dialog box

Select the face or datum plane to start 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, 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.

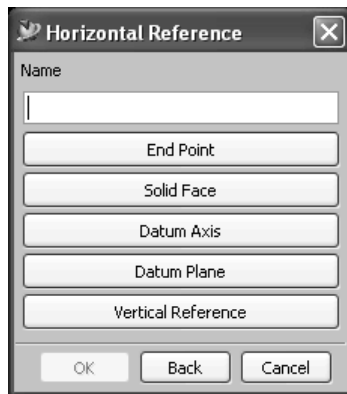


Figure 8-12 The **Horizontal Reference** dialog box

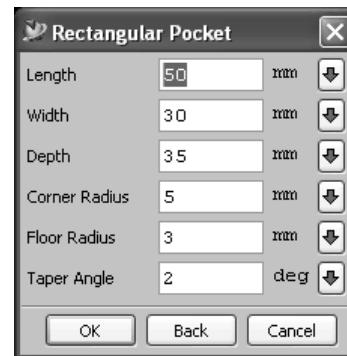


Figure 8-13 The **Rectangular Pocket** dialog box

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-degrees.

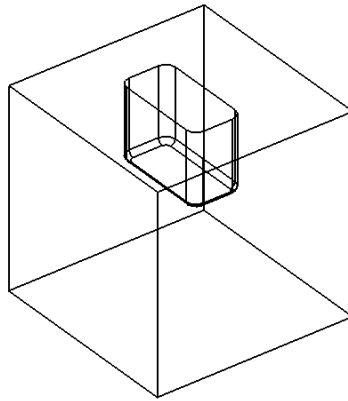


Figure 8-14 A rectangular pocket feature

Creating General Pockets

Menu: Insert > Design Feature > Pocket
Toolbar: Form Feature > Pocket



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 nonplanar. 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 **Form Feature** toolbar; the **Pocket** dialog box will be displayed, as shown in Figure 8-15.

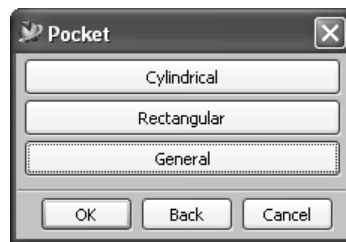


Figure 8-15 The Pocket dialog box

Choose the **General** button from the **Pocket** dialog box; the **General Pocket** dialog box will be displayed, as shown in Figure 8-16. By default, the **Placement Face** button from the **Selection Steps** area is chosen 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 8-17. 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 8-17. 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 8-17. 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 8-17. 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 8-17. 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 8-18 shows a general pocket feature created using the selection made in Figure 8-17.

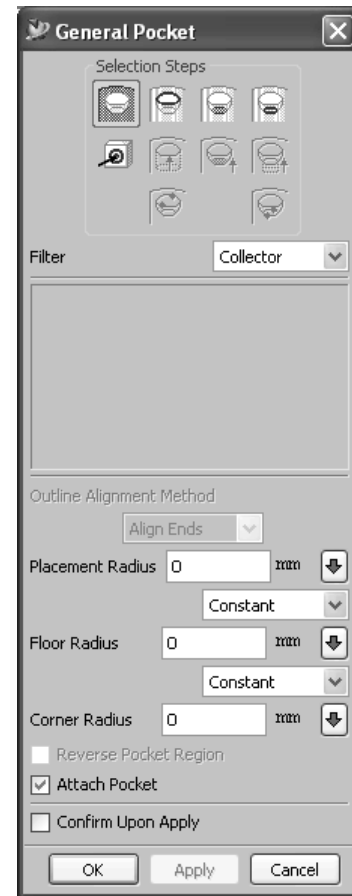


Figure 8-16 The **General Pocket** dialog box

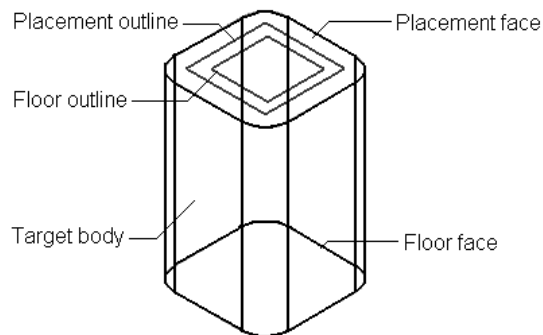


Figure 8-17 Objects to be selected

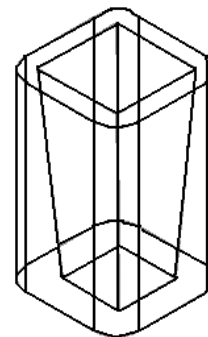


Figure 8-18 The general pocket feature

Creating Pad Features

Pad is defined as the process of adding material to the model in rectangular or user-defined cross-sections. In NX, you can create two types of pad features: rectangular and general. The **Pad** button from the **Form Feature** toolbar is used for creating both types of pads. The procedure for creating different types of pads is the same as creating the pocket features and is discussed next.

Creating Rectangular Pads

Menu: Insert > Design Feature > Pad
Toolbar: Form Feature > Pad



Pad

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 the rectangular pad, choose the **Pad** button from the **Form Feature** toolbar; the **Pad** dialog box will be displayed, as shown in Figure 8-19.



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

Choose the **Rectangular** button and then **OK**; the **Rectangular Pad** dialog box will be displayed, as shown in Figure 8-20, and you will be prompted to select a planar placement face.



Figure 8-20 The **Rectangular Pad** dialog box

You can select the datum plane or a 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, as shown in Figure 8-21, 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 8-22, 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

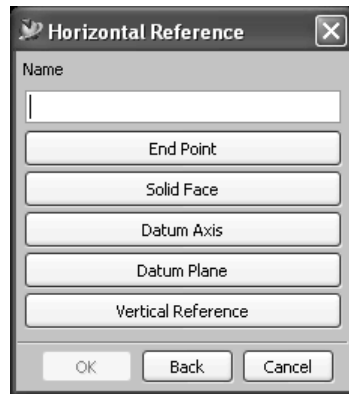


Figure 8-21 The **Horizontal Reference** dialog box

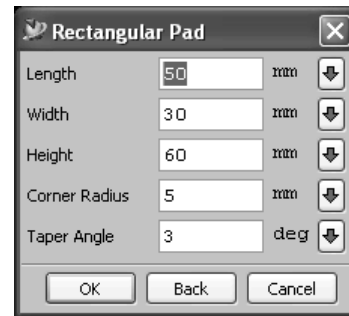


Figure 8-22 The **Rectangular Pad** dialog box

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 8-23 shows the rectangular pad feature of length 50, width 30, height 60, corner radius 5, and taper angle 3-degrees.

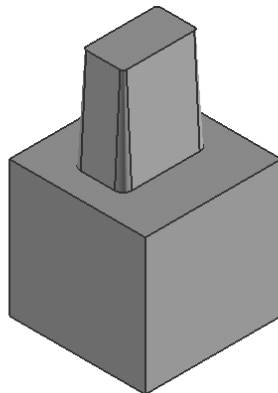


Figure 8-23 The resulting rectangular pad feature



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

Menu: Insert > Design Feature > Pad
Toolbar: Form Feature > Pad



Using a general pad option, you can add the material with a greater flexibility than with the **Rectangular** pad option. 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 **Form Feature** toolbar; the **Pad** dialog box will be displayed, as shown in Figure 8-24.

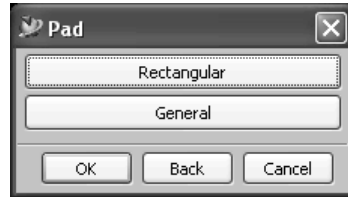


Figure 8-24 The **Pad** dialog box

Choose the **General** button from the **Pad** dialog box; the **General Pad** dialog box will be displayed, as shown in Figure 8-25. 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 8-26. 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 8-26. Choose the **Floor face** button from the **Selection Steps** area; you will be prompted to select the floor faces. Select the bottom face of the model, as shown in Figure 8-26. 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 that will define the shape of the pad feature at the bottom, as shown in Figure 8-26. Choose the **Target body** button from the **Selection Steps** area; you will be prompted to select the optional target body. Select the model to create the pad feature, as shown in Figure 8-26. Choose **Apply** and then the **Cancel** button from the **General Pad** dialog box. A general pad feature is shown in Figure 8-27.



Note

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

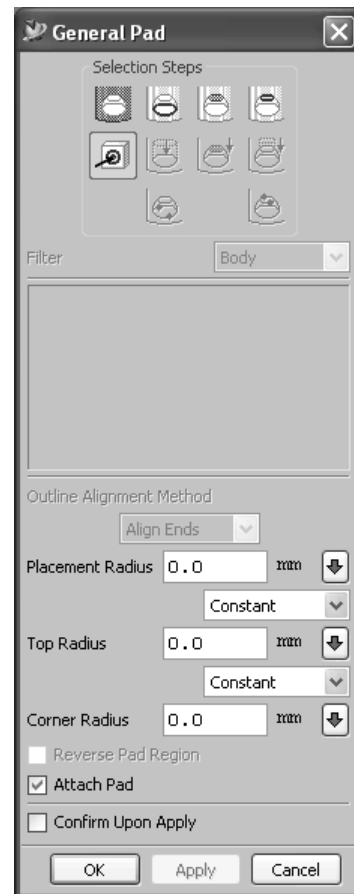


Figure 8-25 The **General Pad** dialog box

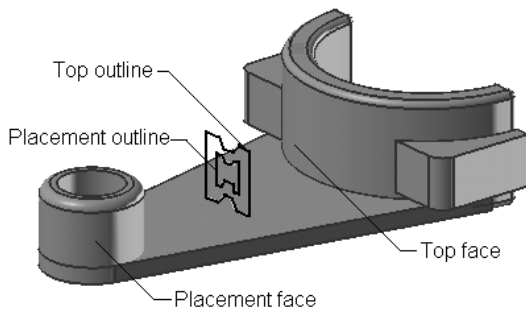


Figure 8-26 Objects to be selected

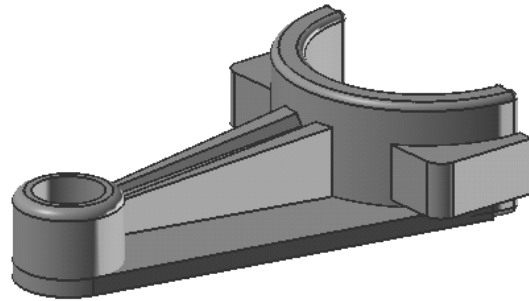


Figure 8-27 The resulting general pad feature

Creating Drafts

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



This tool is used to create a draft on an existing model. In NX, you can create four types of drafts: Draft from Stationary Plane, Draft from Stationary Edges, Draft Tangent to Faces, and Draft To Parting Edges. The **Draft** button from the **Feature Operation** toolbar is used to create all the four types of drafts. Before creating the draft feature, you need to understand the options used in this tool. These options are discussed next.

Draw Direction

This option allows you to select the draft direction vector. You can use the **Vector Method** drop-down list to select the draw direction. The **Reverse Direction** button is used to flip the direction of the vector.

Stationary Plane

This option allows you to select a planar face and a datum plane normal to the draw direction. It is not available for the Draft From Stationary Edges and Draft Tangent to Faces drafts.

Faces to Draft

This option allows you to select the faces involved in the draft operation. It is available only for the Draft From Stationary Plane draft.

Stationary Edges

This option allows you to select edges that are to remain unchanged throughout the draft operation. It is available only for the Draft to Stationary Edges draft.

Variable Angle Point

This option allows you to select points on stationary edges to specify the different draft angles. You can enter different angles for each of the specified points. It is available only for the Draft to Stationary Edges draft.



Tangent Faces to Draft

This option allows you to select tangent faces to draft. It is available only for the Draft From Tangent Faces draft type.

Parting Edges

This option allows you to select the parting edges. It is available only for the Draft To Parting Edges draft type.

The procedure for creating different types of drafts is discussed next.

Creating the Draft From Stationary Plane

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



This 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, as shown in Figure 8-28.

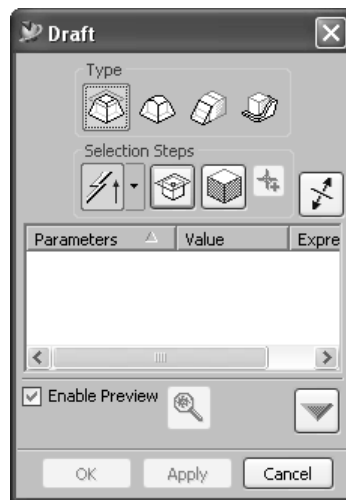


Figure 8-28 The *Draft* dialog box

By default, the **Draft From Stationary Plane** button is chosen from the **Type** area and the **Draw Direction** button is chosen from the **Selection Steps** area. As a result, you will be prompted to specify the draw direction. Select the edge for the draw direction, as shown in Figure 8-29. Alternatively, using the **Vector Method** drop-down list, select the draw direction. You can flip the draw direction by choosing the **Reverse Direction** button from the **Draft** dialog box. Next, you will be prompted to select the planar face. Select the face of the model, as shown in Figure 8-29. Next, you will be prompted to select the faces; select the faces, as shown in Figure 8-29. Enter the angle value in the **Set1 A** edit box. Choose **Apply** and then the **Cancel** button to create the draft. The **Enable Preview** check box allows you to see the preview of the resultant taper. The resulting model, after creating the draft feature, is shown in Figure 8-30.

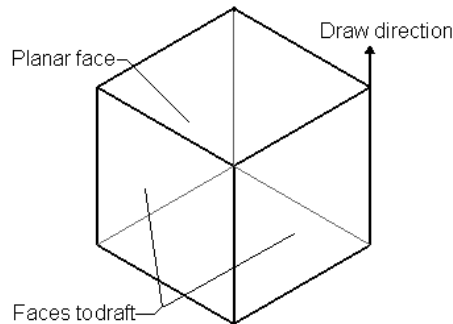


Figure 8-29 Objects to be selected

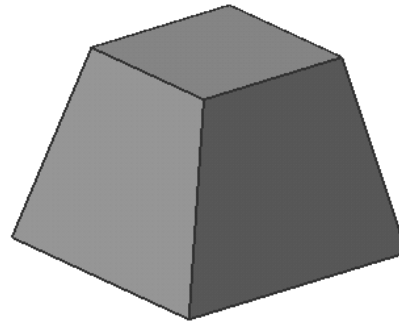


Figure 8-30 The draft created using the Stationary Plane

Creating the Draft From Stationary Edges

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



This option is used to create a draft by selecting the edges of the model. You need to specify the variable points and angles. This option is useful when the edges of the faces to be tapered are not linear. To create this type of draft, choose the **Draft From Stationary Edges** button from the **Type** area of the **Draft** dialog box; you will be prompted to specify the draw direction. Select the edge of the model to specify the draw direction, as shown in Figure 8-31. Alternatively, you can use the options in the **Vector Method** drop-down list to specify the draw direction. You can flip the draw direction by choosing the **Reverse Direction** button from the **Draft** dialog box. Next, you will be prompted to select the stationary edges; select the edges, as shown in Figure 8-31. Choose the **Variable Angle Point** button from the **Selection Steps** area; you will be prompted to select the point to specify the draft angle. You can select any number of points on the edges to specify the different angle values. Select the variable angle points, as shown in Figure 8-31. Enter the different angle values in the **Variable Angle** edit box. Choose **Apply** and then the **Cancel** button to create the taper. The model, after creating the draft feature, is shown in Figure 8-32.

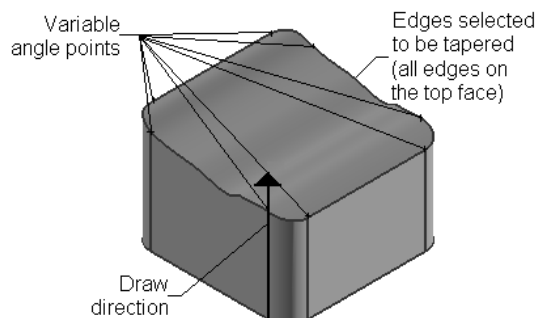


Figure 8-31 Objects to be selected

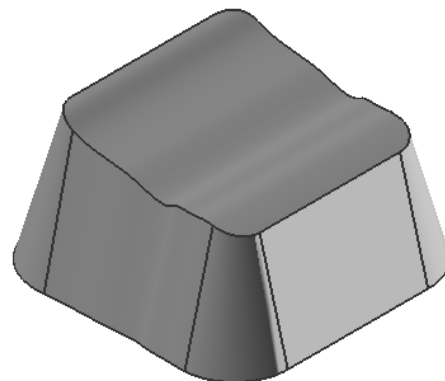


Figure 8-32 The draft created using the Stationary Edges

Creating the Draft Tangent to Faces

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



This option is used to create a draft that is tangent to the selected faces. You need to specify the draw direction and the tangent face. To create this type of draft, choose the **Draft Tangent to Faces** button from **Type** area; you will be prompted to specify the draw direction. Select the edge of the model, as shown in Figure 8-33. Alternatively, you can use the options in the **Vector Method** drop-down list to select the draw direction. Next, you will be prompted to select the tangent faces; select the face, as shown in Figure 8-33. Enter the angle values in the **Set1 A** edit box. Choose the **Apply** button and then the **Cancel** button to create the taper. The resulting model, after creating the taper feature, is shown in Figure 8-34.

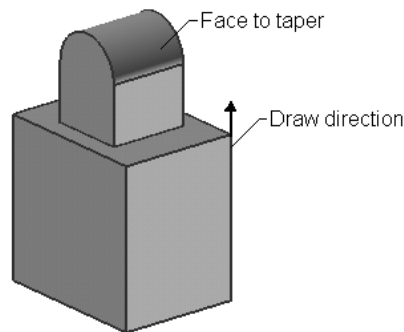


Figure 8-33 Objects to be selected

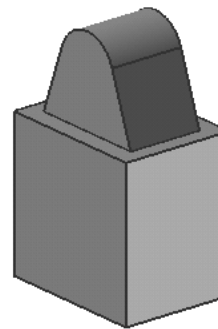


Figure 8-34 The draft created using the Tangent to Faces

Creating the Draft To Parting Edges

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



This type is used to create the draft along the selected set of edges by specifying an angle, draw direction, stationary plane, and the parting edges. To create this type of draft, choose the **Draft To Parting Edges** button from the **Type** area; you will be prompted to specify the draw direction. Select the edge of the model, as shown in Figure 8-35. You can also use the options in the **Vector Method** drop-down list to specify the draw direction. Next, you will be prompted to select the planar face; select it, as shown in Figure 8-35. Next, you will be prompted to select the parting edges; select the edges, as shown in Figure 8-35. Enter the angle values in the **Set1 A** 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 8-36.

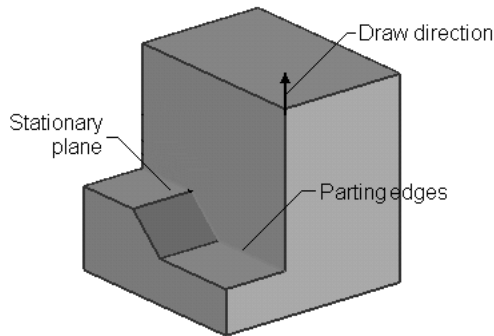


Figure 8-35 Objects to be selected

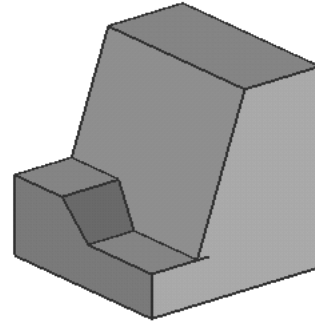


Figure 8-36 The resulting Draft To Parting Edges draft feature

More Options in the Draft Dialog box



To display more options in the **Draft** dialog box, choose the **More Options** button. The **Draft** dialog box will expand and display more options. The **Draft All Instances** check box allows you to specify whether to draft only that instance or all the instances in the array.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 8-37. The dimensions of this model are given in Figure 8-38. After creating the solid model, save it with the name `\NX 4\c08\c08tut1.prt` (Expected Time: 30 min)

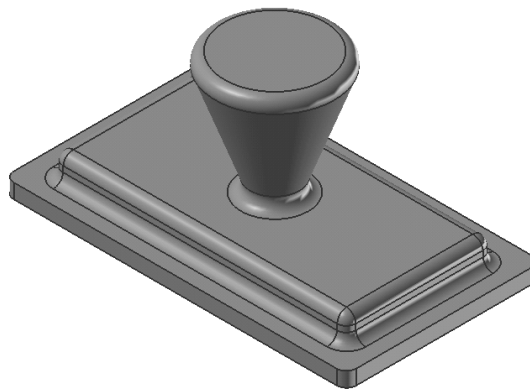


Figure 8-37 Solid model for Tutorial 1

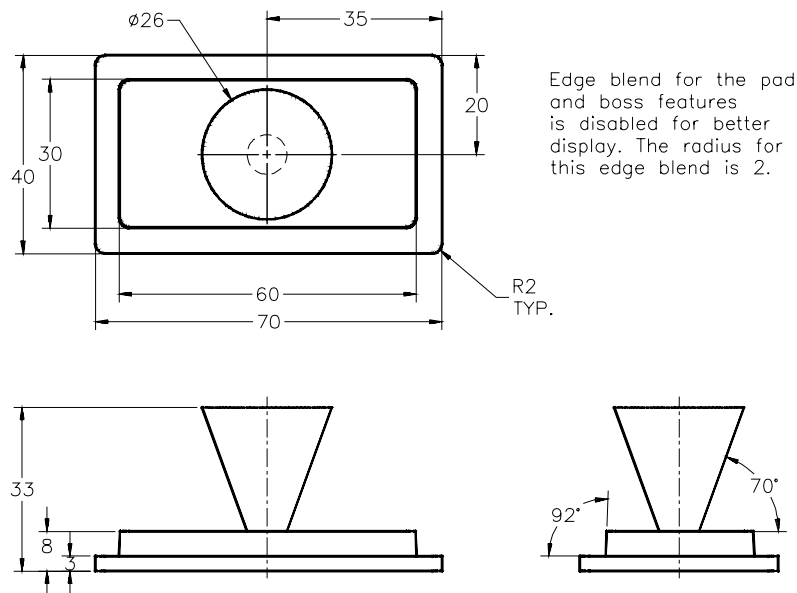


Figure 8-38 Views and dimensions for Tutorial 1

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

- Start a new file in NX.
- Draw the sketch for the base feature and extrude the same, refer to Figure 8-39.
- Create the rectangular pad using the **Pad** tool, refer to Figure 8-41.
- Create the boss feature using the **Boss** tool, refer to Figure 8-42.
- Create the draft on the boss feature, refer to Figure 8-43.
- Create the edge blend on the rectangular pad and the boss feature, refer to Figures 8-44 and 8-45, and then save the file.

Starting a New File

- Start a new file using the **Master Model Template** and save it as *c08tut1* in the *c08* folder.

Creating the Base Feature of the Model

- Choose the **Sketch** button from the **Form Feature** toolbar; the **Sketch Icon Options** are displayed. Select the **XC-YC Plane** as the sketching plane and create the sketch for the base feature, as shown in Figure 8-39.
- Choose the **Finish Sketch** button from the **Sketcher** toolbar and exit the **Sketcher** task environment. Next, you need to extrude the sketch created for the base feature.
- Choose the **Extrude** button from the **Form Feature** toolbar; you are prompted to select the section geometry. Select the sketch created for the base feature.

4. Enter the value as 3 in the **End** edit box and choose the **OK** button. Turn off the display of the sketch and the datum planes. The resulting base feature is shown in Figure 8-40.

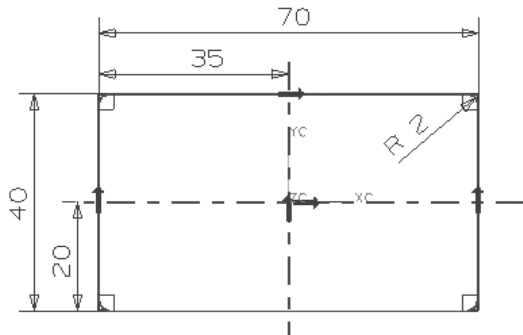


Figure 8-39 Sketch for the base feature

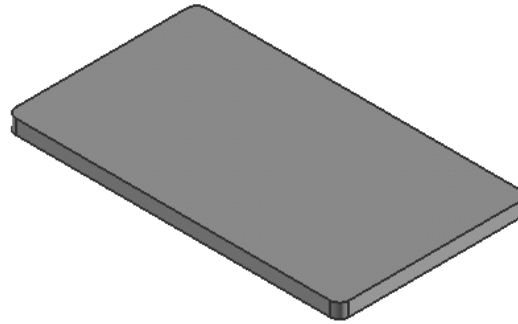


Figure 8-40 Base feature of the model

Creating the Rectangular Pad Feature

1. Choose the **Pad** tool from the **Form 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 measures 70; the **Rectangular Pad** dialog box is displayed. Enter the values in the edit boxes as length **60**, width **30**, depth **5**, corner radius **2**, and taper angle **2**.
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 5 as the value 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 8-41.

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 **Form 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 in the edit boxes as diameter **8**, height **25**, and taper angle **0**.
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; 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 created boss feature is shown in Figure 8-42.

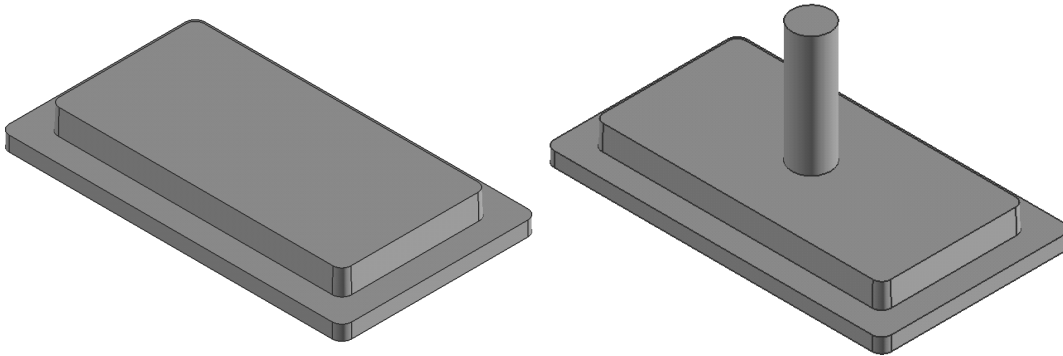


Figure 8-41 Model after creating the pad feature Figure 8-42 Model after creating the boss feature

Creating the Draft Feature

Next, you need to create the 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 **Draft From Stationary Plane** button is chosen from the **Type** area and you are prompted to specify the draw direction.
2. Select the **- ZC Axis** option from the **Vector Method** drop-down list.
3. Next, you are prompted to select the planar face; select the top face of the rectangular pad feature.

- Next, you are prompted to select the faces for draft; select the cylindrical face of the boss feature.
- Enter **20** as the value in the **Set1 A** edit box and choose the **OK** button. The draft feature is created, as shown in Figure 8-43.

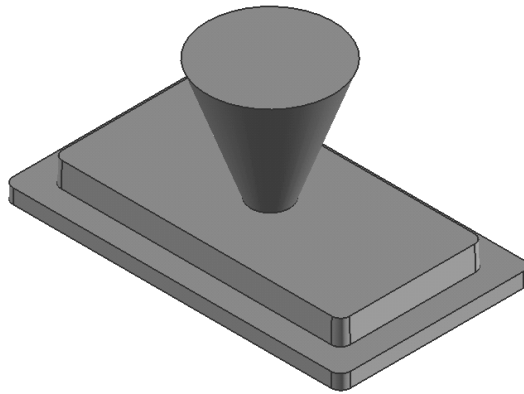


Figure 8-43 The model after creating the taper feature

Creating the Edge Blend Feature

Create the edge blend feature by using the following steps:

- Choose the **Edge Blend** button from the **Feature Operation** toolbar; the **Edge Blend** dialog box is displayed.
- Select the edges of the model, as shown in Figure 8-44, and enter the value as 2 in the **Set1 R** edit box. Choose the **OK** button from the **Edge Blend** dialog box.
- Choose **File > Close > Save and Close** to save and close the part file. The final model for tutorial 1 is shown in Figure 8-45.

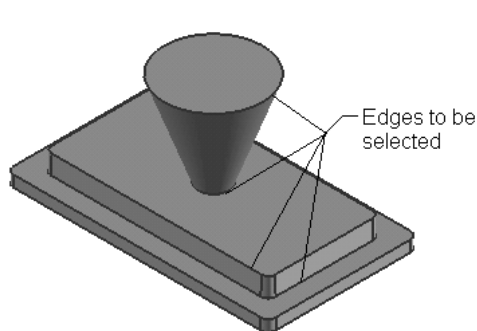


Figure 8-44 Edges to be selected

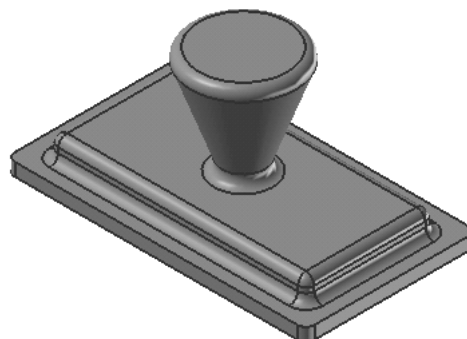


Figure 8-45 The final model

Tutorial 2

In this tutorial, you will create a model whose dimensions are shown in Figure 8-46. The solid model is shown in Figure 8-47. After creating the solid model, save it in the name `\NX 4\c08\c08tut2.prt` (Expected Time: 30 min)

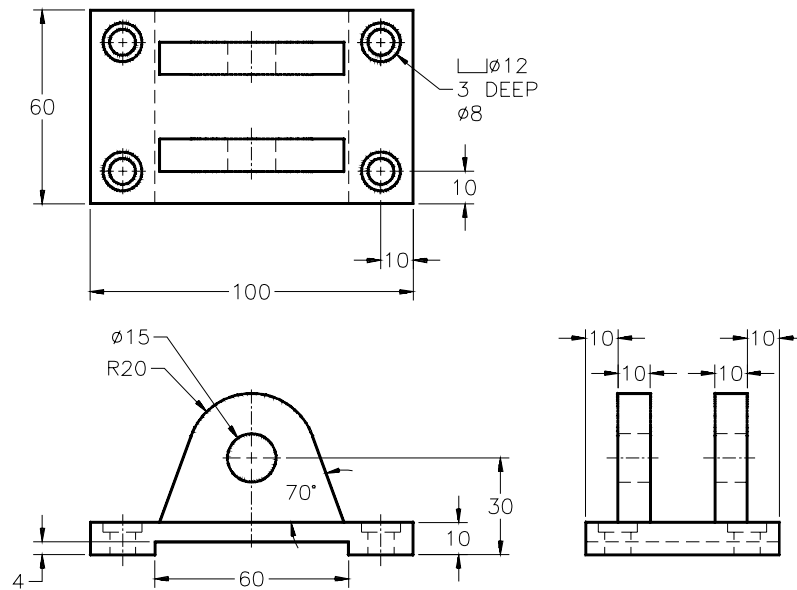


Figure 8-46 Views and dimensions for Tutorial 2

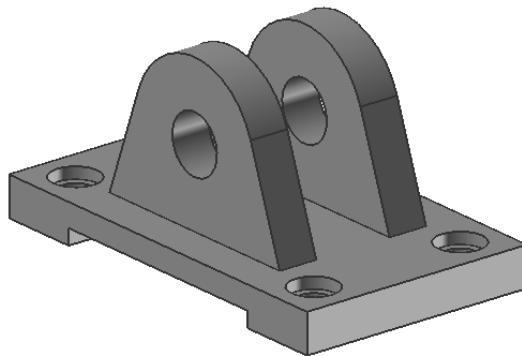


Figure 8-47 The solid model for Tutorial 2

The following steps are required to complete this tutorial:

- Start a new file in NX.
- Draw the sketch for the base feature and extrude the same, refer to Figures 8-48 and 8-49.
- Create the rectangular pocket using the **Pocket** tool, refer to Figure 8-50.

- d. Create the rectangular pad using the **Pad** tool, refer to Figure 8-51.
- e. Create the edge blend on the pad feature using the **Edge Blend** tool, refer to Figure 8-52
- f. Create the draft on the edge blend surface of the rectangular pad using the **Draft** tool, refer to Figure 8-53.
- g. Create the cylindrical pocket in the rectangular pad feature using the **Pocket** tool, refer to Figure 8-54.
- h. Mirror the features with respect to the datum plane using the **Instance Feature** tool, refer to Figure 8-55.
- i. Create a counterbore hole using the **Hole** tool, refer to Figure 8-56.
- j. Create the rectangular array of a hole feature using the **Instance Feature** tool, refer to Figure 8-57.
- k. Save the file.

Starting a New File

1. Start a new file using the **Master Model Template** and save it as *c08tut1* in the *c08* folder.

Creating the Base Feature of the Model

1. Choose the **Sketch** button from the **Form Feature** toolbar; the **Sketch Icon Options** are displayed. Select the **XC-YC Plane** as the sketching plane and create the sketch for the base feature, as shown in Figure 8-48.
2. Choose the **Finish Sketch** button from the **Sketcher** toolbar and exit the sketcher task environment. Next, you need to extrude the sketch created for the base feature.
3. Choose the **Extrude** button from the **Form Feature** toolbar; you are prompted to select the section geometry.
4. Select the sketch created for the base feature and enter the value as **10** in the **End** edit box.
5. Choose the **OK** button to create the base feature. Turn off the display of all the entities, except the base feature. The resulting model is shown in Figure 8-49.

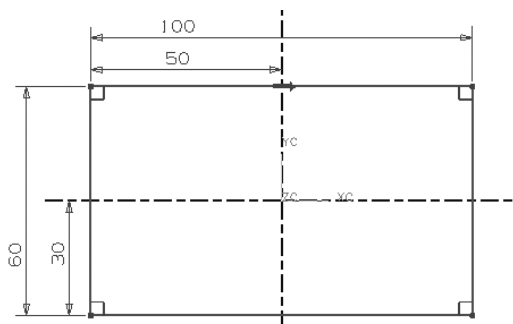


Figure 8-48 Sketch for the base feature

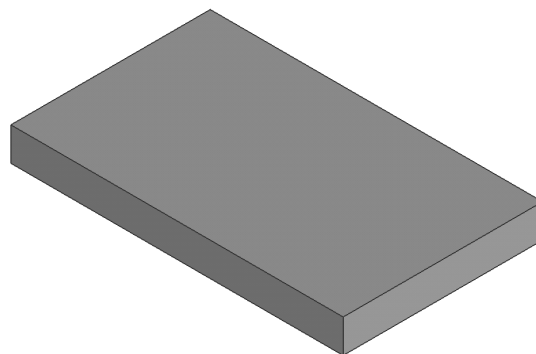


Figure 8-49 Base feature of the model

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 **Form Feature** toolbar; the **Pocket** dialog box is displayed and you are prompted to select the pocket type.
2. Choose the **Rectangular** button; the **Rectangular** dialog box is displayed and you are prompted to select the planer placement face.
3. Select the front face of the model that has the length as 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 in the edit boxes as length **60**, width **4**, and depth **60**.
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 of 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 8-50.

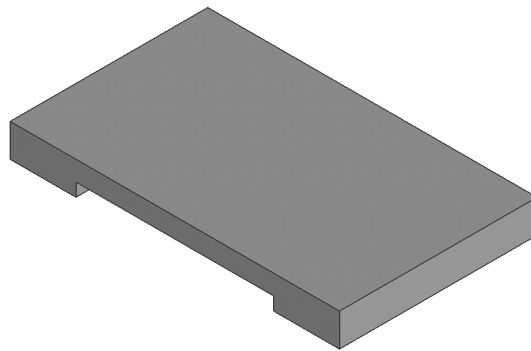


Figure 8-50 The model after creating the rectangular pocket feature

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 **Form 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 planer 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 in the edit boxes as length **40**, width **10**, and depth **40**.
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 8-51.

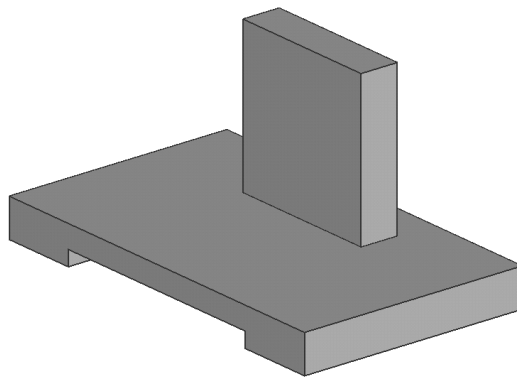


Figure 8-51 The model after creating the rectangular pad feature

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 **20** as the value in the **Set1 R** 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 8-52.

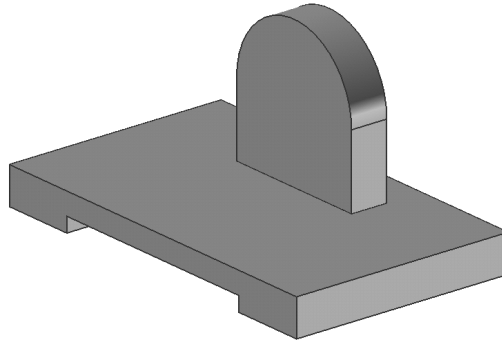


Figure 8-52 The model after creating the edge blend feature

Creating the Draft Feature

Next, you need to create the draft feature on the side faces of the pad feature. This is done using the steps given next.

1. Choose the **Draft** tool from the **Feature Operation** toolbar; the **Draft** dialog box is displayed. By default, the **Draft From Stationary Plane** button is chosen from the **Type** area.
2. Choose the **Draft Tangent to Faces** button from the **Type** area; you are prompted to specify the draw direction. Select the **ZC Axis** option from the **Vector Method** 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 as **20** in the **Set1 A** edit box. Choose the **Apply** button and then the **Cancel** button. The resulting model is shown in Figure 8-53.

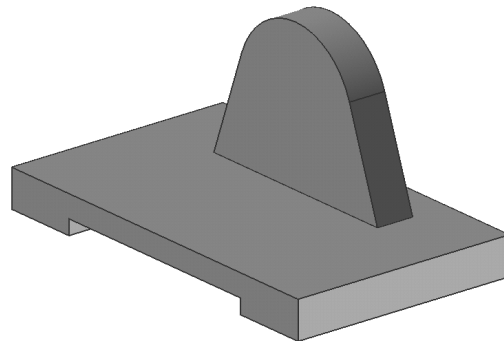


Figure 8-53 The model after creating the taper feature

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 **Form 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 in the edit boxes as pocket diameter 15 and depth 10. 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 shown in Figure 8-54.

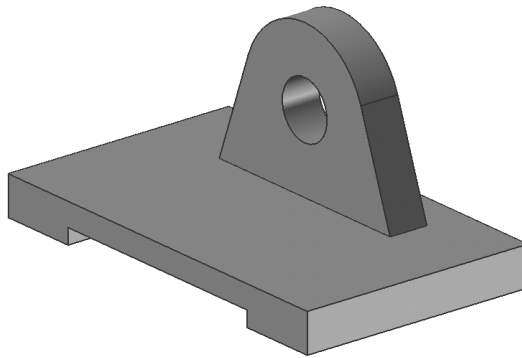


Figure 8-54 The model after creating the cylindrical pocket feature

Mirroring Features

1. Choose the **Instance Feature** tool from the **Feature Operation** toolbar; the **Instance** dialog box is displayed.
2. Choose the **Mirror Feature** button; the **Mirror Feature** dialog box is displayed. By default, the **Feature to Mirror** button is chosen from the **Selection Steps** area and you are prompted to select the feature to mirror.
3. Select the **RECTANGULAR_PAD** feature and all features below it from the **Features in**

Part selection area and choose the **Add** button; the features are added in the **Features in Mirror** area.

4. Choose the **Mirror Plane** button from the **Selection Steps** area; you are prompted to select the plane to mirror. Press the SHIFT+CTRL+K keys to invoke the **Unblank Selected** tool and turn on the display of the **XC-ZC Plane**. When you exit the **Unblank Selected** tool, the **Mirror Feature** dialog box is redisplayed.
5. 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 8-55.

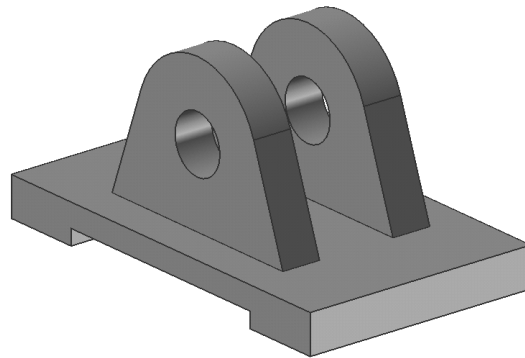


Figure 8-55 The model after creating the mirror feature

Creating the Counterbore Hole Feature on the Base Feature

1. Choose the **Hole** button from the **Form Feature** toolbar; the **Hole** dialog box is displayed and you are prompted to select the placement face.
2. Select the top face of the base feature as the placement face. Next, you need to select the hole type.
3. Choose the **Counterbore** button from the **Type** area.
4. Enter **12** as the value in the **C-Bore Diameter** edit box, **3** in the **C-Bore Depth** edit box, **8** in the **Hole Diameter** edit box, and **10** in the **Hole Depth** edit box.
5. Choose the **Apply** button; the **Positioning** dialog box is displayed. Choose the **Perpendicular** button from the **Positioning** dialog box and select the edge of dimension 100 mm of the base feature. Enter **10** as the value in the **Current Expression** edit box and press ENTER.
6. Again, choose the **Perpendicular** button from the **Positioning** dialog box and from the top face of the base feature, select the edge that measures 60 mm. Enter **10** as the value in the **Current Expression** edit box. Choose the **OK** button. The counterbore hole is created, as shown in Figure 8-56.

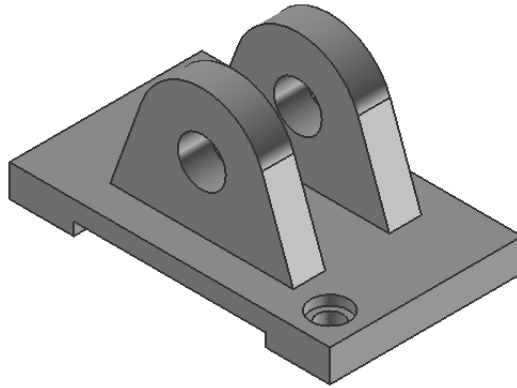


Figure 8-56 The model after creating the hole feature

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 8-57.
5. Choose **File > Close > Save and Close** to save and close the part file.

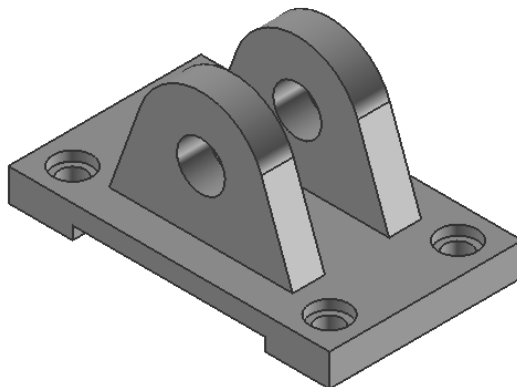


Figure 8-57 The final model

Tutorial 3

In this tutorial, you will create the model shown in Figure 8-58. The dimensions of this model are shown in Figure 8-59. After creating the solid model, save it in the name `\\NX 4\\c08\\c08tut3.prt`
(Expected Time: 30 min)

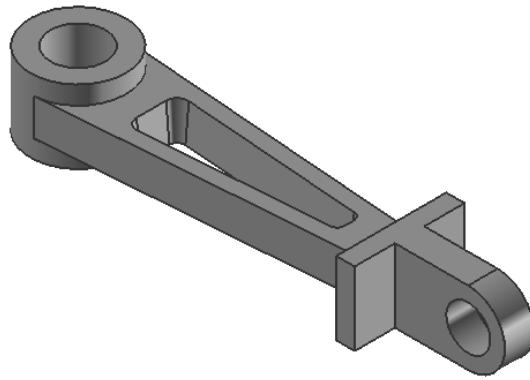


Figure 8-58 The solid model for Tutorial 3

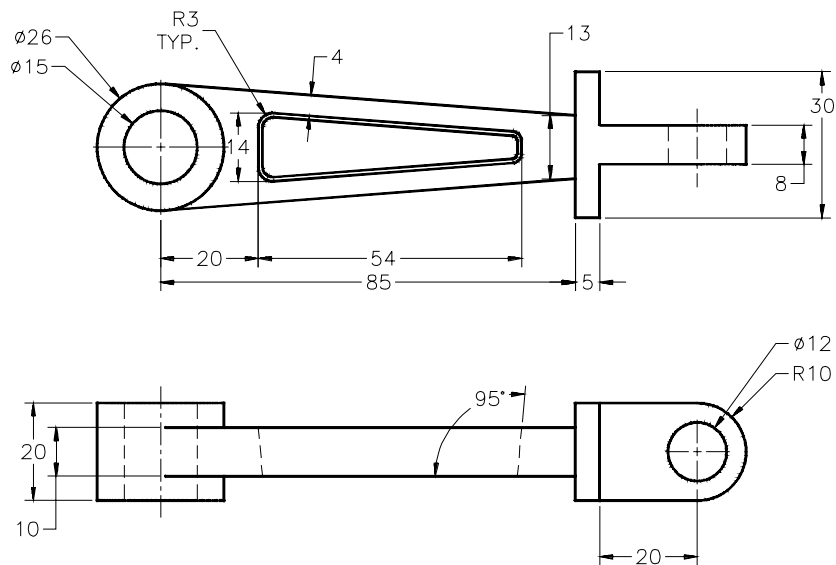


Figure 8-59 Views and dimensions of the model for Tutorial 3

The following steps are required to complete this tutorial:

- Start a new file in NX.
- Draw the sketch for the base feature and extrude the same, refer to Figure 8-61.

- c. Draw the sketch for the second feature and extrude the same, refer to Figure 8-63.
- d. Create the cylindrical pocket in the second feature using the **Pocket** tool, refer to Figure 8-64.
- e. Create the cut extrude feature, refer to Figure 8-65.
- f. Create the rectangular pad using the **Pad** tool, refer to Figure 8-67.
- g. Create the rectangular pad using the **Pad** tool, refer to Figure 8-68.
- h. Create the edge blend using the **Edge blend tool**, refer to Figure 8-69.
- i. Create the cylindrical pocket in the rectangular pad using the **Pocket** tool, refer to Figure 8-70.
- j. Save the file.

Starting a New File

1. Start a new file using the **Master Model Template** and save it as *c08tut3* in the *c08* folder.

Creating the Base Feature of the Model

1. Invoke the **Sketch** tool and select the **XC-YC Plane** as the sketching plane. Create the sketch for the base feature, as shown in Figure 8-60.
2. Choose the **Finish Sketch** button from the **Sketcher** toolbar and exit the **Sketcher** task environment.
3. Choose the **Extrude** button from the **Form Feature** toolbar; the **Extrude** dialog box is displayed. Select the sketch created for the base feature.
4. Select the **Symmetric Value** option from the **Start** drop-down list of the **Limits** area. Enter 5 as the value in the edit box at the right of the **Start** drop-down list. Choose the **OK** button to create the base feature. Turn off the display of all the entities, except the base feature. The resulting base feature is shown in Figure 8-61.

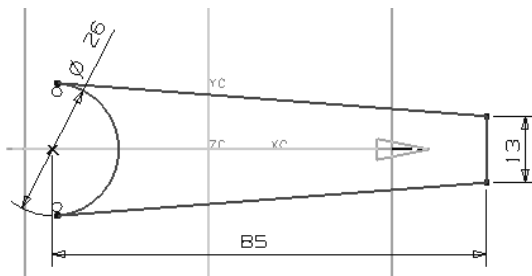


Figure 8-60 Sketch for the base feature

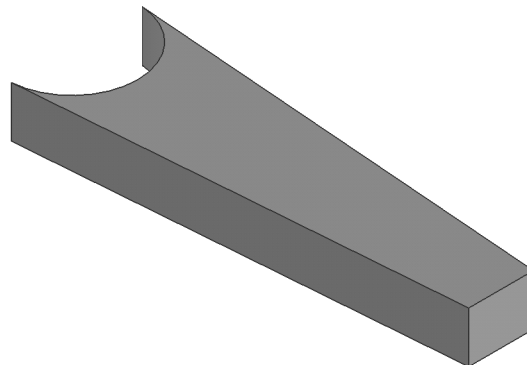


Figure 8-61 Base feature of the model

Creating the Second Feature using the Extrude tool

1. Draw the sketch shown in Figure 8-62 on the **XC-YC Plane**.

2. Choose the **Finish Sketch** button from the **Sketcher** toolbar and exit the **Sketcher Task** environment.
3. Invoke the **Extrude** tool; the **Extrude** dialog box will be displayed. Select the sketch created for the base feature.
4. Select **Unite** from the **Boolean** drop-down list in the **Extrude** dialog box.
5. Select the **Symmetric Value** option from the **Start** drop-down list of the **Limits** area and enter **10** as the value in the edit box on the right of the **Start** drop-down list.
6. Choose the **OK** button to create the second feature. The resulting model is shown in Figure 8-63.

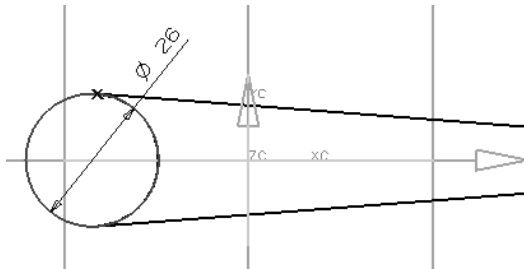


Figure 8-62 Sketch for the second feature

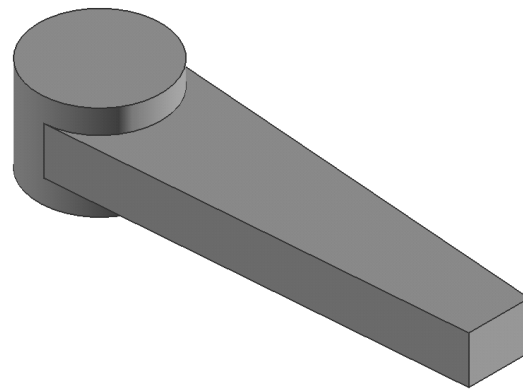


Figure 8-63 The model after creating the second feature

Creating the Cylindrical Pocket Feature

Create the cylindrical pocket feature using the following steps:

1. Choose the **Pocket** tool from the **Form 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 shown in Figure 8-64.

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. Choose the **Sketch** button from the **Form Feature** toolbar and select the **top** surface of the base feature as the sketching plane from the **Sketch Icon Options**. Create the sketch for the cut extrude feature, as shown in Figure 8-65.

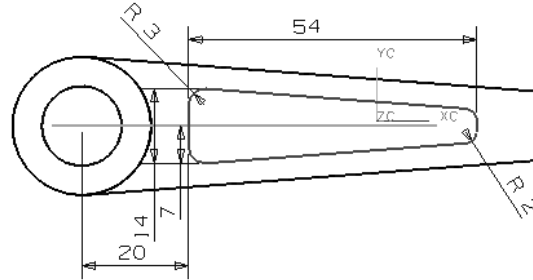
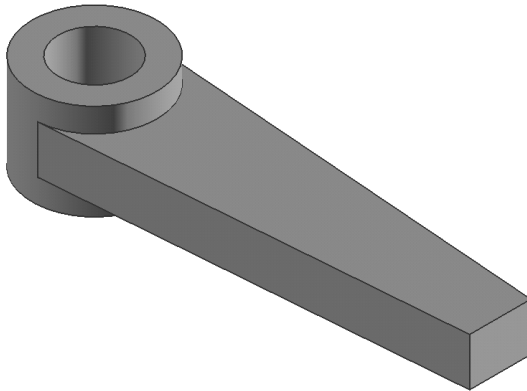


Figure 8-64 Model after creating the pocket feature **Figure 8-65** Sketch for the cut extrude feature

2. Choose the **Finish Sketch** button from the **Sketcher** toolbar and exit the **Sketcher Task** environment.
3. Invoke the **Extrude** tool from the **Form Feature** toolbar; the **Extrude** dialog box is displayed and you are prompted to select the section geometry to extrude. Select the sketch.
4. Reverse the direction of extrusion and select the **Through All** termination option from the **End** drop-down list.
5. Select **Subtract** from the **Boolean** drop-down list.
6. Select the **Draft** check box and enter **5** as the angle value.
7. Choose the **OK** button to create the cut extrude feature.
8. Turn off the display of all the entities, except the model. The model, after creating the cut extrude feature, is shown in Figure 8-66.

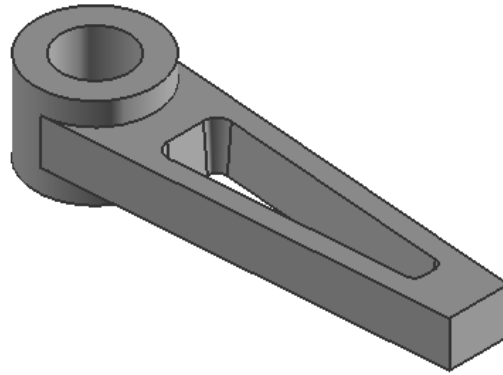


Figure 8-66 The model after creating the general pocket feature

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 **Form 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. Chose 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 shown in Figure 8-67.

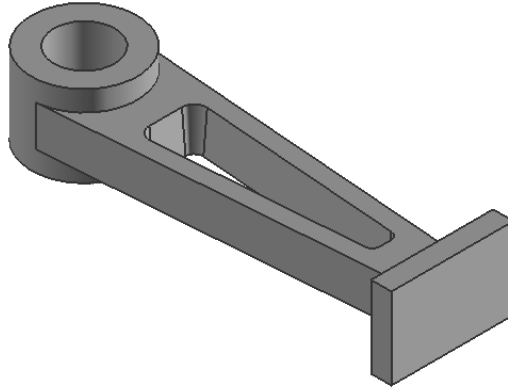


Figure 8-67 The model after creating the pad feature

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 previous rectangular pad feature.

1. Choose the **Pad** tool from the **Form 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 planer 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 values in the edit boxes as length **20**, width **8**, and depth **30**.
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 shown in Figure 8-68.

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 **Set1 R** 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 8-69.

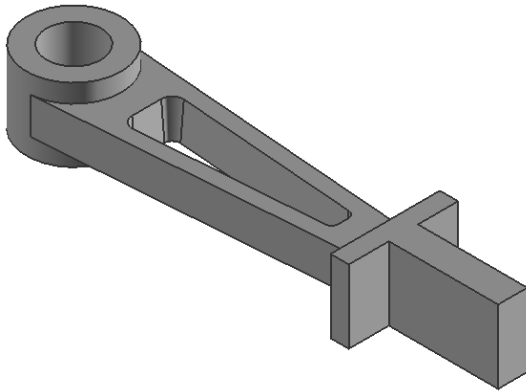


Figure 8-68 Model after creating the pad feature

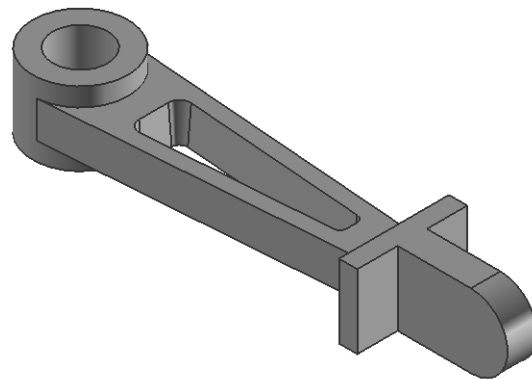


Figure 8-69 Model after creating the edge blend feature

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 **Form 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 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 8-70.

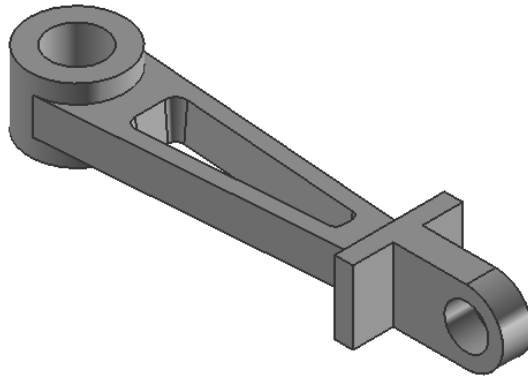


Figure 8-70 The final model for Tutorial 3

7. Choose **File > Close > Save and Close** to save and close the part file.

Self-Evaluation Test

Answer the following questions and then compare your answers with those given at the end of the chapter:

1. The advanced feature tools are used to reduce the time taken in creating the design. (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 **Draft 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 choose whether to taper only that instance or all the instances in the array features.
9. The **Draft 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. **Circular**
 - b. **Rectangle**
 - c. **Square**
 - d. None of the above
2. Which tool is used to create the cutout feature?
 - a. **Pad**
 - b. **Pocket**
 - c. **Boss**
 - d. None of the above
3. Which of the following pocket types needs a sketch?
 - a. **Cylindrical pocket**
 - b. **General pocket**
 - c. **Rectangular pocket**
 - d. None of the above
4. Which draft type is used to create a draft on the nonuniform edges?
 - a. **Face taper**
 - b. **Edge taper**
 - c. **Split-line taper**
 - d. None of the above
5. What 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 of the above
6. Which tool is used to create the cylindrical extrusion feature?
 - a. **Boss**
 - b. **Pad**
 - c. **Pocket**
 - d. None of the above
7. The **Vector Method** 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 through out 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 8-71. The drawing views and dimensions of the model are given in Figure 8-72. After creating the model, save it in the name specified below.

\\NX 4\\c08\\c08exr1.prt

(Expected Time: 30 min)

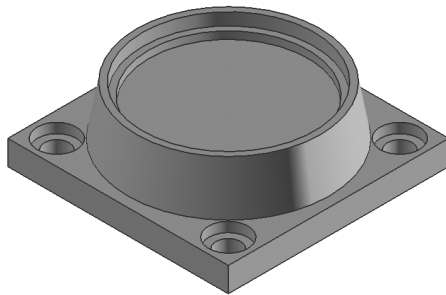


Figure 8-71 The solid model for Exercise 1

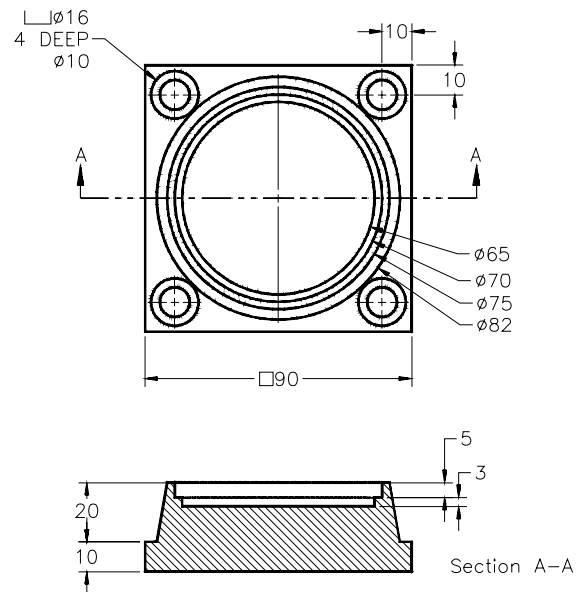


Figure 8-72 Views and dimensions for Exercise 1

Exercise 2

Create the model shown in Figure 8-73. The drawing views and dimensions of the model are given in Figure 8-74. After creating the model, save it in the name specified below.

|NX 4|c08|c08exr2.prt

(Expected Time: 30 min)

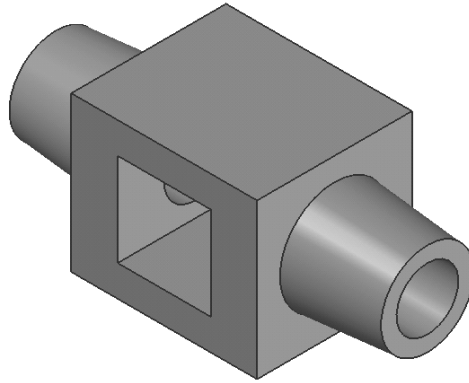


Figure 8-73 The solid model for Exercise 2

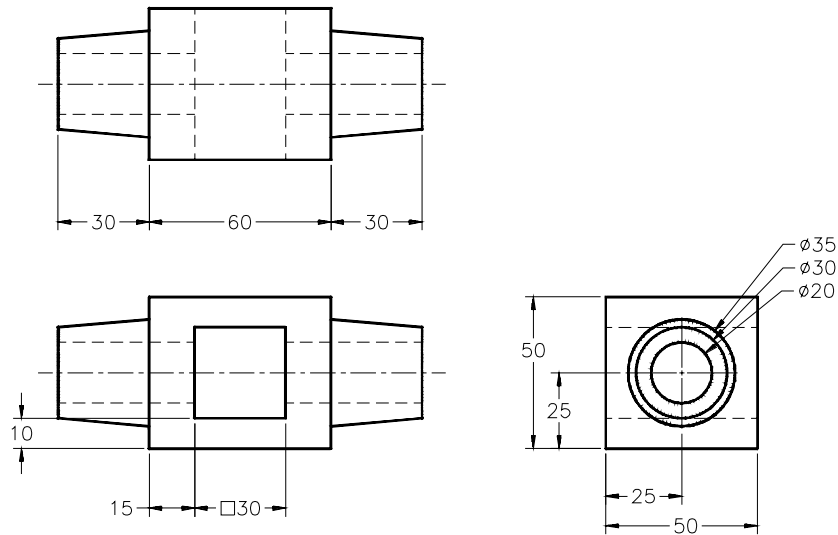


Figure 8-74 Views and dimensions for Exercise 2

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. T, 5. 118, 6. Planar and Conical, 7. 5, 8. Placed, 9. Conical, 10. Guide way