



# **Chapter 4**

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## ***Editing, Extruding, and Revolving Sketches***

### **Learning Objectives**

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

- *Edit sketches using the editing tools.*
- *Edit sketched entities by dragging.*
- *Convert sketches into base features by extruding and revolving.*
- *Hide and show objects.*
- *Rotate the view of a model dynamically in 3D space.*
- *Change the view and display of models.*

## EDITING SKETCHES

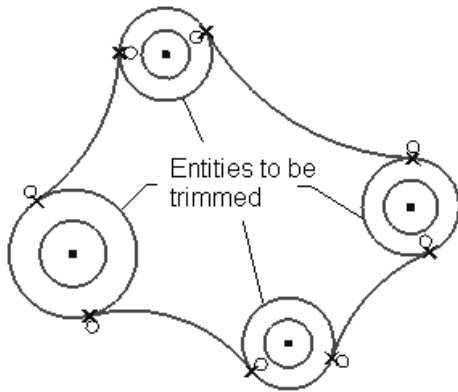
Editing is a very important part of sketching in any design or manufacturing program. You need to edit the sketches during various stages of a design. NX provides you with a number of tools that can be used to edit the sketched entities. These tools are discussed next.

### Trimming Sketched Entities

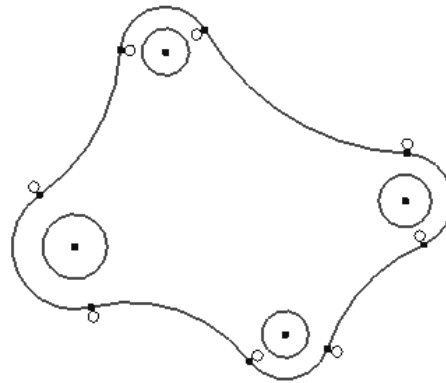
**Menu:** Edit > Curve > Quick Trim  
**Toolbar:** Sketch Tools > Quick Trim



This tool enables you to remove a portion of a sketch by chopping it off. Figure 4-1 shows the sketched entities before trimming entities and Figure 4-2 shows the sketch after trimming entities. Note that when used on an isolated entity, this tool deletes it.

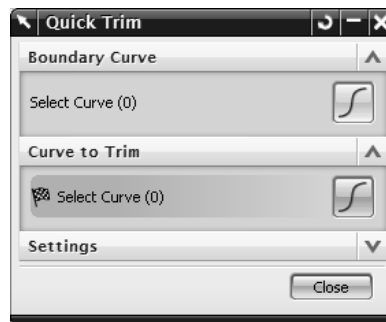


**Figure 4-1** Selecting the entities to be trimmed



**Figure 4-2** Sketch after trimming entities

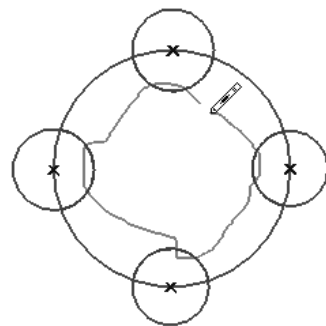
To trim the entities, invoke the **Quick Trim** tool from the **Sketch Tools** toolbar; the **Quick Trim** dialog box will be displayed, as shown in Figure 4-3.



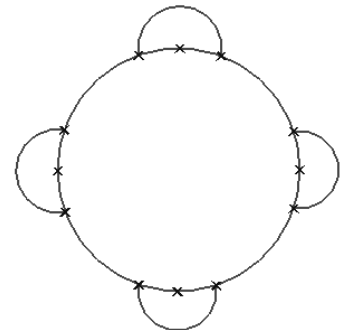
**Figure 4-3** The **Quick Trim** dialog box

Move the cursor over the portion to be trimmed; it will be highlighted. Click to trim the highlighted portion. You will again be prompted to click on the entity to be trimmed. After trimming all entities, press ESC to exit this tool.

To trim multiple entities, press and hold the left mouse button and drag the cursor over the entities to be trimmed. As you move the cursor over them, all the selected entities will be trimmed. Figure 4-4 shows multiple entities being trimmed by dragging the cursor over them and Figure 4-5 shows the resulting sketch.



**Figure 4-4** Dragging the cursor to trim multiple entities



**Figure 4-5** The resulting sketch

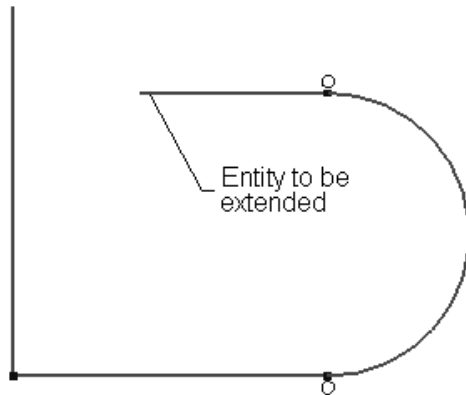
NX also allows you to select a sketched entity as the knife edge and then trim the other entities using this cutting edge. To do so, choose the **Boundary Curve** button from the **Boundary Curve** rollout; you will be prompted to select the boundary curve. Select the curve; this curve will now become the knife edge. Next, choose the **Curve to Trim** button from the **Curve to Trim** rollout; you will be prompted to select the curve to be trimmed. Similarly, select other curves that intersect the boundary curve; the selected curves will be trimmed.

### Extending Sketched Entities

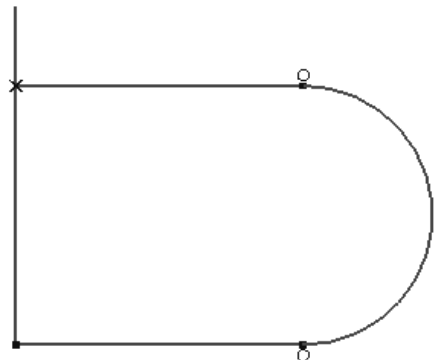
**Menu:** Edit > Curve > Quick Extend  
**Toolbar:** Sketch Tools > Quick Extend



The **Quick Extend** tool enables you to extend or lengthen an open sketched entity up to the next entity that it intersects. Figure 4-6 shows the sketched entity before extending and Figure 4-7 shows the sketch after extending the entity. Note



**Figure 4-6** Selecting the entity to be extended



**Figure 4-7** Sketch after extending the entity

that this tool will not work on an entity that does not intersect with any existing sketched entity when extended.

You can also extend multiple entities by pressing and holding the left mouse button and dragging the cursor over them. All the other options of this tool are same as in the **Quick Trim** tool.

## Making a Corner between Sketched Entities

**Menu:** Edit > Curve > Make Corner  
**Toolbar:** Sketch Tools > Make Corner



You can create a corner between two entities by using the **Make Corner** tool.

Draw sketch before invoking this tool. Next, invoke the **Make Corner** tool from the **Sketch Tools** toolbar; the **Make Corner** dialog box will be displayed, as shown in Figure 4-8. In this dialog box, the **Curve** button in the **Curve** rollout is chosen by default. Select the first curve; you will be prompted to select the second curve. Select the second curve; a corner will be created between the two sketched entities. Figure 4-9 shows the two sketched entities selected for creating corner and Figure 4-10 shows the resultant sketch created using these entities.



Figure 4-8 The **Make Corner** dialog box



Figure 4-9 Two entities selected for creating a corner

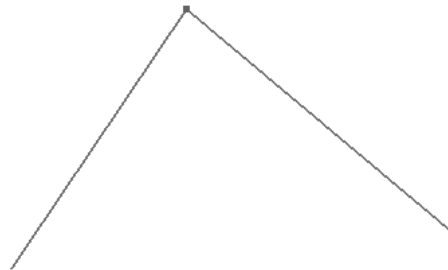


Figure 4-10 Sketch created using the selected entities

## Mirroring Sketched Entities

**Menu:** Insert > Curve from Curves > Mirror Curve  
**Toolbar:** Sketch Tools > Mirror Curve (Customize to add)



This tool enables you to create mirrored copies of the selected sketched entities. Before invoking this tool, you need to draw a line that will be the mirror line. This line will be converted into a reference line after the mirroring operation is completed. To mirror the sketched entities, invoke the **Mirror Curve** tool; the **Mirror Curve**

dialog box will be displayed, as shown in Figure 4-11. The **Mirror Centerline** button is chosen by default in the **Mirror Centerline** rollout. You will be prompted to select a linear object or a plane for the centerline. Select the mirror centerline. The **Curve to Mirror** button will be chosen in the **Curve to Mirror** rollout and you will be prompted to select the curves to mirror. Select the curves to be mirrored and choose the **OK** button; the entities will be mirrored and the mirror line will be converted into the centerline. Figure 4-12 shows the mirror line and entities selected to be mirrored and Figure 4-13 shows the sketch after mirroring entities. The mirror line is automatically converted into a reference line after mirroring.

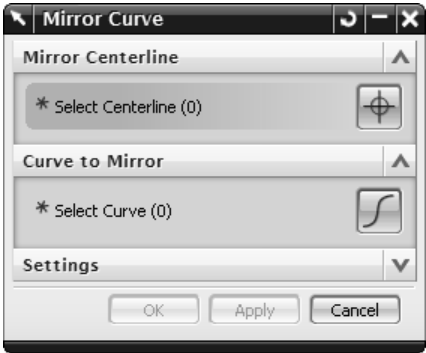


Figure 4-11 The *Mirror Curve* dialog box

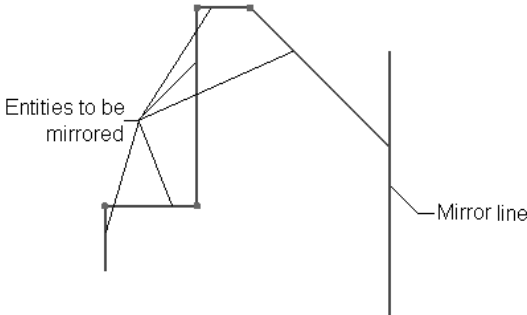


Figure 4-12 Mirror line and entities selected to be mirrored

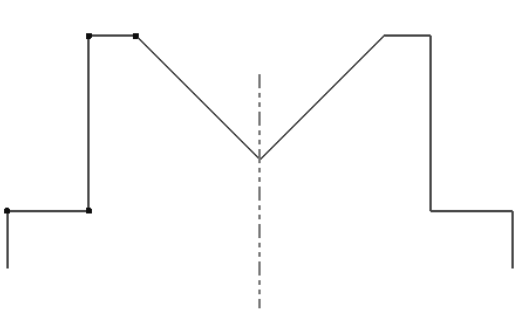


Figure 4-13 Sketch after mirroring entities



**Tip.** After mirroring, if you select and drag any entity from the original portion of the sketch, the same change will also be reflected dynamically in its mirrored portion. However, this relationship ends if you delete the mirror centerline.

### Copying, Moving, and Rotating Sketched Entities

<b>Menu:</b>	Edit > Move Object
<b>Toolbar:</b>	Standard > Move Object (Customize to add)



In NX6, you can dynamically move, rotate, or copy solid objects as well as sketched entities by using the **Move Object** tool. It is a new tool introduced in NX6. To invoke this tool, choose **Edit > Move Object** from the menu bar; the **Move Object** dialog box will be displayed, as shown in Figure 4-14. The options in this dialog box are discussed next.

#### Objects Rollout

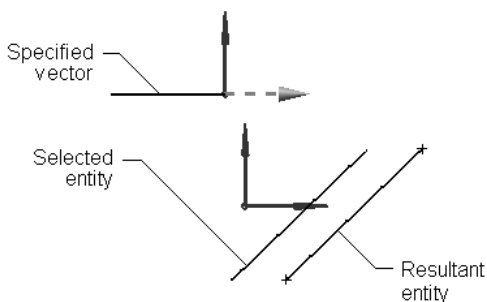
In this rollout, the **Select Object** button is chosen by default. As a result, you will be prompted to select objects to move. Select the entity to which you want to move, rotate, or copy.

## Transform Rollout

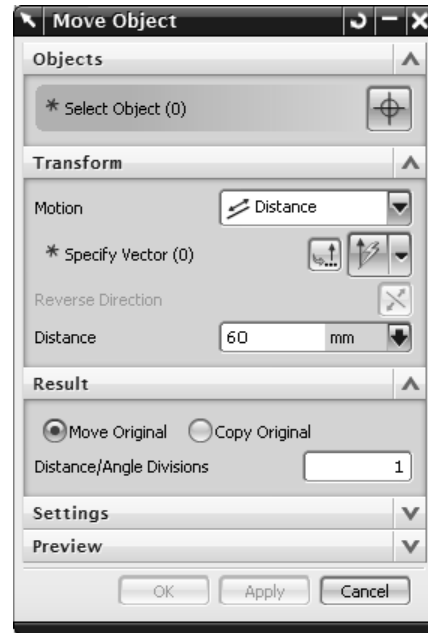
The **Transform** rollout is used to transform object using various methods. The **Motion** drop-down list in this rollout is used to specify the transform method to be used. The options in the **Transform** rollout keep on changing according to the options selected from the **Motion** drop-down list. These options are discussed next.

### Distance

This option is used to move the selected object by a specified distance along the direction of the selected vector. Select this option from the **Motion** drop-down list; you will be prompted to specify the direction vector along which the selected object will be moved or copied. Select an existing edge, sketched entity, datum axis, or datum plane to specify the vector direction. Next, enter the required value in the **Distance** edit box and then press ENTER ; the selected object will be moved or copied by the specified distance value along the specified vector direction. Also, the preview of the entity will be displayed in the graphic window, as shown in Figure 4-15. You can flip the direction of the vector by choosing the **Reverse Direction** button or by double-clicking on the green arrowhead displayed in the graphic area.



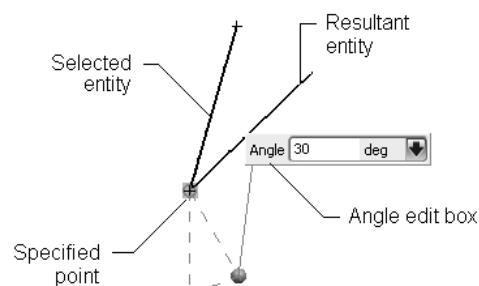
**Figure 4-15** Entity transformed by using the **Distance** option



**Figure 4-14** The **Move Object** dialog box

### Angle

The **Angle** option in the **Motion** drop-down list is used to rotate the selected entity with respect to the specified point. Select this option; you will be prompted to select the object to infer point. Select the point about which you want change the orientation of the object; you will be prompted to select the objects to move. Select the required entity to orient. Next, enter the angle value in the **Angle** edit box and press ENTER. The selected object will be oriented by the specified angle with respect to the specified point. Also, the preview of the resultant entity will be displayed in the graphic window, as shown in Figure 4-16.



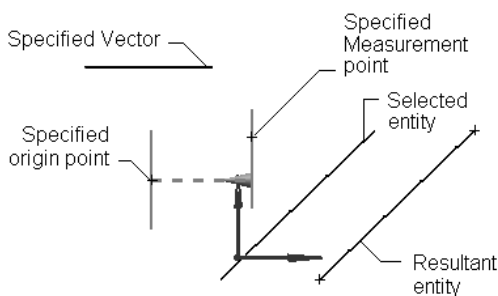
**Figure 4-16** Entity transformed by using the **Angle** Option

### Distance between Points

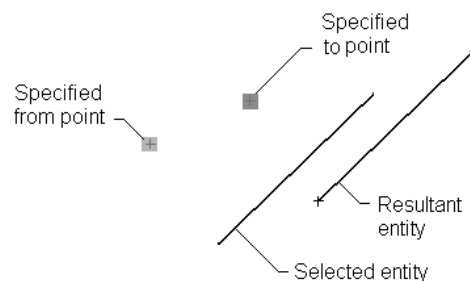
This option is used to move entity from one specified point to another specified point along the specified vector. Select this option from the **Motion** drop-down list; you will be prompted to select the object to infer point. Also, the **Specify Origin Point** option will be highlighted in the **Transform** rollout. The origin point is a point relative to which you want to move the selected entity. Select a point to define the origin point; the **Specify Measurement Point** option will be highlighted in the **Transform** rollout. The measurement point is used to define the distance along the specified vector. Specify the measurement point by selecting another point upto which you want to move the entity; the **Specify Vector** option will be highlighted in the **Transform** rollout. Select an existing edge, sketched entity, datum axis, or datum plane to specify the vector direction; you will be prompted to select the object. Select the entity that you want to move; the preview of the selected entity will be displayed in the graphic window, as shown in Figure 4-17. You can flip the direction of the vector by choosing the **Reverse Direction** button or by double-clicking on the green arrowhead displayed in the graphic area. The shifted distance will be displayed in the distance edit box. You can also overwrite the required distance in this edit box.

### Point to Point

This option is used to move the selected entity from one position to another with respect to one specified point to another. Select this option from the **Motion** drop-down list; you will be prompted to select the object to infer point. Also, the **Specify From Point** option will be highlighted in the **Transform** rollout. Specify the start point of the moving entity; you will be prompted to select the object to infer point. Also, the **Specify To Point** option will be highlighted in the **Transform** rollout. Next, specify the point upto which you want to move the entity; you will be prompted to select the object to move. Next, select the entity upto which you want to move an entity; the selected entity will be moved as specified, and the preview of the entity will be displayed in the graphic window, as shown in Figure 4-18.



**Figure 4-17** Entity transformed by using the *Distance between Points* option



**Figure 4-18** Entity transformed by using the *Point to Point* option

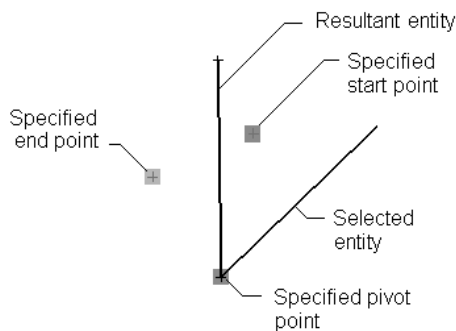
### Rotate by Three Points

This option is used to rotate an entity by using three points. Select this option from the **Motion** drop-down list; you will be prompted to select the object to infer point. Also, the **Specify Pivot Point** option will be highlighted in the **Transform** rollout. Next, specify

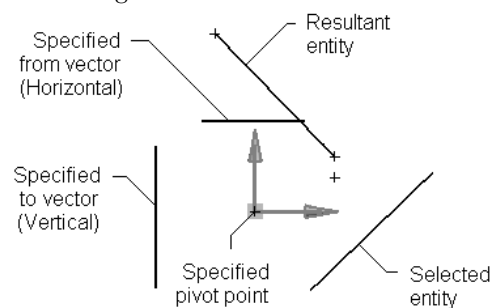
the pivot point relative to which the entity will change its orientation; The **Specify Start Point** area will be highlighted in the **Transform** rollout. As a result, you will be prompted to select the object to infer point. Specify the start point from where the orientation will start; you will be prompted to select the object to infer point and also the **Specify End Point** option will be highlighted in the **Transform** rollout. Specify the end point; you will be prompted to select the object to rotate. Select the entity to be rotated; the preview of the resultant entity will be displayed in the graphic window, as shown in Figure 4-19.

### Align Axis to Vector

This option is used to rotate an object from one specified vector to another specified vector about the specified pivot point. Select this option from the **Motion** drop-down list; you will be prompted to select the object to infer vector. Next, specify the reference vector from which the rotation angle will be measured. After specifying the reference vector for the rotation angle; you will be prompted to select the object to infer a vector. Specify the reference vector for the rotation angle. Next, choose the **Inferred Point** button from the **Specify Pivot Point** option to choose the pivot point; else, it will take the pivot point automatically. Choose the pivot point; you will be prompted to select the object to move. Select the object to be moved; the preview of the resultant entity will be displayed in the graphic window, as shown in Figure 4-20.



**Figure 4-19** Entity transformed by using the **Rotate by Three Points** option

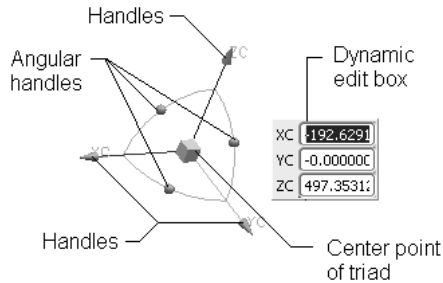


**Figure 4-20** Entity transformed by using the **Align Axis to Vector** option

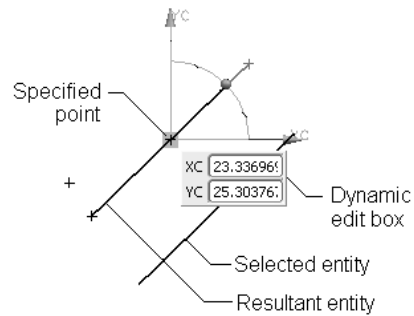
### Dynamic

This option is used to move or rotate an object dynamically. Select this option from the **Motion** drop-down list; a dynamic triad will be displayed at origin, as shown in Figure 4-21. Choose the **Object** button from the **Objects** rollout and select the object to move, or rotate; the triad will be placed at the center of the selected object. You can use this triad to move or rotate the selected entity dynamically by using its handles or angular handles. If you drag the handles, the object will move linearly. If you drag the angular handles, the object will move angularly. Also, you can place the selected object from one point to another by selecting the center of the triad and then clicking at the point where you want to place the object. Note that instead of dragging the handles, you can specify the values in their respective edit boxes, Refer to Figure 4-22. You can select **Move Handles Only** check box in the **Transform** rollout to move the dynamic triad only.





**Figure 4-21** Dynamic triad



**Figure 4-22** Entity transformed by using the *Dynamic* option

## Result Rollout

The options in this rollout are discussed next.

### Move Original

This radio button is selected by default. As a result, the original entity itself will be transformed.

### Copy Original

This radio button toggles with the **Move Original** radio button. Select the radio button, if you want to create multiple copies of the entity such that the original entity is retained in its position.

### Distance/Angle Division

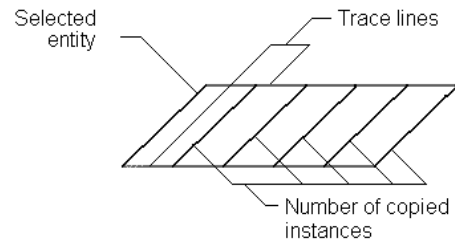
This edit box is used to divide the specified distance or the angle value according to the numbers specified in it.

### Number of Unassociative Copies

This edit box will be available only when the **Copy Original** radio button is selected. In this edit box you can specify a number of instances to be created, excluding the original one.

## Settings Rollout

In this rollout, the **Create Trace Lines** check box is clear by default. If you select this check box, the trace lines of object will be created whenever the object shifts from one position to another, as shown in Figure 4-23.



**Figure 4-23** Trace lines is created

## Transforming Sketched Entities

**Toolbar:** Standard > Transform (*Customize to add*)



NX allows you to perform editing operations on sketches using the **Transform** tool. To invoke this tool, choose the **Transform** button from the **Standard** toolbar; the **Transform** dialog box will be displayed, as shown in Figure 4-24 and you will be prompted to select objects to transform.

The **Transform** dialog box is used to select objects to perform a particular operation. Various rollouts in this dialog box are discussed next.

### Objects Rollout

The options in this rollout are used to define the selection methods, which are discussed next.

#### Select Objects

This button is used to select the objects one by one.

#### Select All

This button is used to select all objects in the drawing window such as sketching entities, datum coordinate systems, work planes, and so on.

#### Invert Selection

This button is used to invert the selection.

### Other Selection Methods Rollout

This rollout provides some additional selection methods.

#### Select by Name

Enter the name of the sketch that you need to select in this text box and press the ENTER key; the specified sketch will be selected.

#### Select Chain

This button is used to select the curves in chain. You need to select the first and the last curve from the sketch; all the sketched entities of the sketch (lines, arcs, and fillets) between these two specified curves will be selected.

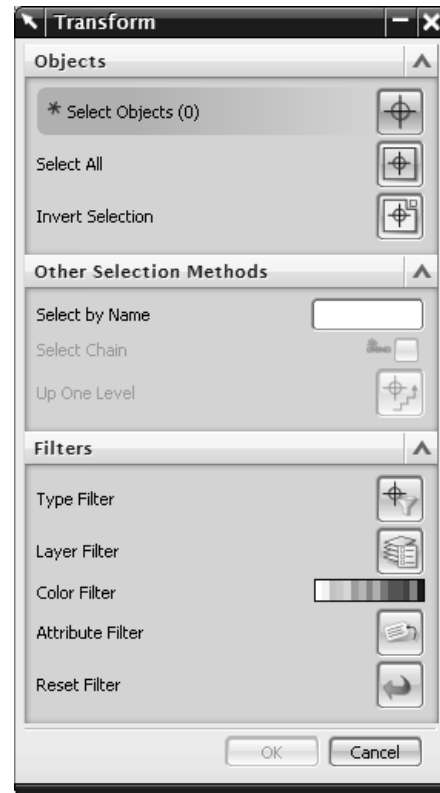


Figure 4-24 The Transform dialog box

### Filters Rollout

This rollout is used to filter out the selection procedure. The options in this rollout have already been discussed in Chapter 2. Select the entities to be transformed and choose the **OK** button; the **Transformations** dialog box will be displayed, as shown in Figure 4-25.

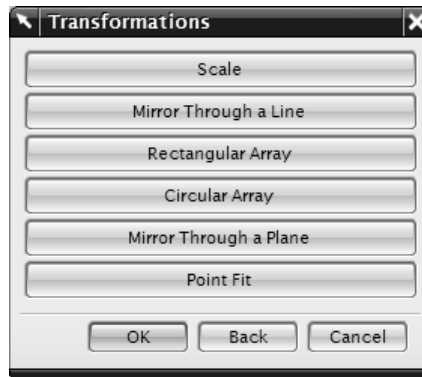


Figure 4-25 The *Transformations* dialog box

Some of the transformation tools in this dialog box are discussed next and the rest will be discussed in the later chapters.

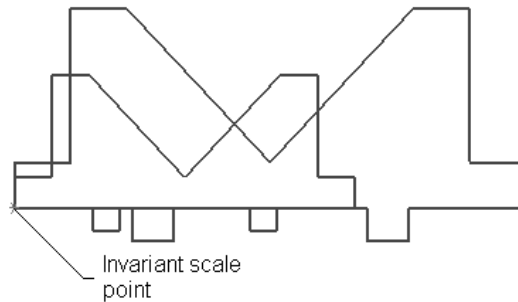
### Scaling the Copied Entities Using the Transformations Dialog Box

The **Scale** option allows you to create scaled copies of the selected objects. If required, you can also scale the objects nonuniformly in the X, Y, and Z directions.

The steps required to scale the sketched entities using the **Transformations** dialog box are discussed next.

1. Choose the **Transform** button from the **Standard** toolbar and then select entities using the **Transform** dialog box. Choose the **OK** button in this dialog box; the **Transformations** dialog box will be displayed.
2. Choose the **Scale** button in the **Transformations** dialog box; the **Point** dialog box will be displayed and you will be prompted to select the object to infer point.
3. Select the base point of the scaling in the drawing window. You can select any point on the screen or any inferred point from the sketch. Next, choose the **OK** button; the modified **Transformations** dialog box will be displayed.
4. Enter the scale factor in the **Scale** edit box. If you need to scale the sketched entities nonuniformly, choose the **Non-uniform Scale** button; the dialog box will modify and display the edit boxes to specify the different value scale factor in the X, Y, and Z directions.
5. After entering the required values, choose **OK**; the **Transformations** dialog box will expand. Choose the **Copy** button; the scaled copy of the selected entities will be displayed.

Figure 4-26 shows the uniformly scaled copy of the original sketch. The scale factor is 1.5.



*Figure 4-26 Uniformly scaled entities*

### Mirroring Entities Using the Transformations Dialog Box

The **Transformations** dialog box allows you to mirror the sketched entities using three methods. These methods are discussed next.

#### Mirroring Using Two Points

Invoke the **Transformations** dialog box and then choose the **Mirror Through a Line** button; the **Transformations** dialog box will get modified. From the modified dialog box, choose the **Two Points** button; the **Point** dialog box will be displayed. Specify two points on the screen to define an imaginary line about which the sketch will be mirrored and choose **OK**. Then, choose the **Move** button in the **Transformations** dialog box if you need to delete the original sketch after mirroring it. Choose the **Copy** button, if you need to retain the original sketch along with mirrored copy.

#### Mirroring Using an Existing Line

Invoke the **Transformations** dialog box and choose the **Mirror Through a Line** button; the **Transformations** dialog box will be modified. From the modified dialog box, choose the **Existing Line** button; the **Transformations** dialog box will be modified again and you will be prompted to select the line to mirror about. Select the line about which the sketch will be mirrored and choose the **Move** or the **Copy** button.

#### Mirroring Using a Point and a Vector

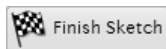
Invoke the **Transformations** dialog box and choose the **Mirror Through a Line** button; the **Transformations** dialog box will be modified. From the modified dialog box, choose the **Point and Vector** button; the **Point** dialog box will be displayed and you will be prompted to select the object to infer point. Select a point on the screen or an inferred point from the sketch; the **Vector** dialog box will be displayed. To mirror about the X-axis, choose the **XC Axis** button. Similarly, to mirror about the Y-axis, choose the **YC Axis** button and choose the **Move** or the **Copy** button.

## Editing Sketched Entities by Dragging

You can also edit the sketched entities by dragging them. Depending upon the type of entity selected and the point of selection, the object will be moved or stretched. For example, if you select a line at any point other than the endpoints and drag the mouse, the line will be moved. However, if you select a line at its endpoint, it will be stretched to a new size. Similarly, if you select an arc at its circumference or its endpoints, it will be stretched. But if you select the arc at its center point, it will be moved. Therefore, editing the sketched entities by dragging depends entirely upon their selection points. The following table gives the details of the operation that will be performed when you drag various entities. Note that while editing the sketched entities using the keypoints, all the related entities will also be moved or stretched.

Entity	Selection point	Operation
Circle	On circumference	Stretch
	Center point	Move
Arc	On circumference/endpoints	Stretch
	Center point	Move
Isolated line or multiple lines selected together	Anywhere other than the endpoints	Move
	Endpoints	Stretch
Curve	Any point other than the keypoints	Move
	Keypoints	Stretch
Rectangle	All lines selected together	Move
	Any one line or any endpoint	Stretch
Ellipse	Anywhere on the circumference or center point	Move

## EXITING THE SKETCHER ENVIRONMENT



After drawing and dimensioning the sketch, you need to exit the **Sketcher** environment and invoke the **Modeling** environment to convert the sketch into a feature. To exit the **Sketcher** environment, choose the **Finish Sketch** button from the **Sketcher** toolbar. On exiting the **Sketcher** environment and entering the **Modeling** environment, you will notice that the **Sketch Tools** toolbar is replaced by the **Feature** and **Feature Operation** toolbars. Also, the dimensions of the sketch are hidden. Note that the current view will change to the trimetric view.

As mentioned in the earlier chapters, most designs are a combination of various sketched, placed, and reference features. The first feature, generally, is a sketched feature. Already you have learned to draw sketches for these base features and add constraints and dimensions to them. After drawing and dimensioning a sketch, you need to convert it into the base feature. NX provides you with a number of tools such as **Extrude**, **Revolve**, and so on to convert these base sketches into base features. In this chapter, you will learn to use the **Extrude** and **Revolve** tools. The remaining tools will be discussed in the later chapters. The base features are created in the **Modeling** environment.

## CHANGING THE VIEW OF THE SKETCH

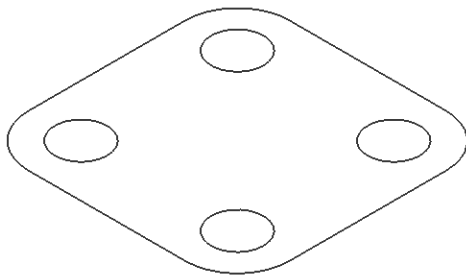
Sometimes you need to change the view of the sketch for better visualization. To change the view, choose the down arrow on the right of the **Trimetric** button in the **View** toolbar; a flyout will be displayed with various view options. Select the required view from this flyout; the current view will be changed to the selected view.

## CREATING BASE FEATURES BY EXTRUDING

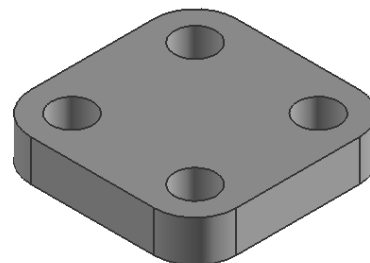
**Menu:** Insert > Design Feature > Extrude  
**Toolbar:** Feature > Extrude



Extrude is defined as the process of creating a feature from a sketch by adding the material along the direction normal to the sketch or any other specified direction. Figure 4-27 shows the isometric view of a closed sketch and Figure 4-28 shows the extruded feature created using this sketch.



*Figure 4-27 Sketch for the extrude feature*



*Figure 4-28 Resulting extruded feature*

When you invoke the **Extrude** tool, the **Extrude** dialog box will be displayed, as shown in Figures 4-29. You will be prompted to select the planar face to sketch or the section geometry to be extruded. If you select the sketch at this stage, the preview of the extruded feature created using the default values will be displayed on the screen. If you select the sketch plane, the **Sketcher** environment will be invoked. Draw the sketch and exit the **Sketcher** environment; the preview of the extruded feature will be displayed in the **Modeling** environment.

## Extrude Dialog Box Options

The options in this dialog box are discussed next.



Figure 4-29 The **Extrude** dialog box

### Section Rollout

The options in this rollout are used to sketch the section or select the section. By default, both the **Sketch Section** and **Curve** buttons will be chosen in this rollout and you will be prompted to select the planar face to sketch or the section geometry to be extruded. These options are discussed next.

#### Sketch Section

This button is used to draw the sketch for extrusion. When you choose this button, the **Create Sketch** dialog box will be displayed and you will be prompted to select the object for the sketch plane. You can select a datum plane or the face of a solid body as the sketching plane.

#### Curve

By default, this button is also chosen from the **Section** rollout and it is used to select the already drawn section sketch.

### Direction Rollout

By default, the direction of extrusion will be normal to the selected section. The buttons in this rollout are used to define the direction of extrusion. These options are discussed next.

**Vector Constructor**

If you choose this button, the **Vector** dialog box will be displayed. You can specify the extrude direction using this dialog box.

**Inferred Vector Drop-down List**

This drop-down list is used to specify the direction of extrusion. The default direction is normal to the selected section.

**Reverse Direction**

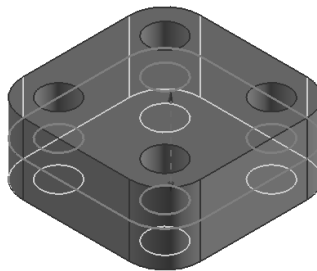
This button is chosen to flip the current extrusion direction.

**Limits Rollout**

The options in this rollout are used to specify the start and termination of the extrusion. These options are discussed next.

**Start Drop-down List**

This drop-down list allows you to specify the start point of the extrusion. You can select the **Value** and **Symmetric Value** options from this drop-down list. The **Value** option allows you to specify the distance from the sketching plane at which the extruded feature will start. You need to enter this value in the **Distance** edit box. If you enter a positive value, it will be taken as the offset value between the sketch and the start of the extrusion feature. If you enter 0, the extruded feature will start from the sketch plane. If you enter a negative value, the extruded feature will start from below the sketch plane. The **Symmetric Value** option allows you to extrude the sketch symmetrically in both the directions of the current sketching plane. When you select this option, the edit boxes on the right of the **Start** and the **End** edit boxes will show identical values and the preview will also be modified dynamically. Figure 4-30 shows the preview of a sketch being extruded symmetrically in both the directions.



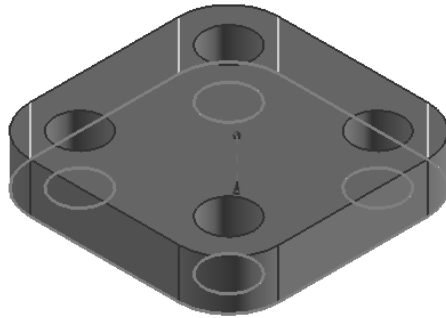
*Figure 4-30 Preview of the symmetric extrusion*

**End Drop-down List**

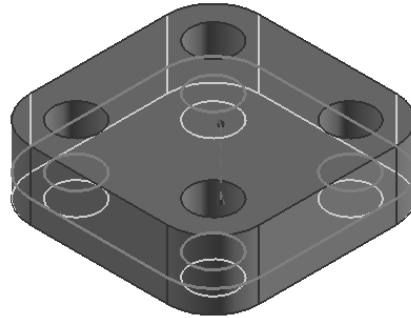
This drop-down list allows you to specify the extrusion termination in the direction of extrusion. For the base feature, only the **Value** and **Symmetric Value** options will be available in this drop-down list. By default, the **Value** option will be selected, and the value entered last will be displayed in the **Distance** edit box. As a result, the sketch will be extruded only in the specified direction. Note that you need to enter a positive value in the **Distance** edit box.



Figure 4-31 shows the preview of the extrusion in only one direction and Figure 4-32 shows the preview of the extrusion with different values in both directions. In this figure, the extrusion value in the upward direction is 10 and in the downward direction is -5.



**Figure 4-31** Preview of the extrusion in only one direction



**Figure 4-32** Preview of the extrusion in two opposite directions with different values



**Tip.** You can also use the start and end drag handles in the preview of the extruded feature to modify the extrusion values in the start and end directions.



#### Note

The other extrusion termination options are discussed in the next chapter.

### Boolean Rollout

Options in this rollout allow you to select the boolean operation that you need to perform. These options in this rollout are discussed in the next chapter.

### Draft Rollout

The options in this rollout are used to specify a draft angle to the extrusion feature. The options in this area will be available only when you select the section to extrude. Various draft options in this rollout are discussed next.

#### Angle

This edit box allows you to specify the draft angle.

#### Draft

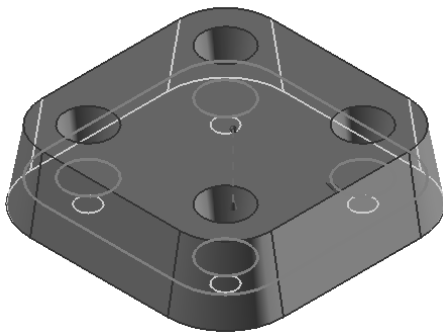
This drop-down list allows you to specify the type of draft to be applied to the feature. The options in this area are discussed next.

#### From Start Limit

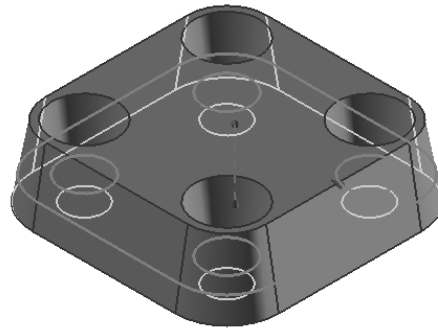
This option adds the draft from the start section to the end section of the extruded feature. As a result, the dimension of the feature at the start section is the same as that of the original sketch and it tapers toward the end section. Figure 4-33 shows the preview of the extruded feature drafted using this option. It is evident from this figure that the bottom section of the extruded feature is the same as that of the original sketch and the feature tapers as it goes toward the top section.

**From Section**

This option is used to taper the extruded surface in such a way that the cross-section of the extruded feature remains the same at the sketching plane, as shown in Figure 4-34.



**Figure 4-33** Preview of the tapered extrusion using the **From Start Limit** option



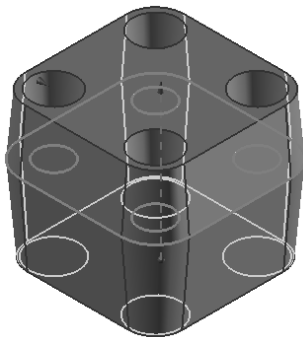
**Figure 4-34** Preview of the tapered extrusion using the **From Section** option

**From Section - Symmetric Angle**

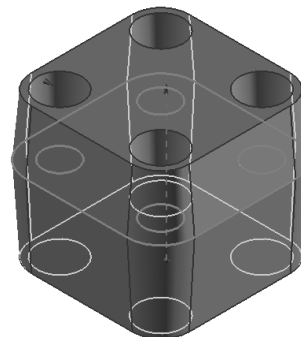
This option is available only when you select the **Symmetric Value** option from the **Limits** rollout or specify the values in both the start and the end directions. This option adds a symmetric taper in both directions of the sketch, as shown in Figure 4-35. In this draft type, if the distance value in one of the directions is more than the other, the section in that direction will also be smaller in size.

**From Section - Matched Ends**

This option is also available only when you select the **Symmetric Value** option from the **Limits** rollout or specify the values in both the start and the end directions. This option tapers the model such that the end sections in both the directions are of similar size, irrespective of the distance values in both directions, as shown in Figure 4-36.



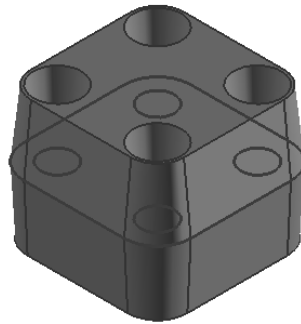
**Figure 4-35** Preview of the extrusion tapered using the **From Section - Symmetric Angle** option



**Figure 4-36** Preview of the extrusion tapered using the **From Section - Matched Ends** option

**From Section - Asymmetric Angle**

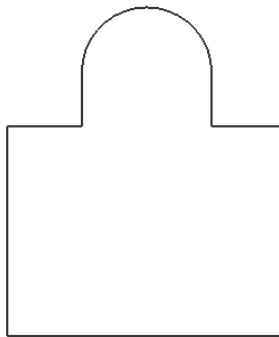
This option is also available only when you select the **Symmetric Value** option from the **Limits** rollout or specify the values in both the start and the end directions. This option adds different tapers in both directions of the sketch, as shown in Figure 4-37. When you select this option, the **Front Angle** and **Back Angle** edit boxes will be available in the **Draft** rollout. The front and back angle values will be applied at the front and back sides of the sketching planes used to create the extruded feature.



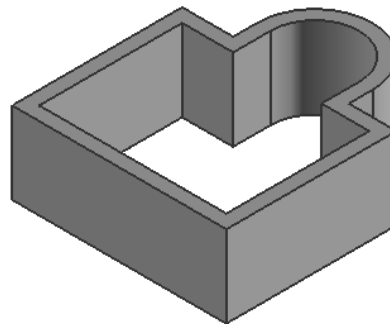
*Figure 4-37 Preview of the extrusion tapered using the **Asymmetric** option*

**Offset Rollout**

NX also allows you to create thin base features by extruding open or closed sketches. For example, refer to the closed sketch shown in Figure 4-38. A thin feature created using this sketch is shown in Figure 4-39. Similarly, Figure 4-40 shows an open sketch and Figure 4-41 shows the resulting thin feature.

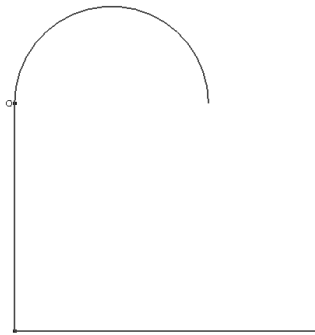


*Figure 4-38 Top view of a single closed sketch*

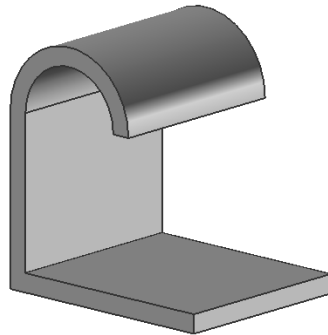


*Figure 4-39 Isometric view of the resulting thin extruded feature*

To create thin features, select the **Offset** rollout in the **Extrude** dialog box; the **Offset** drop-down list will be displayed. This drop-down list contains three offset methods. These methods are discussed next.



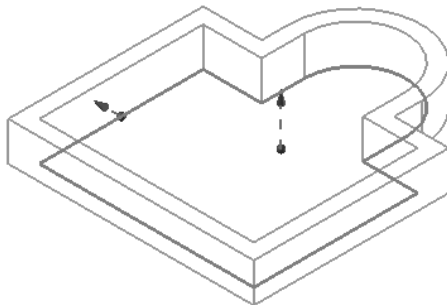
**Figure 4-40** Front view of an open sketch



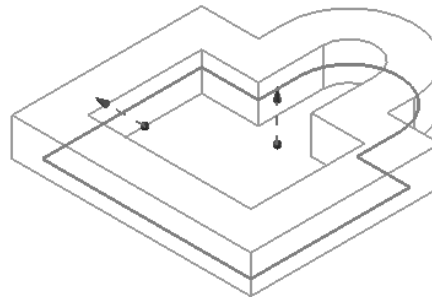
**Figure 4-41** Isometric view of the resulting thin extruded feature

### Two-Sided

This option is used to create a thin feature by offsetting the sketch in two directions. Select this option; the **Start** and **End** edit boxes will be displayed. If you enter the positive value in the **End** edit box, the sketch will offset outward and vice-versa. Figure 4-42 shows the preview of a thin feature with an offset only in the end direction and Figure 4-43 shows the preview of the same feature with an offset in both the directions.



**Figure 4-42** A thin feature with an offset only in the end direction



**Figure 4-43** A thin feature with an offset in the end and start directions



### Note

The display type of models in Figures 4-42 and 4-43 are changed. You will learn more about changing the display type later in this chapter.

### Single-Sided

This option will be enabled only when you create a thin feature using a closed sketch with no nested closed sketch in it. If you select this option, the inner portion of the sketch will be filled automatically. As a result of this, there will be no cavity inside the model. It will be similar to the solid extrusion from inside. However, you can also add some offset to the outer side of the sketch.

### Symmetric

This option is used to offset the material symmetrically on both sides of the sketch to create the thin feature.

### Settings Rollout

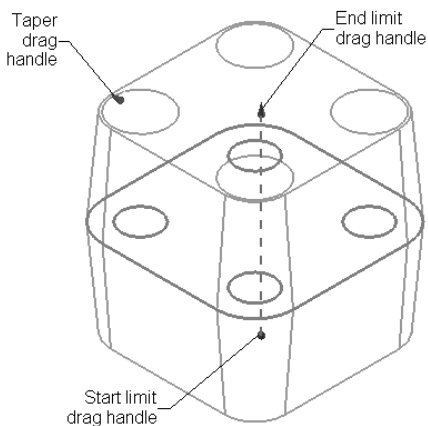
The options in this rollout are used to specify whether you need the extruded feature to be a sheet body or a solid body. To get a solid body, the section must be a closed profile or an open profile with an offset. If you use a **Single-Sided** offset, you will not be able to get a sheet body. You can select the required option from the **Body Type** drop-down list.

### Preview Rollout

This rollout is used to preview the model dynamically while modifying the values in the **Extrude** dialog box. If you select the **Preview** check box, it will allow you to dynamically preview the changes in the model as you modify the values of the extrusion. The **Show Result** button is used to view the final model. The **Undo Result** button is used to go back to the preview mode.

After setting the values in the **Extrude** dialog box, choose **OK** to create the extruded feature and exit the dialog box. If you need to extrude more than one sketches, choose the **Apply** button; the selected sketch will be extruded and the dialog box will be retained. Also, you will be prompted to select the section geometry. Select the other sketch to extrude and choose the **OK** button.

You can also set and modify the values of extrusion using the drag handles that will be displayed in the preview of the extrusion feature, refer to Figure 4-44. The start drag handle will be a filled circle and the end drag handle will be an arrow. To modify the start limit, end limit, or draft angle values, click on their respective drag handles, and then press and hold the left mouse button and drag the mouse. You can also enter the new values in the edit boxes that will be displayed after clicking on the respective handles. To modify the type of limits or taper, right-click on their respective drag handles and select the type from the shortcut menu.



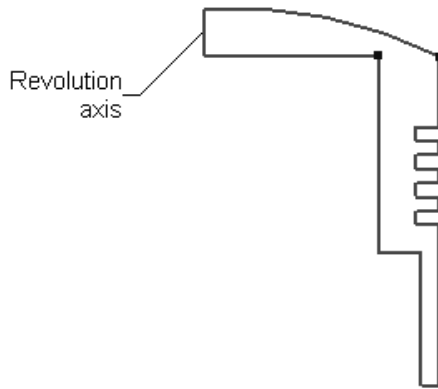
**Figure 4-44** Various drag handles in the preview of the extrusion feature

## CREATING SOLID REVOLVED BODIES

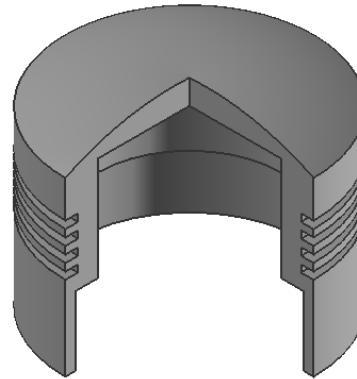
**Menu:** Insert > Design Feature > Revolve  
**Toolbar:** Feature > Revolve



The **Revolve** tool allows you to create a solid body by revolving a sketch around the revolution axis, which could be a sketched line or an edge of an existing feature. Figure 4-45 shows a sketch for the revolved feature and Figure 4-46 shows the Isometric view of the resulting feature revolved through an angle of 270-degree.



**Figure 4-45** Sketch for creating the revolved feature and the revolution axis



**Figure 4-46** Isometric view of the resulting feature revolved through an angle of 270-degree

To convert a sketch into a revolved body, you need to invoke the **Revolve** tool. This tool works in the following three steps:

- Step 1: Select the sketch to be revolved
- Step 2: Select the revolution axis
- Step 3: Specify the revolution parameters

To invoke the **Revolve** tool, choose the revolve button in the **Feature** dialog box; the **Revolve** dialog box will be displayed, as shown in Figure 4-47. The options in this dialog box are same as the options in the **Extrude** dialog box, except the ones that are explained next.

### Axis Rollout

The options in this rollout are used to specify the revolution axis. These options are discussed next.

### Specify Vector

The options in this area are used to specify the revolution axis using the **Vector Constructor** button or the **Inferred Vector** drop-down list.

#### Vector Constructor

When you choose this button, the **Vector** dialog box will be displayed. You can specify the revolution axis by using this dialog box.

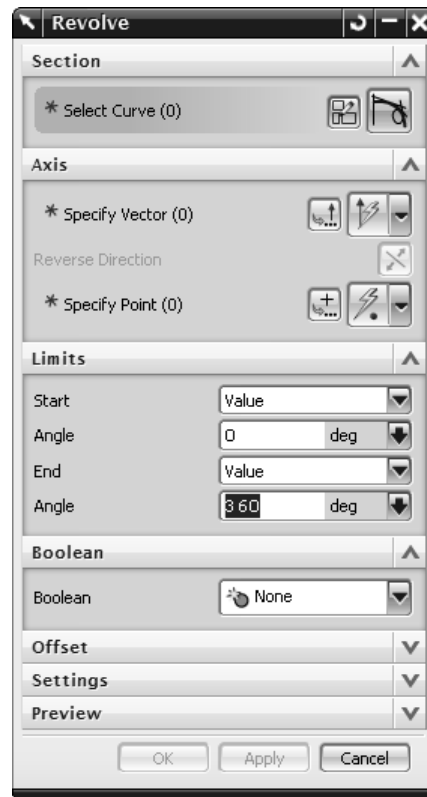


Figure 4-47 The *Revolve* dialog box

#### **Inferred Vector**

This drop-down list is a shortcut to specify the revolution axis.

#### **Reverse Direction**

You can choose this button to flip the direction of revolution.

#### **Specify Point**

The options in this area are used only when you use the vector method to specify the revolution axis.

#### **Point Constructor**

When you choose this button, the **Point** dialog box will be displayed. You can specify the point to define the revolution axis using this dialog box.

#### **Inferred Point**

This drop-down list contains the snap point options that are used to automatically snap the keypoints of the previously sketched entities or features.

## Limits Rollout

The options in this rollout are used to specify the start and termination angles of revolution. These options are discussed next.

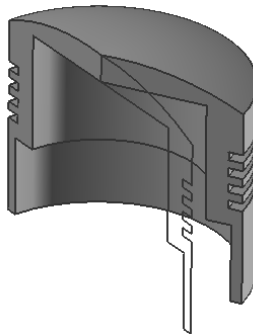
### Start Drop-down List

This drop-down list allows you to specify the start angle of the revolution feature. You can select the **Value** and **Until Selected** options from this drop-down list. The **Value** option allows you to enter the value of the start angle in the **Angle** edit box. You need to enter a positive value of the angle. This value will be taken as the offset value between the sketch and the start of the revolved feature. The **Until Selected** option allows you to start the revolve feature from the selected plane, face, or body. When you select this option, the **Face, Body, Datum Plane** button will be chosen and you will be prompted to select the face, body, or datum plane to start the revolved feature.

### End Drop-down List

This drop-down list allows you to specify the termination angle of the revolution feature. You can select the **Value** and **Until Selected** options from this drop-down list. The **Value** option allows you to enter the value of the end angle in the **Angle** edit box. You need to enter a positive value of the angle. This value will be taken as the offset value between the sketch and the end of the revolved feature. The **Until Selected** option allows you to terminate the revolve feature using the selected plane, face, or body. When you select this option, the **Face, Body, Datum Plane** button will be chosen and you will be prompted to select the face, body, or datum plane to start the revolved feature.

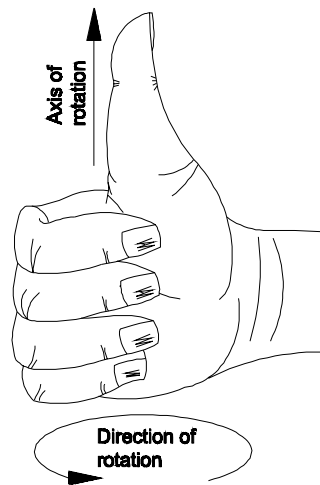
The default value of the end angle is the value that you have used to create the last revolved feature. Figure 4-48 shows a revolved feature with the start angle as 30 degree and the end angle as 180-degree. The sketch used to create this feature is also displayed.



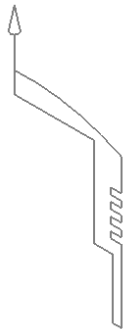
**Figure 4-48** Sketch revolved with start angle as 30 degree and end angle as 180-degree

Note that NX uses the right-hand thumb rule to determine the direction of revolution. This rule states that if the thumb of your right hand points in the direction of the axis of revolution, then the direction of the curled fingers will define the direction of revolution, refer to Figure 4-49. Figure 4-50 shows the sketch and an arrow pointing in the direction of the axis of revolution and Figure 4-51 shows the resulting feature revolved through an angle of 180 degree.

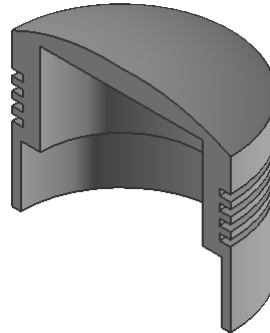




**Figure 4-49** The right-hand thumb rule



**Figure 4-50** Sketch for creating the revolved feature and the direction of the revolution axis

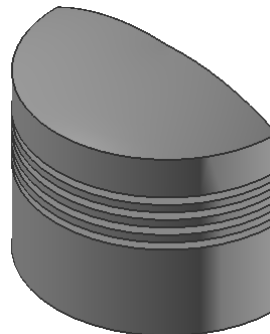


**Figure 4-51** Isometric view of the resulting feature revolved through an angle of 180 degree

Figure 4-52 shows the sketch and an arrow pointing in the direction of the axis of revolution and Figure 4-53 shows the resulting feature revolved through an angle of 180-degree.



**Figure 4-52** Sketch for creating the revolved feature and the direction of the revolution axis

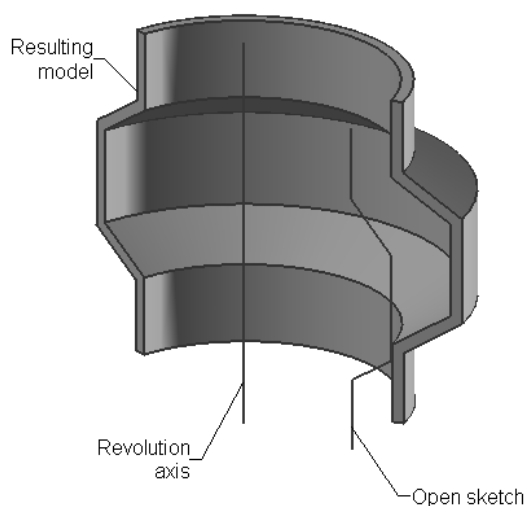


**Figure 4-53** Isometric view of the resulting feature revolved through an angle of 180 degree

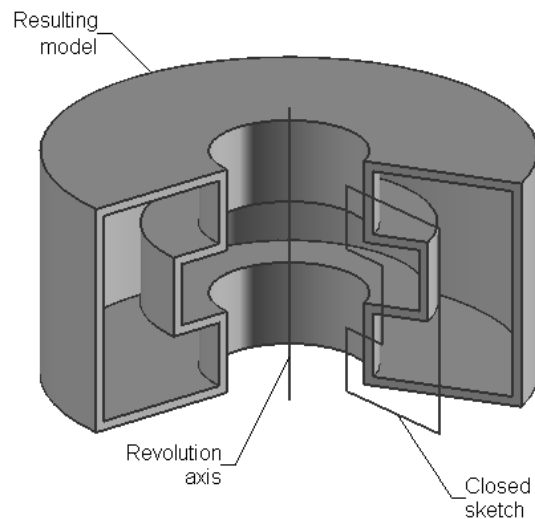
## Offset Rollout

NX also allows you to create thin revolved bodies using the open and closed sketches. This process is similar to that of creating solid extruded features. Select the **Offset** rollout in the **Revolve** dialog box; the rollout will expand and display the **Offset** drop-down list. There is only one option, **Two-Sided**, available in this drop-down list. Select this option; the **Start** and the **End** edit boxes will be available. Enter the start and end offset values in the respective edit boxes. Figure 4-54 shows a thin revolved model with the open sketch and the revolution axis used to create it. In this model, the start angle is 30 degree, the end angle is 180-degree, and the start offset value is 2.

Figure 4-55 shows a thin revolved model with the closed sketch and the revolution axis used to create it. In this model, the start angle is 45 degree, the end angle is 270-degree, and the start offset value is 2.



**Figure 4-54** A thin revolved feature with the original open sketch and the axis of revolution



**Figure 4-55** A thin revolved feature with the original closed sketch and the axis of revolution

## HIDING ENTITIES

**Toolbar:** Utility > Hide



Whenever you create a sketch-based feature, the sketch used to create it is retained on the screen, even after the feature is created. NX allows you to hide the sketches or any other entity on the screen using the **Hide** tool. To invoke this tool, press the CTRL+B keys; the **Class Selection** dialog box will be displayed. Alternatively, you can invoke the **Hide** tool from the **Utility** toolbar. Select the sketch or any other entity from the screen using this dialog box and choose the **OK** button; the selected entities will be hidden.

## SHOWING HIDDEN ENTITIES

**Toolbar:** Utility > Show



To restore the display of the hidden entities, press the SHIFT+CTRL+K keys; the **Class Selection** dialog box and the hidden entities will be displayed. Also, you will be prompted to select the objects to be displayed. Select the entities to be displayed and then choose the **OK** button.



**Note**  
The **Show** tool from the **Utility** toolbar can also be used to restore the display of the hidden entities.

## HIDING ALL ENTITIES USING A SINGLE TOOL

**Toolbar:** Utility > Show and Hide



NX allows you to hide all entities (all datum planes, coordinate systems, sketches, faceted bodies, solid bodies, and so on) from the drawing window, using a single tool. To do so, choose the **Show and Hide** button from the **Utility** toolbar; the **Show and Hide** dialog box will be displayed, as shown in Figure 4-56.

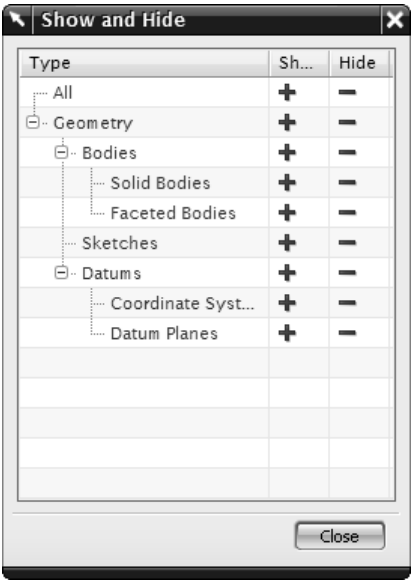


Figure 4-56 The **Show and Hide** dialog box

All entities are divided into three categories, Bodies, Sketches, and Datums. Select the minus sign (-) from the respective rows; the corresponding entities will be hidden. For example, if you need to hide all the sketches in the drawing window, select the minus sign (-) from the **Sketches** row; all the sketches will be hidden.

Similarly, to show hidden entities, select the plus sign (+) from the respective row; all entities under that category will be redisplayed in the drawing window.



The **Show and Hide** tool is very useful while working with complicated models and assemblies, where datums planes, coordinate systems, and sketches are in large numbers.

You can also invoke this tool by pressing the CTRL+W keys.

## ROTATING THE VIEW OF A MODEL IN 3D SPACE

**Toolbar:** View > Rotate



NX provides you with an option of rotating the view of a solid model freely in 3-dimensional (3D) space. This enables you to visually maneuver around the solid model and view it from any direction. To do so, choose the **Rotate** button from the **View** toolbar; the cursor changes to the rotate view cursor and you will be prompted to drag the cursor to rotate the model. Next, press and hold the left mouse button and drag the cursor; the view of the model will be rotated and you can visually maneuver around it.

You can also rotate the view around the X, Y, or Z-axis of the current view. To rotate the view around the X-axis of the current view, invoke the **Rotate** tool and move the cursor close to the left or right edge of the drawing window; the cursor changes to the X-rotate cursor. Press and hold the left mouse button and drag the cursor; the view will be rotated around the X-axis of the current view. Move the cursor close to the bottom edge of the drawing window and drag the cursor to rotate the view around the Y-axis of the current view. Similarly, move the cursor close to the top edge of the drawing window and drag the cursor to rotate the view around the Z-axis of the current view. Figure 4-57 shows the X, Y, and Z rotate cursors.



Figure 4-57 The X, Y, and Z rotate cursors



### Note

You can restore any standard view by choosing its corresponding button from the flyout that is displayed, when you choose the down arrow on the right of the **Trimetric** button in the **View** toolbar.

## SETTING DISPLAY MODES

You can set the display modes for the solid models using the buttons in the **View** toolbar. Figure 4-58 shows the partial display of the **View** toolbar with various buttons and flyout options that you can use to set the display modes of the model.

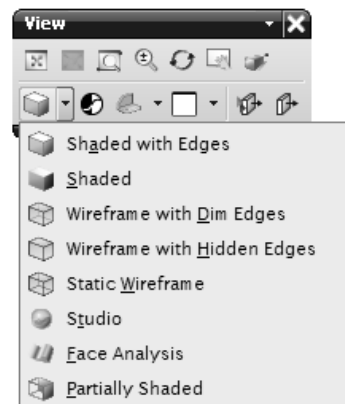


Figure 4-58 Partial view of the **View** toolbar to display various options to set the display modes

## TUTORIALS

### Tutorial 1

In this tutorial, you will create the model, as shown in Figure 4-59. The dimensions of the model are shown in Figure 4-60. The depth of extrusion of the model is 45.

(Expected time: 30 min)

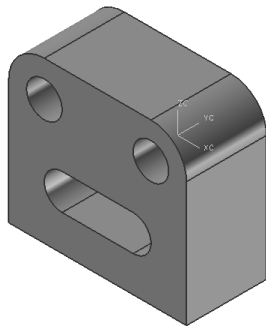


Figure 4-59 Model for Tutorial 1

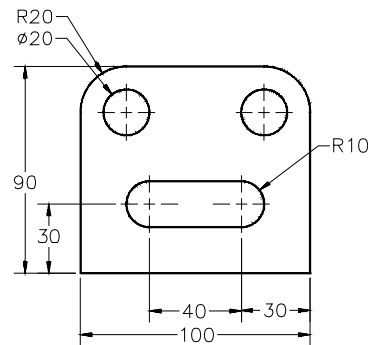


Figure 4-60 Dimensions of the model

The following steps are required to complete this tutorial:

- Start a new part file using the Model template and then draw the profile of the outer loop. Add the required constraints.
- Draw the inner loops and add the required dimensions to them.
- Exit the **Sketcher** environment and invoke the **Extrude** tool. Define the depth of the extrusion to create the final model.
- Rotate the view in 3D space.
- Save the file and close it.

### Drawing the Sketch of the Model

The sketch of this model will be created on XC-ZC plane. As mentioned earlier, when you extrude the sketch with nested closed loops, the inner loops are automatically subtracted from the outer sketch.

- Start a new file using the Model template. Save the file in the C:\NX 6\c04 folder with the name *c04tut1.prt*.
- Create three fixed datum planes (YC-ZC, XC-ZC, and XC-YC) and make the display of WCS on.
- Invoke the **Sketcher** environment using the XC-ZC plane.
- Using the **Zoom In/Out** tool, zoom out such that the sketching plane appears almost half its original size.

5. Draw the sketch of the outer loop and add the required geometric and dimensional constraints to it, as shown in Figure 4-61.
6. Draw the inner loops and then add the required geometric and dimensional constraints to them, as shown in Figure 4-62.

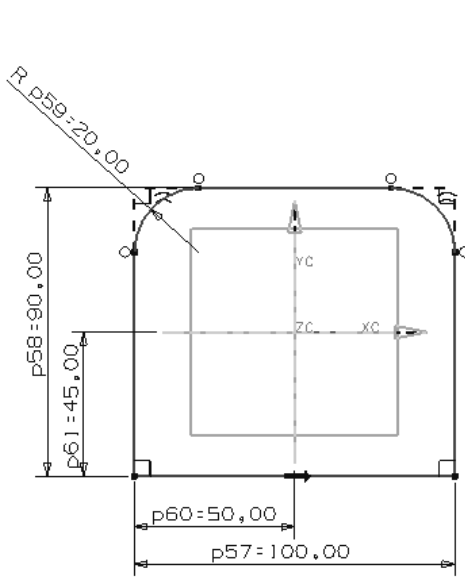


Figure 4-61 Dimensioned sketch of the outer loop

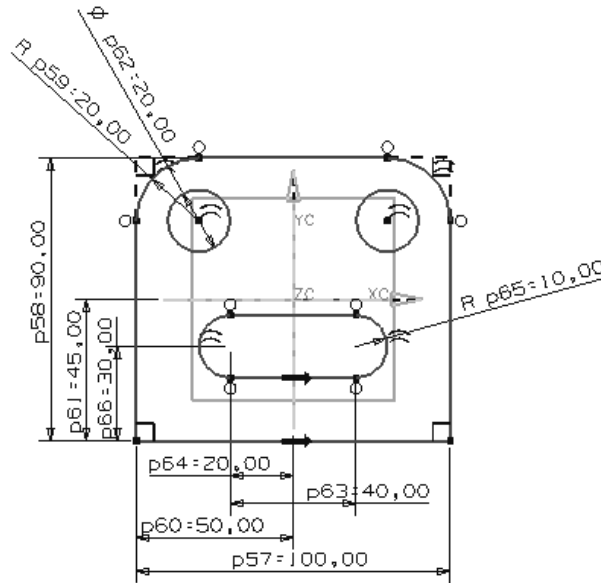



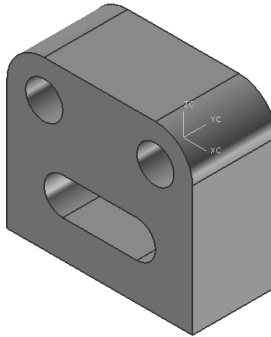
Figure 4-62 Final sketch of the base feature

### Converting the Sketch into the Base Feature

Next, you need to convert the sketch into the base feature. This is done using the **Extrude** tool.

1. Choose the **Finish Sketch** button to exit the **Sketcher** environment; the current view is automatically changed to the Trimetric view.
2. Right-click and choose **Fit** from the shortcut menu to make sure that the sketch fits in the screen.
3. Invoke the **Extrude** tool from the **Feature** toolbar; the **Extrude** dialog box is displayed and you are prompted to select the planar face to sketch or select the section geometry. 
4. Select the sketch; the preview of the extruded feature is displayed and the **End** edit box is displayed on the feature.
5. Enter **45** as the value in the **End** edit box and press the ENTER key; the preview is modified accordingly.
6. Choose the **OK** button in the **Extrude** dialog box; the extrude feature is created and displayed in the drawing window.


7. Using the CTRL+B keys, hide the display of the datum axes, datum plane, and the sketch. The extruded model is shown in Figure 4-63.



*Figure 4-63 Extruded model for Tutorial 1*

### Rotating the View of the Model

Next, you need to rotate the view of the model so that you can maneuver around it and view it from different directions.

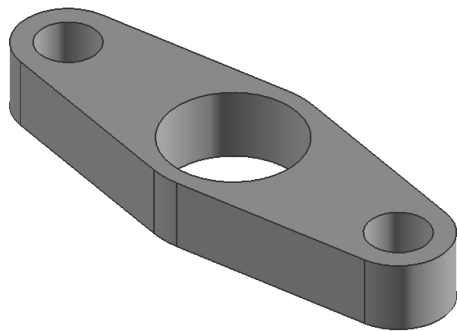
1. Choose the **Rotate** button from the **View** toolbar to invoke this tool; the cursor changes to the rotate view cursor.
2. Press and hold the left mouse button and drag the cursor in the drawing window to rotate the view of the model.
3. Invoke the **Rotate** tool again to finish rotating the model.
4. Choose the **Isometric** button in the **View** toolbar to restore the Isometric view. If this is not the default button, then choose the down arrow on the right of the default button and choose the **Isometric** button from the flyout. 
5. Right-click in the drawing window and choose **Fit** from the shortcut menu; the model fits in the drawing window.

### Saving and Closing the File

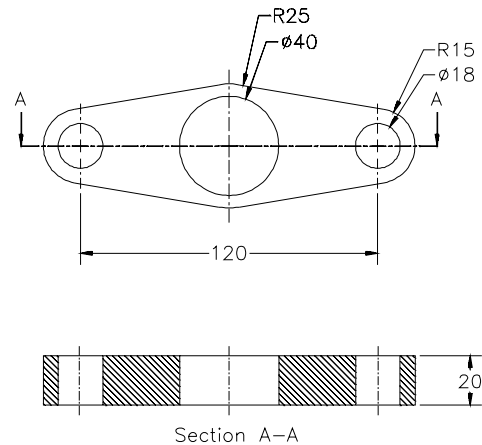
1. Choose **File > Close > Save and Close** from the menu bar to save and close the file.

## Tutorial 2

In this tutorial, you will create the model, as shown in Figure 4-64. Its dimensions are given in the drawing views shown in Figure 4-65. **(Expected time: 30 min)**



**Figure 4-64** Model for Tutorial 2



**Figure 4-65** Top and sectioned front views showing the dimensions of the model

Before creating the model, it is recommended that you outline the steps that are required to create it.

The following steps are required to complete this tutorial:

- Start a new part file using the Model template and then draw the profile of the outer loop.
- Add the required dimensions and constraints to the profile.
- Draw the inner circles and add the required dimensions to them.
- Exit the **Sketcher** environment and invoke the **Extrude** tool. Define the depth of the extrusion to create the final model.
- Rotate the view in 3D space, save the file, and close it.

### Drawing the Sketch of the Model

The sketch of this model can be created using the **Sketch** tool. The inner circles will be automatically subtracted from the outer profile on extruding.

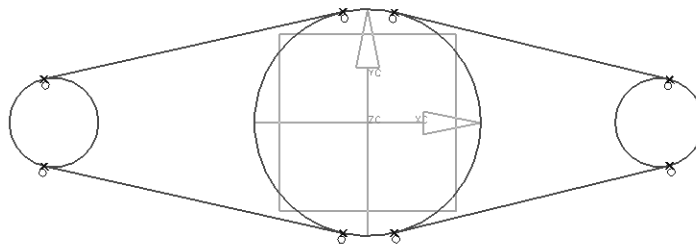
- Start NX 6 and then start a new file using the Model template. Save the file in the C:\NX 6\c04 folder with the name *c04tut2.prt*.
- Create three fixed datum planes (YC-ZC, XC-ZC, and XC-YC) and turn on the display of WCS.
- Invoke the **Sketcher** environment using the XC-YC plane as the sketching plane.



4. Invoke the **Zoom In/Out** tool and zoom out the screen such that the sketching plane appears almost one-third of its original size.
5. Draw the sketch using the **Circle by Center and Diameter** option of the **Circle** tool and the **Line** tool, as shown in Figure 4-66. Make sure that the Tangent and Point On Curve constraints are applied between the lines and the circles at all points where the lines intersect the circles. The Tangent constraint is designated by a small circle and the Point On Curve constraint is designated by a cross, as shown in Figure 4-66.

**Note**

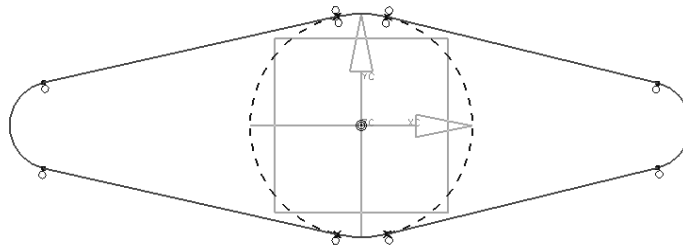
To make sure that both the **Tangent** and the **Point on curve** constraints are applied, specify the endpoint of the line only when the symbol of the **Tangent** constraint is displayed on the left of the cursor and the symbol of the **Point on curve** constraint is displayed on the right of the cursor.



**Figure 4-66** Initial sketch for the base feature

Next, you need to trim the unwanted portion of the circles to retain the outer profile of the model. The sketch is trimmed using the **Quick Trim** tool.

6. Choose the **Quick Trim** button from the **Sketch Curve** toolbar; you are prompted to select the curve to be trimmed.
7. Press and hold the left mouse button inside the right circle and drag it horizontally to the left, close to the center of the left circle.
8. Release the left mouse button to trim the unwanted portions of the circles, refer to Figure 4-67.



**Figure 4-67** Sketch after trimming the unwanted portion of circles

**Note**

If the entire circle is highlighted in red, the **Point on curve** constraint is not applied between the lines and the circle. Therefore, you need to manually add this constraint to the circle and the lines.

Next, you need to add constraints to the sketch.

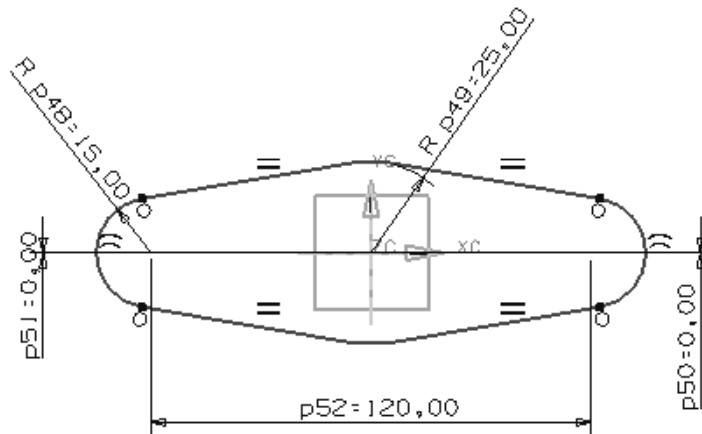
9. Choose the **Constraints** button from the **Sketch Constraints** toolbar and make the radius of the left arc equal to that of the right arc. Similarly, make all lines equal using the Equal Length constraint. Note that you also need to add the **Point on Curve** constraint between the center of the central arcs and the XC and YC axes. To select the axes for applying constraints, use the selection filters.

**Note**

You can choose the **Show All Constraints** button from the **Sketch Constraints** toolbar to view all the constraints that are applied to the sketch.

Next, you need to add the dimensions to the sketch.

10. Choose **Preference > Sketch** from the menu bar; the **Sketch Preferences** dialog box will be displayed. Enter **2** in the **Decimal Places** edit box and then select **Expression** from the **Dimension Label** drop-down list. Choose the **OK** button to exit the dialog box.
11. Add the dimensions to the sketch using the **Inferred Dimensions** tool, as shown in Figure 4-68. You may also have to make the vertical distance value between the center of the left and the right arcs zero to get the exact sketch. Similarly, you may have to make the vertical distance between the center of the central arcs and one of the outer arcs zero. Next, you need to draw the inner circles. You can use the center points of the arcs to draw them.



**Figure 4-68** Sketch after adding relationships and dimensions

12. Next, draw three circles using the center points of the arcs, refer to Figure 4-69.

13. Add the **Equal Radius** constraint to the left and right circles. Now, add the required dimensions to the circles to complete the sketch. The final sketch of the model is shown in Figure 4-69.

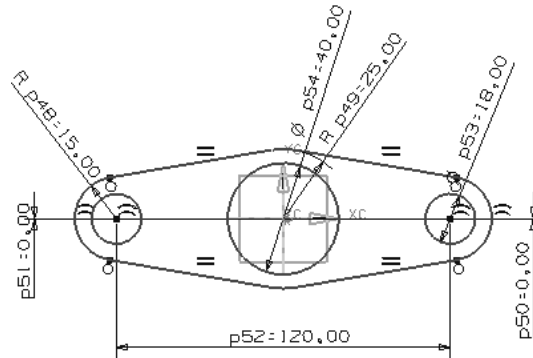
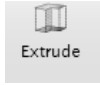


Figure 4-69 Final sketch for Tutorial 2

### Converting the Sketch into the Base Feature

Next, you need to convert the sketch into the base feature. This is done using the **Extrude** tool.

1. Choose the **Finish Sketch** button from the **Sketcher** toolbar to exit the **Sketcher** environment; the current view is automatically changed to the Trimetric view.
2. Invoke the **Extrude** tool from the **Feature** toolbar; the **Extrude** dialog box is displayed and you are prompted to select the planar face to sketch or select the section geometry to extrude. 
3. Select the sketch; the preview of the extruded feature is displayed and the **End** edit box is displayed on the feature.
4. Enter **20** as the value in the **End** edit box and press the ENTER key; the preview is modified accordingly.
5. Choose the **OK** button in the **Extrude** dialog box; the extruded feature is created and displayed in the drawing window, as shown in Figure 4-70.

Notice that the datum plane, sketch, and datum axes are still displayed in the sketch. For a better visualization of the model, turn off the display of these entities.

6. Press the CTRL+B keys; the **Class Selection** dialog box is displayed. Select all the unwanted entities and then choose **OK**.

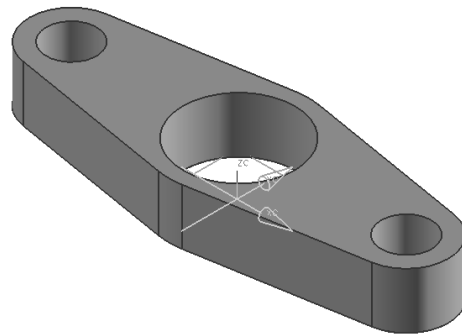

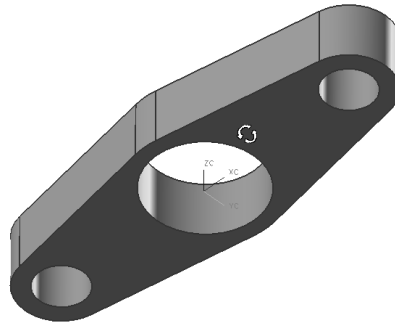


Figure 4-70 Model after extruding the sketch

### Rotating the View of the Model

Next, you need to rotate the model to maneuver and view it from different directions.

1. Choose the **Rotate** button from the **View** toolbar to invoke this tool; the cursor changes to the rotate view cursor. 
2. Press and hold the left mouse button and drag the cursor in the drawing window. Figure 4-71 shows a model being rotated using the **Rotate** tool.



*Figure 4-71 Rotating the model in 3D space*

3. Choose the **Rotate** button again to finish rotating the model.

Next, you need to restore the Isometric view of the model, which has been changed by the **Rotate** tool.

4. Choose the **Isometric** button in the **View** toolbar to restore the Isometric view. If this is not the default button, then choose the down arrow on the right of the default button and choose the **Isometric** button from the flyout.
5. Now, right-click in the drawing window and choose **Fit** from the shortcut menu; the model fits in the drawing window.

### Saving and Closing the File

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

## Tutorial 3

In this tutorial, you will create the model, as shown in Figure 4-72. This is a revolved body created using the sketch shown in Figure 4-73. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- a. Start a new part file using the Model template and draw the sketch of the revolved model. Add the required geometric and dimensional constraints.
- b. Exit the **Sketcher** environment and invoke the **Revolved Body** tool. Select the sketch to be revolved and then the direction of the axis of revolution.

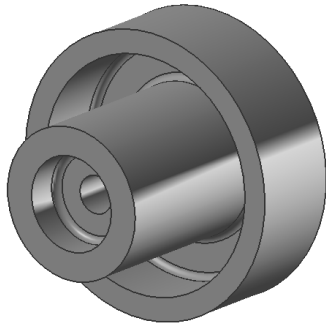


Figure 4-72 Model for Tutorial 3

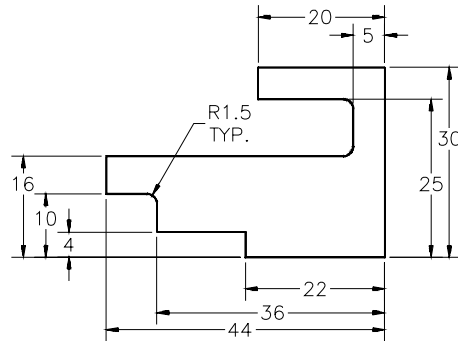


Figure 4-73 Dimensions of the model

- c. Define the other revolution parameters to create the final model.
- d. Rotate the view in 3D space.
- e. Save and close the file.

### Drawing the Sketch of the Model

The sketch of the revolved model will be created on the YC-ZC plane.

1. Start a new file using the Model template. Save the file in the C:\NX 6\c04 folder with the name *c04tut3.prt*.
2. Create three fixed datum planes (YC-ZC, XC-ZC, and XC-YC) and make the display of WCS on.
3. Invoke the **Sketcher** environment using the YC-ZC plane.
4. Draw the sketch of the revolved feature and add the required geometric and dimensional constraints to it, as shown in Figure 4-74.

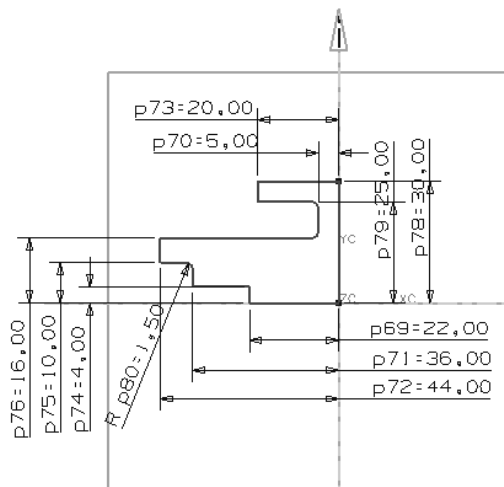

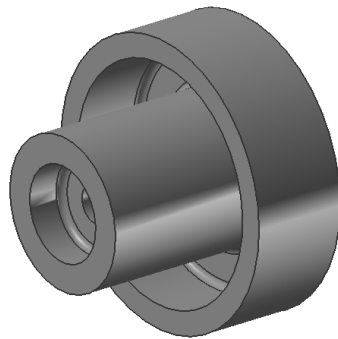


Figure 4-74 Dimensioned sketch of the revolved feature

### Converting the Sketch into the Revolved Feature

Next, you need to convert the sketch into a revolved feature. This is done using the **Revolve** tool.


1. Choose the **Finish Sketch** button to exit the **Sketcher** environment; the current view is automatically changed to the Trimetric view.
2. Right-click and choose **Fit** from the shortcut menu to make sure that the sketch fits in the screen.
3. Invoke the **Revolve** tool from the **Features** toolbar; the **Revolve** dialog box is displayed and you are prompted to select the planar face to sketch or select the section geometry. 
4. Select the sketch drawn and choose the **Inferred Vector** button from the **Axis** rollout; you are prompted to select the objects to infer vector.
5. Select the bottom horizontal line as the axis of revolution; the preview of the revolved body is displayed.
6. Accept the default options in the **Revolve** dialog box and choose the **OK** button; the revolved feature is created.
7. Press the CTRL+B keys to hide the sketch, axes, and the sketching plane. Fit the model in the screen. The revolved model is shown in Figure 4-75.



*Figure 4-75 Revolved model of Tutorial 3*

### Rotating the View of the Model

Next, you need to rotate the model to maneuver and view it from different directions.

1. Choose the **Rotate** button from the **View** toolbar to invoke this tool; the cursor changes to the rotate view cursor. 
2. Press and hold the left mouse button and drag the cursor in the drawing window to rotate the view of the model. Invoke the **Rotate** tool again to finish rotating the model.

3. Choose the **Isometric** button in the **View** toolbar to restore the Isometric view. If this is not the default button, then choose the down arrow on the right of the default button; a flyout is displayed. Now, choose the **Isometric** button from this flyout.
4. Right-click in the drawing window and choose **Fit** from the shortcut menu. The model fits in the drawing window.

### Saving and Closing the File

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

### Self-Evaluation Test

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

1. The **Quick Trim** tool is used to remove portion of the sketch by chopping it off. (T/F)
2. In NX, while extruding a sketch, you can add a draft to it. (T/F)
3. The **Quick Extend** tool is used to extend or lengthen an open sketched entity up to infinity. (T/F)
4. You can set the display modes for solid models using the buttons in the **Standard** toolbar. (T/F)
5. After invoking the **Quick Trim** tool, you can drag the cursor to trim \_\_\_\_\_ entities.
6. The \_\_\_\_\_ option is selected when you need to extrude the sketch symmetrically on both sides of the plane, on which the sketch is created.
7. The \_\_\_\_\_ option adds the draft that is aligned with the profile.
8. NX uses the \_\_\_\_\_ rule to determine the direction of revolution.
9. You can create thin features using the \_\_\_\_\_ or \_\_\_\_\_ sketches.
10. You can restore standard views by choosing various buttons from the \_\_\_\_\_ toolbar.

### Review Questions

Answer the following questions:

1. Which of the following tools in NX 6 allows you to dynamically move, rotate, or copy solid objects as well as sketched entities?
  - (a) **Transform**
  - (b) **Modify**
  - (c) **Move Object**
  - (d) **None**

2. Which one of the following views is the default view in NX?
- (a) Trimetric (b) Isometric  
(c) Top (d) None
3. Which rollout is used to create a thin extruded feature by offsetting in two directions?
- (a) **Offset** (b) **Symmetric**  
(c) **Two Sided** (d) None
4. From where can you invoke the **Move Object** tool?
- (a) **Edit > Move Object** (b) **Insert > Move Object**  
(c) **Preference > Move Object** (d) **Tool > Move Object**
5. Which one of the following rules is used to determine the direction of revolution?
- (a) Right-hand rule (b) Right-hand thumb rule  
(c) Left-hand rule (d) Left-hand thumb rule
6. In which one of the following toolbars can you set the display modes for solid models?
- (a) **Standard** (b) **Sketch**  
(c) **Visible** (d) **View**
7. After choosing **Finish Sketch** from the **Sketcher** toolbar to exit the **Sketcher** environment, the **Modeling** environment will be invoked. (T/F)
8. When you mirror entities using the **Mirror Curve** tool, by default a mirrored copy of the selected entities is created and original entities are deleted. (T/F)
9. In the **Results** rollout of the **Move Object** dialog box, you can set for to move the original entity or copy it by using the **Move Original** or **Copy Original** radio button. (T/F)
10. You can rotate a model in 3D space using the **Rotate** tool. (T/F)

## Exercises

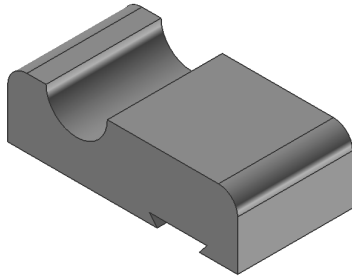
### Exercise 1

Open the sketch drawn in Tutorial 3 of Chapter 3 and convert it into a full revolved body. After creating the model, use the **Rotate** tool to rotate its view. Save the file with a different name in the folder of this chapter. **(Expected time: 15 min)**

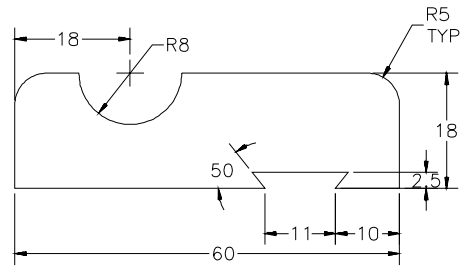


## Exercise 2

Create the model shown in Figure 4-76. The dimensions of the model are shown in Figure 4-77. The depth of extrusion is 30. After creating the model, use the **Rotate** tool to rotate its view. Before closing the file, restore the Isometric view of the model. **(Expected time: 30 min)**



*Figure 4-76 Model for Exercise 2*



*Figure 4-77 Dimensions of the model*

**Answers to Self-Evaluation Test**

1. T, 2. T, 3. F, 4. F, 5. multiple, 6. Symmetric Value, 7. From Section, 8. Right-hand thumb rule, 9. open, close, 10. View