

Chapter 3

Adding Constraints and Creating Procedures in the Part Modeler

Learning Objectives

After completing this chapter, you will be able to:

- *Add geometric and dimensional constraints.*
- *Understand the concept of well constrained sketches.*
- *Create new procedures.*
- *Create new construction planes.*
- *Create added procedures.*
- *Create holes, fillets, chamfers, and drafts.*
- *Create patterned procedures.*
- *Edit procedures.*

CONSTRAINING SKETCHES

In the previous chapter, you learned to draw and edit the sketches in the sketcher environment. In this chapter, you will learn to apply constraints to sketches in order to restrict their degrees of freedom and make them stable. The stability ensures that the size, shape, and location of the sketches do not change unexpectedly with respect to the surroundings. You should always constrain the sketches to restrict their degrees of freedom. You can do this by applying logical constraints to the sketch. Some of these are applied automatically while drawing. In EdgeCAM, you can also apply dimensions while drawing by selecting the check boxes next to the value spinners. You also need to add additional dimensional constraints, if required, using the tools in the **Application Menu** toolbar.

In EdgeCAM, the tools to apply logical constraints and dimensions are present in the **Assign 2d Constraints** dialog box, shown in Figure 3-1. This dialog box is invoked using the **Assign 2d Constraints** tool. Various logical constraints are discussed next.



Figure 3-1 The Assign 2D Constraints dialog box

Applying Logical Constraints

Menu: General Construction > Constraints > Assign 2D Constraints

Toolbar: Application Menu > Assign 2D Constraints



Logical constraints are applied to the sketched elements to define their size and position with respect to the other elements. They are applied using two methods, automatic constraining and manual constraining. While drawing the sketch, some constraints are automatically applied to it, if the **Automatic constraints** button is chosen in the **Locate** toolbar. For applying constraints manually, you need to invoke the **Assign 2d Constraints** dialog box by choosing the **Assign 2d Constraints** button from the **Application Menu** toolbar and selecting the appropriate option.

The logical constraints options in the **Assign 2D Constraints** dialog box are discussed next.

Fix Constraint



This constraint is used to fix the orientation or location of the selected curve or point with respect to the coordinate system of the current drawing. If you apply this constraint to a line or an arc, you cannot move them from their current location. However, you can change their length by selecting one of their endpoints and then dragging it. If you apply this constraint to a circle or an ellipse, you cannot edit them by dragging.

Datum Constraint



Applying the **Datum constraint** ensures that the selected entity will not change its orientation when other entities are constrained with respect to this entity. However, you can drag a datum entity to modify it.

Horizontal Constraint



The **Horizontal constraint** forces the selected line segment, ellipse axis, or two points to become horizontal irrespective of their original orientation. When you invoke this constraint, you will be prompted select a line, an ellipse, or a spline. If you select them, they will become horizontal.

Vertical Constraint



The **Vertical constraint** is similar to the **Horizontal constraint**, with the difference that this will force the selected entities to become vertical.

Coincident Constraint



This option forces two selected entities to coincide with each other. You can coincide two points, a point and a line, a point and a curve, or a point and a spline. If you apply the coincident constraint between two arcs or two circles, the centers will coincide and not the endpoints. Also, the radius of the two curves will become equal. Note that the first or the second entity selected should be a point. The points include the endpoints of a line or an arc or the center point of a circle.

Concentric Constraint



This option forces the selected arcs or circles to share the same center point. The curves that can be made concentric include arcs, circles, and ellipses. When you invoke this constraint, you will be prompted to select the first arc, ellipse, or circle. After making the first selection, you will be prompted to select the second arc, circle, or ellipse.

Parallel Constraint



The **Parallel constraint** forces the selected entity to become parallel to the specified entity. The entities to which this constraint can be applied are lines and ellipses. When you invoke this constraint, you will be prompted to select the first line or ellipse. After making the first selection, select the second line or ellipse.

Perpendicular Constraint



This option is used to make two entities perpendicular to each other. The entities can be two lines or a line and an ellipse. Select the two entities; the entities will become perpendicular to each other.

Equal Radius



This option is used to change two circles, two arcs, or a circle and an arc of different radii into curves with the same radius. To apply this constraint, choose the **Equal Radius** button from the **Assign 2d Constraints** dialog box and select two arcs from the drawing area.

Equal Length



This option forces two lines to become equal in length. To apply this constraint, choose the **Equal Length** button from the **Assign 2 D Constraints** dialog box and select the two lines in succession from the drawing area.

Tangent Constraint



The **Tangent Constraint** makes the selected entities tangent to each other. After invoking this constraint, first select a line, an arc, or a spline. Next, select an arc or an ellipse to which the tangency has to be maintained.



Note

If you try to manually apply any constraints that are already applied automatically, a warning box will be displayed stating that the entity already has the constraint of this type.



Tip. You can apply a constraint to a sketched element or between two or more than two entities. You can also apply it between a sketched element and an edge or a vertex of the model.

Applying Dimensional Constraints

Menu: General Construction > Constraints > Assign 2D Constraints

Toolbar: Application Menu > Assign 2D Constraints



After drawing sketches and adding relations, dimensioning is the most important step in creating a design. As mentioned earlier, EdgeCAM is a parametric software. This property of EdgeCAM ensures that irrespective of the original size, the selected entity is driven by the dimension value you specify. Therefore, when you apply and modify a dimension of an entity, it is forced to change its size in accordance with the specified dimension value. In EdgeCAM, while creating a sketch, if you select the check box next to the spinners that define dimensions such as radius, height, length, and so on, the dimensional constraints will be applied automatically. In addition to this, dimensional constraints can also be added by using various tools in the **Assign 2d Constraints** dialog box. When you dimension the entities, the **Driven** check box will be available. Select the check box to make the dimension a driven dimension. The driven dimension will be discussed later in this chapter. Various tools in the **Dimension** area of the **Assign 2d Constraints** dialog box are discussed next.

Minimum Distance Dimensioning

 To dimension the shortest distance between two entities, the **Minimum Distance Dimension** tool is used. To apply this constraint, choose the **Minimum Distance Dimension** button from the **Assign 2d Constraints** dialog box. Choose two points from the drawing area; the dimension will be attached to the cursor. Move the cursor and place the dimension at an appropriate place using the left mouse button. Enter the new value in the spinner above the **Apply** button and choose the **Apply** button. You can select two points, a curve and a point, or a position on a curve.

Horizontal/Vertical Dimensioning

 These two tools are used to add the horizontal or vertical dimensions to a selected line or between two points. The points can be endpoints of lines or arcs, or the center points of circles, arcs, ellipses, parabola, or hyperbola. Choose the **Horizontal** or **Vertical Distance Dimension** button from the **Assign 2d Constraints** dialog box. Then select the two points for which dimensioning is required; the dimension will be attached to the cursor. Move the cursor and place it at an appropriate position, as shown in Figure 3-2. Enter the new value in the spinner above the **Apply** button and choose the **Apply** button.

Length Dimensioning

 This type of dimensioning is used to measure the actual distance of a line. You cannot use the **Horizontal** or **Vertical Distance Dimension** button to measure an inclined line. With the **Length Dimension** button, you can directly select the inclined line to apply the dimension. Choose the **Length Dimension** button from the **Assign 2d Constraints** dialog box. Then, select the line from the drawing area; the dimension will be attached to the cursor. Move the cursor parallel to the line and place the dimension at an appropriate place, as shown in Figure 3-2. Enter a new value in the spinner and choose the **Apply** button.

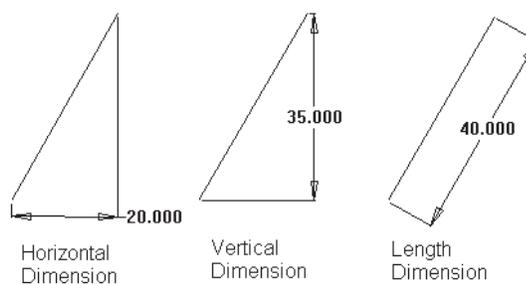


Figure 3-2 Applying the horizontal, vertical, and length dimension to inclined lines



Tip. You can also enter the arithmetic symbols directly into the value spinner. For example, if you have a dimension as a complex arithmetic function such as $(220 * 12.5) - 3 + 150$, enter the statement instead of entering the final result.

Angular Dimensioning



To apply angular dimensions between two lines, choose the **Angle Dimension** button from the **Assign 2d Constraints** dialog box. Next, select the two lines in succession; the dimension will be attached to the cursor. Place the angular dimension and enter the new value in the spinner in the **Assign 2d Constraints** dialog box. You need to be very careful while placing the angular dimension. This is because depending on the location of the dimension placement, the interior angle, exterior angle, major angle, or minor angle will be displayed. Figure 3-3 illustrates various angular dimensions depending on the dimension placement.

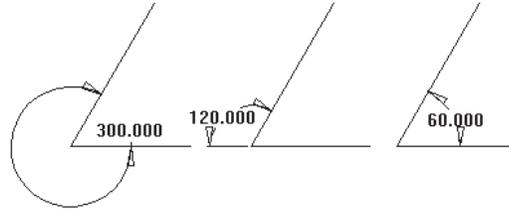


Figure 3-3 Angular dimension displayed according to the dimension placement point

Radial Dimensioning



A radial dimension is applied to dimension a circle or an arc in terms of its radius. To apply the radius dimension, choose the **Radius Dimension** button from the **Assign 2d Constraints** dialog box. Select a circle or an arc to dimension; the radial dimension will be attached to the cursor. Using the left mouse button, place the dimension at an appropriate place, as shown in Figure 3-4. Enter the new value in the spinner and choose **Apply**.

Dimensioning an Ellipse



This type of dimensioning is used to measure the semimajor and semiminor axis of an ellipse. Choose the **Minor/Major Radius Dimension** button from the **Assign 2d Constraints** dialog box. Next, move the cursor near the ellipse and select it as it gets highlighted. The dimension will be attached to the cursor. Place it at an appropriate place using the left mouse button. Enter a new value in the spinner, if required, and choose the **Apply** button.

Figure 3-5 shows the sketch after applying the dimensions discussed above.

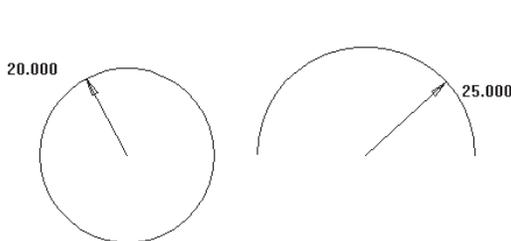


Figure 3-4 Radial dimensioning of a circle and an arc

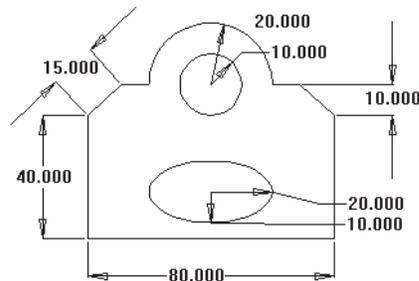


Figure 3-5 The sketch with all dimensions

CONCEPT OF FULLY CONSTRAINED SKETCHES

It is necessary for you to understand the concept of fully constrained sketches. While creating a model, first draw the sketch of the base feature and then create other features. After creating the sketch, add the required relations and dimensions to constrain it with respect to the sketched entities and the surrounding environment. Once you add the required relations and dimensions, the sketch may exist in any one of the three states discussed below.

Fully Constrained

A fully constrained or well constrained sketch is the one in which all the entities of the sketch and their positions are well defined by relations or dimensions, or both. Also, all the degrees of freedom are defined using relations and dimensions. So, the sketched entities cannot move or change their size and location unexpectedly. On the other hand, if a sketch is not fully constrained, it can change its size or position at any time during the design because all the degrees of freedom are not defined completely.

Over Constrained

An over constrained sketch is the one in which some of the dimensions, relations, or both conflict or they have exceeded their required number. Therefore, you need to delete the extra and conflicting relations or dimensions. You should not create the feature with an over defined sketch. You can change an over constrained sketch into a fully constrained or under constrained sketch by deleting the conflicting relations or dimensions. The process of deleting the over defining relation or dimension is discussed later in this chapter.

Under Constrained

An under constrained sketch is the one in which some of the dimensions or relations are not defined and the degrees of freedom are not well constrained. In these sketches, the entities may move or change their size unexpectedly. You can add some required constraints to make them well constrained.



Tip. In EdgeCAM, it is not necessary to fully constrain the sketches before converting them into create the features of the model. However, it is recommended that you fully constrain the sketches before proceeding to create the feature.

Deleting an Over defined Constraint

In EdgeCAM, when a manually added constraint over defines a sketch, the **Overconstrained Dialog** dialog box is displayed, as shown in Figure 3-6. It informs you that the geometry is over defined and you need to delete some constraints. To delete a constraint, select it from the dialog box and choose the **Delete** button. If you choose the **Delete All** button, all the constraints applied will be deleted.

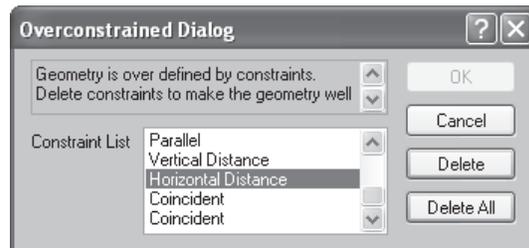


Figure 3-6 The **Overconstrained Dialog** dialog box

Driving and Driven Dimensions

A dimension is called a driving dimension if it can force an entity to change its length and orientation. While adding a dimension, select the **Driven** check box in the **Assign 2d Constraints** dialog box to make a dimension driven dimension. A driven dimension is one whose value depends on the value of the driving dimension. It is displayed in the parenthesis and cannot be modified. If you change the value of the driver dimension, the value of the driven dimension changes automatically.

Editing a Dimension

In EdgeCAM, on moving the cursor near a dimension, a name is displayed. This is the default name assigned to the dimension. You can modify the name and value by invoking the **Enter Distance Value** dialog box, as shown in Figure 3-7. To invoke this dialog box, move the cursor near the dimension and double-click. Enter a new value in the value spinner and a name in the **Name** edit box. To find the value from a given mathematical expression, enter the same directly instead of calculating and entering the final result. Select the **Driven Dimension** check box to make the dimension the driven dimension. It will be displayed in parenthesis, as shown in Figure 3-8.

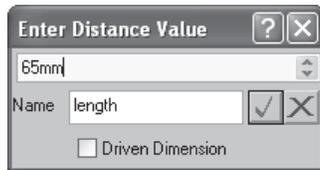


Figure 3-7 The **Enter Distance Value** dialog box

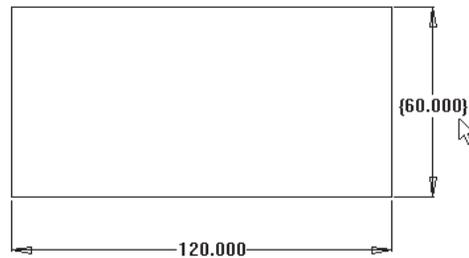


Figure 3-8 The cursor selecting the driven dimension



Note

The name assigned to a dimension should be unique. You cannot assign the same name to two different dimensions.

Models of the same shape but different dimensions can be made by writing equations. Equations are mathematical expressions in which the parameters are equated with the algebraic or trigonometrical functions. To create equations, first assign a name to the dimensional constraint. Then, name the second dimensional constraint and relate its value to the first entity by specifying the name of the first entity in the parenthesis followed by mathematical expressions. For example, to create a rectangle of different width and height, double-click on the width dimension and name it **Width**. Enter a new value in the value spinner, if required. Then, double-click the height dimension and enter the expression = **{Width}/2**. Also, enter the name **Height**. When you change the value of the width, the height is also modified automatically. Also, when you move the cursor near the height dimension, the modified name, **Height**, is displayed.



Note

In the EdgeCAM Part Modeler, move the cursor to the yellow-shaded region of the spinner and right-click. Select the **Equation** option to invoke the **Equation** dialog box.

Editing a Constraint

Menu: General Construction > Constraints > Edit 2D Constraints
Toolbar: Application Menu > Assign 2D Constraints > Edit 2D Constraints



Certain constraints are applied automatically, when you perform an editing operation. This may prevent you from performing some operations. For example, when you mirror an entity, a symmetric constraint will be automatically applied. This will prevent you from performing the trim operation effectively. To remove a constraint, choose the **Edit 2d Constraints** button from the **Application Menu > Assign 2D Constraints** toolbar; the **Edit Constraints** dialog box will be displayed, as shown in Figure 3-9. Next, select the curve from the drawing area; all the constraints defining the curve will be displayed in the **Constraint types on curve** list box. Select a particular constraint; the entities on which this constraint is applied will be highlighted in the drawing area. Choose the **Delete** button to remove the selected constraint from the active construction plane. If you choose the **Delete List** button, all the constraints of the selected entity will be removed. Choose the **Delete All On Plane** button to remove all the constraints in the active construction plane.

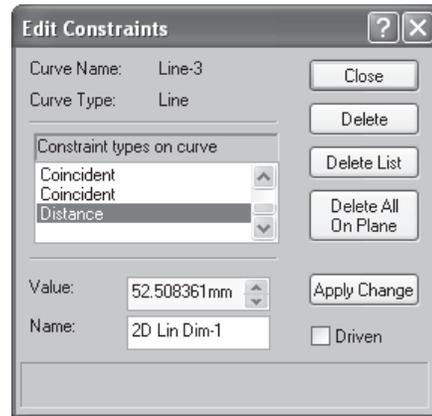


Figure 3-9 The *Edit Constraints* dialog box

If you have applied a dimension to the selected entity, it will be termed as **Distance** in the **Constraint types on curve** list box. Modify the value by selecting it in the list box. When you select **Distance**, the **Value** spinner and the **Name** edit box will be available. Enter a new value in the **Value** spinner and a name, if required, in the **Name** edit box. Select the **Driven** check box to make the dimension a driven dimension. If you have specified equations to any dimensions in the **Value** spinner, there will be a button available instead of the spinner. If you select the button, the **Equation** dialog box will be displayed. You can change the equation in the dialog box. Choose the **Apply Change** button to preview the modified dimension and choose the **Close** button to accept the changes.

CREATING BASE FEATURES

The sketches that you have drawn until now can be converted into base features by choosing **3D Construction > New Component** from the menu bar. The primary feature created from the sketch is called the base feature. Then features are added, if required, to complete the model. When you choose any tool from the **New Component** cascading menu, the sketching environment is converted into the modeling environment. When you create a feature, a node with the name, **Component-1** is created in the **Model** window. A new feature is added under the node. So, you should create only the base feature with the tools in **New Component**.

Based on the sketch for the new component, the resulting feature can be a solid, surface, sheet, or a shell feature. If the sketch drawn is an open sketch, then you can create only the

Surface or the **Sheet** feature. The characteristics of the different features that can be created are discussed next.

Solid & Shell	Sheet	Surface
Need closed sketch	Sketch can be closed or open	Sketch can be closed or open
Has definite mass properties	Has definite mass properties	N/A
Has definite volume	Has definite volume	N/A
Thickness more than 12 mm	Thickness less than 12 mm	No thickness
Two different solid bodies can be joined by a boolean operation	Two different sheets can be joined by a boolean operation	Two different surfaces can be healed

Shelling is a method of creating a hollow model with walls of a specified thickness and a cavity inside.



Note

1. You can also select the tools from the **Application Menu** toolbar to create the base feature.
2. The tools to create the base feature are grouped together in the **Application Menu** toolbar in the **New Component Extrude** flyout.

CREATING EXTRUDED FEATURES

Menu: 3D Construction > New Component > Extrude
Toolbar: Application Menu > New Component Extrude



After drawing and dimensioning the sketch, choose the **New Component Extrude** button from the **Application Menu** toolbar; the **New Component Extrude** dialog box will be displayed, as shown in Figure 3-10. Specify the depth and direction in the corresponding spinners and choose the **OK** button. You can preview the feature displayed in the temporary graphics by choosing the **Preview** button.

The spinners, drop-down lists, and radio buttons in each tab are discussed next.

Extrude Tab

The drop-down lists and spinners in the **Extrude** tab are discussed next.

Direction

The **Direction** drop-down list provides the options to define the termination of the extruded

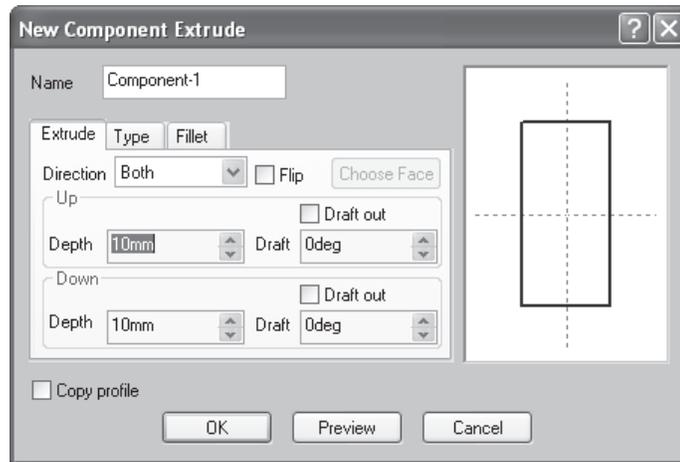


Figure 3-10 The New Component Extrude dialog box

feature. While creating the base feature, some of the options in this drop-down list will not be used. The options to define the termination of the base feature are discussed next.

Up/Down

The **Up** or **Down** option is used to define the termination of the base feature by specifying the depth of extrusion. The depth of extrusion is specified in the **Depth** spinner in the **Up** or **Down** area. If you enter the draft angle in the **Draft** spinner, the resulting feature will have a taper. Select the **Draft out** check box to taper the resulting feature outward.

Next Face

The **Next face** option is used to extrude the sketch from the sketching plane to the next surface that intersects the feature. If a draft angle is required for the feature, it is entered in the **Draft** spinner.

Both

This option is used to create the base feature by extruding the sketch on both directions of the plane on which the sketch is drawn. The depth of the feature in both directions will be specified separately in the **Depth** spinner in the **Up** and **Down** areas. You can enter the draft angle for the feature in the **Draft** spinner to create a taper. Select the **Draft out** check box to taper the resulting feature outward.

To Face

The **To Face** option is used to define the termination of the extruded feature up to a selected face or surface. When you select this option from the drop-down list, the **Choose Face** button is available. Choose this button to select the face or the surface from the drawing area. If a draft angle is required for the feature, it is entered in the **Draft** spinner.

To Face w/Offset

This option is used to define the termination of the extruded feature on a virtual surface created at an offset distance from the selected surface. When this option is selected from

the drop-down list, the **Choose Face** button is available. Choose the button to select the surface from the drawing area. Set the offset distance in the **Offset** spinner. If a draft angle is required for the feature, it is entered in the **Draft** spinner.

To Plane

The **To Plane** option is used to define the termination of the extruded feature to another face. Choose the face or plane using the **Choose Face** button. The draft angle is specified in the **Draft** spinner, if a taper is needed in the feature.

Type Tab

The options in the **Type** tab (Figure 3-11) are discussed next.

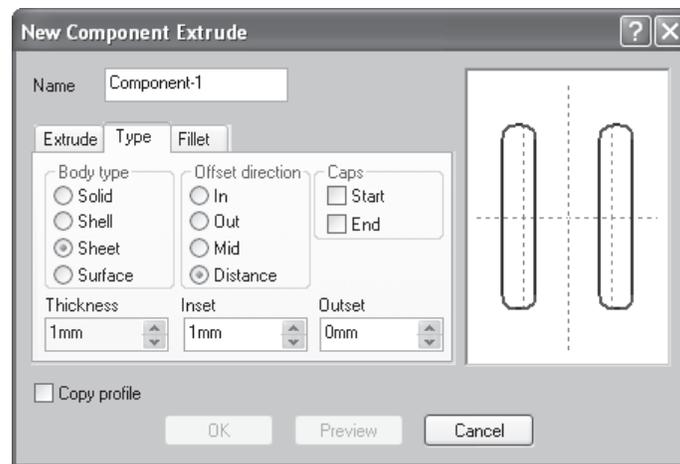


Figure 3-11 Options in the Type tab of the New Component Extrude dialog box

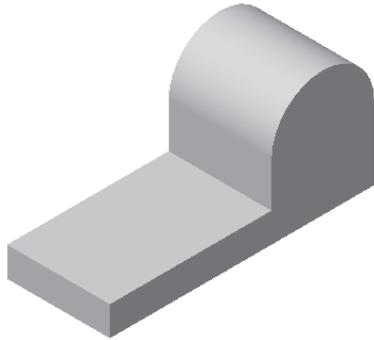
Body type Area

The radio buttons in the **Body type** area decide the type of body to be generated. If the sketch is closed, it can be converted into any one of the four bodies. Select the radio button depending on the requirement. However, if the sketch is open, it will be converted only into a sheet or a surface body. The characteristics of the four types of solid bodies have been discussed earlier.

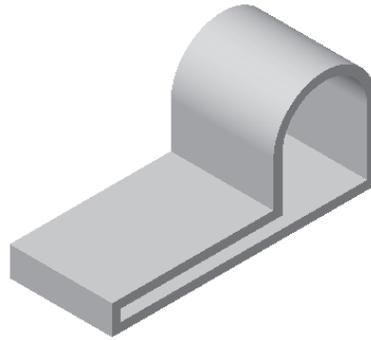
Figure 3-12 shows the base feature created by selecting the **Solid** radio button. Figure 3-13 shows the feature created by selecting the **Sheet** radio button.

Offset direction Area

The radio buttons in the **Offset direction** area will be available if you select the **Sheet** or the **Sheet** radio buttons from the **Body** area, as shown in Figure 3-11. The offset direction specifies the thickness of the shell or the sheet generated. The thickness of the material is calculated from a neutral plane. Here, the sketch drawn will act as the neutral plane. The radio buttons in the **Offset direction** area are discussed next.



*Figure 3-12 Feature created with the **Solid** radio button selected*



*Figure 3-13 Feature created with the **Sheet** radio button selected*

In

Select this radio button to add the material inside the sketch.

Out

Select this radio button to add the material outside the sketch.

Mid

Select this radio button to add the material equally on both sides of the sketch.

Distance

Select this radio button to thicken the surface on both sides to the specified distance.

Thickness

This spinner is used to specify the thickness of the shell or the sheet body.

Inset/Outset

This spinner specifies the thickness of the sheet on both sides of the sketch. The **Inset** spinner specifies the thickness of the material toward the axis from the sketch. The **Outset** spinner specifies the thickness of the material away from the axis and the sketch.

Caps Area

The **Caps** area will be available only if you select the **Sheet** and the **Surface** radio buttons. Select the **Start** and **End** check boxes to cap the ends with the same thickness as of the sheet or the surface.

The **Fillet** tab is selected to automatically fillet the edges of the top and bottom faces. Select the **Start** and the **End** check boxes to fillet the lower and upper edges, respectively. The fillet radius is specified in the corresponding spinners.

If you select the **Copy profile** check box, a copy of the profile used to create the base feature will be created. It will be visible after you create the base feature.

CREATING BASE FEATURES BY REVOLVING SKETCHES

Menu: 3D Construction > New Component > Revolve
Toolbar: Application Menu > New Component Extrude > New Component Revolve



The sketches you have drawn can also be converted into base features by revolving using the **New Component Revolve** tool. This tool is in the **Application Menu** toolbar.

Using this tool, you can revolve the sketch about selected revolution axis. The revolution axis can be an axis, a line of the sketch, or an edge of another feature. Whether you use a centerline or an edge to revolve the sketch, the sketch should be drawn on one side of the axis.

In EdgeCAM, the right-hand thumb rule is used while determining the direction of revolution. The right-hand thumb rule states that if the thumb of your right hand points in the direction of the axis of revolution, the direction of the curled fingers will determine the default direction of revolution.

After you have completed drawing and dimensioning the sketch, choose the **New Component Revolve** button from the **Application Menu > New Component Extrude** toolbar; the **New Component Revolve** dialog box will be displayed, as shown in Figure 3-14. Choose the **Select Axis** button from the **Axis** area; the **Lines** and the **Points** radio buttons will be available. Select the radio button depending on the requirement and select the axis from the drawing area. To select a line as the axis, move the cursor near it and press the left-mouse button as it gets highlighted. To select the axis through points, move the cursor in the drawing area and select two points individually as they get highlighted. When you select the axis, the **Select Axis** button will change to **Change Axis** button and the selected axis will be described next to it.

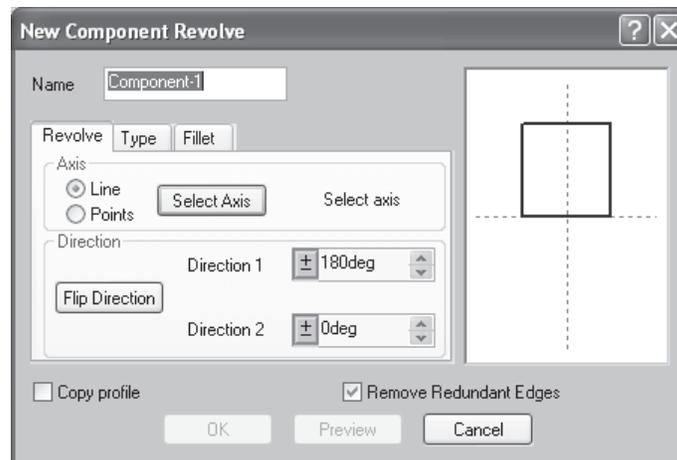


Figure 3-14 The New Component Revolve dialog box

The spinners in the **Direction** area are used to define the angle of revolution about the

selected axis. As mentioned earlier, the default direction of revolution is given by the right-hand thumb rule. The **Direction 1** spinner is used to specify the angle to revolve the sketch on any one side of the plane on which it is drawn. Similarly, the **Direction 2** spinner is used to specify the angle of revolution on the other side of the plane. However, the direction of revolution can be changed by selecting the **Flip Direction** button. The **Remove Redundant Edges** check box is used while revolving the sketch in both directions. If this check box is selected, the revolve feature in both the halves will be merged. If it is cleared, the resulting feature will have two different sections.

The radio buttons and the spinners available when you choose the **Type** and the **Fillet** tab are the same as those of the **New Component Extrude** feature. Figure 3-15 shows the base feature created in one direction by selecting the **Solid** radio button. Figure 3-16 shows the base feature created in one direction by selecting the **Sheet** radio button.

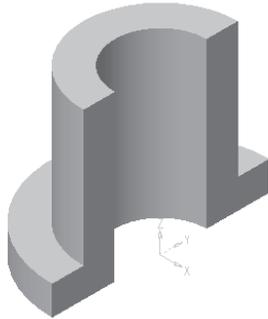


Figure 3-15 Solid feature created by revolving the sketch through an angle of 180-degrees

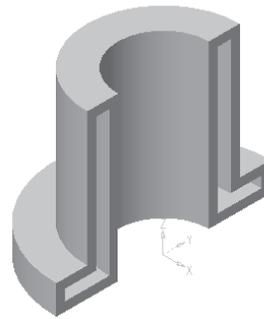


Figure 3-16 Sheet feature created by revolving the sketch through an angle of 180-degrees

DYNAMICALLY ROTATING THE VIEW OF THE DRAWING

Menu: View > View Rotate
Toolbar: Viewing > View Rotate



In EdgeCAM, you can dynamically rotate the model in the 2D or 3D space to view it from all directions. This allows you to visually maneuver around the model so that all its features can be clearly viewed. This tool can be invoked even when you are using some other tool. For example, you can invoke this tool when the **New Component Extrude** dialog box is displayed, and then freely rotate the model in the 3D space to see the preview.

To rotate the model freely in the 2D or 3D space, choose the **Rotate View** button from the **Viewing** toolbar; the cursor will be replaced by the rotate view cursor. Press and hold the left mouse button and drag the cursor to rotate the view. Alternatively, you can rotate the view by pressing and dragging the right-mouse button. You can also rotate the model without invoking the **Rotate View** tool. To do so, press and hold the right mouse button down and drag the cursor.

CREATING PRIMITIVE FEATURES AS BASE FEATURES

EdgeCAM provides you a set of primitive features which can be used as base features. These can be used to create models of simple geometries such as a slab, cone, or sphere. To create the primitive features, you need to specify the parameters that control the basic dimension of the geometrical shape. This process can be effectively used to create basic solid models in a lesser time. The tools used to create primitives models are in the **Application Menu** toolbar. The procedure to create the primitive features is discussed next.

Creating a Slab

Menu: 3D Construction > New Component > Slab
Toolbar: Application Menu > New Component Extrude > New Component Slab



The **Slab** tool is used to create a box of the specified length, width, and height. To create a box as the base feature, select a plane. Then, choose the **New Component Slab** button from the **Application Menu > New Component Extrude** toolbar. Specify a position to place the slab using the left mouse button; a **Slab** of default dimensions will be displayed. Also, the **Slab** dialog box will be displayed, as shown in Figure 3-17. Enter the dimensions of the slab in the **Length[X]**, **Width[Y]**, and **Height[Z]** spinners. Specify a name for the slab, if required, in the **Name** edit box and press ENTER.

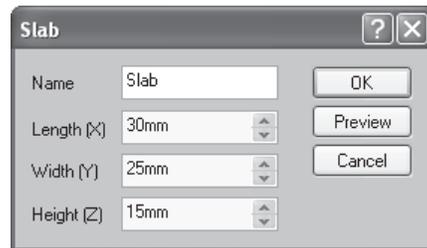


Figure 3-17 The Slab dialog box

Creating a Sphere

Menu: 3D Construction > New Component > Sphere
Toolbar: Application Menu > New Component Extrude > New Component Sphere



To create a sphere as the base feature, select the **New Component Sphere** button from the **Application Menu > New Component Extrude** toolbar. Specify the center of the sphere in the drawing area; a circle with the cursor attached to its periphery will be displayed. Also, the **Sphere** dialog box will be displayed, as shown in Figure 3-18. Enter the radius of the sphere in the **Radius** dialog box. Specify a name for the sphere, if required, in the **Name** edit box and choose the **OK** button. You can also move the cursor and locate the radius using the **Locate** toolbar.

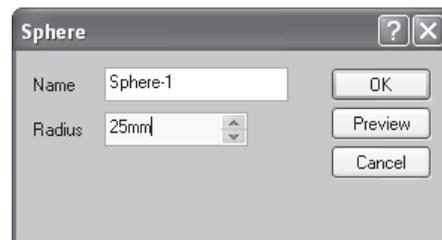


Figure 3-18 The Sphere dialog box

Creating a Cone

Menu: 3D Construction > New Component > Cone
Toolbar: Application Menu > New Component Extrude > New Component Cone



To create the base feature of conical shape, choose the **New Component Cone** button from the **Application Menu > New Component Extrude** toolbar. Specify the center of the base circle of the cone using the left mouse button; the base circle of the cone, with the cursor attached to its periphery will be displayed. Also, the **Cone** dialog box will be displayed, as shown in Figure 3-19. Enter the radius of the base circle in the **Base radius** spinner. You can also move the cursor and locate the radius using the **Locate** toolbar. Next, define the cone by specifying its height or by specifying the angle between the slant height and the vertical height. To define a cone by its height, enter the value in the **Height** spinner, the angle will be calculated automatically and vice versa. To create a truncated cone, specify the radius in the **Top radius** spinner. You will notice the height is also modified automatically.

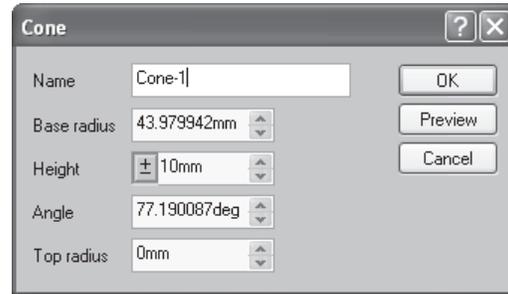


Figure 3-19 The Cone dialog box

Creating a Cylinder

Menu: 3D Construction > New Component > Cylinder
Toolbar: Application Menu > New Component Extrude > New Component Cylinder



To create a cylinder as the base feature, select the **New Component Cylinder** button from the **Application Menu > New Component Extrude** toolbar. Specify the center point of the cylinder; a circle of default radius will be displayed. Also, the **Cylinder** dialog box will be displayed, as shown in Figure 3-20. Specify the radius of the cylinder in the **Radius** spinner and the height of the cylinder in the **Height** spinner. Enter a name for the cylinder in the **Name** edit box, if required, and choose **OK**. After specifying the center, you can also move the cursor and locate the other end of the axis and the radius using the **Locate** toolbar.

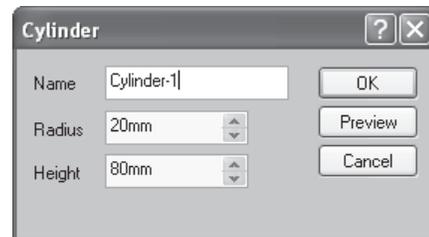


Figure 3-20 The Cylinder dialog box

Creating a Torus

Menu: 3D Construction > New Component > Torus
Toolbar: Application Menu > New Component Extrude > New Component Torus



To create a torus as the base feature, choose the **New Component Torus** button from the **Application Menu > New Component Extrude** toolbar. Specify the center of the torus in the drawing area; a torus of default radius will be displayed. Also, the **Torus** dialog box will be displayed, as shown in Figure 3-21. Specify the radius of the torus in the **Major Radius** spinner. Specify the radius of the tube in the **Minor Radius** spinner. Enter a name for the torus, if required, in the **Name** edit box and choose the **OK** button. You can also move the cursor and locate the radius using the **Locate** toolbar.

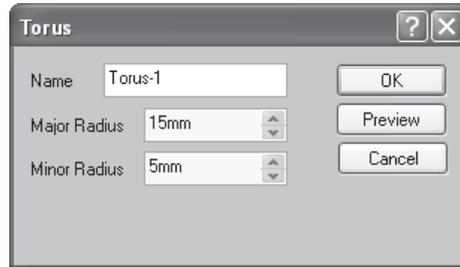


Figure 3-21 The Torus dialog box

Figure 3-22 shows the slab, sphere, and the truncated cone generated using the corresponding tools. Figure 3-23 shows the cylinder and the torus created using the **New Component Cone** and the **New Component Torus** tools from the **Application Menu** toolbar.

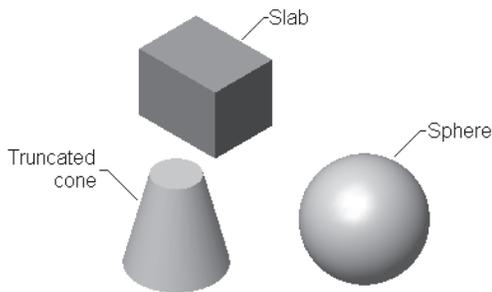


Figure 3-22 The slab, sphere, and the truncated cone

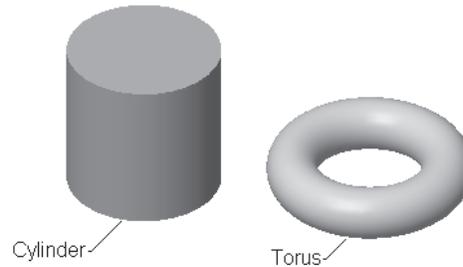


Figure 3-23 The cylinder and the torus

MODIFYING THE VIEW ORIENTATION

In EdgeCAM, you can change the view orientation using some predefined standard views or user-defined views. To invoke these standard views, choose the **Views** button from the **Viewing** toolbar; the **Display View** dialog box will be displayed. You can select the required standard view from the list. Select the **Add** button to save a view other than the standard view. When you choose the **Add** button, the **Add display view** dialog box will be displayed. Enter a new name in the **Name** edit box and choose the **OK** button; the name will be displayed in the dialog box, as shown in Figure 3-24. You can also change this name by choosing the **Rename** button. To rename an added view, select it from the dialog box and choose the **Rename** button; the **Rename display view** dialog box will be displayed. Enter the name and choose **OK**. Select the **Zoom Extents** check box to fit the display of the view on the screen. Select the **Maintain view's position/scale** check box to maintain the scale of the solid.

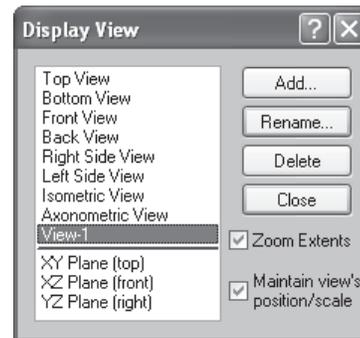


Figure 3-24 The Display View dialog box with an added view

Maintain view's position/scale

DISPLAY MODES OF THE MODEL

EdgeCAM provides you with the various predefined modes to display the model. You can also select any of these display modes from the **Display Mode** toolbar. These modes are discussed next.

Wireframe



When you choose the **Wireframe** button; all the hidden lines are displayed, along with the visible lines in the model. Sometimes, for complex models, it becomes difficult to recognize the visible lines and the hidden lines, if you set this display mode.

Wireframe HLD



When you choose the **Wireframe HLD** button, the model will be displayed in the wireframe and the hidden lines in the model will be displayed as dashed lines.

Wireframe HLR



When you choose the **Wireframe HLR** button, the hidden lines in the model will not be displayed. Only the edges of the faces visible in the current view of the model will be displayed.

Shaded



In this mode, the model is shaded and the edges of the model are not displayed.

Shaded w/HLD Edges



The **Shaded w/HLD Edges** mode is used to display the shaded model with the hidden lines displayed as dashed lines.

Shaded w/HLR Edges



When you choose this mode, the model is shaded and all the hidden lines of the model are removed.

Perspective View



Using this button, you can display the model using the perspective viewing. Once you choose this button, it will be active until you select it again to clear it.

IMPORTANCE OF SKETCHING PLANES

In the previous section, you created a sketch on the principal planes. But most of the mechanical designs consist of multiple sketched features that lie on different planes. On the basis of the design requirement, you can select any one of the principal planes to create the basic feature. To create additional sketched features, you need to select an existing plane or a planar surface or you need to create a new plane that will be used as the sketching plane.

Creating New Planes

In EdgeCAM, there are ten methods of creating planes. In this book, you will learn some of these tools. To create new planes, choose the **Construction Planes** button from the **Application Menu** toolbar. The **Construction Planes** dialog box will be displayed. When you define the parameters for the creation of a new plane, the **Transform** button will be available. Choose the **Transform** button to expand the dialog box, as shown in Figure 3-25. The spinners in the **Plane Offset** area are used to modify the position of the new plane. The **Translation** spinners are used to move the new plane along the X, Y, and Z directions. The **Rotation** spinners are used to rotate the plane along the X, Y, and Z axes. Choose the **Reset X-Form** button to revert to the default position. The methods in the **Construction Planes** dialog box to create new planes are discussed next.

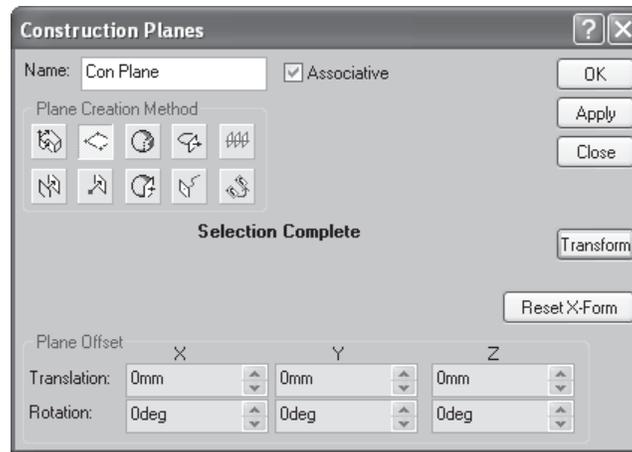


Figure 3-25 The Construction Planes dialog box

Creating a Plane on a Planar Surface



The **On Planar Face** option is used to create a plane on the existing planar surface. As this option is selected by default, move the cursor over any surface and select it as it gets highlighted.

Creating a Plane Using the Position - Line Option



The **Position - Line** option is used to create a plane that passes through two edges or lines, an edge and a point, or three points. The selected point can be a sketched point or a vertex. Note that you need to hold the SHIFT or CTRL key down to make multiple selections. To create a plane using this option, choose the **Position - Line** button from the **Construction Planes** dialog box. Select the first entity from the drawing area; the name of the selected entities will be displayed in the dialog box. Next, hold the CTRL key down and select the second and third entities, if needed; the preview of the plane will be displayed. Choose the **OK** button to create the plane. Select the **Associative** check box to make the orientation of the new plane relative to the referenced entities. Figure 3-26 shows the plane created by selecting three points.

Creating a Plane Tangent to a Surface



The **Tangent to Face at Position** option is used to create a plane tangent to a selected non-planar surface. On selecting this option, you will be prompted to select a non-planar surface. Select the surface from the geometry area; the **U-Value** and **V-Value** spinners will be available and they can be used to place the plane at the required position. The system divides the diameter and the height of the surface into ten parts. **U-Value** is used to snap a point along its diameter and **V-Value** is used to snap a point along its height. Set the value in the spinners and choose the **OK** button. Select the **Associative** check box to make the orientation of the new plane relative to the referenced entities. Figure 3-27 shows the plane created tangent to a selected cylindrical surface.

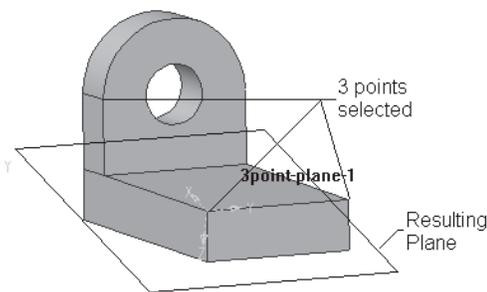


Figure 3-26 Plane created by selecting three points

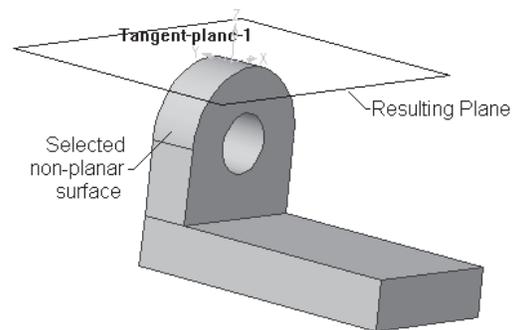


Figure 3-27 Plane created tangent to the selected curved surface

Creating a Plane Normal to a Curve



The **Normal to Curve** option is used to create a plane that is perpendicular to the selected curve. On selecting this option, you will be prompted to select the reference curve. Select the **Chain** check box to select the chain of curves. Select the **Chain Tangent** check box to select the curves that are tangential to the endpoints of the selected curve. Select a curve or an edge of a feature from the geometry area. Specify the number of planes required in the **Total Planes** spinner. To create the plane at an offset, choose the **Transform** button. Enter the offset distance in terms of the percentage from the initial position in the **Distance %** spinner. If you have multiple planes, the plane to be offset is specified in terms of the number in the **Plane** spinner. Figure 3-28 shows the plane created normal to a spline curve and positioned at its start point.



Note

To select an edge as the reference curve, select the feature or the component from the **Model** window.

Creating a Plane at an Offset from an Existing Plane/Planar Face



Choose the **Offset** button from the **Construction Planes** dialog box to create a plane at an offset from an existing plane or planar surface. On selecting this button, you

will be prompted to select a plane or a planar surface. Select a reference entity from the geometry area; the **Total Planes** and **Distance** spinners will be available in the dialog box. Enter the number of planes required in the **Total Planes** spinner. The offset distance is specified in the **Distance** spinner. Select the **Associative** check box to make the orientation of the new planes relative to the selected reference. Figure 3-29 shows the plane created at an offset from the reference plane.

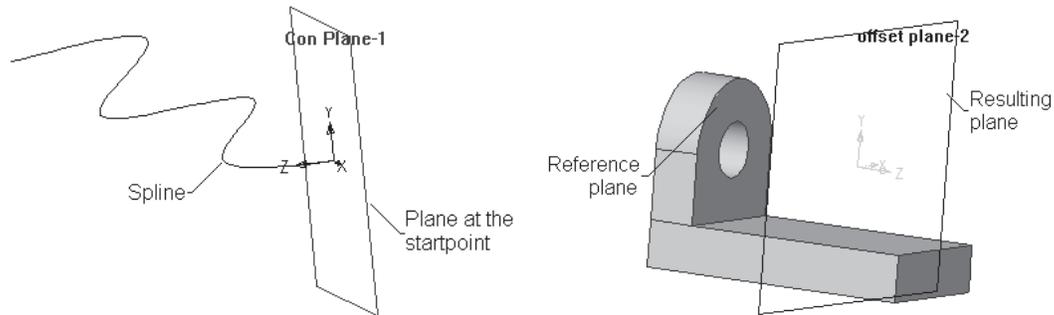


Figure 3-28 Plane created normal to a spline *Figure 3-29 The plane created at an offset from the reference plane*

Creating a Plane Parallel to an Existing Plane and Passing Through a Point



The **Position Parallel to Plane** option is used to create a plane that is parallel to a reference plane or planar face and passes through a specified point. To create a plane using this option, select the **Position Parallel to Plane** option from the **Construction Planes** dialog box; you will be prompted to select a plane or a planar surface and a sketched point or a vertex. Select the plane from the graphics area. Next, press and hold the CTRL key down and select the point through which the plane has to pass; the plane will be created, as shown in Figure 3-30. You can also select the principle plane as the reference plane.

Creating a Plane Tangent to a Non-planar Face and Parallel to a Planar Face



The **Tangent to Face-Parallel to plane** option in the **Construction Planes** dialog box is used to create a plane tangent to the selected surface and parallel to a selected reference plane or planar face. To create a plane using this option, choose the **Tangent to Face-Parallel to plane** button from the **Construction Planes** dialog box; you will be prompted to select a non-planar surface and the reference plane, in any order, from the geometry area. On selecting the entities, temporary planes will be created. Choose the **OK** button to create the plane, as shown in Figure 3-31.



Note

For multiple selections, use the **SHIFT** key.

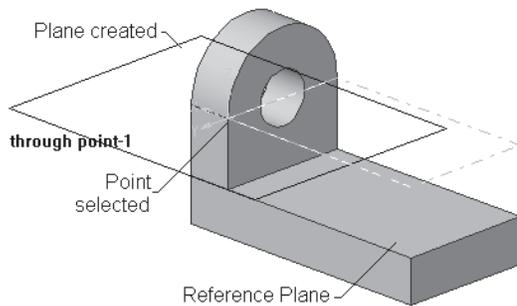


Figure 3-30 Plane created passing through a point and a reference plane

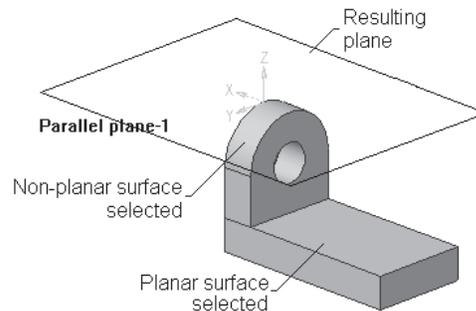


Figure 3-31 Plane created tangent to a nonplanar surface and parallel to a planar surface

CREATING BOSS/CUT/DIVIDE FEATURES

Most engineering designs consist of more than one feature. So, after creating the base feature, you have to add or remove material to complete the design. These added features will be known as the second feature, the third feature, and so on. In EdgeCAM, the base feature is created using the options in the **New Component** tool. If you use the options in **New Component** to create the second feature, a new component will be created and the **Component-2** node will be added in the **Model** window. Also, **Component-2** will be considered as the new base feature that is to be joined with **Component-1** using the **Boolean** operation. So, it is recommended that you choose the **Boss/Cut/Divide** tools from the **Application Menu** toolbar to create additional features. Also, the sketches for these added features will be drawn on the new plane. The **Boss** option is used to add the material and the **Cut** option is used to remove the material from the feature. The **Divide** option is used to divide the surface into two or more solid bodies.

Creating Boss/Cut Extrude Features

Menu: 3D Construction > Boss/Cut/Divide > Extrude
Toolbar: Application Menu > Boss/Cut/Divide Extrude



To create the extruded cut, join, or divide feature, invoke the **Boss/Cut/Divide Extrude** tool in the **Application Menu** toolbar; the **Boss/Cut Extrude** dialog box will be displayed, as shown in Figure 3-32. Select the **Boss** radio button to add the material to the base feature or the plane on which the sketch is drawn. If you select the **Cut** radio button, the material will be removed from the base feature. If the sketch for this feature was drawn on a face of an existing feature, the target body will be automatically selected. However, if the sketch is drawn on a principle plane, then the **Set Target** button will be displayed on the left of the **OK** button in the dialog box. You need to choose this button and select a target body to add or remove the material.

The options in the **Extrude**, **Type**, **Fillet** tabs are the same as those in the **New Component Extrude** dialog box. The options in the **Boss/Cut options** tab are available when you choose the **Surface** radio button from the **Type** tab. These options are used to divide the base feature. The radio buttons in the **Fence** area of the **Boss/Cut options** tab are discussed next.

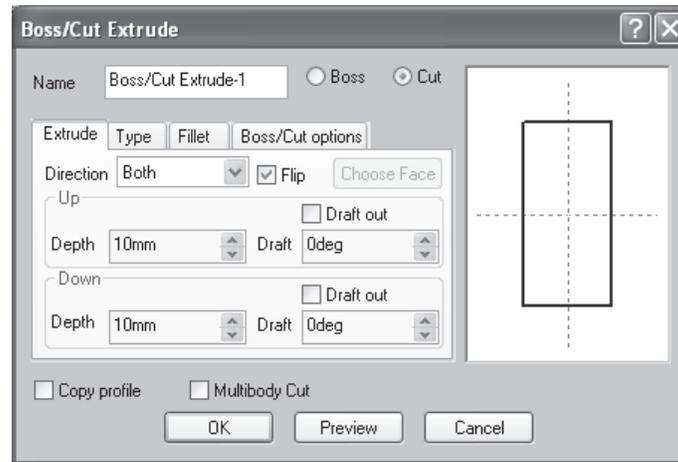


Figure 3-32 The Boss/Cut Extrude dialog box

Both

Select this radio button to keep both the surfaces.

In Front

This radio button is selected to keep the front portion of the surface.

Back

Select this radio button to retain the back surface of the selected surface.

Auto

This radio button is available only if both the entities are surfaces. If you select this radio button, the software selects the best surfaces to retain.

Multibody cut

Select this check box if the cut has to pass through multiple bodies.

Creating Boss/Cut Revolve Features

Menu:	3D Construction > Boss/Cut/Divide > Revolve
Toolbar:	Application Menu > Boss/Cut/Divide Extrude > Boss/Cut/Divide Revolve



The sketches drawn on the new planes are converted into features by rotating them about an axis. Choose the **Boss/Cut/Divide Revolve** tool in the **Application Menu** > **Boss/Cut/Divide Extrude** toolbar; the **Boss/Cut Revolve** dialog box will be displayed, as shown in Figure 3-33. Select the **Boss** radio button to add the material. If you select the **Cut** radio button, the material will be removed. Set the parameters and choose the **OK** button. The options in the **Extrude**, **Type**, and **Fillet** tabs are the same as the **New Component Extrude** dialog box. Also, the options in the **Boss/Cut Options** tab are the same as that of the **Boss/Cut Extrude** dialog box.

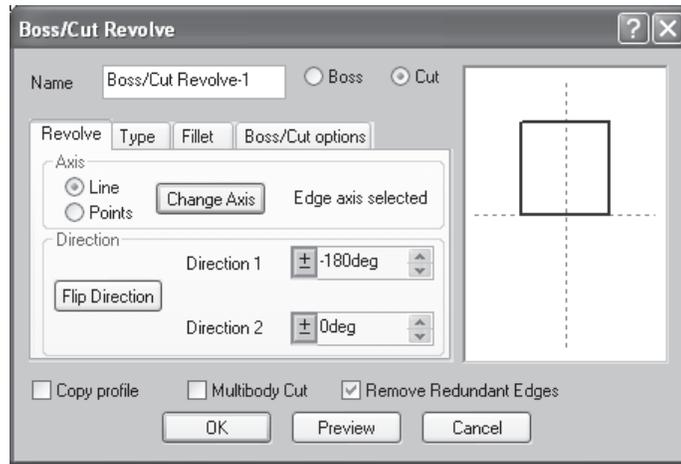


Figure 3-33 The Boss/Cut Revolve dialog box

Figure 3-34 shows the profile and the additional feature created by the **Boss/Cut Extrude** tool. Figure 3-35 shows the section view of the model with the sketch and the additional feature created by the **Boss/Cut Revolve** tool.

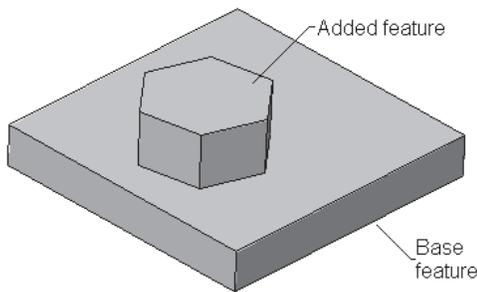


Figure 3-34 Feature created by the Boss/Cut Extrude tool

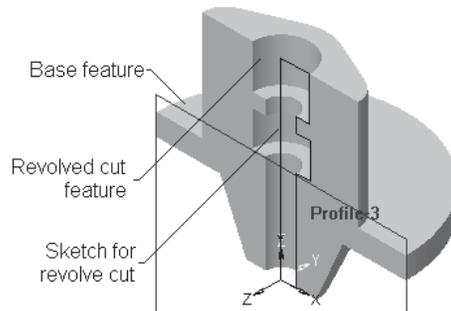


Figure 3-35 Feature created by the Boss/Cut Revolve tool

ADVANCED MODELING TOOLS

EdgeCAM provides you various advanced modeling tools that help in creating a better and accurate design by capturing the design intent in the model. For example, instead of creating a hole by the cut revolve options, you can place a hole directly at the required location using the **Drill Hole** option. The advanced modeling tools are discussed next.

Creating Fillets

Menu: 3D Construction > Feature > Fillet Blend
Toolbar: Application Menu > Fillet Blend Feature



The fillet tool is used to round an internal or external face or an edge of a model. You can also use the advanced fillet options to add advanced fillets to the model. In EdgeCAM, you can add fillets as a feature to the model using the **Fillet Blend Feature** tool. As discussed earlier, you can also add fillets within a sketch. But adding fillets to a sketch is not a good practice from the design point of view. This is because you have to keep the sketch as simple as possible. To create a fillet, choose the **Fillet Blend Feature** button from the **Application Menu** toolbar; the **Fillet Blend Parameters** dialog box will be displayed. Enter the fillet radius in the **Radius** spinner and select the edges from the drawing area. Choose the **OK** button; the fillets will be created according to the specified radius. To create a fillet at different edges and of different radii, choose the **Apply** button, change the radius and select the other edges. The **Go Back** button is used to revert the fillet applied. If the **Propagate** check box is selected, you can select the chain of tangentially edges connected by selecting any one of the edges.



Note

To select the loop of edges, move the cursor near a corner and press the left mouse button when the cursor changes to the loop cursor.

To create a fillet of different radii, select the **Variable Radius** check box; the expanded form of the dialog box will be displayed, as shown in Figure 3-36. Also, the start point and the endpoint of the fillet will be represented as nodes at the vertices. The buttons and the spinners in the **Variable Radius Options** area are used to create fillets of different radii and are discussed next.

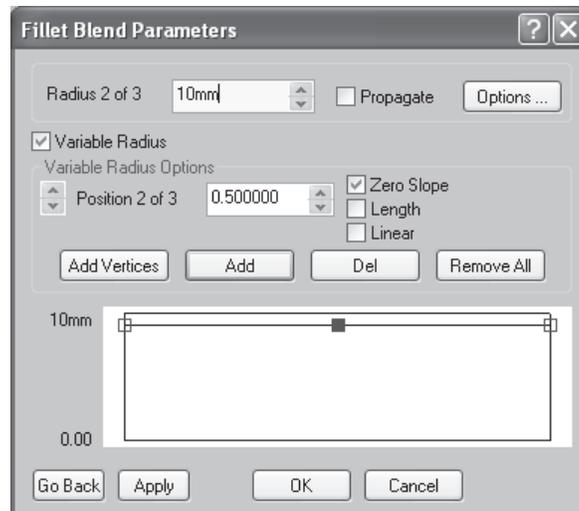


Figure 3-36 The *Fillet Blend Parameters* dialog box

Up/Down Arrows

The up and down arrows allow you to choose the nodes created. The selected node will be highlighted in the graphics window and on the model. You can select the node directly from the graphics window also.

Position Spinner

The value in the position spinner decides the placement of the node. The check boxes next to it are used for defining the method of placement and are discussed next.

Zero Slope

Select this check box if you do not require a slope at the selected node.

Length

Select this check box to position the node using the current measuring units.

Linear

Select this radio button to position the node in terms of the percentage of its length.



Note

1. You can select all the three check boxes also.
2. You can also specify the position and slope by selecting the corresponding check boxes for each node.
3. The radius for each node is specified in the radius spinner after selecting the node in the graphics window.

Add Vertices

If you have selected multiple edges, the nodes will be at only two ends. Select the **Add Vertices** button to create nodes at all the vertices between the two existing nodes.

Add

Choose the **Add** button to create additional nodes between the vertices of the selected edge. The nodes will be created to the right of the selected node. If you have multiple nodes, select the node from the graphics window that represents the vertex of the selected edge.

Del

This button is used to remove a node from an edge. Choose this button after selecting the node from the graphics window.

Remove All

Choose this button to remove all the nodes except those at the ends.

When the radius of the fillet is too large, the fillet spreads over the adjacent faces. This can be controlled by choosing the **Options** button from the **Fillet Blend Parameters** dialog box. On choosing the **Options** button, the **Blend Options** dialog box will be displayed, as shown in Figure 3-37. The radio buttons in the **Blend Options** dialog box are discussed next.

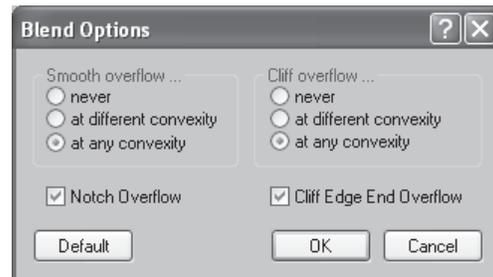


Figure 3-37 The **Blend Options** dialog box

Smooth overflow Area

never

If you select this radio button, the fillet is created only on the selected face.

at different convexity

Select this radio button to make the fillet curve flow over the other surface only if the face curvature points in a different direction.

at any convexity

Select this radio button to make the fillet curve flow over any adjacent surface.

Cliff overflow Area

never

If this radio button is selected, the blend goes beyond the face and it is extended or trimmed.

at different convexity

Select this radio button to make the cliff flow over the opposite surface if the face curvature points in a different direction.

at any convexity

Select this radio button to make the fillet curve flow over an adjacent surface.

Notch overflow

Select this check box to make the fillet flow over a cliff and an edge.

Cliff edge end overflow

Select this check box to trim the fillet if it flows over the other edge.

Figure 3-38 shows the fillet created by selecting the **never** radio button from the **Blend Options** dialog box. Figure 3-39 shows the fillet created by selecting the **at any convexity** radio button from the **Blend Options** dialog box.

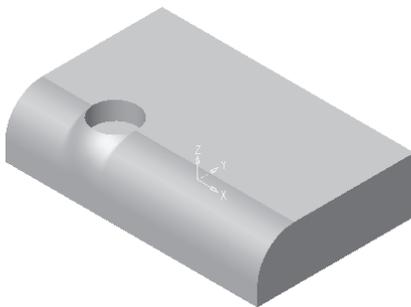


Figure 3-38 Fillet created by selecting the **never** radio button from the **Blend Options** dialog box

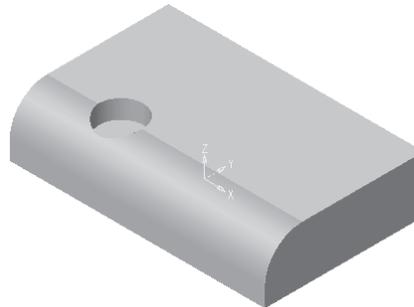


Figure 3-39 Fillet created by selecting the **at any convexity** radio button from the **Blend Options** dialog box

Creating Chamfers

Menu: 3D Construction > Feature > Chamfers
Toolbar: Application menu > Fillet Blend Feature > Chamfer Feature



Chamfering is defined as a process in which the sharp edges are beveled in order to reduce the area of stress concentration. The process also eliminates the sharp edges and corners that are not desirable. In EdgeCAM, a chamfer is created using the

Chamfer Feature tool in the **Application Menu > Fillet Blend Feature** toolbar. When you invoke this tool, the **Solid Chamfer** dialog box will be displayed, as shown in Figure 3-40. The radio buttons in the **Methods** area decide the type of chamfer creation. The spinners in the **Value** area specify the dimension for the parameters to create a chamfer. Select the radio button from the **Methods** area, based on the parameters of the chamfer, as shown in Figure 3-41. Enter the corresponding chamfer distance and angle in the spinners in the **Value** area. Then, select the edge to which the chamfer has to be applied and choose the **OK** button; the chamfer will be created. To create the chamfer at different lengths and different angles, choose the **Apply** button, change the dimensions, and select the other edges. The **Go Back** button is used to revert the chamfer applied. If the **Propagate** check box is selected, you can select the chain of tangentially connected edges by selecting any one edge from the chain. Select the **Flip** button to toggle between the edges. Various types of chamfers options available in the **Solid Chamfer** dialog box are discussed next.

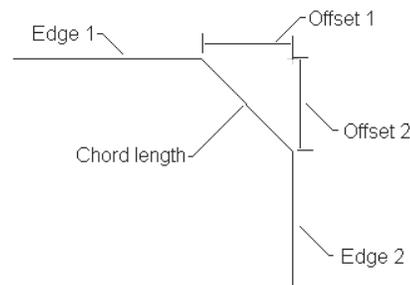
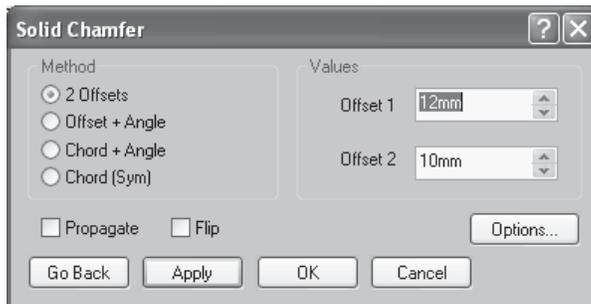


Figure 3-40 The Solid Chamfer dialog box

2 Offsets

This radio button is selected by default and is used to apply chamfers to an edge, as shown in Figure 3-42. Enter the chamfer distance in the **Offset1** and **Offset 2** spinners.

Offset + Angle

Select this radio button to apply the chamfer by specifying the chamfer distance from one face on a selected edge and at an angle with the adjoining face, as shown in Figure 3-43. Enter the chamfer distance in the **Offset** spinner and the angular value in the **Angle** spinner.

Chord + Angle

Select this radio button to apply the chamfer by specifying the chamfer length and the angle it makes with one of the selected faces. Enter the length of the chamfer in the **Chord** spinner and the angle in the **Angle** spinner.

Chord Sym

This radio button is selected to create a chamfer of a specified chord length and a constant angle (45-degrees) on both the faces. Enter the chord length in the **Chord** spinner.

When you choose the **Options** button, the **Blend Options** dialog box will be displayed. The functions of the options are the same as those discussed in the **Fillet Blend Parameters** dialog box.

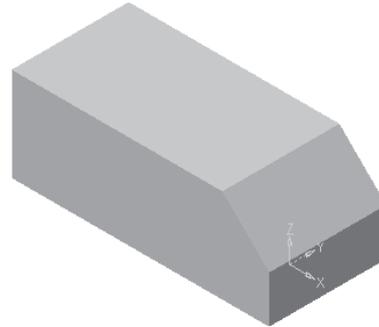
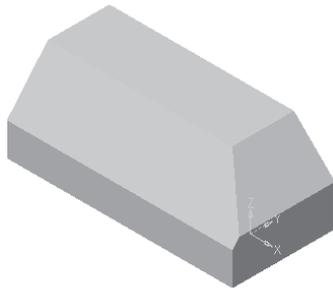


Figure 3-42 Chamfer created using the 2 Offsets *Figure 3-43 Chamfer created using the Offset + Angle option*

Adding a Draft to the Faces of the Model

Menu:	3D Construction > Feature > Draft
Toolbar:	Application Menu > Fillet Blend Feature > Draft Feature



A draft is defined as the process of adding a taper angle to the faces of the model. Adding a draft to the faces of the model is one of the most important operations, especially for the components that need to be cast, molded, or formed. To create a draft surface, choose the **Draft** button from the **Application Menu > Fillet Blend Feature** toolbar; the **Draft Feature** dialog box will be displayed, as shown in Figure 3-44. Select the reference plane from the model, as the **Ref Plane** button is chosen by default. Next, select the faces to be drafted as the **Target Face** button is automatically chosen. Enter the draft angle in the **Angle** spinner. Choose the **+/-** button in the spinner to give a negative value. On choosing the **Preview** button, the draft to be created will be displayed, as shown in Figure 3-45. Choose **OK** after setting the required parameters. The options in the **Draft Faces** drop-down list are used to define the faces to be selected as the target surface. The options are discussed next.

Only Selected Faces

This option allows you to apply a draft to the selected face.

All Faces

This option is selected to apply a draft to all the faces adjoining the reference plane.

Faces Outside Ref Plane

This option is selected to apply a draft to the surfaces outside the reference plane.

Faces inside Ref Plane

Select this option to apply a draft to the surfaces inside the reference plane.

All filleted Faces

Select this option to apply a draft to the filleted faces adjoining the reference plane.

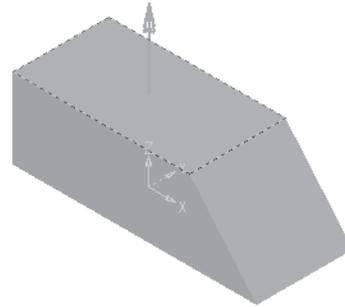
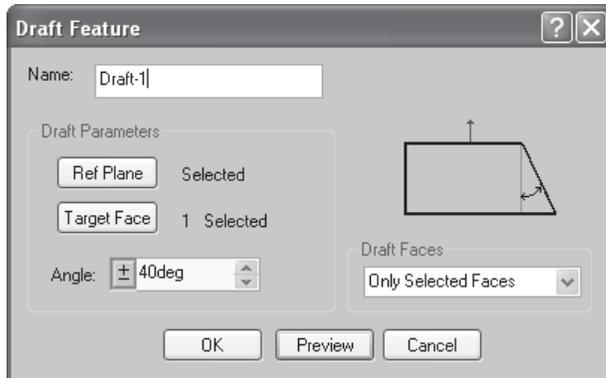


Figure 3-44 The Draft Feature dialog box

Creating a Shell Feature

Menu: 3D Construction > Feature > Shell

Toolbar: Application Menu > Fillet Blend Feature > Shell feature



The **Shell** tool is used to scoop out the material from the model and remove the selected faces, resulting in a thin walled structure. To create a shell feature, choose the **Shell Feature** button from the **Application Menu > Fillet Blend Feature** toolbar; the **Shell** dialog box will be displayed, as shown in Figure 3-46. Enter the required wall thickness in the **Thickness** spinner. Select the **Inward** check box to make the wall inside the selected face. Select a face that you need to remove from the model. The **Change Open Face(s)** button is used to change the selected face.

To create a multithickness shell feature, select the **Enable Multi Thickness** check box and choose the **Multi-Thickness** tab. Hold the SHIFT key down and select the faces to apply a different thickness. The selected surfaces will be numbered in the order of selection and displayed in the window. Select the surface from the window and change the thickness by entering a new value in the **Thickness** spinner. Select the **Change M-Thick Face(s)** button to change the selected faces, if any. A model shelled with a different wall thickness is shown in Figure 3-47.

Creating a Linear Array

Menu: 3D Construction > Feature > Linear Array

Toolbar: Application Menu > Fillet Blend Feature > Linear Array



As discussed in the previous chapter, you can arrange the sketched entities in a rectangular arrangement or pattern. In the same manner, you can also arrange the

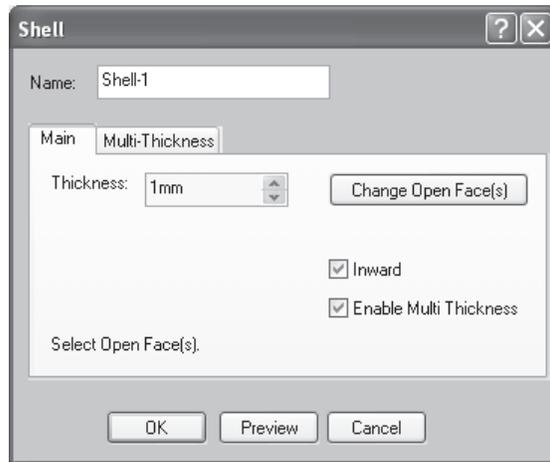


Figure 3-46 The *Shell* dialog box

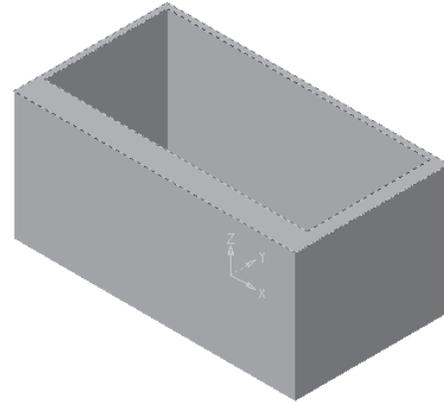


Figure 3-47 Shell feature with variable thickness

features, faces, and bodies in a rectangular pattern. To create a linear pattern, choose the **Linear Array** button from the **Application Menu** > **Fillet Blend Feature** toolbar; the **Linear Array** dialog box will be displayed, as shown in Figure 3-48. The **Select Planar Face Reference** button is selected by default, so select the plane in which the arrayed components will be placed. Now, the buttons in the **Selected Items** area will be available. Choose the **Procedures** button to select the feature from the **Model** window. To select a particular face, choose the **Faces** button and select the feature or the face to be arrayed from the **Model** window or from the geometry area. Set the distance and the number of instances required along the horizontal axis in the **X Offset** spinner and the **X Instances** spinner, respectively. Also, set the distance and the number of instances required along the vertical axis in the **Y Offset** spinner and the **Y Instances** spinner, respectively. If the instances are to be placed at an angle, set the angular value in the **Angle** spinner in the **X-Direction** and **Y-Direction** areas. The **Select X Edge** and **Select Y Edge** buttons are used to select an edge in order to arrange the instances in the specified angle with respect to the horizontal and vertical axes, respectively. Choose the **Reset Edge** button to revert back to the

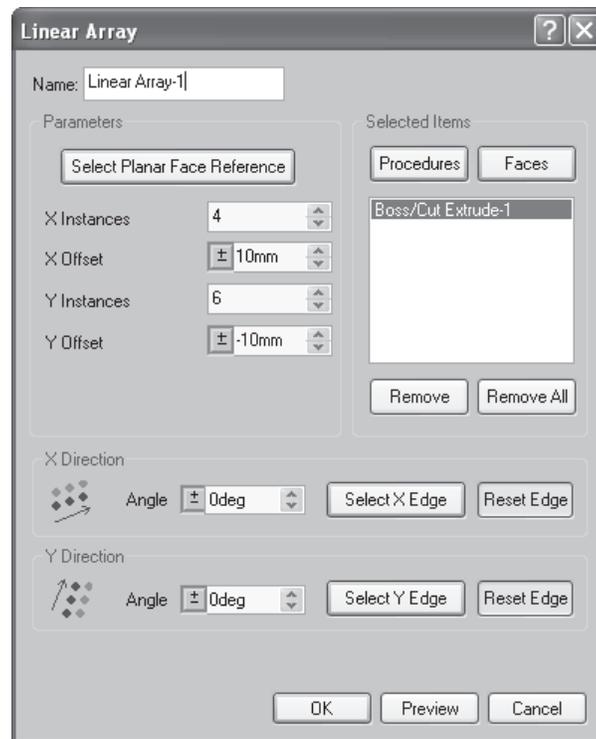


Figure 3-48 The *Linear Array* dialog box

initial position. Figure 3-49 shows the feature to be arrayed and Figure 3-50 shows the resulting linear array.

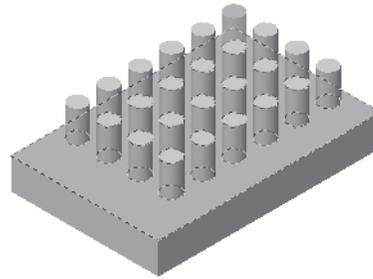
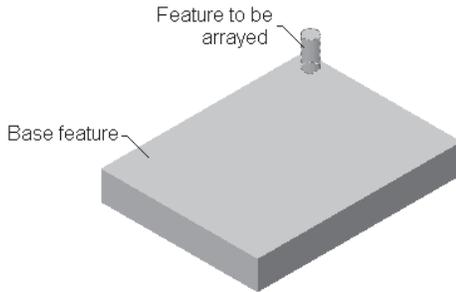


Figure 3-49 The feature to be patterned

Figure 3-50 Resulting linear array

Creating a Circular Array

Menu: 3D Construction > Feature > Circular Array
Toolbar: Application Menu > Fillet Blend Feature > Circular Array

 You can also create instances of the features, faces, and bodies and arrange them in a circle. This is called a circular array. It is also called the polar array. To create a circular array, choose the **Circular Array** button from the **Application Menu > Fillet Blend Feature** toolbar; the **Circular Array** dialog box will be displayed, as shown in Figure 3-51. The **Select Reference Axis** button is chosen by default. Select the reference axis about which the circular pattern will be created. To select the axis of a cylinder or a cone, select the nonplanar surface of the cylinder or the cone. Alternatively, select a planar face and a point. After selecting an axis, choose the **Procedures** button or the **Faces** button from the **Selected Items** area. Select the procedure or the faces to be arranged from the **Model** window. You can also select from the model environment. When you select the procedures or the faces, they will be listed in the window below the buttons. Specify the number of copies required in the **Angular Instances** spinner. Also, specify the angle at which the copies are to be placed in the **Angular Offset** spinner. Choose the **+/-** button to assign a negative value. Choose the **Preview** button to view the proposed arrangement and choose the **OK** button to accept it.

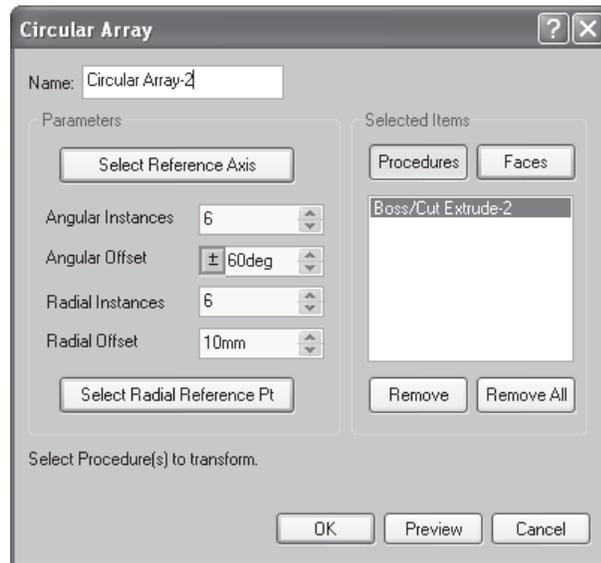


Figure 3-51 The Circular Array dialog box

To create instances radially, set the number of copies in the **Radial Instances** spinner. Also, set the radial offset distance in the **Radial Offset** spinner. To remove any selected face or procedure, select it in the **Selected Items** list box and choose **Remove**. Choose the **Remove all** button to clear the entire selection. Figure 3-52 shows the feature to be patterned and Figure 3-53 shows the resulting circular array.

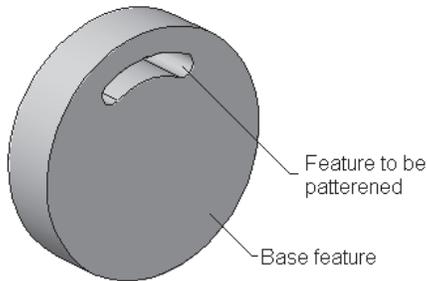


Figure 3-52 The feature to be patterned

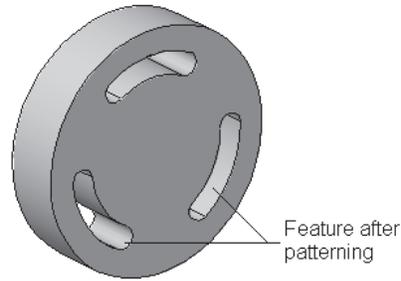


Figure 3-53 Resulting circular array

Creating Simple Holes

Menu: 3D Construction > Feature > Drill Hole
Toolbar: Application Menu > Fillet Blend Feature > Drill Hole



The **Drill Hole** option is used to add standard holes such as counter bore, counter sink, simple, and tapered holes. Before creating holes, define the location of the holes using the **Point** tool and make sure it is selected. Next, choose the **Drill Hole Feature** button from the **Application Menu > Fillet Blend Feature** toolbar; the **Drill Hole** dialog box will be displayed, as shown in Figure 3-54. Selecting the **Copy Point(s)** check box forces the point to become a part of the feature. If you select the **Create X and Y seams** check box, the drill seams will be created in the 2d drawing file while creating the manufacturing drawing. The **Diameter Depth** radio button is selected to specify the shoulder depth of the hole. The **Point Depth** radio button is selected to specify a hole up to the tip. Select the hole standard required from the **Standard** drop-down list. Also, select the standard size of the hole type from the **Nominal Size** drop-down list. The dimensioning units are selected from the **Units** area. To create a hole of a specified depth, select the **Blind** radio button and enter the value in the **Depth to Shoulder** spinner. Select the **Through** radio button to create a through hole. Also, specify the number of walls to drill if there are multiple walls. Select the **Tapped** check box to expand the dialog box. The tapping parameters will be specified in the expanded area. The size of the drill required to create the thread is entered in the **Drill Size** spinner. Set the pitch of the thread in the **Pitch/TPI** spinner. Select the **Full Depth** check box to create full length holes. To create the thread up to a particular depth, specify it in the **Depth** spinner. Select the **Right** or the **Left** radio button to specify the direction of the thread.



Note

To know the name of a spinner near the graphics window, move the cursor in the value region; the name of the spinner will be displayed.

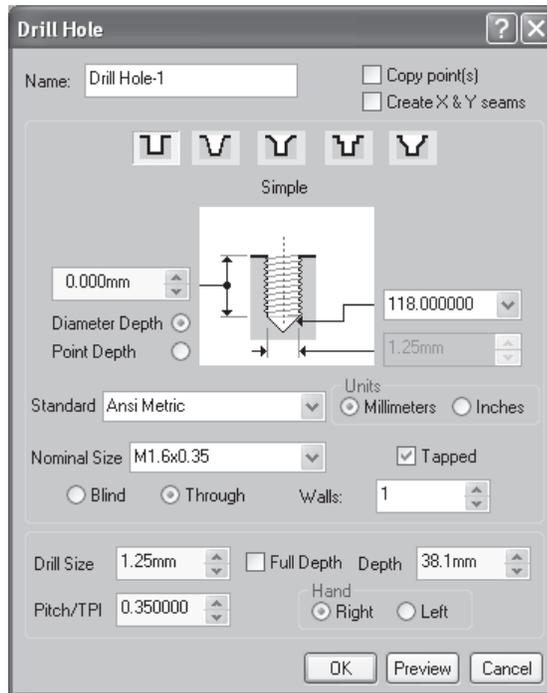


Figure 3-54 The Drill Hole dialog box

Various options for creating a hole are discussed next.

Creating Simple Holes

To create a drill hole, choose the **Simple Hole** button; the parameters for the simple hole will be displayed in the graphics window, as shown in Figure 3-55. Also, the points on the selected surface will be highlighted. Specify the parameters in the corresponding spinners and choose the **OK** button.

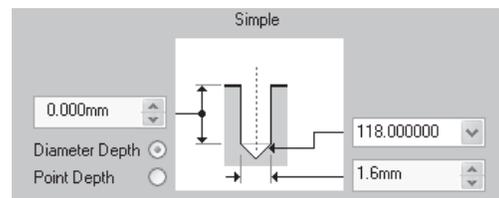


Figure 3-55 The parameters for the drill holes

Creating Tapered Holes

To create a tapered hole, choose the **Tapered Hole** button; the parameters for the tapered hole will be displayed in the graphics window, as shown in Figure 3-56. Also, the points on the selected surface will be highlighted. Specify the parameters in the corresponding spinner and choose the **OK** button.

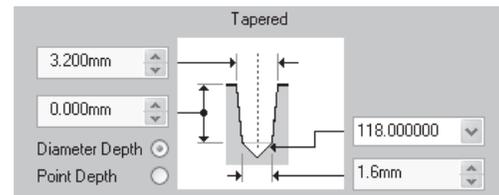


Figure 3-56 The parameters for the tapered holes

Creating Counter Sink Holes

 To create a counter sink hole, choose the **Counter Sink Hole** button; the parameters for the counter sink hole will be displayed in the graphics window, as shown in Figure 3-57. Also, the points on the selected surface will be highlighted. Specify the parameters in the corresponding spinner and choose the **OK** button.

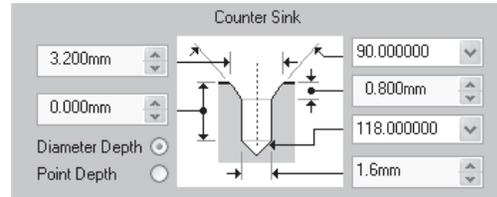


Figure 3-57 The parameters for the counter sink holes

Creating Counter Bore Holes

 To create a counter bore hole, choose the **Counter Bore Hole** button; the parameters for the counter bore hole will be displayed in the graphics window, as shown in Figure 3-58. Also, the points on the selected surface will be highlighted. Specify the parameters in the corresponding spinner and choose the **OK** button.

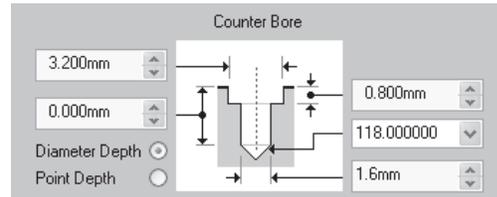


Figure 3-58 The parameters for the counter bore holes

Creating Counter Drill Holes

 To create a counter drill hole, choose the **Counter Drill Hole** button; the parameters for the counter drill hole will be displayed in the graphics window, as shown in Figure 3-59. Also, the points on the selected surface will be highlighted. Specify the parameters in the corresponding spinner and choose **OK**.

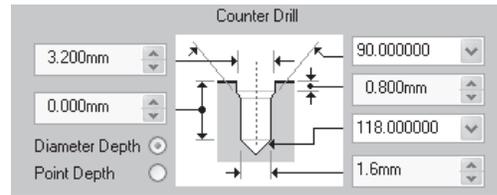


Figure 3-59 The parameters for the counter drill holes

Figure 3-60 shows the sectional view of a model with a tapered hole and a simple hole. Figure 3-61 shows the sectional view of a model with the counter sink hole, counter bore hole, and counter drill hole.

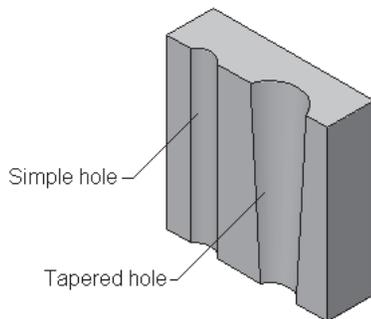


Figure 3-60 The cross-section of a model with the tapered hole and the simple hole.

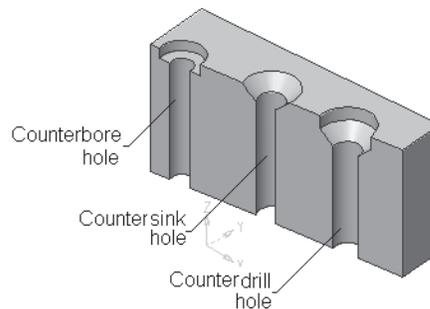


Figure 3-61 The cross-section of a model with the counter sink hole, counter bore hole, and counter drill hole.

EDITING FEATURES OF THE MODEL

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, EdgeCAM Part Modeler is a parametric software. Also, the design created in EdgeCAM Part Modeler is a combination of individual features integrated together to form a solid model. All these features can be edited individually. For example, Figure 3-62 shows a base plate with simple holes.

To replace the four simple holes with four counter sink holes, you need to perform an editing operation. Using the editing operation, you will change the simple holes to the counter sink holes. For editing the holes, select the hole feature from the **Model** window and right-click to invoke the shortcut menu. Choose the **Edit Procedure** option from the shortcut menu to invoke the **Drill Hole** dialog box. Choose the **Counter Sink Hole** button and set the parameters. The drilled holes will be automatically replaced by counter sink holes. Figure 3-63 shows the base plate with drilled holes modified to the counter sink holes.

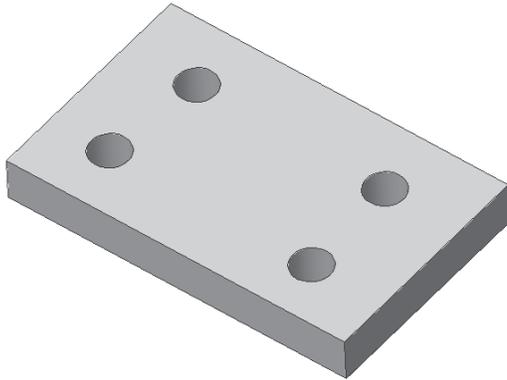


Figure 3-62 Part with four simple holes

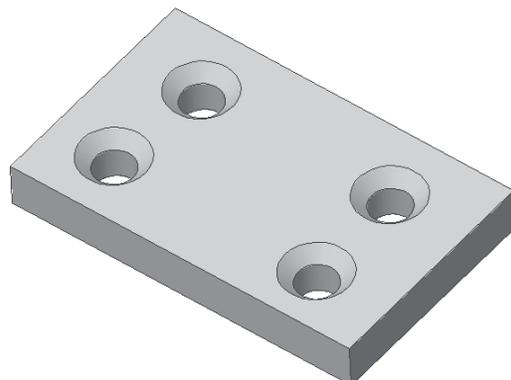


Figure 3-63 Modified part with counter sink holes

To edit the sketch of a feature, click on the + sign on the left to expand the feature in the **Model** window. Select the **Profile** node; the corresponding sketch will be available in the drawing area. Using the standard sketching tools, edit the sketch and choose the **Rebuild** button from the **Standard** toolbar to update the changes made in the sketch. You can also select the **Rebuild** option by invoking the shortcut menu by right-clicking the mouse button. Similarly, to edit the construction plane, choose the **Profile** node, right-click and select the **Edit Construction Plane** from the shortcut menu. The **Construction Planes** dialog box will be displayed. Enter the distance or the angle in the corresponding spinners in the **Planes offset** area and choose the **OK** button.



Note

For a better display of the sketch, choose the **2D Wireframe** button below the drawing area while editing. Choose the **Component** button to view the model only and the **All** button to view both the sketch and the model.

Suppressing Features

Sometimes, you may not require a feature to be displayed in the model or in its drawing views. Instead of deleting it, you can suppress the feature. When you suppress a feature, it will not be visible in the model or in the drawing views. You can view the feature by unsuppressing it. To suppress a feature, select it from the **Model** window, right-click and clear the **Enable Procedure** option. Similarly, to unsuppress the feature, invoke the shortcut menu and select the **Enable Procedure** option.



Note

If you suppress a feature that has other features dependent on it, the dependent features will also be suppressed. Therefore, it is recommended that the dependent feature should be suppressed before suppressing the parent feature.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model and section it. The model is shown in Figure 3-64. The views and dimensions of the model are shown in Figure 3-65. **(Expected Time: 45 min)**

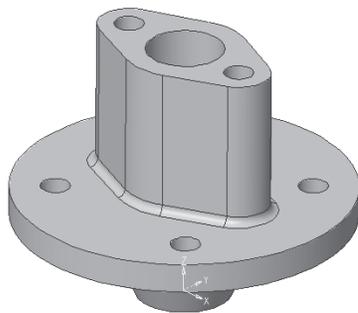


Figure 3-64 Model for Tutorial 1

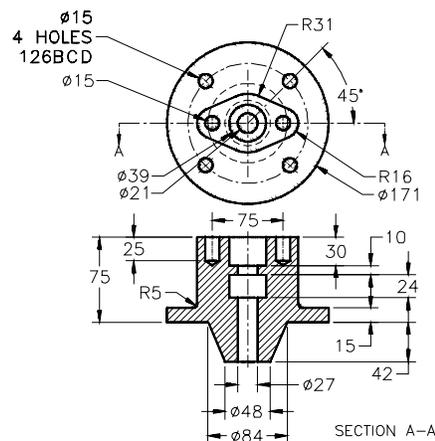


Figure 3-65 Dimensions for the model

The steps to be followed to complete this tutorial are listed next.

- Create the base feature of the model by revolving the sketch around its central axis, refer to Figure 3-66 and Figure 3-67 .
- Draw the sketch for the second feature on the top face of the base feature and extrude it to the given dimension, refer to Figure 3-68.
- Create the revolve cut feature, refer to Figures 3-70 and 3-71
- Create a counter bore hole using the **Drill Hole Feature** tool, refer to Figure 3-72.
- Create a hole using the **Drill Hole Feature** tool and pattern it using the **Circular Array** tool, refer to Figure 3-73.

- f. Apply the fillet and section the model.

Creating the Base Feature

First, you will create the base feature of the model by revolving the sketch created on the XZ Plane (front).

1. Start a new model application by choosing the **New** button from the **Standard** toolbar.
2. Select the **XZ Plane (front)** plane from the **Model** window and choose the **Set the View to Const. Plane** button from the **Viewing** toolbar. 
3. Draw the sketch of the base feature on the XZ plane. Add the required relations and dimensions to the sketch, as shown in Figure 3-66.
4. Invoke the **New Component Revolve** tool from the **Application Menu > New Component Extrude** toolbar. Choose the **Select Axis** button and select the left vertical line of the sketch as the axis for the revolve feature. 
5. Set **360-degrees** as the angle of revolution in the **Direction 1** spinner. Select the **Type** tab and select the **Solid** radio button from the **Body Type** area, if it is not already selected.
6. Select the **Remove Redundant Edges** check box and choose **OK**.
7. Choose the **Shaded w/HLR Edges** button from the **Display Mode** toolbar. 
8. Choose the **Views** button from the **Viewing** toolbar; the **Display View** dialog box is displayed. Select the **Isometric View** option. Now press and hold the right mouse button and drag the cursor. A 3D view of the base feature created, after revolving the sketch, is shown in Figure 3-67.

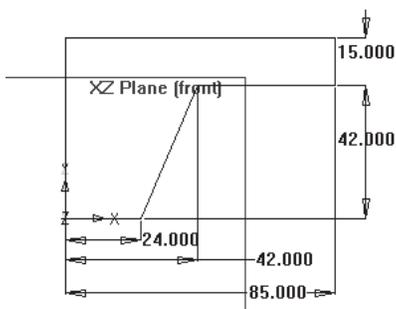


Figure 3-66 Sketch for the base feature



Figure 3-67 3D View of the base feature

Creating the Second Feature

The second feature of this model is an extruded feature. It is created by extruding the sketch created on the top planar face of the base feature.

1. Select the top planar face of the base feature as the sketching plane, draw the sketch for the second feature, and apply the required relations and dimensions to it, as shown in Figure 3-68.
2. Invoke the **Boss/Cut/Divide Extrude** tool from the **Application Menu** toolbar.  Select the **Up** option from the **Direction** drop-down list.
3. Set the **Depth** spinner in the **Up** area to **75 mm** and choose **OK**; the model is created, as shown in Figure 3-69.

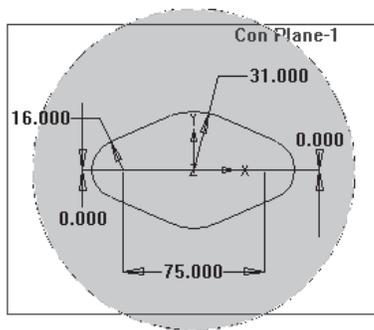


Figure 3-68 Sketch for the second feature

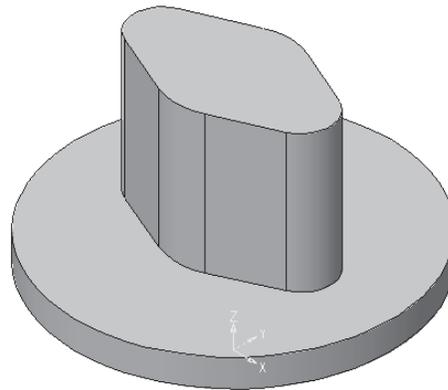


Figure 3-69 Second feature added to the model

Creating the Third Feature

The third feature of the model is created by revolving the sketch using the **Cut** option. The sketch for this feature is created on the **XZ Plane (front)**.

1. Select the **XZ Plane (front)** plane from the **Model** window and choose **Set the View to Const. Plane** button from the **Viewing** toolbar.
2. Choose the **Wireframe** button from the **Display Mode** toolbar. Draw the sketch for the revolved cut feature and apply the required relations and dimensions to it, as shown in Figure 3-70. 
3. Invoke the **Boss/Cut/Divide Revolve** tool from the **Application Menu > Boss/Cut/Divide Extrude** toolbar and select the **Cut** radio button. 
4. Choose the **Select Axis** button and select the vertical line of 132 mm as the axis of revolution. Set **360-degrees** as the angle of revolution in the **Direction 1** spinner.
5. Choose the **Type** tab and from the **Body type** area and select the **Solid** radio button. Select the **Remove Redundant Edges** check box.
6. Choose the **Set Target** button and select the base feature. Choose the **OK** button. The third feature is created and displayed, as shown in Figure 3-71.

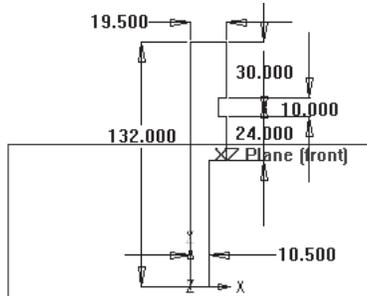


Figure 3-70 Sketch for the revolve cut feature

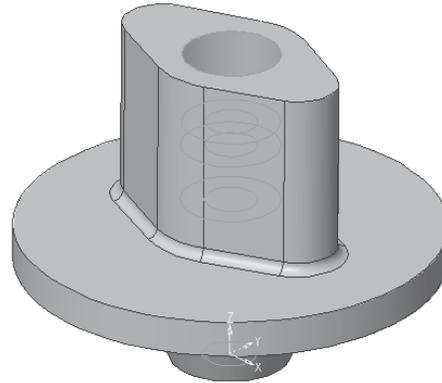


Figure 3-71 Third feature added

Creating the Hole Feature

The holes are created by choosing the **Hole Drill Feature** button. Before choosing the **Hole Drill Feature** tool, you need to specify the location of the hole.

1. Locate two points on the top surface of the second feature to create the 15 mm diameter hole. You can make the points concentric with the arcs.
2. Invoke the **Drill Hole Feature** tool and select the **Custom** option from the **Standard** drop-down list. Set the value as **15 mm** in the **Drill diameter** spinner. 
3. Select the **Diameter Depth** radio button and set the value of **25 mm** in the **Depth to shoulder** spinner and choose **OK**; the holes are created, as shown in Figure 3-72.

To create holes along 126 BCD, you need to select the top surface of the base feature, draw a circle, and place the points at an angle using the **Line** tool. Then, convert the circle and the line to construction entities.

4. Select the top face of the feature and draw a circle of radius 63 mm using the **Circle** tool. The center of the circle should be at the origin.
5. Place a point on the circle at an angle close to 45-degrees and apply the coincident relation to it.
6. Draw two lines, one passing through the center of the circle and the point, and the other passing through the center and coincident to the X-axis.
7. Apply a dimension of 45-degrees between the two lines.
8. Hold the CTRL key down and select the lines and the circle and right-click to invoke the shortcut menu.

9. Choose the **Toggle Construction** option from the shortcut menu.
10. Invoke the **Drill Hole Feature** tool. Select the **Custom** option from the **Standard** drop-down list and set the value as **15 mm** in the **Drill diameter** spinner.
11. Select the **Diameter Depth** radio button and set the value of **25 mm** in the **Depth to shoulder** spinner and choose **OK**. The model, after creating the hole is displayed, as shown in Figure 3-72.

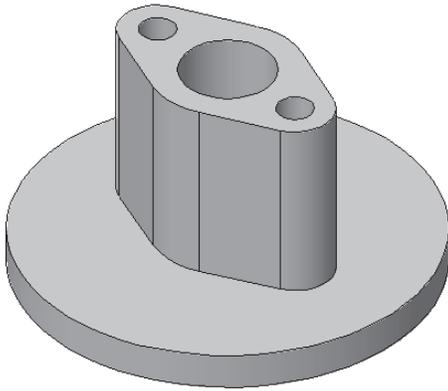


Figure 3-72 Model after creating the counter bore hole

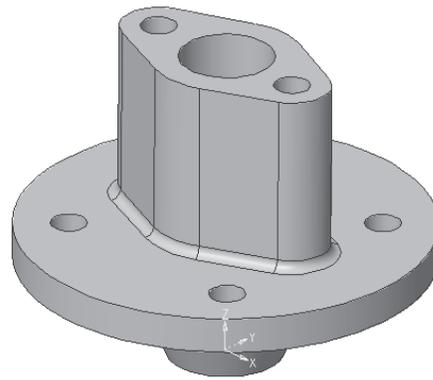


Figure 3-73 Model after creating four holes and the fillet

Next, you need to create the circular pattern of the hole using the **Circular Array** tool.

12. Invoke the **Circular Array** tool and select the face of the hole with a 39 mm radius in the second feature to create an axis. 
13. Choose the **Procedure** button and select the hole created in step 9.
14. Set the angular instance as **4** and the angular offset as **90** degrees in the corresponding spinners and choose **OK**.
15. Invoke the **Fillet Blend Feature** tool from the **Application Menu** toolbar. Select the **Propagate** check box. 
16. Move the cursor near the bottom edge of the second feature. Press the left-mouse button when the edge is highlighted.
17. Set **4 mm** in the **Radius** spinner and choose the **OK** button. The model, after creating the hole and the fillet, is shown in Figure 3-73.

Creating a Sectional View

To create the sectional view, you need to draw a rectangle on the **XZ Plane (front)** plane and extrude it using the **Cut** option.

1. Select the **XZ Plane (front)** plane from the **Model** window and choose the **Set the View to Const. Plane** button from the **Viewing** toolbar.
2. Using the **Rectangle** tool, draw a rectangle that encloses the entire model.
3. Choose the **Boss/Cut/Divide Extrude** button from the **Application Menu** toolbar. 
4. Select the **Cut** radio button and set the value as **90 mm** in the **Depth** spinner.
5. Choose the **Set Target** button and select a surface on the model. Reverse the direction by clearing the **Flip** button, if needed be.
6. In the **Name** edit box, assign the name as **Section view** and choose **OK**. The isometric view of the sectional view is shown in Figure 3-74.
7. Choose the **Section view** feature from the **Model** window and right-click.
8. Clear the **Enable Procedure** option from the shortcut menu displayed. The isometric view of the final model is displayed, as shown in Figure 3-75.

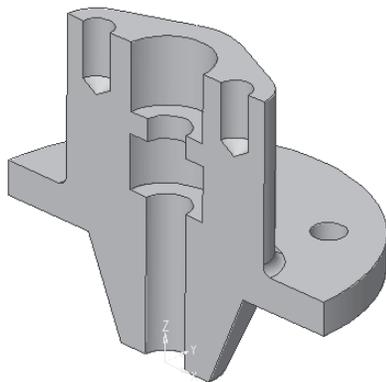


Figure 3-74 Sectional view of the model

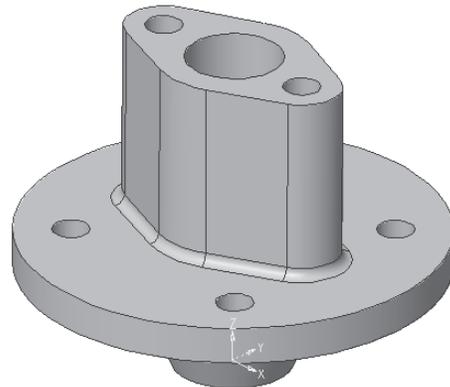


Figure 3-75 Isometric view of the model

Saving the Model

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below.

`\\My Documents\EdgeCAM\c03\c03tut1.epmod`

2. Choose **File > Close** from the menu bar to close the document.

Tutorial 2

In this tutorial, you will create the model whose dimensions are shown in Figure 3-76. The solid model is shown in Figure 3-77. **(Expected time : 30min)**

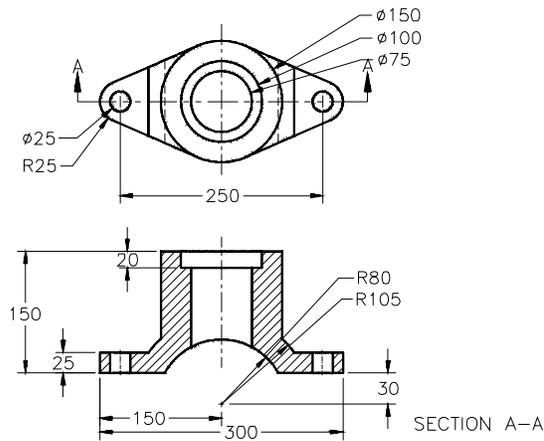


Figure 3-76 Dimensions for Tutorial 2

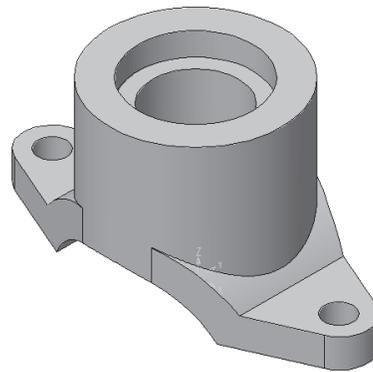


Figure 3-77 Model for Tutorial 2

The steps to complete this tutorial are listed next.

- Create the base feature by extruding the sketch drawn on the **XZ Plane (front)** plane, refer to Figures 3-78 and 3-79.
- Extrude the sketch created on the **XY Plane (top)** plane to create a cut feature, refer to Figure 3-81.
- Create a plane at an offset distance from the **XY Plane (top)** plane.
- Draw a sketch on the newly created plane and extrude it to the selected surface, refer Figure 3-83.
- Create a counter bore using the **Drill Hole Feature** tool, refer to Figure 3-84.
- Create the holes using the cut feature, refer to Figure 3-85.
- Save the document and close it.

Creating the Base Feature

It is evident from the model that its base comprises a complex geometry. You need to create the base feature and then apply the cut feature to the base of the model to get the desired shape. The sketch of the base feature needs to be created on **XZ Plane (front)**. After drawing the sketch, you need to extrude it using the **Both** option from the **Direction** drop-down list to complete the feature creation.

- Start a new EdgeCAM document and select the **XZ Plane (front)** plane as the construction plane.
- Using the standard sketching tools, draw the sketch of the base feature and then apply the required relations and dimensions to the sketch, as shown in Figure 3-78.

3. Invoke the **New Component Extrude** tool and select the **Both** option from the **Direction** drop-down list. 
4. Set the value as **75 mm** in the **Depth** spinners in the **Up** and the **Down** areas.
5. Choose the **Type** tab and select the **Solid** radio button from the **Body type** area.
6. Choose the **Preview** button and rotate the component using the **View Rotate** button or by holding down the right mouse button.
7. Choose the **Shaded** button and the **Views** button from the **Viewing** toolbar; the **Display View** dialog box is displayed. Select the **Isometric View** option and choose **OK**. The model is displayed, as shown in Figure 3-79. 

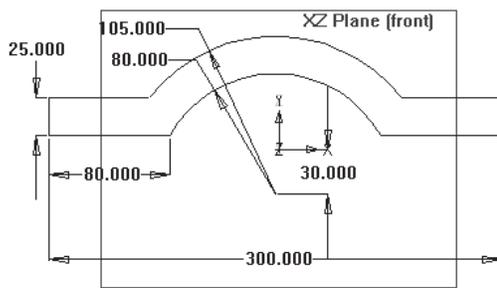


Figure 3-78 Sketch for the base feature

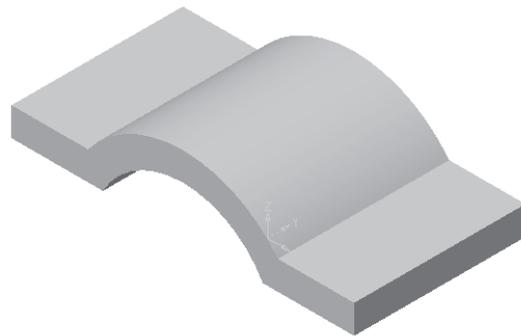


Figure 3-79 Base feature of the solid model

Creating the Cut Feature

Next, you need to create a cut feature to get the required shape of the base feature. The sketch for this cut feature is to be created using a construction plane defined tangent to the curved face of the previous feature.

1. Choose the **Construction Planes** button from the **Application Menu** toolbar and choose the **Tangent to Face at Position** button from the **Construction Planes** dialog box. 
2. Select the upper curved face of the existing feature and set the value as **0.5** in the **U-Value** and **V-Value** spinners. Choose **OK** to create the plane.
3. Using the recently drawn plane as the sketch plane, draw the sketch for the cut feature and apply the required relations and dimensions to the sketch, as shown in Figure 3-80.
4. Choose the **Boss/Cut/Divide Extrude** tool from the **Application Menu** toolbar to invoke the **Boss/Cut Extrude** dialog box. Change the current view to the isometric view. 

5. Select the **Cut** radio button. Select the **Through All** option from the **Direction** drop-down list. Also, select the **Flip** check box, if required.
6. Choose the **Type** tab and select the **Surface** radio button from the **Body type** area.
7. Choose the **Boss/Cut Options** tab and select the **In Back** radio button from the **Fence** area.
8. Choose the **OK** button. The isometric view of the model, after creating the base feature, is shown in Figure 3-81.

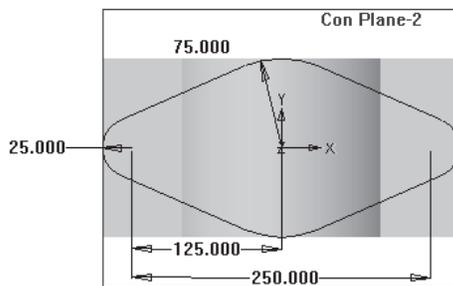


Figure 3-80 Sketch for the cut feature

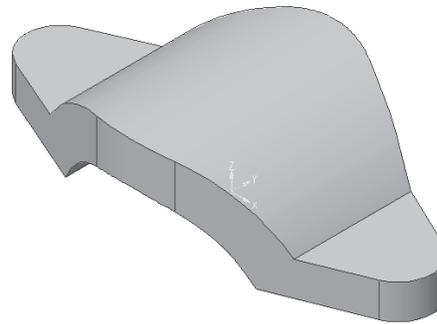


Figure 3-81 Cut feature added to the feature

Creating a Plane at an Offset Distance and Extruding the Feature

After creating the base feature of the model, you need to create a plane at an offset distance of 150 mm from the **XY Plane (top)**. This newly created plane is used as a sketching plane for the next feature.

1. Choose the **Construction Plane** button from the **Application Menu** toolbar to invoke the **Construction Planes** dialog box.
2. Choose the **Offset** button and select the **XY Plane (top)** plane from the **Model** window as the reference plane. 
3. Set the value of **1** and **190 mm** in the **Total Planes** and **Distance** spinner, respectively, and choose **OK**; the plane is created.
4. Draw the sketch of the circle and apply the dimensions, as shown in Figure 3-82.
5. Invoke the **Boss/Cut/Divide Extrude** tool; the **Boss/Cut Extrude** dialog box is displayed.
6. Select the **To Face** option from the **Direction** drop-down list. Choose the **Choose Face** button and select the top curved face of the base feature.
7. Choose the **Type** tab, select the **Solid** radio button from the **Body type** area, and choose **OK**.

8. Choose the **Set Target** button. Select the same face and choose **OK**; the feature will be created, as shown in Figure 3-83.

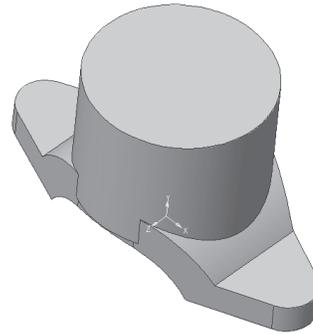
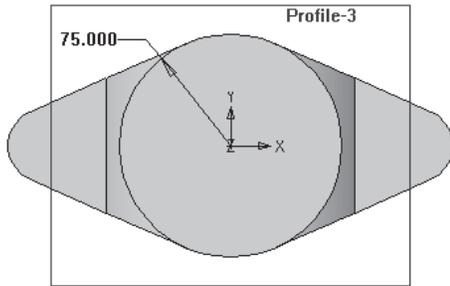


Figure 3-82 Sketch created for the third feature Figure 3-83 Model after adding the third feature

Creating the Counter bore Hole

Next, you need to create the counter bore hole. It will be created using the **Drill Hole Feature** tool.

1. Select the top surface of the extrude feature created in the last step and place a planar point at the center of the top surface using the **Point** tool.
2. Select the **Drill Hole Feature** tool to invoke the **Drill Hole** dialog box. Choose the **Counter Bore Hole** button and enter the name as **C.Bore** in the **Name** edit box. 
3. Select the **Custom** option from the **Standard** drop-down list. Set the value of **100 mm** in the **C.Bore/CSink Diameter** spinner. Also, set **20 mm** in the **Head depth** spinner.
4. Set the value as **75 mm** in the **Drill diameter** spinner. Select the **Through** radio button and choose **OK**. The counter bore hole is created as shown in Figure 3-84.

To create the drill hole on the base feature, you need to select the top surface of the base feature and place two points to locate the drill hole. Then, create the hole using the **Drill Hole Feature** tool.

5. Select the top surface of the base feature; a construction plane is created.
6. Choose the **Point** button and place two points for the holes to be created. Add dimensions to them.
7. Invoke the **Drill Hole Feature** tool; the **Drill Hole** dialog box is displayed. Choose the **Simple Hole** button and enter the name as **Holes 25x2** in the **Name** edit box. 

8. Select the **Custom** option from the **Nominal** size drop-down list and set the value of **25 mm** in the **Drill Diameter** spinner.
9. Select the **Through** radio button. Check the preview by selecting the **Preview** button and choose **OK**. The final model is shown in Figure 3-85.

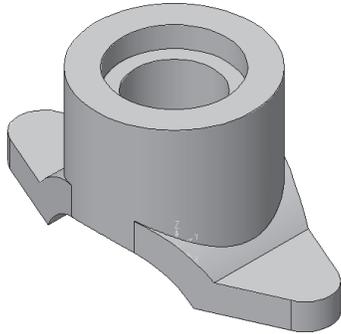


Figure 3-84 Counter bore added to the model

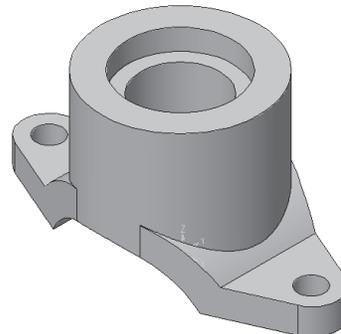


Figure 3-85 Final model

Saving the Model

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given next `|My Documents|EdgeCAM|c03|c03tut2.epmod`
2. Choose **File> Close** from the menu bar to close the document.

Self-Evaluation Test

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

1. A sketch drawn for the first time in the sketching environment is drawn on the **XZ Plane (front)** plane. (T/F)
2. If the sketch is open, it will be converted into

(a) Sheet	(b) Surface
(c) Both	(d) None
3. Which relation forces the selected arc to share the same center point with the center point of another arc?

(a) Coincident	(b) Concentric
(c) Coradial	(d) Merge Points

4. Only dimensional constraints can be applied while drawing a sketch. (T/F)
5. You cannot modify the **Driven** dimension. (T/F)
6. Using _____ tool, you can create a simple hole, tapered hole, counter bore, and counter sink hole.
7. The _____ check box is selected to select the chain of curves while creating fillets.
8. In EdgeCAM, you cannot create a multithickness shell feature. (T/F)
9. When you choose the _____ button, all the hidden lines will be displayed along with the visible lines in the model.
10. You can invoke the **View Rotate** tool while the other tool is in operation. (T/F)

Review Questions

Answer the following questions:

1. The _____ dialog box will be displayed when the added constraint over define the sketch.
2. Which is the correct method to add an equation to a dimension?
 - (a) $= [] * 2$
 - (b) $\{ \} * 2$
 - (c) $= () * 2$
 - (d) $= \{ \} * 2$
3. You cannot edit the sketch of a sketched feature. (T/F)
4. You cannot rename a feature in the **Model** window. (T/F)
5. Before creating a hole feature, you need to specify the location by points. (T/F)
6. Which check box is selected to remove the edges generated by revolving a sketch?
 - (a) **Remove redundant edges**
 - (b) **Edges**
 - (c) **Remove Edges**
 - (c) **None**
7. To create a sphere, revolve a circle along its center. (T/F)
8. To invoke the **Equation** dialog box, move the cursor in the spinner, right-click, and select the **Equation** option. (T/F)
9. By default, the _____ radio button is selected in the **Chamfer** dialog box.

10. To transform a plane, choose the _____ button in the **Construction Planes** dialog box.

EXERCISE

Exercise 1

In this exercise, you will create the model shown in Figure 3-86. The dimensions of this model are also shown in the same figure. **(Expected time: 30 min)**

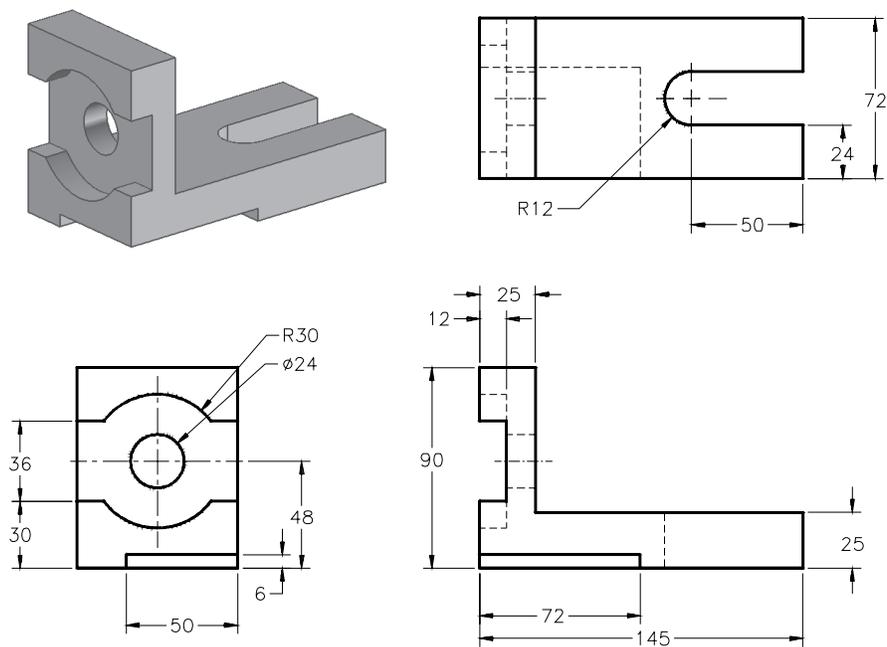


Figure 3-86 Model for Exercise 1 and its dimensions

Answers to Self-Evaluation Test

1. F, 2. Both, 3. Concentric, 4. F, 5. T, 6. Drill Hole, 7. Propagate, 8. F, 9. Wireframe, 10. T