Chapter 3

Adding Relations and Dimensions to the Sketches

Learning Objectives

After completing this chapter you will be able to:

- *Add geometric relations to the sketch.*
- *Dimension the sketches.*
- *Modify the dimensions of the sketch.*
- *Understand the concept of fully defined sketch.*
- *View and examine the relations applied to the sketches.*
- *Open an existing file.*

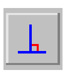
ADDING GEOMETRIC RELATIONS TO THE SKETCH

As discussed earlier, the geometric relations are the logical operations that are performed to add a relationship (such as tangent or perpendicular) between the sketched entities, planes, axes, edges, or vertices. The relations applied to the sketched entities are used to capture the design intent. The geometric relations constrain the degree of freedom of the sketched entities. There are two methods to apply the relations to the sketch. These two methods are

1. Add Relations PropertyManager
2. Automatic Relations

Adding Relations Using the Add Relations PropertyManager

Toolbar:	Sketch Relations > Add Relation
Menu:	Tools > Relations > Add



The **Add Relations PropertyManager** is widely used to apply relation to the sketch in the sketcher environment of SolidWorks. The **Add Relations PropertyManager** is invoked using the **Add Relation** button from the **Sketch Relations** toolbar. The **Add Relations PropertyManager** is displayed on the left of the drawing area as soon as you choose the **Add Relation** button. You can also invoke the **Add Relations PropertyManager** by right-clicking in the drawing area and choosing the **Add Relation** option from the shortcut menu. The **Add Relations PropertyManager** is invoked as shown in Figure 3-1. The confirmation corner is also displayed at the top right corner of the drawing area. The various options available in the **Add Relations PropertyManager** are discussed next.

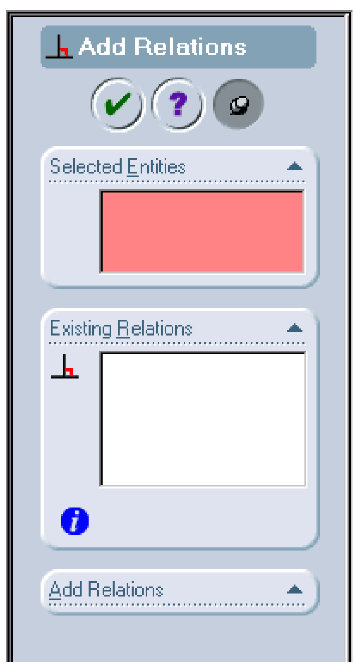
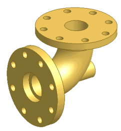


*Figure 3-1 The **Add Relations PropertyManager***

Selected Entities

The **Selected Entities** rollout displays the name of the entities that are selected to apply the relations. The entities are added in the area under the **Selected Entities** rollout by selecting them from the graphics area. The selected entities are displayed in green color. You can remove the selected entity from the selection set by selecting the same entity from the drawing area using the left mouse button. You will notice that the color of the entity is changed from green to blue.



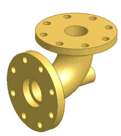
Tip. *You can also remove the selected entity from the selection set by choosing that entity in the* ***Selected Entities*** *rollout and use the right mouse button to invoke the shortcut menu. Choose the* ***Delete*** *option from this menu to remove the entity from the selection set. If you choose the* ***Clear Selections*** *option from the shortcut menu then all the entities will be removed from the selection set.*

Existing Relations

The **Existing Relations** rollout displays the relations that are already applied to the selected sketch entities. It also shows the status of the sketch entities. You can delete the already existing relation from the **Existing Relations** rollout. Select the already existing relation from the selection area and right-click to display the shortcut menu. Choose the **Delete** option from this shortcut menu to delete the selected relation. If you choose the **Delete All** option from the shortcut menu, all the relations displayed in the selection area of the **Existing Relations** rollout will be deleted. The status of the sketch entity is displayed below the selection area. You will learn more about the status of the selected sketch entity and the status of the entire sketch later in this chapter.

Add Relations

The **Add Relations** rollout is used to apply the relations to the selected entity. The list of relations that can be applied to the selected entity or entities is shown in the **Add Relations** rollout. Note that only the relations that can be applied to the selected entity or entities are displayed in the list. The most appropriate relation for the selected entities appears in bold letters.



Tip. *You can apply a relation to a single entity or a relation between two or more than two entities. For applying a relation between two or more than two entities at least one entity should be a sketched entity. The other entity or entities can be sketch entities, edges, faces, vertices, origins, plane, or axes. The sketch curves from other sketches that form lines or arcs when projected on the sketch plane can also be included in the relation.*

You will learn more about projected sketch curves and planes in the later chapters.

The relations that can be applied to the sketches using the **Add Relations** rollout are discussed next.

Horizontal

The **Horizontal** relation forces the selected line or lines to become horizontal. A line can be a line, or center line in the sketch, or an external entity such as an edge, plane, axis, or sketch curve on an external sketch that projects as a line in the sketch. Using the **Horizontal** relation you can also force two or more points to become horizontal. A point can be a sketch, a centerpoint, an endpoint, a control point of a spline, or an external entity such as origin, vertex, axis, or point in an external sketch that projects as a point. To use this relation, choose the **Add Relation** button from the **Sketch Relations** toolbar to display the **Add Relations PropertyManager**. Select the entity or entities to apply the **Horizontal** relation. Choose the **Horizontal** button from the **Add Relations** rollout provided in the **Add Relations PropertyManager**. You will notice that the name of the horizontal relation is displayed in the **Existing Relations** rollout.

Vertical

The **Vertical** relation forces the selected line or lines to become vertical. Using the **Vertical** relation you can also force two or more points to become vertical. To use this relation, choose the **Add Relation** button from the **Sketch Relations** toolbar to display the **Add Relations PropertyManager**. Select the entity or entities to apply the **Vertical** relation. Choose the **Vertical** button from the **Add Relations** rollout provided in the **Add Relations PropertyManager**. You will notice that the name of the vertical relation is displayed in the **Existing Relations** rollout.

Collinear

The **Collinear** relation forces the selected lines to lie on the same infinite line. To use this relation invoke the **Add Relations PropertyManager**. Select the line to apply the **Collinear** relation. Choose the **Collinear** button from the **Add Relations** rollout.

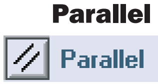
Coradial

The **Coradial** relation forces the selected arcs to share the same radius and the same centerpoint. An arc can be an arc or a circle in a sketch, or an external entity that projects as an arc or a circle in the sketch. To use this relation, invoke the **Add Relations PropertyManager**. Select two arcs or circles, or an arc and a circle to apply the **Coradial** relation. Choose the **Coradial** button from the **Add Relations** rollout.

Perpendicular

The **Perpendicular** relation forces the selected lines to become perpendicular to each other. To use this relation, invoke the **Add Relations PropertyManager**. Select two lines to apply the **Perpendicular** relation. Choose the **Perpendicular** button from the **Add Relations** rollout. Figure 3-2 shows two lines before and after applying the perpendicular relation.

Parallel



The **Parallel** relation forces the selected lines to become parallel to each other. To use this relation, invoke the **Add Relations PropertyManager**. Select two lines to apply the **Parallel** relation. Choose the **Parallel** button from the **Add Relations** rollout to apply the parallel relation. Figure 3-3 shows two lines before and after applying this relation.

ParallelYZ

The **ParallelYZ** relation forces a line in the 3D sketch to become parallel to the YZ plane with respect to the selected plane. To use this relation, invoke the **Add Relations PropertyManager**. Select the line in the 3D sketch and

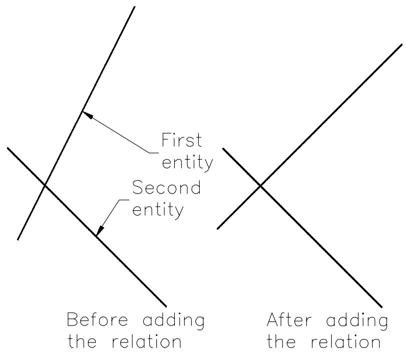


Figure 3-2 *Entities before and after applying the Perpendicular relation*

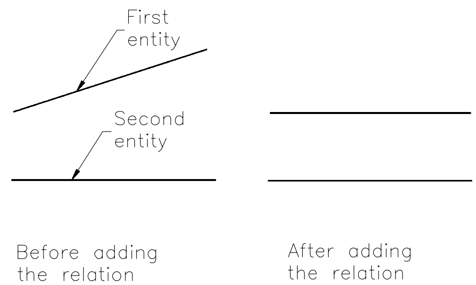


Figure 3-3 *Entities before and after applying the Parallel relation*

a plane and choose the **ParallelYZ** button from the **Add Relations** rollout.

ParallelZX

The **ParallelZX** relation forces a line in the 3D sketch to become parallel to the ZX plane with respect to the selected plane. To use this relation, invoke the **Add Relations PropertyManager**. Select the line in the 3D sketch and a plane and choose the **ParallelZX** button from the **Add Relations** rollout.

AlongZ

The **AlongZ** relation forces a line in the 3D sketch to become normal to the selected plane. To use this relation, invoke the **Add Relations PropertyManager**. Select the line in the 3D sketch and a plane and choose the **AlongZ** button from the **Add Relations** rollout.

Tangent

The **Tangent** relation forces the selected arc, circle, spline, or ellipse to become tangent to other arc, circle, spline, ellipse, line, or edge. To use this relation, invoke the **Add Relations PropertyManager**. Select two entities to apply the **Tangent** relation. Choose the **Tangent** button from the **Add Relations** rollout. Figures 3-4 and 3-5 show the entities before and after applying this relation.

Concentric

The **Concentric** relation forces the selected arc or circle to share the same centerpoint with other arc, circle, point, vertex, or a circular edge. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entity to apply the **Concentric** relation. Choose the **Concentric** button from the **Add Relations** rollout.

Midpoint

The **Midpoint** relation forces the selected point to move at the midpoint of the selected line. To use this relation, invoke the **Add Relations PropertyManager**.

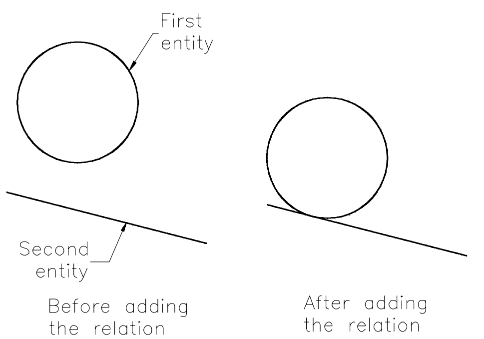


Figure 3-4 Entities before and after applying the Tangent relation

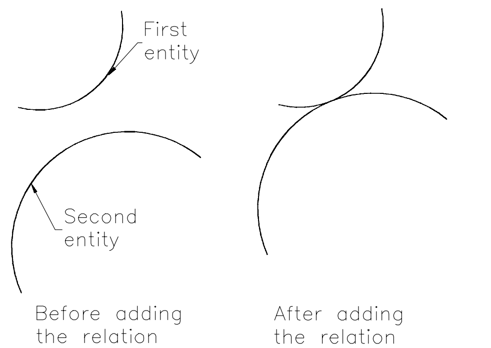


Figure 3-5 Entities before and after applying the Tangent relation

Select the required entity to apply the **Midpoint** relation. Choose the **Midpoint** button from the **Add Relations** rollout.

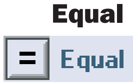
Intersection

The **Intersection** relation forces the selected point to move at the intersection of the two selected lines. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entity to apply the **Intersection** relation. Choose the **Intersection** button from the **Add Relations** rollout.

Coincident

The **Coincident** relation forces the selected point to be coincident with the selected line, arc, circle, or ellipse. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entity to apply the **Coincident** relation. Choose the **Coincident** button from the **Add Relations** rollout.

Equal



The **Equal** relation forces the selected lines to have equal length and selected arcs, circles, or an arc and a circle to have equal radii. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entity to apply the **Equal** relation. Choose the **Equal** button from the **Add Relations** rollout.

Symmetric

The **Symmetric** relation forces two selected lines, arcs, points, and ellipses to remain equidistant from a center line. This relation also force the arcs to have the same radii. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entity to apply the **Symmetric** relation and select a centerline. Choose the **Symmetric** button from the **Add Relations** rollout.

Fix

The **Fix** relation forces the selected entity to fix at the specified position. If you apply this relation to a line or an arc, its location is fixed but you can change its size by dragging the endpoints. To use this relation, invoke the **Add Relations**

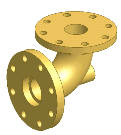
PropertyManager. Select the required entity to apply the **Fix** relation. Choose the **Fix** button from the **Add Relations** rollout.

Pierce

The **Pierce** relation forces a sketch point to be coincident where an axis, line, arc, edge, or spline pierce the sketch plane. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entities to apply the **Pierce** relation. Choose the **Pierce** button from the **Add Relations** rollout.

Merge Points

The **Merge Points** relation forces two sketch points or endpoints to merge in a single point. To use this relation, invoke the **Add Relations PropertyManager**. Select the required entities to apply the **Merge Points** relation. Choose the **Merge Points** button from the **Add Relations** rollout.



Tip. *You can also apply the relations using the* ***Properties PropertyManager****. Using the left mouse button select the entities to add relation. When you select the entities, the* ***Properties PropertyManager*** *will be displayed. If you select a single entity then the* ***PropertyManager*** *of that particular entity will be displayed. The possible relations for the selected geometry will be displayed in the* ***Add Relations*** *rollout. Choose the relation that you want to apply to the selected geometry. The existing relation will be displayed in the* ***Existing Relations*** *rollout.*

Another method of applying relations to the sketches is by selecting the entity or entities on which you have to apply the relation and then right-clicking to display the shortcut menu. The relation that can be applied to the selected entities will be displayed in the shortcut menu. Choose the relation from the shortcut menu.

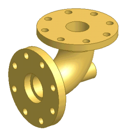
Automatic Relations

The automatic relations are applied automatically to the sketch while drawing. You can activate the automatic relations option if it is not available. Invoke the **System Options - Sketch** dialog box by choosing **Tools** > **Options** > **System Options** > **Sketch**. The **System Options - Sketch** dialog box is displayed. Select the **Automatic Relations** check box from the **System Options - Sketch** dialog box and choose the **OK** button.

The automatic relations are applied to the entities while sketching. For example, you will notice that when you specify the startpoint of the line and move the cursor horizontally toward the right or left, a symbol **H** is displayed below the line cursor. This is the symbol of the Horizontal relation that is applied to the line while drawing. If you move the cursor vertically downwards or upwards, the **V** symbol for Vertical relation is displayed below the line cursor. If you move the cursor to the intersection of two or more than two sketched entities, the intersection symbol appears below the cursor. Similarly, other relations are also automatically applied to the sketch while creating the sketch. The relations that are applied automatically are listed below:

1. Horizontal
2. Vertical
3. Coincident

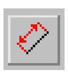
4. Midpoint
5. Intersection
6. Tangent
7. Perpendicular



Tip. *You will observe that while sketching, two types of inferencing lines are displayed. One is displayed in blue and the second in brown. The brown inferencing line indicates that relation is applied automatically to the sketch. The blue inferencing line indicates that no automatic relation is applied.*

DIMENSIONING THE SKETCH

Toolbar:	Sketch > Dimension
Menu:	Tools > Dimension



After drawing the sketches and adding the relations, dimensioning is the most important step in creating a design. As mentioned earlier, SolidWorks is a parametric software. The parametric nature of SolidWorks ensures that irrespective of the original size, the selected entity is driven by the dimension value that you specify. Therefore, when you apply and modify a dimension on an entity, it is forced to change its size in accordance with the specified dimension value. The type of dimension that will be applied depends on the type of entity selected. For example, if you select a line, linear dimension will be applied and if you select a circle, diametric dimension is applied. Similarly, if you select an arc, a radial dimension is applied. While dimensioning, you can set the priority to edit the dimension value as soon as you place it. To set this priority, choose **Tools** > **Options** to display the **System Options - General** dialog box. Select the **Input dimension value** check box and choose **OK**.

Now, when you select an entity to apply the dimension, the **Modify** dialog box will be displayed, as shown in Figure 3-6, as soon as you place the dimension. You can enter a new value in this box to modify the dimension.

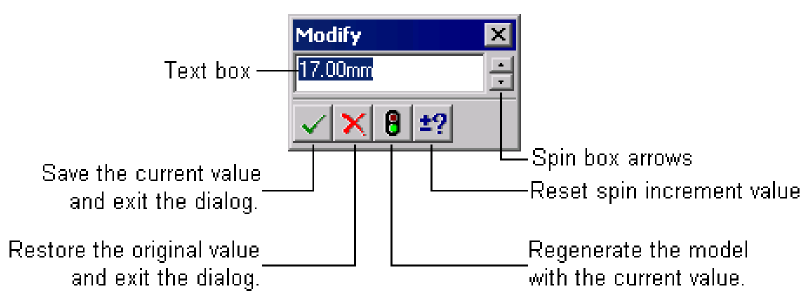


***Figure 3-6 Modify** box*

You can modify the default dimension value using the spinner arrows or by entering a new value in the text box available in the **Modify** dialog box. The buttons available in the **Modify** dialog box are discussed next.

The **Save the current value and exit the dialog** button is used to accept the current value and exit the dialog box. The **Restore the original value and exit the dialog** button is used to restore the last dimensional value applied to the sketch and exit the dialog box. The **Regenerate the model with the current value** button is used to preview the geometry of the sketch with the new modified dimensional value. The **Reset spin increment value** button is used to enter a new spin increment value. This is the value that is added or subtracted from the current value when you click once on the spinner arrow. When you choose this button the **Increment** dialog box is displayed as shown in Figure 3-7.

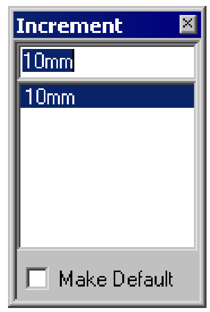
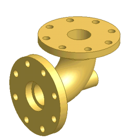


Figure 3-7 Increment *box*

Insert a new value in the text box provided in the **Increment** dialog box and press ENTER. If you select the **Make Default** check box, the new increment value will become the default spin increment value. By default, 10 mm is the default increment value while working with metric units. Next time when you modify the dimension using the spin increment arrows, the dimension will increase or decrease with the increment value that you saved as the default value.



Tip. *You can also enter the arithmetic symbols directly into the text box to calculate the dimension. For example, if you have a dimension as a complex arithmetic function such as (220*12.5)-3+150, which is equal to 1247, you do not need to calculate this using the calculator. Just enter the statement in the text box provided in the* ***Modify Dimension*** *dialog box and press ENTER. SolidWorks will automatically solve the function to get the value of the dimension.*

Linear Dimensioning

Linear dimensions are defined as the dimensions that define the shortest distance between two points. You can apply the linear dimension directly to a line, two points, or two objects. The points can be the endpoints of lines or arcs, or the centerpoints of circles, arcs, ellipses, or parabolas. You can dimension a vertical or a horizontal line by directly selecting them. Choose the **Dimension** button from the **Sketch** toolbar or right-click in the drawing area. Choose the **Dimension** option from the shortcut menu to activate the **Dimension** tool. When you move the cursor on the line, it is highlighted and turns red in color. As soon as you select the line, it turns to green, and the dimension is attached to the cursor. Move the cursor and place the dimension at an appropriate place using the left mouse button. Since you have already set the priority of editing the dimension as it is placed, the **Modify** dialog box is displayed with the default value in it. Enter the new value of dimension in the **Modify** dialog box and press ENTER. Figure 3-8 shows linear dimensioning of horizontal and vertical lines.

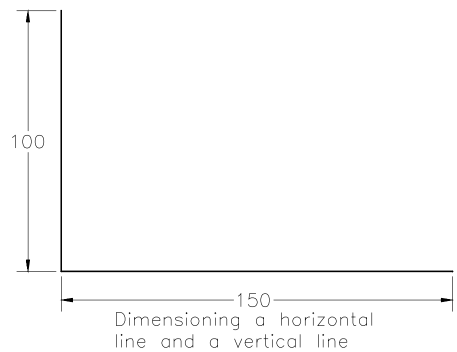


Figure 3-8 *Linear dimensioning of lines*

If the dimension is selected in the drawing area, the **Dimension PropertyManager** is displayed at the left of the drawing area as shown in Figure 3-9. The various options available in the **Dimension PropertyManager** are discussed next.

Part Favorite

The **Part Favorite** rollout is used to create, save, delete, and retrieve the dimension style in the current document. You can also retrieve the dimensions styles saved earlier using this rollout. The **Part Favorite** rollout is shown in Figure 3-10. The options available in this rollout are discussed next:

Apply the default attributes to selected dimensions

The **Apply the default attributes to selected dimensions** button is used to apply the default attributes to the selected dimension or dimensions. The attributes include the tolerance, precision, arrow style, dimension text, and so on. This option is generally used when you have modified the settings applied to a dimension and then you want to restore the default settings on that dimension.

Add or Update a Favorite

The **Add or Update a Favorite** button is used to add a dimension style to the current document for a selected dimension. After invoking the **Dimension PropertyManager**, set the attributes using various options provided in this **PropertyManager**. Now choose the **Add or Update a Favorite** button. The **Add or Update a Favorite** dialog box is displayed in Figure 3-11. Enter the name of the dimension style in the edit box and press ENTER or choose the **OK** button from this dialog box. The dimension style will be added to the current document.

You can apply the new dimension style to the selected dimension by selecting the dimension style from the drop-down list in the **Part Favorite** rollout. You can also update the dimension style. To update a dimension style, select the dimension and set the options of the dimension style according to your need and then choose the **Add or Update a Favorite** button to invoke the **Add or Update a Favorite** dialog box. Select the dimension style to update from the drop-down list provided in the dialog box. The two radio button in this dialog box are enabled. Select the **Update all the annotations linked to this favorite** radio button and

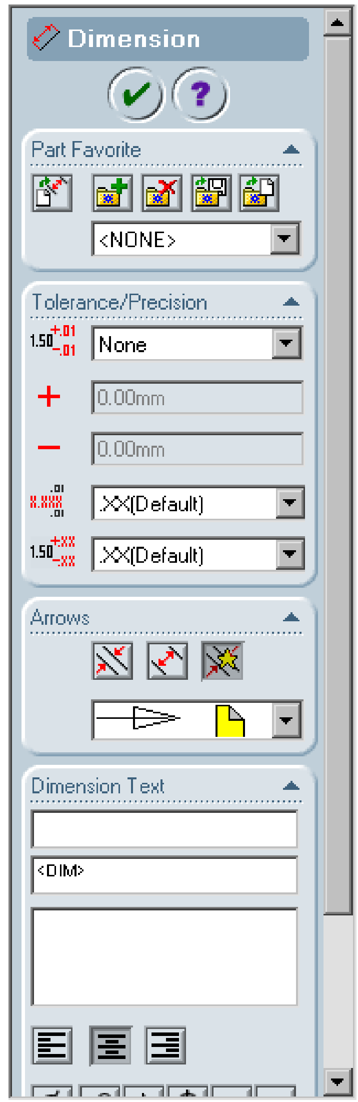


Figure 3-9 *The* ***Dimension PropertyManager***

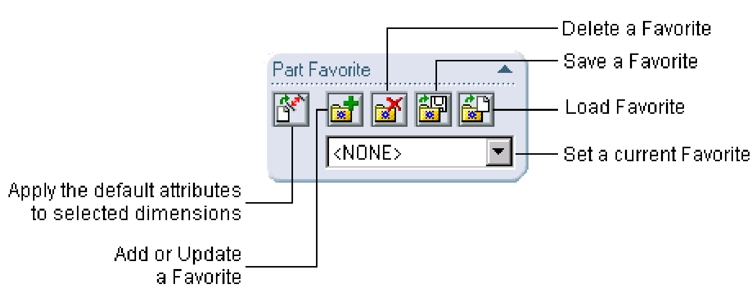


Figure 3-10 *The* ***Part Favorite*** *rollout*

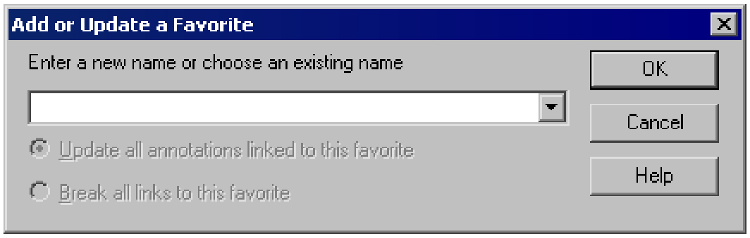


Figure 3-11 *The* ***Add or Update a Favorite*** *dialog box*

choose the **OK** button to update all the dimensions linked with the selected favorite. If you select the **Break all links to this favorite** radio button and choose the **OK** button to update the dimension style used in the selected dimension, then the link between the other dimensions having the same favorite and the selected favorite will be broken.

Delete a Favorite

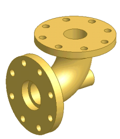
The **Delete a Favorite** button is used to delete a dimension style. Select the dimension style or favorite from the **Set a current Favorite** drop-down list and choose the **Delete a Favorite** button. Note that even after you delete the dimension style, the properties of the dimensions will be the same as with the deleted favorite. You can set the properties of a dimension to the document default using the **Apply the default attributes to the selected dimension** button.

Save a Favorite

The **Save a Favorite** button is used to save a dimension style so that it can be retrieved in some other document. Select the dimension style or favorite from the **Set a current Favorite** drop-down list and choose the **Save a Favorite** button. The **Save As** dialog box will be displayed. Browse the folder in which you want to save the favorite and enter the name of the favorite in the **File name** edit box. Choose the **Save** button from the **Save As** dialog box. The extension for the file in which the favorite is saved is *.sldfvt*.

Load Favorite

The **Load Favorite** button is used to open a saved favorite in the current document and the properties of that favorite will be applied to the selected dimension. To load a favorite, choose the **Load favorite** button to invoke the **Open** dialog box. Browse the folder in which the favorite is saved. Now, select the file with the extension *.sldfvt* and choose the **Open** button. The **Add or Update a Favorite** dialog box is displayed. Choose the **OK** button from this dialog box.



Tip. *You can load more than one favorite by pressing the CTRL key and selecting the favorites from the* ***Open*** *dialog box. All the favorites will be displayed in the* ***Set a current Favorite*** *drop-down list.*

Tolerance/Precision

The **Tolerance/Precision** rollout is used to specify the tolerance and precision in the dimensions. This rollout is shown in Figure 3-12. The various options available in the **Tolerance/Precision** rollout are discussed next.

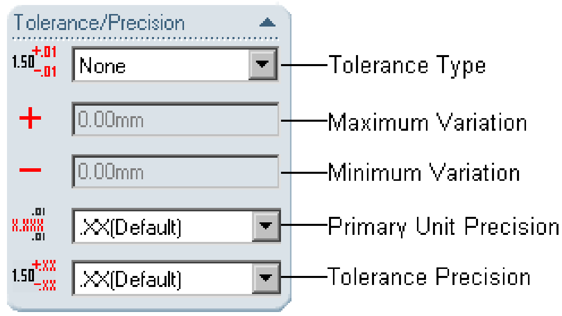


*Figure 3-12 The **Tolerance/Precision** rollout*

Tolerance Type

The **Tolerance Type** drop-down list is used to apply the tolerance to the dimension. Various tolerance methods are available in this list. By default, the **None** option is selected, which means that tolerance is not applied to the dimension. The other tolerance methods available in the **Tolerance Type** drop-down list are discussed next.

Basic

The **Basic** option is used to display the basic dimension. To display a basic dimension, select a dimension that you want to display as a basic dimension and then select the **Basic** option from the **Tolerance Display** drop-down list. You will notice that the dimension is enclosed in a rectangle, suggesting that it is a basic dimension. The basic dimension is shown in Figure 3-13.

Bilateral

The **Bilateral** option is used to display the bilateral tolerance along with the dimension. This type of tolerance provides the maximum and the minimum variation in the value of the dimension that is acceptable in the design. For applying the bilateral tolerance, select the dimension and then select the **Bilateral** option from the **Tolerance Type** drop-down list. The **Maximum Variation** and the **Minimum Variation** edit boxes are enabled. These edit boxes are used to apply the value of the maximum and minimum variation to a dimension. If you select the **Show parentheses** check box, the bilateral tolerance will be displayed with parentheses. This check box is available when you apply the bilateral tolerance. The dimension with bilateral tolerance is shown in Figure 3-14.

Note

The dimension standard used in the drawings in this book is the ISO standard.

Limit

The **Limit** option is used to display the limits with the dimension. In the limit dimension, the dimension is displayed as its maximum and minimum values that are allowed in the design. Select the dimension to display as limit dimension and select the **Limit** option. The **Maximum Variation** and the **Minimum Variation** edit boxes are enabled to enter the value of maximum and minimum variation. The dimension along with limit tolerance is shown in Figure 3-15.

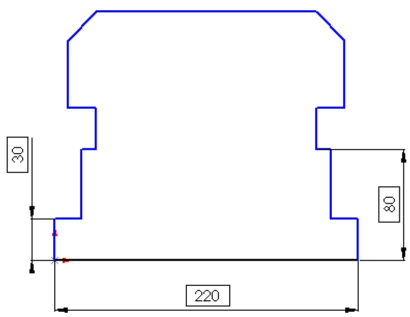


Figure 3-13 Basic dimension

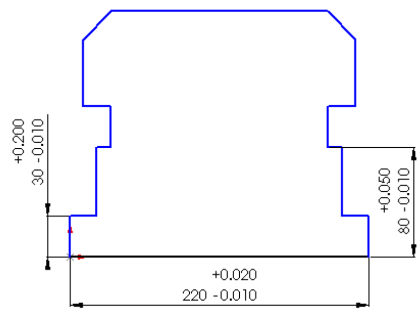


Figure 3-14 Bilateral tolerance

Symmetric

The **Symmetric** option is used to display the symmetric dimensional tolerance. This type of tolerance is displayed with plus and minus sign. To use this tolerance, select the dimension and select the **Symmetric** option. The **Maximum Variation** edit box will be displayed to enter the value of tolerance. You can select the **Show parentheses** check box to show the tolerance in parentheses. The dimension along with the symmetric tolerance is shown in Figure 3-16.

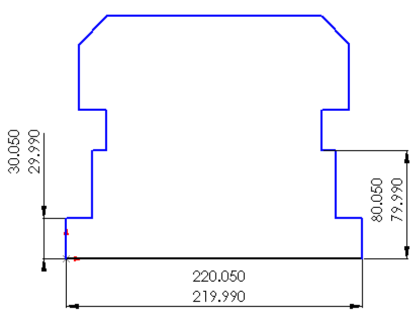


Figure 3-15 Limit tolerance

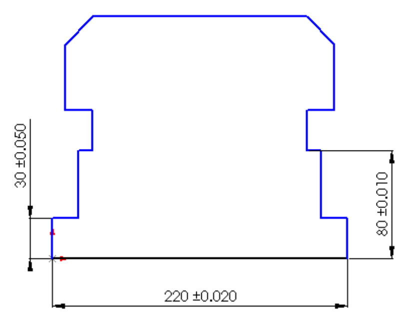


Figure 3-16 Symmetric tolerance

MIN

The **MIN** option in the **Tolerance Type** drop-down list is used to display the minimum allowable value of the dimension. In this type of dimensional tolerance, the **min.** symbol is added to the dimension as a suffix. This implies that the dimensional value is the minimum value that is allowable in the design. To display this dimensional tolerance, select the dimension and select the **MIN** option from the **Tolerance Type** drop-down list. The dimension along with the minimum tolerance is shown in Figure 3-17.

MAX

The **MAX** option in the **Tolerance Type** drop-down list is used to display the maximum allowable value of the dimension. In this type of dimensional tolerance, the **max.** symbol is added to the dimension as suffix. This implies that the dimensional value is the maximum value that is allowable in the design. To display this dimensional tolerance, select the dimension and select the **MAX** option from the **Tolerance Type** drop-down list. The dimension along with the maximum tolerance is shown in Figure 3-18.

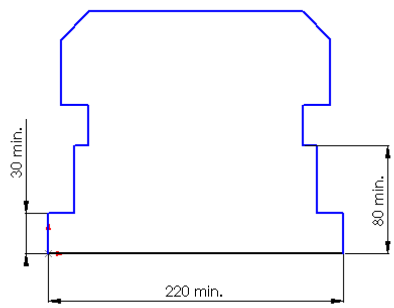


Figure 3-17 Minimum tolerance

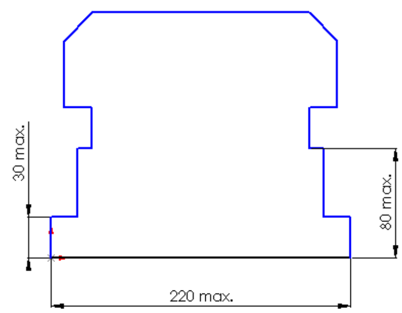


Figure 3-18 Maximum tolerance

Fit

The **Fit** option is used to apply the fit according to the **Hole Fit** and the **Shaft Fit** system. The **Tolerance/Precision** rollout with the **Fit** option selected is shown in Figure 3-19. Specify the type of fit from the **Classification** drop-down list. The **Classification** drop-down list is used to define **User Defined** fit, **Clearance** fit, **Transitional** fit, and **Press** fit. To apply the fit using the hole fit system or the shaft fit system, select the dimension and select the **Fit** option from the **Tolerance Type** drop-down

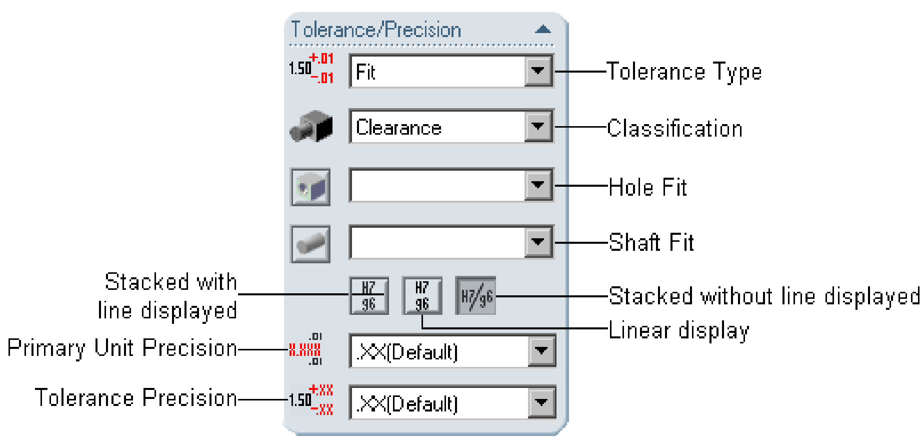


***Figure 3-19** The **Tolerance/Precision** rollout with the **Fit** option selected from the **Tolerance Type** drop-down list.*

list. The **Classification** drop-down list, the **Hole Fit** drop-down list, and the **Shaft Fit** edit drop-down list are displayed below the **Tolerance Type** drop-down list. Choose the type of fit from the **Classification** drop-down list and select the standard of fit from the **Hole Fit** drop-down list or the **Shaft Fit** drop-down list. If you choose the **Clearance**, **Transitional**, or **Press** option from the **Classification** drop-down list and you choose the standard of fit from the **Hole Fit** drop-down then only the standards that match with the selected hole fit will be displayed in the **Shaft Fit** drop-down list and vice-versa. If you choose the **User Defined** option from the **Classification** drop-down list then you can choose any standard from the **Hole Fit** and the **Shaft Fit** drop-down lists.

The **Stacked with line display** button provided under the **Shaft Fit** drop-down list is selected by default. You will notice that the tolerance is displayed as stacked with a line. You can also display the tolerance as stacked without a line using the **Stacked without line display** button. If you choose the **Linear display** button, the tolerance will be displayed in linear form. The dimension along with the hole fit and the shaft fit is shown in Figure 3-20.

Fit with tolerance

The **Fit with tolerance** option in the **Tolerance Type** drop-down list is used to display the tolerance along with the hole fit and shaft fit in a dimension. To apply the fit with the tolerance, select the dimension and select the **Fit with tolerance** option from the **Tolerance Type** drop-down list. Choose the type of fit from the **Classification** drop-down list. Now, select the fit standard from the **Hole Fit** drop-down list or the **Shaft Fit** drop-down list. The tolerance will be displayed with the fit standard only if you select only one fit system either from the hole drop-down list or from the shaft drop-down list. The tolerance will be displayed along with the fit standard in the drawing area. In this release of SolidWorks the tolerance is calculated automatically depending upon the type of fit selected and the standard of fit selected. The show parentheses check box is available, and you can select this check box if you want to show the parentheses. The dimension along with fit and tolerance is shown in Figure 3-21.

Note

The diametrical dimension will be discussed later in this chapter.

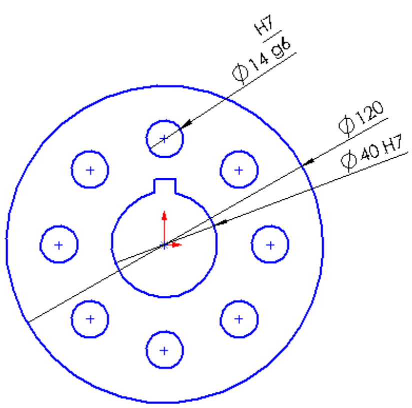


Figure 3-20 *Hole fit and shaft fit*

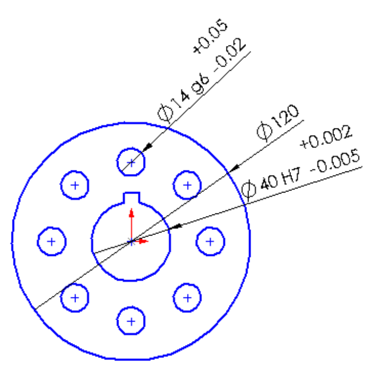


Figure 3-21 *Fit with tolerance*

Primary Unit Precision

The **Primary Unit Precision** drop-down list is used to specify the precision of the number of places after the decimal for dimensions. By default, the selected precision is two places after the decimal.

Tolerance Precision

The **Tolerance Precision** drop-down list is used to specify the precision of the number of places after the decimal for tolerance. By default, the selected precision is two places after the decimal.

Arrows

The **Arrows** rollout is used to specify the arrow style in the dimensions. The **Arrows** rollout is shown in Figure 3-22. Various options available in this rollout are discussed next.

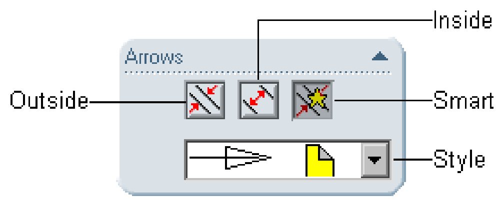


Figure 3-22 *The **Arrows** rollout*

Outside

The **Outside** button available in the **Arrows** rollout is used to display the arrows outside the dimension line. Select a dimension from the drawing area and choose the **Outside** button from the **Arrows** rollout.

Inside

The **Inside** button available in the **Arrows** rollout is used to display the arrows inside the dimension line. Select a dimension from the drawing area and choose the **Inside** button from the **Arrows** rollout.

Smart

The **Smart** button available in the **Arrows** rollout is used to display the dimension inside or outside the dimension line, depending on the surrounding geometry. The **Smart** button is selected by default in the **Arrows** rollout.

Style

The **Style** drop-down list is used to select the arrowhead style. The unfilled triangular arrow is select by default. You can select any arrow style for a particular dimension or dimension style. To change the arrow style, select the dimension from the drawing area and choose the arrow style from the **Style** drop-down list.

Dimension Text

The **Dimension Text** rollout is used to add the text and symbols in the dimension. The **Dimension Text** rollout is displayed in Figure 3-23. Three text boxes are provided in this rollout. These three text boxes are the three lines in which the text or symbols can be added. The first text box in this rollout is used to enter the text or symbols in the first line, which is above the dimension value. The second text box is used to enter the text or symbols in the second line, in which the dimension is also displayed in the second line. The third text box is used to enter the text or symbols in the third line, which is below the dimension value.

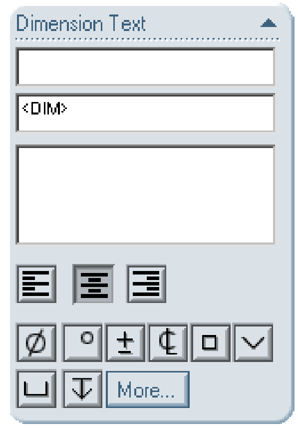


Figure 3-23 Dimension Text rollout

In this rollout various buttons are provided to add the symbols such as **Diameter**, **Degree**, **Plus/Minus**, **Centerline**, and so on to the dimension text. You can invoke the **Symbols** dialog box to add more symbols by choosing the **More Symbols** button from the **Dimension Text** rollout. The **Symbols** dialog box is displayed in Figure 3-24.

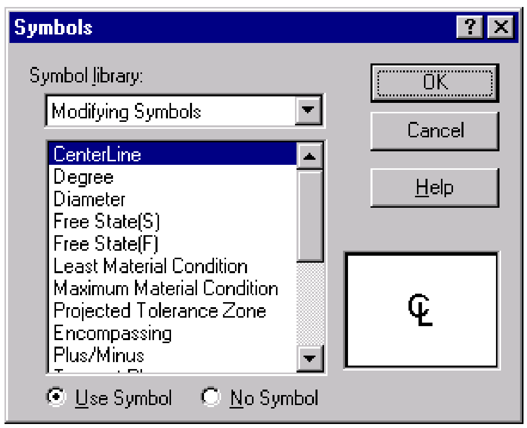


*Figure 3-24 The **Symbols** dialog box*

The **More Properties** button is used to invoke the **Dimension Properties** dialog box to modify the properties of the dimension. All the options available in the **Dimension PropertyManager** are available in the **Dimension Properties** dialog box with some additional options. You can modify the dimension properties from the **Dimension PropertyManager** or from the **Dimension Properties** dialog box.

Linear Dimensioning Between Points

You can add a linear dimension between two points; one point can be a sketch point, an endpoint, or a centerpoint and the second point can be a sketch point, an endpoint, a centerpoint, an origin, or a vertex. For creating the linear dimension between two points, choose the **Dimension** button from the **Sketch** toolbar. Select the first point, and then select the second point. Move the cursor to the right or left of the sketched entities to get the vertical dimension or move the cursor to the top or bottom of the sketched entities to get the horizontal dimension. Specify a point to place the dimension. The **Modify** dialog box will be displayed. Enter a new dimension value in this dialog box and press ENTER. Figures 3-25 and 3-26 show the horizontal and vertical linear dimensioning between points.

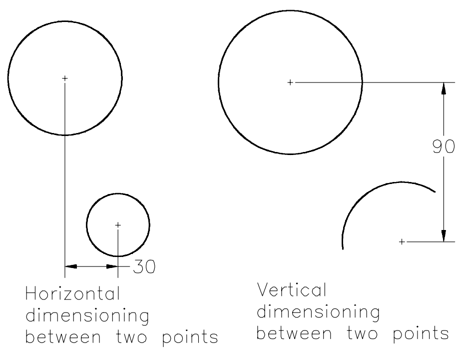


***Figure 3-25** Linear dimensioning between points*

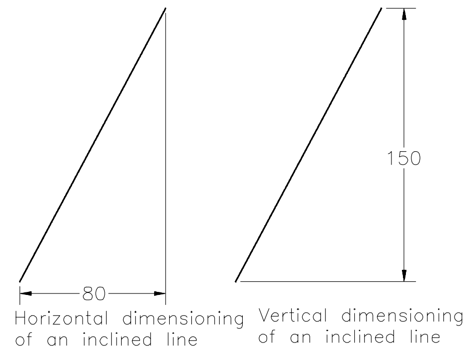


***Figure 3-26** Linear dimensioning of lines*

Linear Dimensioning of a Circle

You can also dimension a circle using the linear dimensioning method. Sketch the circle and choose the **Dimension** button from the **Sketch** toolbar. Using the left mouse button, select the circle. The dimension will be attached to the cursor. If you want to create the vertical dimension, move the cursor to the right or left of the sketch. If you want to create the horizontal dimension, move the cursor to the top or bottom of the sketch. Using the left mouse button, place the dimension and enter a new value in the **Modify** dialog box. The linear dimensioning of the circle is shown in Figure 3-27.

Aligned Dimensioning

Aligned dimensions are used to dimension lines that are at an angle with respect to the X axis and the Y axis. This type of dimensioning measures the actual distance of the inclined lines. You can directly select the inclined line to apply this dimension or select two points. The points that can be used to apply aligned dimension include the endpoints of line, arc, parabolic arc, or spline and the centerpoints of arcs, circles, ellipse, or parabolic arc. To apply an aligned dimension to an inclined line, choose the **Dimension** button from the **Sketch** toolbar and select the line. Move the cursor at an angle such that the dimension line is parallel to the inclined line. Using the left mouse button place the dimension at an appropriate place and enter a new value in the **Modify** dialog box.

To apply an aligned dimension between two points, choose the **Dimension** button and select the first point to which you have to apply the dimension. Now select the second point to apply the dimension. The dimension will be attached to the cursor. Move the cursor such that the dimension line is parallel to the imaginary line that joins two points. Now, using the left mouse button place the dimension at an appropriate location and enter a new value in the **Modify** edit box and press ENTER. Figure 3-28 shows the aligned dimensioning of a line and an aligned dimension between two points.

Angular Dimensioning

Angular dimensions are used to dimension angles. You can select two line segments to apply the angular dimensions or use three points to apply the angular dimensions. You can also use angular dimensioning to dimension an arc. All these options of angular dimensioning are discussed next:

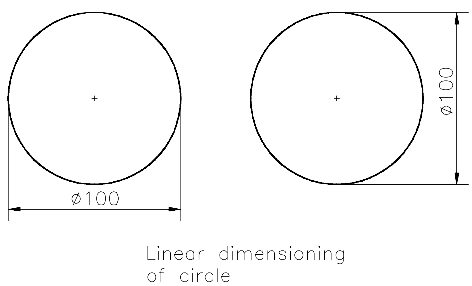


Figure 3-27 Linear dimensioning of circle

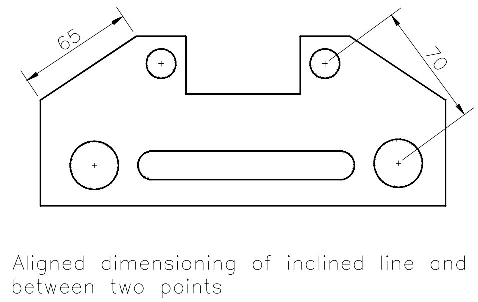


Figure 3-28 Aligned dimensioning

Angular Dimensioning Using Two Line Segments

You can select two line segments to apply angular dimensions. Choose the **Dimension** button from the **Sketch** toolbar and select the first line segment using the left mouse button. A dimension is attached to the cursor. Now, select the second line segment. An angular dimension will be attached to the cursor. Place the angular dimension and enter the new value of angular dimension in the **Modify** edit box. You have to be very careful while placing the angular dimension. This is because depending upon the location of the dimension placement, the interior angle, exterior angle, major angle, or minor angles are displayed. Figures 3-29 through 3-32 illustrate various angular dimensions depending upon the dimension placement point.

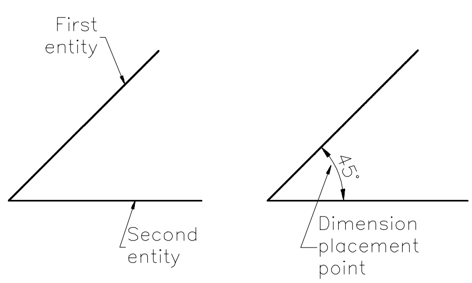


Figure 3-29 Angular dimension displayed according to the dimension placement point

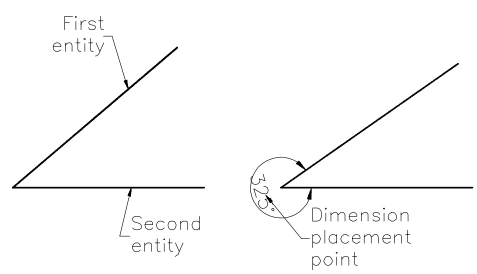


Figure 3-30 Angular dimension displayed according to the dimension placement point

Angular Dimensioning Using Three Points

You can also apply angular dimensions using three points. Choose the **Dimension** button from the **Sketch** toolbar. Select the first point using the left mouse button. This is the angle vertex point. Select the second point. A linear dimension is attached to the cursor. Next, select the third point; an angular dimension is attached to the cursor. Place the angular dimension and enter a new value of angular dimension in the **Modify** edit box. The points that can be used to apply the angular dimensions include the endpoints of lines or arcs, centerpoint of circles, and

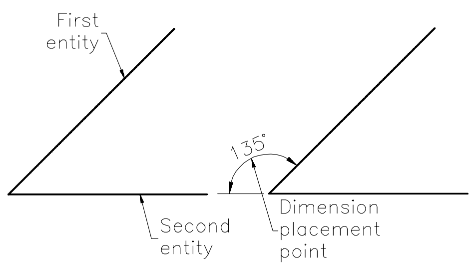


Figure 3-31 *Angular dimension displayed according to the dimension placement point*

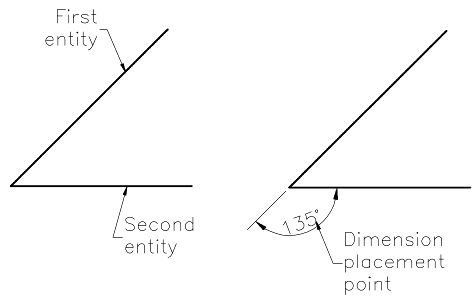


Figure 3-32 *Angular dimension displayed according to the dimension placement point*

endpoints of ellipse, parabola, and so on. Figure 3-33 shows the angular dimensioning using three points.

Angular Dimensioning of an Arc

You can use angular dimensions to dimension an arc. In case of arcs, the three points that should be used are the endpoints of the arc and the centerpoint of the arc. Figure 3-34 shows the angular dimensioning of an arc.

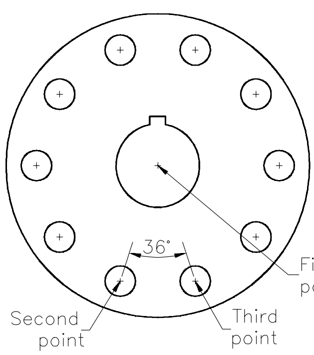


Figure 3-33 *Angular dimension displayed according to the dimension placement point*

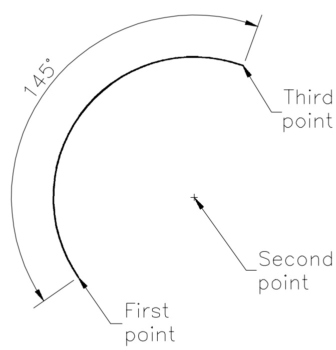


Figure 3-34 *Angular dimension displayed according to the dimension placement point*

Diameter Dimensioning

Diameter dimensions are applied to dimension a circle or an arc in terms of its diameter. For creating a diameter dimension, choose the **Dimension** button from the **Sketch** toolbar and select the entity to add the diametrical dimension. Now, using the left mouse button, place the dimension. In SolidWorks, when you select a circle to dimension by default, the diameter dimension is applied to it. However, when you select an arc, the radius dimension is displayed to an arc. To apply the diameter dimension to an arc, select the arc and place the radius dimension. Next right-click to display the shortcut menu and choose the **Properties** option from the shortcut menu. The **Dimension Properties** dialog box is displayed. Select the **Diameter**

dimension check box and choose **OK**. Figure 3-35 shows a circle and an arc with the diameter dimension.

Radius Dimensioning

Radius dimensions are applied to dimension a circle or an arc in terms of its radius. As mentioned earlier, by default the dimension applied to a circle is in the diameter form and the dimension applied to an arc is a radius dimension. To apply radius dimension, choose the **Dimension** button from the **Sketch** toolbar and select the arc. A radius dimension will be attached to the cursor. Using the left mouse button place the dimension at an appropriate place. To convert the radius dimension to the diameter dimension you need to select the **Diameter dimension** check box from the **Dimension Properties** dialog box. Figure 3-36 illustrates the radial dimensioning of a circle and an arc.

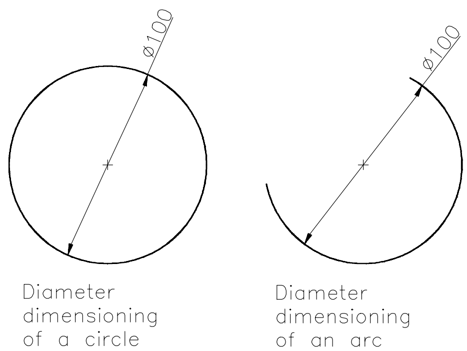


Figure 3-35 Diametrical dimensioning of a circle and an arc

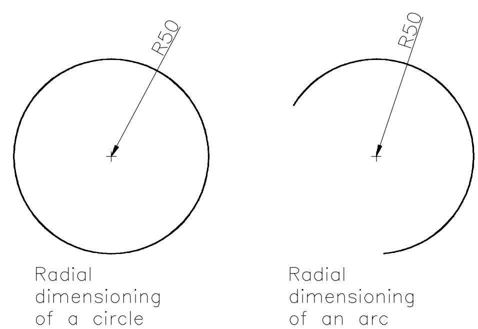


Figure 3-36 Radial dimensioning of a circle and an arc

Linear Diameter Dimensioning

Linear dimensioning is used to dimension the sketch of a revolved component. An example of a revolved component is shown in Figure 3-37. The sketch for a revolved component is drawn using simple sketcher entities as shown in Figure 3-38. If you dimension the sketch of the base feature of the given model using the linear dimensioning method then the same dimensions will be generated in drawing views. This may be confusing because in the shop floor drawing, you need diameter dimension of a revolved model. To overcome this problem, it is recommended that you create a linear diameter dimension as shown in Figure 3-38. For creating the linear diameter dimension, choose the **Dimension** button from the **Sketch** toolbar. Select the entity to be dimensioned and then select the center line around which the sketch will be revolved. Move the cursor to the other side of the centerline. A linear diameter dimension is displayed. Using the left mouse button place the dimension and enter a new value in the **Modify** dialog box.

Additional Dimensioning Options

In SolidWorks, you are also provided with some other dimensioning options other than those discussed earlier. The main additional dimensioning option is discussed next.

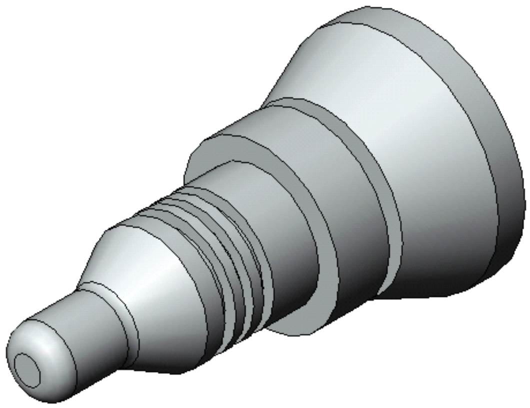


Figure 3-37 *A revolved component*

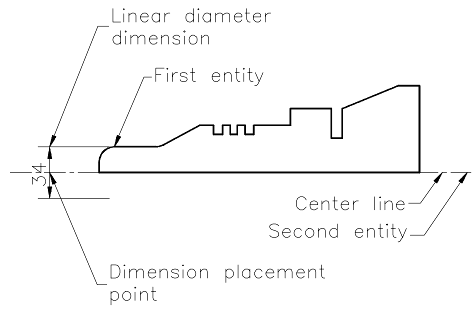


Figure 3-38 *Sketch for the revolved feature and linear diameter dimension*

Dimensions Between Arcs or Circles

By default, the dimension between two arcs, two circles, or between an arc and a circle is placed from centerpoint to centerpoint. For dimensioning two circles, choose the **Dimension** button from the **Sketch** toolbar and using the left mouse button select the first circle. A diameter dimension is attached to the cursor. Now, select the second circle. A linear dimension between the centerpoints of two circles is attached to the cursor; place the dimension using the left mouse button. Enter a new value of dimension in the **Modify** edit box and press ENTER. Figure 3-39 shows the dimensioning of two circles using this method.

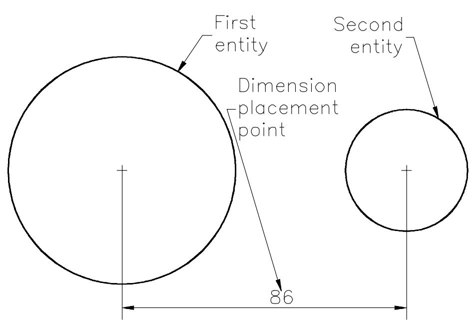
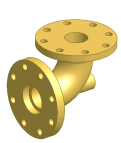


Figure 3-39 *Dimension with first arc condition as* ***Center*** *and second arc condition as* ***Center***



Tip. *You can also choose* ***Tools > Dimension > Parallel / Horizontal / Vertical*** *to add the dimensions to the sketch instead of using the* ***Dimension*** *button from the* ***Sketch*** *toolbar. But it is recommended that you use the* ***Dimension*** *button because using this button you can create any type of dimension whether it is a parallel, horizontal, vertical, diametrical, or radial dimension. In this book you will be using the* ***Dimension*** *button to dimension the sketches.*

Now, right-click to display the shortcut menu. Choose the **Properties** option from the shortcut menu to display the **Dimension Properties** dialog box. You will notice that in the **Dimension**

Properties dialog box, the **Center** radio buttons are selected in the **First arc condition :** and **Second arc condition :** areas. Therefore, the dimension displayed using this combination is the dimension between the centerpoint between two circles or arcs. If you choose the **Min** radio button from the **First arc condition :** area and the same from the **Second arc condition :** area, the resultant dimensioning will be displayed as the minimum distance between two circles or arcs as shown in Figure 3-40. If you select the first arc condition **Max** and also the second arc condition as **Max**, the dimensional value will be displayed as the maximum distance between two circles as shown in Figure 3-41. Similarly, you can adjust the first end condition and the second end condition accordingly.

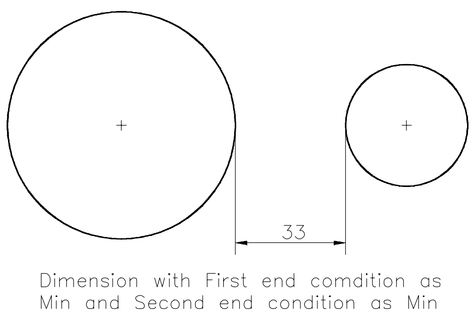


Figure 3-40 *Dimension with first and second arc condition as* ***Min***

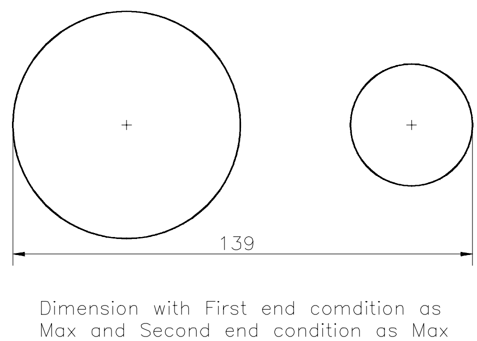


Figure 3-41 *Dimension with first and second arc condition as* ***Max***

CONCEPT OF FULLY DEFINED SKETCH

It is very necessary for you to understand the concept of fully defined sketches. While creating a model, first you have to draw the sketch for the base feature and then proceed further for creating other features. This is the reason sketching is the basic concept of modeling. After creating the sketches, you have to add the required relations and dimensions to constrain the sketch with respect to the surrounding environment. After creating the sketch and adding the required relations and dimensions, the sketch may exist in one of the six states. The six states of the sketch are discussed below:

1. Fully Defined
2. Over Defined
3. Under Defined
4. Dangling
5. No Solution Found
6. Invalid Solution Found

Fully Defined

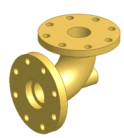
A fully defined sketch is a sketch in which all the entities of the sketch and their positions are fully defined by the relations or dimensions, or both. In the fully defined sketch, all the degrees of freedom of a sketch are constrained using relations and dimensions and the sketched entities cannot move or change their size and location unexpectedly. If the sketch is not fully defined, it can change its size or position at any time during the design because all the degree of freedom are not constrained. A fully defined sketch is displayed in black.

Over Defined

An over defined sketch is a sketch in which some of the dimensions, relations, or both are conflicting or the dimension or relations in the sketch have exceeded the required number. The over defined sketch is displayed in red. The over defined sketch geometry is constrained by too many dimensions and/or relations. Therefore, you need to delete the extra and conflicting relations or dimensions. It is recommended that you do not proceed further for creating the feature with an over defined sketch. The over defined sketch is solved to fully defined or under defined sketch by deleting the conflicting relations or dimensions. Deleting the over defining relations or dimension is discussed later in this chapter.

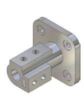
Under Defined

An under defined sketch is a sketch in which some of the dimensions or relations are not defined and the degree of freedom of the sketch is not fully constrained. In the under defined sketch, the dimensions and relations are not defined adequately and entities may move or change size unexpectedly. The sketched entities of the under defined sketch are displayed in blue. This is the reason the sketch is displayed in blue while drawing. This means the sketch is under defined and it needs some relations and dimensions to constrain its degree of freedom. When you add the relations and dimensions, the sketch changes to black color, suggesting that the sketch is fully defined. If the entire sketch is in black and only some of the entities are shown in blue, this means that the entities in blue require some dimension or relation.



Tip. *In SolidWorks, it is not necessary that you fully dimension or define the sketches before you use them to create the features of the model. However, it is recommended that you fully define the sketches before you proceed further for creating the feature.*

If you want to always use fully defined sketches before proceeding further, you can set the option by choosing ***Tools > Options*** *from the menu bar to display the* ***System Option - General*** *dialog box. Choose the* ***Sketch*** *option from the* ***System Options*** *tab. Select the* ***Use fully defined sketches*** *check box and choose* ***OK*** *from this dialog box.*



Note

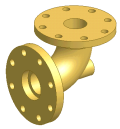
From this chapter onwards, you will work with fully defined sketches. Therefore, follow the above procedure to use the fully defined sketches in future.

Dangling

In the dangling sketch, the dimensions or relations lose their reference because of the deletion of an entity from which it was referenced. These entities are displayed dashed in brown color. You need to delete the dangling entities, dimensions, or relations that conflict.

No Solution Found

In the no solution found state, the sketch is not solved with the current constraints. Therefore, you need to delete the conflicting dimensions or relations and add other dimensions or relations.



Tip. *The status bar of the SolidWorks window is divided into four areas while working in the sketching environment. The **Sketch Definition** area of the status bar always displays the status of the sketch, dimension, and relation. If the sketch is under defined the status area will display the **Under Defined** message; if the sketch is over defined then the message displayed in the status area will be **Over Defined**; if the sketch is fully defined then the message displayed in the status area will be **Fully Defined**.*

The sketched entity, dimension, or relation will be displayed in pink.

Invalid Solution Found

In the invalid solution found state, the sketch is solved but the sketch will result in invalid geometry such as a zero length line, zero radius arc, or self-intersecting spline. The sketch entities for this state are displayed in yellow.

Sketch Dimension or Relation Status

In SolidWorks when you are applying the dimensions and relations to the sketches, sometimes you apply the dimensions or relations that are not compatible with the geometry of the sketched entities or that make the dimensioned entity over defined. The sketch dimensions or relations may have any of the following states:

1. Dangling
2. Satisfied
3. Over Defining
4. Not Solved
5. Driven

Dangling

A dangling dimension or relation is the one that cannot be resolved because the entity to which it was referenced is deleted. The dangling dimension appears in brown color.

Satisfied

A satisfied dimension is the one that is completely defined and is displayed in black.

Over Defining

An over defining dimension or relation overdefines one or more entities in the sketch. The over defining dimension appears in red.

Not Solved

The not solved dimension or relation is not able to determine the position of the sketched entities. A not solved dimension appears in pink.

Driven

The driven dimension's value is driven by other dimensions in the sketch that solve the sketch. The driven dimension appears in gray.

DELETING THE OVER DEFINING DIMENSIONS

In SolidWorks, when you add a dimension that over defines a sketch, the sketch and dimension turns red. Also, the **SolidWorks** warning window is displayed as shown in Figure 3-42.

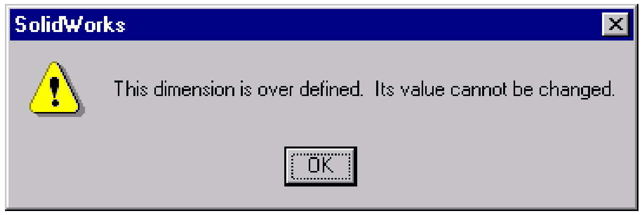


Figure 3-42 *The box warning you about over defined dimensions*

Choose the **OK** button from this window. The **Make Dimension Driven?** dialog box is displayed as shown in Figure 3-43. If you select the **Make This Dimension Driven** radio button from this dialog box and choose **OK**, then that dimension will become a driven dimension. The driven dimension is displayed in gray and you cannot modify a driven dimension. Its value depends upon the value of the driver dimension. If you change the value of the driver dimension, the value of the driven dimension will be automatically changed.

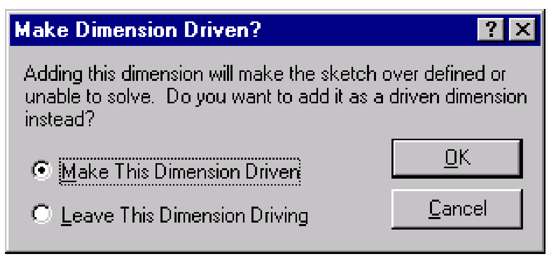


Figure 3-43 *The* ***Make Dimension Driven?*** *dialog box*

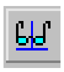
If you select the **Leave This Dimension Driving** radio button from the **Make Dimension Driven?** dialog box and choose **OK**, then some of the entities and dimension in the sketch will be displayed in red. You need to delete either the red sketched entity or the red dimension to make sure the sketch is no more over defined. The **SolidWorks** information dialog box will be displayed as shown in Figure 3-44 when you delete the over defining entity or dimension. The message displayed in this dialog box is **The sketch is no longer over defined.** Choose **OK** from the **SolidWorks** information dialog box. The sketch will be displayed in black, which indicates that the sketch is fully defined.

Figure 3-44 *The* ***SolidWorks*** *information dialog box*

You can also prevent the sketch from being over defining by choosing the **Cancel** button from the **Make Dimension Driven?** dialog box. If you choose **Cancel** from the **Make Dimension Driven?** dialog box, then the **SolidWorks** information dialog box will be displayed, as shown above, with a message displaying **The sketch is no longer over defined**.

Displaying and Deleting Relations

Toolbar:	Sketch Relations > Display/Delete Relations
Menu:	Tools > Relations > Display/Delete



If the sketch is over defined after adding the dimensions and relations, then you need to delete some of the over defining, dangling, or not solved relations or dimensions. You can view the relations applied to the sketch using the **Sketch Relations PropertyManager**. You can also delete the unwanted relations using this **PropertyManager**. To invoke the **Sketch Relations PropertyManager**, choose the **Display/Delete Relations** button from the **Sketch Relations** toolbar. You can also right-click in the drawing area to display the shortcut menu and choose the **Display/Delete Relations** option. The **Sketch Relations PropertyManager** will be displayed as shown in Figure 3-45. The confirmation corner is also displayed at the upper right corner of the drawing area. The options available in the **Sketch Relations PropertyManager** are discussed next.

Relations

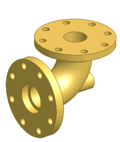
The **Relations** rollout is used to check, delete, and suppress the unwanted and conflicting relations. The status of the sketch or the selected entity is displayed in the **Information** area of this rollout. The various options of the **Relations** rollout are discussed next.

Filter

The **Filter** drop-down list is used to select the filter to show the relations in the **Sketch Relation PropertyManager**. The various options available in the **Filter** drop-down list are discussed next.

All in this sketch

The **All in this sketch** option is used to display all the relations applied to the sketch. This option is selected by default in the **Filter** drop-down list. The first relation displayed in the list will be selected by default and will appear with blue background. The status of the selected relations is displayed in the **Information** area of the **Relations** rollout. The overdefined relations are highlighted in red color. If you select a relation highlighted red in color in the **Relations** area, you will notice that the status of the selected relation is displayed as **Over Defining**. The dangling relation is highlighted in brown. When you select the dangling relation, the status of the relation will be displayed as **Dangling** in the **Information** area. Similarly, the not solved relation will be highlighted in yellow and the driven relation in gray.



Tip. *When you move the cursor on the relations displayed in the* ***Relations*** *area of the* ***Relations*** *rollout, the tooltip will inform you about the type of relation.*

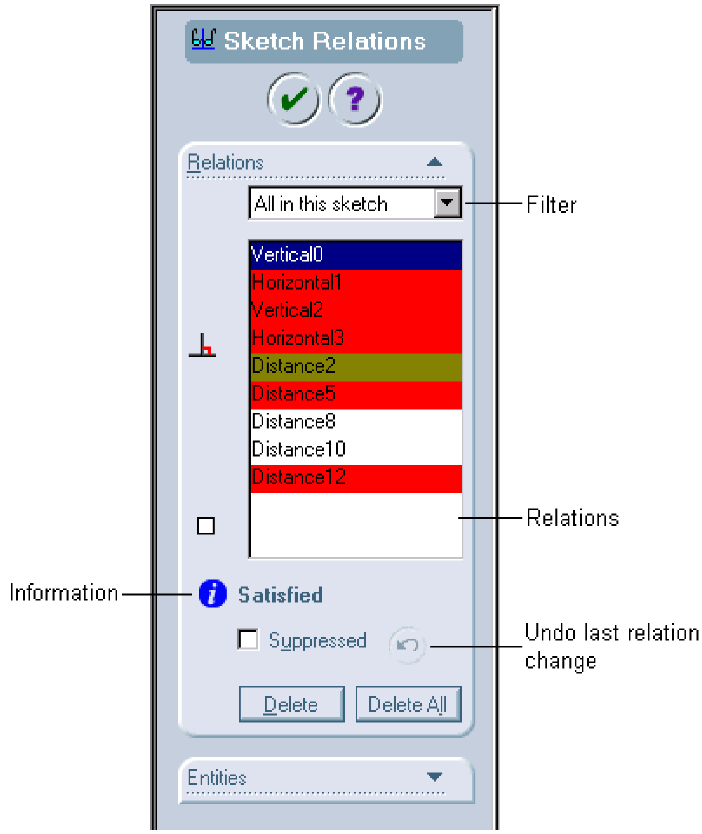


*Figure 3-45 The **Sketch Relations PropertyManager***

Dangling
The **Dangling** option is used to display only the dangling relations applied to the sketch.

Overdefining/Not Solved
The **Overdefining/Not Solved** option is used to display only the over defining and not solved relation. The dangling relations are also not solved relations. Therefore, they will also be displayed.

External
The **External** option is used to display the relations that have a reference with an entity outside the sketch. This entity can be an edge, vertex, or origin within the same model or it can be an edge, vertex, or origin of a different model within an assembly.

Design In Context
The **Design In Context** option is used to display only the relations that are in the context of a design. They are the relations between the sketch entity in one part and an

entity in another part. These relations are defined while working with the top-down assemblies.

Locked

The **Locked** option is used to display only the locked relations.

Broken

The **Broken** option is used to display only the broken relations.

Note

*The **Locked** and the **Broken** relations are applied while creating a part within the assembly environment. You will learn more about these relations in the later chapters.*

Selected Entities

The **Selected Entities** option is used to select the entities to display the relations. When you select this option from the **Filter** drop-down list, the **Selected Entities** area is displayed in the **Relations** rollout. When you select an entity to display the relations, the name of the selected entity is displayed in the **Selected Entities** area and the relation applied to this entity is displayed in the relation. To remove the selected entity from the selection set, select the entity to be removed and right-click to display the shortcut menu. Choose the **Delete** option from the shortcut menu. If you choose the **Clear Selections** option, all the entities will be removed from the selection set.

Suppressed

The **Suppressed** check box is selected to suppress the selected relation of the current configuration. When you suppress a relation, the relation is displayed in gray in the **Relations** area. The status of the suppress relation is displayed as **Satisfied** or **Driven** in the information area. If you suppress over defining dimensions the **SolidWorks** information dialog box will be displayed with a message that **The sketch is no longer over defined.** Choose the **OK** button from this dialog box.

Note

The configurations used in the part and assembly modeling will be discussed in the later chapters.

Delete

The **Delete** button is used to delete the relation selected in the **Relations** area.

Delete All

The **Delete All** button is used to delete all the relations that are displayed in the **Relations** area.

Undo last relation change

The **Undo last relation change** button is used to undo the **Delete**, **Replace**, and **Suppressed** options used earlier. The replace option is discussed later in this chapter.

Entities

The **Entities** rollout is used to display the entities that are referred to in the selected relation.

This rollout is also used to display the status of the selected relation and the external reference, if any. By default, the **Entities** rollout is closed. Move the cursor to the black triangle at the right of the **Entities** rollout. The **Entities** rollout is shown in Figure 3-46. The various options in the **Entities** rollout are discussed next.

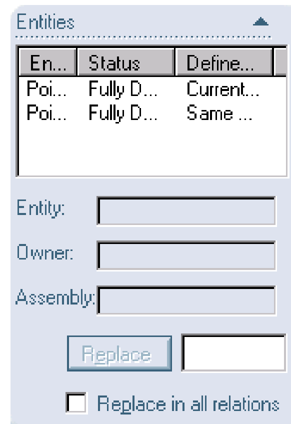


*Figure 3-46 The **Entities** rollout*

Entities used in the selected relation

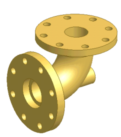
The **Entities used in the selected relation** area is used to display the information about entities used in the selected relation. The information about the name of the entity, the status of the entity, and the place where the entity is defined is provided. This area is divided in three columns. The three columns available in the **Entities** rollout are discussed next.

Entity.
The **Entity** column is used to display the entity or entities on which the selected relation is applied.

Status.
The **Status** column is used to display the status of the selected relation. The status displayed in the **Status** column can be **Fully Defined**, **Dangling**, **Over Defined**, or **Not Solved**.

Defined In.
The **Defined In** column is used to display the placement of the entity. The entity can be placed in any of the following places:

Current Sketch.
The **Current Sketch** option is displayed in the **Defined In** column when the entity is placed in the same sketch.



Tip. *By default the **Override Dims on Drag** option is selected from **Tools > Sketch Settings** menu in the menu bar. As a result, if you drag a sketched entity, the value of dimension applied to the sketched entity will be changed automatically. If you clear this option you cannot change a dimensioned sketched entity by dragging because it will not override the dimension. However, if the sketch is not fully defined, the entities that are not properly dimensioned or constrained will move.*

*By default, the **Automatic Solve** option is selected from the **Tools > Sketch Settings** menu in the menu bar. This option helps you to solve the relations and dimensions automatically when you drag or modify a sketched entity. If you clear this option, a message that **The sketch cannot be dragged because Auto Solve Mode is off. To drag the sketch, please turn the Auto Solve Mode on.** appears. If you modify the dimension value using the **Modify** dialog box, the dimension will not update automatically, and you have to update the new dimension manually. To update and solve the dimension you have to choose the **Rebuild** button from the **Standard** toolbar or press CTRL+B using the keyboard.*

Same Model.
The **Same Model** option is displayed in the **Defined In** column when the entity is defined as placed in the same model. This means that the entity is placed in the same model but outside the sketch. This entity can be an edge, vertex, or an origin of the same model.

External Model.
The **External Model** option is displayed in the **Defined In** column when the entity is placed in some other model but within the assembly. This entity can be the edge, vertex, or the origin of a different model but in the same assembly.

Entity

The **Entity** display box is used to display the name of the entity and the name of the part in which the selected entity is placed. This entity is selected in the **Entity** column of the **Entities used in the selected relation** area. The selected entity is also highlighted in the drawing area.

Owner

The **Owner** display box is used to display the name of model in which the entity is placed when the **External Model** option is displayed in the **Defined In** column.

Assembly

The **Assembly** display box is used to display the path of the assembly in which the entity is placed when the **External Model** option is displayed in the **Defined In** column.

Replace

The **Replace** button is used to replace the selected entity from the **Entity** column with some other entity from the drawing area. When you select the entity from the drawing area, the entity will be displayed in the display box provided below the **Replace** button. Choose the **Replace** button to replace the entity. If the sketch is overdefined, you will be given a warning message. Sometimes after replacing the entity, the status of the entity is changed to not solved or overdefining. You need to undo the last operation. The **Replace in all relations** check box is selected to replace the entity in all the relations.

VIEWING AND EXAMINING RELATIONS

You can view and examine the relations applied to the sketch. To view and examine the relations applied to a particular entity, select that entity. The **PropertyManager** of that entity will be displayed. The **Existing Relations** area will display the relations that are already applied to the selected entity. The **Information** area will inform you about the status of the sketch. Select the relations one by one from the **Existing Relations** area. The callout of the relations will be displayed in the drawing area as shown in Figure 3-47. The callout that is displayed in the drawing area is divided in two parts. The first part displays the symbol of the relation applied to the selected entity and the second part displays the name of the entity. If you double-click the relation symbol then the **Sketch Relations PropertyManager** will be displayed. You can analyze, examine, and delete the unwanted, over defining, dangling, and not solved relations using this option.

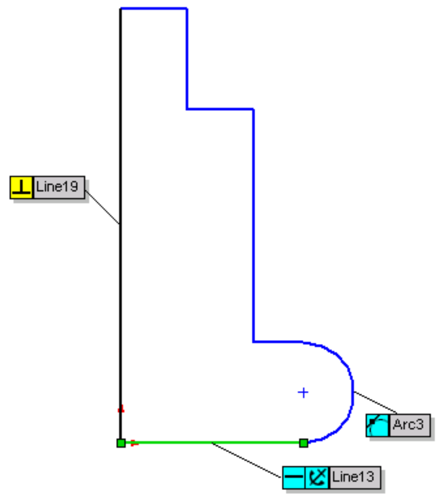
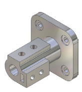
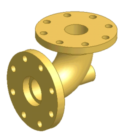


Figure 3-47 *Callout of relations displayed in the drawing area*



Note
To exit the callout, click anywhere on the screen.



Tip. *You can also display the relation callouts in the drawing area without invoking the **Sketch Relations PropertyManager**. Double-click on the sketched entity and the relation callouts will be displayed in the drawing area.*

OPENING AN EXISTING FILE

Toolbar:	Standard > Open
Menu:	File > Open

The **Open** dialog box is used to open an existing SolidWorks part, assembly, or drawing document. You can also use this dialog box to import files from other applications saved in some standard file formats. Choose the **Open** button from the **Standard** toolbar, or press CTRL+O keys to invoke the **Open** dialog box. The **Open** dialog box is shown in Figure 3-48. Various options available in this dialog box are discussed next.

Look in drop-down list

The **Look in** drop-down list is used to specify the drive or directory in which the file is saved. The location of the file and the folder that you browse is shown in this drop-down list.

File name

The name of the file selected is shown in the **File name** edit box. You can also enter the name of the file to open in this edit box.

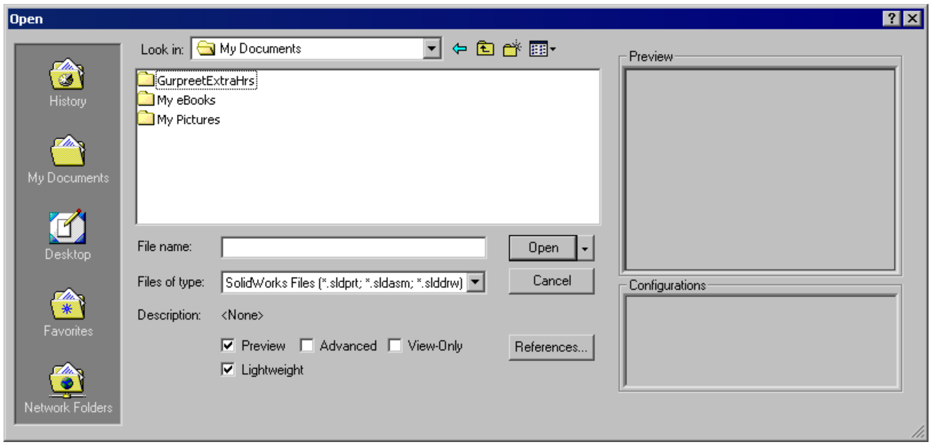


*Figure 3-49 The **Open** dialog box*

Files of type

The **Files of type** drop-down list is used to specify the type of file to open. Using this drop-down list, you can select a particular type of file type such as the part file, assembly file, drawing file, all SolidWorks files, and so on. You can also define the file standard format in this drop-down list to import the files saved in those file formats. You will learn more about importing the files in the later chapters.

Open as read-only

The **Open as read-only** option is selected to open the document as a read only file. This option is available after invoking the flyout by selecting the down arrow available at the right of the **Open** button. If you modify the design in a read-only file, the changes are saved in a new file. The original file is not modified. This also allows another user to access the document while the document is open on your system.

Preview

The **Preview** check box is selected to display the **Preview** area in this dialog box. In the **Preview** area you can preview the selected part, assembly, or drawing document before opening.

Advanced

The **Advanced** check box is selected to display the configurations available in the selected file. The configurations available in the selected file are displayed in the **Configurations** area. You will learn more about configurations in the later chapters.

View-Only

The **View-Only** check box is selected to open a SolidWorks document in the view only format. When you open a view-only file, only the tools related to viewing the models are enabled. Rest of the tools are not available. This is the reason you cannot do any modification in the view-only document. You can only use the zoom, pan, or dynamically rotate tools. If you want to edit the design, right-click in the drawing area and choose the **Edit** option from the shortcut menu to edit the design.

Lightweight

The **Lightweight** check box is selected to open the assembly document using the lightweight parts. You will learn more about the lightweight parts in the later chapters.

TUTORIALS

Tutorial 1

In this tutorial you will draw the sketch of the model shown in Figure 3-49. This is the same sketch that was drawn in Tutorial 1 of Chapter 1. In this tutorial you will draw the sketch using the mirror line and then add the required relations and dimensions. The sketch is shown in Figure 3-50. The solid model is given only for reference. (**Expected time: 30 min**)

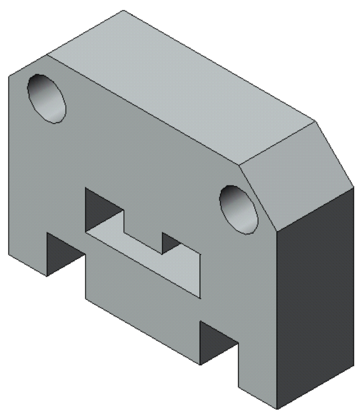


Figure 3-49 *Solid Model for Tutorial 1*

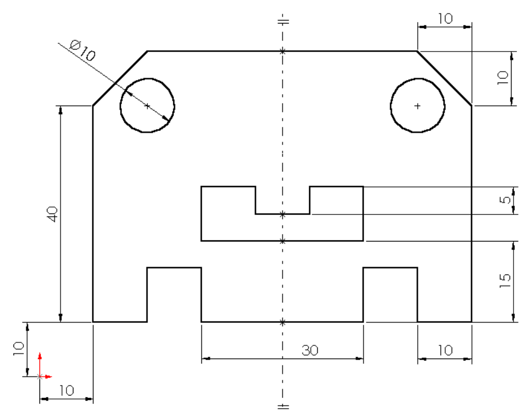


Figure 3-50 *Sketch of the model*

The steps that will be followed to complete this tutorial are listed below:

a. Start SolidWorks and then open a new part file.
b. Maximize the part file document and then switch to the sketching environment.
c. Create a mirror line using the **Centerline** and the **Mirror** tool.
d. Draw the sketch of the model on one side of the mirror line so that it is automatically drawn on the other side, refer to Figure 3-51 through Figure 3-56.
e. Add the required relations to the sketch, refer to Figures 3-57 and Figure 3-58.
f. Add the required dimensions to the sketch and fully define the sketch, refer Figure 3-59.
g. Save the sketch and then close the file.

Starting SolidWorks and Opening a New Part Document

1. Start SolidWorks by choosing **Start** > **Programs** > **SolidWorks 2003** > **SolidWorks 2003** or by double-clicking the shortcut icon of SolidWorks available on the desktop of your computer.

   The **Welcome to SolidWorks 2003** window is displayed and also the **Tip of the Day** dialog box. As mentioned earlier, the tips that are displayed in the **Tip of the Day** dialog are very useful in making the best use of SolidWorks.

2. Close the **Tip of the Day** dialog box by choosing the **Close** button and then choose the **New Document** option from the **Welcome to SolidWorks** window.

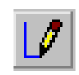
   The **New SolidWorks Document** dialog box is displayed.

3. Select the **Part** option and then choose the **OK** button from the **New SolidWorks Document** dialog box.

   A new SolidWorks part document is opened. But the part document window is not maximized in the SolidWorks window.

4. Choose the **Maximize** button available on the upper right corner of the part document window to maximize the document window.

5. Choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment.



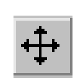
6. Set the units for measuring linear dimensions to **Millimeters** and the units for angular dimensions to **Degree** using the **Document Properties - Detailing** dialog box. If you selected **Millimeters** as units while installing SolidWorks, then you can skip this point.

Drawing the Mirror Line

In this tutorial, you will draw the sketch of the given model with the help of a mirror line. As mentioned earlier, the sketches that are symmetrical along any axis are recommended to be drawn using the mirror line. The mirror line is drawn using the **Centerline** and **Mirror** tools. When you draw an entity on one side of the mirror line, the same entity is drawn automatically on the other side of the mirror line. The entity drawn on the other side is the mirror image of the entity you draw. A symmetrical relation is applied to the entities on both the sides of the mirror line. Therefore, if you modify an entity on one side of the mirror line, the same modification is reflected in the mirrored entity and vice versa. First, you need to create a mirror line.

The origin of the sketcher environment is placed in the center of the drawing area and you have to create the sketch in the first quadrant. Therefore, it is recommended that you modify the drawing area such that the area in the first quadrant in increased. This can be done using the **Pan** tool.

1. Choose the **Pan** tool button from the **View** toolbar. The select cursor will be replaced by the pan cursor.



2. Press and hold the left mouse button down and drag the cursor toward the bottom left corner of the screen.

   You will notice that the origin also moves toward the bottom left corner of the screen. This increase the drawing area in the first quadrant.

3. After dragging the origin close to the lower left corner, release the left mouse button.

4. Choose the **Centerline** button from the **Sketch Tools** toolbar.

   The pan cursor is replaced by the line cursor.

5. Move the line cursor to a location whose coordinates are close to 45mm 70mm 0mm. You do not need to move the cursor to exactly this location. You can move it to a point close to this location.

6. Specify the startpoint of the centerline and move the line cursor vertically downward. Specify the endpoint of the centerline when the length of the line cursor shows a value close to 80.

   As soon as you specify the endpoint of the center line, a rubber-band line is attached to the line cursor. Double-click any where in the drawing area or right-click to display the shortcut menu and choose the **End chain** option from the shortcut menu to end line creation.

7. Choose the **Zoom to Fit** button from the **View** toolbar to fit the sketch on the screen.

8. Choose the **Select** button from the **Sketch** toolbar to toggle back to the selection mode and select the center line.

9. Choose the **Sketch Mirror** button from the **Sketch Tools** toolbar to convert the center line into a mirror line and invoke the automatic mirror option.

Drawing the Sketch

You will draw the sketch on the right side of the mirror line and the same sketch will be automatically drawn on the other side of the mirror line.

1. Choose the **Line** button from the **Sketch Tools** toolbar.

   The arrow cursor is replaced by the line cursor.

2. Move the line cursor to a location where the coordinates are close to 45mm 10mm 0mm. The line cursor turns yellow in color and an orange bulb is displayed. This suggests that the cursor snaps the mirror line.

3. Specify the startpoint of the line at this point and move the cursor horizontally toward the right. Specify the endpoint of the line when the length of the line above the line cursor shows a value close to 15. As soon as you press the left mouse button to specify the endpoint of the line, a line of the same length is drawn automatically on the other side of the mirror line. Figure 3-51 shows the mirror image created automatically on the other side of the mirror line. A rubber-band line is attached to the cursor.

   You will notice that the mirror image that is automatically created on the left of the mirror line is merged with the line drawn on the right. Therefore, the entire line becomes a single entity. As mentioned earlier, the mirror image of the line is merged with the line that you draw only if one of the endpoints of the line you draw is coincident with the mirror line.

4. Move the cursor vertically upward. Specify the endpoint when the length of the line on the line cursor displays a value close to 10. Figure 3-52 shows the sketch after drawing the vertical lines.

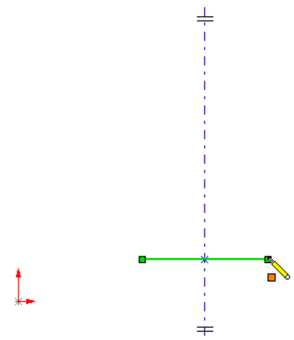


Figure 3-51 *After releasing the left mouse button*

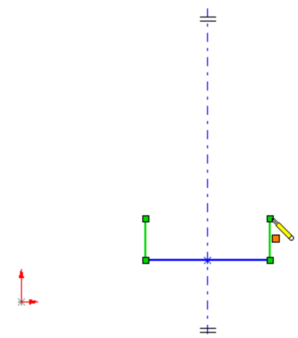


Figure 3-52 *After releasing the left mouse button*

A rubber-band line is attached to the cursor.

5. Move the line cursor horizontally toward the right. Specify the endpoint of the line when the length of the line on the line cursor displays a value close to 10.

6. Move the line cursor vertically downward. Specify the endpoint when the length of the line on the line cursor displays a value close to 10.

7. Move the line cursor horizontally toward the right. Specify the endpoint when the length of the line on the line cursor displays a value close to 10.

8. Move the line cursor vertically upward. Specify the endpoint when the length of the line on the line cursor displays a value close to 40.

9. Move the line cursor such that the line is drawn at an angle close to 135-degree, see Figure 3-53. Specify the endpoint when the value of the length of the line is close to 14.14.

10. Move the line cursor horizontally toward the left. Specify the endpoint when the cursor snaps to the mirror line and the line cursor turns yellow in color. A rubber-band line is attached to the cursor. Double-click anywhere on the screen to end line creation. The sketch after completing the outer loop is shown in Figure 3-54.

    Next, you will draw the sketch of the inner cavity. To draw the sketch of the inner cavity, you will start drawing the lower horizontal line.

11. Move the line cursor to a location whose coordinates are around 45mm 25mm 0mm.

12. Specify the startpoint of the line at this point and move the cursor horizontally toward the

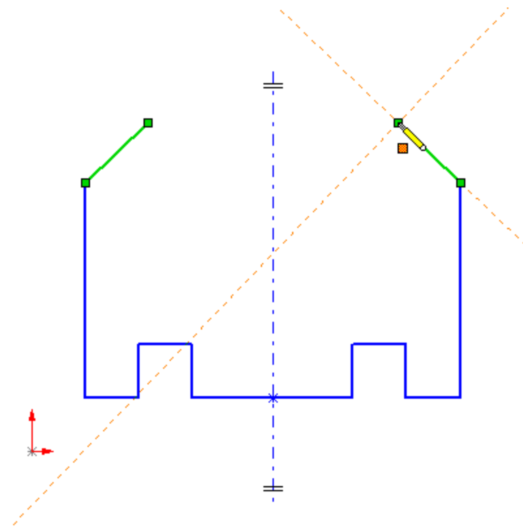


Figure 3-53 *Sketch after drawing the aligned line*

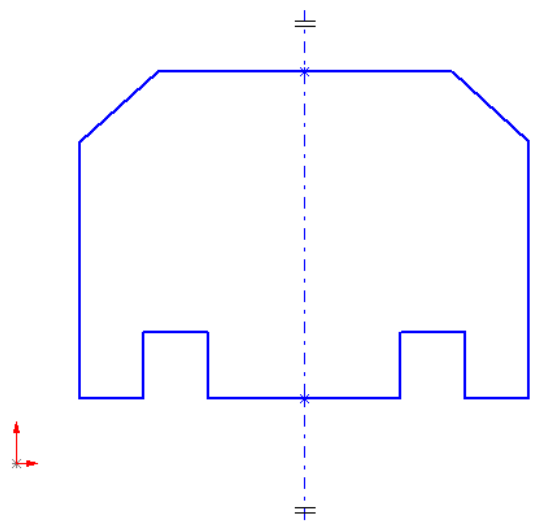


Figure 3-54 *Sketch after completing the outer profile of the sketch*

right. Specify the endpoint when the length of the line above the line cursor shows a value close to 15.

13. Move the line cursor vertically upwards. Specify the endpoint when the length of the line on the line cursor displays a value close to 10.

14. Move the line cursor horizontally toward the left. Specify the endpoint when the length of the line on the line cursor displays a value close to 10.

15. Move the line cursor vertically downwards. Specify the endpoint when the length of the line

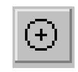
on the line cursor displays a value close to 5.

16. Move the line cursor horizontally toward the left. Specify the endpoint when the line cursor snaps to the mirror line.

17. Double-click anywhere on the screen to end line creation. The sketch after completing the inner cavity is shown in Figure 3-55.

Next, you will draw the circles using the **Circle** tool from the **Sketch Tools** toolbar.

18. Choose the **Circle** button from the **Sketch Tools** toolbar to invoke the circle tool.



When you invoke the circle tool the select cursor will be replaced by the circle cursor.

19. Move the circle cursor to the point where the inferencing line originating from the endpoints of the right inclined line meet.

20. Specify the center of the circle at this point and move the circle cursor toward the left to define the circle. Press the left mouse button when the radius of the circle above the circle cursor shows a value close to 5.

The mirror image of the circle is automatically drawn on the other side of the mirror line. The sketch after drawing the circle is shown in Figure 3-56.

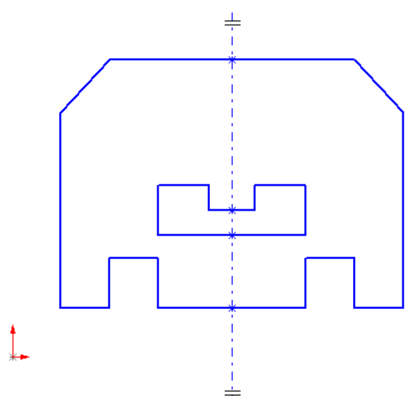


Figure 3-55 *Sketch after drawing the inner cavity*

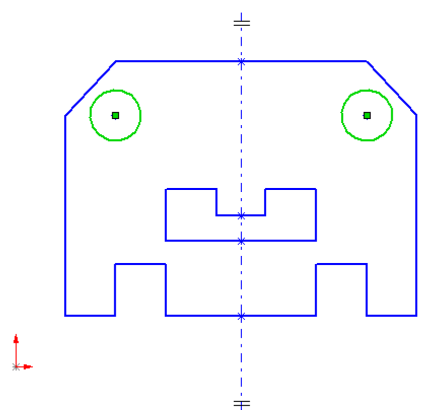
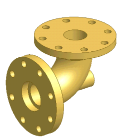


Figure 3-56 *Sketch after drawing the circle*

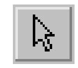


Tip. *Sometimes unwanted inferencing lines are displayed when you are drawing a sketch. You can remove the unwanted inferencing lines by choosing **View > Redraw** from the menu bar. You can also redraw the screen by pressing CTRL+R key.*

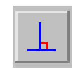
Adding the Required Relations

After drawing the sketch, you need to add the relations using the **Add Relations PropertyManager**. The relations are applied to a sketch to constrain its degree of freedom, to reduce the number of dimensions in the sketch, and also to capture the design intent in the sketch.

1. Choose the **Select** button from the **Sketch** toolbar to remove the circles created previously from the selection set.



2. Choose the **Add Relation** button from the **Sketch Relations** toolbar to invoke the **Add Relations PropertyManager**. The confirmation corner is also displayed at the upper right corner of the drawing window.



3. Select the centerpoint of the circle on the right and then select the lower endpoint of the right inclined line. The name of the selected entities are displayed in the **Selected Entities** area of the **Add Relations PropertyManager**.

   The relations that can be applied to the two selected entities are displayed in the **Add Relations** area of the **Add Relations PropertyManager** as shown in Figure 3-57. The **Horizontal** option is highlighted, suggesting that the horizontal relation is the most appropriate relation for the selected entities.

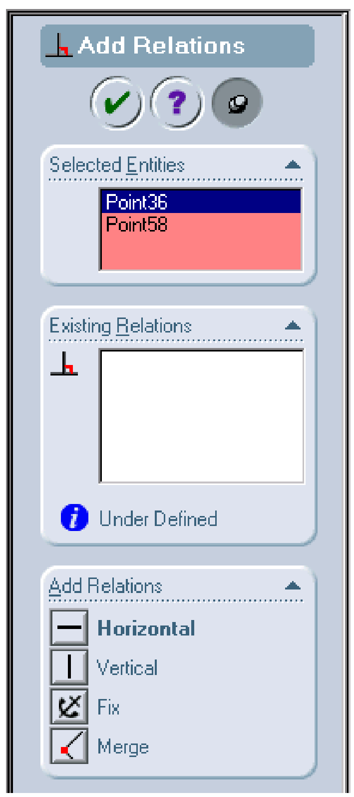


Figure 3-60 Add Relations PropertyManager

Note
The name of the entities displayed in the ***Selected Entities*** *area of the* ***Add Relations PropertyManager*** *may be different from those displayed on your screen.*

4. Choose the **Horizontal** button from the **Add Relations** area to apply the **Horizontal** relation to the selected entities.

5. Move the cursor to the drawing area and right-click to invoke the shortcut menu. Choose the **Clear Selections** option to remove the selected entities from the selection set.

6. Select the centerpoint of the circle on the right and the upper endpoint of the right inclined line.

   The relations that can be applied to the selected entities are displayed and the **Vertical** option is highlighted.

7. Choose the **Vertical** button from the **Add Relation** area of the **Add Relations PropertyManager**. Right-click in the drawing area and choose the **Clear Selection** option.

8. Select the entities shown in Figure 3-58. Choose the **Equal** button from the **Add Relations** area of the **Add Relations PropertyManager**.

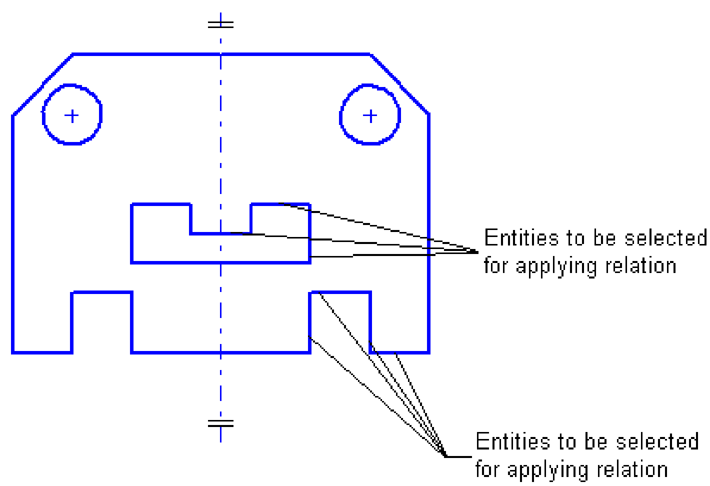


Figure 3-58 *Entities to be selected to apply the equal relation*

9. Choose the **OK** button from the **Add Relations PropertyManager** or choose the **OK** icon from the confirmation corner to close the **PropertyManager**. Specify a point on the screen to clear the selected entities.

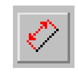
Applying the Dimensions to the Sketch

Next, you will apply the dimensions to the sketch and fully define the sketch. As mentioned earlier, the sketched entities are shown in blue, suggesting that the sketch is under defined.

It will be changed to black after applying the required dimensions to the sketch, suggesting that the sketch is fully defined.

1. Choose **Tools** > **Options** from the menu bar to display the **System Options - General** dialog box. Select the **Input dimension value** check box if cleared and choose **OK** from the **System Options - General** dialog box. This check box is selected to invoke the modify dialog box to enter a new dimension value and modify the sketch as you place the dimension.

2. Choose the **Dimension** button from the **Sketch** toolbar or right-click in the drawing area to display the shortcut menu. Choose the **Dimension** option to invoke the dimension option.



   The cursor is replaced by the dimension cursor.

3. Move the dimension cursor to the lower right horizontal line. The lower right horizontal line is highlighted in red color.

4. Select the line. A linear dimension is attached to the cursor.

5. Move the cursor downwards and using the left mouse button, place the dimension below the line, refer to Figure 3-59.

   As you place the dimension, the **Modify** dialog box is displayed.

6. Enter the dimension value of **10** in this dialog box and press ENTER. The dimension is placed and the length of line is also modified to 10.

7. Move the dimension cursor to the lower middle horizontal line. Select the line when the line changes to red. A dimension is attached to the cursor.

8. Move the cursor downwards and place the dimension using the left mouse button. Enter a value of **30** in the **Modify** edit box and press ENTER.

9. Move the cursor to the outer left vertical line and when the color of the line changes to red, select the line. A dimension is attached to the cursor.

10. Move the cursor to the left and use the left mouse button to place the dimension. Enter a value of **40** in the **Modify** dialog box and press ENTER.

11. Select the right inclined line. A dimension is attached to the cursor. Move the cursor vertically upwards to apply the horizontal dimension to the selected line. Using the left mouse button place the dimension at an appropriate place, see Figure 3-59.

12. Enter a value of **10** in the **Modify** dialog box and press ENTER.

13. Again, select the right aligned line. A dimension is attached to the cursor. Move the cursor horizontally toward the right to apply the vertical dimension for the selected line. Using the

left mouse button place the dimension at an appropriate place, see Figure 3-59.

18. Enter a value of **10** in the **Modify** dialog box and press ENTER.

19. Move the cursor to the left circle and when the circle is highlighted in red, select it. A diameter dimension is attached to the cursor. Move the cursor outside the sketch.

20. Place the diameter dimension. Enter a value of **10** in the **Modify** dialog box and press ENTER.

21. Select the lower horizontal line of the inner cavity. A linear dimension is attached to the cursor. Select the lower horizontal line of the outer loop.

    A vertical dimension between the lower horizontal line of the inner cavity and the lower horizontal line of the outer sketch is attached to the cursor.

22. Move the cursor horizontally toward the right and place the dimension. Enter a value of **15** in the **Modify** dialog box and press ENTER.

23. Select the inner right vertical line of the cavity and place the dimension outside the sketch. Enter a value of **5** in the **Modify** dialog box and press ENTER.

24. Select the lower horizontal line of the outer sketch and the origin using the left mouse button.

25. Move the cursor horizontally toward the left and place the dimension. Enter a value of **10** in the **Modify** dialog box.

    Notice, that some of the entities are displayed in black. This suggests that these entities are now fully defined. But you have to fully define the entire sketch. So you need to add some more dimensions.

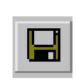
26. Select the outer left vertical line of the outer sketch and the origin.

27. Move the cursor vertically downward and place the dimension using the left mouse button. Enter a value of **10** in the **Modify** dialog box.

    Notice that all the entities are displayed in black. This suggests that the sketch is fully defined. If the sketch is not fully defined then you have to add a dimension between the outer right vertical line and the outer left vertical line. The value of the dimension should be maintained 70. The fully defined sketch, after applying all the required relations and dimensions, is shown in Figure 3-59.

Saving the Sketch

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.



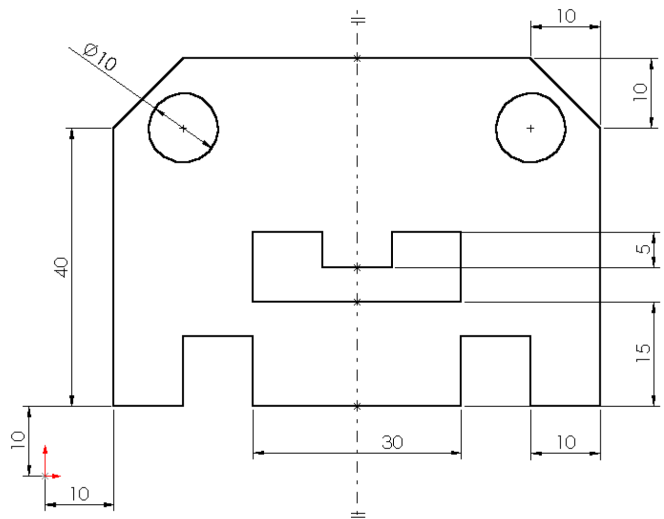
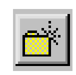


Figure 3-59 *Fully defined sketch after applying all the required relations and dimensions*



2. Double-click the **SolidWorks** directory. Choose the **Create New Folder** button from the **Save As** dialog box. Enter the name of the folder as *c03* and press ENTER.

3. Enter the name of the document as *c03-tut01.sldprt* in the **File name** edit box and choose the **Save** button.

   The document will be saved in the *\My Documents\SolidWorks\c03* directory.

4. Close the file by choosing **File** > **Close** from the menu bar.

Tutorial 2

In this tutorial you will draw the sketch of the model shown in Figure 3-60. You will draw the sketch using the mirror line and then add the required relations and dimensions. The sketch is shown in Figure 3-61. The solid model is given only for reference. **(Expected time: 30 min)**

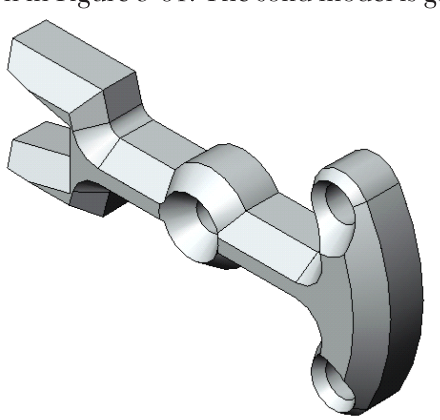


Figure 3-60 *Solid Model*

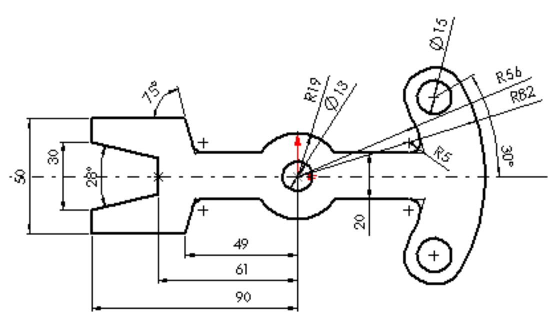
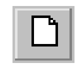


Figure 3-61 *Sketch of the solid model*

The steps that will be followed to complete this tutorial are listed below:

a. Start SolidWorks and then open a new part file.
b. Maximize the part file document and then switch to the sketching environment.
c. Create a mirror line.
d. Draw the sketch on one side of the mirror line, refer to Figure 3-62.
e. Trim the arcs and circles and add the fillets, refer to Figure 3-63 through Figure 3-65.
f. Add the required relations.
g. Add the dimensions and fully define the sketch, refer Figure 3-66.

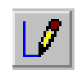
Opening a New File



1. Choose the **New** button from the **Standard** toolbar to invoke the **New SolidWorks Document** dialog box.

2. Select the **Part** option and then choose the **OK** button from the **New SolidWorks Document** dialog box.

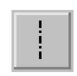
   A new SolidWorks part document is opened.



3. Choose the **Sketch** button from the **Sketch** toolbar to invoke the sketching environment.

Drawing the Mirror Line

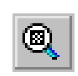
Similar to the last tutorial, you will draw the sketch of the given model with the help of a mirror line.



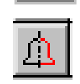
1. Choose the **Centerline** button from the **Sketch Tools** toolbar.

2. Move the line cursor to a location whose coordinates are close to -102mm 0mm 0mm. You do not need to move the cursor to exactly this location. You can move it to a point close to this location.

3. Specify the startpoint of the centerline at this point and move the line cursor horizontally toward the right. Specify the endpoint of the center line when the length of the line shows a value close to 204. Double-click anywhere in the drawing area to end line creation.



4. Choose the **Zoom to Fit** button from the **View** tollbar to fit the sketch on the screen.

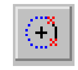


5. Invoke the **Select** tool and select the centerline. Choose the **Sketch Mirror** button from the **Sketch Tools** toolbar to create the mirror line and enable the automatic mirror option.

Drawing the Sketch

You will draw the sketch on the upper side of the mirror line and the same sketch will be automatically drawn on the other side of the mirror line.

1. Choose the **Centerpoint Arc** button from the **Sketch Tools** toolbar.



   The arrow cursor is replaced by the arc cursor.

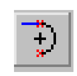
2. Move the arc cursor close to the origin. Specify the centerpoint of the arc when the cursor turns yellow in color. Move the cursor horizontally toward the right. The cursor snaps to the mirror line. As you move the cursor, a reference circle is drawn. Specify the startpoint of the arc when the radius of the arc on the arc cursor shows a value close to 82.

3. Move the arc cursor in the counterclockwise direction. Specify the endpoint of the arc when the value of the angle above the arc cursor shows a value close to 30-degree. The mirror image of the sketched entity is automatically created on the other side of the mirror line.

4. Move the arc cursor to the origin. Specify the centerpoint of the arc. Move it horizontally toward the right. The cursor snaps to the mirror line and a reference circle is drawn. Specify the startpoint of the arc when the radius of the arc on the arc cursor shows a value close to 56.

5. Move the arc cursor in the counterclockwise direction. Specify the endpoint of the arc when the value of the angle above the arc cursor shows a value close to 30-degree. The mirror image of the sketched entity is automatically created on the other side of the mirror line.

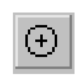
6. Choose the **Tangent Arc** button from the **Sketch Tools** toolbar.



7. Move the cursor to the upper endpoint of the left arc and when cursor turns to yellow, specify the first point of the tangent arc. Move the cursor close to the upper endpoint of the right arc and when the cursor turns yellow, specify the second point of the tangent arc.

   A rubber-band arc is attached to the cursor. Double-click anywhere in the drawing area to end arc creation.

8. Choose the **Circle** button from the **Sketch Tools** toolbar. The arc cursor will be replaced by the circle cursor



9. Move the cursor to the centerpoint of the upper arc and when the cursor turns yellow, specify the centerpoint of the circle. Move the cursor horizontally toward the right. Press the left mouse button when the radius of the circle above the circle cursor shows a value close to 7.5.

10. Move the cursor to the origin and when the cursor turns yellow, specify the centerpoint of the circle. Move the cursor horizontally toward the right. Press the left mouse button when the radius of the circle above the circle cursor shows a value close to 6.5.

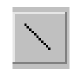
    The **SolidWorks** warning dialog box is displayed. This warning message will inform you “**Unable to create the symmetric element**”. This message is displayed because the circle is created on both sides of the mirror line. Therefore, a symmetric element is not created in this case.

11. Choose **OK** from the **SolidWorks** warning dialog box.

12. Move the cursor to the origin and when the cursor turns yellow in color, specify the centerpoint of the circle. Move the cursor horizontally toward the right. Press the left mouse button when the radius of the circle above the circle cursor displays a value close to 13. The **SolidWorks** warning dialog box will be displayed.

13. Choose **OK** from the **SolidWorks** warning dialog box.

14. Choose the **Line** button from the **Sketch Tools** toolbar.



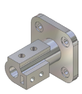
15. Move the line cursor to a location whose coordinates are close to 55mm 10mm 0mm. The cursor snaps to the left arc.

16. Specify the startpoint of the line at this point and move the cursor horizontally toward the left. Specify the endpoint of the line when the line cursor snaps the outer circle. Double-click anywhere in the drawing area to end line creation.

17. Move the cursor to a location whose coordinates are close to -10mm 10mm 0mm.

18. Specify the startpoint of the line on this location and move the cursor horizontally toward the left. Specify the endpoint when the length of the line above the line cursor shows a value close to 42.

19. Move the line cursor such that the line is drawn at an angle close to 105^{o}. Specify the endpoint when the length of line is close to 15.



Note

*You may need to scroll down the **Line PropertyManager** to view the angle. Move the cursor to the scroll bar, press and hold down the left mouse button, and drag the cursor vertically downwards to scroll down the **Line PropertyManager**.*

20. Move the line cursor horizontally toward the left. Specify the endpoint when the length of the line above the line cursor shows a value close to 34.

21. Move the line cursor vertically downwards. Specify the endpoint when the length of the line above the line cursor shows a value close to 10.

22. Move the line cursor such that the line is drawn at an angle close to 346-degree. Specify the endpoint when the length of the line is around 27.

23. Move the line cursor vertically downwards. Specify the endpoint when the line cursor snaps the mirror line. Double-click anywhere in the drawing area to end line creation.

The sketch after drawing the required arcs, circles, and lines is shown in Figure 3-62.

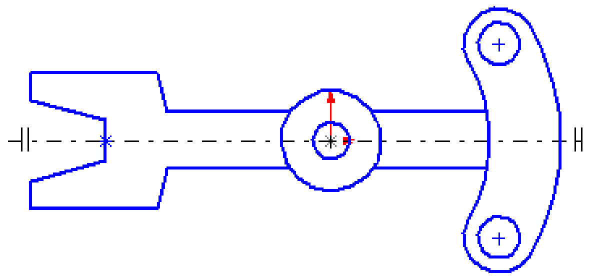
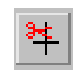


Figure 3-62 *Complete sketch*

Trimming the unwanted entities

After creating the sketch, you need to trim some of the unwanted sketched entities using the **Sketch Trim** tool.

1. Choose the **Sketch Trim** button from the **Sketch Tools** toolbar to invoke the trim tool.



The line cursor is replaced by the trim cursor.

2. Select the entities to be trimmed as shown in Figure 3-63. The entities are dynamically trimmed.

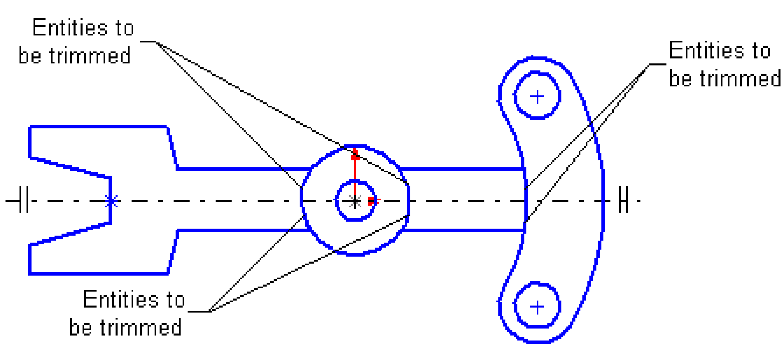
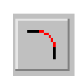


Figure 3-63 *Entities to be trimmed*

Adding the Fillets to the Sketch

Next, you need to add fillets to the sketch. The fillets are generally applied to avoid the stress concentration at sharp corners.

1. Choose the **Sketch Fillet** button from the **Sketch Tools** toolbar.



The **Sketch Fillet PropertyManager** is displayed and the trim cursor is replaced by the select cursor.

2. Set the radius spinner to **5**.

3. Using the left mouse button, select the entities shown in Figure 3-64 to apply the fillet.

4. Choose the **OK** button from the **Sketch Fillet PropertyManager** to exit the **Fillet** tool.

   The sketch after applying the fillets to the sketch is shown in Figure 3-65.

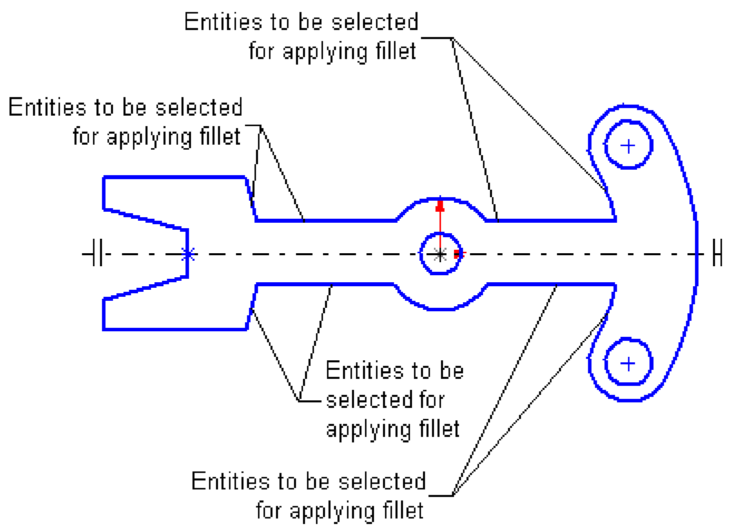


Figure 3-64 *Entities to be selected to apply fillet*

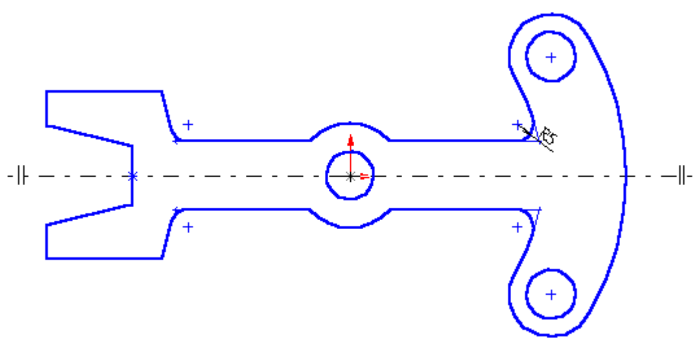
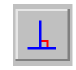


Figure 3-65 *Sketch after creating the fillets*

Adding the Relations to the Sketch

Next, you need to add the required relations to the sketch.



1. Choose the **Add Relations** button from the **Sketch Relations** toolbar to invoke the **Add Relations PropertyManager**.

2. Select the lower left arc and the lower tangent arc. The **Tangent** button is highlighted bold in the **Add Relations** rollout of the **Add Relations PropertyManager**. This suggests that the **Tangent** relation is the most appropriate relation for the selected entities.

3. Choose the **Tangent** button from the **Add Relations PropertyManager**.

4. Right-click in the drawing area and choose the **Clear Selections** option from the shortcut menu to clear the selections from the selection set. Select the right arc and the lower tangent arc. The **Tangent** button is highlighted.

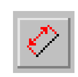
5. Choose the **Tangent** button from the **Add Relations PropertyManager**. Clear the current selections from the selection set.

6. Select the right horizontal line and the left horizontal line that are coincident with the trimmed circle. Choose the **Collinear** button from the **Add Relations PropertyManager**.

7. Choose the **OK** button from the **Add Relations PropertyManager** or choose the **OK** icon from the confirmation corner.

Adding the Dimensions to the Sketch

Next, you will apply the dimensions to the sketch and fully defined it.



1. Choose the **Dimension** button from the **Sketch** toolbar to invoke the dimension tool. The arrow cursor is replaced by the dimension cursor.

2. Select the right arc. A radius dimension is attached to the cursor. Move the cursor away from the sketch toward the right and place the dimension.

3. Enter a value of **82** in the **Modify** dialog box and press ENTER.

4. Select the left upper arc. A radius dimension is attached to the cursor. Move the cursor away from the sketch toward the right and place the dimension.

5. Enter a value of **56** in the **Modify** dialog box and press ENTER.

6. Select the **Origin** and the centerpoint of the upper circle. A dimension is attached to the cursor. Next, select the right endpoint of the mirror line. An angular dimension is attached to the cursor. Place the angular dimension outside the sketch.

7. Enter a value of **30** in the **Modify** dialog box and press ENTER.

8. Select the upper right circle and a diameter dimension is attached to the cursor. Place the dimension outside the sketch.

9. Enter a value of **15** in the **Modify** dialog box and press ENTER.

10. Select the upper right horizontal line that coincides the trimmed circle and the lower right horizontal line that coincides the trimmed circle. A linear dimension is attached to the cursor. Move the cursor vertically downwards and using the left mouse button place the dimension.

11. Enter a value of **20** in the **Modify** dialog box.

12. Select the smaller circle at the center. A diameter dimension is attached to the cursor. Move the cursor upwards and place the dimension outside the sketch.

13. Enter a value of **13** in the **Modify** dialog box.

14. Select the outer trimmed circle and place the radius dimension outside the sketch.

15. Enter a value of **19** in the **Modify** dialog box.

16. Select the upper left inclined line. A dimension is attached to the cursor. Now, select the upper left horizontal line. An angular dimension is attached to the cursor. Place the dimension above the horizontal line.

17. Enter a value of **75** in the **Modify** dialog box.

18. Select the origin and the lower endpoint of the lower left inclined line. Move the cursor vertically downwards and place the dimension.

19. Enter a value of **49** in the **Modify** dialog box.

20. Select the origin and the middle left vertical line. Move the cursor vertically downwards and place the dimension below the last dimension.

21. Enter a value of **61** in the **Modify** dialog box.

22. Select the origin and select the lower endpoint of the outer left vertical line. Move the cursor vertically downwards and place the dimension below the last dimension.

23. Enter a value of **90** in the **Modify** dialog box.

24. Select the upper left inclined line, and the lower left inclined line, refer to Figure 3-66. An angular dimension is attached to the cursor. Move the cursor horizontally toward the left and place the dimension.

25. Enter a value of **28** in the **Modify** dialog box.

26. Select the upper left horizontal line and the lower left horizontal line. A linear dimension is attached to the cursor. Move the cursor horizontally toward the left and place the dimension.

27. Enter a value of **50** in the **Modify** dialog box.

28. Select the lower endpoint of the upper left vertical line and the upper endpoint of the lower left vertical line. A linear dimension is attached to the cursor. Move the cursor horizontally toward the left and using the left mouse button place the dimension.

29. Enter a value of **30** in the **Modify** dialog box.

    All the entities are displayed in black, suggesting that the sketch is fully defined. The fully defined sketch is shown in Figure 3-66.

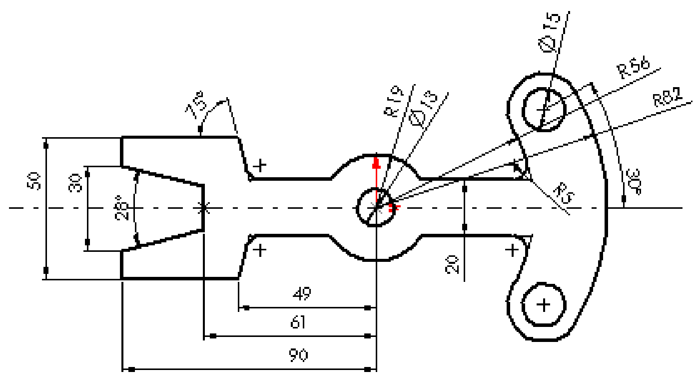
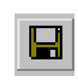


Figure 3-66 *Sketch after applying all relations and dimensions*

Saving the Sketch

Next, you need to save the document in the *\My Documents\SolidWorks\c03* directory.

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.



2. Enter the name of the document as *c03-tut02.sldprt* in the **File name** edit box and choose the **Save** button.

   The document will be saved.

3. Close the file by choosing **File** > **Close** from the menu bar.

Tutorial 3

In this tutorial you will draw the sketch of a revolved model shown in Figure 3-67. The sketch is shown in Figure 3-68. The solid model is given only for your reference.

(Expected time: 30 min)

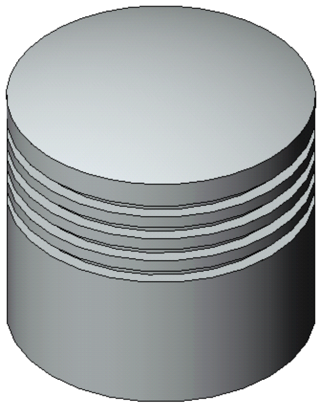


Figure 3-67 Solid model of the piston

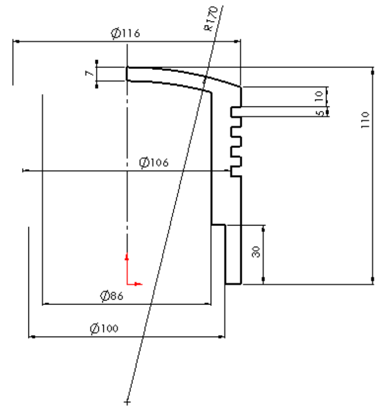


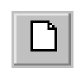
Figure 3-68 The sketch of the base feature

The steps that will be followed to complete this tutorial are listed below:

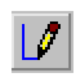
a. Open a new part file and then switch to the sketching environment.
b. Create a center line that will act as the axis of revolution when you will create the base feature using this sketch.
c. Create the sketch using the various sketching tools.
d. Use the offset tool to offset the required lines, refer to Figure 3-69.
e. Draw the arcs and trim the unwanted entities, refer Figure 3-70.
f. Add the required relations.
g. Add the required dimensions and fully define the sketch, refer Figure 3-71.

Opening a New Part File

1. Choose the **New** button from the **Standard** toolbar and open a new part file using the **New SolidWorks Document** dialog box.



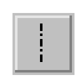
2. Choose the **Sketch** button from the **Sketch** toolbar to switch to the sketching environment for drawing the sketch.



Creating a Center line

You need to create a center line around which the sketch of the base feature will be revolved.

1. Choose the **Centerline** button from the **Sketch Tools** toolbar.



2. Move the cursor to the origin and specify the startpoint when the line cursor turns yellow. Move the cursor vertically upwards. Specify the endpoint when the length of the line cursor shows a value close to 120.

Drawing the Sketch

Next, you will draw the sketch of the piston.

1. Right-click in the drawing area and choose the **Line** option from the shortcut menu. Move the cursor to a location whose coordinates are close to 58mm 0mm 0mm.

2. Draw a vertical line of dimension close to 100. Choose the **End chain** option from the shortcut menu.

3. Move the cursor to the lower endpoint of the line created earlier. Specify the startpoint of the line when the cursor turns yellow. Move the cursor horizontally toward the left and create a horizontal line of dimension close to 8.

4. Move the line cursor vertically upwards and create a vertical line of dimension close to 30.

5. Move the cursor horizontally toward the left and create a horizontal line of dimension close to 7.

6. Move the line cursor vertically upwards and create a vertical line of dimension close to 70.

7. Right-click and choose the **3 Point Arc** option from the shortcut menu. Move the cursor near the upper endpoint of the right vertical line.

8. Specify the first point of the arc when the cursor changes to yellow color. Move the cursor horizontally toward the left. The reference arc is attached to the cursor. Specify the second point of the arc when the value of the length is close to 116.

9. Move the cursor vertically upwards. Specify the third point of the arc when the value of the radius is close to 170.

10. Right-click and choose the **Line** option from the shortcut menu. Move the line cursor to a location whose coordinates are close to 58mm 90mm 0mm. Specify the startpoint of the line at this location and move the cursor horizontally toward the left to create a line of dimension close to 5. Double-click anywhere in the drawing area.

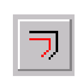
11. Choose the **Zoom To Fit** button from the **View** toolbar to fit the sketch to the drawing area.

Offsetting the lines

Using the **Offset Entities** tool you will offset the entities created earlier.

1. Select the line created previously using the select tool.

2. Choose the **Offset Entities** button from the **Sketch Tools** toolbar to invoke the **Offset Entities PropertyManager**. The confirmation corner is also displayed at the upper right corner of the drawing area.



3. Choose the **Keep Visible** button from the **Offset Entities PropertyManager** to pin the **PropertyManager**.

4. Set the value of **Offset Distance** spinner to **5**. Now, select the **Reverse** check box because you need to offset the entity in the reverse direction.

   As soon as you select the **Reserve** check box you will observe that the preview of the entity to be offset is modified in the drawing area.

5. Choose the **OK** button from the **Offset Entities PropertyManager**. You will notice that an entity is created at an offset distance of 5 from the original entity.

   You will notice that a dimension is also attached to the newly created entity and the original entity with a value of **5**. This dimension is the offset distance between the two entities.

6. Select the newly created entity. Move the cursor vertically downwards. The preview of the entity and the direction of offset creation is also displayed. Press the left mouse button to offset the selected line.

   Repeat this procedure of offsetting the entities unless you get the eight entities including the original entity.

7. Set the value of **Offset Distance** spinner to **7** and clear the **Select chain** check box. Select the upper arc. The preview of the offset arc is shown in the background.

8. Choose the **OK** button twice on the **Offset Entities PropertyManager**.

Completing the Remaining Sketch

Next, you will complete the remaining sketch using the line tool.

1. Right-click and choose the **Line** option from the shortcut menu. Move the line cursor to the left endpoint of the upper smaller right horizontal line. When the cursor turns yellow in color, create a vertical line that snaps the endpoint of the last offseted line. Double-click to end line creation.

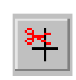
2. Move the cursor to the intersection point of the upper arc and the centerline. When the cursor snaps the intersection, create a vertical line that snaps the intersection point of the lower arc and the centerline.

   The sketch after creating the entities using the various sketch tools and the offset tool is shown in Figure 3-69.

Trimming the Unwanted Entities

Next, you will trim the unwanted entities using the trim tool.

1. Choose the **Trim** button from the **Sketch Tools** toolbar.



2. Using the left mouse button trim the unwanted entities. The sketch after trimming the unwanted entities is shown in Figure 3-70.

Adding the Required Relations

Now, you will add the required relations to the sketched entities.

1. Right-click and choose the **Select** option from the shortcut menu. Press and hold down the CTRL key from the keyboard and select one of the endpoints of the lower horizontal line

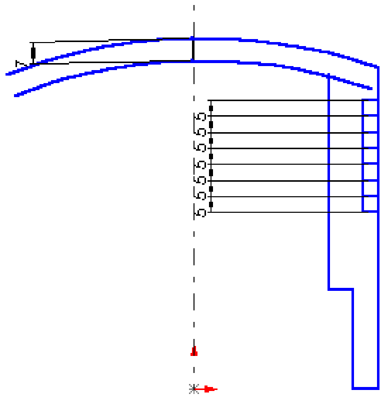


Figure 3-69 *Sketch after creating various entities*

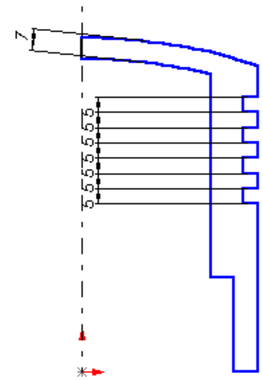


Figure 3-70 *Sketch after trimming the unwanted entities*

and then select the origin. Release the CTRL key after selection. Right-click and choose the **Make Horizontal** option from the shortcut menu to add the horizontal relation to the selected entities. Click anywhere in the drawing area to clear the selection set.

2. Press and hold down the CTRL key and select the small horizontal lines in the right of the sketch. Release the CTRL key after making the selection and right-click to display the shortcut menu. Move the cursor to two down arrows displayed at the end of the shortcut menu. Keep the cursor on this location for a couple of seconds to expand the shortcut menu. Choose the **Make Equal** option from the shortcut menu to apply the equal relation.

3. Similarly, apply the **Equal** relation to all the small vertical lines.

   The **SolidWorks** warning message window will be displayed. This warns you that the solution cannot be determined for the sketch. You will notice that some the entities of the sketch turn red. This indicates that the sketch is over defined.

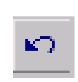
4. Choose **OK** from this dialog box.

   Another SolidWorks dialog box is displayed. This warns you that applying this relation will result in over defining the sketch.

5. Choose **OK** from this dialog box.

   As discussed earlier, the overdefined sketch is not used to create any feature; therefore, you have to delete the conflicting relations. The sketch is over defined after applying the previous relation. Therefore, if you delete the last applied relation using the **Undo** button from the **Standard** toolbar then the sketch will not be over defined.

6. Choose the **Undo** button from the **Standard** toolbar. Click anywhere in the drawing area to clear the selections from the selection set.



As evident from the color of the entities, which is green, you still have to add some more relations or dimensions to fully define the sketch. The fully defined sketch is displayed in black color.

7. Zoom in using the **Zoom In/Out** tool and apply the coincident relation between the centerpoint of the upper arc and the centerline.

Adding the Dimensions to the Sketch

After creating, editing, and applying the relations to the sketch, you will add the required dimensions to the sketch to fully define the sketch.

1. Select the dimension with a value of **7** that is placed between the upper arcs and press the DELETE key from the keyboard.

   The SolidWorks dialog box is displayed. This warns you that if you delete the offset dimension, the offset relation will also be deleted from the sketch.

2. Choose the **Yes** button from this dialog box.

   You have to delete this dimension because during the design and manufacturing practices the dimension between the tangents should be avoided.

3. Right-click and choose the **Dimension** option from the shortcut menu to invoke the **Dimension** tool.

4. Select the outer upper right vertical line. A dimension is attached to the cursor. Now, select the centerline and move the cursor to the other side of the centerline. You will notice that the diameter dimension is displayed along the cursor.

5. Using the left mouse button place the dimension above the sketch and enter a value of **116** in the **Modify** edit box and press ENTER.

6. Now, select the inner left vertical line and then select the centerline. Move the cursor to the other side of the centerline and place the dimension using the left mouse button below the sketch.

7. Enter a value of **86** in the **Modify** edit box and press ENTER.

   Add the remaining dimensions to fully defined the sketch. Refer to Figure 3-71. Using the left mouse button select one of the dimensions that are created after offsetting the entities and drag the cursor toward the right and using the left mouse button place the dimension at an appropriate place. Arrange all the dimensions using the above method. The fully defined sketch is shown in Figure 3-71.

Saving the Sketch

Since the document has not been saved once until now, therefore, when you choose the **Save**

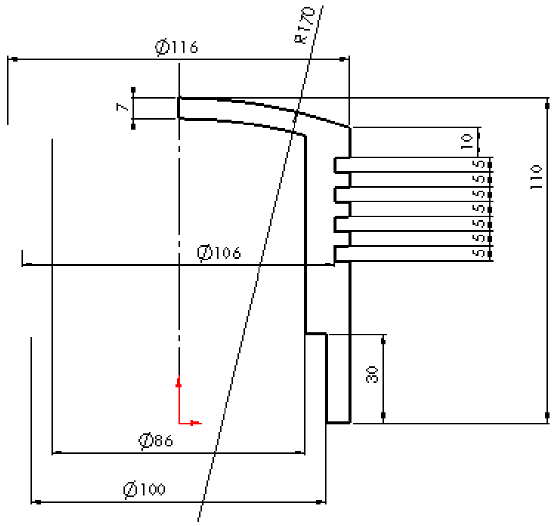
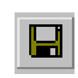


Figure 3-71 *Fully defined sketch after applying all the relations and dimensions*

button from the **Standard** toolbar, the **Save As** dialog box will be displayed. You can enter the name of the document in this dialog box.



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

   \My Documents\SolidWorks\c03\c03-tut02.SLDPRT.

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

SELF-EVALUATION TEST

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

1. The **Trim** option is also used to extend the sketched entities. (T/F)

2. In the sketching environment you can apply fillets to two parallel lines. (T/F)

3. You can apply a fillet to two nonintersecting entities. (T/F)

4. You cannot offset a single entity; you have to select a chain of entities to create an entity using the offset tool. (T/F)

5. You can do modifications in the view-only file. (T/F)

6. The __________ **PropertyManager** is displayed using the **Add Relation** button from the **Sketch Relations** toolbar.

7. The __________ dimension is used to dimension a line that is at an angle with respect to the X axis or the Y axis.

8. The __________ defined sketch is a sketch in which all the entities of the sketch and their positions are described by the relations or dimensions, or both.

9. The __________ dimensions or relations are not able to determine the position of one or more sketched entities.

10. The __________ option is displayed in the **Defined In** column when the entity is defined as placed in the same sketch.

REVIEW QUESTIONS

Answer the following questions:

1. You can invoke the **Sketch Relations PropertyManager** using the **Display/Delete Relations** button from __________ toolbar.

2. The **Document Properties - Grid/Snap** dialog box is displayed using the __________ button from the **Sketch** toolbar.

3. The __________ option is used to display all the relations of the sketch in the **Sketch Relations PropertyManager**.

4. The __________ sketch geometry is constrained by too many dimensions and/or relations. Therefore, you have to delete the extra and conflicting relations or dimensions.

5. The __________ relation forces two selected lines, arcs, points, or ellipses to remain equidistant from a centerline.

6. In SolidWorks, by default, the dimensioning between two arcs, two circle, or between an arc and a circle is done from

   (a) Centerpoint to centerpoint
   (b) Centerpoint to tangent
   (c) Tangent to tangent
   (d) None

7. Which relation forces the selected arc to share the same centerpoint with another arc or a point?

   (a) **Concentric**
   (b) **Coradial**
   (c) **Merge points**
   (d) **Equal**

8. Which **PropertyManager** is displayed when you choose the **Fillet** button from the **Sketch Tools** toolbar?

   (a) **Sketch Fillet** (b) **Fillet**
   (c) **Surface Fillet** (d) **Sketching Fillet**

9. Which dialog box is displayed when you modify a dimension?

   (a) **Modify Dimensional Value** (b) **Insert a value**
   (c) **Modify** (d) None

10. When you add an extra dimension to a sketch or add an extra relation that over defines the sketch, then which dialog box is displayed?

    (a) **Over defining** (b) **Delete relation**
    (c) **Make Dimension Driven?** (c) **Add Geometric Relations**

EXERCISES

Exercise 1

Create the sketch of the model shown in Figure 3-72. Create the sketch and apply the required relations and dimensions and fully define the sketch. The sketch is shown in Figure 3-73. The solid model is given only for reference. (**Expected time: 30 min**)

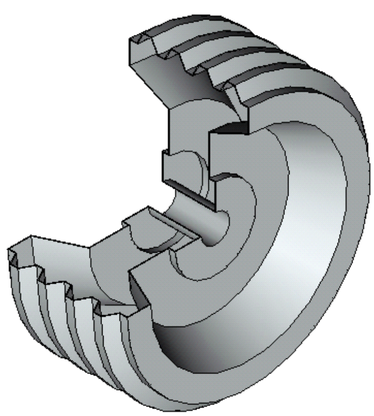


Figure 3-72 *Solid model for Exercise 1*

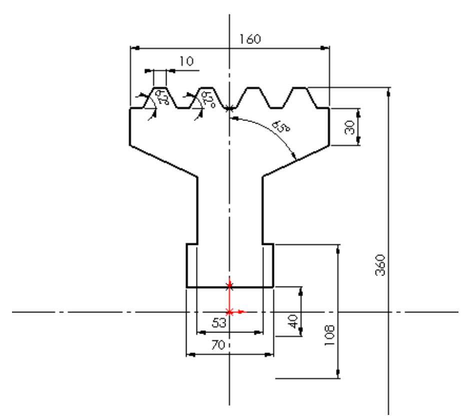


Figure 3-73 *Sketch for Exercise 1*

Exercise 2

Create the sketch of the model shown in Figure 3-74. Create the sketch and apply the required relations and dimensions and fully define the sketch. The sketch is shown in Figure 3-75. The solid model is given only for reference. (**Expected time: 30 min**)

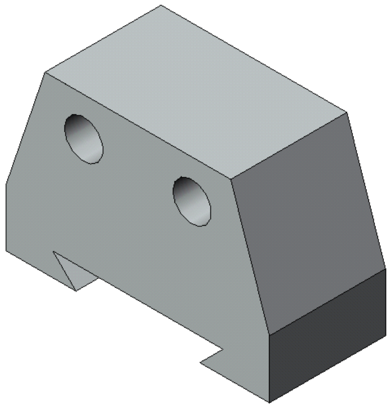


Figure 3-74 Solid model for Exercise 2

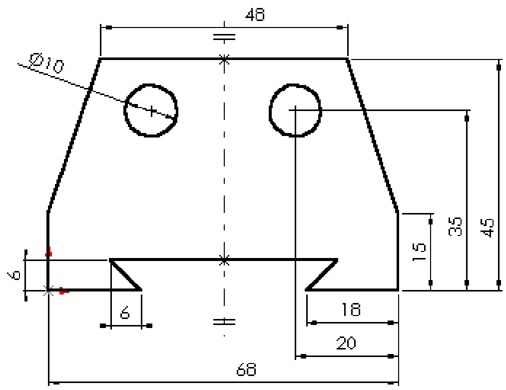


Figure 3-75 Sketch for Exercise 2

Exercise 3

Create the sketch of the model shown in Figure 3-76. Create the sketch and apply the required relations and dimensions and fully define the sketch. The sketch is shown in Figure 3-77. The solid model is given only for reference. (**Expected time: 30 min**)

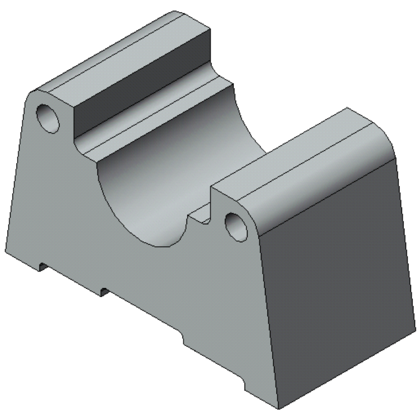


Figure 3-76 Solid model for Exercise 3

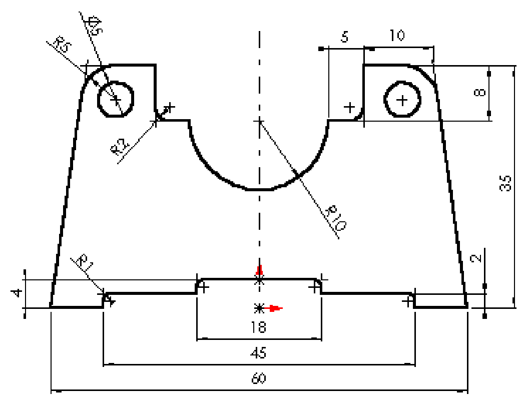


Figure 3-77 Sketch for Exercise 3

Answers to Self-Evaluation Test

1. T, **2.** F, **3.** T, **4.** F, **5.** T, **6. Add Relations**, **7.** aligned, **8.** fully defined, **9. Dangling**, **10. All in this sketch**