Chapter *13*

Working With Drawing Views-II

Learning Objectives

After completing this chapter you will be able to:

- *Generate Annotations in the Drawing Views.*
- *Create Reference Dimensions.*
- *Add Notes to the Drawing Views.*
- *Add Surface Finish Symbols to the Drawing Views.*
- *Add Datum Feature Symbols to the Drawing Views.*
- *Add Geometric Tolerance to the Drawing Views.*
- *Add Datum Target Symbols to the Drawing Views.*
- *Add Center Marks and Center Lines to the Drawing Views.*
- *Add Hole Callouts to the Drawing Views.*
- *Add Cosmetic Threads to the Drawing Views.*
- *Add Weld Symbols to the Drawing Views.*
- *Add Multi-jog Leader to the Drawing Views.*
- *Add Dowel Pin Symbol to the Drawing Views.*
- *Edit the Annotations.*
- *Add BOM and Balloons to the Assembly.*
- *Add New Sheets in the Drawing Document.*
- *Edit Sheet Format.*
- *Create User-Defined Sheet Format.*

ADDING ANNOTATIONS TO THE DRAWING VIEWS

After generating the drawing views, you need to generate the dimensions in the drawing views and add the other annotations such as notes, surface finish symbols, geometric tolerance, and so on. There are two types of annotations that can be displayed in the drawing views. The first type of annotation is the generative annotation. These annotations are added while creating the part in the part mode. For example, the dimensions that you add to the sketch and features of the part. The second type of annotations are added manually to the geometry of drawing views such as reference dimensions, notes, surface finish symbols, and so on. Both these types of annotations are discussed next.

Generating Annotations Using the Model Items Tool

Toolbar:	Annotation > Model Items
Menu:	Insert > Model Items

The **Model Items** tool is used to generate the annotations that were added while creating the model in the part mode. To invoke this tool, choose the **Model Items** button from the **Drawing** toolbar or choose **Insert > Model Items** from the menu bar. The **Insert Model Items** dialog box is displayed as shown in Figure 13-1. The various options available in this dialog box are discussed next.

Annotations

The **Annotations** area of this dialog box is used to select the annotations that you need to generate in the drawing views. Using the options available in the **Annotation** area you can generate the cosmetic thread, datums, datum targets, dimensions, geometric tolerances, notes, surface finish, and weld symbols. Select the annotation to be generated from this area.

Reference geometry

The **Reference geometry** area is used to generate the reference geometries that were used in creating the model. You can generate axes, curves, planes, surfaces, and so on. Select the check box of the reference geometry that you need to generate in the drawing views.

Import from

The options available in the **Import from** area are used to define the options from where the annotation should be imported. The options available in this area are discussed next.

Entire model

The **Entire model** radio button is selected to import the annotations from the entire model when a feature of a model is selected from the **FeatureManager Design Tree** flyout in the **Drawing** mode. In case of assemblies, the annotations from the entire assembly are imported even if you select a single component of the assembly.

Selected component

The **Selected component** radio button is active if you select a component of the assembly from the **FeatureManager Design Tree**. This radio button is selected to import the annotations from only the selected component.
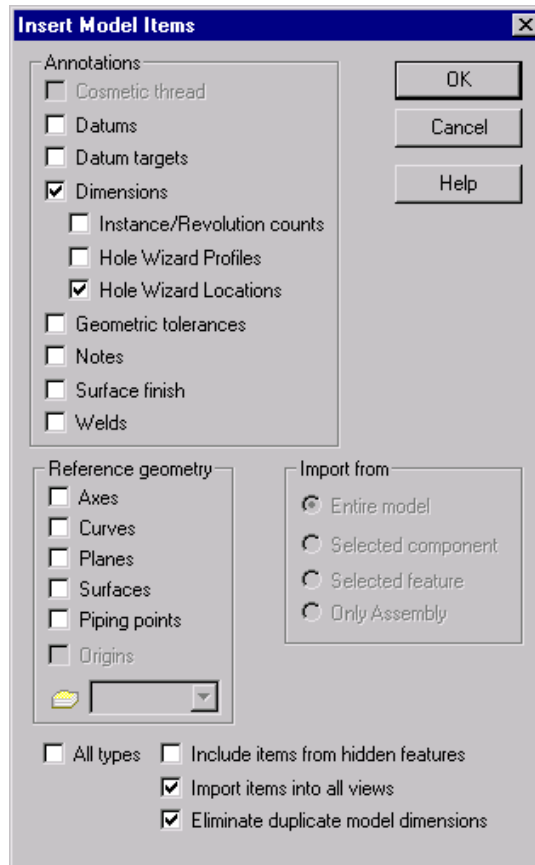
Figure 13-1 *The Insert Model Items dialog box*

Selected feature
The **Selected feature** radio button is active if you select a feature of the component from the **FeatureManager Design Tree**. This radio button is selected to import the annotations from only the selected feature or features.

Only Assembly
The **Only Assembly** radio button is active when you generate the drawing views of an assembly. This radio button is selected to import the annotations that are applied to the assembly in the **Assembly** mode such as offset distance and so on.

All types
The **All types** check box is selected if you need to display all the annotations applied to the part or assembly.

Include items from hidden features
The **Include items from hidden features** check box is selected to display the annotation that belongs to a hidden feature of the model. By default, this check box is cleared. It is

recommended that you clear this check box because it will eliminate the display of unwanted annotations.

Import items into all views

The **Import items into all views** check box is selected to display the annotation in all the views. The annotations are automatically distributed in all the drawing views. If you need to display the annotations in a selected view or views only, clear this check box. When you clear this check box, the **Import to views** display area is displayed below the **Help** button. You can select the views in which you need to display the annotations. You can also preselect the views in which you need to display the annotations before invoking the **Insert Model Items** dialog box.

Eliminate duplicate model dimensions

The **Eliminate duplicate model dimensions** check box is selected to prevent the display of the same annotation more than once.

After setting all the parameters in the **Insert Model Items** dialog box, choose the **OK** button to display the annotations.

Tip. *When you generate the annotations, mostly the annotations overlaps one over the other. Therefore, you may need to move the annotations after generating. To move an annotation, move the cursor on the annotation; a red rectangle is displayed. Press and hold down the left mouse button and drag the cursor to place the annotation at the desired location. Release the left mouse button when the cursor is placed at the desired location.*

*The dimensions generated while generating the annotations are parametric dimensions and these dimensions are the link of bidirectional associativity in SolidWorks. To modify the dimensions, double-click the dimension; the **Modify** dialog box is displayed. Using this dialog box you can modify the value of dimension. After modifying the dimensions choose the **Rebuild** button from the **Standard** tool bar or choose CTRL+B from the keyboard.*

*The modification implemented in the **Drawing** mode will also be reflected in the **Part** mode and the **Assembly** mode.*

*If you need to delete a particular generated annotation, select the annotation from the drawing sheet and press the DELETE key from the keyboard. The annotation deleted from the drawing view will not be deleted from the part in the **Part** mode.*

Creating the Reference Annotations

In SolidWorks, you can create the reference annotations in the drawing views. The reference annotations include reference dimensions, notes, surface finish symbols, datum feature symbol, geometric tolerance, and so on. The method of creating the reference annotations is discussed next.

Creating Reference Dimensions

Toolbar:	Sketch Relations > Dimension
Menu:	Tools > Dimensions

In the **Drawing** mode of SolidWorks, you can use the **Dimension** tool to add reference dimensions to the drawing views. This tool is similar to the **Dimensions** tool discussed in the sketcher environment of SolidWorks. The additional dimensioning option is provided in the **Drawing** mode of SolidWorks is discussed next.

Chamfer Dimension

The **Chamfer Dimension** option is used to add the reference dimension to the chamfers available in the drawing view. To add a chamfer dimension choose **Insert > Dimensions > Chamfer Dimension** from the menu bar. You can also invoke this tool by choosing the **Dimension** button from the **Sketch Relations** toolbar and then choosing the **Chamfer Dimension** option by invoking the shortcut menu. The cursor is replaced by the chamfer dimension cursor. Now, select the inclined chamfered edge and then select a horizontal or vertical edge. The chamfer dimension will be attached to the cursor. Select a point on the sheet to place the dimension. Figure 13-2 shows a chamfer dimension created using the **Chamfer Dimension** option.
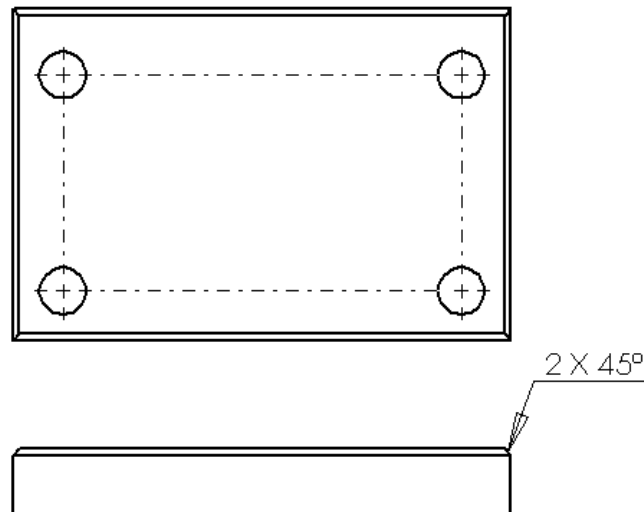
*Figure 13-2   Chamfer dimension created using the **Chamfer Dimension** tool*

Adding Notes to the Drawing Views

Toolbar:	Annotation > Note
Menu:	Insert > Annotations > Note

In SolidWorks, you can also add notes to the drawing views in the **Drawing** mode. To add notes to the drawing view, choose the **Note** button from the **Annotation** toolbar

or choose **Insert > Annotations > Note** from the menu bar. The **Note PropertyManager** is displayed as shown in Figure 13-3.
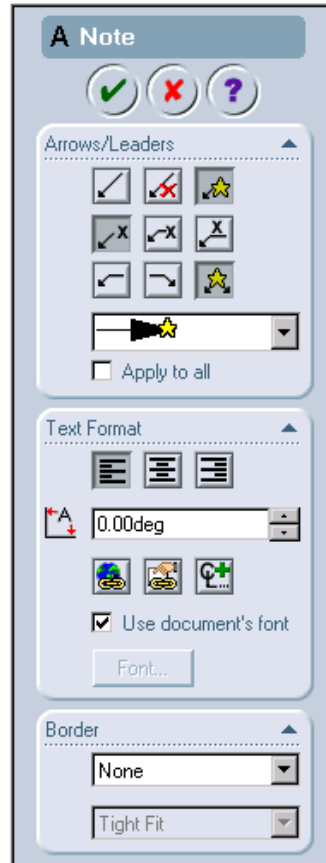

*Figure 13-3 The **Note PropertyManager***

When you invoke the **Note PropertyManager**, a text box is attached to the cursor. When you move the cursor close to an existing entity in the drawing sheet, a leader will be displayed attached to the text box. This is because the **Auto Leader** button is chosen in the **Arrows/ Leaders** rollout. Place the text box at the desired location. As you place the text box, the text edit box will be invoked and you can enter the text. After entering the text, choose **OK** from the **Note PropertyManager**. The other options available in the **Note PropertyManager** are discussed next.

Arrows/Leaders
The options available in the **Arrows/Leaders** rollout are used to define the style of arrows and leaders that are displayed in the notes.

Text Format
The **Text Format** rollout is used to set the format of text such as font, size, justification,

and rotation of the text. You can also add the symbols and hyperlink to the text using the options available in this rollout.

Border

The options available in the **Border** rollout are used to define the border in which the note text will be displayed. You can use various types of borders from the **Style** drop-down list. The **Size** drop-down list available in this rollout is used to define the size of the border in which the text will be placed.

Adding Surface Finish Symbols to the Drawing Views

Toolbar:	Annotation > Surface Finish
Menu:	Insert > Annotations > Surface Finish

You can add the surface finish symbols to the drawing views using the **Surface Finish** tool. To add the surface finish symbols, choose the **Surface Finish** button from the **Annotation** toolbar or choose **Insert > Annotations > Surface Finish** from the menu bar. The **Properties** dialog box is displayed as shown in Figure 13-4.
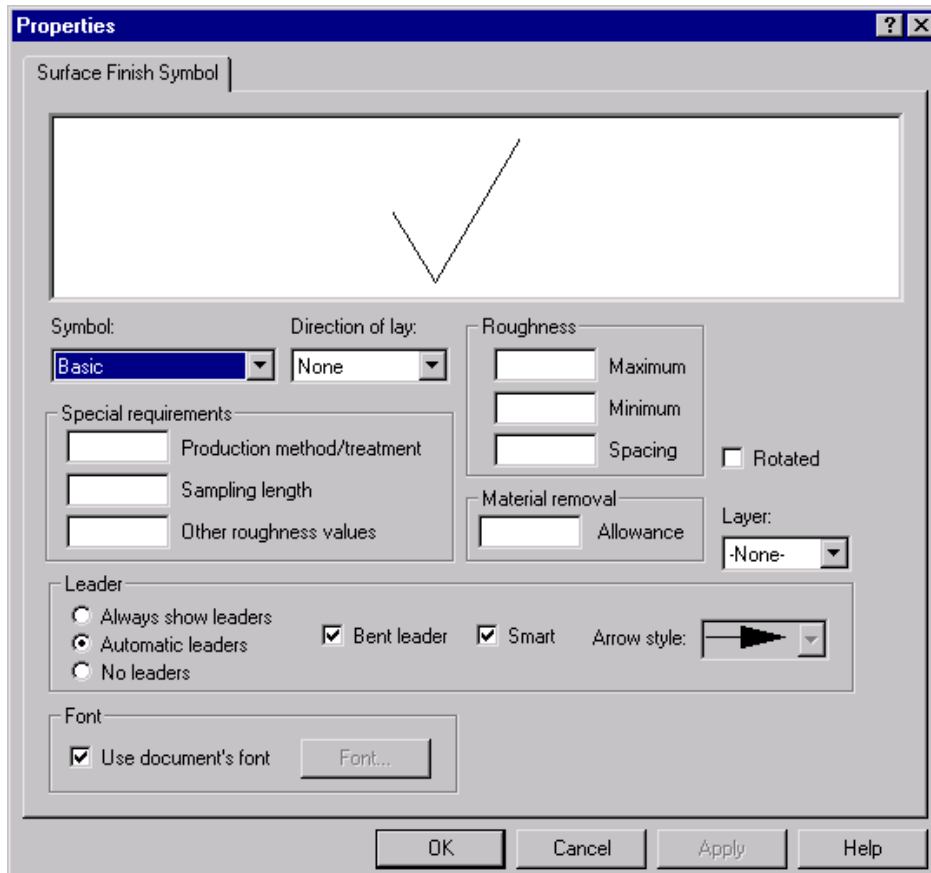

Figure 13-4 *The* **Properties** *dialog box*

As you invoke this tool, a surface finish symbol is attached to the cursor. The options available in the **Properties** dialog box are discussed next.

Preview Area
The preview area is used to preview the surface finish symbol with the current parameters. As you modify the parameters of the surface finish symbol, the preview updates dynamically in the preview area.

Symbol
The **Symbol** drop-down list is used to define the type of surface finish symbols that you can add to the drawing views. The surface finish symbols include the **Basic** surface finish symbol, **Machining Required**, **Machining Prohibited**, **JIS Surface Texture 1**, **JIS Surface Texture 2**, **JIS Surface Texture 3**, **JIS Surface Texture 4**, and **JIS No Machining**.

Direction of lay
The **Direction of lay** drop-down list is used to define the direction of lay in the surface finish symbol. This drop-down list is available when only the **Basic**, **Machining Required**, or **Machining Prohibited** option is selected from the **Symbol** drop-down list.

Roughness
The **Roughness** area available in the **Properties** dialog box is used to define the surface roughness values to be displayed in the surface finish symbol.

Rotated
The **Rotated** check box is selected to rotate the surface finish symbol at an angle of 90º.

Special requirements
The **Special requirements** area is used to enter the information such as production method/ treatment, sampling length, and some other roughness values, if there is any.

Material removal
The **Material removal** area is used to define the material removal allowance to be displayed in the surface finish symbol.

Leader
The **Leader** area is used to define the style and properties of the leader and the arrow that will be attached to the surface finish symbol.

Font
The **Font** area is used to define the font and the font size to be used in the text added in the surface finish symbol.

Adding Datum Feature Symbol to the Drawing Views

Toolbar: Annotation > Datum Feature Symbol
Menu: Insert > Annotations > Datum Feature Symbol

The **Datum Feature Symbol** tool is used to add the datum feature symbol to the drawing views. The datum feature symbols are used as datum references while adding the geometric tolerances in the drawing view. To add the datum feature symbol, choose the **Datum Feature Symbol** button from the **Annotation** toolbar or choose **Insert > Annotations > Datum Feature Symbol** from the menu bar. The **Properties** dialog box is displayed and a datum feature symbol with default parameters is attached to the cursor. The **Properties** dialog box used to add the **Datum Feature Symbol** is displayed in Figure 13-5.
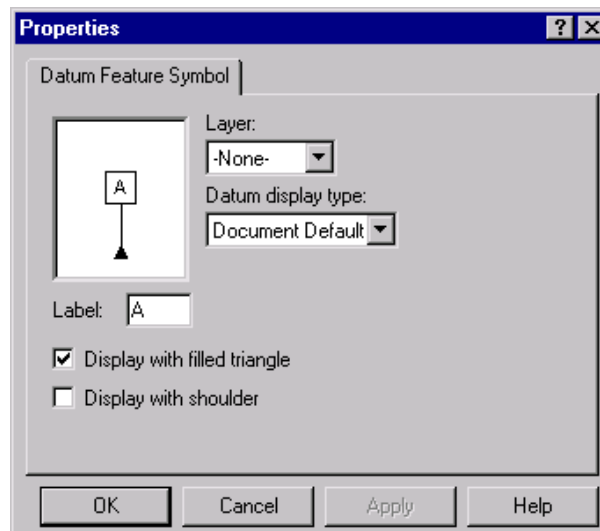
Figure 13-5 *The **Properties** dialog box used to add datum feature symbol to the drawing views*

The options available in the **Properties** dialog box to add the datum feature symbol are discussed next.

Preview Area
The preview area available in this dialog box is used to preview the datum feature symbol. By default, the datum symbol feature is displayed in the preview area with the default parameters. The preview of the datum feature symbol updates dynamically as you modify the parameters of the datum feature symbol.

Datum display type
The **Datum display type** drop-down list is used to define the border that will be displayed around the label of the datum feature symbol. The border of the datum feature symbol is displayed as a square by default. You can also display the border of the datum feature symbol as a circle using this drop-down list.

Label
The **Label** edit box is used to define the label to be used in the datum feature symbol.

Display with filled triangle
The **Display with filled triangle** check box is selected to display a filled triangle at the bottom of the datum feature symbol.

Display with shoulder
The **Display with shoulder** check box is selected to display the datum feature symbol with a shoulder created at 90º.

After defining all the parameters of the datum feature symbol, specify a point on an existing entity in the drawing sheet and then move the cursor to define the length and the placement of the datum feature symbol. As you place one datum feature symbol, another datum feature symbol is attached to the cursor. Therefore, you can place as many datum feature symbols as you want using the **Properties** dialog box. The sequence of the names of datum feature symbols automatically follows the alphabetical order as you place multiple datum feature symbols.

Adding Geometric Tolerance to the Drawing Views

Toolbar:	Annotation > Geometric Tolerance
Menu:	Insert > Annotations > Geometric Tolerance

In a shop floor drawing, you need to provide various other parameters along with the dimensions and dimensional tolerance. These parameters can be geometric condition, surface profile, material condition, and so on. All these parameters are defined using the geometric tolerance. To add the geometric tolerance to the drawing views, choose the **Geometric Tolerance** button from the **Annotation** toolbar or choose **Insert** > **Annotations** > **Geometric Tolerance** from the menu bar. The **Properties** dialog box will be displayed. It will be used to define the parameters of the geometric tolerance. When you invoke this tool, a geometric tolerance is attached to the cursor. The **Properties** dialog box is shown in Figure 13-6.

Various options available in the **Properties** dialog box to add the geometric tolerance to the drawing views are discussed next.

Feature control frames
The **Feature control frames** area is used to define the features of geometric tolerance to be added in the frames. The features of the geometric tolerance that can be added in the frames are geometric condition symbols, diameter symbol, value of tolerance, material condition, and datum references. The options available in the **Feature control frames** area are discussed next.

GCS

The **GCS** button available in the **Feature control frames** area is used to define the geometric condition symbol. When you choose this button, the **Symbols** dialog box is displayed, which is used to define the geometric condition symbols in the geometric tolerance. The **Symbols** dialog box is shown in
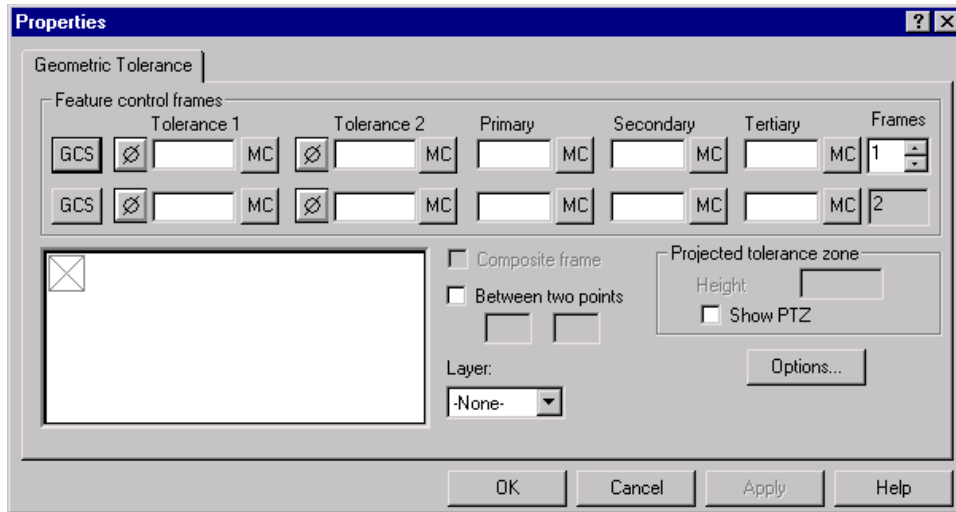
Figure 13-6 The **Properties** *dialog box used to apply geometric tolerance*

Figure 13-7. You can specify the standard of the geometric condition symbol from the **Symbol library** drop-down list. After selecting the appropriate geometric condition symbol, choose the **OK** button from this dialog box. The preview of the geometric tolerance is displayed in the preview area.
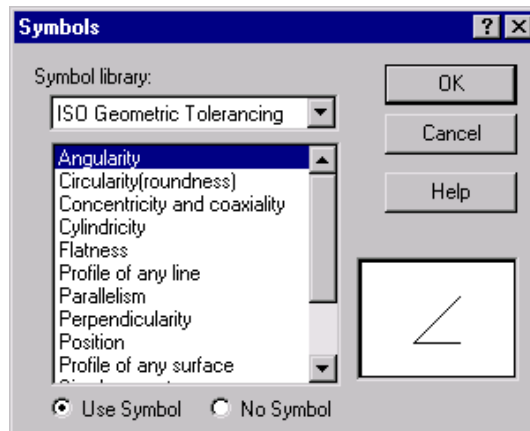

Figure 13-7 The **Symbols** *dialog box used to define the geometric condition symbols*

Tolerance 1

The **Tolerance 1** edit box is used to specify the tolerance value with respect to the geometric condition defined using the **GCS** button. Using the **Diameter** button, you can add a diameter symbol with the tolerance value. Using the **MC** button available next to the **Tolerance 1** edit box, you can specify the material condition in the geometric tolerance. When you choose the **MC** button, the **Symbols** dialog box is displayed, see Figure 13-8. This dialog box is used to define the symbol for the material condition.
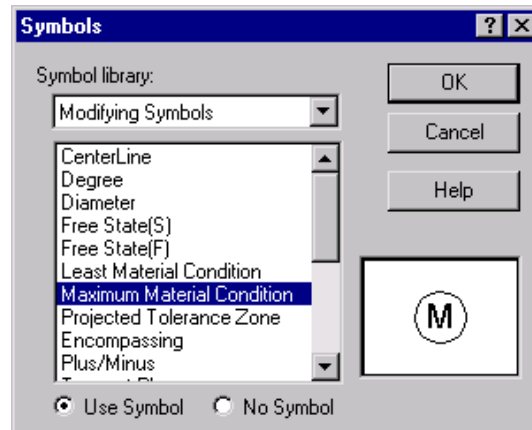
*Figure 13-8 The **Symbols** dialog box used to define
the symbol for material condition*

Tolerance 2
The options available in the **Tolerance 2** edit box are the same as those discussed
earlier. This edit box is used to define the second geometric tolerance, if required.

Primary
The **Primary** edit box is used to specify the alphabets to define the datum reference
added to the drawing view using the **Datum Feature Symbol** tool. Similarly, you can
define the **Secondary** datum reference and the **Tertiary** datum reference.

Frames
The **Frames** spinner is used to increase the frames for applying more complex
geometric tolerances.

Projected tolerance zone
The **Projected tolerance zone** area is used to define the height of the projected tolerance.
Select the **Show PTZ** check box from the **Projected tolerance zone** area; the **Projected
tolerance zone** edit box will be displayed. Specify the zone height in this edit box.

Options
The **Options** button is used to define the settings of the leader, arrow, font, and so on.
When you choose this button, the **Geometric Tolerance Options** dialog box is displayed.
Using this dialog box you can set the parameters for leader style, arrow style, font, and so
on.

Between two points
The **Between two points** check box is selected if the geometric tolerance to be applied is
between two points of entities. To apply the geometric tolerance between two points or
entities select the **Between two points** check box and specify the tolerance in the edit
boxes provided in the **Between two points** area.

Figure 13-9 shows a drawing after adding annotations to the drawing.
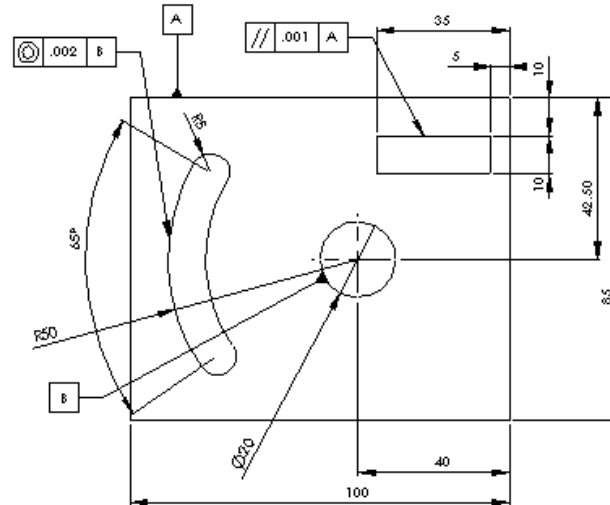

Figure 13-9 *A drawing after adding some annotations*

Adding Datum Target Symbols to the Drawing Views

Toolbar:	Annotation > Datum Target	(*Customize to Add*)
Menu:	Insert > Annotations > Datum Target	

The **Datum Target** tool is used to add a datum target to the drawing views. To add the datum targets to the drawings, choose the **Datum Target** button from the **Annotations** toolbar or choose **Insert > Annotations > Datum Target** from the menu bar. The cursor is replaced by the datum target cursor. Select a model face, an edge, or a line from the view on which you need to add the datum target. The **Properties** dialog box is displayed as shown in Figure 13-10. This dialog box is used to define the properties of the datum target symbol. Set the parameters of the datum target symbol in the **Target area** of this dialog box. You can set the target shape as a point, circle, or rectangle; you can also define the diameter of the target area if the shapes of the target are selected as point and circle using the **Target area**. If the shape of the target is selected as rectangle then you need to define the length and width of the rectangle. Using the **Datum referrence(s)** edit box you can specify the datum references to the datum target symbol. Figure 13-11 shows the datum target symbols added to drawing view.
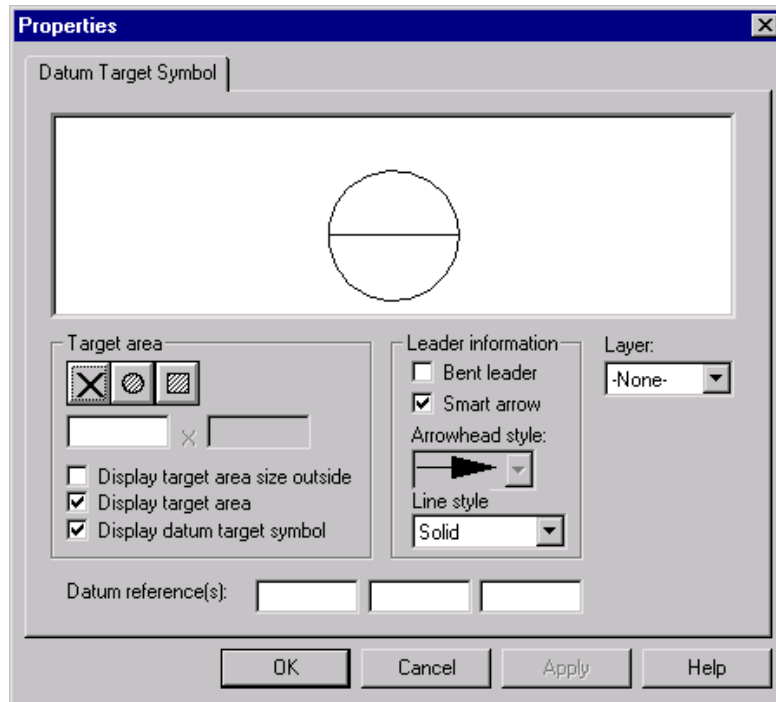
Figure 13-10 *The* **Properties** *dialog box used to add the datum target symbol to the drawing views*
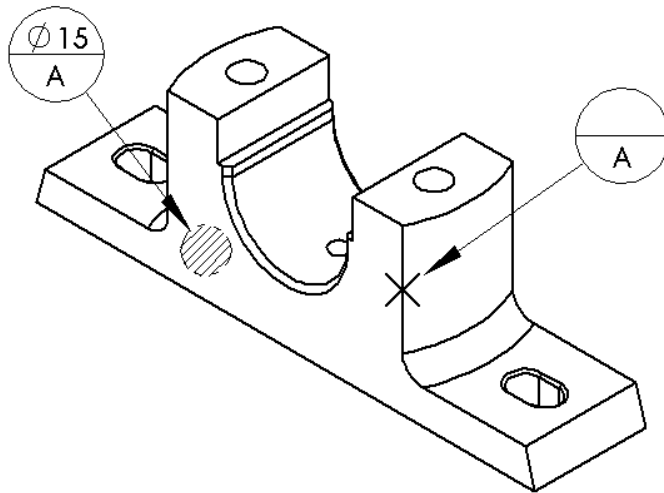

Figure 13-11 *Datum target symbols added to the model*

Adding Center Marks to the Drawing Views

Toolbar:	Annotation > Center Mark
Menu:	Insert > Annotations > Center Mark

The **Center Mark** tool is used to add center marks to the circular entities. As discussed earlier, center marks are automatically generated when you generate the model. But if the center marks are not generated while generating the drawing view, or if you have sketched a view, you can use this tool to add the center marks to the drawing views. To add the center marks to the drawing views, choose the **Center Mark** button from the **Annotations** toolbar or choose **Insert > Annotation > Center Mark** from the menu bar. The **Center Mark PropertyManager** will be displayed as shown in Figure 13-12.
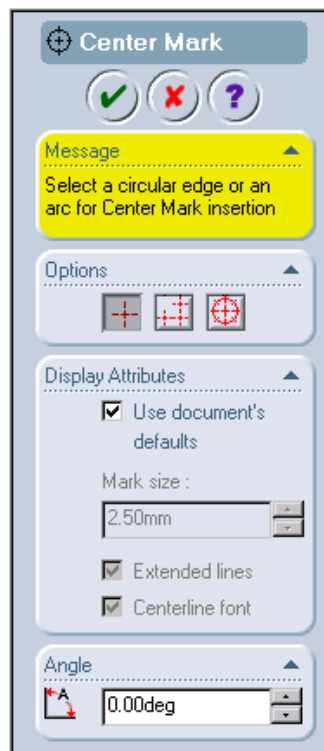

*Figure 13-12 The **Center Mark PropertyManager***

When you invoke the **Center Mark PropertyManager**, the cursor is replaced by the center mark cursor and you are prompted to select a circular edge or an arc for the Center Mark insertion. By default, the **Single Center Mark** button is selected in the **Options** rollout of the **Center Mark PropertyManager**. Select the circular edge or arc to add the center mark. Figure 13-13 shows the center marks added using the **Single Center Mark** button selected from the **Options** rollout.

If you add center marks to a hole that is part of a linear or circular pattern, the **Propagate** button appears on the center mark. If you choose this button the center mark is added to all the remaining instances of the pattern.

The center mark in rectangular and circular patterns can also be applied using the **Linear Center Mark** and **Circular Center Mark** buttons. If you select the **Linear Center Mark** button from the **Options** rollout, then you can add the center mark in the linear pattern format. Since the **Connection lines** check box is selected by default, the center mark will be connected using the center lines. Figure 13-14 shows the center marks added using the **Linear Center Mark** button with the **Connection lines** check box selected.
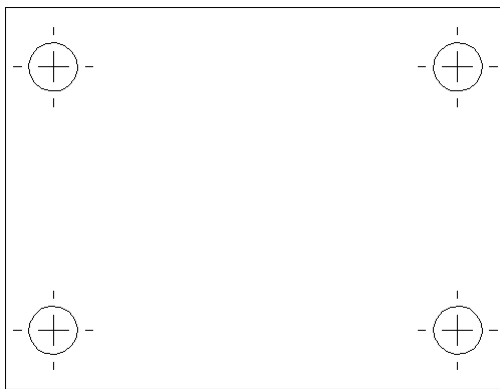
Figure 13-13 *Center marks added using the* ***Single Center Mark*** *option*
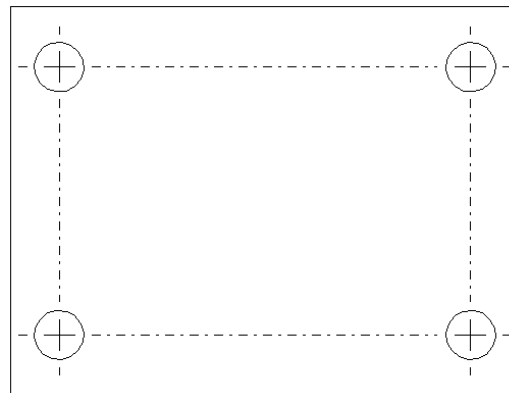
Figure 13-14 *Center marks added using the* ***Linear Center Mark*** *option*

Using the **Circular Center Mark** button, you can create the center mark in the circular pattern format. When you select the **Circular Center Mark** button from the **Options** rollout, the **Circular lines** check box, **Radial lines** check box, and the **Base center mark** check box are displayed. The **Circular lines** check box is used to create a circular line around the centers of the circle arranged in the form of circular pattern. The **Radial lines** check box is used to display the radial lines with the center mark. The **Base center mark** check box is used to display the base center mark. Figure 13-15 shows the center mark created with the **Base center mark** check box cleared. Figure 13-16 shows the center mark created with the **Base center mark** check box selected. Figure 13-17 shows the center mark created with the **Radial lines** check box selected.

The **Display Attributes** rollout available in the **Center Mark PropertyManager** is used to define the size of the center mark and the extended lines. The **Angle** rollout is used to rotate the center mark at an angle.
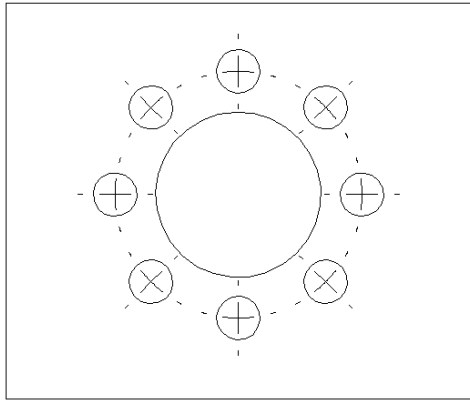
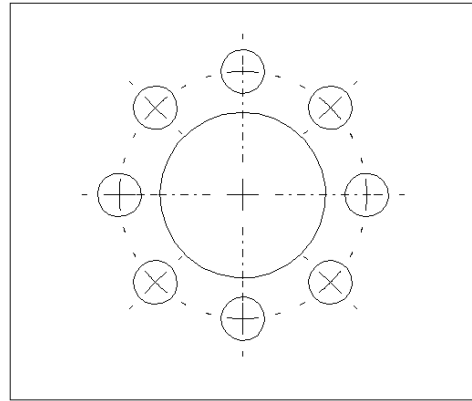***Figure 13-15*** *Center marks created with the **Base center mark** check box cleared*

Figure 13-16 *Center marks created with the **Base center mark** check box selected*
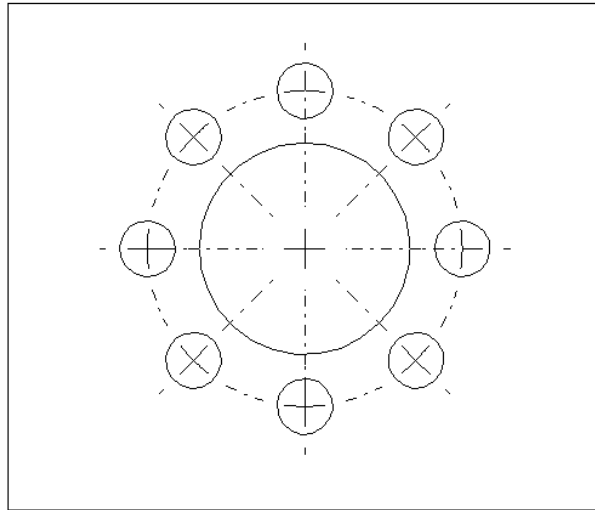

Figure 13-17 *Center marks created with the **Radial lines** check box selected*

Adding Center Lines to Views

Toolbar:	Annotation > Centerline
Menu:	Insert > Annotations > Centerline

The **Centerline** tool is used to create the centerlines in the views by selecting two edges/sketch segments or a single cylindrical/conical face. To create a centerline, choose the **Centerline** button from the **Annotation** toolbar or choose **Insert > Annotations > Centerline** from the menu bar. The **Centerline PropertyManager** is invoked and it prompts you to select two edges/sketch segments or a single cylindrical/conical face. Select the entity or entities from the view to add the centerline. Figure 13-18 shows the center lines added to the drawing views by selecting the surface.
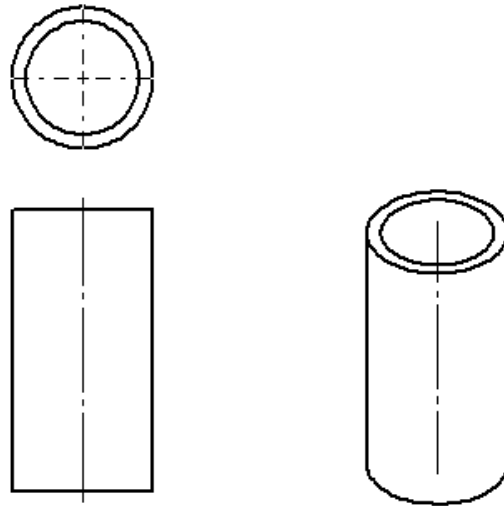
*Figure 13-18 Centerlines added using the **Center Mark** tool*

Tip. *You can also set the option for the automatic creation of centerline while generating the drawing views. Choose **Tools > Options** from the menu bar to invoke the **System Options- General** dialog box. Select the **Document Properties** tab from this dialog box. Select the **Centerlines** check box from the **Auto insertion on view creation** area and choose **OK** to close the dialog box.*

You can also directly select a view to add the centerlines.

Adding Hole Callout to the Views

Toolbar:	Annotation > Hole Callout
Menu:	Insert > Annotations > Hole Callout

The **Hole Callout** tool is used to generate the hole callouts that are created in **Part** mode using the **Hole PropertyManager**, **Hole Wizard**, or cut option. To generate a hole callout, choose the **Hole Callout** button from the **Annotation** toolbar or choose **Insert > Annotations > Hole Callout** from the menu bar. The cursor will be replaced by the hole callout cursor. Select the hole from the drawing views; the hole callout will be attached to the cursor. Pick a point on the drawing sheet to place the hole callout. Figure 13-19 shows a drawing view with hole callouts generated using the **Hole Callout** tool.

Adding the Cosmetic Threads to the Drawing Views

Toolbar:	Annotation > Cosmetic Thread (*Customize to Add*)
Menu:	Insert > Annotations > Cosmetic Threads

The **Cosmetic Thread** tool is used to add cosmetic threads that will display the thread conventions in the drawing views. To add a cosmetic thread in the drawing view, select the circular edge from the drawing view on which you need to apply the cosmetic thread. Now, choose the **Cosmetic Thread** button from the **Annotation** toolbar or choose
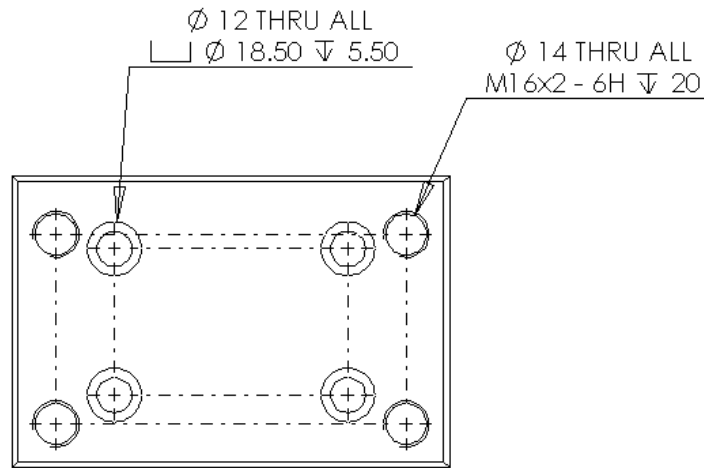
Figure 13-19 *Hole callouts generated using the **Hole Callout** tool*

Insert > **Annotations** > **Cosmetic Thread** from the menu bar. The **Cosmetic Thread PropertyManager** is displayed as shown in Figure 13-20.
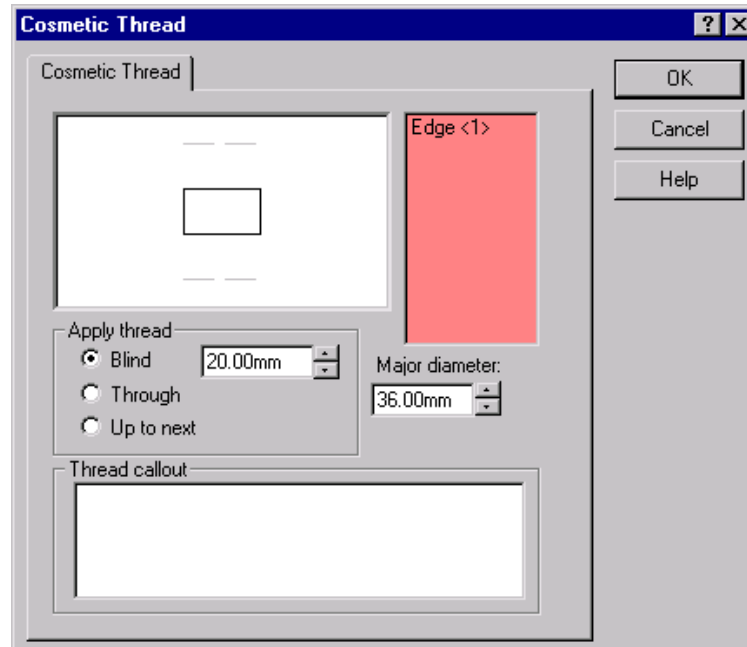

Figure 13-20 *The **Cosmetic Thread** dialog box*

The options available in the **Cosmetic Thread** dialog box are discussed next.

Preview Area
The preview area is used to display the preview of the cosmetic thread. The preview of the cosmetic thread modifies dynamically as you modify the parameters of the cosmetic thread.

Selection Area
The selection area is used to display the name of the selected entity on which you need to apply the cosmetic thread.

Apply thread
The **Apply thread** area is used to define the depth of the cosmetic thread.

Major diameter
The **Major diameter** spinner is used to define the major diameter of the thread.

Thread callout
The **Thread callout** edit box is used to specify the text to be used in the thread callout for the cosmetic thread.

After setting all the parameters, choose the **OK** button from the **Cosmetic Thread** dialog box. When you add a cosmetic thread to a generated drawing view, the thread convention will be displayed in all the drawing views. Figure 13-21 shows the cosmetic thread added to the model.
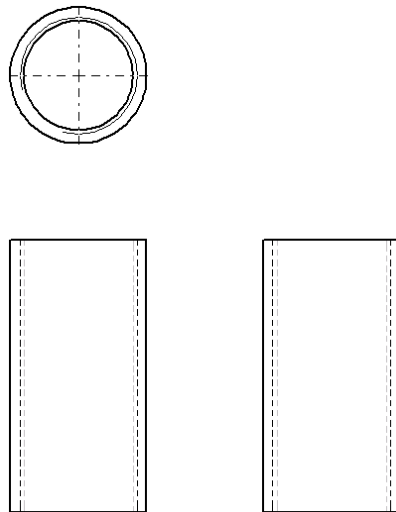
Figure 13-21 *Cosmetic threads added to the drawing views*

Tip. *You can change the model display setting from hidden lines removed to hidden lines visible, or wireframe, shaded using the options available in the **View** toolbar.*

Note

*The thread conventions once added in the drawing view can only be deleted from the part file. You cannot delete the thread convention from the drawing file. To delete the thread convention in the part file, expand the **Hole** or **Cut** feature in the **FeatureManager Design Tree**. Now, select the thread convention and delete it.*

Adding Weld Symbols to the Drawing Views

Toolbar:	Annotation > Weld Symbol	(*Customize to Add*)
Menu:	Insert > Annotations > Weld Symbol	

The **Weld Symbol** tool is used to add the welding symbol to the drawing views. To add the welding symbol, select the edge or the vertex where you need to add the weld symbol and choose the **Weld Symbol** button from the **Annotation** toolbar or choose **Insert > Annotations > Weld Symbol** from the menu bar. The **Properties** dialog box will be displayed as shown in Figure 13-22.
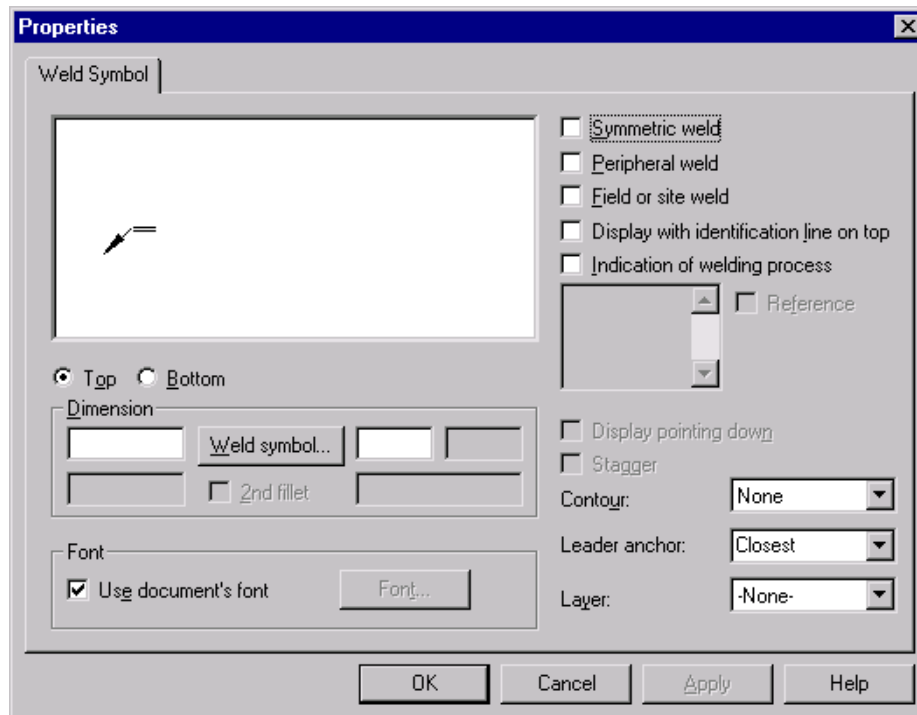

*Figure 13-22 The **Properties** dialog box used to add the welding symbols to the drawing views*

The options available in the dialog box are discussed next.

Dimension

The **Dimension** area is used to specify the dimension of the weld and the welding symbol. The **Weld Symbol** button available in the this area is used to define the welding symbols

from the **Symbols** dialog box displayed in Figure 13-23. Using the edit boxes available in this area you can define the dimension of weld.
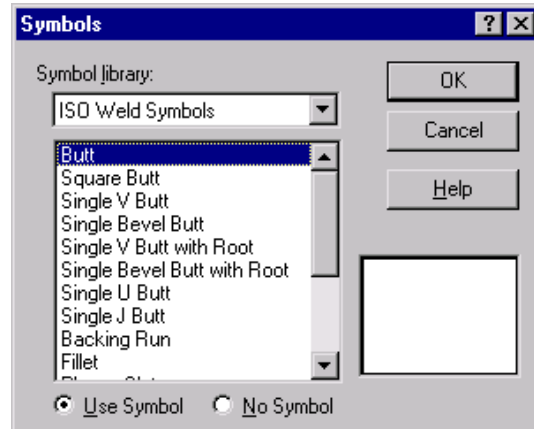

Figure 13-23 *The **Symbols** dialog box used to define the weld symbols*

Top/Bottom
Using the **Top** radio button and the **Bottom** radio button you can specify if you need to display the welding symbols on the top or at the bottom.

Contour
The **Contour** drop-down list is used to define the shape of the cross-section of the weld contour to be flat, convex, or concave.

The other options available in this dialog box are used to define if you need a symmetric weld, peripheral weld, field, site weld, identification line, and indication of welding process.

Stagger
The **Stagger** check box is selected to define the stagger in the weld symbol.

After specifying all the parameters in the **Properties** dialog box, choose the **OK** button to exit the dialog box and place the weld symbol. Figure 13-24 shows the weld symbol added to the drawing view.

Adding the Multi-jog Leader to the Drawing Views

Toolbar:	Annotation > Multi-jog Leader (*Customize to Add*)
Menu:	Insert > Annotations > Multi-jog Leader

Using the **Multi-jog Line** tool you can add a multi-jog leader line to the drawing views. A multi-jog leader is basically a leader in which you can add multiple jog lines. To add a multi-jog leader, choose the **Multi-jog Leader** button from the **Annotation** toolbar or choose **Insert > Annotations > Multi-jog Leader** from the menu bar. The cursor will be replaced by the multi-jog line cursor. Select a point on the sheet or on an entity from where you need to start the leader. Now, specify the points on the drawing sheet to specify the
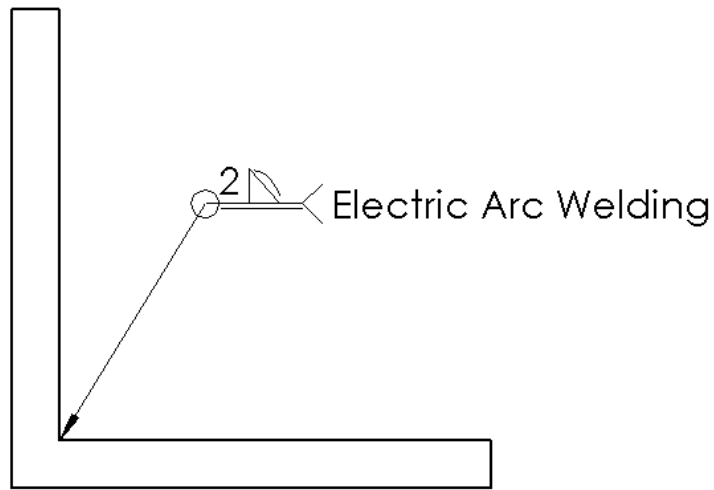
Figure 13-24 *Welding symbol added to the model*

jogs and then select the second entity where the second end of leader will be placed. A multi-jog leader will be created. Double-click the sheet to end the multi-jog leader creation. Figure 13-25 shows a multi-jog leader added to the drawing view.
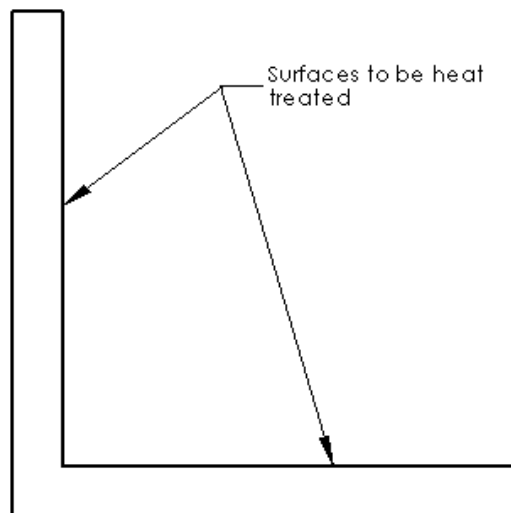
Figure 13-25 *Multi-jog leader added to the drawing view*

Adding the Dowel Pin Symbols to the Drawing Views

Toolbar:	Annotation > Dowel Pin Symbol (*Customize to Add*)
Menu:	Insert > Annotations > Dowel Pin Symbol

The **Dowel Pin Symbol** tool is used to add the dowel pin symbol to the holes in the drawing views. The dowel pin symbol is used to confirm the size of the selected hole.

To create a dowel pin symbol, select a hole or circular edge from the drawing view and choose the **Dowel Pin Symbol** button from the **Annotation** toolbar or choose **Insert > Annotations > Dowel Pin Symbol** from the menu bar. The **Dowel Pin Symbol PropertyManager** is displayed. Using the **Flip symbol** check box in the **Display Attributes** rollout, you can flip the direction of the dowel pin symbol.

EDITING THE ANNOTATIONS

You can edit the annotation added to drawing view by selecting them or by double-clicking them to display their respective **PropertyManager** or dialog box. Using the **PropertyManager** or dialog box you can edit the parameters of the annotations.

ADDING THE BOM AND BALLOONS IN THE DRAWING

Menu: Insert > Bill of Materials

The BOM, also known as Bill of Materials, is a table that provides you the information related to the number of components in an assembly, their name, their quantity, and other related information. You can add the BOM in the drawing sheet to display the part list of the components used in the assembly. The BOM placed in the drawing document is parametric in nature. Therefore, if you add or delete a part from the assembly, the change will be reflected in the BOM in the assembly document. To insert a BOM in the drawing file having the drawing views of an assembly, select any one view from the drawing document and choose **Insert > Bill of Materials** from the menu bar. The **Select BOM Template** dialog box is invoked. Using this dialog box you can select the default BOM templates provided in SolidWorks. Select the **bomtemp.xls** file from the **Select BOM Template** dialog box and choose the **Open** button. The **Bill of Materials Properties** dialog box will be displayed as shown in Figure 13-26.
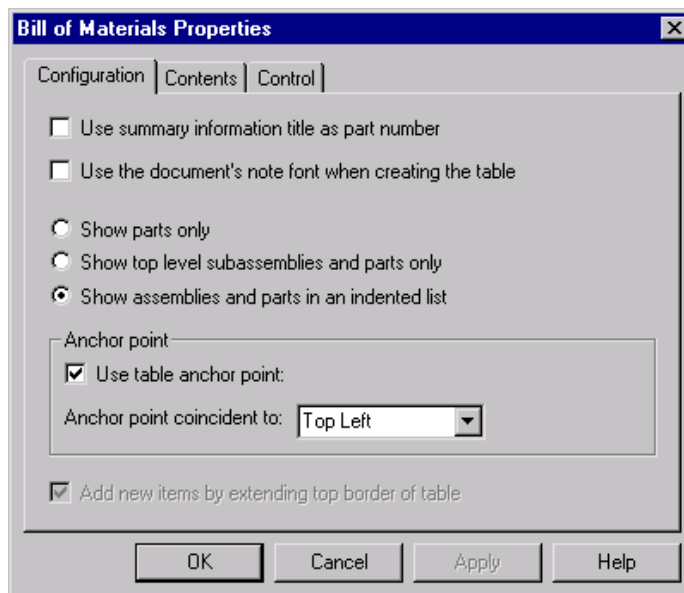

Figure 13-26 *The* **Bill of Materials Properties** *dialog box*

The options available in this dialog box are discussed next.

Configuration Tab

The options available in the **Configuration** tab are used to specify the information to be included in the BOM and the parts and the subassemblies to be included in the BOM. The options available in this dialog box are discussed next.

The **Anchor point** area available in this dialog box is used to define the point of placement of the BOM. The **Use table anchor point** check box is used to coincide the anchor point of the drawing sheet with the anchor point of the BOM. You will learn more about anchor point later in this chapter. The **Anchor point coincident to** drop-down list is used to define the point of the BOM that should coincide with the drawing sheet anchor point.

Contents

The **Contents** tab is used to display the content of the parts to be used in the BOM. When you select the **Contents** tab from this dialog box, the preview of the BOM attached to the top left corner of the drawing sheet is displayed. The **Contents** tab selected from the **Bill of Materials Properties** dialog box is displayed in Figure 13-27.
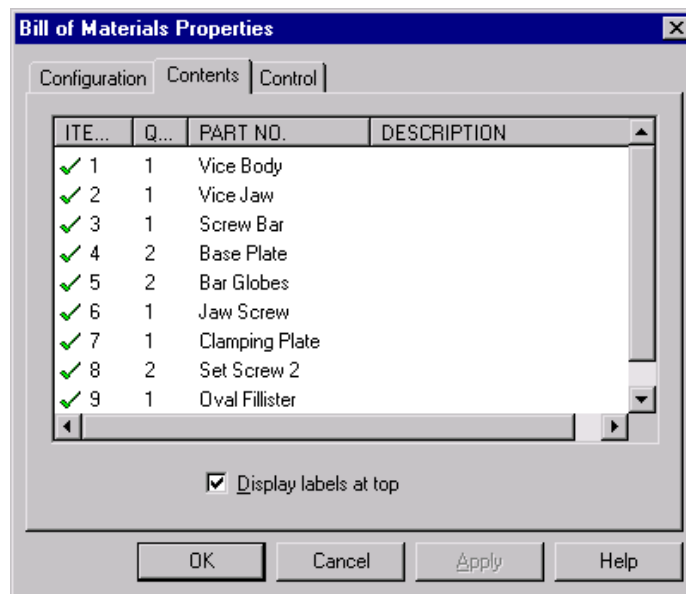

*Figure 13-27 The **Contents** tab of the **Bill of Materials Properties** dialog box*

The parts that will be displayed in the BOM are checked with a green check mark in the **Contents** tab of the **Bill of Materials Properties** dialog box. You can clear the check mark from the parts that need not be included in the BOM.

Control

The **Control** tab is used to define the style from the row numbers. You can also split the table using the options available in this tab.

After setting all the parameters, choose **OK** from this dialog box. The BOM will be placed at the top-right corner of the drawing sheet. Figure 13-28 shows the BOM added to the drawing sheet.
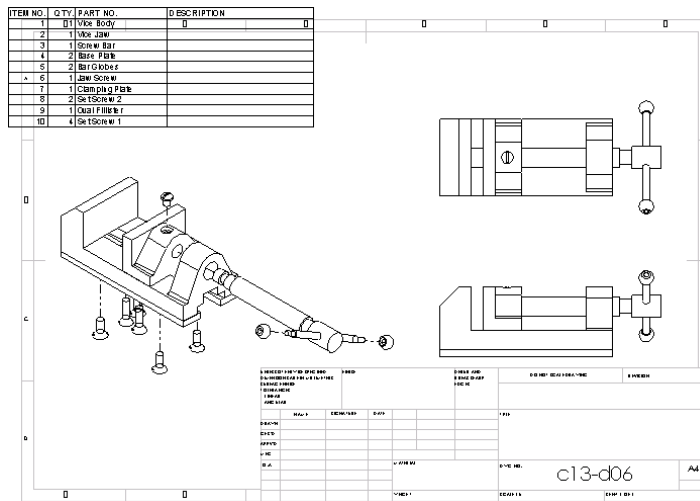

Figure 13-28 *BOM added to the drawing sheet*

Tip. *You can edit the BOM placed in the drawing sheet by just double-clicking the BOM; the Microsoft Excel environment will be invoked and using the tools available in Microsoft Excel, you can edit or modify the BOM. Double-click the drawing sheet to return to the assembly environment.*

*You can set the anchor of the drawing sheet by expanding **Sheet Formats 1** from the **FeatureManager Design Tree**. Select the **Bill of Materials Anchor** and right-click to invoke the shortcut menu. Choose the **Set Anchor** option from the shortcut menu. The current anchor will be displayed as a point on the drawing sheet. Select a point in the drawing sheet where you need to specify the anchor point for placing the BOM. Now, select **Sheet 1** from the **FeatureManager Design Tree** and choose the **Edit Sheet** option by invoking the shortcut menu to return to the edit sheet environment.*

*You can also unlock the BOM from the anchor point so that it can be moved freely anywhere. To unlock the BOM, right-click it and choose **Anchor > Unlock** from the shortcut menu. The shortcut menu also has some other options such as **Hiding** or **Editing** the BOM.*

Adding Balloons to the Drawing Views

Toolbar:	Annotation > Balloon
Menu:	Insert > Annotations > Balloon

The **Balloon** tool is used to add the balloons to the components of the assembly in the drawing view. The naming of balloons depends on the sequence of the parts in BOM. To add balloons to the drawing view, choose the **Balloon** button from the **Annotation** toolbar or choose **Insert > Annotations > Balloon** from the menu bar. The **Balloon PropertyManager** is displayed as shown in Figure 13-29.
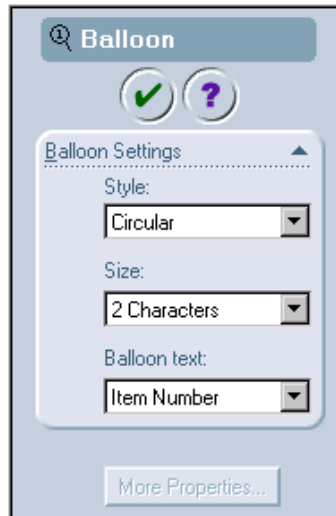

*Figure 13-29 The **Balloon PropertyManager***

After invoking the **Balloon PropertyManager**, you are prompted to select one or more locations to place balloons. Select the components from the assembly drawing view to add the balloons. If you select the face of a component, the balloon will have a filled circle at the attachment point. However, if you select an edge of the component, the balloon will have a closed fillet arrow. After placing all the balloons, choose the **OK** button from the **Balloon PropertyManager**. You may have to select and move the balloons after placing them. Figure 13-30 shows the drawing sheet in which balloons are added.

ADDING NEW SHEETS TO THE DRAWING VIEWS

You can also add new sheets to the drawing document. A multi-sheet drawing document is generally used when you need to generate the drawing views of all the components and the drawing views of the assembly in the same document. You can switch between the sheets easily to refer to the drawings of different parts of an assembly within the same document and you do not need to open the separate drawing documents. To add a sheet to the drawing document, select **Sheet 1** from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Add Sheet** option from the shortcut menu to add a sheet to the drawing document or choose **Insert > Sheet** from the menu bar. The **Sheet Setup** dialog box is displayed. Using
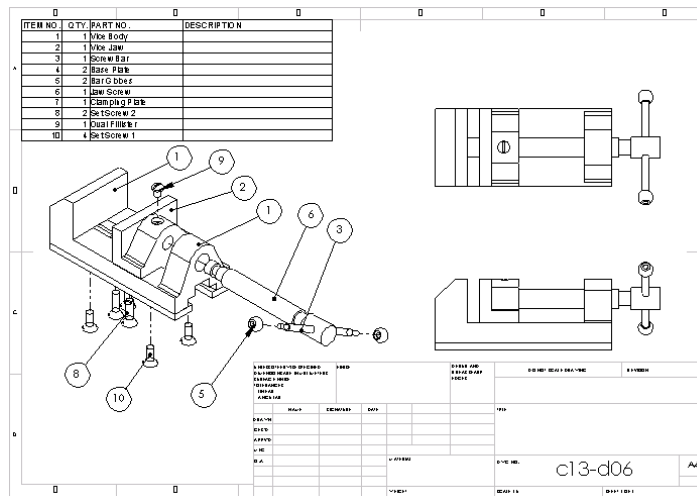
Figure 13-30 *Balloons added to the assembly drawing view*

this dialog box you can specify the name, standard sheet format, and scale projection type for the drawing sheet to be added. Choose the **OK** button from this dialog box; a new sheet will be added in the drawing document. Figure 13-31 shows a drawing document with four drawing sheets added and **Sheet 4** as the active sheet.
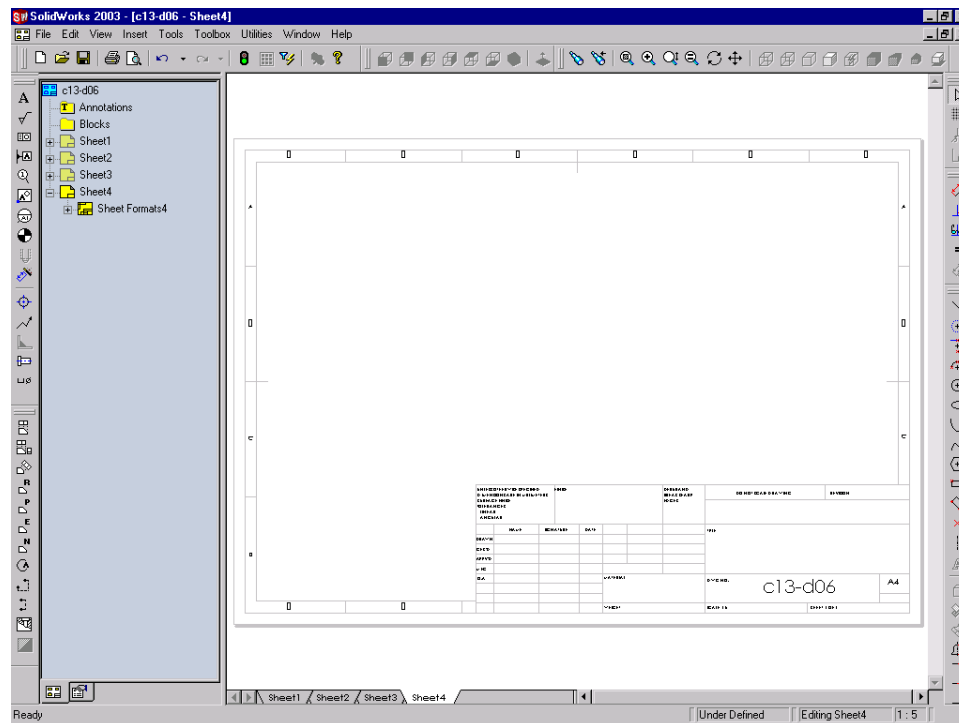

Figure 13-31 *Drawing sheets added to the drawing document*

To activate a drawing sheet, select the sheet from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Active** option from the shortcut menu to activate the selected sheet. You can also activate the sheet by selecting the sheet tab from the bottom of the drawing document.

EDITING THE SHEET FORMAT

You can edit the default standard sheet format according to your requirement. To edit the standard sheet format, select **Sheet 1** from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Edit Sheet Format** option from the shortcut menu. All the entities, annotations, and views will disappear from the drawing sheet. Using the sketching tools available in the **Sketch Tools** toolbar you can edit the sheet format. After editing the sheet format select **Sheet 1** and invoke the shortcut menu. Select the **Edit Sheet** option from the shortcut menu to switch back to the edit sheet environment.

CREATING A USER-DEFINED SHEET FORMAT

In SolidWorks you can also create a user-defined sheet format. To create a user-defined sheet format, create a new drawing document with standard size with the **No Sheet Format** option from the **Sheet Format To Use** dialog box. Select **Sheet 1** from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Edit Sheet Format** option from the shortcut menu. The edit sheet format environment is invoked. Using the standard sketching tools available in the **Sketch Tools** toolbar you can create the sheet format. After creating the sheet format switch back to the edit sheet environment. Choose **File > Save Sheet Format** from the menu bar. The **Save Sheet Format** dialog box is displayed. Select the **Custom sheet format** radio button from this dialog box and browse the location where you need to save the sheet format. Specify the name of the sheet format and choose the save button from the **Save As** dialog box. The location and name of the sheet format will be displayed in the **Custom sheet format** edit box. Choose the **OK** button from the **Save Sheet Format** dialog box. Next time when you invoke the **Sheet Format To Use** dialog box, you can retrieve the saved sheet format by selecting the **Custom sheet format** radio button and browse the location using the **Browse** button. Figure 13-32 shows a user-defined sheet format.
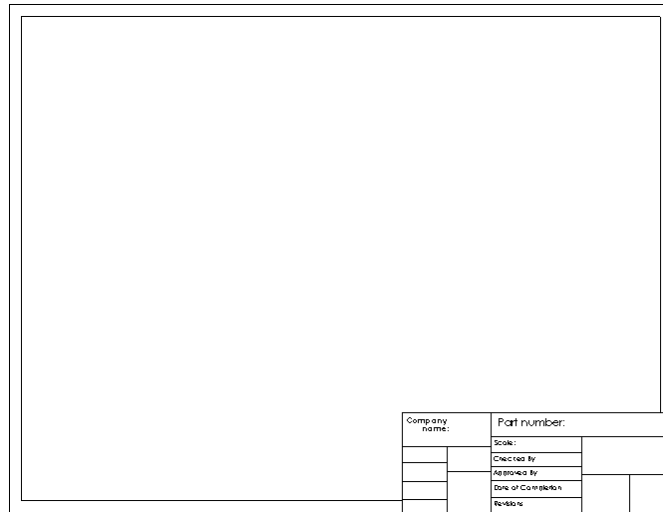
Figure 13-32 A user-defined sheet format

TUTORIALS

Tutorial 1

In this tutorial you will open the drawing created in Tutorial 2 of Chapter 7 and generate the dimensions and add the annotations to the drawing. After that you will change the display of the front and the right view to hidden lines visible, and the display of the isometric and aligned section view to shaded mode.   **(Expected time: 45 min)**

The steps to be followed to complete this tutorial are discussed next:

a.   Copy the model and the drawing document in the current directory.
b.   Configure the font settings and generate the dimensions using the **Insert Model Items** tool.
c.   Generate and arrange the dimensions and delete the unwanted dimensions, refer to Figures 13-34 and 13-35.
d.   Add the datum symbol and geometric tolerance to the drawing views, refer Figures 13-36 and 13-37.
e.   Change the model display state of the drawing views, refer to Figure 13-38.

Copying the Model in the Current Directory

First, you need to copy the model and the drawing document in the current directory.

1.   Create a directory with the name *c13* in the *SolidWorks* directory and copy *c07tut02.sldprt* from the */My Document/SolidWorks/c7* directory to this directory.

Opening the Drawing Document

After copying the part and the assembly document, you need to open the drawing document in the SolidWorks window.

1. Invoke the **Open** dialog box and open *c12tut01.sldprt* document.

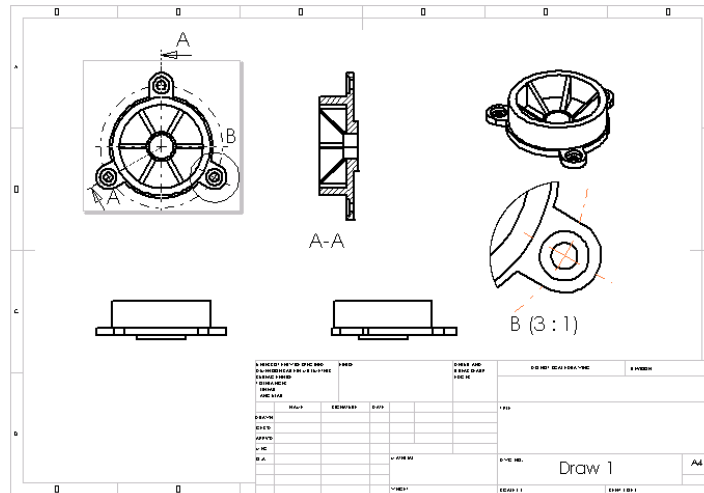   The drawing document in which you need to add the detailing is displayed in Figure 13-33.

Figure 13-33 Drawing document in which you need to add the detailing

Applying the Documents Settings

Before you start generating dimensions, you need to configure the document settings.

1. Invoke the **Document Properties - Grid/Snap** dialog box and select the **Annotations Font** option from this dialog box.

2. The **Note/Balloon** option is selected in the **Annotation type** area. Select this option again. The **Choose Font** dialog box will be invoked.

3. Select the **Points** radio button from the **Height** area and set the value of the font size to **9** from the list box.

4. Choose the **OK** button from the **Choose Font** dialog box. Similarly, set the font size for dimension, detail, and section to **9**.

5. Now, select the **Arrow** option from the **Document Properties** dialog box. The options related to size and shape of the arrows are displayed.

6. Set the value of the **Height** as **1**, **Width** as **3**, and **Length** as **3** in the **Size** area.

7. In the **Section/View size** area, set the value of the **Height** as **3**, **Width** as **6**, and **Length** as **12**.

8. Choose the **OK** button from the **Document Properties - Grid/Snap** dialog box.

After applying the document setting, you need to generate the dimensions of the drawing view.

Generating the Dimensions

Next, you need to generate the dimensions using the **Insert Model Items** tool. As discussed earlier, if you do not select any view and generate the dimensions using the **Insert Model Items** tool, all the dimensions will be displayed in all the views. Therefore, sometimes the dimensions overlap each other and it becomes confusing to judge which dimension is of which view. Therefore, you will select the view in which you need to generate the dimension and then you will invoke the **Insert Model Items** tool.

1. Select the top view and choose the **Model Items** button from the **Annotation** toolbar. The **Insert Model Items** dialog box is displayed. The name of the selected view is displayed in the **Import to views** display area.

2. Choose the **OK** button from the **Insert Model Items** dialog box.

    The dimensions of the model that can be displayed in the selected view are generated. If a radial dimension is attached to the counterbore hole in the top view, it needs to be deleted because you will add a hole callout to this counterbore hole.

3. Select the radial dimension and press the DELETE key from keyboard to delete the dimension. Similarly, delete the dimension of diameter **20**.

    The dimension arrows are displayed as no filled, but need to be filled dimension arrows.

4. Drag a window such that all the dimensions are enclosed inside the window. Release the left mouse button to select all the generated dimensions.

    The **Dimension PropertyManager** is displayed.

5. Select the **Filled Arrow** option from the **Style** drop-down list of the **Arrow** rollout.

    The arrows will be changed to filled arrows. The dimensions are scattered arbitrarily on the drawing sheet. Therefore, you need to arrange the dimensions by moving them to the required locations.

6. Select the dimension one by one and drag them to the desired location, refer to Figure 13-34.

7. Now, select the aligned section view and generate the dimensions using the **Insert Model Items** tool and move the dimensions to place them at appropriate places, refer to Figure 13-35.

8. Choose the **Hole Callout** button from the **Annotation** toolbar and select the outer circle of one of the counter bore feature. The hole callout will be attached to the cursor. Pick a point on the drawing sheet to place the hole callout.
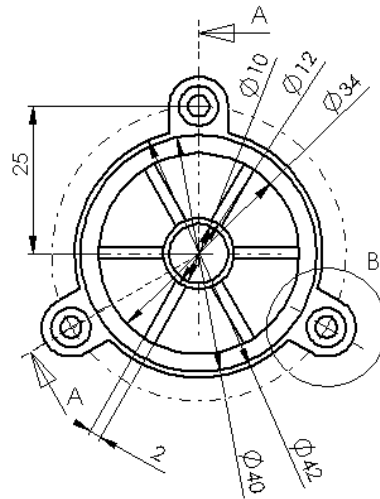
Figure 13-34 *The dimensions generated using the* ***Insert Model Items*** *tool*
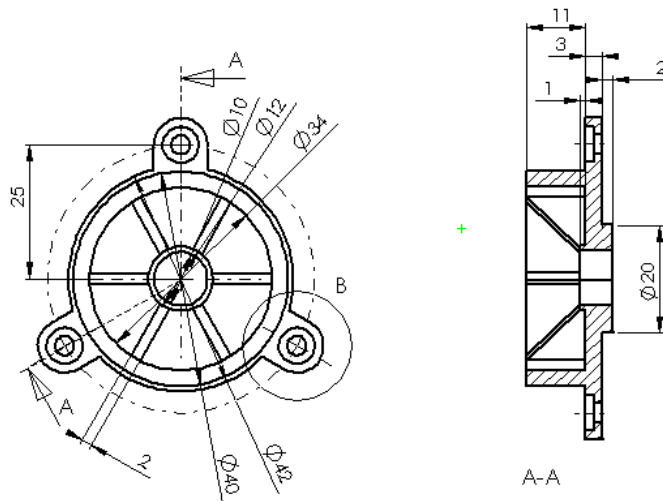
Figure 13-35 *The dimensions generated by selecting the aligned section view*

Adding the Datum Feature Symbol to the Drawing View

After generating the dimensions, you need to add the datum feature symbol to the drawing view. Datum feature symbols are used as the datum reference for adding the geometric tolerance to the drawing views.

1. Select the edge of the outer cylindrical feature from the top view and then choose the **Datum Feature Symbol** button from the **Annotation** toolbar.

The **Properties** dialog box will be displayed, which is used to define the parameters of the datum feature symbol.

2. Choose the **OK** button from the **Properties** toolbar.

    The datum feature symbol is placed at some default location.

3. Select the datum feature symbol and drag and place it at a location as shown in Figure 13-36.
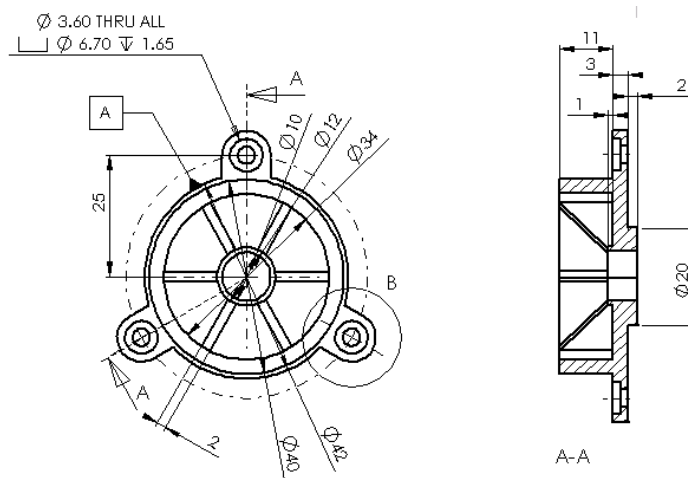

Figure 13-36 Datum feature symbol added to the drawing view

Adding the Geometric Tolerance to the Drawing View

After defining the datum feature symbol, you will add the geometric tolerance to the drawing view.

1. Select the circular edge that has a diameter of 12 from the top view and choose the **Geometric Tolerance** button from the **Annotation** toolbar.

    The **Properties** dialog box is displayed. It is used to specify the parameters of the geometric tolerance.

2. Choose the **GCS** button from the **Feature control frames** area. The **Symbols** dialog box is displayed.

3. Choose the **Concentricity and coaxiality** option from the **ISO Geometric Tolerancing** list box and choose the **OK** button from the **Symbols** dialog box.

4. Set the value of tolerance in the **Tolerance 1** edit box to **.002**.

5. Specify **A** in the **Primary** edit box to define the primary datum reference.

6. Choose the **OK** button from the **Property** dialog box. The geometric tolerance will be placed attached to the selected circular edge.

    You may need to move the geometric tolerance if it overlaps the drawing view. Figure 13-37 shows the drawing view after adding the geometric tolerance.
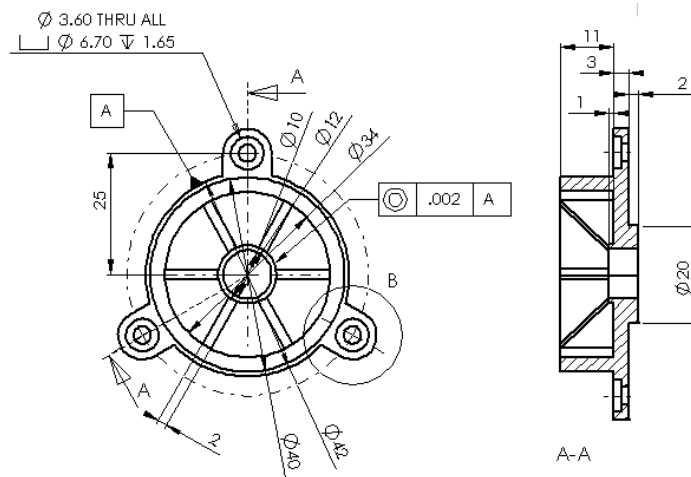
Figure 13-37 Geometric tolerance added to the drawing view

Changing the Display View Options

After adding all the annotations to the drawing views, you need to change the model display setting as discussed in the description of this tutorial.

1. Press and hold down the CTRL key and select the front view and the right-side view from the drawing sheet.

2. Choose the **Hidden Lines Visible** button from the **View** toolbar.

    The hidden lines will be displayed in the selected drawing views.

3. Now, select the isometric view and the aligned section view from the drawing sheet.

4. Choose the **Shaded** button and the **Display HRL Edges In Shaded Mode** button from the **View** toolbar.

    Figure 13-38 shows the final drawing sheet after changing the display view settings.

5. Save the drawing file.

Figure 13-38 *Final drawing sheet*

Tutorial 2

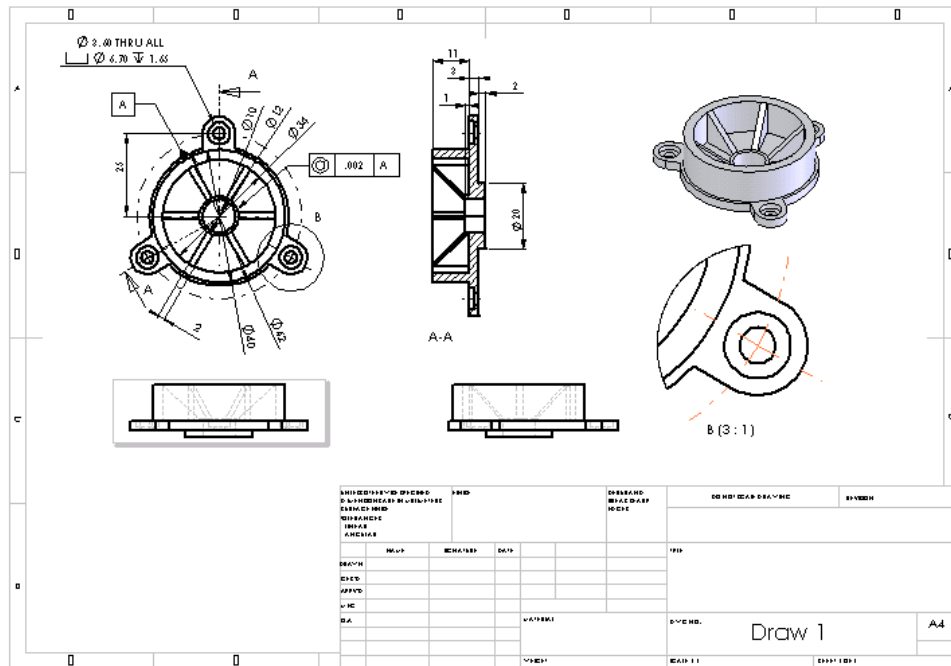In this tutorial you will generate the BOM of the Bench Vice assembly and then add balloons to the isometric view in the exploded state.   **(Expected time: 45 min.)**

The steps to be followed to complete this tutorial are discussed next:.

a. Copy the Bench Vice folder that contains parts, assembly, and the drawing document in the current directory.
b. Delete the views that are not required in the drawing sheet, refer to Figure 13-40.
c. Move to arrange the views in the drawing sheet, refer to Figure 13-40.
d. Set the anchor on the drawing sheet where the BOM will be attached.
e. Generate the BOM, refer to Figure 13-41.
f. Apply the balloons to the drawing view, refer to Figure 13-42.

Copying the Folder in the Current Directory

First, you need to copy the Bench Vice folder in the current directory.

1. Copy the Bench Vice folder from the */My Document/SolidWorks/c12* directory to the current directory.

Opening the Drawing Document

After copying the part and the assembly document, you need to open the drawing document in the SolidWorks window.

1. Open *c12tut02.slddrw* document.

   The drawing document in which you need to generate the BOM and balloons is displayed in Figure 13-39.
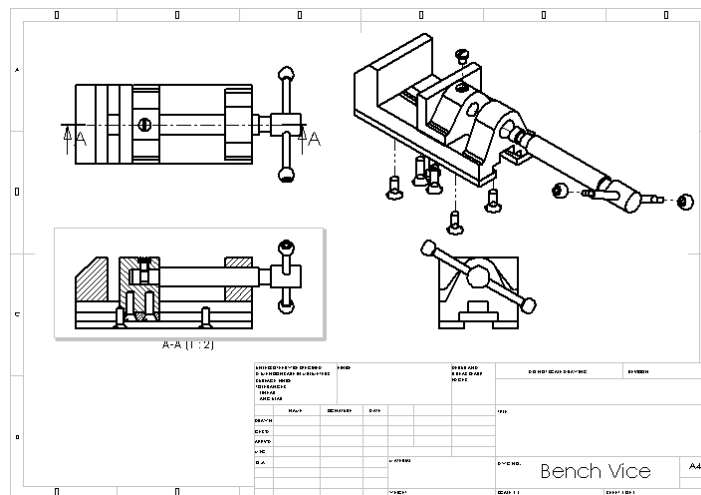
Figure 13-39 Drawing document in which you need to generate BOM and balloons

Deleting the Unwanted View

You need to delete the right-side view, as this view is not required in this tutorial.

1. Select the right-side view and press the DELETE key from the keyboard.

   The **Confirm Delete** dialog box will be displayed.

2. Choose the **Yes** button from this dialog box. The view will be deleted from the current drawing sheet.

Moving the Drawing View

You need to move the exploded isometric view because the BOM will be generated at the top right corner of the drawing sheet.

1. Select the isometric view; the border of the view is displayed in green.

2. Move the cursor to the border. The cursor will be replaced by the move cursor.

3. Drag the cursor to move the drawing view and place the view at the desired location, refer to Figure 13-40.

Figure 13-40 *Drawing sheet after moving the drawing view*

Setting the Anchor for the BOM

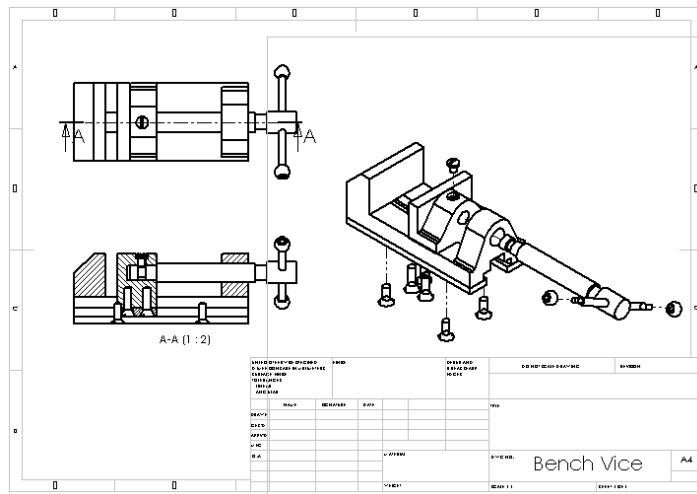Before generating the BOM, you need to set the anchor of the BOM. The anchor is a point on the drawing sheet with which one corner of the BOM coincides. By default, the anchor is defined at the top left corner of the drawing sheet. But in this tutorial you need to add the BOM on the top right corner of the drawing sheet. Therefore, you will set anchor before generating the BOM.

1. Expand **Sheet 1** from the **FeatureManager Design Tree** and then expand **Sheet Formats 1** to display the **Bill of Materials Anchor 1** option.

2. Select the **Bill of Materials Anchor 1** option and invoke the shortcut menu. Choose the **Set Anchor** option from the shortcut menu.

    As you move the cursor on the **Bill of Materials Anchor 1** option, the anchor is displayed as a point in the upper right corner of the drawing sheet.

3. Specify the anchor point on the inner top right corner of the drawing sheet. A point will be placed at the selected location.

    After setting the anchor, you need to switch back to the edit sheet environment.

4. Select **Sheet 1** from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Edit Sheet** option from the shortcut menu.

Generating the BOM

Next, you need to generate the BOM. As discussed earlier, the BOM generated in SolidWorks is parametric in nature. If a component is deleted or added in the assembly, the change is automatically reflected in the BOM.

1. Select the isometric view and choose **Insert > Bill of Materials** from the menu bar.

   The **Select BOM Template** dialog box will be displayed.

2. Select the *bomtemp.xls* file and choose the **Open** button from the **Select BOM Template** dialog box.

   The **Bill of Materials Properties** dialog box will be displayed.

3. Select the **Top Right** option from the **Anchor point coincident to** drop-down list.

4. Choose the **OK** button from the **Bill of Materials Properties** dialog box. The BOM is generated. You will notice that the **Description** column does not have any data. You can add more description by double-clicking the BOM.

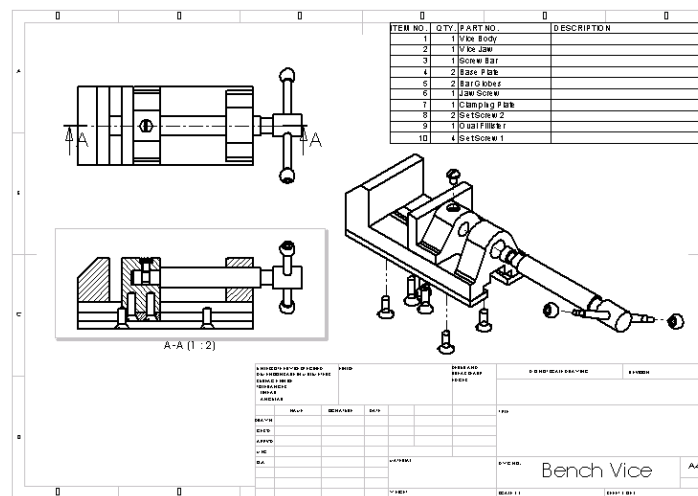   The drawing sheet after generating the BOM is displayed in Figure 13-41.

Figure 13-41 *Drawing sheet after generating the BOM*

Adding the Balloons to the Drawing View

After generating the BOM you need to add the balloons to the drawing views.

1. Choose the **Balloon** button from the **Annotation** toolbar.

   The **Balloon PropertyManager** will be displayed. The cursor is replaced by the balloon cursor.

2.   Select the Vice Body from the isometric view displayed in shaded mode. A balloon will be attached to the Vice Body with the sequence of the balloon as the item number displayed in the BOM.

3.   Select the Vice Jaw; the balloon will be attached to the Vice Jaw.

4.   Similarly, add balloons to all the components in the assembly. You may need to move the balloons, refer to Figure 13-42.

5.   Now, drag a window and select all the balloons. Select the **Tight Fit** option from the **Size** drop-down list in the **Balloon PropertyManager**. The final drawing sheet after adding balloons to the drawing view is shown in Figure 13-42.
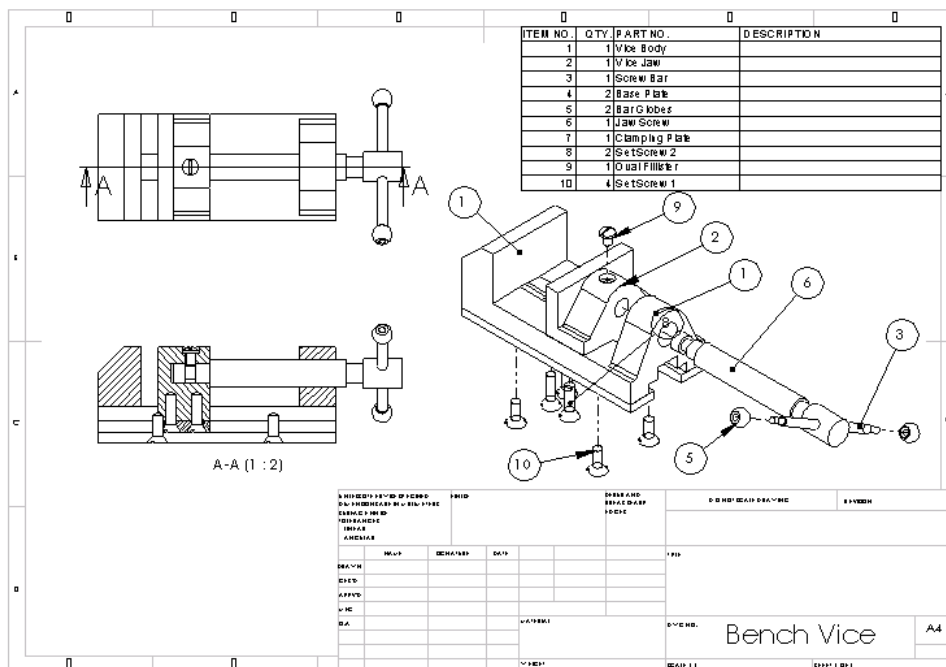
6.   Save the drawing.

Figure 13-42 *Final drawing sheet after adding balloons*

SELF-EVALUATION TEST

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

1.   You cannot add annotations while creating the part in SolidWorks. (T/F)

2. You can add the surface finish symbols to the drawing views. (T/F)

3. The **Projected tolerance zone** area is used to define the quality of the projected tolerance. (T/F)

4. You can set the target shape as a point, circle, or rectangle while adding a datum target. (T/F)

5. You can also set the option for the automatic creation of centerline while generating the drawing views. (T/F)

6. The _____ spinner is used to define the major diameter of the thread.

7. The _____ tool is used to add the balloons to the components of the assembly in the drawing view.

8. The _____ drop-down list available in the **Properties** dialog box to define the weld symbol is used to define the shape of the cross-section of the weld contour to be flat, convex, or concave.

9. The _____ tab of the **Bill of Materials Properties** dialog box is used to display the content of the parts to be used in the BOM.

10. The _____ dialog box is invoked to select the default BOM templates provided in SolidWorks.

REVIEW QUESTIONS

Answer the following questions:

1. The _____ tool is used to add the cosmetic threads that are used to display the thread conventions in the drawing views.

2. The _____ tab available in this dialog box is used to define the row numbers and you can also split the table using the options available in this tab.

3. You can change the model display setting from hidden lines removed to hidden lines visible, or wireframe, or shaded using the options available in the _____ toolbar.

4. Select the _____ check box from the **Auto insertion on view creation** area to automatically create centerlines while generating the view.

5. The _____ tool is used to create the centerlines in the views.

6. Which dialog box is invoked when you choose the **Weld Symbol** button from the **Annotation** toolbar?

    (a) **Weld Properties**   (b) **Weld Symbol**
    (c) **Properties**   (d) **Weld Joints**

7. Which dialog box is displayed when you choose the **Cosmetic Thread** button from the **Annotation** toolbar?

    (a) **Cosmetic Thread Properties**   (b) **Cosmetic Thread**
    (c) **Cosmetic Thread Convention**   (d) None of these

8. Which **PropertyManager** is used to add center marks to the drawing views?

    (a) **Add Center Mark**   (b) **Create Center Mark**
    (c) **Center Mark**   (c) **Cosmetic Thread**

9. Which area available in the **Cosmetic Thread** dialog box is used to define the depth of the cosmetic thread?

    (a) **Apply Thread**   (b) **Thread Depth**
    (c) **Cosmetic Thread**   (d) None of these

10. Which **PropertyManager** is used to add balloons in the drawing views?

    (a) **Add Balloons**   (b) **Balloon Properties**
    (c) **Balloons**   (d) None of these

EXERCISE

Exercise 1
In this exercise you will generate the isometric view in exploded view of the assembly created in Tutorial 1 of Chapter 11 on the standard A4 sheet format. The scale of the view will be 1:5. After generating the view generate the BOM and add balloons to the assembly view shown in Figure 13-43.   **(Expected time: 30 min)**

ITEM NO	QTY.	PART NO.	DESCRIPTION
1	1	master-rod	
2	1	piston	
3	1	piston-pin	
4	1	rod-bush-upper	
5	1	master-rod-bearing	
6	2	piston-pin-plug	
7	4	piston-articulated-rod-subassembly	
	1	articulation-rod	
	1	piston	
	4	piston-ring	
	2	piston-pin-plug	
	1	piston-pin	
	1	rod-bush-upper	
	1	rod-bush-lower	
8	4	link-pin	
9	4	piston-ring	

Figure 13-43 *Drawing view for Exercise 1*

Answers to Self-Evaluation Test

1. T, **2.** T, **3.** F, **4.** T, **5.** T, **6. Major Diameter**, **7. Balloon**, **8. Contour**, **9. Contents**, **10. Select BOM Template**