Chapter 14
Working with Drawings

In this chapter, you will learn the following:

• Invoking Drawing Environment by using the New tool
• Creating the Base/Model View of a Model
• Invoking Drawing Environment from the Part or the Assembly Environment
• Creating a Model View
• Creating a Projected View
• Creating 3 Standard Views
• Working with Angle of Projection
• Defining the Angle of Projection
• Editing the Sheet Format
• Creating a Section View
• Creating an Auxiliary View
• Creating a Detail View
• Creating a Broken-out Section View
• Creating a Break view
• Creating a Crop View
• Creating the Alternate Position View
• Applying Dimensions
• Modifying the Driving Dimension
• Controlling the Dimension and the Arrow Style
• Adding Notes
• Adding the Surface Finish Symbol
• Adding the Weld Symbol
• Adding the Hole Callout
• Adding the Center Mark
• Adding Centerlines
• Creating the Bill of Material (BOM)
• Adding Balloons

After creating parts and assemblies, you need to generate 2D drawings. 2D drawings are the technical drawings, which are used to fully and clearly communicate the information about the end product to be manufactured. 2D drawing is not only a drawing, but also a language of engineers that communicates ideas and information about engineered

products with each other. By using 2D drawings, a designer can communicate the information about the component to be manufactured to the engineers on the shop floor. Underscoring importance of 2D drawings from the designers and engineers point of views, the role of designers is very important in generating accurate or error-free drawings for production. Inaccurate or missing information about a component in drawings can lead to wrong production. Keeping this in mind, SOLIDWORKS provides you with an environment that allows you to generate error-free 2D drawings. This environment is known as Drawing environment.

You can invoke the Drawing environment for generating 2D drawings by using the **New** tool available in the **Standard** toolbar as well as in the **File** menu of the SOLIDWORKS menus. You can also invoke the Drawing environment by using the **Make Drawing from Part/Assembly** tool, which is available within the Part and Assembly environments. Different methods of invoking Drawing environment are discussed next.

Invoking Drawing Environment by using the New Tool

To invoke the Drawing environment by using the **New** tool, click on the **New** tool in the **Standard** toolbar. The **New SOLIDWORKS Document** dialog box appears, see Figure 14.1. Click on the **Drawing** button in this dialog box and then click on the **OK** button. The **Sheet Format/Size** dialog box appears, see Figure 14.2. The options in this dialog box are used to select sheet size/format to be used for creating drawings. The options are as follows:
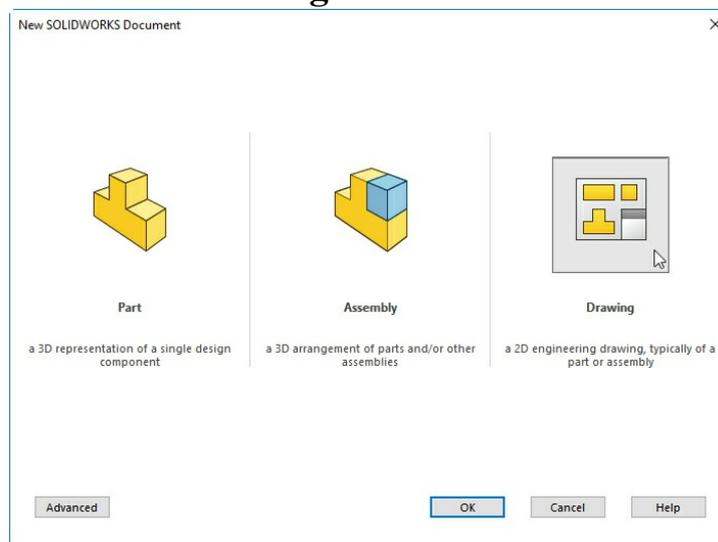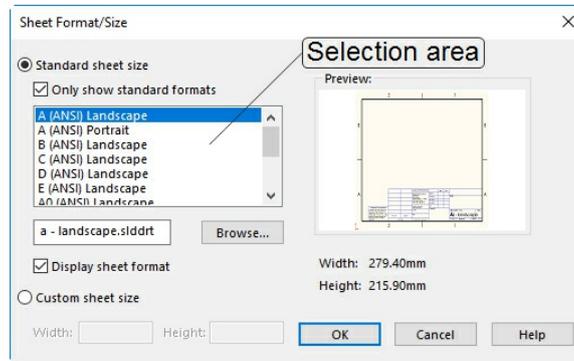
Figure 14.1

Figure 14.2

Standard sheet size

By default, the **Standard sheet size** radio button is selected in the **Sheet Format/Size** dialog box. As a result, a list of standard sheet sizes appears in the **Selection** area of the dialog box, see Figure 14.2. You can select a required standard sheet size from this area for creating drawing views. Note that if the **Only show standard formats** check box is selected in the dialog box, then the list of standard sheet sizes appears in the **Selection** area as per the current drawing standard only, see Figure 14.2. If this check is unchecked, all the standard sheet sizes are listed in this area.

Display sheet format

By default, the **Display sheet format** check box is selected in the dialog box. As a result, a drawing sheet will be displayed with the default standard sheet format. You can also select a sheet format other than the default one by clicking on the **Browse** button in the dialog box. When you click on the **Browse** button, the **Open** dialog box appears. In this dialog box, select the required sheet format and then click on the **Open** button. The preview of the sheet format appears in the **Preview** area of the dialog box. If you uncheck the **Display sheet format** check box, a blank drawing sheet will be displayed for creating drawings. Note that the **Display sheet format** check box is enabled only if the **Standard sheet size** radio button is selected in the dialog box.

Custom sheet size

On selecting the **Custom sheet size** radio button in the dialog box, the **Width** and **Height** fields are enabled and the other options in the dialog box are disabled, see Figure 14.3. In the **Width** and **Height** fields, you can specify the custom width and height values for the drawing sheet.

Figure 14.3
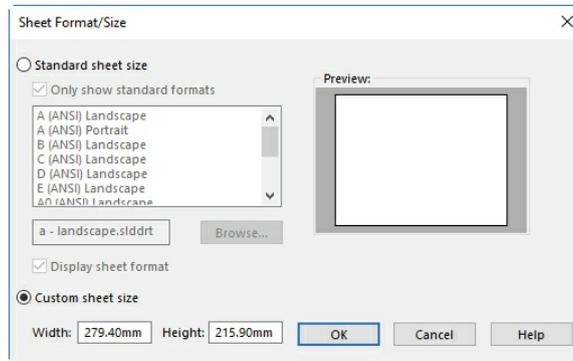
After selecting the required sheet size by using the options of the **Sheet Format/Size** dialog box, click on the **OK** button. The Drawing environment is invoked with the drawing sheet of specified size/format. Also, the **Model View PropertyManager** appears on its left, see Figure 14.4.

Figure 14.4
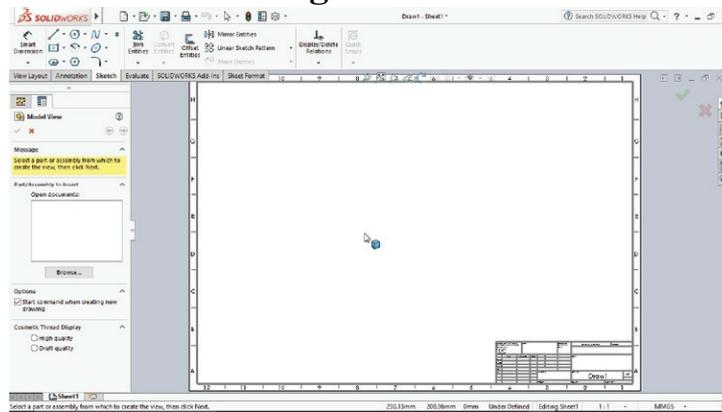
The options of the **Model View PropertyManager** are used to create the base/model view of a component or an assembly. The method of creating the base/model view of a component or an assembly by using this PropertyManager is as follows:
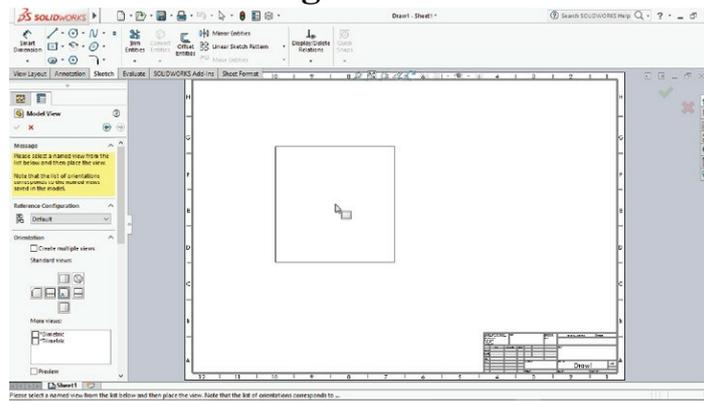

Note:

*The **Model View PropertyManager** appears each time on invoking the Drawing environment. This is because, in the **Options** rollout of the **Model View PropertyManager**, the **Start command when creating new drawing** check box is selected, by default, see Figure 14.4. If you uncheck this check box, next time when you invoke the Drawing environment, the **Model View PropertyManager** will not appear. In such a case, you can invoke the **Model View PropertyManager** by clicking on the **Model View** tool.*

Creating the Base/Model View of a Model `Updated`

To create the base view of a model, click on the **Browse** button in the Part/Assembly to Insert rollout of the **Model View PropertyManager**. The **Open** dialog box appears. In this dialog box, browse to the location where the model, whose drawing view is to be created, has been saved and then select it. Next, click on the **Open** button in the dialog box. A rectangular box representing the base/model view of the model is attached to the cursor, see Figure 14.5. Also, the options of the PropertyManager are modified. The options are as follows:

Figure 14.5

Tip:

*You can also select a component or an assembly, whose drawing views is to be created from the **Open documents** field of the **Part/Assembly to Insert** rollout in the PropertyManager. Note that only the models, which are opened in the current session of SOLIDWORKS are displayed in the **Open documents** field of the PropertyManager. To select a model from the **Open documents** field, double-click on the name of the model in this field.*

Reference Configuration

The drop-down list in the **Reference Configuration** rollout of the PropertyManager contains a list of all the configurations of the selected model. You can select the required configuration of the model for creating its drawing views. Note that if the selected model does not have any configuration created, then only the **Default** option is available in this drop-down list, see Figure 14.6.

Orientation

The options in the **Standard views** area of the **Orientation** rollout are used to select a standard view of the model to be created. By default, the **Front** button is activated in the **Standard views** area, see Figure 14.6. You can click on the required button in this area to create the respective front, top, right, left, back, bottom, or isometric drawing view.

You can also create dimetric or trimetric drawing view of the model by using the **More views** field of this rollout, see Figure 14.6. To create the dimetric or the trimetric view, select the respective check box in the **More views** field of the rollout.

By default, the **Preview** check box of the **Orientation** rollout is unchecked, see Figure 14.6. As a result, an empty rectangular box appears attached to the cursor, which represents the selected view of the model. On selecting this check box, the preview of the selected standard view appears in the rectangular box.

You can also create multiple drawing views of the selected component or the assembly by selecting the **Create multiple views** check box of this rollout, see Figure 14.6.
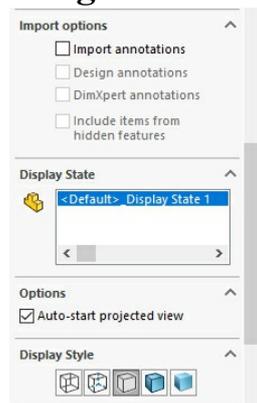
Figure 14.6
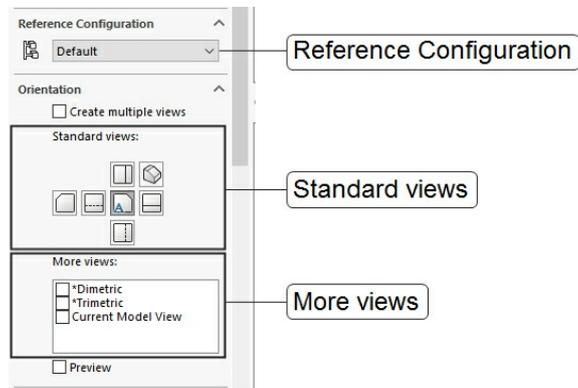
Import Options

The options in the **Import options** rollout are used to import annotations of the model into the drawing view, see Figure 14.7. On selecting the **Import annotations** check box, the **Design annotations**, **DimXpert annotations**, and **Include items from hidden features** check boxes are enabled in the rollout. Depending upon the check boxes selected in this rollout, the annotations of the model are imported into the drawing view.

Figure 14.7

Options

By default, the **Auto-start projected view** check box is selected in the **Options** rollout, see Figure 14.7. As a result, soon after creating the base/model view, the **Projected View PropertyManager** appears automatically. Also, a projected view of the model is attached to the cursor. The **Projected View PropertyManager** is used to create the projected views of the model. You will learn more about projected views later in this chapter.

Display Style

The options in the **Display Style** rollout are used to select the type of display for the drawing view, see Figure 14.7. On selecting the **Wireframe** button, all the visible and hidden edges of the model appear as continued lines in the drawing view, see Figure 14.8 (a). If you select the **Hidden Lines Visible** button, the visible edges appear as continued lines and the hidden edges appear as dotted lines in the drawing view, see Figure 14.8 (b). On selecting the **Hidden Lines Removed** button, only the visible edges of the model appear in the drawing view as continued lines, see Figure 14.8 (c). On selecting the **Shaded With Edges** button, the drawing view is displayed in the shaded

display style with the appearance of visible edges, see Figure 14.8 (d). If you select the **Shaded** button, the drawing view is displayed in the shaded model with the appearance of visible and hidden edges turned off, see Figure 14.8 (e).
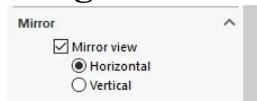
Figure 14.8

Different display styles of a model

a)   b)   c)   d)   e)
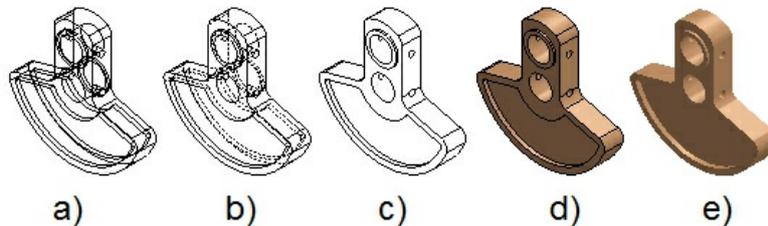
Mirror

The **Mirror** rollout of the PropertyManager is used to create the horizontal or vertical mirror view of the model relative to the selected standard view. To create the horizontal or vertical mirror view of the model, expand the **Mirror** rollout of the PropertyManager and then select the **Mirror view** check box, see Figure 14.9. Next, select the **Horizontal** or **Vertical** radio button to create the respective mirror view of the model relative to the selected standard view.

Figure 14.9

Scale

By default, the **Use sheet scale** radio button is selected in the **Scale** rollout, see Figure 14.10. As a result, the scale of the drawing view is the same as that of the scale of the drawing sheet. On selecting the **Use custom scale** radio button, the **Scale** drop-down list is enabled. By using this drop-down list, you can select a pre-defined scale value of the drawing view. Also, if you select the **User defined** option from the **Scale** drop-down list, then the **Scale** field gets enabled. In this field, you can specify the user-defined scale value of the drawing view.

Figure 14.10
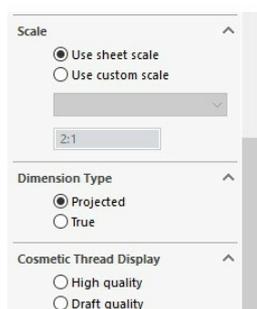
Dimension Type

The **Dimension Type** rollout is used to specify the type of dimensions: true or projected for the drawing view. The projected dimensions appear as 2D dimensions in an isometric drawing view. It is mainly used in orthogonal views such as front, top, and right, see Figure 14.11. The true dimensions appear as accurate model dimensions in an isometric, dimetric, and trimetric drawing views, see Figure 14.12.
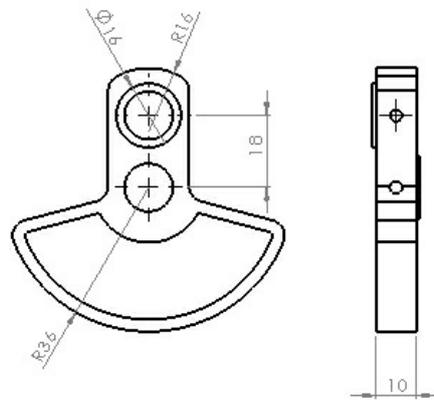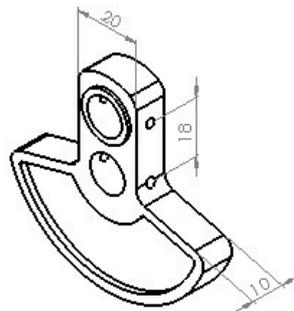
**Figure 14.11**

Figure 14.12

Cosmetic Thread Display

The **High quality** and **Draft quality** radio buttons of the **Cosmetic Thread Display** rollout are used to control the display of cosmetic threads in the drawing view. On selecting the **High quality** radio button, the cosmetic threads appear in precise line fonts. On selecting the **Draft quality** radio button, the cosmetic threads appear with less detail in the drawing views.

After specifying the required settings for creating the base view such as type of view, display style, and scale factor, click on the drawing sheet to position the drawing view. The drawing view is created and placed in the specified position on the drawing sheet. Also, the **Projected View PropertyManager** appears, automatically. Notice that on moving the cursor, a projected view appears attached to the cursor. You can create projection views by specifying the placement points in the drawing sheet. Most of the options of the **Projected View PropertyManager** are the same as those discussed earlier and are used to specify the settings for the projected views. Figure 14.13 shows different projected views that can be created from the base view. Once you have created the required projected views, press the ESC key to exit the creation of projection views.

Figure 14.13

Top View

Left Side View   Front View   Right Side View

Bottom View

Third Angle of Projection
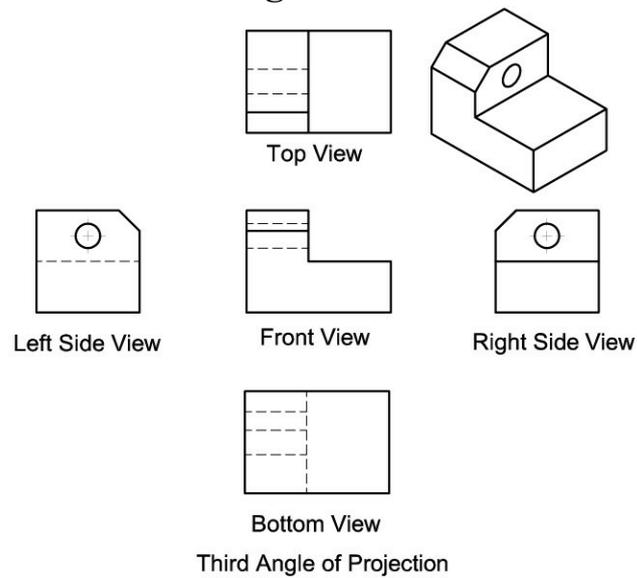
Note:

*You can also control or modify the settings such as display style and scale factor for a drawing view that has been already placed in the drawing sheet. To modify the settings of a drawing view, click on the drawing view in the drawing sheet. The **Drawing View PropertyManager** appears. By using the options of this PropertyManager, you can control the settings of the selected drawing view. All options of this PropertyManager are the same as those discussed earlier.*

# Invoking Drawing Environment from the Part or the Assembly Environment

Similar to invoking the Drawing environment by using the **New** tool and creating different drawing views of a model, you can also invoke the Drawing environment from the Part modeling environment or the Assembly environment. If you are in the Part modeling environment or in the Assembly environment, you can directly invoke the Drawing environment from there and start creating drawing views of the model, which is currently available in the respective environment. To invoke the Drawing environment from the Part modeling environment or the Assembly environment, click on the down arrow available next to the **New** tool in the **Standard** toolbar. A flyout appears, see Figure 14.14.

Figure 14.14

Click on the **Make Drawing from Part/Assembly** tool in this flyout. The **Sheet Format/Size** dialog box appears. The options in this dialog box are used to specify the required format/size of the drawing sheet and are same as those discussed earlier. After defining the format/size of the drawing sheet, click on the **OK** button. The Drawing environment is invoked with the display of **View Palette Task Pane** on the right of the drawing sheet, see Figure 14.15.

Figure 14.15
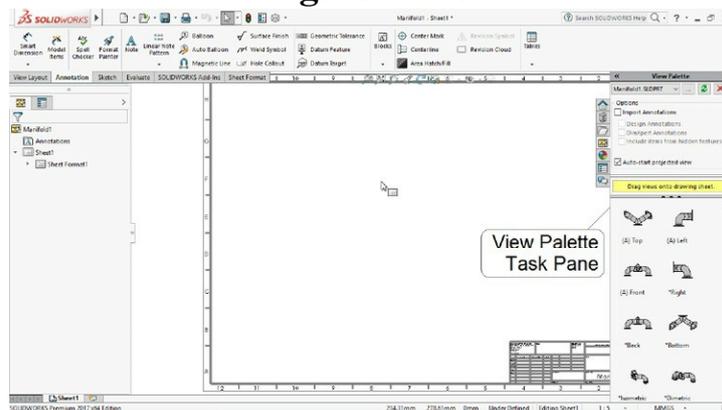
Note that the bottom half area of the **View Palette Task Pane** displays drawing views of the model that were opened in the Part modeling environment or the Assembly environment. By using this task pane, you can drag and drop the required drawing view of the model on the drawing sheet. The first placed view in the drawing sheet is known as a model, base, or parent view. As soon as you place the model/base view in the drawing sheet, the **Projected View PropertyManager** appears on the left of the drawing sheet. This is because the **Auto-start projected view** check box is selected in the upper half of the **View Palette Task Pane**, see Figure 14.15. Now, on moving the cursor in the drawing sheet, a projected view appears and attaches with the cursor,

automatically. You can click on the drawing sheet to specify the position of the projected view. You can create multiple projected views one after another by clicking the left mouse button. Once you have created the required projected views, press the ESC key to exit the creation of projection views.

Creating a Model View

A model view is an independent view of a model. It is also known as base, first or parent view. You can create a model view of a model by using the **Model View PropertyManager** and the **View Palette Task Pane**, which appears automatically on invoking the Drawing environment, as discussed earlier. Moreover, you can also invoke the **Model View PropertyManager** by clicking on the **Model View** tool in the **View Layout CommandManager** for creating the model/base view of a model, see Figure 14.16.

Figure 14.16
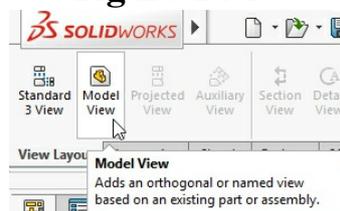
Procedure for Creating a Model/Base View

1. Invoke the **Model View PropertyManager** by clicking on the **Model View** tool, if not invoked, by default.
2. Click on the **Browse** button in the **Part/Assembly to Insert** rollout of the PropertyManager.

Tip:

*If the component or the assembly whose drawing views is to be created appears in the **Open documents** field of the **Part/Assembly to Insert** rollout in the PropertyManager then you can directly select it from this field by double-clicking on it for creating the model view. The **Open documents** field displays a list of models, which are opened in the current session of SOLIDWORKS.*

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3. Select a part or an assembly, whose drawing views are to be created and then click on the **Open** button in the dialog box. A rectangular box appears attached to the cursor, which represents the model/base view of the selected part or assembly.
4. Specify the required settings such as standard view, display style, and scale factor for the drawing view by using the options in the PropertyManager.
5. Click on the drawing sheet to specify the position for the model/base view in the drawing sheet. The model view is created. Also, the **Projected View PropertyManager** appears. By using the **Projected View PropertyManager**, you can create the projected views of the model.
6. After creating the required views, press the ESC key to exit the creating of drawing

views.

Creating a Projected View

Projected views are orthogonal views of an object, which are created by viewing the object from its different projection sides such as top, front, and side. Figure 14.17 and Figure 14.18 shows different projected views of an object.
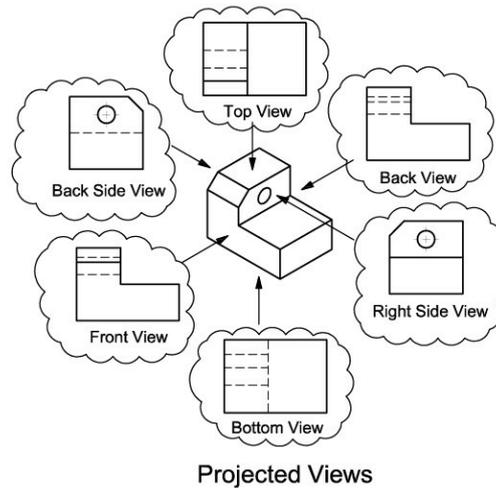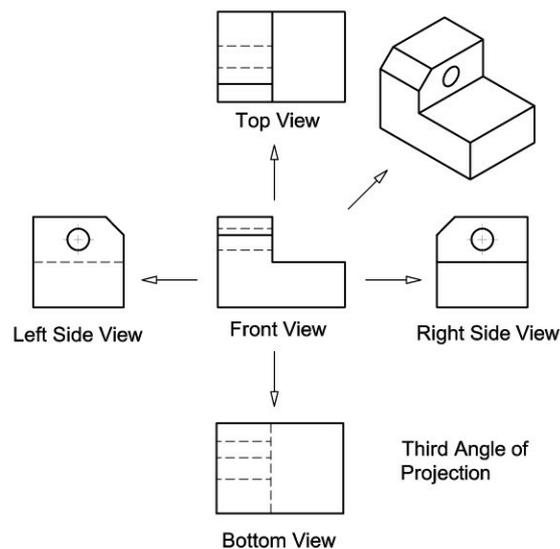
Figure 14.17

Projected Views

Figure 14.18

You can create projected views by using the **Projected View PropertyManager**. As discussed earlier, the **Projected View PropertyManager** is invoked automatically as soon as you create the model/base view of a model by using the **Model View PropertyManager**. You can also invoke this PropertyManager by clicking on the **Projected View** tool in the **View Layout CommandManager**. The options in the **Projected View PropertyManager** are the same as those discussed earlier and are used to create the projected views of a selected model/base view.

Procedure for Creating Projected Views

1. Invoke the **Projected View PropertyManager** by clicking on the **Projected View** tool.
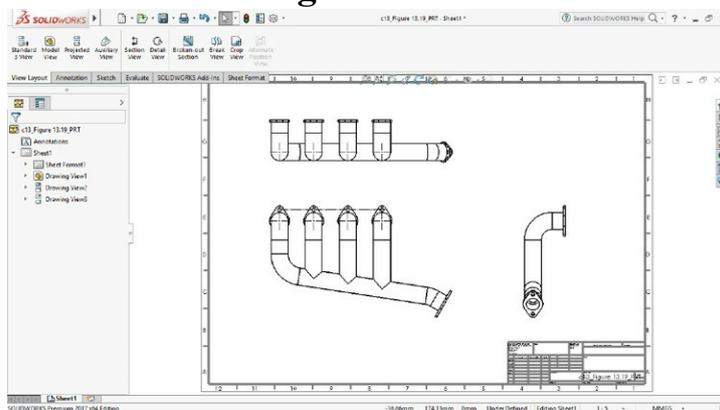2. Select a view as the model/base view whose projected views are to be created.

3. Move the cursor to the required location in the drawing sheet and then click to specify the placement point for the projected view attached. You can continue creating other projected views by clicking the left mouse button in the drawing sheet.
4. Once you have created the projected views, press the ESC key.

Creating 3 Standard Views

In addition to creating drawing views by using the **Model View PropertyManager** and the **Projected View PropertyManager**, you can create three standard orthogonal views: front, top, and side by using the **Standard 3 View** tool of the **View Layout CommandManager**. On clicking this tool, the **Standard 3 View PropertyManager** appears. If the part or assembly, whose drawing views are to be created is displayed in the **Open documents** field of the PropertyManager then double-click on it. The three standard views are created automatically in the drawing sheet, see Figure 14.19. If the model is not displayed in the **Open documents** field, then click on the **Browse** button. The **Open** dialog box appears. In this dialog box, browse to the location where the required part or assembly has been saved and then select it. Next, click on the **Open** button in the dialog box.

Figure 14.19

Working with Angle of Projection

Engineering drawings follow two types of angle of projection: first angle of projection and the third angle of projection. In the first angle of projection, the object is assumed to be kept in the first quadrant and the viewer views the object from the direction shown in Figure 14.20. As the object has been kept in the first quadrant, its projections of views are on the respective planes as shown in Figure 14.20. Now on unfolding the planes of projections, the front view appears on the upper side and the top view appears on the bottom side. Also, the right side view appears on the left and the left side view appears on the right side of the front view, see Figure 14.21. Similarly, in the third angle of projection, the object is assumed to be kept in the third quadrant, see Figure 14.20 and the projection of the front view appears on the bottom and the projection of the top view appears on the top side in the drawing. Also, the right side view appears on the right and the left side view appears on the left of the front view, see Figure 14.22.
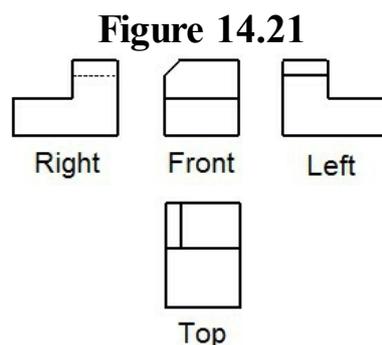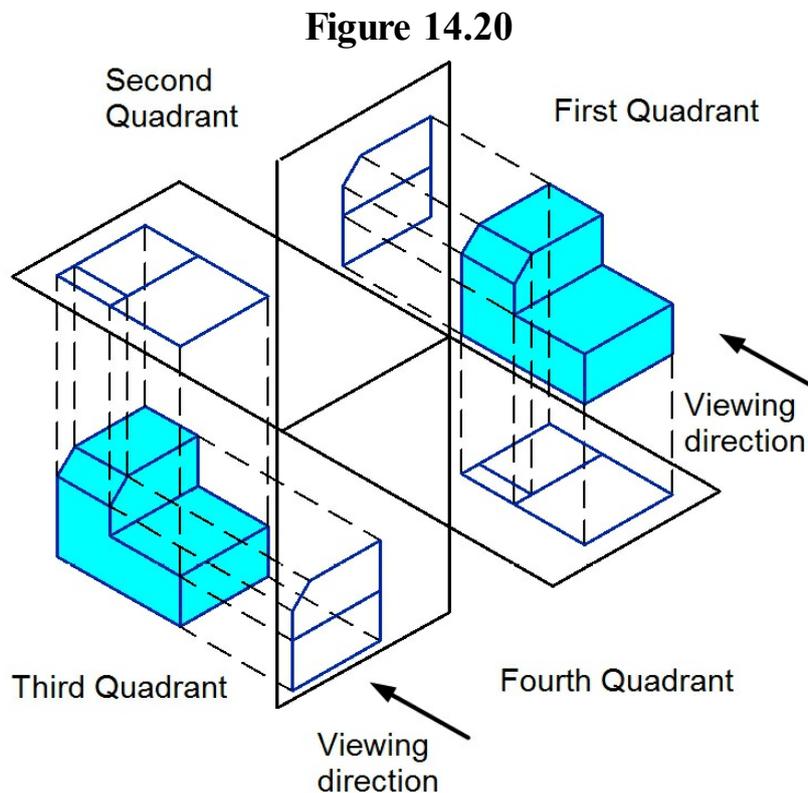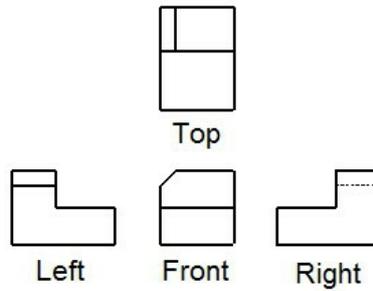
Figure 14.20

Figure 14.21

Figure 14.22

Top

Left Front Right

Defining the Angle of Projection

In SOLIDWORKS, to define the required angle of projection for creating drawing views, select the **Sheet** node from the FeatureManager Design Tree and then right-click. A shortcut menu appears, see Figure 14.23. Next, click on the **Properties** option in the shortcut menu. The **Sheet Properties** dialog box appears, see Figure 14.24. In this dialog box, you can select the type of projection to be followed for creating drawing views by selecting the respective radio button from the **Type of projection** area of the dialog box. Next, click on the **OK** button to accept the change and to exit the dialog box.
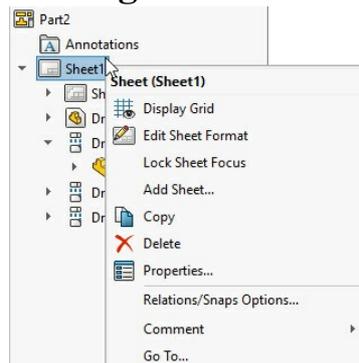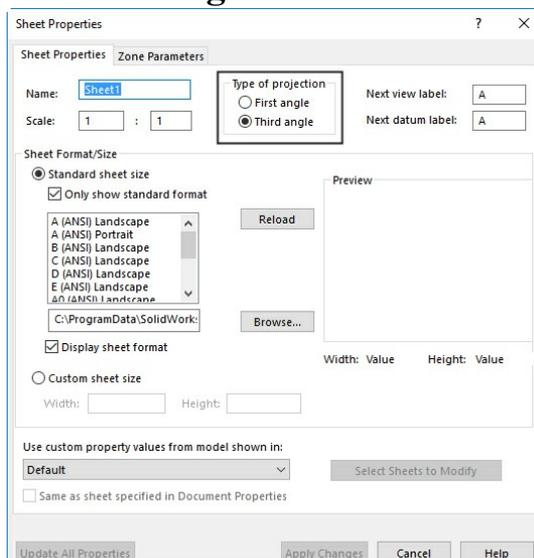
Figure 14.23

Figure 14.24

Editing the Sheet Format

While invoking the Drawing environment, you can select the required sheet size and sheet format. Note that the sheet format contains title block, which has drawing information such as project name, drawn by, checked by, approved by, date, sheet number, and so on. You can create or edit the sheet format such that it matches the standard format of your company. To edit the sheet format of a sheet, select the **Sheet** node in the FeatureManager Design Tree and then right-click to display a shortcut menu, refer to Figure 14.23. Next, click on the **Edit Sheet Format** option in the shortcut menu. The editing mode for editing sheet format is invoked, see Figure 14.25. Now, you can edit or modify the existing text and lines of the title block. Also, you can add new lines and text in the title block by using the sketching tools of the **Sketch CommandManager**. To delete existing lines and text, select the lines and text of the title block to be deleted and then press the DELETE key. To edit the existing text, double-click on the text to be edited. The editing mode is invoked such that the text appears in the edit field. Now, you can write new text in the edit field. Once you have edited the sheet format, click on the Confirmation corner on the upper right corner of the drawing area to exit the editing mode and switch back to the drawing sheet.

Figure 14.25
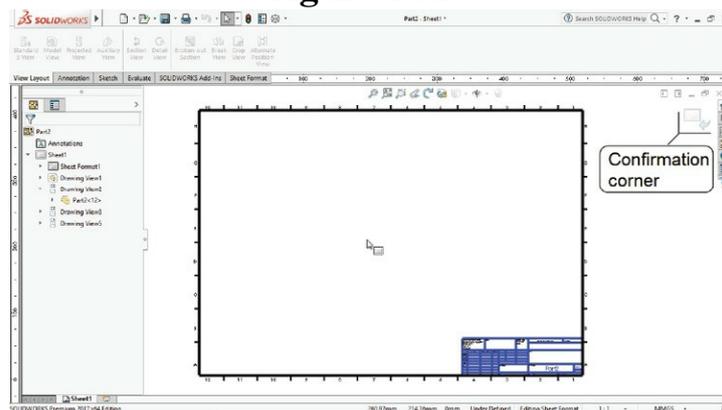
After modifying the sheet format or creating a new sheet format, as per the requirement, you can also save it for future use in other drawings. To save the sheet format, click on **Files > Save Sheet Format** in the SOLIDWORKS menus. The **Save Sheet Format** dialog box appears. In this dialog box, specify the name and location for the sheet format and then click on the **Save** button.

Creating other Drawing Views

In SOLIDWORKS, in addition to creating orthogonal views such as front, top, and right of an object, you can also create the following types of drawing views:
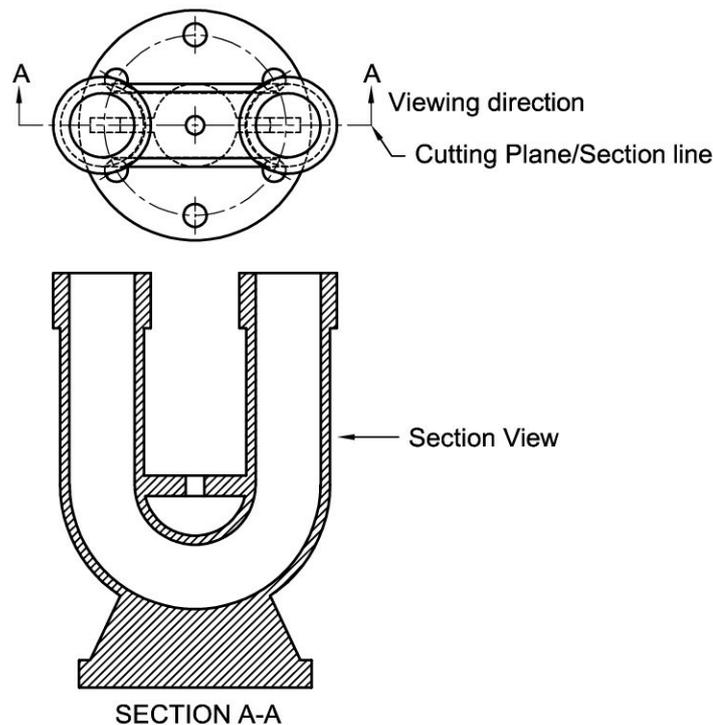
- Section View
- Aligned Section View
- Auxiliary View
- Detail View

- Broken View
- Crop View
- Alternate Position View
- Broken-out Section View

Creating a Section View `Updated`

A section view is created by cutting an object by using an imaginary cutting plane or a section line and then viewing the object from the direction normal to the cutting plane. Figure 14.26 shows a section view created by cutting an object using a cutting plane. A section view is used to illustrate internal features of the object clearly. It also reduces the number of hidden-detail lines, facilitates the dimensioning of internal features, shows cross-section, and so on. In SOLIDWORKS, you can create full section view and half section view by using the **Section View** tool of the **View Layout CommandManager**.

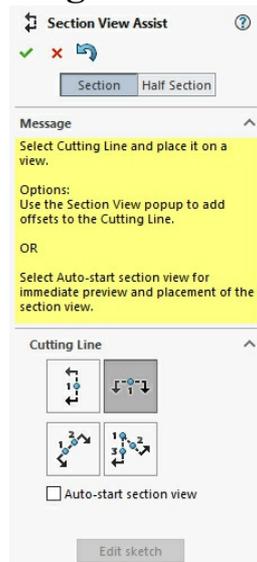**Figure 14.26**

SECTION A-A

Creating a Full Section View

Full section views are the most widely used section views in engineering drawings. In a full section view, an object is assumed to be cut through all its length by an imaginary cutting plane or a section line, refer to Figure 14.26. In SOLIDWORKS, you can create four types of full section views: horizontal, vertical, auxiliary, and aligned by using the **Section View** tool.

To create a full section view, click on the **Section View** tool in the **View Layout CommandManager**. The **Section View Assist PropertyManager** appears, see Figure 14.27.

Figure 14.27

The **Cutting Line** rollout of the PropertyManager is used to select the type of cutting line for creating the respective full section view. You can create a vertical section view, horizontal section view, auxiliary section view, and aligned section view by using the respective buttons of the **Cutting Line** rollout. Different types of full section views are as follows:

Creating a Horizontal/Vertical Section View

To create a horizontal section view, click on the **Horizontal** button in the **Cutting Line** rollout of the PropertyManager. A horizontal cutting line attached to the cursor appears. Next, select the **Auto-start section view** check box in the **Cutting Line** rollout. Now, move the cursor toward an existing drawing view and then click to specify the placement point for the horizontal cutting line. The preview of the horizontal section view appears attached to the cursor. Also, the **Section View PropertyManager** appears with additional options. Note that the direction of the arrows of the cutting section line represents the viewing direction. You can reverse the viewing direction by clicking on the **Flip Direction** button in the PropertyManager. In SOLIDWORKS 2017, you can emphasize the outlines of the cutting faces such that they appear thicker than the object lines in the section view by selecting the **Emphasize outline** check box in the **Section View** rollout of the PropertyManager. Also, if needed, you can scale the hatch pattern of the section view by selecting the **Scale Hatch Pattern** check box of the **Section View** rollout in the PropertyManager. Next, click to specify the placement point for the horizontal section view at the

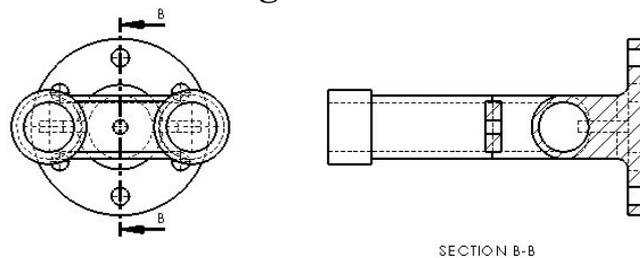required location on the drawing sheet. The horizontal section view is created, refer to Figure 14.26.

Figure 14.28

Similar to creating a horizontal section view, you can create a vertical section view by using the **Vertical** button of the **Cutting Line** rollout. Figure 14.29 shows a vertical section view created.

Figure 14.29

SECTION B-B

Note that if you are creating a section view of an assembly or a component having rib features, then soon after specifying the placement point for the section line, the **Section View** dialog box appears, see Figure 14.30.

Figure 14.30
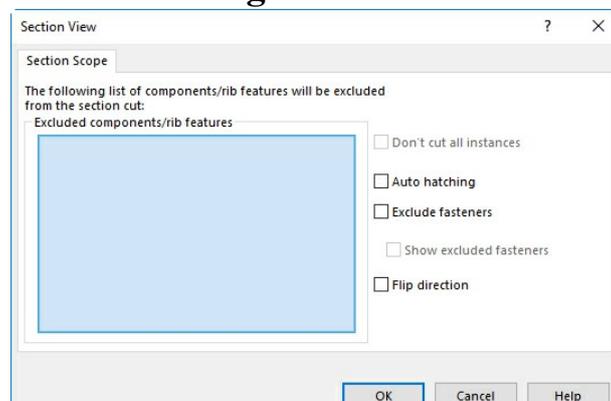
In the **Section View** dialog box, the **Excluded components/rib features** field is activated. As a result, you can select components like fasteners to be excluded from the section cut. You can also select features like ribs to be excluded from the section

cut. Once you have selected components/features to be excluded, click on the **OK** button in the dialog box. The preview of the section view appears attached to the cursor, without cutting the selected components/features. Next, click to specify the placement point for placing the section view in the drawing sheet.


Tip:

You can set the line font for the emphasized outlines of the cutting faces in the section view. To set the font for the emphasized outlines, click on the ***Options*** *tool in the* ***Standard*** *toolbar to invoke the* ***System Options*** *dialog box. Next, click on the* ***Document Properties*** *tab in the dialog box and then click on the* ***Line Font*** *option. Next, select the* ***Emphasized Section Outline*** *option in the* ***Type of edge*** *area of the dialog box and then set the font/style for the emphasized outlines using the options that appear in the dialog box.*

Creating a Auxiliary Section View

An auxiliary section view is created by cutting an object using a sight line which is not parallel to any of the principal projection planes: frontal, horizontal, or profile and then viewing the object from the direction normal to the sight line, see Figure 14.31.

To create an auxiliary section view, click on the **Section View** tool to invoke the **Section View Assist PropertyManager**. Next, click on the **Auxiliary** button in the **Cutting Line** rollout. A section line is attached to the cursor. Move the cursor to an existing drawing view and then click to specify the placement point. The preview of the auxiliary section view is attached to the cursor. If the **Section View** Pop-up toolbar appears soon after specifying the placement point then click on its green tick mark to display the preview of the auxiliary section view. You can reverse the default viewing direction by clicking on the **Flip Direction** button in the PropertyManager. Next, click to specify the placement point for the auxiliary section view at the required location in the drawing sheet. The auxiliary section view is created, see Figure 14.31.

Figure 14.31
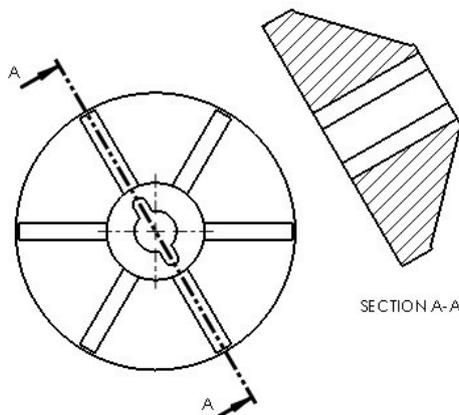
SECTION A-A

Creating a Aligned Section View

An aligned section view is created by cutting an object using the cutting section line, which comprises of two non-parallel lines and then straightening the cross-section by

revolving it around the center point of the section line, see Figure 14.32.

Procedure for Creating a Aligned Section View
1. Invoke the **Section View Assist PropertyManager** by clicking on the **Section View** tool.
2. Click on the **Aligned** button in the **Cutting Line** rollout. A cutting section line, which comprises of two non-parallel lines is attached to the cursor.
3. Move the cursor toward an existing drawing view as the parent view for creating the aligned section view.
4. Click to specify the center point for the cutting section line, see Figure 14.32.
5. Move the cursor for a little distance and then click to specify the position for the first cutting line, see Figure 14.32.
6. Move the cursor for a little distance and then click to specify the position for the second cutting line, see Figure 14.32. The preview of the aligned section view is attached to the cursor.
7. Move the cursor to the required location and then click to specify the placement point for the aligned section view in the drawing sheet, see Figure 14.32.
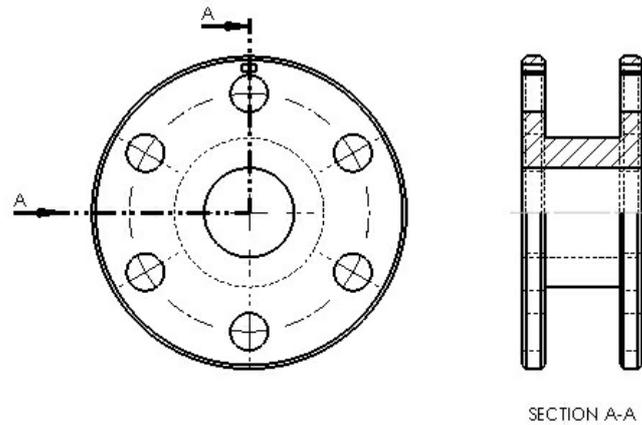8. Press ESC to exit.

Figure 14.32
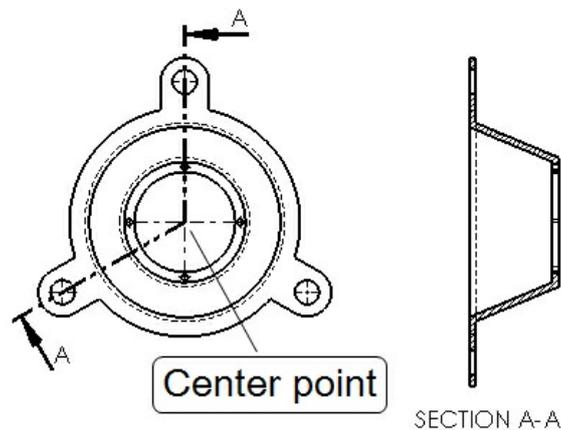
Creating a Half Section View
A half section view is created by cutting an object using an imaginary cutting plane or section line that passes halfway through the object, see Figure 14.33. In SOLIDWORKS, you can create half section view by using the **Section View** tool.
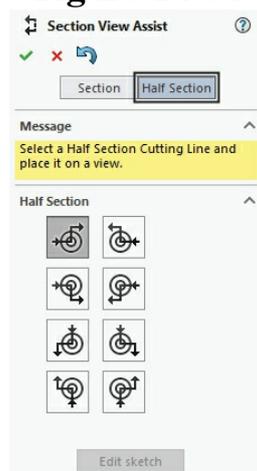
Figure 14.33

SECTION A-A

Procedure for Creating a Half Section View

1. Click on the **Section View** tool. The **Section View Assist PropertyManager** appears.

2. Click on the **Half Section** tab available at the top of the PropertyManager, see Figure 14.34. The options to create pre-defined shape of half section views appears in the PropertyManager.
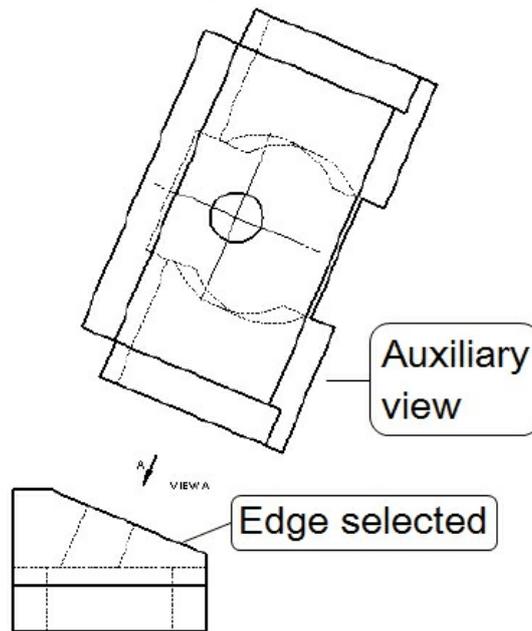
Figure 14.34

3. Click on the required button in the **Half Section** rollout of the PropertyManager. The respective half section cutting line is attached to the cursor.

4. Move the cursor over an existing view in the drawing sheet and then click to specify the placement point for the half section cutting line at the required location. The preview of the half section view is attached to the cursor.

5. If you want to flip the viewing direction, click on the **Flip Direction** button in the **Section Line** rollout of the PropertyManager, else skip this step.

6. Click to specify the position for the half section view at the required location in the drawing sheet. The half section view is created. Next, press the ESC key.

Creating an Auxiliary View

An auxiliary view is a projected view, which is created by projecting the edges of an object normal to the edge of an existing drawing view, see Figure 14.35. You can create an auxiliary view by using the **Auxiliary View** tool in the **View Layout**

CommandManager.

Figure 14.35

Auxiliary view

Edge selected

Procedure for Creating an Auxiliary View
1. Click on the **Auxiliary View** tool in the **View Layout CommandManager**.
2. Select an edge of an existing drawing view, see Figure 14.35. The preview of the auxiliary view is attached to the cursor.
3. Click to specify the position of the auxiliary view in the drawing sheet, see Figure 14.35.
4. Press ESC to exit.

Creating a Detail View Updated
A detail view is used to show a portion of an existing drawing view in an enlarged scale, see Figure 14.36. You can define the portion of an existing drawing view to be enlarged by creating a circle or a closed sketch. You can create the detail view of a portion of an existing view by using the **Detail View** tool.

Figure 14.36
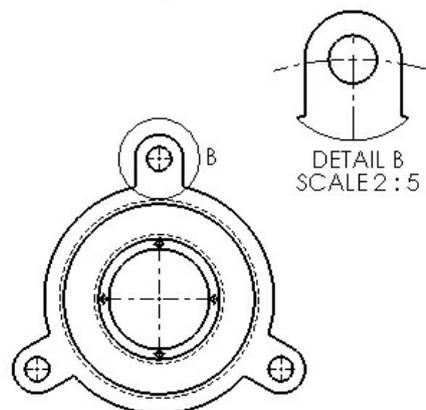
DETAIL B
SCALE 2 : 5

Procedure for Creating a Detail View
1. Click on the **Detail View** tool in the **View Layout CommandManager**. The **Detail**

View PropertyManager appears. Also, you are promoted to draw a circle to define the portion of an existing drawing view to be enlarged.

2. Draw a circle around the portion of an existing view to be enlarged. The enlarged view of the portion of the existing view is attached to the cursor. Also, the **Detail View PropertyManager** appears with additional options.

3. You can increase or decrease the default scale factor for the attached detail view by using the options of the **Scale** rollout in the PropertyManager, as required.

Note:

*In SOLIDWORKS 2017, you can display the detail view with full outline around it by selecting the **Full outline** check box in the **Detail View** rollout of the PropertyManager, see Figure 14.37. Also, you can display the detail view with no outline by selecting the **No outline** check box in the **Detail View** rollout of the PropertyManager, see Figure 14.37. Moreover, in SOLIDWORKS 2017, you can also display the detail view with a jagged outline by selecting the **Jagged outline** check box in the **Detail View** rollout, see Figure 14.37.*
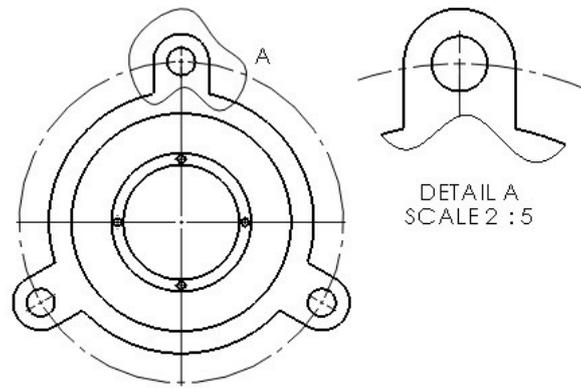

Figure 14.37
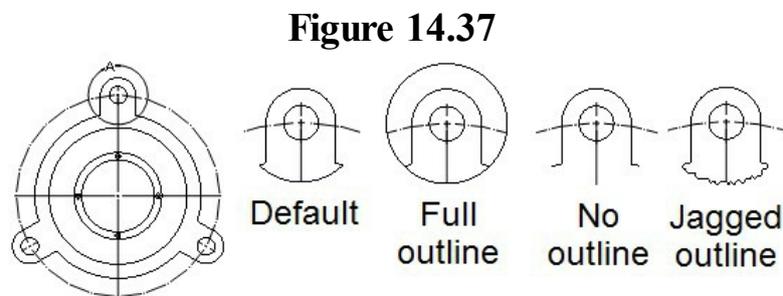
4. After specifying the required properties for the detail view in the PropertyManager, move the cursor to the required location in the drawing sheet and then click to specify the placement point in the drawing sheet. The detail view is created.

5. Press the ESC key to exit.

Note:

*In addition to defining the portion of a view to be enlarged by drawing a circle, you can also use a closed sketch, which defines the portion of a view to be enlarged. To do so, before invoking the **Detail View** tool, first select an existing view and then draw a closed sketch by using the sketching tools of the **Sketch CommandManager**. Once the closed sketch has been drawn, select it and then click on the **Detail View** tool. Figure 14.38 shows the detail view created by a closed sketch, which is drawn by using the **Spline** tool.*
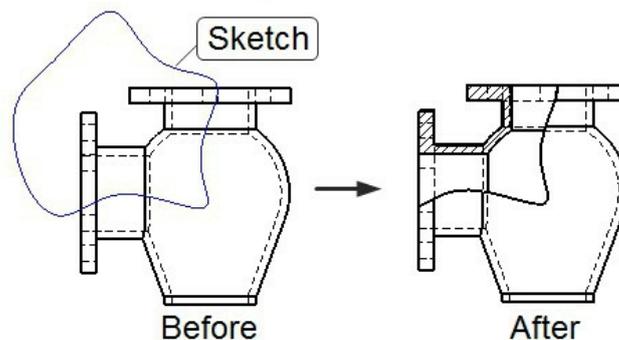

Figure 14.38

DETAIL A
SCALE 2 : 5

Creating a Broken-out Section View

A broken-out section view is created by removing the portion of an existing view up to a specified depth in order to view inner details of the object, see Figure 14.39. You can define the portion of an existing view to be removed by drawing a closed sketch.
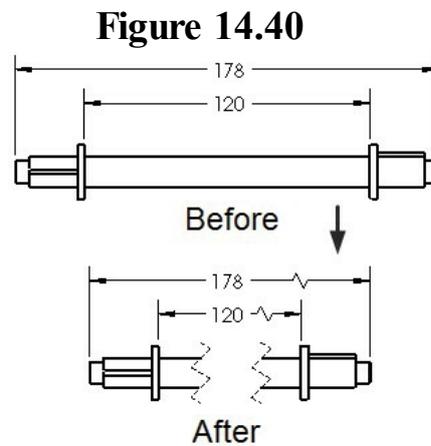
Figure 14.39

Sketch

Before   After

Procedure for Creating a Broken-out Section

1. Click on the **Broken-out Section** tool in the **View Layout CommandManager**. The **Spline** tool is invoked to create a closed sketch for defining the portion to be removed.
2. Move the cursor over the existing drawing view whose portion is to be removed up to a specified depth.
3. Draw a closed profile around the portion to be removed. As soon as you draw a closed profile, the **Broken-out Section PropertyManager** appears.
4. Enter the depth value in the **Depth** field of the PropertyManager up to which the material is to be removed. Alternatively, you can select an edge or an axis of the view to define the depth by using the **Depth Reference** field of the PropertyManager.
5. Select the **Preview** check box to display the preview of the broken-out section view in the drawing sheet.
6. Click on the green tick mark ☑ in the PropertyManager to accept the defined settings and to exit the PropertyManager. The broken-out section view is created, refer to Figure 14.39.

Creating a Break View `Updated`

A break view is created by breaking an existing view using a pair of break lines such that the portion existing between the breaking lines is removed, see Figure 14.40. A break view is used to display a large scaled view on a small scale sheet by removing a portion of the view that has the same cross-section, see Figure 14.40.

Figure 14.40

Note:

The dimension applied to a break view represents its actual dimension, see Figure 14.40. It is evident from this figure that even on breaking the view, the dimension value associated with it remains the same. You will learn more about applying dimensions later in this chapter.

Procedure for Creating a Break View
1. Click on the **Break View** tool. The **Broken View PropertyManager** appears.
2. Move the cursor over an existing drawing view to be broken and then click on it. The first vertical or horizontal break line is attached to the cursor.

Note:

*The display of break line (vertical or horizontal) depends upon whether the **Add vertical break line** or the **Add horizontal break line** button is activated in the PropertyManager. If the **Add vertical break line** button is activated, a vertical break line appears and if the **Add horizontal break line** button is selected, a horizontal break line appears.*

3. Make sure that the required button: **Add vertical break line** or **Add horizontal break line** is activated in the **Broken View PropertyManager**.
4. Specify the placement point for the first break line by clicking the left mouse button on the required location of the drawing view. The second break line is attached to the cursor.
5. Specify the placement point for the second break line in the view. The view is broken and the portion inside the break lines is removed, refer to Figure 14.40.

Note:

*You can also control the gap between the break lines by using the **Gap size** field of the PropertyManager. You can also select a required style or type*

6. Click on the green tick mark ☑ in the PropertyManager to confirm the creation of break view and to exit the PropertyManager.

Creating a Crop View Updated

A crop view is created by cropping an existing view by using a closed sketch in such a way that only the portion that is lying inside the closed sketch is retained in the view, see Figure 14.41. You can create a crop view by using the **Crop View** tool. Note that to create a crop view, you first need to create a closed sketch by using the sketching tools of the **Sketch CommandManager**.
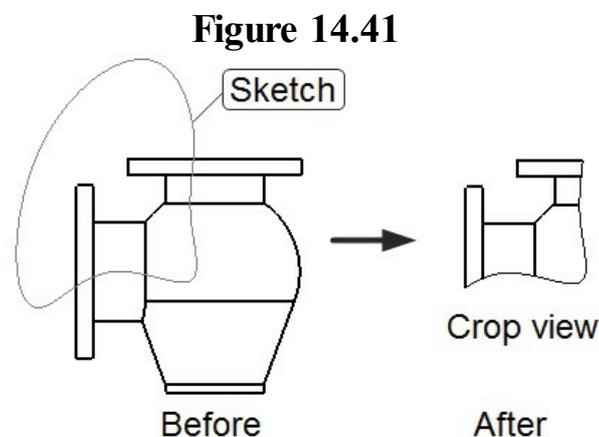
Figure 14.41

Procedure for Creating a Crop View
1. Select the view to be cropped.
2. Draw the closed sketch around the portion of the view to be cropped, refer to Figure 14.41.
3. Make sure that the sketch created is selected.
4. Click on the **Crop View** tool. The crop view is created by retaining only the portion that lies inside the closed sketch. Next, press the ESC key.

Note:

*In SOLIDWORKS 2017, after creating a crop view, you can display it with no outline or with jagged outline. To display a crop view with no outline or jagged outline, select the crop view in the drawing sheet. The **Drawing View PropertyManager** appears. In this PropertyManager, expand the **Crop View** rollout and then select the required check box: **No outline** or **Jagged outline**.*

Creating the Alternate Position View

In SOLIDWORKS, you can show or create the alternate position of an assembly in a drawing view by using the **Alternate Position View** tool, see Figure 14.42. You can also create multiple positions of an assembly in a drawing view by using this tool.

Figure 14.42
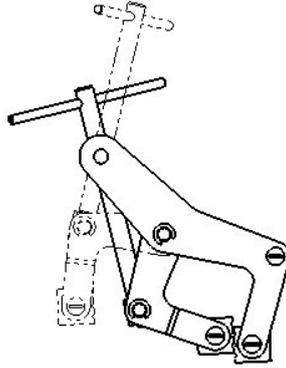
Procedure for Creating the Alternative View

1. Click on the **Alternative Position View** tool in the **View Layout CommandManager**. The **Alternate Position PropertyManager** appears.
2. Select a drawing view of an assembly. The **Alternate Position PropertyManager** is modified.

3. Make sure that the **New configuration** radio button is selected in the **Configuration** rollout.
4. Accept the default settings and then click on the green tick mark ☑ in the PropertyManager. The Assembly environment is invoked with the display of the **Move Component PropertyManager** on the left of the graphics area.

5. Drag to rotate or move the components of the assembly whose alternate position is to be created to the desired position.
6. Once the desired position of the assembly components has been achieved, click on the green tick mark ☑ in the **Move Component PropertyManager**. The Drawing environment is invoked again and the alternative position of the assembly components is created in the drawing view, refer to Figure 14.42. Note that the alternative position of the components is displayed in dotted lines.
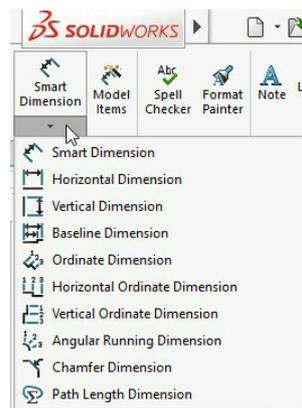
Applying Dimensions

After creating various drawing views of a part or an assembly, you need to apply dimensions to them. In SOLIDWORKS, you can apply two types of dimensions: reference dimensions and driving dimensions. Reference dimensions are applied manually by using the dimension tools such as **Smart Dimension**, **Horizontal Dimension**, and **Vertical Dimension** whereas driving dimensions are generated automatically by retrieving the model dimensions. You can apply driving dimensions by using the **Model Items** tool of the **Annotation CommandManager**. The methods of applying both types of dimensions are as follows:

Applying Reference Dimensions

You can apply reference dimensions by using the dimension tools such as **Smart Dimension**, **Horizontal Dimension**, and **Vertical Dimension** available in the **Dimension** flyout of the **Annotation CommandManager**, see Figure 14.43.

Figure 14.43

Applying reference dimension is the manual method of applying dimensions to drawing views and is same as discussed in the Sketching environment while dimensioning sketch entities. For example, to apply dimension to a linear edge in a view, click on the **Smart Dimension** tool and then select the edge. The dimension value of the selected edge is attached to the cursor. Next, place the dimension to the required location in the drawing sheet.
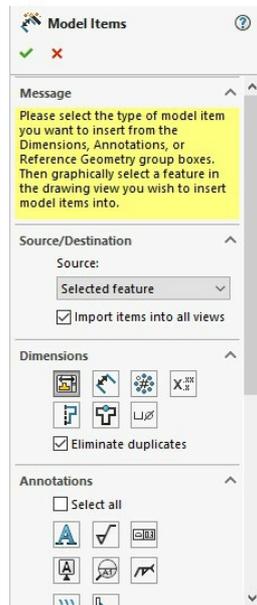
Applying Driving Dimensions

Driving dimensions are applied automatically in drawing views by retrieving the model dimensions, which are applied in the sketches and features of the model. Note that on modifying a driving dimension, the respective sketch or feature of the model is also modified and vice-versa.

To apply driving dimensions, click on the **Model Items** tool in the **Annotation CommandManager**. The **Model Items PropertyManager** appears, see Figure 14.44. By using the options in the PropertyManager, you can retrieve dimensions, symbols,

annotations, and other elements of the model in the drawing views. Some of the options of this PropertyManager are as follows:

Figure 14.44

Source/Destination

The options in the **Source/Destination** rollout of the PropertyManager are used to select features or the model as the source and destination for retrieving and applying dimensions, symbols, annotations. By default, the **Selected feature** option is selected in the **Source** drop-down list. As a result, you can select a feature of the model in a drawing view as the source and destination for retrieving and applying dimensions, respectively. As soon as you select a feature in a drawing view, the dimensions of the selected feature are retrieved and applied in the drawing views. In Figure 14.45, the driving dimensions are applied to a hole feature by selecting it as the source and destination feature in the drawing view.

Figure 14.45
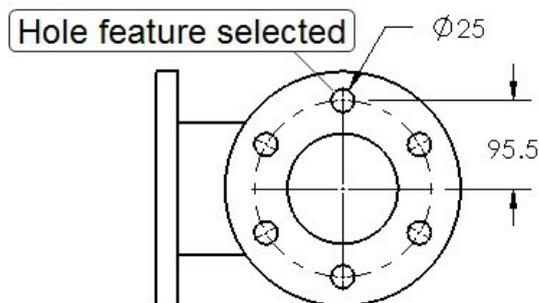
On selecting the **Entire model** option in the **Source** drop-down list, all dimensions, symbols, and annotations, which are applied in the model, get retrieved and applied in the drawing views. By default, the **Import items into all views** check box is selected in the **Source/Destination** rollout. As a result, items such as dimensions and annotations apply to all views present in the drawing sheet.

Dimensions

The buttons in the **Dimensions** rollout of the PropertyManager are used to select the type of dimensions to be retrieved from the model and applied in the drawing view. On selecting the **Eliminate duplicates** check box in this rollout, applying duplicate dimensions in the drawing views will be eliminated.

Annotations

The buttons in the **Annotations** rollout are used to select the type of annotations to be retrieved from the model and applied in the drawing view. You can click on the buttons to activate them in order to retrieve respective annotations in the drawing views. If you select the **Select all** check box, all the buttons of this rollout get activated, automatically.

Reference Geometry

The buttons in the **Reference Geometry** rollout are used to select the type of reference geometry such as planes, axis, and origin to be retrieved from the model and applied in the drawing views.

After selecting the required options in the **Model Items PropertyManager**, click on the green tick mark in the PropertyManager. The respective dimensions, annotations, and reference geometries are retrieved and applied in the drawing views, see Figure 14.46.

Figure 14.46
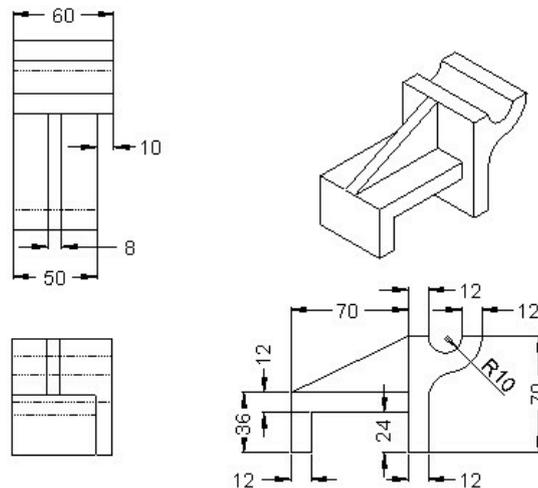
Note:

Sometimes the driving dimensions applied in the drawing views do not appear in the required position, nor does they maintain uniform spacing in the drawing views. You can drag the dimensions and place in the required position for maintaining proper spacing between them.

Modifying the Driving Dimension

On modifying a driving dimension in the Drawing environment, the same modification is reflected in the model as well. To modify a driving dimension, double-click on it in the

drawing view. The **Modify** dialog box appears, see Figure 14.47. Enter the new dimension value in the field of this dialog box and then click on the green tick mark. The respective dimension and feature of the model are modified accordingly.

Figure 14.47
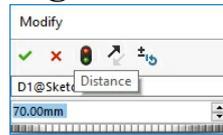
Controlling the Dimension and the Arrow Style

In SOLIDWORKS, you can control dimension and arrow styles such as dimension font, dimension height, and arrow height by using the options in the **Document Properties - Dimensions** dialog box. To invoke this dialog box, click on the **Options** tool in the **Standard** toolbar. The **System Options - General** dialog box appears. Next, click on the **Document Properties** tab in the dialog box and then click on the **Dimensions** option. The **Document Properties - Dimensions** dialog box appears, see Figure 14.48.

In this dialog box, click on the **Font** button in the **Text** area. The **Choose Font** dialog box appears. By using this dialog box, you can specify the required font, style, and height for dimensions text. Next, click on the **OK** button in the **Choose Font** dialog box.

You can control the dimension arrow height by using the options in the **Arrows** area of the **Document Properties - Dimensions** dialog box, see Figure 14.48.

Figure 14.48
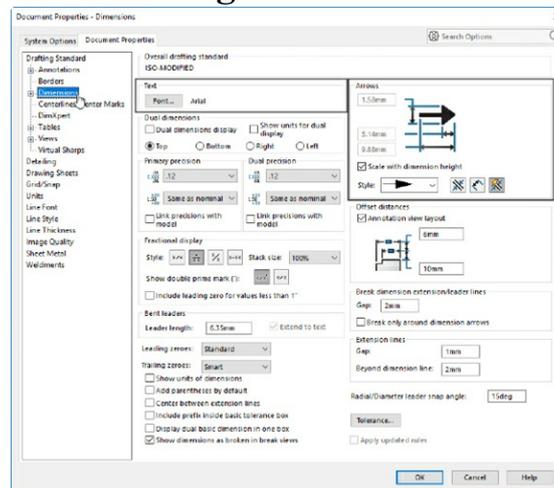
Adding Notes

In SOLIDWORKS, you can add notes on a drawing sheet by using the **Note** tool. Generally, adding notes in drawings is used to convey/provide additional information that are not available in the drawing views.

To add a note, click on the **Note** tool in the **Annotation CommandManager**. The **Note PropertyManager** appears, see Figure 14.49. Also, a rectangular box is attached to the

cursor. Now, specify the required settings such as text style, text format, type of leader, leader style, so on for the note by using the options in the PropertyManager. Once the required settings for the note have been specified, move the cursor over an entity of a drawing view to add the note. The preview of the note with the leader attached to the entity of the drawing view appears in the drawing sheet. Next, click on the entity for adding the note. The leader arrow is attached to the selected entity. Now, move the cursor to the required location and then click to place the note. An edit box and the **Formatting** toolbar appear. Now, you can write text as note in the edit box. You can use the **Formatting** toolbar to control the formatting of the text such as font, style, height, and alignment. Next, click anywhere on the drawing sheet. The note is added in the selected entity of the drawing view, see Figure 14.50. You can also add note anywhere in the drawing sheet without selecting any entity.
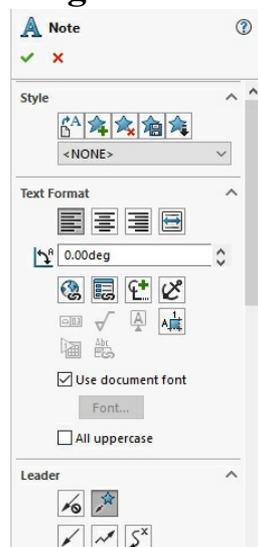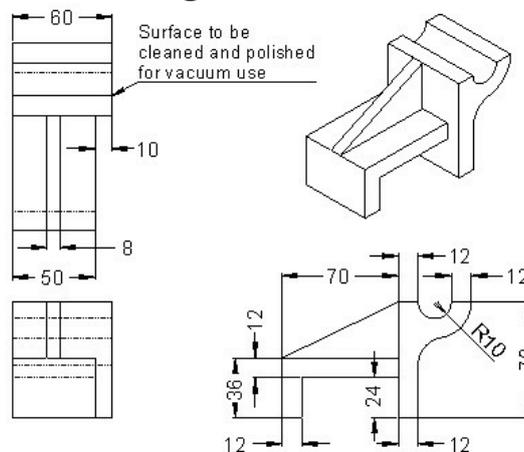
Figure 14.49
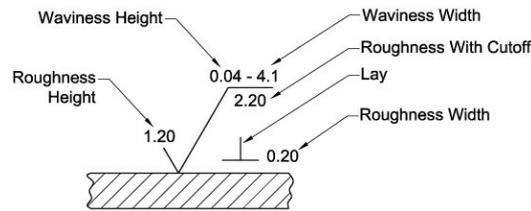

Figure 14.50

Adding the Surface Finish Symbol

In SOLIDWORKS, you can add surface finish symbol to specify the surface texture/finish for a face of a model. A surface finish symbol has three components: surface roughness, waviness, and lay, see Figure 14.51. Specifications for the surface

finish given in a surface finish symbol are used to machine the respective surface of the object. You can add surface finish symbol to an edge of a surface in a drawing view by using the **Surface Finish** tool.
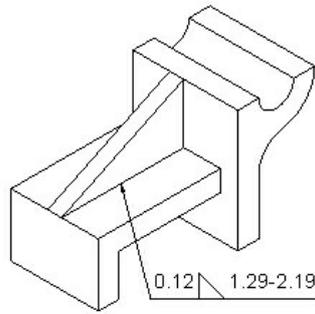
Figure 14.51

To add a surface finish symbol, click on the **Surface Finish** tool in the **Annotation CommandManager**. The **Surface Finish PropertyManager** appears, see Figure 14.52. Also, the default selected surface finish symbol is attached to the cursor. Select the required type of surface finish symbol to be added from the **Symbol** rollout of the PropertyManager. Next, in the **Symbol Layout** rollout, specify the required specification for the surface finish (roughness, waviness, and lay) in the respective fields.

Figure 14.52
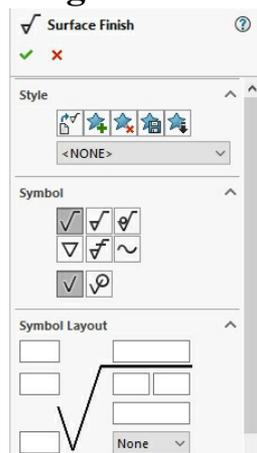
If needed, you can rotate the surface finish symbol at an angle by using the options in the **Angle** rollout of the PropertyManager. By using the options of the **Leader** rollout, you can select the type of leader to be attached to the surface finish symbol. Once you have specified the surface finish specification, move the cursor over the required edge in a drawing view for applying the surface finish symbol and then click on the edge when it highlights. The surface finish symbol is applied and attached to the selected edge, see Figure 14.53.

Figure 14.53

Adding the Weld Symbol

A weld symbol is added in a drawing view in order to represent the welding specification used while welding two parts of a model. To add a weld symbol, click on an edge of a model in the drawing view for adding a welding symbol, and then click on the **Weld Symbol** tool in the **Annotation CommandManager**. The **Properties** dialog box appears, see Figure 14.54. Also, the **Weld Symbol PropertyManager** appears on the left of the drawing sheet. By using the **Properties** dialog box, you can specify the welding properties to be included in the weld symbol. Note that the availability of options in the **Properties** dialog box for specifying the welding properties depends upon the type of drafting standard selected in the **Document Properties - Drafting Standard** dialog box. Once you have specified the welding properties, click on the **OK** button in the dialog box. The weld symbol is added in the selected edge of the model, see Figure 14.55.

Figure 14.54

Figure 14.55

0.12 1.29-2.19

Adding the Hole Callout `Updated`

In SOLIDWORKS, you can add the hole callout to a hole in a drawing view. A hole
callout contains hole specifications such as diameter and type of hole, see Figure 14.56.
Note that on modifying the hole parameters of a model in the Part modeling environment,
the respective hole callout gets updated accordingly in the Drawing environment.

Figure 14.56


2X Ø 3.26 THRU ALL
⌴ Ø 6.75 ▼ 1.73
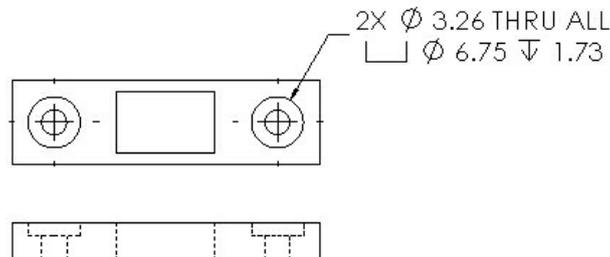
To add a hole callout to a hole in a drawing view, click on the **Hole Callout** tool in the
Annotation CommandManager. The symbol of hole callout is attached to the cursor.
Move the cursor over the hole for adding the hole callout in a drawing view and then
click the left mouse button on the hole when it highlights. The preview of the hole
callout is attached to the cursor. Now, move the cursor to the required location and then
click to specify the placement point for the hole callout, see Figure 14.56. In
SOLIDWORKS 2017, a hole callout also includes mirrored holes in the count.

Adding the Center Mark

Center marks are used as references for dimensioning circular edges, slot edges, or
circular sketch entities in drawing views. You can add center marks on circular edges,
slot edges, or circular sketch entities by using the **Center Mark** tool. To add center
marks, click on the **Center Mark** tool in the **Annotation CommandManager**. The
**Center Mark PropertyManager** appears, see Figure 14.57. By using this

PropertyManager, you can add center marks automatically as well as manually in a drawing view.
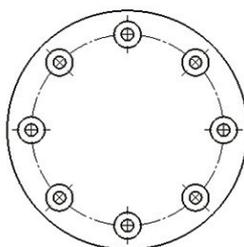
Figure 14.57
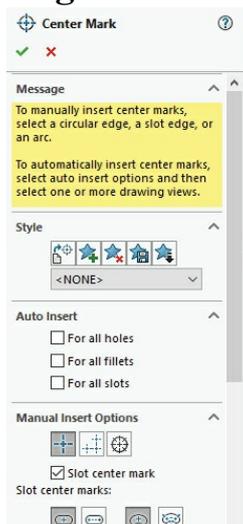
To add center marks automatically to all holes, fillets, slots, or all of them in a drawing view, select the respective check box or check boxes such as **For all holes** and **For all fillets** in the **Auto Insert** rollout of the PropertyManager. Next, select a drawing view for adding center marks. The center marks are added automatically in the drawing view, depending upon the check box or check boxes selected. In the automatic method of adding center marks, you can further control the connection among center marks by using the check boxes: **Connection lines**, **Circular lines**, **Radial lines**, and **Base center mark** in the **Options** area of the **Auto Insert** rollout, see Figures 14.58 through 14.61. Note that the **Options** area appears as soon as you select a check box in the **Auto Insert** rollout of the PropertyManager, see Figure 14.62.
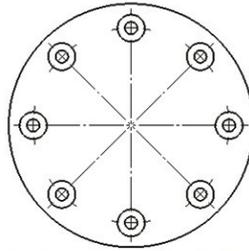
Figure 14.58
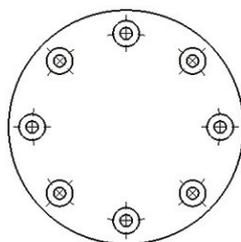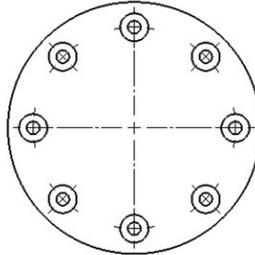

Center mark with connection lines

Figure 14.59

Center mark with circular lines
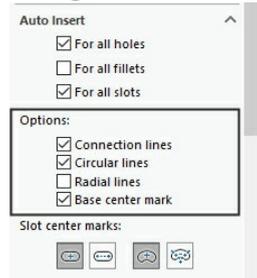
Figure 14.60

Center mark with radial lines

Figure 14.61

Center mark with base center mark

Figure 14.62

To add center marks manually, invoke the **Center Mark PropertyManager** and then move the cursor over a circular edge, a slot edge, or a circular sketch entity in a drawing view. Next, click on the entity when it highlights. The center mark is added. Similarly, you can add center marks to the other entities of drawing views manually. In the manual method of adding center marks, you can select the type of center marks to be added by activating the respective button in the **Manual Insert Options** rollout of the PropertyManager, see Figure 14.63.

Figure 14.63
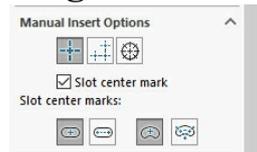
Once you have added center marks either by manual or automatic method, click on the green tick mark ☑ in the PropertyManager.

Adding Centerlines

Centerlines are used as references for dimensioning circular cut features and hole features in drawing views, see Figure 14.64. In SOLIDWORKS, you can add centerline between two linear edges that represent the edges of a circular cut or hole feature in a drawing view by using the **Centerline** tool, see Figure 14.64.

Figure 14.64
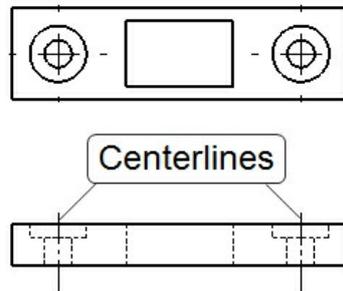
To add center lines, click on the **Centerline** tool in the **Annotation CommandManager**. The **Centerline PropertyManager** appears. Select two linear edges one by one by clicking the left mouse button. The centerline is added at the center of the two selected edges, see Figure 14.64. You can also select two sketch segments, or a single cylindrical, conical, toroidal, or swept feature for adding centerline.

Creating the Bill of Material (BOM)

After creating all the required drawing views of an assembly in the Drawing environment, you need to create Bill of Material. A Bill of Material (BOM) contains all the required information such as the number of parts used in an assembly, part number, quantity of each part, material, and so on of an assembly. Since Bill of Material (BOM) contains all the information, it serves as a primary source of communication between the manufacturer and the vendors as well as the suppliers.

To create Bill of Material (BOM), click on the down arrow below the **Tables** tool in the **Annotation CommandManager**. A flyout appears, see Figure 14.65. In this flyout, click on the **Bill of Materials** tool. The **Bill of Materials PropertyManager** appears. Next, click on the drawing view of the assembly whose Bill of Material has to be created. The **Bill of Materials PropertyManager** gets modified and appears as shown in Figure 14.66. The options of this PropertyManager are used to set the parameters for the Bill of Material. Some of the options of this PropertyManager are as follows:
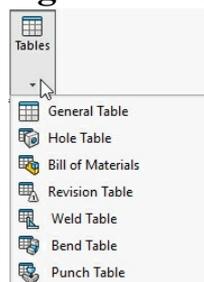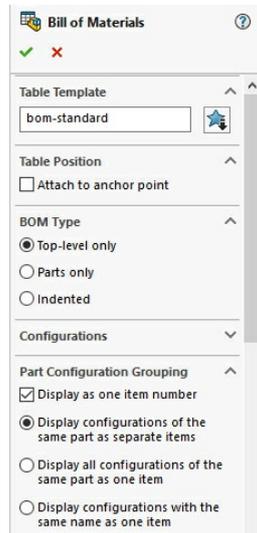
Figure 14.65

Figure 14.66

Table Template

The **Table Template** rollout is used to specify template for the Bill of Material (BOM). By default, bom-standard template is selected in this rollout. You can select the template other than the default one by clicking on the **Open table template for Bill of Materials** button of this rollout. As soon as you click on this button, the **Open** dialog box appears such that all the template files are displayed in it. You can select a required template for the BOM in this dialog box and then click on the **Open** button.

Table Position

The **Table Position** rollout is used to specify the position for the BOM table in the drawing sheet. By default, the **Attach to anchor point** check box is unchecked in this rollout. As a result, on clicking the green tick mark of the PropertyManager, the BOM gets attached to the cursor and you need to define its position in the drawing sheet by specifying the placement point. However, on selecting the **Attach to anchor point** check box, the BOM is placed directly in the drawing sheet such that the top left corner of the BOM is attached to the anchor point present in it.

Note:

*You can define the position of the anchor point in the drawing sheet, as required. To define the anchor point position in the drawing sheet, select the **Sheet** node in the FeatureManager Design Tree and then right-click to display a shortcut menu. Next, click on the **Edit Sheet Format** option in the shortcut menu. The editing mode for defining the anchor point location is invoked, see Figure 14.67. Now, click on an existing vertex or a point of the drawing sheet, see Figure 14.67 and then right-click to display a shortcut menu. In this shortcut menu, select **Set as Anchor > Bill of Materials**, see Figure 14.68. The selected vertex/point is defined as the anchor point for the BOM. In addition to selecting an existing vertex or a point, you can create a new sketch point by using the **Point** tool of the **Sketch CommandManager** and then define that point as the anchor point. Once you have defined the anchor point, exit the exiting mode by clicking on the Confirmation corner available at the upper right corner of the drawing sheet.*
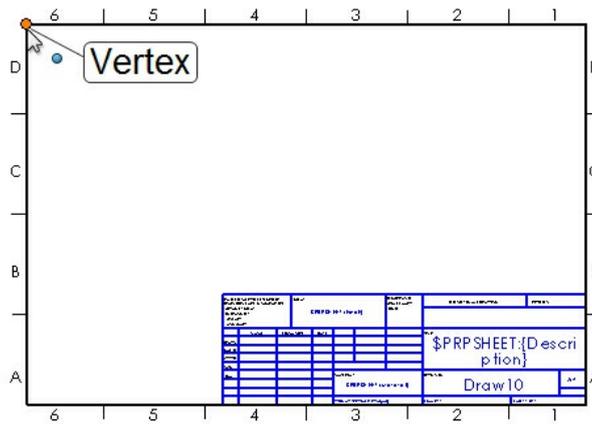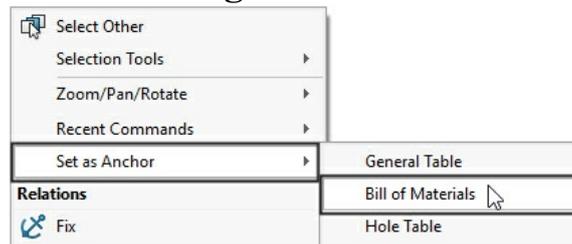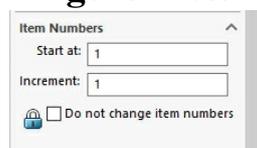
Figure 14.67

Figure 14.68

Item Number

The **Start at** field of the **Item Numbers** rollout in the PropertyManager is used to specify the start number for the count of components, see Figure 14.69. By default, 1 is entered in this field. As a result, the counting of components starts from number 1 in the BOM. In the **Increment** field, you can specify the incremental value for the count of components. Note that on selecting the **Do not change item numbers** button of this rollout, the components count/numbers assigned are locked.
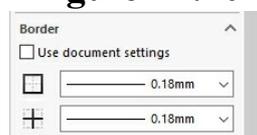
Figure 14.69

Border

The options of the **Border** rollout of the PropertyManager are used to define the thickness of the Bill of Material (BOM) border, see Figure 14.70.
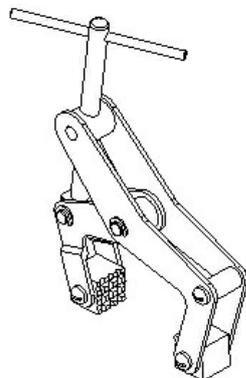
Figure 14.70

Accept the default parameters specified in the PropertyManager and then click on the

green tick mark ☑ in the PropertyManager. The Bill of Material (BOM) is attached to the cursor. Next, click on the drawing sheet to specify the position for the Bill of Material (BOM). The Bill of Material (BOM) is placed at the specified position in the drawing sheet, see Figure 14.71.

Figure 14.71

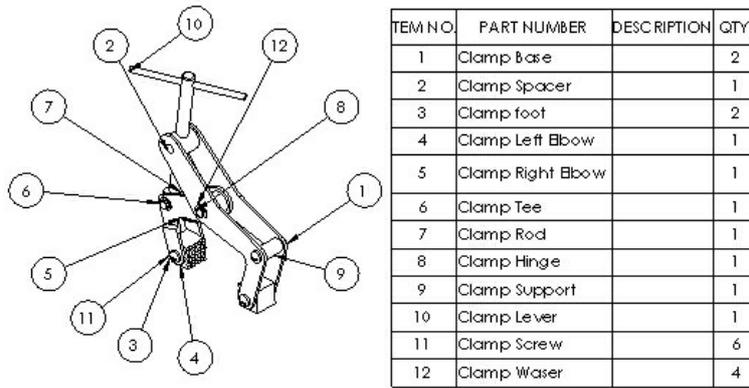| ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Clamp Base		2
2	Clamp Spacer		1
3	Clamp foot		2
4	Clamp Left Elbow		1
5	Clamp Right Elbow		1
6	Clamp Tee		1
7	Clamp Rod		1
8	Clamp Hinge		1
9	Clamp Support		1
10	Clamp Lever		1
11	Clamp Screw		6
12	Clamp Waser		4

Procedure for Creating Bill of Material (BOM)
1. Click on the down arrow below the **Tables** tool in the **Annotation CommandManager**. A flyout appears.
2. Click on the **Bill of Materials** tool in the flyout. The **Bill of Materials PropertyManager** appears.
3. Click on a drawing view of the assembly in the drawing sheet.
4. Accept the default parameters specified in the PropertyManager and then click on the green tick mark ☑ in the PropertyManager. The Bill of Material (BOM) is attached to the cursor.
5. Click to specify the placement point for the Bill of Material (BOM) in the drawing sheet.

Adding Balloons
A Balloon is attached to a component with a leader line and displays the respective part number assigned in the Bill of Material (BOM), see Figure 14.72. In the drawings, balloons are generally added to the individual components of an assembly in order to identify them easily with respect to the part number assigned in the Bill of Materials (BOM).

Figure 14.72

| ITEM NO.	PART NUMBER	DESCRIPTION	QTY.
1	Clamp Base		2
2	Clamp Spacer		1
3	Clamp foot		2
4	Clamp Left Elbow		1
5	Clamp Right Elbow		1
6	Clamp Tee		1
7	Clamp Rod		1
8	Clamp Hinge		1
9	Clamp Support		1
10	Clamp Lever		1
11	Clamp Screw		6
12	Clamp Waser		4

In SOLIDWORKS, you can add balloons to the components of an assembly by using two methods: Automatic and Manual. In the Automatic method, balloons are added automatically to all the components of an assembly with respect to the part number assigned in the BOM. Whereas, in the Manual method, you need to add balloons manually to the components of an assembly one by one. Both these methods of adding balloons are as follows:

Adding Balloons Automatically

To add balloons automatically to the components of an assembly, click on the **Auto Balloon** tool in the **Annotation CommandManager**. The **Auto Balloon PropertyManager** appears, see Figure 14.73. The options of this PropertyManager are used to set parameters for balloons. Some of these options are as follows:

Figure 14.73
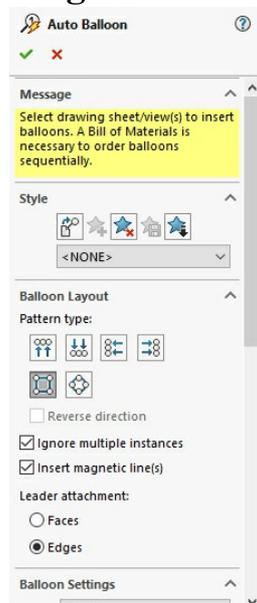
Balloon Layout

The **Balloon Layout** rollout is used to define the type of layout for balloons. The options of this rollout are as follows:

Pattern type

The buttons in the **Pattern type** area of the **Balloon Layout** rollout are used to select

the required type of layouts (Square, Circular, Top, Bottom, Left, and Right) for arranging balloons in the drawing sheet.

Ignore multiple instances

By default, the **Ignore multiple instances** check box is selected in the **Balloon Layout** rollout. As a result, the duplicates are avoided by not adding balloons to all the instances of a component.

Insert magnetic line(s)

By default, the **Insert magnetic line(s)** check box is selected in the rollout. As a result, magnetic lines are inserted along with balloons such that balloons are aligned to each other. Note that this check box is not enabled if the **Layout Balloons to Circular** button is selected in the **Pattern type** area of the rollout.

Leader attachment

By default, the **Edges** radio button is selected in the **Leader attachment** area of the rollout. As a result, the leader lines of balloons are attached to the edges of components. On selecting the **Faces** radio button, balloons are attached to the faces of components through leader lines.

Balloon Settings

The **Balloon Settings** rollout is used to define the settings for balloons such as balloon style, balloon size, and balloon text. The options of this rollout are as follows:

Style

The **Style** drop-down list in the **Balloon Settings** rollout is used to select the required type of style for the border of balloons. By default, the **Circular** option is selected in this drop-down list. As a result, balloons appear with circular borders, refer to Figure 14.72. You can select a style such as triangle, hexagon, and diamond from this drop-down list. Note that on selecting the **None** option, balloons appear without borders, see Figure 14.74. If you select the **Circular Split Line** option, the border of balloons appears such that the circles are split into two areas, see Figure 14.75. By default, its upper area displays part number information and the lowest area displays information about the quantity of component.
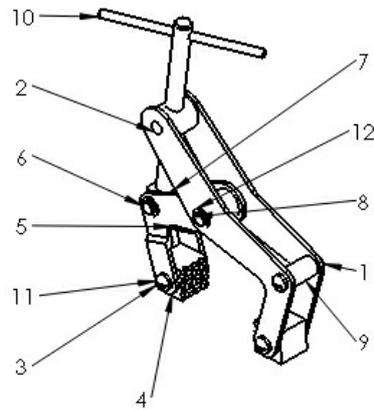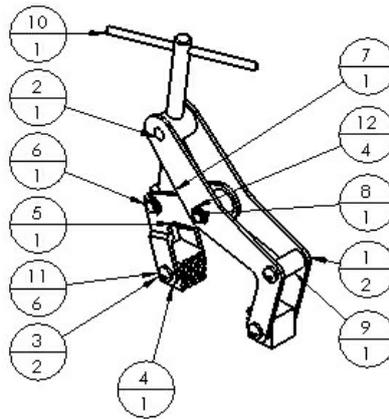
Figure 14.74

Figure 14.75

Size

The **Size** drop-down of the **Balloon Settings** rollout is used to select a required pre-defined size for balloons. In addition to selecting a pre-defined size, you can also select the **User Defined** option from this drop-down list and specify the value for the required size of balloons in the **User defined** field of the rollout.

Balloon text

The **Balloon text** drop-down list is used to select a required text to be displayed in balloons. By default, the **Item Number** option is selected. As a result, balloons appear with part numbers.

Lower text

The **Lower text** drop-down list is used to select a required text to be displayed in the lower area of the balloons. Note that this drop-down list is available only if the **Circular Split Line** option has been selected in the **Style** drop-down list. By default, the **Quantity** option is selected in the **Lower text** drop-down list. As a result, the lower area of balloons displays the quantity information, refer to Figure 14.75.

After specifying the required parameters for balloons, click on the green tick mark ☑ in the PropertyManager. The balloons are attached to the components of the assembly in the drawing view.
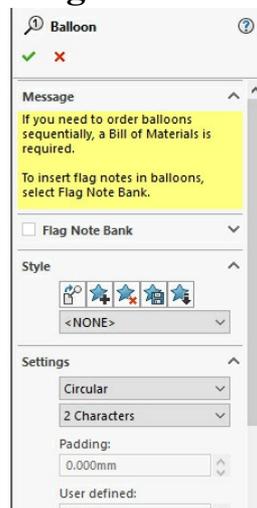
Procedure for Adding Balloons Automatically

1. Click on the **Auto Balloon** tool in the **Annotation CommandManager**. The **Auto Balloon PropertyManager** appears.
2. Click on the drawing view of an assembly, if not selected.
3. Define the pattern layout (square, circular, top, bottom, left, or right) by clicking on the required button in the **Pattern type** area of the **Balloon Layout** rollout.
4. Accept the other default settings in the PropertyManager.
5. Click on the green tick mark ☑ in the PropertyManager. The balloons are added automatically to the components.

Adding Balloons Manually

To add balloons manually to the components of an assembly, click on the **Balloon** tool in the **Annotation CommandManager**. The **Balloon PropertyManager** appears, see Figure 14.76. The options in this PropertyManager are the same as those discussed earlier. By using this PropertyManager, you can add balloons to the components of an assembly one by one by selecting them in the drawing view. As soon as you click on a component in the drawing view, the leader line of the balloon is attached to the component. Next, move the cursor to the required location and then click to specify the location for the balloon in the drawing sheet. Similarly, you can add balloons to all the components of the assembly one by one. Once you have added balloons to all the components, click on the green tick mark ☑ in the PropertyManager.

Figure 14.76

Procedure for Adding Balloons Manually

1. Click on the **Balloon** tool in the **Annotation CommandManager**.
2. Click on a component of the assembly in a drawing view.
3. Move the cursor to the required location and then click to specify the placement point for the balloon. The balloon is added to the component selected.
4. Similarly, add balloons to the remaining components of the assembly one by one.
5. Click on the green tick mark ☑ in the PropertyManager.

Tutorial 1

Open the model created in Tutorial 2 of Chapter 7 and then create different drawing views: front, top, side, isometric, section, and detail as shown in Figure 14.77 in the A3 (ANSI) Landscape sheet size. You also need to apply driving dimensions to the front, top, and right side drawing views of the model.
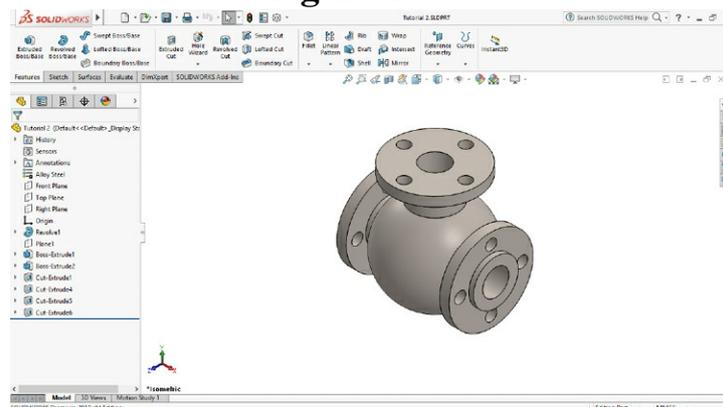
Figure 14.77
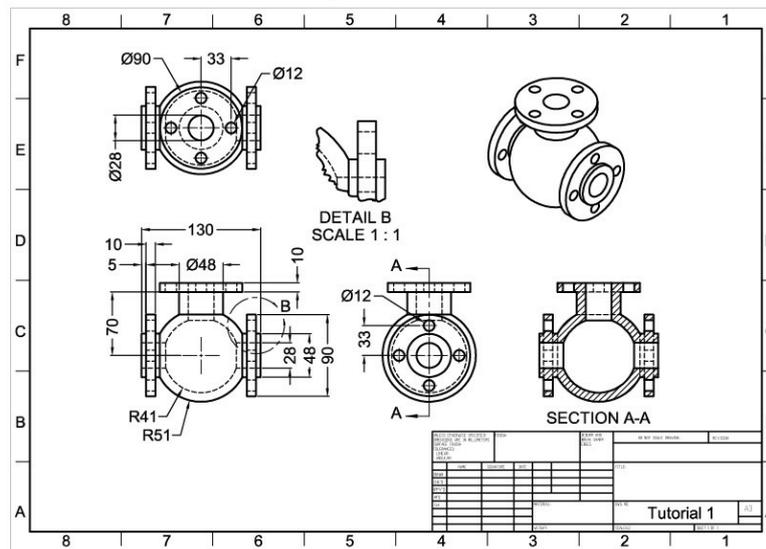
Section 1: Starting SOLIDWORKS

1. Double-click on the SOLIDWORKS icon on your desktop to start SOLIDWORKS.

Section 2: Opening and Saving Model Created in Tutorial 2 of Chapter 7

1. Open the model created in Tutorial 2 of Chapter 7 by using the Open button of the Standard toolbar, see Figure 14.78.

Figure 14.78

2. Click on File > Save As in the SOLIDWORKS menus and then save the model with the name Tutorial 1 inside the *Tutorial* folder of *Chapter 14*. Note that you need to create these folders inside the *SOLIDWORKS* folder.

Section 3: Invoking Drawing Environment

1. Click on the arrow next to the **New** tool in the **Standard** toolbar. A flyout appears, see