

Chapter 12

Generating, Editing, and Dimensioning the Drawing Views

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the Draft environment.*
- *Learn the types of views generated in Solid Edge.*
- *Generate drawing views.*
- *Manipulate drawing views.*
- *Add annotations to drawing views.*
- *Evolving 3D Model from 2D drawings.*
- *Generate the exploded views of assemblies.*
- *Create associative balloons and parts list.*

THE DRAFT ENVIRONMENT

After you have created a solid model or an assembly, you can generate its two-dimensional (2D) drawing views. Note that you can create a solid model in the part environment and assemble it in the **Assembly** environment. Solid Edge has a separate environment called the **Draft** environment, which is used to generate drawing views. This environment contains tools to generate, edit, and modify the drawing views.

To invoke the **Draft** environment, start **Solid Edge ST3** and then choose the **ISO Draft** option from the **Create** area of the welcome screen.

If Solid Edge is running on your computer, invoke the **New** dialog box and select the **iso draft.dft** template from it, see Figure 12-1. The *.dft* is the extension of the files created in this environment. After selecting the required template, choose **OK** in the **New** dialog box to enter the **Draft** environment. You can modify the drawing standards of the current file from the **Draft** environment.

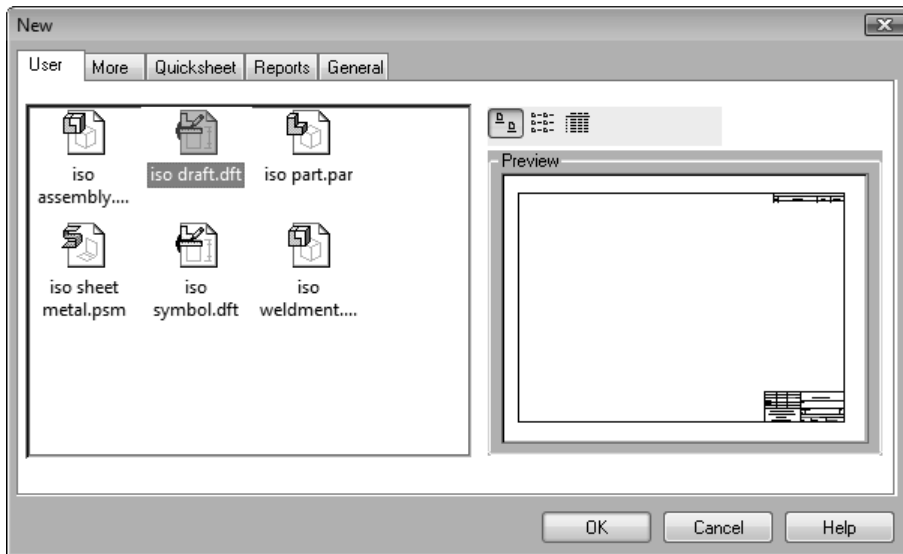


Figure 12-1 Selecting a draft template from the New dialog box

When you enter the **Draft** environment, the drawing sheet and the background sheet with borders will be displayed, as shown in Figure 12-2. This sheet is the one on which you will generate the drawing views. The background sheet is used to add title blocks. You can set the parameters of the drawing sheet in the **Sheet Setup** dialog box which can be invoked by choosing **Sheet Setup** from the **Application Menu**.

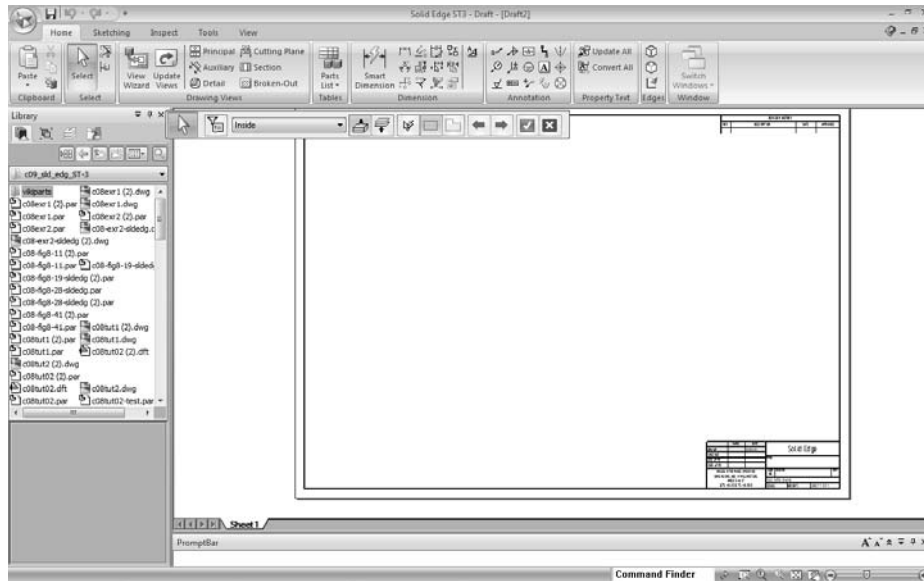


Figure 12-2 The default screen in the **Draft** environment

If you have a part file or an assembly file opened and you want to generate its drawing views, choose **Application Button > New > Create Drawing**; the **Create Drawing** dialog box is displayed. Select the default template and choose the **OK** button; the **Drawing View Creation Wizard** will be displayed to generate the drawing views. You will learn more about this dialog box later in this chapter.



Tip. To use an empty sheet without any margin or title block, choose **Application Button > Sheet Setup**; the **Sheet Setup** dialog box will be displayed. In this dialog box, choose the **Background** tab and select the blank space in the **Background sheet** drop-down list. The preview of an empty sheet will be displayed in the dialog box. Next, choose the **OK** button to exit the dialog box.

TYPES OF VIEWS GENERATED IN SOLID EDGE

In Solid Edge, there are two types of drafting techniques: generative drafting and interactive drafting. In the generative drafting, the views are generated from the part or assembly that is already created. In the interactive drafting, the views are sketched using the sketching tools.



Note

The reason for generating the drawing views is that these views are associative with their respective solid models or assemblies. Therefore, any change in the model updates the drawing views also. On the other hand, the sketched view is not associated with any model. Therefore, the editing of the views is not automatic.

In Solid Edge, you can generate six types of views from a model or an assembly. These views are discussed next.

Base View

The base view is the first view and is generated using a parent model or an assembly. This view is an independent view and is not affected by the changes made in any other view in the drawing sheet. Most of the other views are generated taking this view as the parent view.

Principal View

The principal view is an orthographic view that is generated using any other view present in the drawing sheet. This is the most common view generated after the base view.

Auxiliary View

The auxiliary view is a drawing view that is generated by projecting lines normal to a specified edge of an existing view. These views are mainly used when you want to show the true length of an inclined surface.

Section View

The section view is generated by cutting the part of an existing view using a plane or a line and then viewing the parent view from a direction normal to the section plane. These views are generally used for the models that are complicated from inside and it is not possible to display the inner portion of the part using the conventional views.

Detail View

The detail view is used to display the details of a portion of an existing view. This portion is selected from the parent view. The portion that you select will be scaled and placed as a separate view. The scale can be modified, if needed.

Broken-Out View

The broken-out view is used for the parts that have a high length to width ratio. The broken-out area is specified by adding break lines to an existing orthographic view.

GENERATING DRAWING VIEWS

Ribbon: Home > Drawing Views > View Wizard



In Solid Edge, the first view to be generated is the base view. This view is generated using the **View Wizard** tool. The remaining views are generated by using the base view directly or indirectly. Before you proceed further, you need to set the projection type to the third angle projection. To do so, choose **Application Button > Solid Edge Options**; the

Solid Edge Options dialog box will be invoked. Next, choose the **Drawing Standards** tab from this dialog box. In this tab, select the **Third** radio button from the **Projection Angle** area and then choose **OK**.

Generating the Base View

You can generate the base view by using the **View Wizard** tool. When you invoke this tool, the **Select Model** dialog box will be displayed. In this dialog box, select the model whose drawing views need to be generated and then choose the **Open** button; the **Drawing View**

Creation Wizard will be displayed. Figure 12-3 shows the wizard when a part file is selected. The options in this wizard are discussed next.

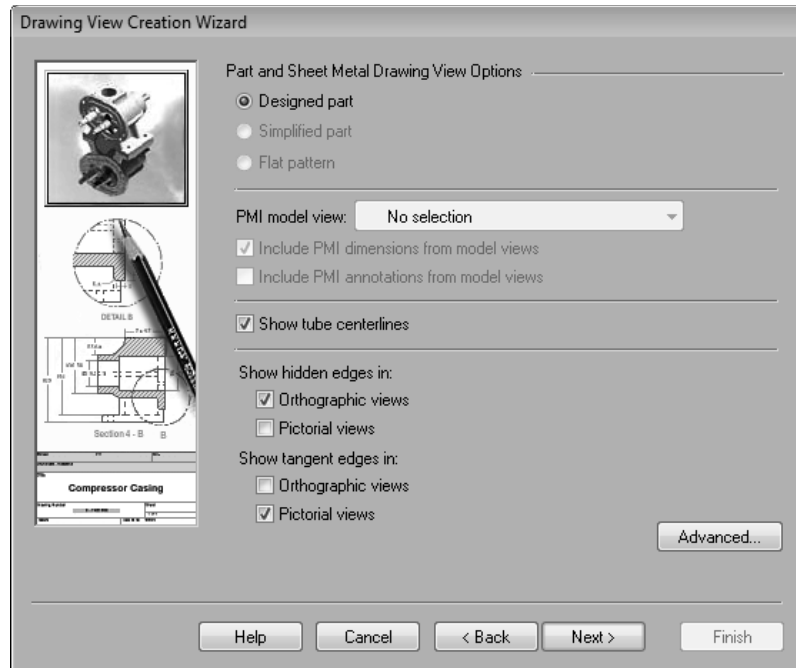


Figure 12-3 The Part and Sheet Metal Drawing View Options page of the Drawing View Creation Wizard displayed

Part and Sheet Metal Drawing View Options Page

The options in this page enable you to specify the parameters related to the display of the drawing view.

Designed part

This radio button is selected by default and is used to specify that you need to generate the drawing views of an existing part.

Simplified part

This radio button is used to generate the drawing views of the simplified version of a model. This button will be in the inactive state if the simplified version of a model does not exist.

Flat pattern

This radio button is used to generate the drawing views of a flat pattern of the sheet metal part. It is available only for the sheet metal parts having a flat pattern.

Show tube centerlines

This check box is selected to display the centerlines in the tube components.

Show hidden edges in Orthographic views

This check box is selected by default and is used to display the hidden edges, if any, in the orthographic drawing views.

Show hidden edges in Pictorial views

Pictorial views are the drawing views other than orthogonal views. This option is used to display hidden edges, if any, in pictorial drawing views. Figures 12-4 and 12-5 show the isometric (pictorial) drawing views with visible hidden-edges and suppressed hidden edges, respectively.

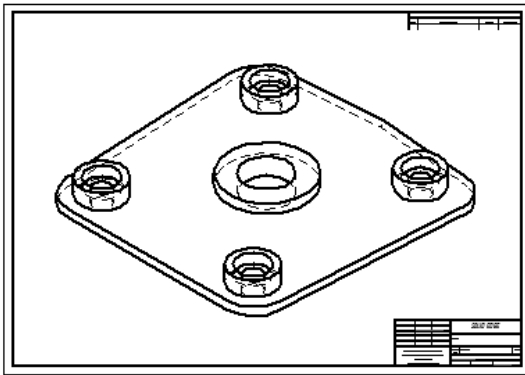


Figure 12-4 Drawing view with hidden edges

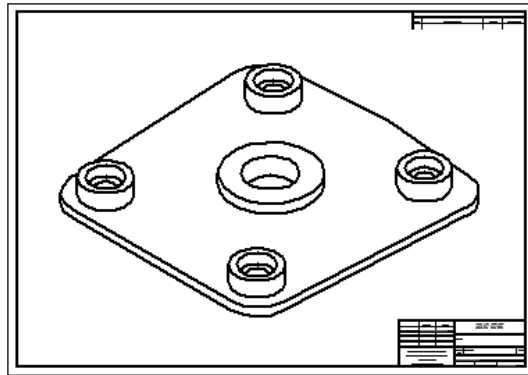


Figure 12-5 Drawing view without hidden edges

Show tangent edges in Orthographic views

Tangent edges are the edges formed by rounds or cylindrical features. This option is used to display the tangent edges, if any in the part, in the orthographic drawing views.

Show tangent edges in Pictorial views

This option is used to display the tangent edges, if any, in the pictorial drawing views.

Drawing View Orientation Page

When you choose the **Next** button from the **Part and Sheet Metal Drawing View Options** page, the **Drawing View Orientation** page will be displayed, as shown in Figure 12-6. The options in this page enable you to specify the standard orientation of the drawing view or the pictorial view. These options are discussed next.

Named Views

This display box consists of options for generating views in the standard orientations. You can select any option from the standard orientations.

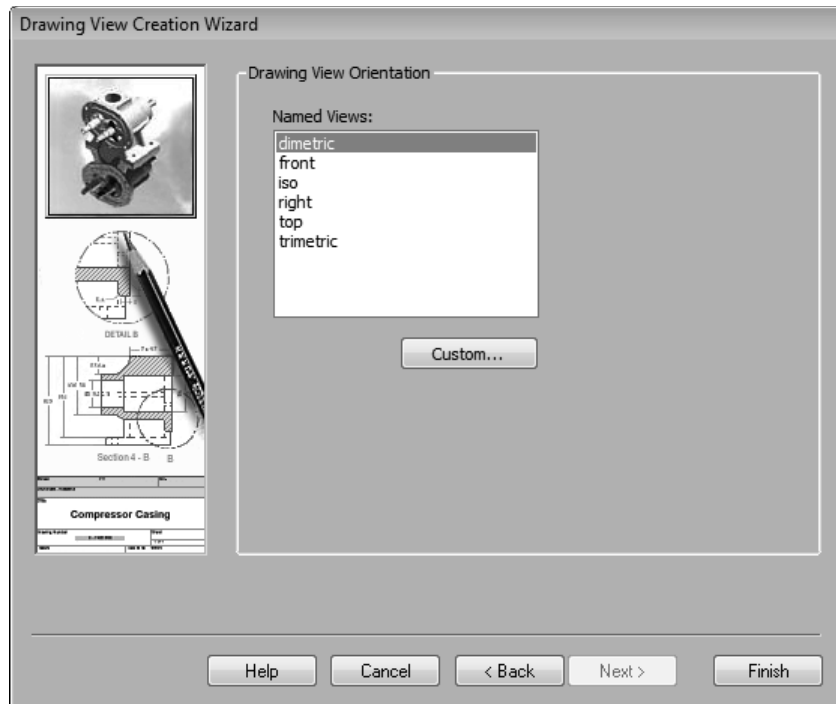


Figure 12-6 The **Drawing View Orientation** page of the **Drawing View Creation Wizard**

Custom

When you choose the **Custom** button, the **Custom Orientation** window will be displayed, as shown in Figure 12-7. This window displays the part or the assembly that you had selected earlier from the **Select Model** dialog box. You can use the drawing display tools available in this window to set the orientation of the model. You can also spin the model using the middle mouse button. After setting the orientation, choose the **Close** button to exit the window.



Note

*The **Next** button is not activated in the **Drawing View Creation Wizard** if the **iso**, **trimetric**, or **dimetric** option is selected from the **Named Views** area.*

If you suppress features of a model whose drawing views are generated, the suppressed features will not be displayed in the drawing views. When you unsuppress the features, they will be displayed in the drawing view.

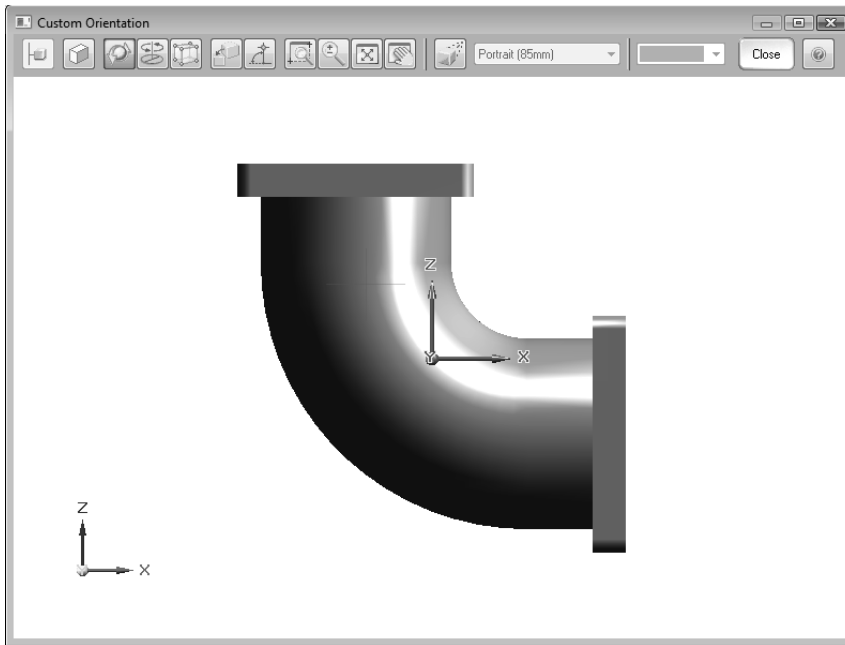


Figure 12-7 The Custom Orientation window



Note

You can create the perspective drawing view of the model by choosing the **Perspective** button from the **Custom Orientation** window and specifying the perspective angle in the **Rotation Angle** list.

Drawing View Layout Page

When you choose the **Next** button from the **Drawing View Orientation** page, the **Drawing View Layout** page will be displayed, as shown in Figure 12-8. The button in the middle represents the orientation of the model that you selected from the **Named Views** display box. You can select more than one view from the **Drawing View Layout** page and choose the **Finish** button to place the views in the drawing sheet. If you select two buttons, in addition to the middle button, three orthogonal views of the model will be displayed.



Tip. When you double-click on a generated drawing view, the part file associated with that view will be opened.

Generating the Principal View

Ribbon: Home > Drawing Views > Principal



The principal view is generated after selecting an existing view. The views that can be selected include a base view or another principal view. To generate the principal view, choose the **Principal** tool from the **Drawing Views** group of the **Home** tab; you will be prompted to select a drawing view. Move the cursor to a drawing view or the source view

and select it as soon as it is enclosed in a red box. On doing so, a red crossed box will be attached to the cursor. Now, as you move the cursor up, down, left, or right, the box will also move with it. You can place the view at the bottom, top, left, or right of the source view. Note that if you move the cursor diagonally, you can generate a pictorial view, such as an isometric view, from the source view.

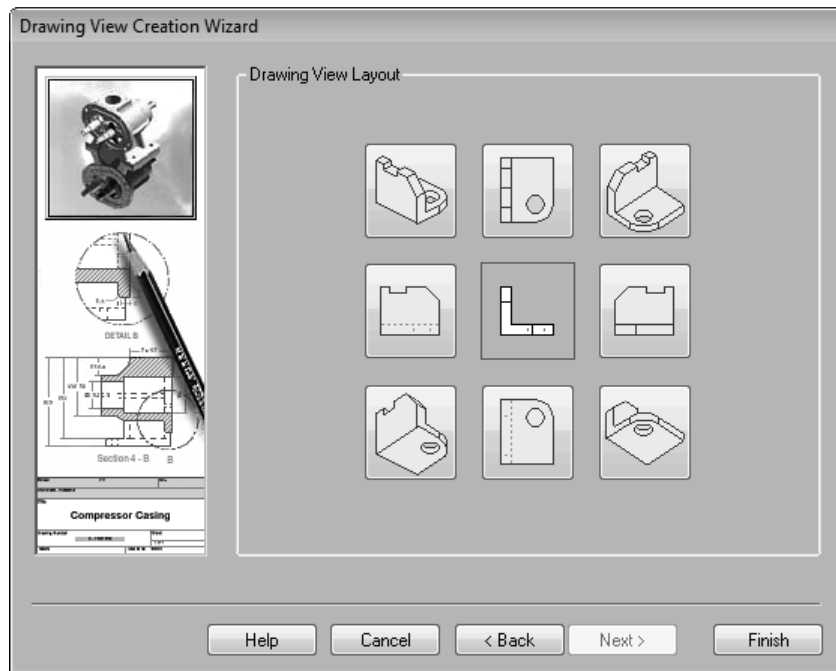


Figure 12-8 The **Drawing View Layout** page of the **Drawing View Creation Wizard**



Note

A principal view cannot be generated from a detail view, section view, or auxiliary view.

By default, Solid Edge generates drawing views in the first angle projection system. However, the third angle projection system has been used throughout the book. The procedure to change the default projection system to the third angle projection system has already been discussed in this chapter.

Figure 12-9 shows the drawing sheet with the base view and the principal views. The base view is the front view and the top and isometric views are generated from the front view using the **Principal** tool.



Note

*When you generate a drawing view on the drawing sheet, it will be displayed with the visible and hidden edges. You can also change the display of the drawing view to the shaded view. To change the display, select the drawing view; various shading buttons will be available in the **Principal** command bar. Choose any shading button and update the drawing view.*

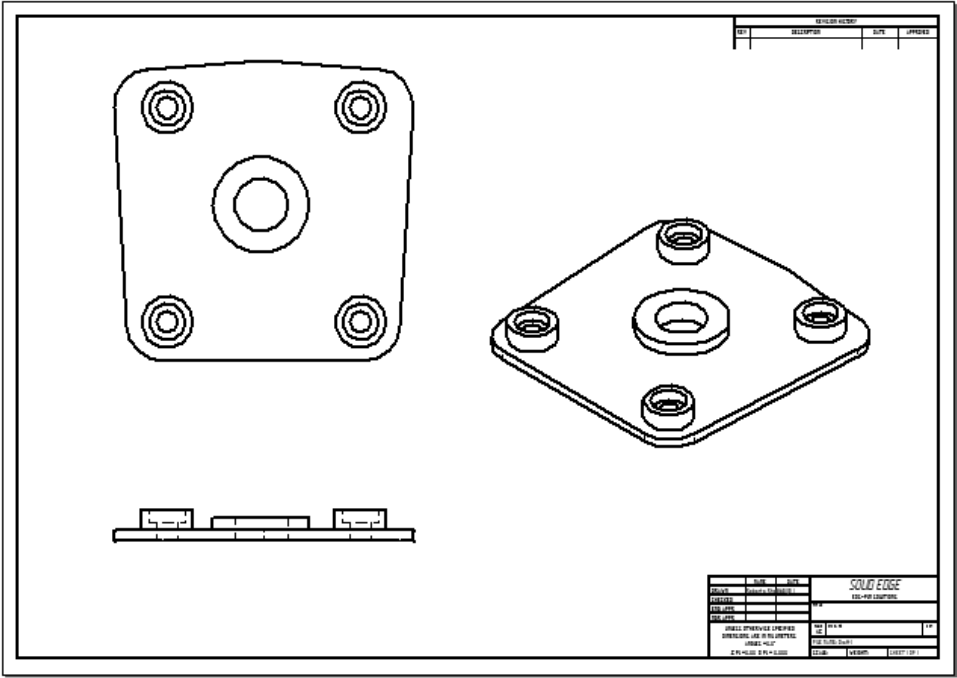


Figure 12-9 Drawing sheet with the base view and the principal views

Generating the Auxiliary View

Ribbon: Home > Drawing Views > Auxiliary



The auxiliary view is a drawing view that is generated by projecting the lines normal to a specified edge of an existing view. To generate the auxiliary view, choose the **Auxiliary** tool from the **Drawing Views** group of the **Home** tab; you will be prompted to click on a fold line or click for the first point of the fold line. The fold line is an edge of the model or an imaginary line normal to which the view will be projected. Select an edge of the model or a keypoint on the edge and then select another keypoint. An imaginary fold line will be formed and the auxiliary view will be projected normal to this fold line. Move the cursor and then click to place the view; an arrow pointing in the direction normal to the fold line will be displayed, as shown in Figure 12-10.

The display of this arrow can be changed to a line with double arrows. To change the display, select the arrow and right-click to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu; the **Viewing Plane Properties** dialog box will be displayed, as shown in Figure 12-11. Select the **Double** radio button from the **View Direction Lines** area in the dialog box and choose **OK**; the arrow will change to a line with double arrows.

Need for Auxiliary View

The need for auxiliary view arises when it becomes impossible to dimension a geometry in the orthographic views. For example, in the model shown in Figure 12-10, the profile on the face of the inclined feature cannot be dimensioned until a view is generated normal to the inclination. After the auxiliary view is generated, the profile can easily be dimensioned with true dimensions.

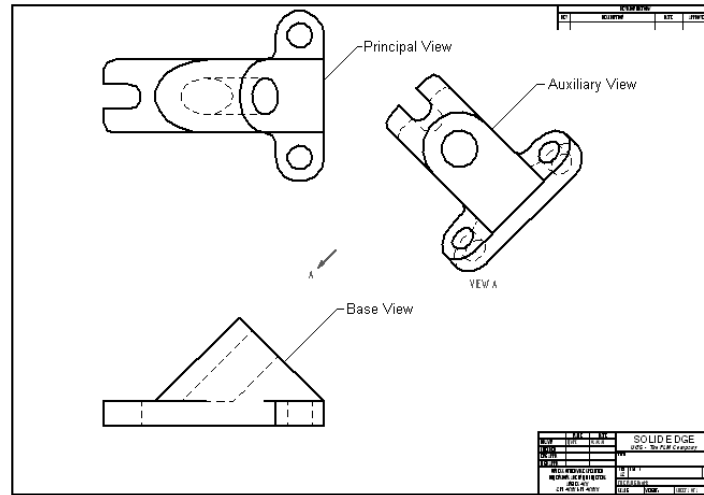


Figure 12-10 Auxiliary view generated from the principal view



Figure 12-11 *The Viewing Plane Properties dialog box*

Generating the Section View

Ribbon: Drawing Views > Section




As mentioned earlier, the section view is generated by cutting a portion of an existing view using a cutting geometry and then viewing the source view from the direction normal to the cutting geometry. In Solid Edge, various types of section views can be generated using the **Section** tool.

To create a section view, you need a geometry that will be used to cut the source view. To create a cutting geometry, the **Cutting Plane** tool is used.

Generating a Simple Section View

The following steps explain the procedure for creating a cutting geometry and generating a simple section view:

1. Choose the **Cutting Plane** tool from the **Drawing Views** group of the **Home** tab; you will be prompted to select a drawing view. This view will be the source view. 
2. Select the source view to activate the sketching environment. This environment contains the sketching tools that you can use to create the cutting geometry.



Note

When you move the cursor over a drawing view to select the source view, it will be enclosed in a red box. The box indicates that the view can be selected. But sometimes, when you move the cursor on the view in the area that does not have any entity, this box will disappear. If you click on this point, the view will not be selected. Therefore, note that the red box will appear only when the cursor is on an entity that composes the drawing view.

3. Draw the sketch for the cutting geometry. Remember that the sketch should be a continuous open sketch. You can use the alignment indicators to sketch the cutting geometry.
4. After drawing the sketch for the cutting geometry, choose the **Close Cutting Plane** button from the **Close** group to exit the cutting plane environment.



Note

When you move the cursor across the cutting geometry, the direction of arrows will also change.

5. Specify the direction of the view by clicking in the direction of the arrows.
6. Choose the **Section** tool from the **Drawing Views** group of the **Home** tab; you will be prompted to select a cutting plane.
7. Select the cutting plane and place the section drawing view parallel to the cutting plane at the desired location on the drawing sheet. A simple section view is shown in Figure 12-12.

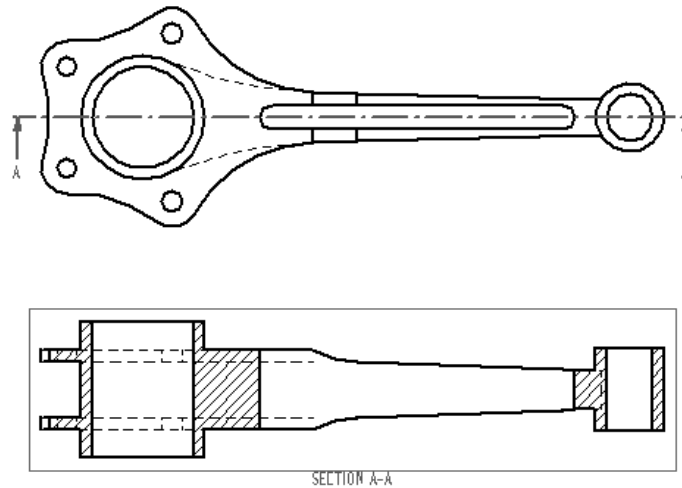


Figure 12-12 The shaded top view and the front section view

Points to Remember while Creating Cutting Planes

The following points should be remembered while creating the cutting planes:

1. The sketch drawn can be a combination of arcs and lines, but an arc cannot be the start or the end entity of the sketch.
2. The sketch must be open and all entities should be connected to each other.
3. Relationships and dimensions can be applied between the sketched entities and the drawing view.
4. The cutting plane can be edited by double-clicking on it or by choosing the **Edit** button, which will be displayed in the command bar when you select the cutting plane.



Note

When you place the section view, it does not matter on which side of the source view you place it. The section view remains the same on either side of the source view; but it varies with the direction of arrows on the cutting plane.

Revolved Section Views



The revolved section views are needed when some features in a model are at a certain angle. In a revolved section view, the section portion revolves around an axis normal to the viewing plane such that it is straightened. For example, Figure 12-13 shows the views of a model that has three outer features at an angle equal with respect to each other. If you want to show the geometry of at least two outer features, you need to generate a revolved section view.

To generate a revolved section view, invoke the **Section** tool and select an existing cutting plane. Since the cutting planes have entities that are at some angle to each other, you need to select the line that will be used as a fold line for generating the section view. For example, in Figure 12-14, the inclined line is used to generate the section view at the top and the vertical line is used to generate the section view on the right. Before placing the view, choose the **Revolved Section View** button from the command bar.

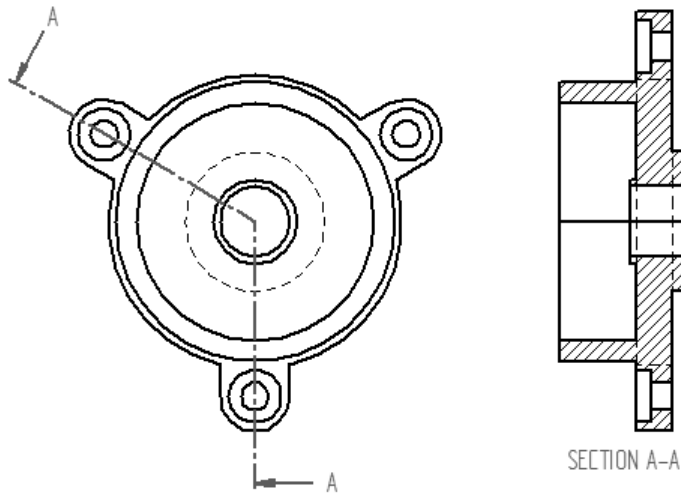


Figure 12-13 The Front view and the right side revolved section view

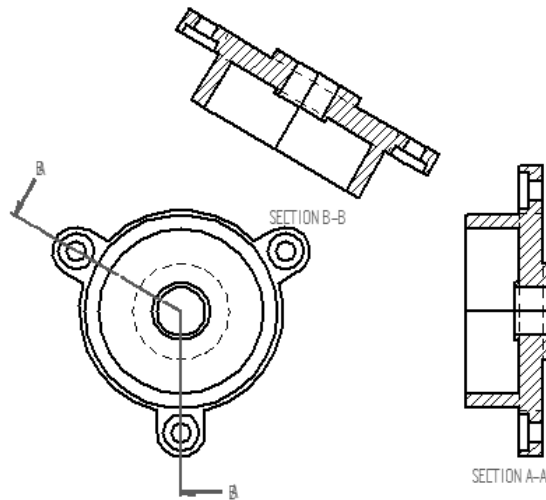


Figure 12-14 Two different revolved section views generated by selecting two different cutting geometries



Note

The cutting plane geometries should be multiline and they should be inclined at an angle to each other.

If a multiline cutting geometry exists, you can select only the first entity and the last entity as the cutting plane.

If the cutting plane geometry consists of an arc, the **Revolved Section View** button in the command bar cannot be used.

Section View that Displays only the Sectioned Geometry



The **Section Only** button in the command bar is used to generate a section view that displays only that area of the model that is sectioned. Figure 12-15 shows the section view which displays only the section area of the model.

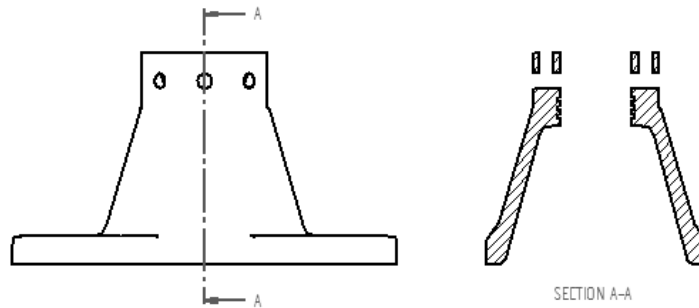


Figure 12-15 Section view displaying only the section area

The following steps explain the procedure for creating this type of view:

1. Choose the **Section** tool from the **Drawing Views** group of the **Home** tab; you are prompted to select a cutting plane.
2. Select the cutting plane; a box attached to the cursor and the options in the **Section** command bar get enabled.
3. Choose the **Section Only** button from the command bar and place the view by clicking on the desired side. The section view is placed, refer to Figure 12-15.

Generating the Broken-Out Section View

Ribbon: Home > Drawing Views > Broken-Out



The broken-out section view is used to display the section of a particular portion of a model without sectioning the model. The **Broken-Out** tool is used to generate the broken section view. This tool is available in the **Drawing Views** group of the **Home** tab. The steps given next explain the procedure for creating this type of view.

1. Choose the **Broken-Out** tool from the **Drawing Views** group of the **Home** tab and select the drawing view where you have to draw the profile for the broken view. You will automatically enter the sketching environment where all the tools that are used to draw the profile are available.
2. Draw the sketch, as shown in Figure 12-16. Next, choose the **Close Broken-Out Section** button from the **Ribbon** to exit the **Broken-Out** environment; you are prompted to specify the distance for the depth of the cut.

- Specify the distance in the **Depth** edit box present on the command bar or on the other orthographic view, as shown in Figure 12-16. After specifying the depth of the cut, you are prompted to select the view that needs to be broken.

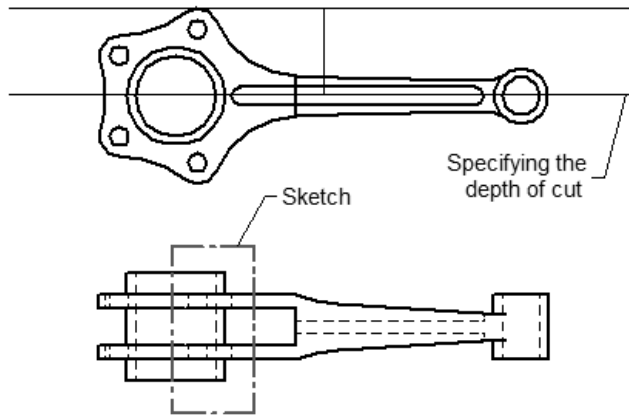


Figure 12-16 Specifying the profile and the depth of the cut

- Select the isometric view (pictorial view) to generate the broken-out section view, as shown in Figure 12-17.

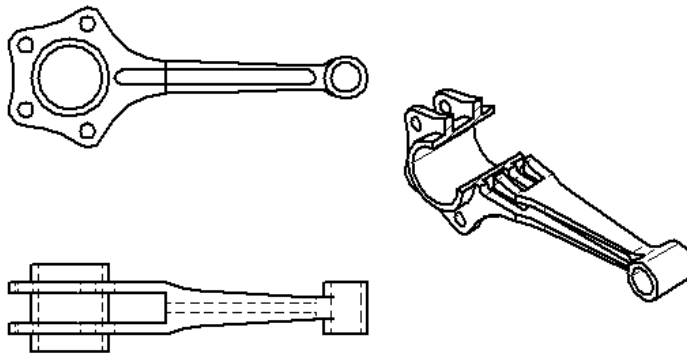


Figure 12-17 Isometric broken-out section view

Generating the Detail View

Ribbon: Drawing Views > Detail



Detailed views are used to provide the enlarged view of a particular portion of an existing view. In Solid Edge, the detailed view is generated by drawing a circle or any other user-defined sketch around the portion whose details are needed. When you choose the **Detail** button, the **Detail** command bar will be displayed and the **Circular Detail View** button will be chosen by default. Also, you will be prompted to select the center of the circle. After specifying the center of the circle, you will be prompted to specify another point to determine the radius of the circle. As soon as you click, a circle, which is actually the detail view, will be attached to the cursor. Place the view on the drawing sheet at the desired location, see Figure 12-18.

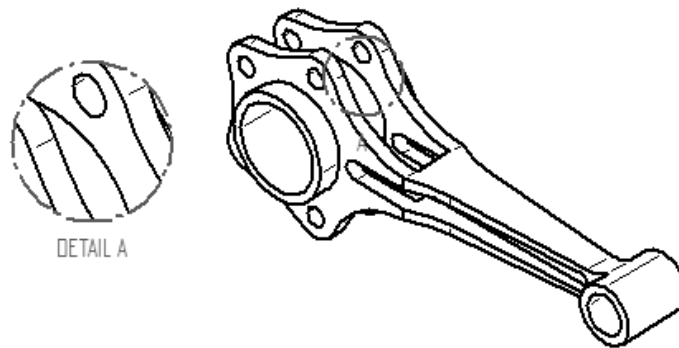


Figure 12-18 Detail view of an isometric view

You can also choose the **Define Profile** button from the command bar to define the boundary of the detail view. When you choose this button, the sketching environment will be activated. Draw the required closed profile using the tools available in this environment. Figure 12-19 shows the user-defined closed profile and the detail view created.

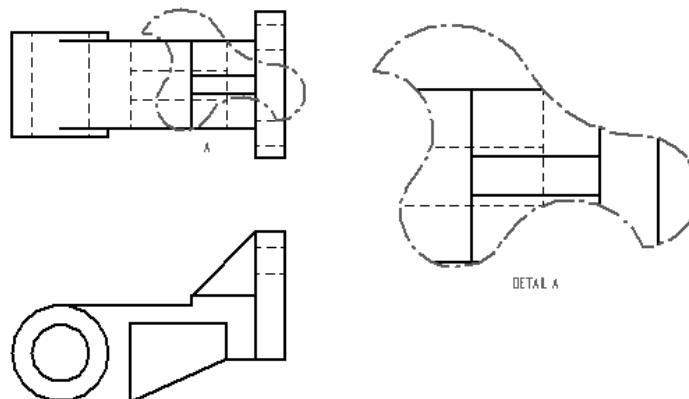


Figure 12-19 User-defined sketch and the resulting detail view



Tip. If you drag and drop a part file from the docking window on the drawing sheet, then depending on the projection angle system set for the current file, the top, front, and right side views of the part will be generated. If you drag and drop an assembly on the drawing sheet, then an isometric view of the assembly will be generated.

You can right-click on a detail view and choose **Convert to Independent Detail View** to convert the view to an independent view.

Generating the Broken View

This type of view is used for the parts that have a high length to width ratio. This view is generated on an existing orthographic or pictorial view. It is created by breaking the existing view along the horizontal or vertical direction using horizontal or vertical lines. Various types of broken views are shown in Figure 12-20.

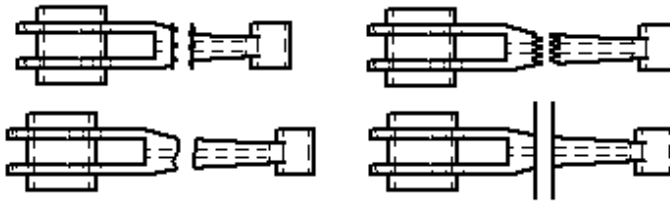


Figure 12-20 Different types of broken views

The following steps explain the procedure for creating this type of view:

1. Move the cursor over the drawing view where you have to add the break lines. When the view is highlighted and enclosed in a red box, right-click to invoke the shortcut menu.
2. From the shortcut menu, choose the **Add Break Lines** option; you are prompted to click in the drawing view at the point where you need to place the first break line. Notice that as you move the cursor, the break line also moves with it.

You can set the style for the break lines from the **Style** drop-down list. By choosing the **Vertical Break** or **Horizontal Break** buttons from the command bar, you can specify whether the drawing view will be broken vertically or horizontally. In the command bar, there are four linetypes that can be selected for the break line type.

3. Before specifying the first point for the break line, you need to specify the line type for the break lines. To do so, choose the **Break Line Type** button from the command bar; a flyout is displayed with four buttons. Choose any of the buttons for selecting a line type. The **Break gap** edit box on the command bar is used to set the distance between the pair of break lines. The **Height** edit box is used to specify the height of the zigzag in the break lines. The **Pitch** edit box is used to specify the pitch of the breaks when you select the zigzag break line type. The **Symbols** edit box, which is available for the last break line type, is used to set the number of symbols that are added to the break lines.



Tip. You can add the views of different parts and assemblies on a single drawing sheet. This helps you view the dimensions of the different models on a single sheet. To add a new model to the sheet, follow the same procedure that was used to generate the base view.

4. Click on the drawing view to specify the location for the first line. After the first break line is placed, click again to specify the location for the second break line.
5. After specifying the second break line, choose the **Finish** button to exit.

**Note**

When you select a view with break lines, the **Show Broken View** button will be displayed in the command bar. This button toggles the display of break lines in the selected drawing view.

WORKING WITH INTERACTIVE DRAFTING

As mentioned earlier, you can also sketch the 2D drawing views in the **Draft** environment of Solid Edge. In technical terms, sketching the 2D drawing views is known as interactive drafting. You can draw the 2D drawing views by choosing the **Draft View** button from the **Quick Access toolbar**, after customizing the **Quick Access toolbar**. To customize the toolbar, right-click on the **Ribbon** and choose the **Customize Quick Access Toolbar** from the shortcut menu displayed; the **Customize** dialog box will be displayed. Next, select the **Commands Not in the Ribbon** option from the **Choose commands from** drop-down list and then select **Draft View** from the list box displayed below the drop-down list. Next, choose **Add** to add the **Draft View** as a button to the toolbar and then close the dialog box. Next, choose the **Draft View** button from the **Quick Access toolbar**; you will be prompted to place the view. Select a point on the drawing sheet; the sketching environment will be invoked. The tools in the sketching environment are the same as those in the sketching environment of the Part environment. After sketching the drawing view, choose the **Close Draw In View** button from the **Close** group of the **Ribbon** to return to the **Draft** environment.

MANIPULATING DRAWING VIEWS

Once the drawing views are generated, it is very important to learn how they can be modified or edited. The following editing operations can be performed on the existing drawing views:

Aligning Drawing Views

When you generate the principal views from the base views, they are aligned automatically. If you move one of the views, then the other view will also move along with the first one. This shows that the two views are aligned. Also, when you select one of the views, a centerline that connects the two views will be displayed.

To unalign a view, select the view and right-click to invoke a shortcut menu. Choose **Maintain Alignment** from the shortcut menu to deactivate it. The alignment indicator in the view shows a zigzag line suggesting the view is no more aligned. Now, if you move any of the previously aligned views, the other corresponding view will not move. Choose this option again to align the views. Note that a straight line is displayed in the drawing when the views are aligned.

To completely delete the alignment, right-click on the required view; a shortcut menu will be displayed. Choose the **Delete Alignment** option from the shortcut menu and select the alignment line to be deleted.

To create a new alignment of the view, choose the **Create Alignment** option from the shortcut menu; the **Create Alignment** command bar will be displayed. You can select a location for the drawing view alignment using the **Alignment position** drop-down list. The buttons available in the command bar are used to specify whether you want to create a vertical, horizontal, parallel, or a perpendicular alignment. Choose the required options and buttons from the command bar and then select the view to which you want to align the current view.

Modifying the Scale of Drawing Views

You can modify the scale of a drawing view by selecting it and changing the value in the command bar. Alternatively, you can choose the **Properties** button from the command bar. When you choose this button, the **High Quality View Properties** dialog box will be displayed. You can set the scale value in this dialog box. You can also invoke this dialog box by choosing the **Properties** option from the shortcut menu that will be displayed when you select a view and then right-click on it.



Note

When you modify the scale of a drawing view that was generated by projecting a view, the scale factors of both the views are modified.

Cropping Drawing Views

Cropping is a technique that is used to view only a particular portion of the drawing view on the drawing sheet. To crop a drawing view, select the drawing view; a box with handles will be displayed. Drag a handle till the required portion of the drawing is visible. This visible portion will be the cropped view. The portion of the view lying inside the associated box is retained and the remaining portion is removed.

To bring back the view to its original display, right-click on the view to display the shortcut menu. In the shortcut menu, choose the **Uncrop** option; the cropping in the view is removed.



Note

You cannot crop a detail view and a broken view.

Moving the Drawing Views

To move a drawing view, select it and drag it on the drawing sheet. To place the view, release the left mouse button at the desired location on the drawing sheet.

Rotating the Drawing Views



The drawing views can be rotated by invoking the **Sketching** tab in the **Ribbon**. To rotate a drawing view, choose the **Rotate** tool from the **Move** drop-down in the **Draw** group of the **Sketching** tab; you will be prompted to select an element to modify. Select the drawing view you need to rotate; you will be prompted to select the center of rotation. Select a point that acts as the center of rotation. Now, select a point from which you want to

start the rotation and then select a point up to which you want to rotate the view. You can also specify the angle of rotation value in the dimension boxes present on the command bar.

Applying the Hatch Pattern

You can also apply a hatch pattern to a closed region by using interactive drafting. To apply the hatch pattern, choose the **Fill** tool from the **Draw** group of the **Sketching** tab; you will be prompted to select the area. As you select a closed region, it will be filled with the hatch pattern. You can modify the hatch pattern by selecting it and right-clicking; a shortcut menu will be displayed. Choose the **Properties** option from the menu displayed.

Modifying the Properties of Drawing Views

After generating a drawing view, you can modify its properties. To set its properties, right-click on the drawing view to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu to display the **High Quality View Properties** dialog box, as shown in Figure 12-21. There are eight tabs in this dialog box and these are discussed next.

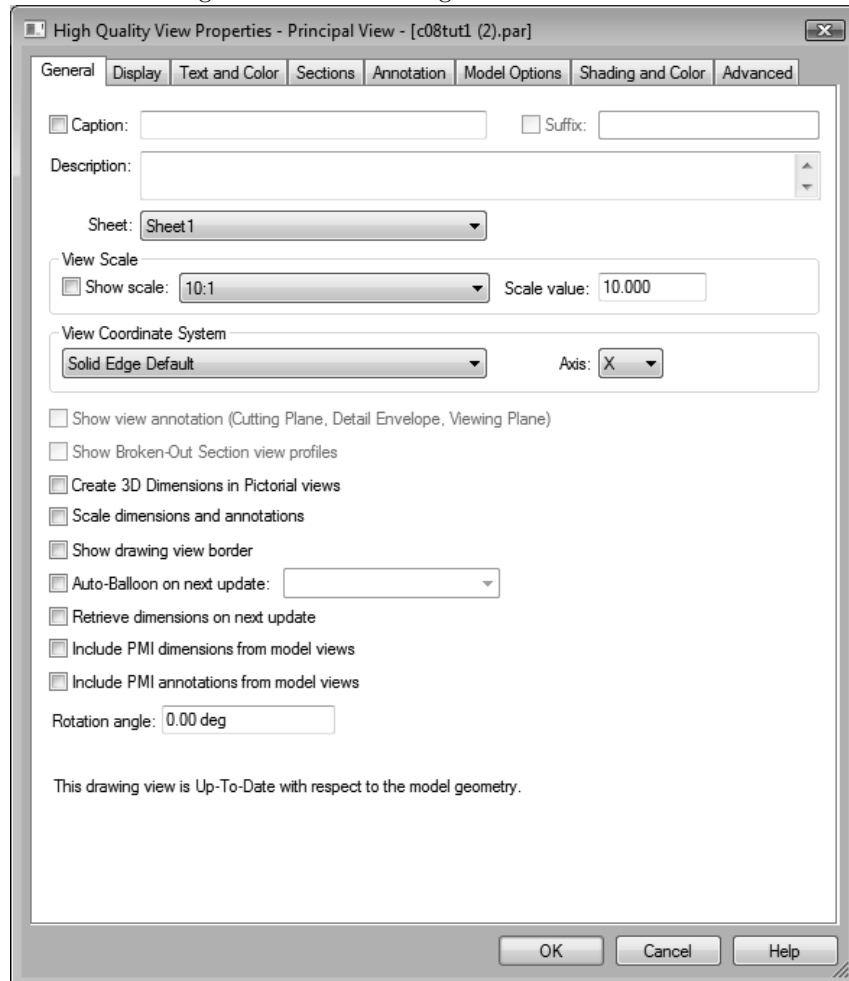


Figure 12-21 The *High Quality View Properties* dialog box for a principal view

The **General** tab contains the options that are used to modify the scale, move the drawing view to another sheet, add caption or description to the drawing view, rotate the drawing view by specifying an angle, and so on. You can also select the options to add 3D dimensions to the pictorial view, retrieve dimensions on the next update of the part, include PMI dimensions as well as annotations from the model views, and so on.

The **Display** tab contains the options that are used to set the display of the drawing view. In this tab, the **Parts list** list box displays the components, subassemblies, and construction surfaces. You can select them in this list and choose the display options from this tab. The **Parts List Options** button enables you to select the items that you want to display in the **Parts list** list box. From the **Selected Parts(s) Display** area, you can set the display style of the entities in the drawing view. The **Restore Default Display Settings** button restores the default settings.

The **Text and Color** tab contains the options that are used to set the dimension style, color, font, font style, and size of text.

The **Sections** tab lists the 3D sections that can be used to generate the section views.

The **Annotation** tab contains the options that enable you to set the style for the center lines and flowlines.

The **Model Options** tab contains the options that are used to specify whether to display the simplified representation of the part in the drawing view.

The **Shading and color** tab contains the options that are used to set the shading of a drawing view of the part. You can also apply the part base colors using the options under this tab.

The **Advanced** tab contains the advanced drawing options.



Tip. To modify the hatch pattern style of a section view, select the view and right-click to invoke a shortcut menu. Choose the **Properties** option from the shortcut menu to display the **High Quality View Properties** dialog box. Next, choose the **Display** tab. Now, from the **Show fill style** drop-down list in the **Selected Part(s) Display** area, select any one of the hatch pattern styles and then choose **OK** to exit the dialog box.

If you need to modify the properties of the hatch such as spacing, angle, and so on, right-click on the drawing view and then choose the **Draw in View** option from the shortcut menu displayed; the view will open in a separate window. Now, right-click on the hatch and then choose the **Properties** option from the shortcut menu; the **Fill Properties** dialog box will be displayed. Using this dialog box, you can modify the parameters of the hatch. After modifying the parameters of the hatch, choose **OK** and then exit the **Draw-in-View** environment; the changes will be displayed in the drawing view.

ADDING ANNOTATIONS TO DRAWING VIEWS

Once you have generated the drawing views, you need to add annotations such as dimensions, notes, surface finish symbols, geometric tolerances, and so on to them. There are two methods of displaying these annotations in the drawing views. The first method is to generate the

annotations that are defined at the time of creating the model such as dimensions. These dimensions are associative in nature and therefore, can be used to modify or drive the dimensions of a model. The second method of displaying the annotations is to manually add them to the drawing views.

**Note**

*You can also create dimensions in the **Draft** environment, but these dimensions cannot drive the dimensions of the part.*

Generating Annotations

Generating annotations is the process of retrieving dimensions, notes, and so on from the parent model. The annotations that were used to create the model are displayed on the orthographic views of the model.



To retrieve dimensions, choose the **Retrieve Dimensions** tool from the **Dimension** group of the **Home** tab; the command bar will be displayed with various dimensioning options and you will be prompted to select a drawing view. As soon as you select a drawing view, the dimensions will be displayed on it. Various buttons available on the command bar are discussed next.

Dimension Style Mapping

This is a toggle button available in the **Format** rollout. It is used to specify whether or not the dimension style mapping set from the **Dimension Style** tab of the **Solid Edge Options** dialog box will be used.

Linear

This button is used to retrieve linear dimensions in the selected drawing view.

Radial

This button is used to retrieve radial dimensions in the selected drawing view.

Angular

This button is used to retrieve angular dimensions in the selected drawing view.

Annotations

This button is used to retrieve annotations that are applied to the model.

Retrieve Duplicate Radial Dimensions

This button is used to retrieve duplicate radial dimensions that have the same value.

Hidden Line Dimensions

This button is used to retrieve dimensions of the hidden edges.

Add Dimensions

If this button is chosen, the dimensions will be added to the drawing view.

Remove Dimensions

If this button is chosen, the dimensions will be removed from the drawing view. Note that the dimensions that are applied using the **Retrieve Dimension** tool can only be removed.

Figure 12-22 shows a drawing view after retrieving dimensions.

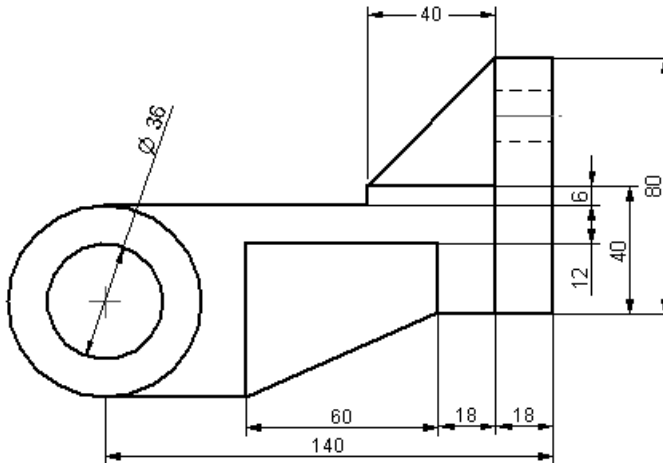


Figure 12-22 Drawing view with dimensions

Displaying Center Marks and Center Lines in a Drawing View



Solid Edge allows you to display center marks and center lines in the drawing views.

To do so, choose the **Automatic Center Lines** tool from the **Annotation** group of the **Home** tab; the command bar will be displayed with various options. If you choose the

Center Line and Center Mark Options button from the command bar, the **Center Line and Center Mark Options** dialog box will be displayed. You can select the entities on which you want to display the center marks and center lines using this dialog box. Note that the **Center Line and Center Mark Options** button is available only when the **Add Lines and Marks** button is chosen.

You can set the options for adding or removing the center lines and center marks using the buttons available in the command bar. It is recommended that you choose the **Center Marks** or the **Center Mark Projection Lines** button from the command bar. Choosing both buttons will place the center lines above the center marks.

Figure 12-23 shows a drawing view with center mark projection lines and Figure 12-24 shows a drawing view with center lines.

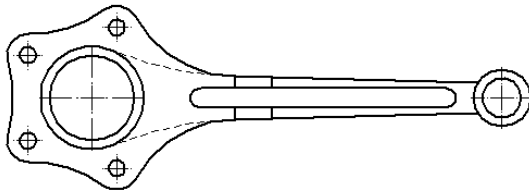


Figure 12-23 Center marks on the holes in a drawing view

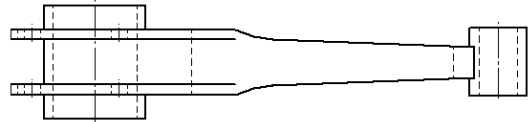


Figure 12-24 Center lines on the holes in a drawing view

Adding Reference Dimensions to Drawing Views

Although retrieving the dimensions from the parent model is the most effective way of dimensioning, sometimes you may also need to dimension the drawing views manually. The options for dimensioning a drawing view are available in the **Dimension** group of the **Ribbon**. These options are similar to those discussed in the sketching environment. The dimension tools that were not discussed earlier are: **Line Up Text**, **Attach Dimension**, and **Chamfer Dimension**. These tools are discussed next.

Line Up Text



This button is available in the **Dimension** group and is used to align the selected entities in line with the base element. The elements can be dimensions, callouts, or balloons. On choosing this button, the **Line Up Text** command bar is displayed. To align the selected entities, select the alignment type and specify the alignment offset distance in the command bar. Next, select the base element to which the selected entities will be aligned. Next, select the entities; the selected entities will be aligned with the base element.

Attach Dimension



This button is available in the **Dimension** group and used to replace the selected dimension with the new element. However, the new element should be of the same type as the selected entity. For example, you can use this tool to replace the dimension of an arc with a new dimension on another arc, but not to a line. Any prefixes, tolerances, or any other formatting of the old dimension can be applied to the new dimension. To attach or replace a dimension, select the dimension and specify the origin and target elements to attach; the dimension gets replaced.

Chamfer Dimension



This button is available in the **Dimension** group of the **Ribbon**. On choosing this button, you will be prompted to select the dimension base line. Select a horizontal or vertical line from which you need to measure the chamfer length; you will be prompted to select the dimension measure line. Select the chamfer line; you will notice that the dimension is attached to the cursor. Click to place the dimension.

ADDING NEW DRAWING SHEETS

In Solid Edge, a drawing file can have multiple drawing sheets. A multisheet drawing file is generally used when you need to generate drawing views of all parts of an assembly in a single drawing file. You can easily switch between the sheets to refer to the drawing views of different parts within the same file, thus avoiding the step of opening separate drawing files.

To add a new sheet to the drawing file, right-click on **Sheet1** at the bottom of the drawing window and choose **Insert** from the shortcut menu; a new sheet named **Sheet2** will be added to the drawing file. Similarly, you can use other options available in the shortcut menu to rename, delete, reorder, and set up a sheet.

EDITING THE DEFAULT SHEET FORMAT

You can edit the default standard sheet format according to your design requirement. To edit the standard sheet format, you need to activate the **Background** and de-activate the **Working** environment. Choose the **Background** button from the **Sheet Views** group of the **View** tab to activate the **Background** and choose the **Working** button from the **Sheet Views** group of the **View** tab to de-activate the **Working**. You will notice that all entities, annotations, and views are removed from the drawing sheet and four sheet tabs, namely **A1-Sheet**, **A2-Sheet**, **A3-Sheet**, and **A4-Sheet** are displayed at the bottom of the drawing sheet. Right-click on **A1-Sheet** at the bottom of the drawing window and choose **Sheet Setup** from the shortcut menu to display the **Sheet Setup** dialog box. You can use the options in this dialog box to modify the size and units of the sheet. You can also delete the existing title block and use the sketching tools to draw a new title block. After editing the sheet, activate the **Working** and de-activate the **Background**.

EVOLVING A 3D MODEL FROM A 2D DRAWING



Create
3D

You can evolve a 3D model from a 2D drawing, using the **Create 3D** tool. This tool enables the solid edge tools to work with 2D drawing views to quickly create the 3D models. Also, you can use the drafting views or the drawing files created in AutoCAD to generate the 3D model. Figure 12-25 shows a 2D drawing and Figure 12-26 shows a 3D model evolved from the 2D drawing.

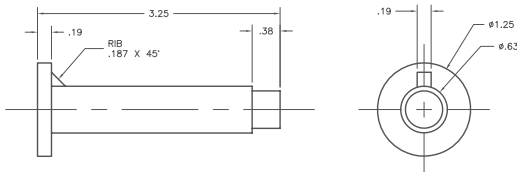


Figure 12-25 A 2D drawing

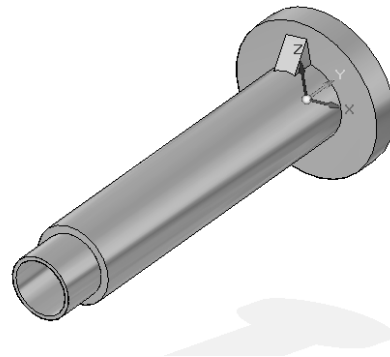


Figure 12-26 The 3D model evolved from the 2D drawing

The steps to be followed to evolve a 3d model from a 2d drawing are given below:

1. Open the 2D drawing in Solid Edge.
2. Choose the **Create 3D** tool from the **Assistants** group of the **Tools** tab to invoke the **Create 3D** dialog box.
3. Specify a template file.
4. Choose the **Options** button and select the desired angle of projection (first or third).
5. Choose the **Next** button and specify the scale for the 3D model, if required.
6. Select the view geometry from the drawing area by drawing a selection window.
7. Choose the **Next** button to select another view geometry, if required.
8. Next, choose the **Set Fold Line** button to draw a fold line. This line defines the position to fold the primary view. This ensures the proper orientation of the sketches.
9. After performing all the steps mentioned above, choose the **Finish** button; the sketches will be oriented as defined.

Now, you can select the profile from the drawing view and perform the required 3d operation on it. You will be more clear about the usage of the **Create 3D** tool in Tutorial 3.

GENERATING THE EXPLODED VIEWS OF ASSEMBLIES

The exploded views are generated by selecting the configuration that was saved at the time of exploding the assembly in the **Assembly** environment. The following steps explain the procedure for generating an exploded view:

1. Choose the **View Wizard** tool from the **Drawing Views** group of the **Home** tab to display the **Select Model** dialog box.
2. Select the assembly from the dialog box and choose the **Open** button to display the **Drawing View Creation Wizard**.
3. From the **Configuration** drop-down list in the dialog box, select the configuration that represents the exploded state of the assembly.
4. Set the other parameters and then choose the **Finish** button from the dialog box.
5. Place the view on the drawing sheet; an exploded view is generated, as shown in Figure 12-27.



Before placing the view, you can also set the display of the parts in the exploded drawing view by choosing the **Model Display Settings** button from the command bar.

When you choose this button, the **Drawing View Properties** dialog box will be displayed, as shown in Figure 12-28. The options in the **Display** tab of this dialog box are discussed next.

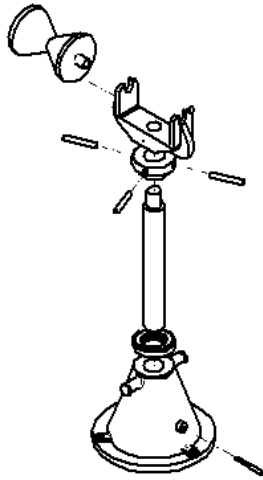


Figure 12-27 Exploded view

Display Tab of the Drawing View Properties Dialog Box

The **Display** tab of this dialog box contains various options for controlling the display of the assembly drawing view. These options are discussed next.

Show

Using the **Show** check box, you can control the display of one or more parts in the assembly. You will also notice that all parts, including the assembly, are selected in the Parts list area.

Display as Reference

This check box, when selected, enables you to display the selected part as reference. You can select the part from the **Parts list** area. The selected part is displayed as dotted in the assembly drawing view.

Visible edge style

This drop-down list enables you to select the edge style of the parts in the assembly drawing view. You can also apply different styles to different parts in the assembly drawing view.

CREATING ASSOCIATIVE BALLOONS AND PARTS LIST



Generally, the drawing view of an assembly also contains the list of parts, material of each part, quantity, and other related information in the form of a table, which is called the bill of material (BOM). In Solid Edge, it is called the parts list. Balloons are added to the parts of assembly and each balloon refers to the part in the parts list.



The parts list can be generated by choosing the **Parts List** tool from the **Part List** drop-down in **Tables** group. The parts list is associative in nature, which means that any modification made in the part files of the assembly will be reflected in the parts list also. The procedure for generating a parts list and balloons is given next.

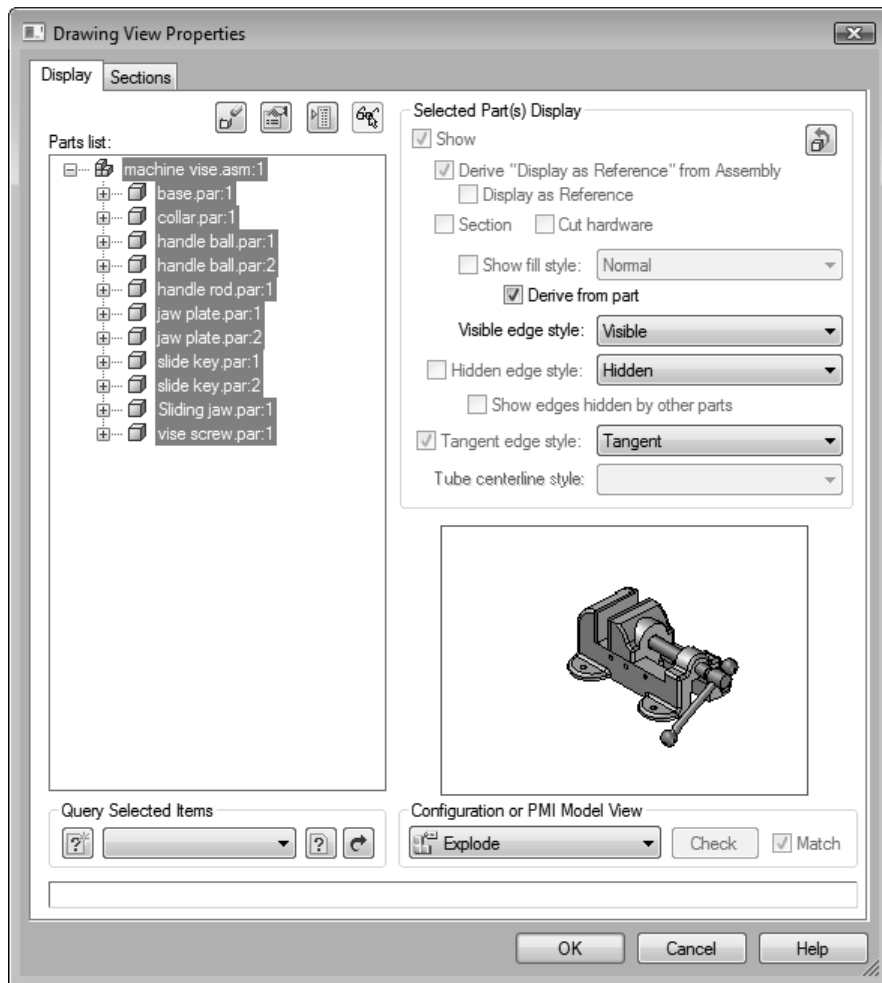


Figure 12-28 The Drawing View Properties dialog box

1. Generate the assembly drawing view.
2. Choose the **Parts List** button from the **Tables** group of the **Home** tab; the command bar will be displayed with the options that can be used to specify whether or not you want to display the balloons and the table. Also, you will be prompted to select a drawing view.
3. Select the assembly drawing view. You can also choose the **Properties** button from the command bar. On doing so, the **Part List Properties** dialog box will be displayed. Set the display properties of the parts list and balloons in this dialog box and choose **OK**.
4. Choose the **Auto-Balloon** button from the command bar, if not already chosen.
5. Click in the drawing area to place the parts list and balloons, as shown in Figure 12-29.

Parts List Properties Dialog Box

While generating a parts list on the drawing sheet, if you choose the **Properties** button from the command bar before placing it, the **Parts List Properties** dialog box will be displayed, as shown in Figure 12-30. This dialog box is used to set the parameters of the parts list and balloons. The options in this dialog box are discussed next

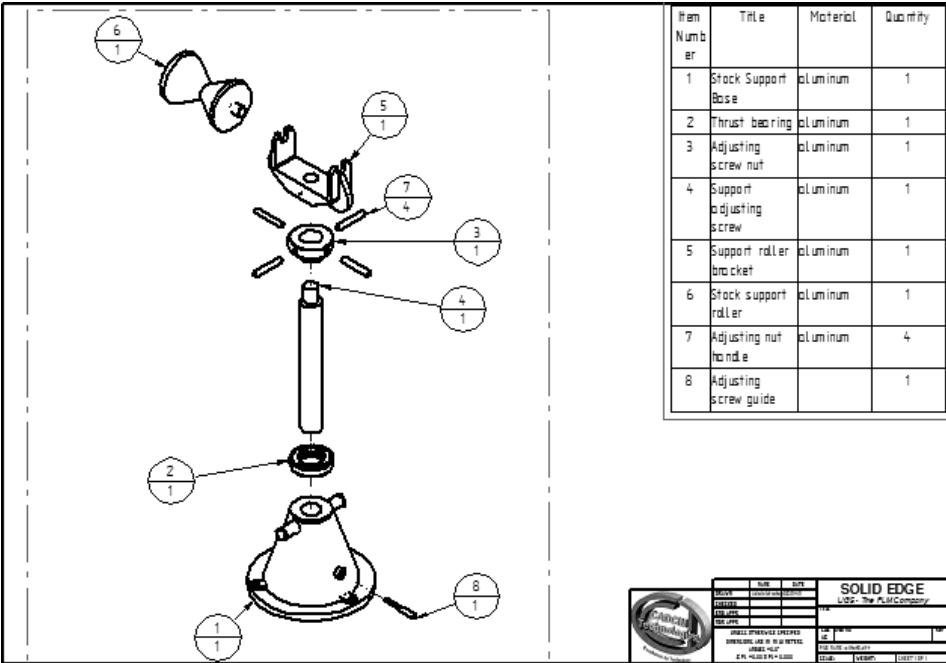


Figure 12-29 The exploded view of the assembly with parts list and balloons

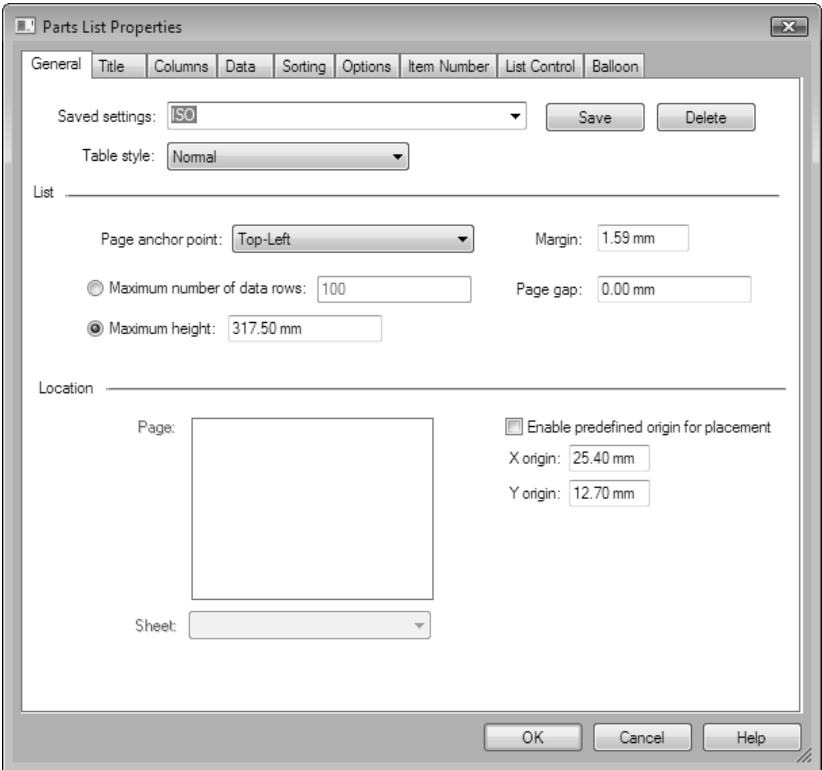


Figure 12-30 The Parts List Properties dialog box

General Tab

The options in this tab are used to specify the text properties of the text in the parts list and are discussed next.

Saved settings

This drop-down list contains the styles that are saved for the parts list. By default, three styles, **ISO**, **Exploded**, and **ANSI**, are saved. If you need to save a new style, enter a name in this drop-down list and choose the **Save** button. You can also select the required table style from the **Table style** drop-down list available below the **Saved settings** drop-down list.

List Area

The options in this area are used to specify the location to place the table in the drawing view, set the maximum number of rows, maximum height, and many other parameters of the table.

Location Area

The **Page** list in this area lists the pages in the table. The **Sheet** drop-down list displays the sheets available in the drawing. On selecting the **Enable pre defined origin for placement** check box in this area, you can specify the x and y coordinates to define the origin of the table.

Title Tab

You can add titles to the table using the options in this tab. On choosing the **Add Title** button, the **Title text** area will get activated. You can enter the desired title in this area. You can also set the number of titles for the table by using the **Number of Titles** edit box. Additionally, you can specify the sequence of titles by using the **Title** spinner. Using the options in the **Position** drop-down list, you can specify the position of each title in the table. You can delete the selected title by using the **Delete Title** button.

Columns Tab

The options in this tab are used to specify various properties of columns in the parts list. These options are discussed next.

Columns

This display box lists all the columns that will appear in the parts list. It also displays the order in which the columns will be displayed. You can change the order of the columns by using the **Move Up** and **Move Down** buttons. You can add columns to this box by selecting the required column from the **Properties** display box and choosing the **Add Column** button. To remove columns from the table, choose the **Delete Column** button.

Properties

This display box lists all column headings that can be displayed in the parts list. You can also select the **User Defined** property to create a column without any heading.

Column Format Area

The options in this area are used to control the column headings to be displayed in the table. You can specify the required headings by using the **Columns** display box. You can

also specify the alignment of the text as well as its position in various cells of the parts list. Moreover, you can set the width of the column in this area.

Data Tab

The options in this tab are used to format the data in the parts list tables. You can insert or delete columns or rows, as well as move the rows up or down by using the options in this tab. You can change the format of any column by choosing the **Format Column** button in this tab. On choosing this button, the **Format Column** dialog box will be displayed. Using this dialog box, you can set the column width, text, position, alignment and other parameters of the header.

Sorting Tab

The options in this tab are used to specify the criteria to sort the parts in the parts list.

Options Tab

The options in this tab are used to control the numbering of items in the parts list. You can also specify the type of parts to be displayed in the list by using the options in this tab.

Item Number Tab

The options under this tab allow you to edit the item number associated with each item in the parts list.

List Control Tab

The options under this tab are used to control the display of parts in the parts list. From these options, you can select the parts that you want to exclude from the assembly. These options are discussed next.

Top-level list (top-level and expanded components)

When this radio button is selected, only the top-level assembly is searched for the parts to list them in the parts list. If the assembly contains subassemblies, they are listed as a single part and the parts of the subassemblies are not listed in the parts list.



Note

Remember that the title in the parts list will be listed only if you have entered it in the file properties of that part.

Atomic List (all parts)

This radio button, when selected, specifies that all parts will be listed in the parts list. If the assembly contains subassemblies, all parts of the subassemblies will also be listed in the parts list.

Exploded List

When this radio button is selected, all parts, subassemblies, and subassembly parts are displayed in the parts list. In this list, a part may be displayed multiple times based on the number of times it has been used in a subassembly. The sub options of exploded list are discussed next.

Use level based item numbers. When this check box is selected, the parts under a subassembly receive the item number based on the hierarchy level of the subassembly. For example, 1.1, 1.2, 1.3.....and so on.

Show top assembly in list. When this check box is selected, a row will be inserted in the part list for entering the name of the top level assembly.

Expand weldment subassemblies. When this check box is selected, the weldment assembly will expand and its parts will be displayed in the part list. If this check box is unselected, the weldment assembly will be treated as the single part.

Selected Item Area

The options in this area are used to exclude or include the selected part from the parts list. To exclude a part, select it from the adjacent tree and then select the **Exclude** radio button.

Subassemblies Area

The options in this area are used to specify if a subassembly exists and also if you want it to be displayed as a single item or along with its parts in the parts list.

Include only ballooned parts in all drawing views

When this check box is selected, only those parts that were ballooned earlier will be listed in the parts list. This check box is cleared by default.

Exclude parts hidden in all drawing views

When this check box is selected, the hidden parts will be excluded from the parts list. This check box is cleared by default.

Exclude parts marked as reference in all drawing views

If this check box is selected, the reference parts will be excluded from the parts list. This check box is selected by default.

Balloon Tab

The options under this tab enable you to set the display properties of the balloons that appear on the assembly drawing view.

Setting the Text Properties

You can set the properties of the text used in the Parts list or BOM. The steps required to modify the text size are as follows:

1. Select the parts list table from the drawing sheet; the **Edit Definition** command bar is displayed with the current table style displayed in it. You need to modify this current table style.
2. Choose the **Styles** button from the **Styles** group of the **View** tab to display the **Style** window.
3. Select **Table** from the **Style type** drop-down list and then select the current table style from the **Styles** list; the parameters of the current table style are displayed in the **Description** box.

4. To modify the parameters of the table style, choose the **Modify** button from the **Style** dialog box; the **Modify Table Style** dialog box is displayed.
5. Choose the **Text** tab from it and then select **Normal** from the **Text styles** list. On doing so, the **Modify** button gets activated.
6. Choose the **Modify** button; the **Modify Text Box Style** dialog box gets invoked. Choose the **Paragraph** tab from it; the text settings are displayed in this tab.
7. Specify the required font size in the **Font size** edit box and press ENTER.
8. Choose **OK** from the **Modify Table Style** dialog box and then **Apply** from the **Style** dialog box to accept and exit the settings.

Steps to Generate Parts List and Balloons

In this section, you will learn how to generate the parts list and balloons by setting the options in the **Parts List Properties** dialog box.

The following steps explain the procedure for generating the parts list and balloons:

1. Choose the **Parts List** tool from the **Tables** group; you are prompted to select the drawing view.
2. Select the drawing view that exists on the drawing sheet.
3. Choose the **Properties** button from the command bar to display the **Parts List Properties** dialog box.
4. Choose the **Columns** tab.
5. In the **Columns** display box, select **Document Number** and choose the **Delete Column** button.
6. Choose the **List Control** tab.
7. From the **Global** area, select the **Atomic List (all parts)** radio button and choose the **OK** button to exit the dialog box.
8. Choose the **Auto-Balloon** button from the command bar, if it is not already chosen.
9. Click in the drawing area to place the BOM and balloons. You will notice that the balloons are displayed showing both the item number and the quantity.
10. To remove the quantity from a balloon, select all the balloons by pressing the CTRL key; the command bar is displayed.
11. Choose the **Item Count** button from the command bar displayed. Now, the balloon will show only the item number.



On performing the above steps, an assembly with parts list and balloons will be generated, as shown in Figure 12-31. This assembly consists of two subassemblies.

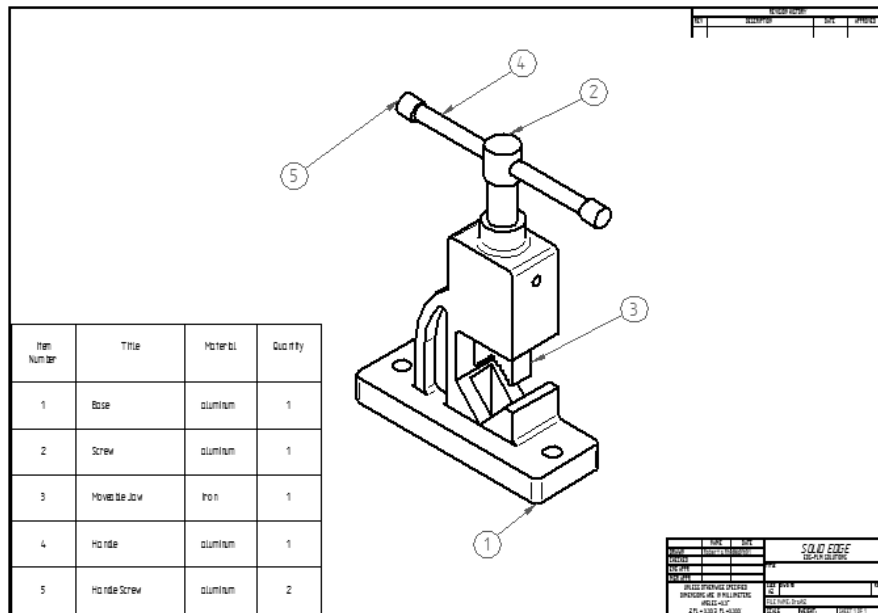


Figure 12-31 Exploded view of the assembly with BOM and balloons

TUTORIALS

Tutorial 1

In this tutorial, you will generate the top view, front view, and right view of the part created in Exercise 1 of Chapter 8, as shown in Figure 12-32. You will use the standard A2 sheet format for generating the drawing views. You will also need to insert your company logo in the sheet.

(Expected time: 1 hr)

The following steps are required to complete this tutorial:

- Start a new draft file.
- Set up the drawing sheet and the background sheet, refer to Figures 12-33 and 12-34.
- Set the projection angle method and save the draft template file.
- Generate the drawing views, refer to Figure 12-36.
- Save the draft file.

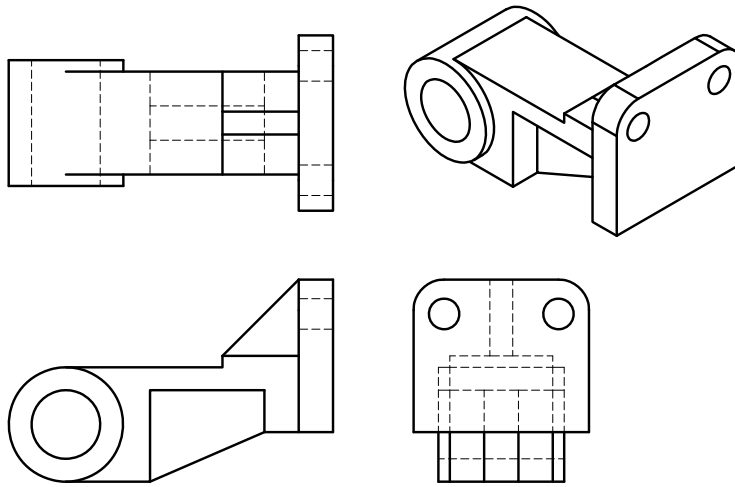


Figure 12-32 Top, front, right side, and isometric views of the model

Starting a New File in the Draft Environment

1. Start Solid Edge ST3 and then choose the **ISO Draft** option from the **Create** area of the welcome screen.

Setting the Drawing Sheet Options

The drawing sheet in Solid Edge is like a blank sheet of paper on which you can draw views. You can add as many sheets as needed in the same drawing file. When you work in the **Draft** environment, there is only one sheet by default. The sheet on which you place the drawing views is called the worksheet and the sheet on which you create the title blocks is called the background sheet. Based on the size of the worksheet you select, the background sheet automatically adjusts itself to the size of the worksheet. To set the worksheet and the drawing sheet for this tutorial, follow the steps given next.

1. Choose **Application Button > Sheet Setup**; the **Sheet Setup** dialog box is displayed. In the **Sheet Size** area of the **Size** tab, the **Standard** radio button is selected by default.
2. In the **Standard** drop-down list, select **A2 Wide (594mm x 420mm)**, if it has not already been selected. Note that in the dialog box, the units are set in millimeters.
3. Choose the **Background** tab in the dialog box. Make sure **A2-Sheet** is selected in the **Background sheet** drop-down list. Choose the **OK** button from the dialog box to exit it.

Next, you need to insert a graphic image as an object in the background sheet. Generally, this method is used to insert a company's logo. As the image has to be inserted in the

background sheet, you need to deactivate the working sheet. Also, note that to insert an image, you should have an image editing program installed on your computer.

4. Choose the **Background** button from the **Sheet Views** group of the **View** tab and then choose the **Working** button from the **Sheet Views** group of the **View** tab to deactivate the Working Sheets option. Notice that four sheets are displayed in the bar below the drawing window.
5. Choose the **A2-Sheet** tab from the bottom of the drawing window and then zoom to fit it on the screen.
6. Now, you need to delete the table at the top. To do so, select the top table, as shown in Figure 12-33. All entities in the table turn magenta in color, which indicates that they are selected. Press the DELETE key to delete the selected entities.

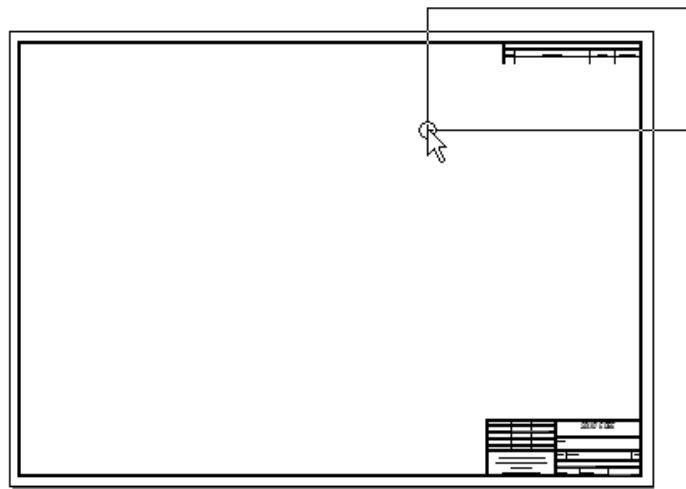


Figure 12-33 Selecting the table

7. Choose the **Image** button from the **Insert** group of the **Sketching** tab to display the **Insert Image** dialog box.
8. Choose the **Browse** button from this dialog box; the **Open a File** dialog box is displayed.
9. Browse and then select the image file of the logo that you want to use, refer to Figure 12-34.

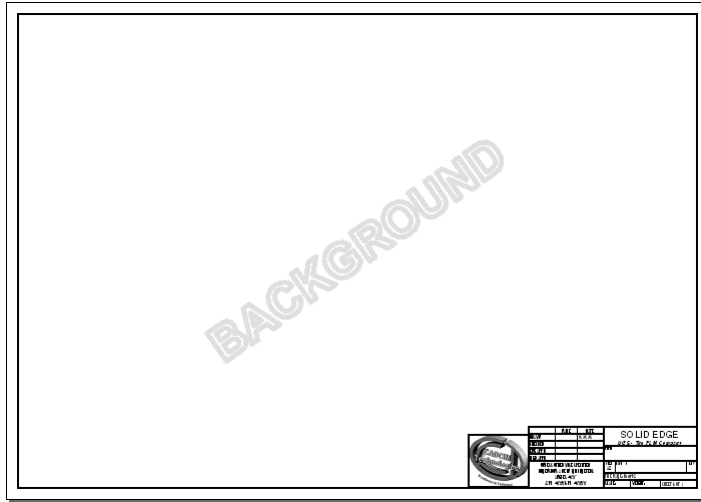


Figure 12-34 Title block with the image

10. Set the transparency option for the logo, if required.
11. After selecting the file, choose the **OK** button from the **Insert Image** dialog box; the image is placed on the sheet.
12. Drag the handles of the image to resize it and move it to the desired location, refer to Figure 12-34.
13. After placing the image, choose **Application Button > Solid Edge Options** to display the **Solid Edge Options** dialog box. Then, choose the **Drawing Standards** tab from this dialog box.
14. In the **Projection Angle** area of this dialog box, select the **Third** radio button to set the current projection type to the third angle projection. Choose **OK** to exit the dialog box.
15. Choose the **Working** button from the **Sheet Views** group of the **View** tab to activate the **Working** button and then choose the **Background** button from the **Sheet Views** group of the **View** tab to deactivate the **Background** button. Next, the drawing views that you generate will be placed on the worksheet and not on the background sheet.

Use the **Fit** button to fit the drawing into the sheet.

16. Save the drawing file with the name *Template.dft* at the following location:
 \Solid Edge ST3\c12.

As the *Template.dft* is to be used in the next tutorial, you need to save it at the following location so that it is displayed in the template list of the **New** dialog box.
 \Program Files\Solid Edge ST3\Template.

You have successfully created a template file. In this textbook, you will further use this drawing file as a template to generate drawing views in the **Draft** environment.

17. Close this file and then start a new file by using the *Template.dft* template.

Generating Drawing Views

The **Drawing View Creation Wizard** enables you to generate multiple views in a single attempt. All views required for this tutorial will be generated at the same time using the options on this page or using this wizard.

1. Choose the **View Wizard** tool from the **Drawing Views** group of the **Home** tab to display the **Select Model** dialog box.
2. Select the part created in Exercise 1 of Chapter 8 and then choose the **Open** button to display the **Drawing View Creation Wizard**.
3. Accept the default options and choose the **Next** button from the **Part and Sheet Metal Drawing View Options** page.
4. Select the **front** view from the **Drawing View Orientation** page and then choose the **Next** button.
5. In the **Drawing View Layout** page, select the views shown in Figure 12-35 and then choose the **Finish** button.

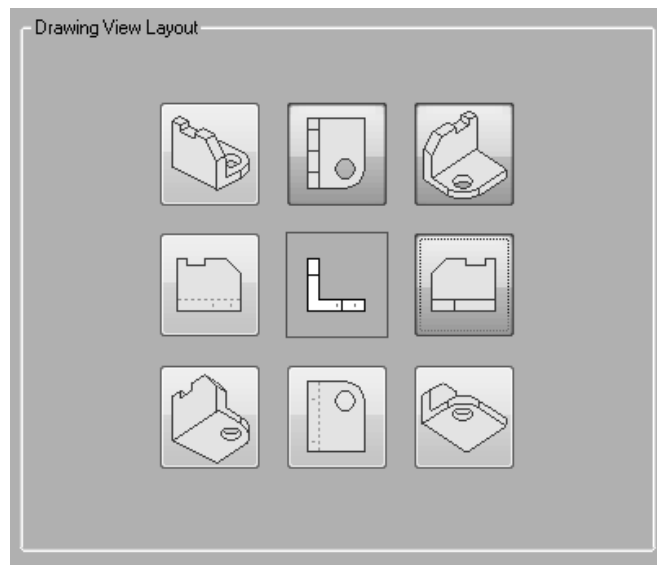


Figure 12-35 The **Drawing View Layout** page of the **Drawing View Creation Wizard**

6. Place the drawing views on the sheet.

Notice that you need to increase the scale of the views. When the scale of any one orthographic view is modified, it changes the scale of other two orthographic views as well.

7. Select the isometric view to display the command bar.
8. Enter **1.2** in the **Scale value** edit box and press ENTER to modify the scale.
9. Similarly, select any one of the orthographic views to invoke the command bar. Modify the value of the scale to **1.2**.
10. You can move the views on the sheet by dragging them. Arrange all views as they are shown in Figure 12-36.

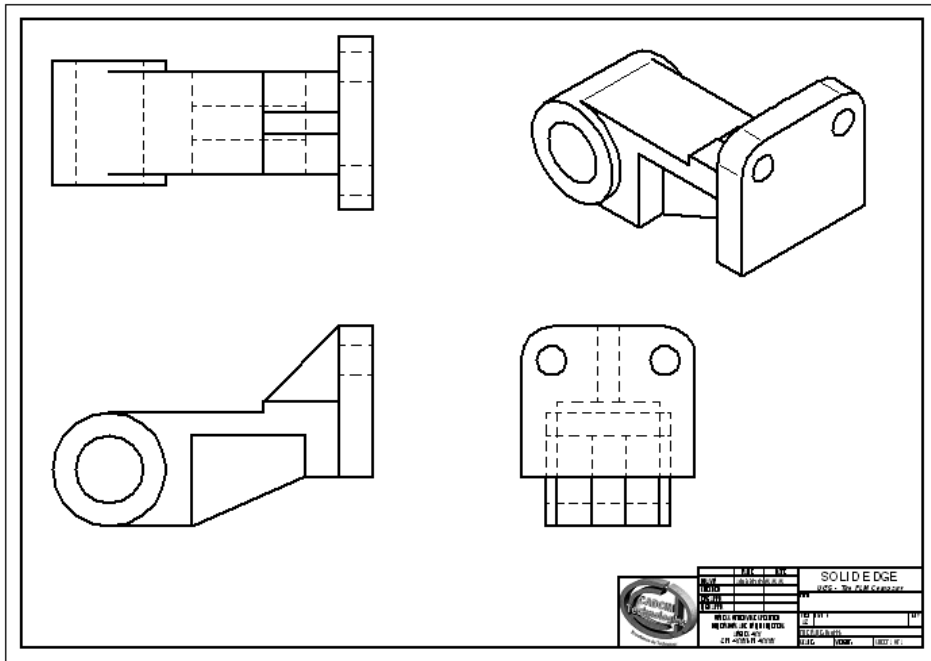


Figure 12-36 Sheet after generating all drawing views

Saving the File

1. Choose **Save** from the **Quick Access toolbar** to display the **Save As** dialog box. Next, save the file with the name *c12tut1.dft* at the following location: *Solid Edge ST3/c12*.
2. Choose **Application Button > Close** to close the file.

Tutorial 2

In this tutorial, you will generate the front view, left-side view, and auxiliary view of the part created in Exercise 2 of Chapter 7. You will also generate dimensions, as shown in Figure 12-37. Use the template that was created in Tutorial 1.

(Expected time: 30 min)

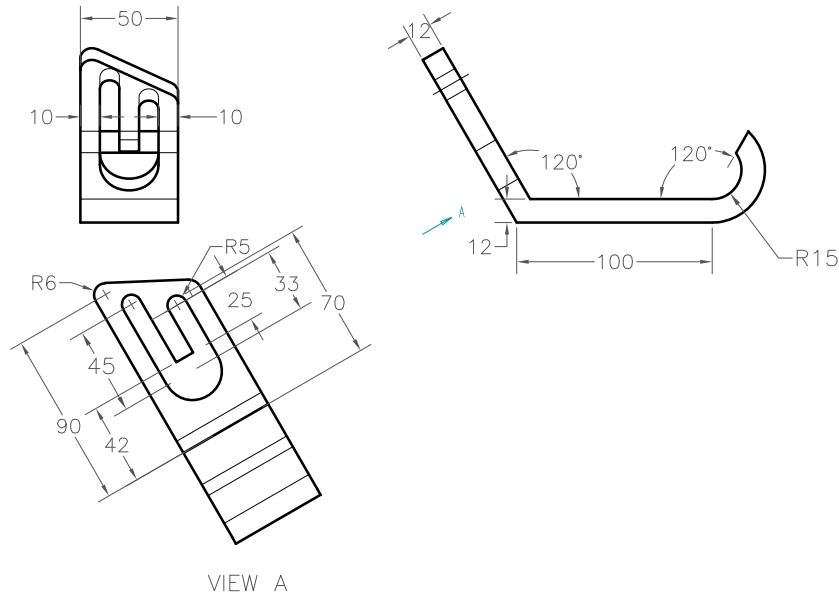


Figure 12-37 Left-side view, auxiliary view, and the front view of the model

The following steps are required to complete this tutorial:

- Start a new draft file.
- Generate drawing views, refer to Figures 12-38 through 12-41.
- Generate dimensions, refer to Figures 12-42 through 12-44.
- Create the remaining dimensions that are not generated, refer to Figure 12-45.
- Save the drawing file and close the window.

Starting a New File in the Draft Environment

You will use the template created in the previous tutorial to generate the drawing views for this tutorial.

- Choose the **New** button from the **Quick Access toolbar** to display the **New** dialog box.
- Select *Template.dft* and choose the **OK** button to exit the dialog box and enter the **Draft** environment.

Generating the Base Drawing Views

As mentioned earlier, the drawing views are generated from their parent part. The following steps are required to generate the drawing views:

1. Choose the **View Wizard** tool from the **Drawing Views** group to display the **Select Model** dialog box.
2. Select the part created in Exercise 2 of Chapter 7 from the dialog box and choose the **Open** button to display the **Drawing View Creation Wizard**.
3. Accept the default options and choose the **Next** button.
4. From the **Drawing View Orientation** page, select **front** and choose the **Next** button.
5. Now, choose the button of the left view in the dialog box and then choose the **Finish** button.
6. Place the drawing views on the sheet.



Note that you need to scale the views. When the scale of any one orthographic view is modified, it changes the scale of other two orthographic views as well.

7. Select one of the views to display the command bar.
8. Modify the value of the scale to **1.2** in the **Scale value** edit box.
9. You can move the views on the sheet by dragging them. Arrange the views as they are shown in Figure 12-38.

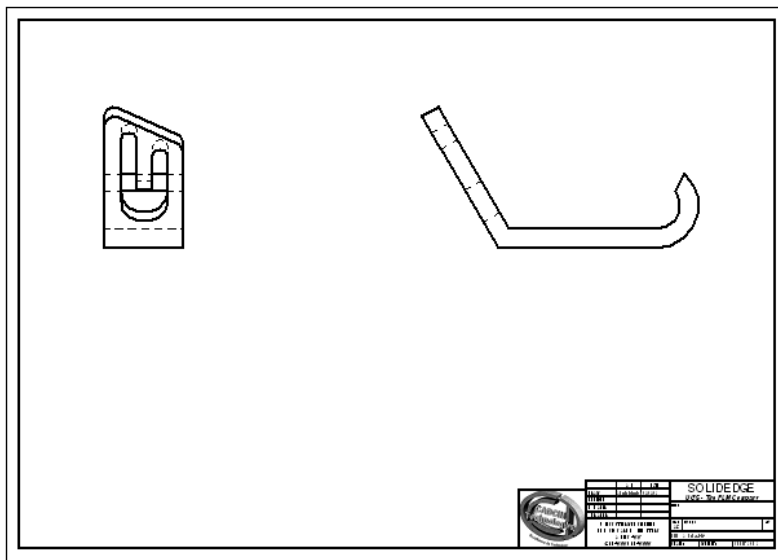



Figure 12-38 Drawing views after scaling

Generating the Auxiliary View

You will generate the auxiliary view by selecting the edge perpendicular to which the view will be projected. This view is created because the true shape of the cut profile can be shown in this view. To generate the auxiliary drawing view, follow the steps discussed next.

1. Choose the **Auxiliary** tool from the **Drawing Views** group; you are prompted to click on the first point of the fold line. The fold line is an imaginary line that is created when you select two keypoints. The auxiliary view is projected about this imaginary line. 
2. Select the edge, as shown in Figure 12-39; an imaginary fold line is formed and the auxiliary view is projected normal to this fold line. Move the cursor to the left to place the view, see Figure 12-40. After you place the view, an arrow pointing in the direction normal to the fold line is displayed, as shown in Figure 12-41.

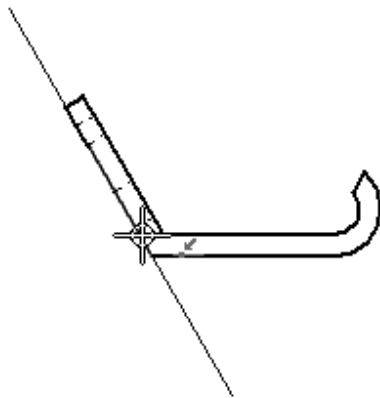


Figure 12-39 Edge to be selected

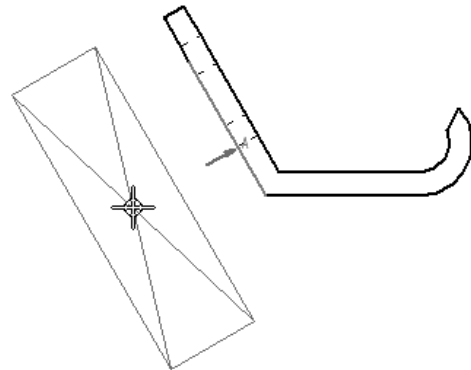


Figure 12-40 Moving the cursor to place the view

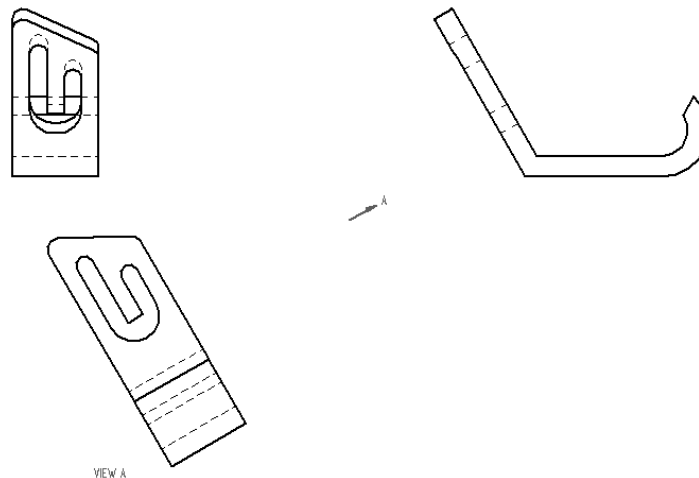




Figure 12-41 Drawing sheet after generating the auxiliary view

Generating the Dimensions

In this section, first the dimensions from the model and then create the remaining dimensions.

1. Choose the **Retrieve Dimensions** tool from the **Dimension** group; you are prompted to select the drawing view. 
2. Select the front view and increase the font size of the dimensions.
3. Exit the current tool and then select a dimension; the current dimension style is displayed in the first drop-down list in the command bar. You need to modify the text size for this dimension style.
4. Press the ESC key to remove the dimension from the selection set. Next, choose the **Styles** button from the **Style** group of the **View** tab to display the **Style** dialog box. Select the dimension style of the current dimensions from the **Styles** list box and then choose the **Modify** button.
5. Choose the **Text** tab and modify the value of the font size to **8.5** in the **Font size** edit box. Choose **OK** and then **Apply** from the **Style** dialog box.
6. Arrange the dimensions, as shown in Figure 12-42, using the dimensioning tools.
7. Choose the **Distance Between** tool from the **Dimension** group of the **Home** tab and then select the **By 2 Points** option from the **Orientation** drop-down list in the command bar. Dimension the auxiliary view with dimensions 90, 42, and 70, as shown in Figure 12-43.
8. To dimension the auxiliary view with dimensions 45, 25, and 33, draw an axis such that it passes through the center of the arc, as shown in Figure 12-43.
9. Select the center of the arc and then the axis, and place 33 as the dimension, see Figure 12-44.
10. Similarly, apply the remaining dimensions to the auxiliary view so that all dimensions are displayed.
11. After dimensioning the auxiliary view, choose the **Retrieve Dimensions** button and select the left-side view to generate the dimensions. You will notice that only dimension 50 is generated. Create the remaining dimensions manually. 

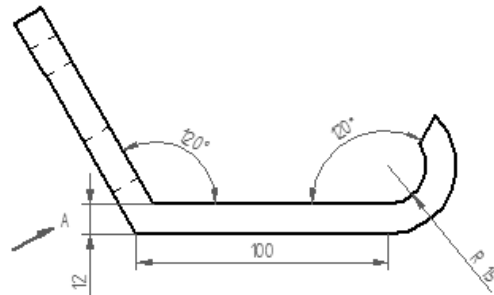


Figure 12-42 Front view after placing the missing dimensions

The drawing sheet after dimensioning the drawing views is shown in Figure 12-45.

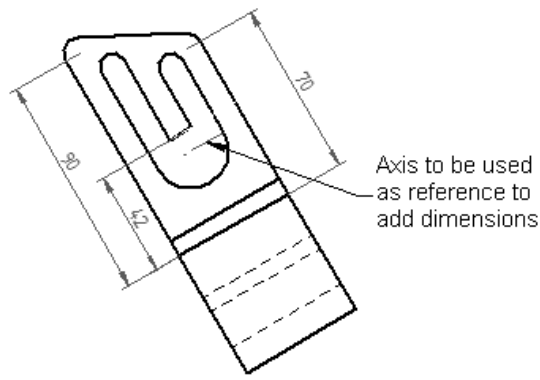


Figure 12-43 Axis drawn to dimension

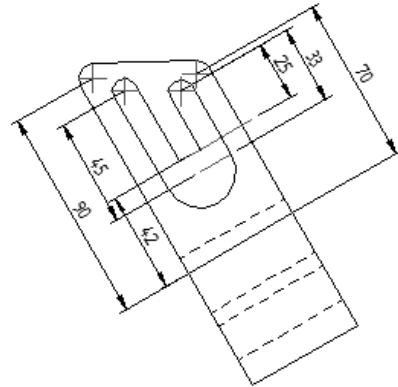


Figure 12-44 Auxiliary view after dimensioning

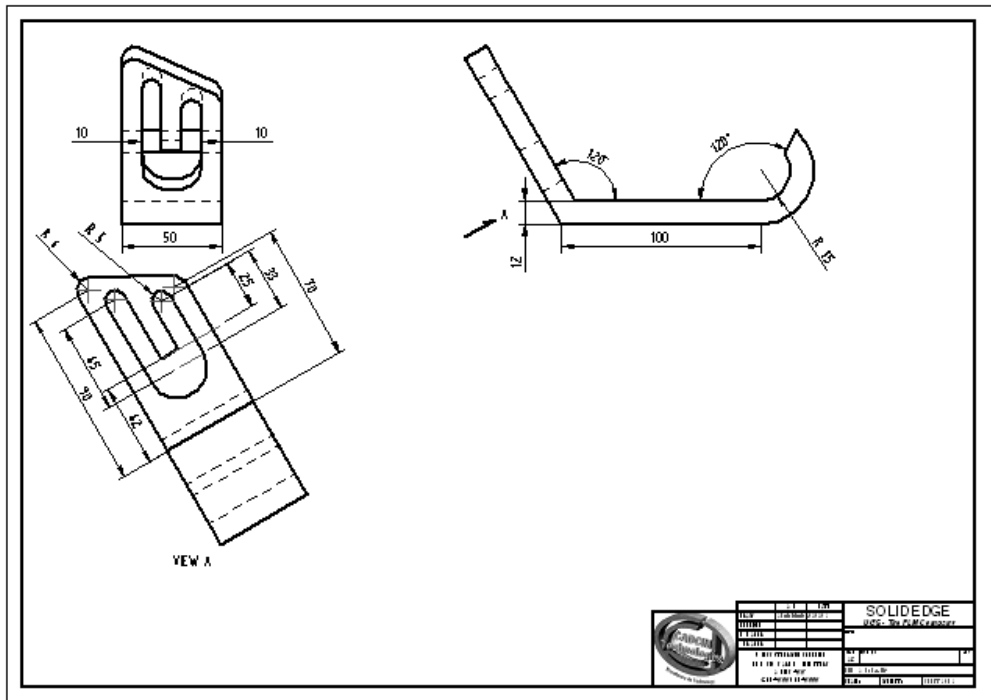


Figure 12-45 Drawing sheet after dimensioning the views

Saving the File

1. Save the file with the name *c12tut2.dft* at the following location: *Solid Edge ST3/c12*.
2. Choose **Application Button > Close** to close the file.

Tutorial 3

In this tutorial, you will create the drawing views shown in Figure 12-46 in AutoCAD. You can also download this drawing file from <http://www.cadcam.com>. The path of the file is as follows: *Textbooks > CAD/CAM > Solid Edge > SolidEdge ST3 for Designers*. Next, you will evolve a 3d model from these drawing views, as shown in Figure 12-47. Finally, you will save the 3D model with the name *c12tut3.par* at the location given below.

\Solid Edge\c12

(Expected time: 45 min)

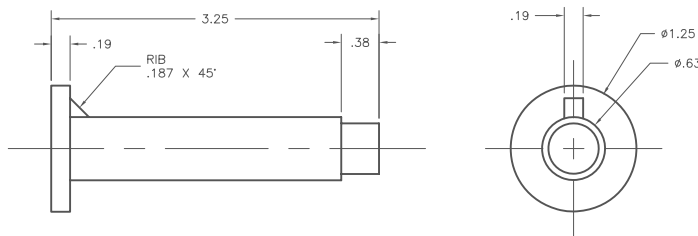


Figure 12-46 The drawing views to be created

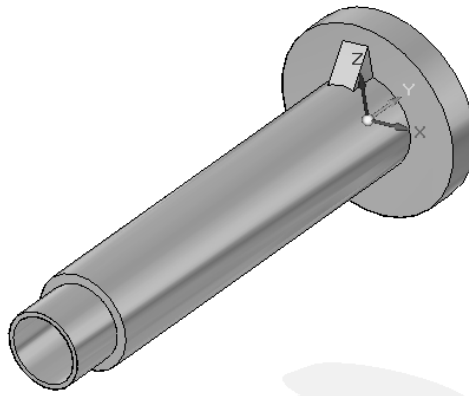


Figure 12-47 Model for Exercise 1

The following steps are required to complete this tutorial:

- Import the drawing views created in AutoCAD into Solid Edge.
- Define the settings for placing the views, refer to Figures 12-45 and 12-46.
- Create the extrusion features using both the drawing views, refer to Figure 12-47 and 12-48.
- Create a hole.
- Create a rib, refer to Figures 12-50 and 12-51 .

Importing the Drawing Views into Solid Edge

You will create the drawing views, as shown in Figure 12-46, in AutoCAD.

1. Create the drawing views in AutoCAD. Make sure the unit used in this drawing is inch. Next, save the file with the name *shaft_bolt.dwg*. Alternatively, download the drawing file from the link specified in the tutorial description.
2. Invoke a new part document by double-clicking on the **ansi part.par** template from the **New** dialog box. This template is used for creating models with inch as unit.
3. Choose the **Open** button from the **Quick Access toolbar**; the **Open File** dialog box is displayed. Next, browse to the *shaft_bolt.dwg* file and choose the **Options** button from the **Open File** dialog box; the **AutoCAD to Solid Edge Translation Wizard** is displayed.
4. Choose the **Preview** button in this wizard; the preview of the drawing is displayed. You can use the display tools like zoom, pan, and fit to view the drawing.
5. Switch off the unwanted layers such as Border, vports, text, if any, and choose the **Next** button.
6. Select **inch** from the **Units** drop-down list in the wizard.
7. In the **Solid Edge template** area of the wizard, choose the **Browse** button; the **New** dialog box is displayed. Choose **ansi draft. dft** from the templates and choose **OK** to exit the dialog box. Next, choose the **Next** button; the **Step 3** page of the wizard is displayed.
8. Select **D Wide(34in x 22in)** as the **Standard** size of the sheet and choose the **Next** button in the wizard; the **Step 4** page of the wizard is displayed with the **Line Type Mapping** area.
9. Accept the default values and choose the **Next** button; the **Step 5** of the wizard is displayed with the **Color to Line Width Mapping** area.
10. Accept the default settings in the **Step 5** of the wizard and then choose the **Next** button. Similarly, accept the default settings in the **Step 6** and **Step 7** of the wizard and then choose the **Next** button; the **Step 8** of the wizard is displayed.
11. Select the **Create a new configuration file** radio button from the **Step 8** of the wizard and then choose the **Copy To** button; the **Save As** dialog box is displayed.
12. Enter a name for the configuration file and then choose the **Save** button to save the file and exit the wizard.
13. After specifying all settings, choose the **Finish** button from the wizard; the **Open File** dialog box is displayed again.
14. Choose the **Open** button to open the AutoCAD file in Solid Edge; the *shaft-bolt.dwg*

is opened in Solid Edge. Choose the **Fit** button to fit the contents into the screen, if required.

15. Choose the **2D Model** tab from the bottom of the window to change the sheet from the Layout view to the Model view.



Note

If you are using the Draft document of Solid Edge, then you can skip the above steps and directly go to the next head, 'Defining Settings for Placing the Views'.

*You can show or hide the layers of the AutoCAD file in Solid Edge. To do so, choose the **Layers** tab of the docking window. Next, select the required layer from the **Layers** area or from the drawing window. Then, choose the **Show Layer** or **Hide Layer** button at the top of the **Layers** area.*

Defining Settings for Placement the Views

In this section, you need to define the settings required for aligning the views.

1. Choose the **Create 3D** tool from the **Assistants** group of the **Tools** tab; the **Create 3D** dialog box is displayed.
2. Make sure the **ansi part.par** template is selected in the **Template file** drop-down list. If not selected, then browse and select it.
3. Choose the **Options** button; the **Create 3D Options** dialog box is displayed.
4. In the **Create 3D Options** dialog box, select the **Third** radio button in the **Projection Angle** area. Next, choose **OK** and then the **Next** button.
5. In the drawing area, select the view at the right by drawing a selection box around it or by using shift and click.
6. Choose the **Next** button from the dialog box and select the left side view by drawing a selection box around it.
7. Choose the **Set Fold Line** button to draw a line at a point from where you need to fold the primary view. Next, create a fold line, as shown in Figure 12-48.



Figure 12-48 Drawing the fold line

8. Choose the **Finish** button; the views are oriented perpendicular to each other, as shown in Figure 12-49.

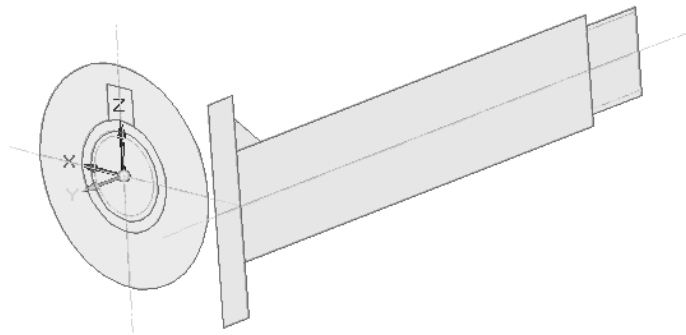


Figure 12-49 Aligning the views perpendicularly

Creating the First, Second, and Third Features

Next, you need to create solid parts using the dimensions of the drawing views.

1. Select the outer circle and invoke the **Extrude** tool; a preview of the extrusion is displayed.
2. Move the cursor toward the front view and snap to the endpoint of the first line segment to define the height of the cylinder, as shown in Figure 12-50. Click when the extrusion height snaps to the endpoint; the first feature is created.



Note

*If you are unable to snap to endpoints, choose the **Keypoints** button from the **QuickBar**; a flyout is displayed. Choose the **All** option from the flyout.*

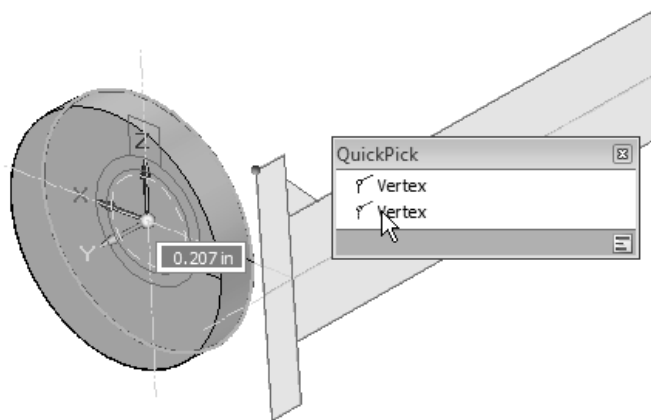


Figure 12-50 Preview of the first feature

3. Next, select the second outer circle and invoke the **Extrude** tool. Now, snap the extrusion height with the endpoint of the second rectangle, as shown in Figure 12-51; the second feature is created.
4. Similarly, create the third feature by snapping the extrusion height of the inner profile with the outermost endpoint.

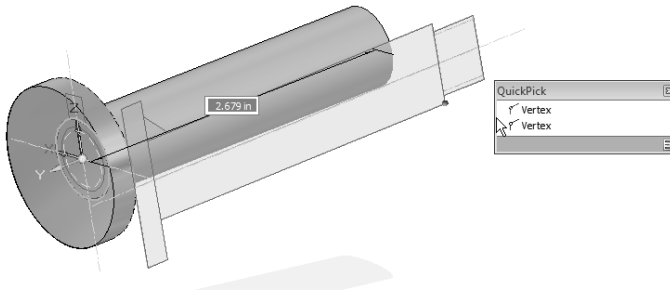


Figure 12-51 Preview of the second feature

Creating the Hole Feature

Next, you need to create a hole or a circular cutout in the third feature.

1. Select the innermost circle that is displayed in the form of hidden lines and move it to the back face of the third feature using the steering wheel.
2. Invoke the **Extrude** tool and select the innermost circle, if it is not selected; the **Extrude QuickBar** is displayed at the top-left of the screen.
3. Choose the **Add** button; a flyout is displayed. Choose the **Remove** button from it.
4. Choose the **From-To** option from the **Extents** drop-down in the **QuickBar**; you are prompted to select the **To** surface. Select the back face of the second feature as the **To** surface; the hole is created.

Creating the Rib Feature

Next, you need to create the rib feature by using the drawing views and then rotate the face to create a taper.

1. Select the rectangular section at the top view and invoke the **Extrude** tool; the preview of the extruded section is displayed.
2. Snap the extrusion height to the rib on the front view, as shown in Figure 12-52; the extrusion feature is created.
3. Now, to create taper, select the back face of the rib; an Extrude handle is displayed. Move the Extrude handle to the bottom edge of the face, so that the Extrude handle is modified into a steering wheel, see Figure 12-53.

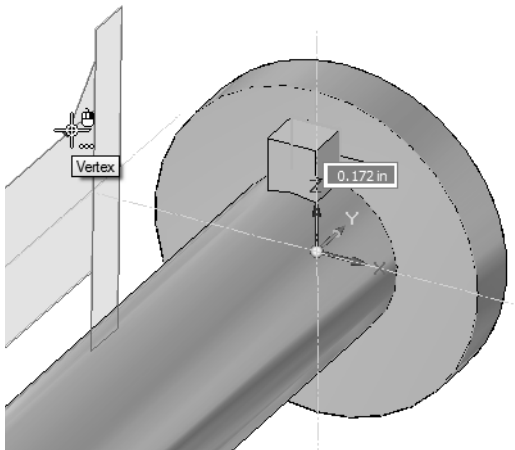


Figure 12-52 Extrusion height of the rib

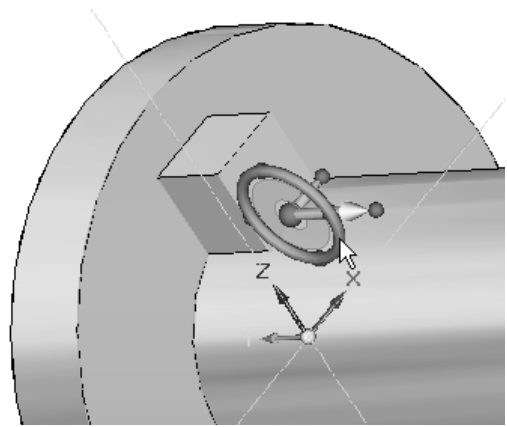


Figure 12-53 Placing the Extrude handle

4. Click on the torus of the wheel such that the back face rotates inward and snaps to the midpoint of the front face of the rib, as shown in Figure 12-54.

The final model after creating all features in the top and front views is shown in Figure 12-55. In this figure, the drawings are hidden because the **Sketches** check box in the **PathFinder** is cleared.

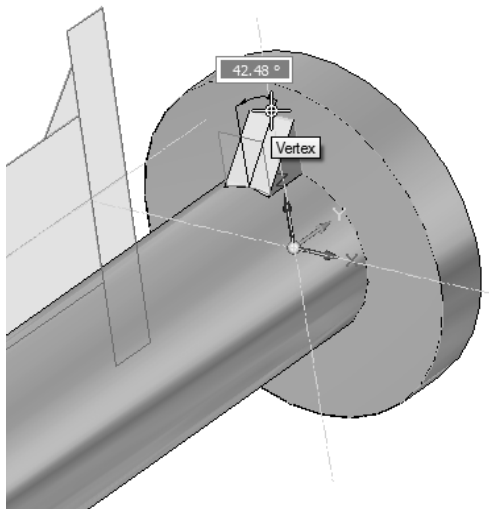


Figure 12-54 Rotating the face for creating rib

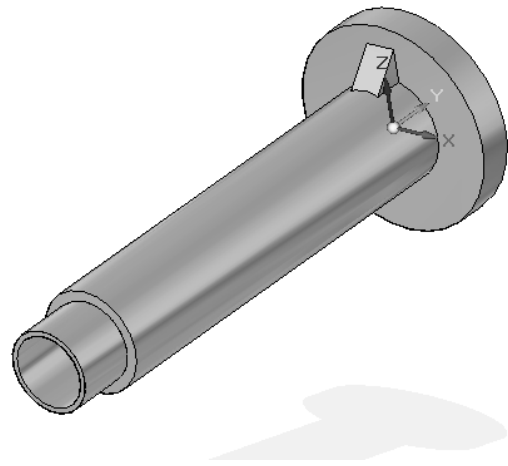


Figure 12-55 The final model

Saving the File

1. Save the file with the name *c12tut2.par* at the following location: *Solid Edge ST3/c12*.

Tutorial 4

In this tutorial, you will generate the exploded drawing view of the assembly created in Chapter 11. You will also add the parts list and balloons to the assembly, as shown in Figure 12-56. **(Expected time: 30 min)**

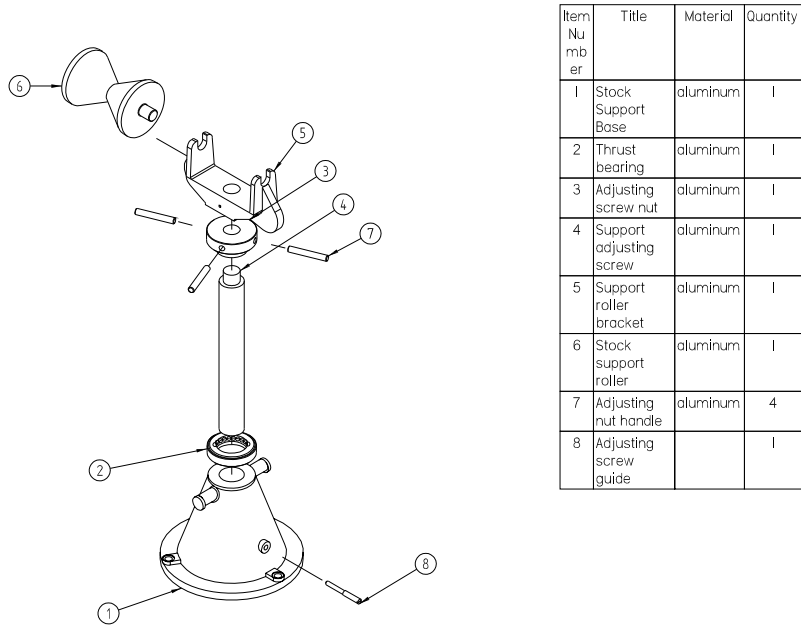


Figure 12-56 Parts list and balloons in the exploded drawing view

The following steps are required to complete this tutorial:

- a. Start a new draft file.
- b. Generate the exploded drawing view.
- c. Generate the parts list and balloons.
- d. Edit balloons and the text in the column, refer to Figure 12-57.
- e. Save the drawing file and close the window.

Starting a New File in the Draft Environment

- 1. Choose the **New** button from the **Quick Access toolbar** to display the **New** dialog box.
- 2. Select **Template.dft** and choose the **OK** button to exit the dialog box. Now, you have entered the **Draft** environment.

Generating the Exploded Drawing View


- 1. Choose the **View Wizard** tool from the **Drawing Views** group of the **Home** tab; the **Select Model** dialog box is displayed.



2. Select **Assembly Document (*.asm)** from the **Files of type** drop-down list.
3. Select the Stock Bracket assembly and choose the **Open** button; the **Drawing View Creation Wizard** is displayed.
4. In the **Assembly Drawing View Options** page, select the **Explode** configuration from the **Configuration** drop-down list. Accept the remaining default options.
5. Choose the **Finish** button to exit the dialog box.
6. Click in the drawing area to place the drawing view.
7. Select the drawing view and then modify the scale of the drawing view to **0.18**.

Generating the Parts List and Balloons

The parts list and balloons can be generated directly from the assembly drawing view.

1. Choose the **Parts List** tool from the **Tables** group; the **Parts List** command bar is displayed and you are prompted to select a view.The icon shows a small table with two columns and two rows, with the text 'Parts List' and a dropdown arrow below it.
2. Select the exploded drawing view.
3. Choose the **Properties** button from the command bar to display the **Parts List Properties** dialog box.
4. Choose the **Columns** tab from the **Parts List Properties** dialog box. In the **Columns** display box, select the **Document Number** option and then choose the **Delete Column** button to remove this column from the table.
5. Select the **Material** option from the **Columns** display box.
6. Next, in the **Column Format** area, change the width value in the **Column width** edit box to **42**. Similarly, change the width value of the **Quantity** column to **42**.

By default, the balloons display the item number and the item count. However in this case, you do not need to display the item count. Therefore, you need to modify the balloon properties accordingly by using the **Balloon** tab.

7. Choose the **Balloon** tab to display the options related to balloons.
8. Modify the value in the **Text size** edit box to **7**.
9. Next, clear the **Item count** check box to make sure that the item counts are not displayed.

10. Choose the **OK** button to exit the dialog box. Make sure that the **Auto-Balloon** button is active in the command bar.
11. Click at the desired location on the drawing sheet to place the parts list table.

Editing the Text Properties

Note that the text in the parts list is too small. Therefore, you need to increase the size of the text in the table. But before doing that, you need to confirm the table style used for the table.

1. Select the parts list table from the drawing sheet; the **Edit Definition** command bar is displayed with the current table style displayed in it.
2. Choose the **Styles** button from the **Style** group of the **View** tab to display the **Style** dialog box.
3. Select **Table** from the **Style type** drop-down list and then select the current table style from the **Styles** list; the parameters of the current table style are displayed in the **Description** box.
4. To modify the parameters of the table style, choose the **Modify** button from the **Style** dialog box; the **Modify Table Style** dialog box is displayed.
5. Choose the **Text** tab in the **Modify Table Style** dialog box and then select **Normal** from the **Text styles** list; the **Modify** button gets activated.
6. Choose the **Modify** button; the **Modify Text Box Style** dialog box is invoked. In this dialog box, choose the **Paragraph** tab; the text settings are displayed in this tab.
7. Enter **7** in the **Font size** edit box and press ENTER.
8. Choose **OK** from the **Modify Table Style** dialog box and then **Apply** from the **Style** dialog box to accept and exit the settings.

The drawing sheet after placing the parts list and balloons is shown in Figure 12-57.

Saving the File

1. Choose **Save** from the **Quick Access toolbar** to display the **Save As** dialog box.
2. Save the file with the name *c12tut4.dft* at the following location:

 |Solid Edge ST3|c12.
3. Choose **Application Button > Close** to close the file.

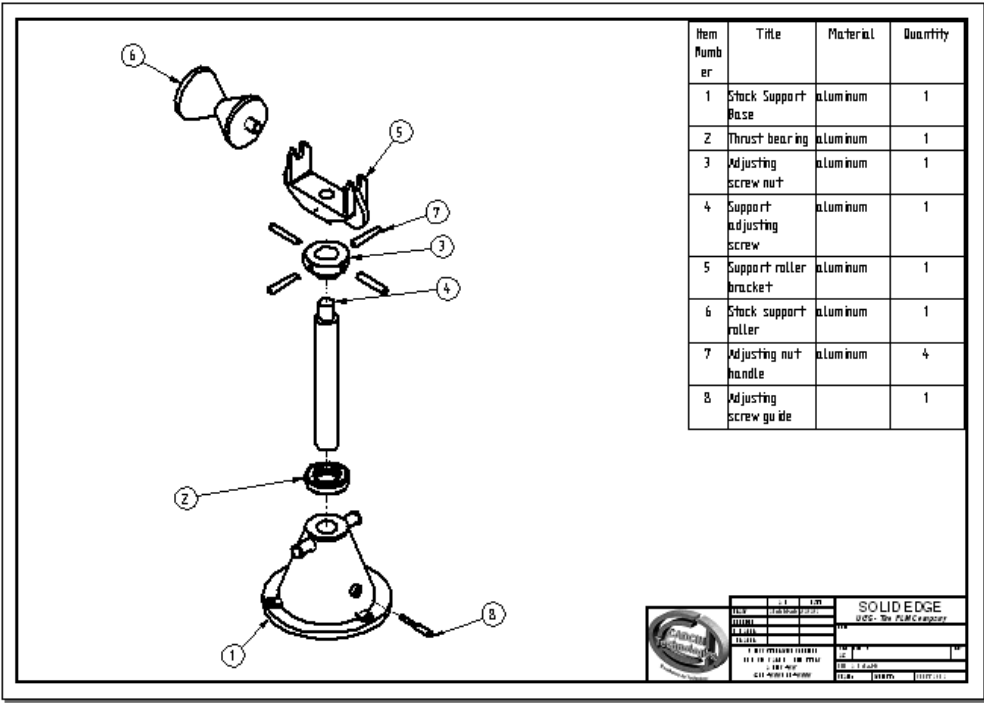


Figure 12-57 Exploded drawing view with parts list and balloons

Self-Evaluation Test

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

1. When you enter the **Draft** environment of Solid Edge, only the drawing sheet is displayed. (T/F)
2. You cannot use an empty sheet for drawing. (T/F)
3. The _____ tool is used to retrieve the dimensions that are applied to a model.
4. The base view is the first view that is generated on the drawing sheet. (T/F)
5. A section view is generated by cutting the part of an existing view using a plane or a line and then viewing the parent view from a direction normal to the section plane. (T/F)
6. The **View Wizard** tool is used to generate the base view. (T/F)

7. _____ is the file extension of the files created in the **Draft** environment of Solid Edge.
8. The _____ view is used for the parts that have a high length to width ratio.
9. A cutting plane can be edited by _____ or by choosing the _____ button from the command bar.
10. To generate a section drawing view of a part, you need a _____.

Review Questions

Answer the following questions:

1. In which of the following views, dimensions cannot be generated from the part?
 - (a) Front
 - (b) Right side
 - (c) Top
 - (d) None of these
2. Which of the following buttons is used to generate the BOM?
 - (a) **Smart Dimension**
 - (b) **Parts List**
 - (c) **Draft View**
 - (d) None of these
3. The _____ tab contains the options that are used to set the dimension style, color of text, font type, font style, and size.
4. Which of the following dialog boxes is displayed when you choose the **View Wizard** tool from the **Drawing Views** group of the **Home** tab?
 - (a) **Properties**
 - (b) **Select**
 - (c) **Drawing View Properties**
 - (d) None of these
5. Before placing the BOM on the drawing sheet, if you choose the **Properties** button from the command bar, the **Parts List Properties** dialog box is displayed. (T/F)
6. When you enter the **Draft** environment, there are two sheets available by default. (T/F)
7. The technique of generating drawing views from a solid model is called generative drafting. (T/F)
8. A detail view is used to display the details of a portion of an existing view. (T/F)

- 9. The need for an auxiliary view arises when it becomes impossible to dimension a geometry in the orthographic view. (T/F)
- 10. In a revolved section view, the section portion revolves about an axis normal to the viewing plane such that it is straightened. (T/F)

Exercises

Exercise 1

Create the exploded view of the assembly that was created in Chapter 11, see Figure 12-58. Generate the BOM and balloons. (Expected time: 30 min)

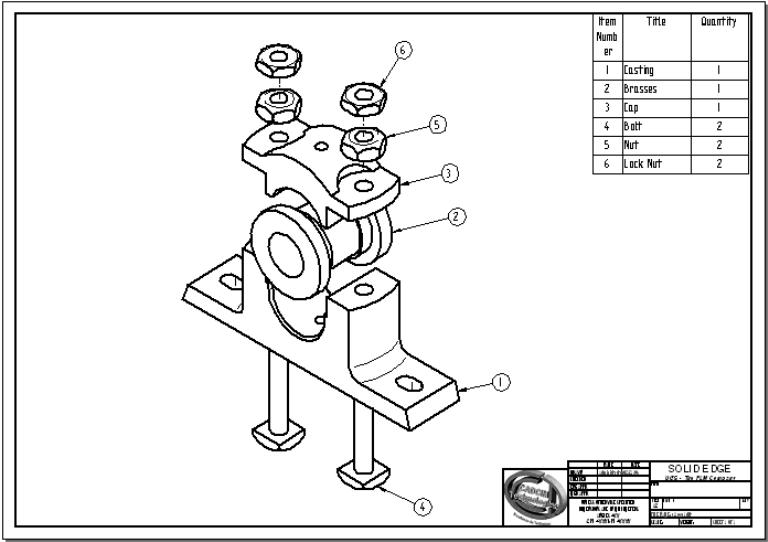


Figure 12-58 Exploded drawing view with the BOM and balloons

Exercise 2

Create the model whose drawing views are shown in Figure 12-59 and then generate the drawing views of the model. Dimension the drawing views, refer to Figure 12-59. (Expected time: 45 min)



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