



Chapter 11

Generating, Editing, and Dimensioning Drawing Views

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the Draft environment.*
- *Learn the types of views that can be generated in Solid Edge.*
- *Generate drawing views.*
- *Manipulate drawing views.*
- *Add annotations to drawing views.*
- *Generate 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 their two-dimensional (2D) drawing views. Solid Edge has a separate environment called the **Draft** environment, which is used for generating drawing views. This environment contains the tools to generate, edit, and modify drawing views.

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

If Solid Edge is running on your computer, invoke the **New** dialog box and select the **Normal.dft** template, see Figure 11-1. 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.

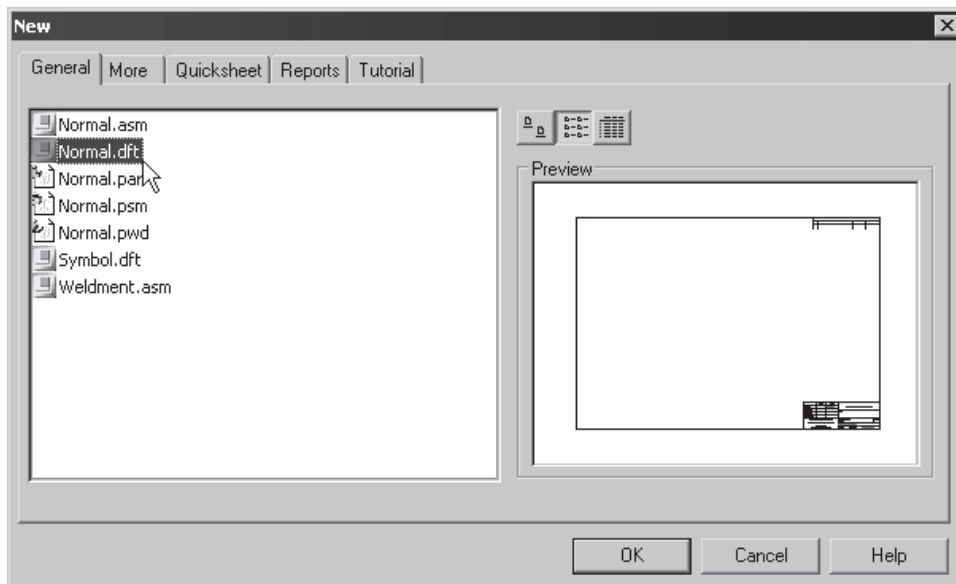


Figure 11-1 Selecting a draft template from the **New** dialog box

When you enter the **Draft** environment, the drawing sheet and the background sheet with borders will be displayed, as shown in Figure 11-2. The sheet displayed is the one on which you will generate the drawing views. Generally, the background sheet is used to add title blocks. The drawing sheet can be modified by choosing the **File > Sheet Setup** option from the menu bar. In the **Draft** environment, you can modify the title block and add more sheets according to your requirement.

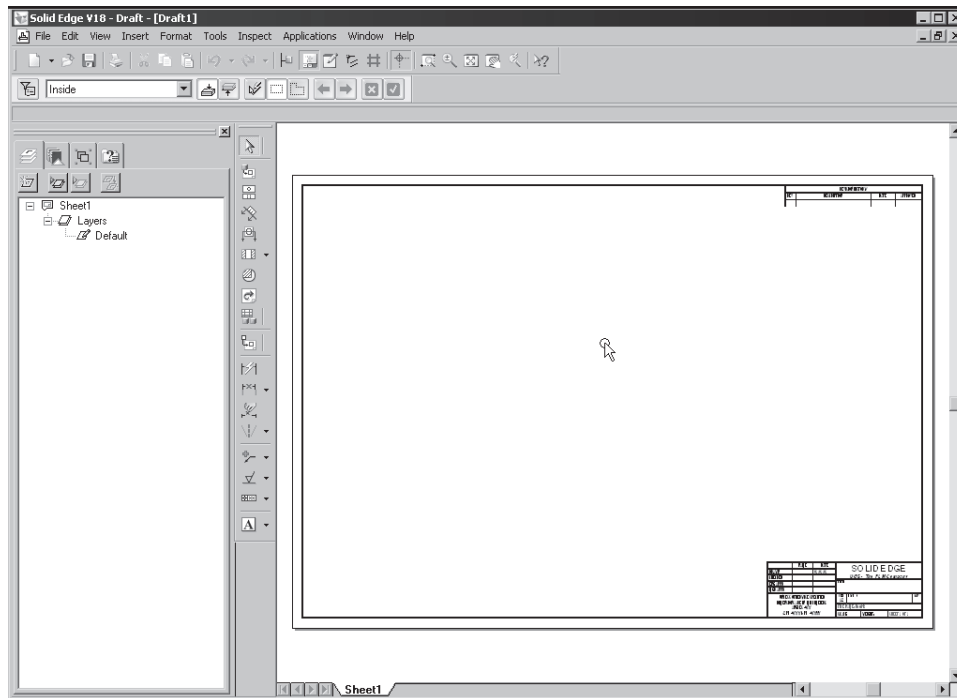


Figure 11-2 The default screen display in the **Draft** environment



Tip. To use an empty sheet without any margin lines or the title block, choose **File > Sheet Setup** from the menu bar; the **Sheet Setup** dialog box will be displayed. Choose the **Background** tab and select the blank space in the **Background sheet** drop-down list. The preview of the sheet in the dialog box shows an empty sheet. Choose the **OK** button to exit the dialog box.

TYPES OF VIEWS THAT CAN BE 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 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. Similarly, any change in the dimensions of the drawing views updates the model. 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 inside 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

In Solid Edge, the base view is generated first. This view is generated using the **Drawing View Wizard** tool. The remaining views are generated by directly or indirectly using the base view. The procedure for generating all six types of views mentioned earlier is discussed next.

Generating the Base View

Toolbar: Drawing Views > Drawing Views Wizard



The **Drawing View Wizard** tool is used to generate the base view. When you choose this button from the **Drawing Views** toolbar, the **Select Model** dialog box will be displayed.

From this dialog box, select the model whose drawing views you need to generate and choose the **Open** button; the **Drawing View Creation Wizard** dialog box will be displayed. Figure 11-3 shows this dialog box when a part file is selected. The options in this dialog box are discussed next.

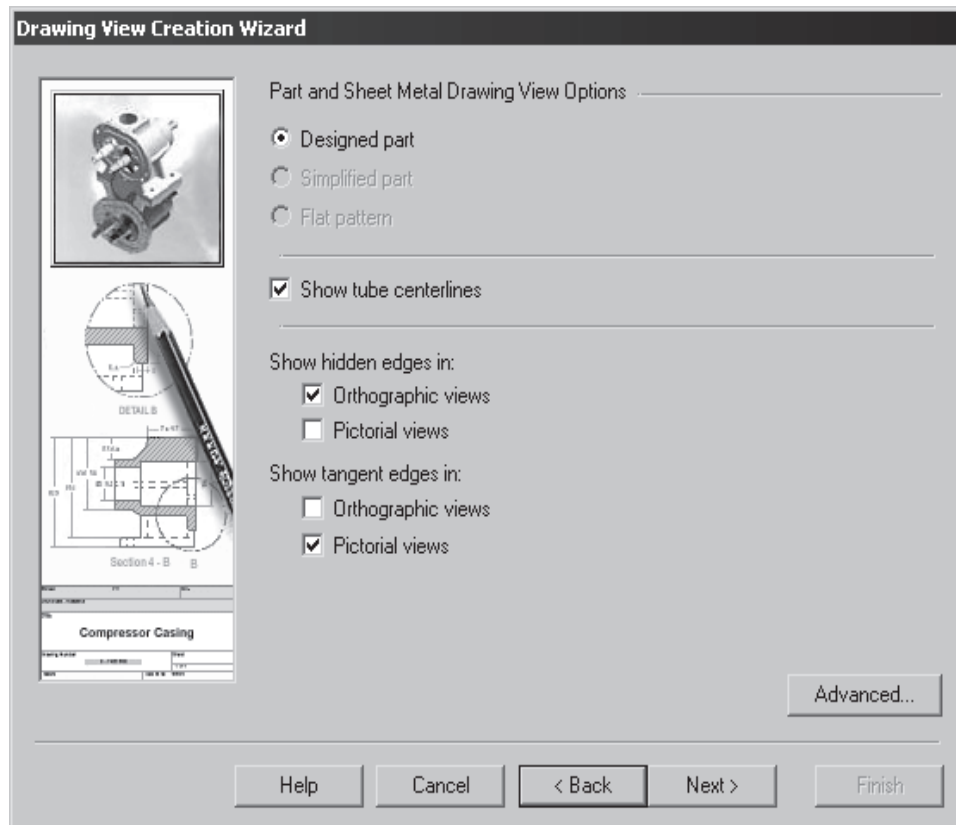


Figure 11-3 The Part/Sheet Metal Drawing View Options page of the Drawing View Creation Wizard dialog box displayed when a part file is selected

Part/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. It is not available 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 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 drawing views other than the orthogonal views. This option is used to display the hidden edges, if any, in the pictorial drawing views. Figures 11-4 and 11-5 show the isometric (pictorial) drawing views with the visible hidden edges and the suppressed hidden edges, respectively.

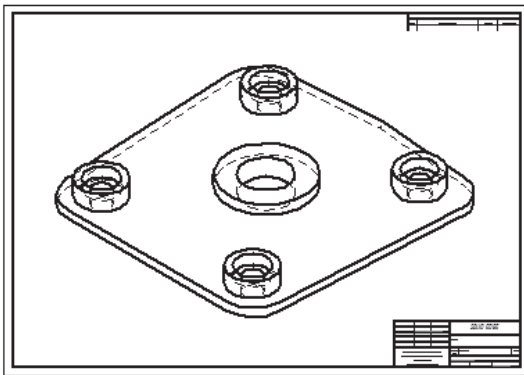


Figure 11-4 Drawing view with hidden edges

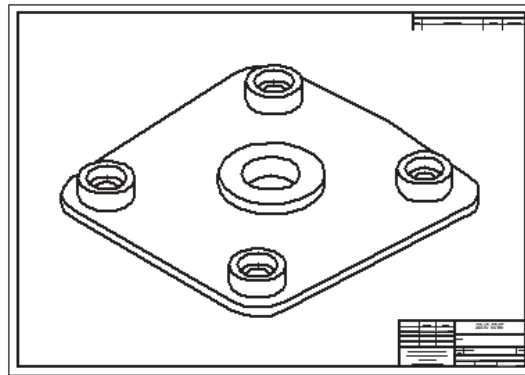


Figure 11-5 Drawing view without hidden edges

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/Sheet Metal Drawing View Options** page, the **Drawing View Orientation** area will be displayed, as shown in Figure 11-6. 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 from any of the standard orientations.

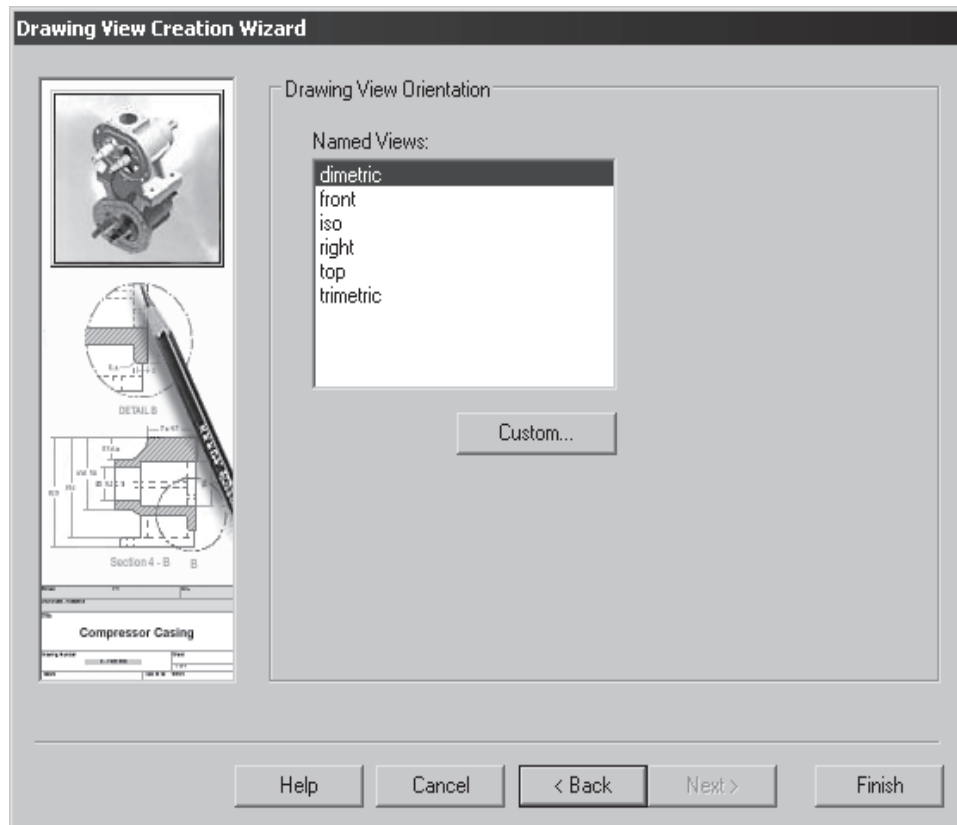


Figure 11-6 The **Drawing View Orientation** page of the **Drawing View Creation Wizard** dialog box

Custom

When you choose the **Custom** button, the **Custom Orientation** window will be displayed, as shown in Figure 11-7. This window displays the part or the assembly that you had earlier selected from the **Select** 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** option is not available in the **Drawing View Creation Wizard** dialog box if the **iso**, **trimetric**, or **dimetric** options are selected from the **Named Views** display box.

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.

Drawing View Layout Page

When you choose the **Next** button from the **Drawing View Orientation** page, the **Drawing**

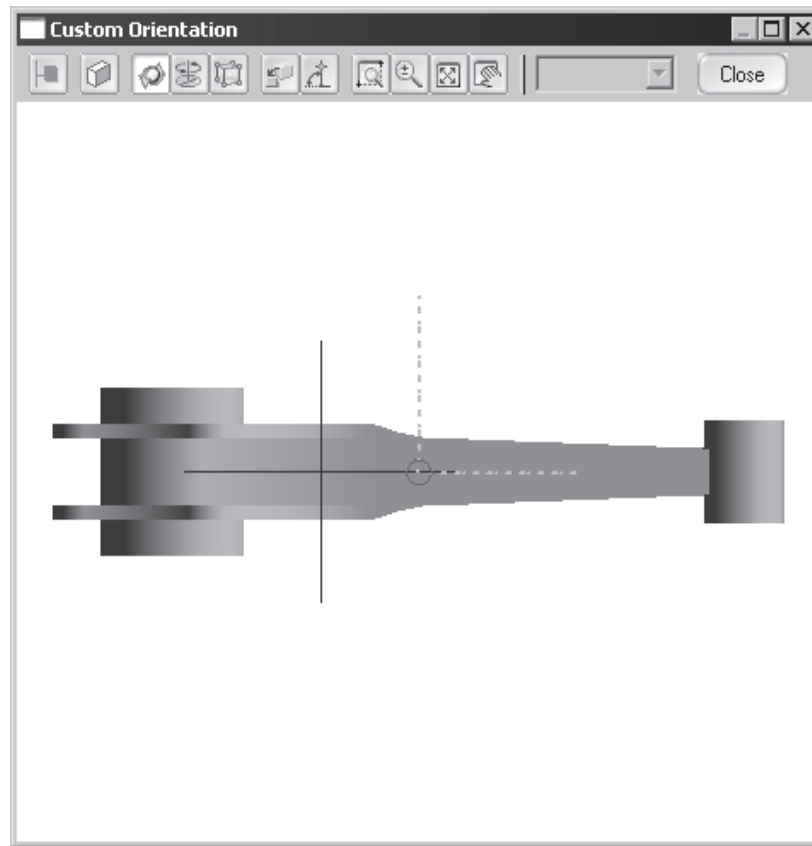


Figure 11-7 The Custom Orientation window

View Layout page will be displayed, as shown in Figure 11-8. The nine orthographic views of a part are displayed. 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

Toolbar: Drawing Views > Principal View



The principal view is generated by selecting an existing view. The view that you can select can be a base view or another principal view. To generate the principal view, choose the **Principal View** button from the **Drawing Views** toolbar; you will be prompted

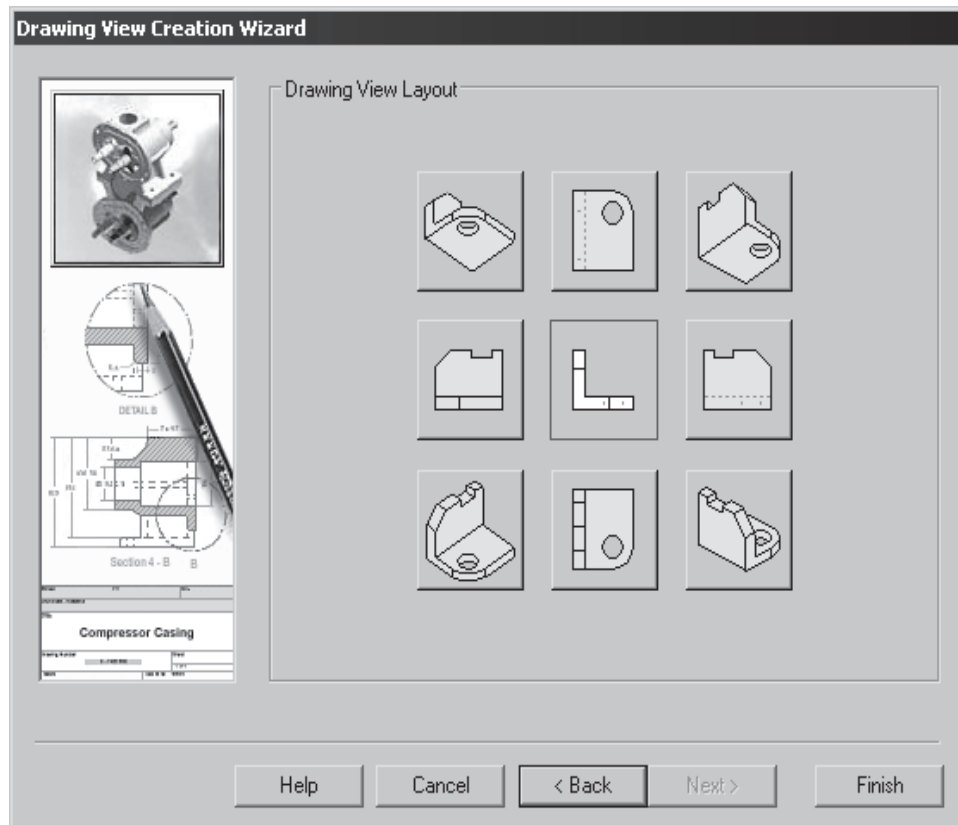


Figure 11-8 The *Drawing View Layout* page of the *Drawing View Creation Wizard* dialog box

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. 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 from the source view.



Note

The principal views cannot be generated from the detail view, section view, and auxiliary view.

*By default, Solid Edge generates drawing views in the first angle projection system. Throughout this book, the third angle projection system is used. To change the default projection system, choose **Tools > Options** from the menu bar; the **Options** dialog box is displayed. Choose the **Drawing Standards** tab and then select the **Third** radio button.*

Figure 11-9 shows the drawing sheet with the base view and principal views. The base view is the front view and the top and isometric views are generated from the front view using the **Principal View** tool.

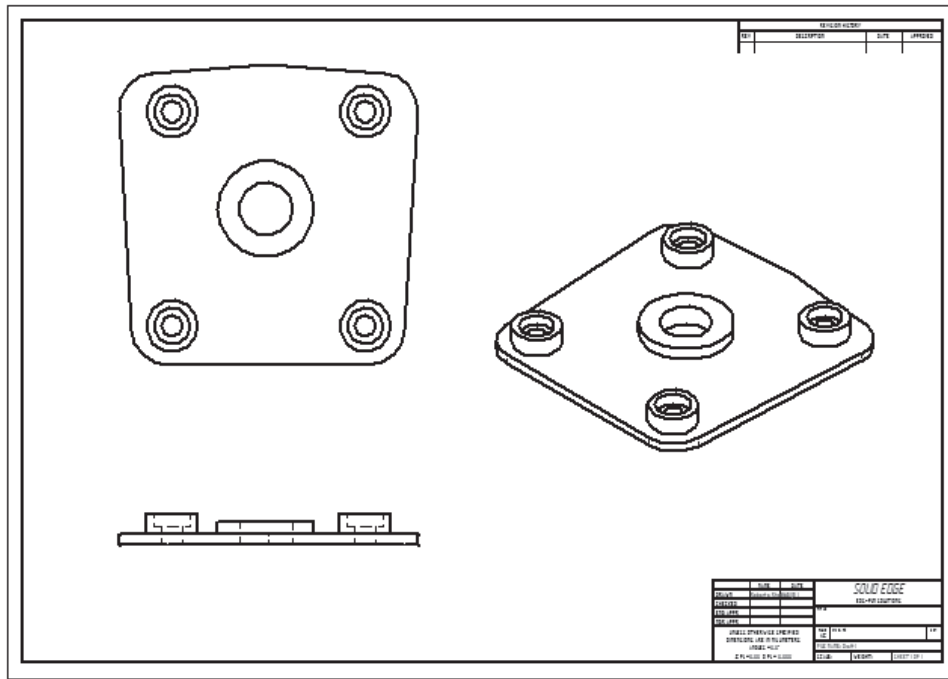


Figure 11-9 Drawing sheet with the base view and principal views



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 shaded. To change the display, select the drawing view and you will find that various shading buttons will be available in the **Drawing View Selection** ribbon bar. Select any shading button and update the drawing view.

Generating the Auxiliary View

Toolbar: Drawing Views > Auxiliary View



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 View** button from the **Drawing View** toolbar; 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 11-10.

The display of this arrow can be changed to a line with double arrows. To change the display, select the arrow and invoke the shortcut menu by right-clicking. Choose the **Properties** option

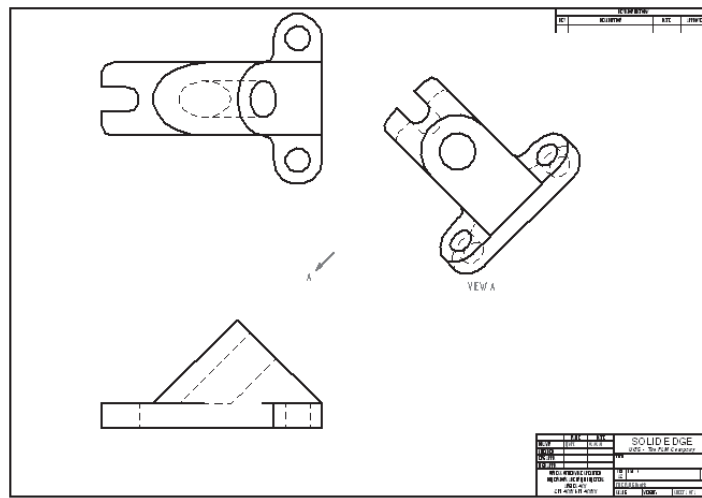


Figure 11-10 Auxiliary view from the principal view

from the menu; the **Viewing Plane Properties** dialog box will be displayed, as shown in Figure 11-11. Select the **Double** radio button from the **View Direction Lines** area in the dialog box and choose **OK**; the arrow will change to double arrows.

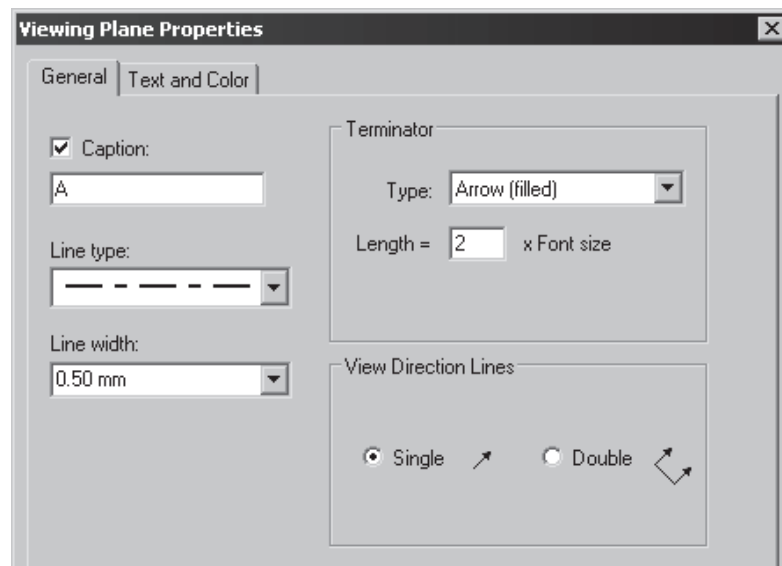


Figure 11-11 Partial view of the **Viewing Plane Properties** dialog box

Need for Auxiliary View

The need for auxiliary view arises when it becomes impossible to dimension a geometry in the orthographic views; for example, refer to Figure 11-10. In the model, the profile that is 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 be easily dimensioned with the true dimensions.

Generating the Section View

Toolbar: Drawing Views > Section View




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 View** tool. For example, a simple section view that requires a cutting plane, revolved section view, and section views that display only the section geometry of the part.

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** button from the **Drawing Views** toolbar; 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 at this point, the view will not be selected. So, it should be noted 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 and an open sketch.
4. After drawing the cutting geometry, choose the **Finish** button from the ribbon bar to exit the cutting plane environment. You will be prompted to specify the direction of viewing.



Note

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

5. Specify the direction of viewing by clicking in the direction pointed by the arrows.
6. Choose the **Section View** button from the **Drawing Views** toolbar; you will be prompted to select a cutting plane.
7. Select the cutting plane and place the section drawing view at the desired location in the drawing sheet. A simple section view is shown in Figure 11-12.

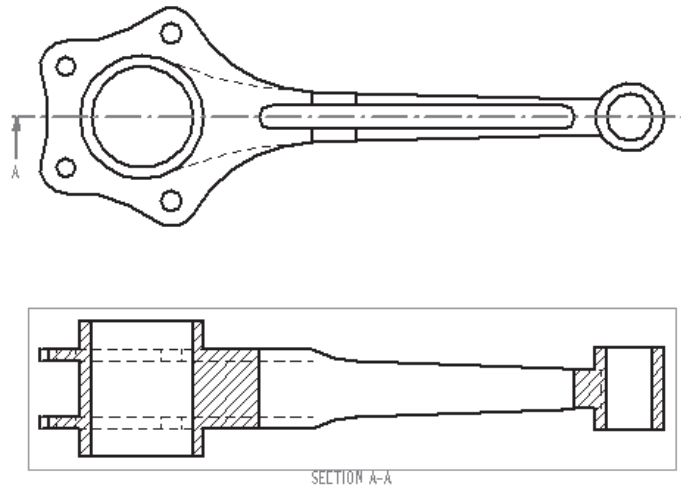


Figure 11-12 Shaded top view and the front section view

Points to Remember for Creating the 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 between the sketch entities and the drawing view can be applied.
4. The cutting plane can be edited by double-clicking on it or by choosing the **Edit** button, which will be displayed in the ribbon 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 sides 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 11-13 shows the views of a model that has three outer features at an equal angle 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, select an existing cutting plane. Because 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 11-14, the inclined line is used to generate the section at the top and the vertical line is used to generate the section view at the right. Before placing the view, choose the **Revolved Section View** button from the ribbon bar.

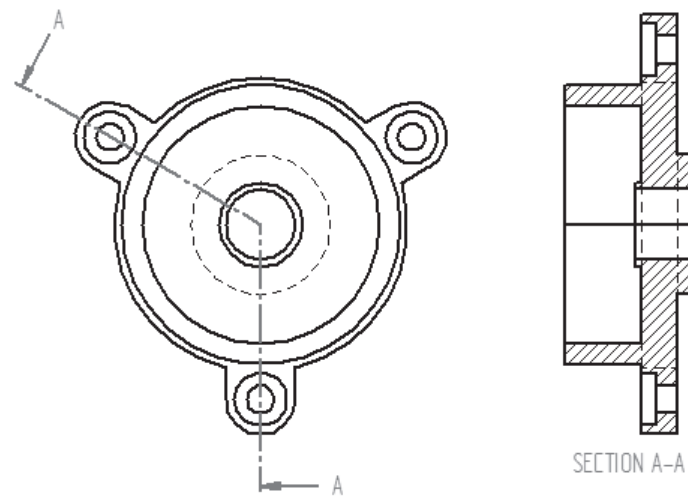


Figure 11-13 Front view and the right-side revolved section view

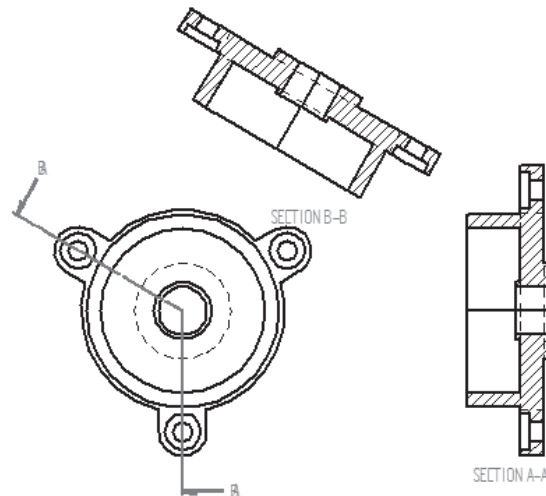


Figure 11-14 Two different revolved section views generated by selecting two different cutting geometries



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 its first and the last entity as the cutting plane.

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

Section View that Displays only the Section Geometry



The **Section Only** button in the ribbon bar is used to generate a section view that displays only that area of the model that is sectioned. Figure 11-15 shows the section view that displays only the section area of the model.

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

1. Choose the **Section View** button from the **Drawing** toolbar; you are prompted to select a cutting plane.
2. Select the cutting plane and move the cursor; a box attached to the cursor also moves along with it.
3. Choose the **Section Only** button from the ribbon bar and place the view by clicking on the right side of the front view. The section view is placed, as shown in Figure 11-15.

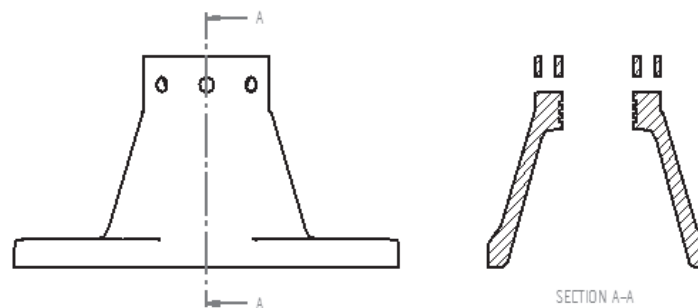


Figure 11-15 Section view displaying only the section area

Generating the Broken-Out Section View

Toolbar: Drawing Views > Section View > Broken-Out Section View



The broken-out section view is used when you want to show a particular portion of the model in section and at the same time not sectioning the remaining model. The **Broken-Out Section View** button is used to generate the broken section view. This button is available on the flyout that will be displayed when you press and hold the left mouse button on the down arrow on the right of the **Section View** button. The following steps explain the procedure for creating this type of view.

1. Invoke the **Broken-Out Section View** button from the **Section View** flyout and select the drawing view where you have to draw the profile for the broken view. You 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 11-16. After drawing the sketch, choose the **Finish** button from the ribbon bar to exit.

3. Next, you are prompted to specify the distance for the depth of the cut. This distance can be specified in the dimension box present on the ribbon bar or on the other orthographic view, as shown in Figure 11-16. After specifying the depth of the cut, you are prompted to select the view that needs to be broken.

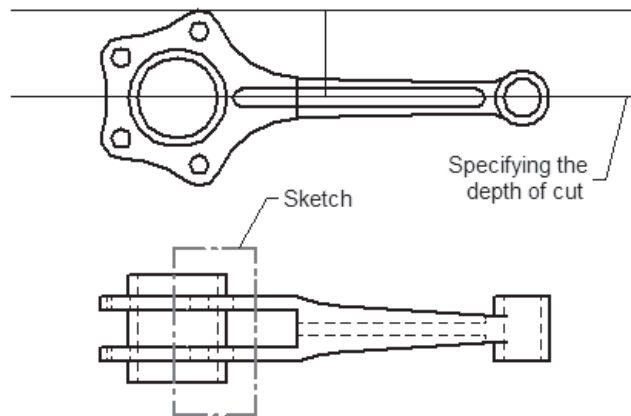


Figure 11-16 Specifying the profile and the depth of the cut

4. Select the isometric view (pictorial view) to generate the broken-out section view, as shown in Figure 11-17.

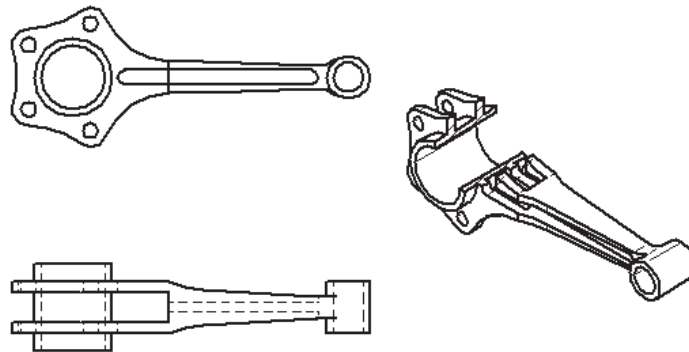


Figure 11-17 Isometric broken section view

Generating the Detail View

Toolbar: Drawing Views > Detail View



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 View button, the **Circular Detail View** button will be chosen by default, and you will be prompted to select the center of the circle. After specifying the center of the circle, you will be prompted to click for the edge 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 11-18.

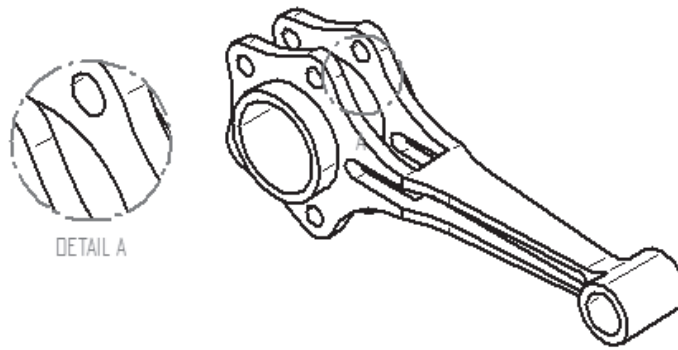
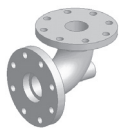


Figure 11-18 Detail view of an isometric view

You can also choose the **Define Profile** button from the ribbon 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 11-19 shows the user-defined closed profile and the detail view created.



Tip: If you drag and drop a part file from the **EdgeBar** 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.

Generating the Broken View

This view is generated on the existing orthographic or pictorial view. It is created by breaking the existing view along the horizontal or vertical direction using the horizontal or vertical lines. This type of view is used for the parts that have a high length to width ratio. Various types of broken views are shown in Figure 11-20.

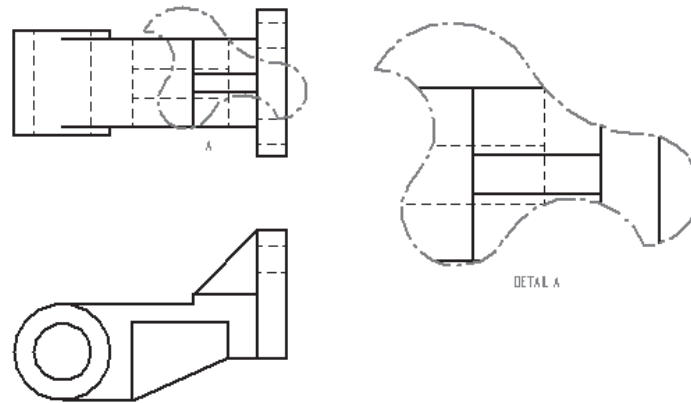


Figure 11-19 User-defined sketch and the resulting detail view

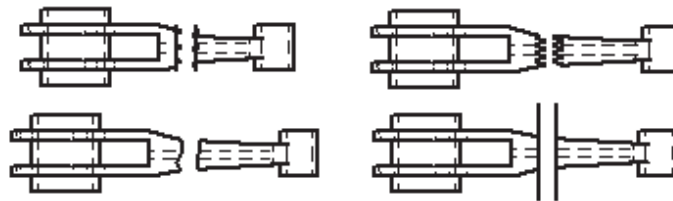


Figure 11-20 Different types of broken views

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** and **Horizontal Break** buttons from the ribbon bar, you can specify whether the drawing view will be broken vertically or horizontally. In the ribbon 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. Choose the **Break Line Type** button from the ribbon bar; a flyout is displayed with four buttons. Select any of these buttons for selecting a line type. The **Break gap** edit box on the ribbon 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 zig zags in the break lines. The **Pitch** edit box is used to specify the pitch of the breaks when you select the short break line type. The **Symbols** edit, which is available for the last break line type, is used to set the

- number of symbols that are added to the break lines.
- 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.
 - 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 ribbon bar. This button toggles the display of break lines in the selected drawing view.



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.

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 2D drawing views is known as interactive drafting. The 2D drawing views can be drawn by choosing the **Draft View** button from the **Drawing Views** toolbar, after customizing the toolbar to add this button. When you do so, you will be prompted to place the view. Select a point on the drawing sheet; the sketching environment will be invoked. The tools available in the sketching environment are the same as those available in the sketching environment of the **Part** environment. After sketching the drawing view, choose the **Return** button from the ribbon bar 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 view. This shows that the two views are aligned. Also, when you select one of the views, a center line that connects the two views will be displayed.

To unalign a view, select the view and right-click to invoke the shortcut menu. The **Maintain Alignment** option has a check mark on its left. Choose this option again from the shortcut menu; the check mark on its left is removed, suggesting that the view is no more aligned. Now, if you move any one of the previously aligned views, the other view does not move.

To completely delete the alignment, choose the **Delete Alignment** option from the shortcut menu and then select the alignment line of the current view.

To create a new alignment of the view, choose the **Create Alignment** option from the shortcut menu; the **Create Alignment** ribbon bar is displayed. You can select a location for the drawing view alignment using the **Alignment position** drop-down list. The buttons available in the

ribbon bar are used to specify whether you want to create a vertical, horizontal, parallel, or a perpendicular alignment. Select the required option and button from the ribbon 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 choosing the **Properties** button from the ribbon bar. When you choose this button, the **Properties** dialog box will be displayed. 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, then the scale factors of both views are modified.

Cropping Drawing Views

A drawing view is cropped to show a particular portion of the drawing view that already exists on the drawing sheet. The portion of the view that lies inside the associated box is retained and the remaining portion is removed. To crop a drawing view, select the view so that it is enclosed inside a boundary. Now, drag one of the boundary's handles until the portion of the drawing you want to display is visible.

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



Note

You cannot crop a detail view and a broken view.

Moving 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 Drawing Views



The drawing views can be rotated by invoking the **Drawing** toolbar. This toolbar can be invoked by choosing the **Draw** button from the **Main** toolbar. Choose the **Rotate** button from this toolbar; 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 ribbon 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** button from the **Drawing** toolbar; you will be prompted to select the area. As you select a closed region, it will be filled with the

hatch pattern. The hatch pattern can be modified by right-clicking on it and selecting the **Properties** option.



Tip: To modify the hatch pattern style of a section view, select the view and right-click to invoke the shortcut menu. Choose the **Properties** option to display the **Drawing View Properties** dialog box. Select the **Display** tab and then from the **Show fill style** drop-down list in the **Selected Part(s) Display** area, select any one of the hatch styles. 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 choose the **Draw in View** option; the view will be opened in a separate window. Now, right-click on the hatch and choose the **Properties** option; the **Fill Properties** dialog box is displayed. Using this dialog box, you can modify the parameters of the hatch. After modifying the parameters, choose the **Return** button from the ribbon bar.

Modifying the Properties of Drawing Views

After generating a drawing view, you can modify its various properties. To set the properties, right-click on the drawing view to invoke the shortcut menu. Choose the **Properties** option to display the **Drawing View Properties** dialog box, as shown in Figure 11-21. There are six tabs in this dialog box.

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, process part intersections, and so on.

The **Display** tab contains the options that are used to set the display of the drawing view. 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 you want to display in the **Parts list** area. 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 of text, font, font style, and size.

The **Annotations** 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 **View Shading** tab contains the options that enable you to set the shading options for the shaded drawing view of the part.

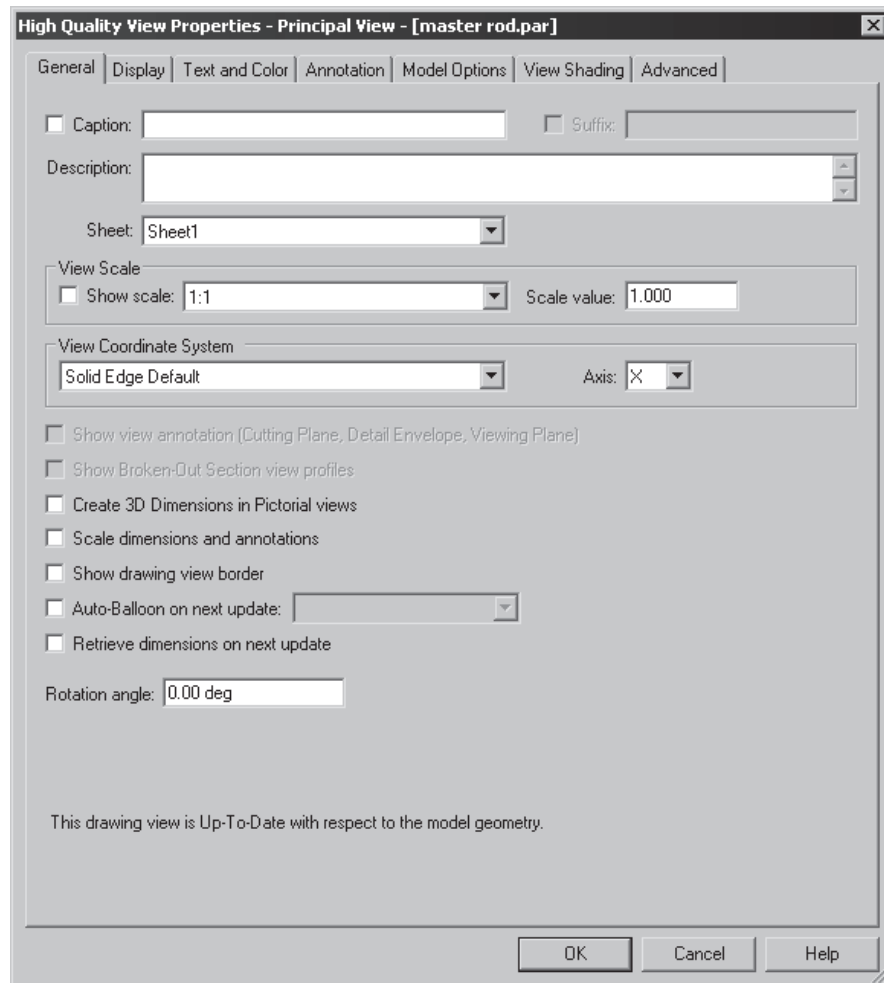


Figure 11-21 The View Properties dialog box for a principal view

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

ADDING ANNOTATIONS TO THE 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. 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 so 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 this type of dimension 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** button from the **Drawing Views** toolbar; the ribbon 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 ribbon bar are discussed next.

Dimension Style Mapping

This button is a toggle and is used to specify whether or not the dimension style mapping set using the **Dimension Style** tab of the **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. However, note that the dimensions that are applied using the **Retrieve Dimension** tool can only be removed using this tool.

Figure 11-22 shows a drawing view after retrieving the dimensions.

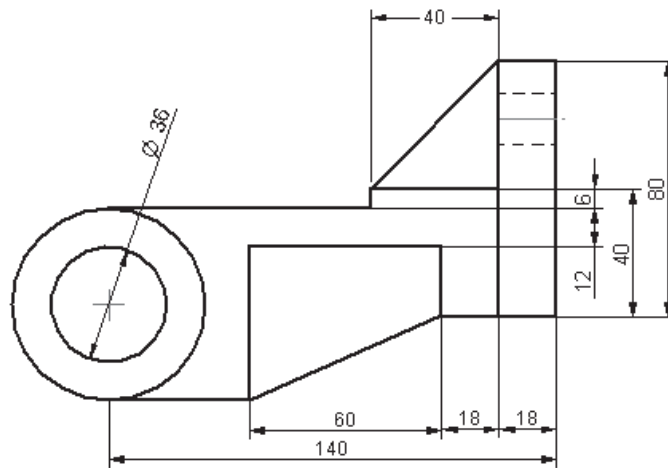


Figure 11-22 Drawing view with dimensions

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 this, choose the **Automatic Center Line** button from the **Drawing Views** toolbar; the ribbon bar will be displayed with various options. If you choose the **Center Line and Center Mark** button from the ribbon 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.

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

Figure 11-23 shows a drawing view with center mark projection lines and Figure 11-24 shows a drawing view with center lines.

Adding Reference Dimensions to the Drawing Views

Although retrieving the dimensions from the parent model is the most effective way of dimensioning, but sometimes you may also need to dimension the drawing views manually. The options for dimensioning a drawing view are available in the **Drawing Views** toolbar. These options are the same as those discussed in sketching. The only dimension tool that is not available in the sketching is the **Chamfer Dimension** tool, which is discussed next.

Chamfer Dimension



This button is available in the **Distance Between** flyout in the **Drawing Views** toolbar. On choosing this button, you will be prompted to select the dimension base line. This line is one of the edges of the chamfer. Next, 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.

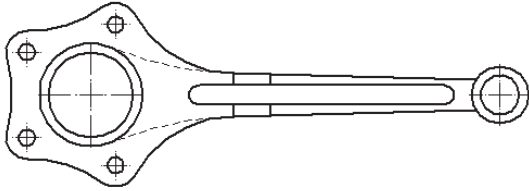


Figure 11-23 Center marks on the holes in a drawing view

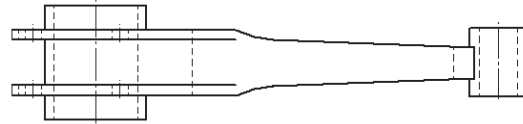


Figure 11-24 Center lines on the holes in a drawing view

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. Alternatively, you can also choose **Insert > New Sheet** from the menu bar; a new sheet named **Sheet2** will be added to the drawing file. Using the other options available in the shortcut menu, you can 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, choose **View > Background Sheet** from the menu bar and then choose **View > Working Sheets** to clear the check mark on the left of this option. 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, choose **View > Working Sheets** to select this option and then choose **View > Background Sheet** to clear it.

GENERATING 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 **Drawing View Wizard** button from the **Drawing Views** toolbar 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** dialog box.
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 shown in Figure 11-25.

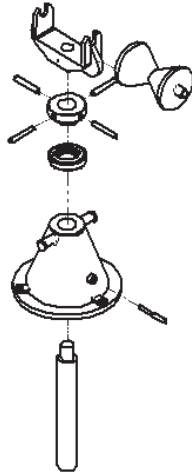


Figure 11-25 Exploded view



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 ribbon bar. When you choose this button, the **Drawing View Properties** dialog box will be displayed, as shown in Figure 11-26. The options in this dialog box are discussed next.

Drawing View Properties Dialog Box

This dialog box contains various options of 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.

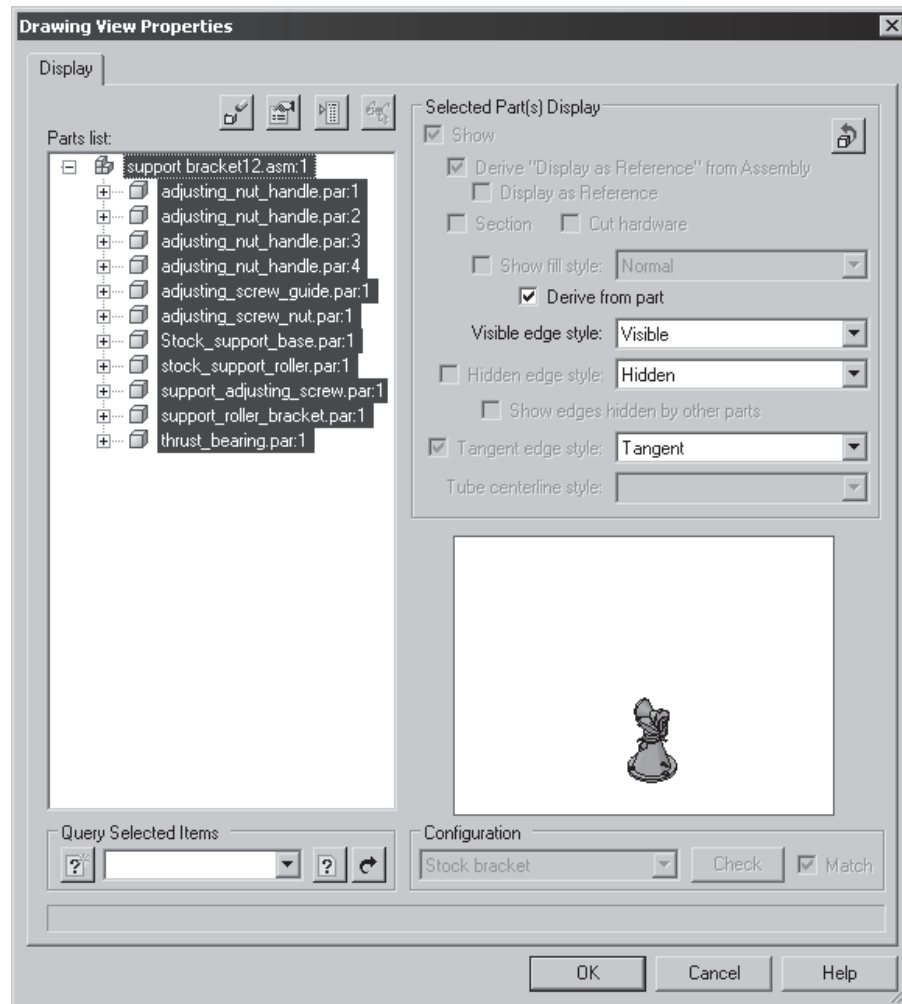


Figure 11-26 The Drawing View Properties dialog box

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** button from the **Drawing Views** toolbar. The parts list created is associative in nature, which means that any modifications made in the part files of the assembly, will be reflected in the parts list also. The following steps explain the procedure for generating a parts list and the balloons.

1. Generate the assembly drawing view.

2. Choose the **Parts List** button from the **Drawing Views** toolbar; the ribbon bar is displayed with the options that can be used to specify whether or not you want to display the balloons and the table. Also, you are prompted to select a drawing view.
3. Select the assembly drawing view. You can also choose the **Properties** button from the ribbon bar and set the display properties of the parts list and balloons.
4. Choose the **Auto-Balloon** button from the ribbon bar.
5. Choose the **Finish** button to place the parts list and balloons, as shown in Figure 11-27.

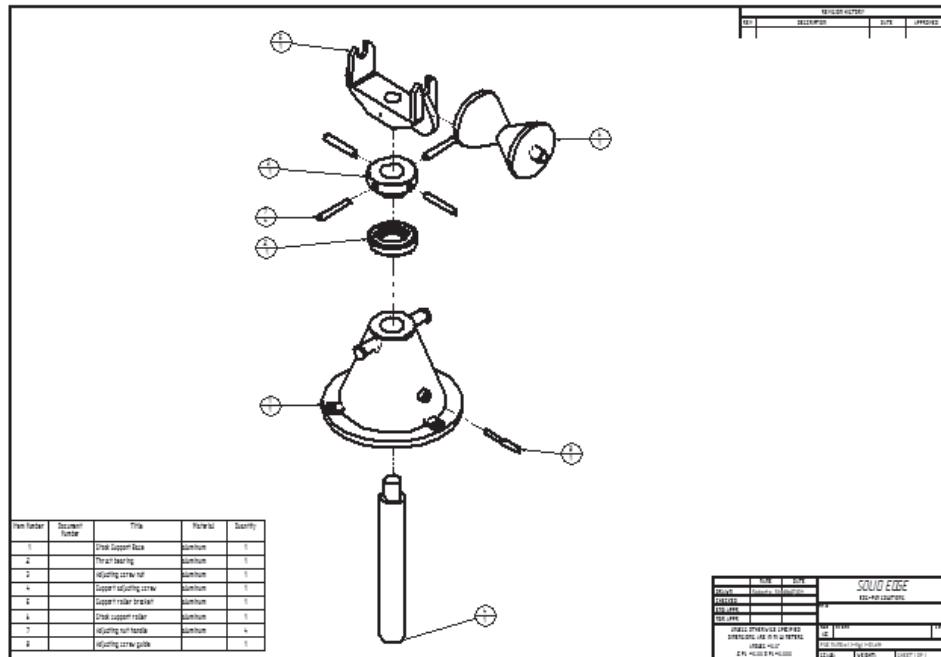


Figure 11-27 Exploded view of the assembly with BOM and balloons

List Properties Dialog Box

While generating the parts list on the drawing sheet and before choosing the **Finish** button, when you choose the **Properties** button from the ribbon bar, the **List Properties** dialog box will be displayed, as shown in Figure 11-28. This dialog box is used to set the parameters of the parts list and balloons. The options available in this dialog box are discussed next.

General Tab

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

Saved settings

This drop-down list contains the styles that are saved for the parts list. By default two styles, **ISO** and **ANSI**, are saved. If you need to save a new style, type a name in this drop-down list and select the **Save** button.

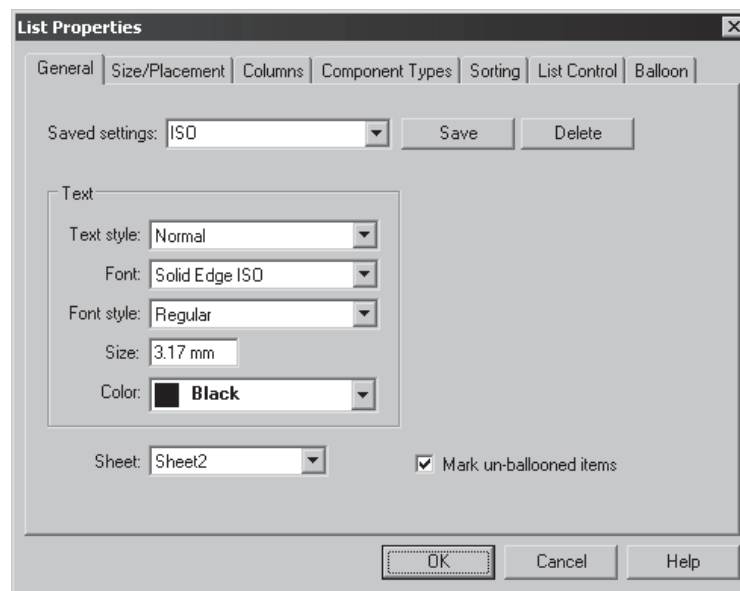


Figure 11-28 The List Properties dialog box

Text Area

This area provides the options that are used to set the properties of the text.

Sheet

This drop-down list is used to specify the sheet on which you want to place the parts list. This option will not be active if there is only one sheet.

Mark un-ballooned items

This check box is selected by default. This indicates that if a balloon is not attached to any part of the assembly, then that part in the parts list will be marked by an asterisk. Balloons are not added to parts that are not visible in the assembly drawing view.

Size/Placement Tab

The options under this tab are used to specify the display of the parts list on the drawing sheet.

Title block

In the title block of a parts list, you specify titles such as the item number, name, quantity, and so on. Using this drop-down list, you can specify the location of the title block in the parts list.

Maximum height of list

You can specify the maximum height of the parts list using this edit box. When the height of this list exceeds this value, the BOM list is divided into multiple sections.

Section gap

This edit box is used to specify the gap between the different sections of the parts list. This is used only when the height of the parts list exceeds the maximum height and it is divided into multiple sections.

Grid color

This drop-down list is used to specify the color of the lines that make up the parts list.

Grid line width

This edit box is used to specify the width of the lines in the parts list.

Text margin

The value in this edit box specifies the gap between the lines and the text in the parts list.

Location Area

The options in this area are used to specify the location of the parts list on the drawing sheet.

Columns Tab

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

Available columns

This display box lists all the column headings that you can display in the parts list.

Columns used

This display box lists all the columns that will appear in the parts list. You can add columns to this box by selecting them from the **Available columns** display box and choosing the **Add** button.

Column Format Area

The options in this area are used to specify a different title for the column heading selected from the **Columns used** display box. You can also specify the alignment of the text in various cells of the parts list.

Component Types Tab

The options under this tab are used to specify the type of components to be included in the parts list.

Sorting Tab

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

List Control Tab

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

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.

Sub-assemblies 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

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 hidden parts

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

Exclude reference parts

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.

Steps to Generate Parts List and Balloons

In this section, you will learn how to generate the parts list and balloons by setting some options in the **List Properties** dialog box. They will be generated, as shown in Figure 11-29. The assembly shown in the figure consists of two subassemblies.

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

1. Choose the **Parts List** button from the **Drawing Views** toolbar; 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 ribbon bar to display the **List Properties** dialog box.
4. Choose the **Columns** tab.

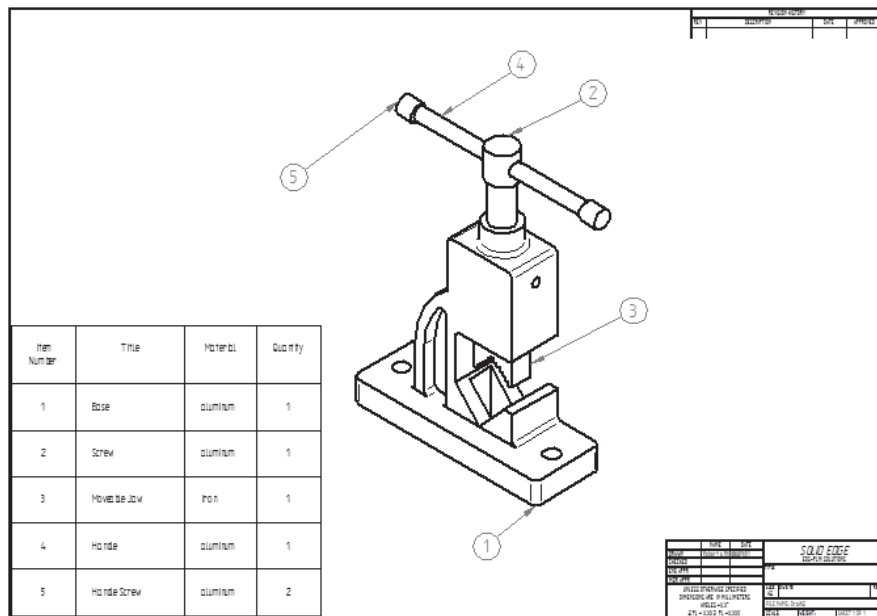


Figure 11-29 Exploded view of the assembly with BOM and balloons

5. In the **Columns used** display box, select **Document Number** and choose the **Remove** 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 ribbon bar, if not already chosen.
9. Choose the **Finish** button from the ribbon bar to display 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 ribbon bar is displayed.
11. Choose the **Item Count** button from the ribbon bar to clear it. Now, the balloon shows only the item number.



TUTORIALS

Tutorial 1

In this tutorial, you will generate the top view, front view, and right-side view of the part that was created in Exercise 1 of Chapter 7 and is shown in Figure 11-30. Use the standard A4 Landscape sheet format for generating the drawing views. You will also insert your company logo in the sheet.
(Expected time: 1 hr)

The following steps are required to complete this tutorial:

- a. Start a new draft file.

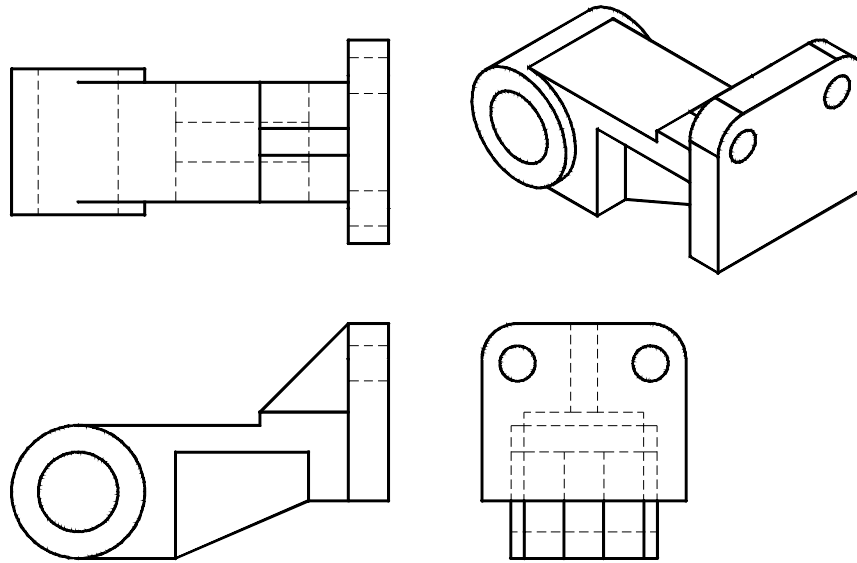


Figure 11-30 Top, front, right-side, and isometric views of the model

- b. Set up the drawing sheet and the background sheet, refer to Figure 11-32.
- c. Set the projection angle method.
- d. Generate the drawing views, refer to Figure 11-34.
- e. Save the draft file.

Starting a New file in the Draft Environment

1. Start Solid Edge and then choose the **Drawing** option from the **Create** area of the welcome screen.
2. After entering the **Draft** environment, choose the **EdgeBar** button to hide it, if it is displayed. This is because you will not use it and also turning its display off increases the work area on the screen.

Setting the Drawing Sheet Options

The drawing sheet in Solid Edge is like a blank sheet of paper on which you draw. You can add as many sheets as needed in the same drawing file. When you work in the **Draft** environment, you have two sheets 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 **File > Sheet Setup** from the menu bar to display the **Sheet Setup** dialog box. In the **Sheet Size** area of the **Size** tab, the **Standard** radio button is selected by default.
2. From the drop-down list, select **A2 Wide (594mm x 420mm)**, if it is not already selected. Note that in the dialog box the units are set in mm.
3. Select 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 will insert a graphic image as an object in the background sheet. Generally, a company logo is inserted using this method. The image has to be inserted in the background sheet, and so, 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 **View > Background Sheet** from the menu bar and then choose **View > Working Sheets** to clear this option. Notice that four sheets are displayed in the bar below the drawing window.
5. Select the **A2-Sheet** tab from the bottom of the drawing window and then zoom to fit it on the screen.
6. Select the top table, as shown in Figure 11-31. All entities in the table turn magenta in color, which indicates that they are selected.

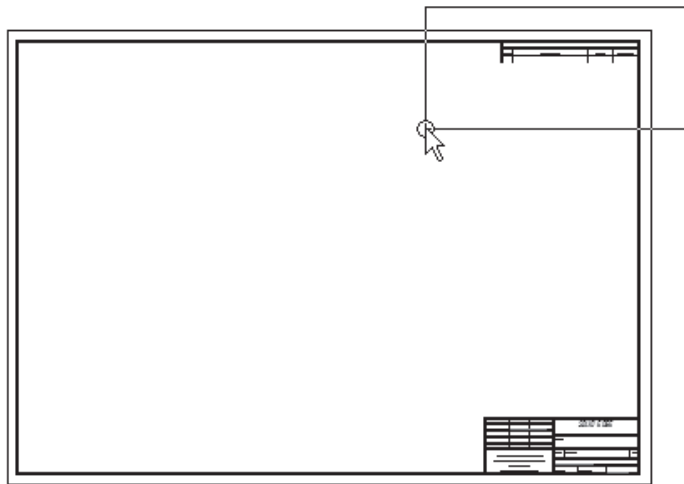


Figure 11-31 Selecting the table

7. Press the **DELETE** key to delete the selected entities.
8. Choose **Insert > Image** from the menu bar to display the **Insert Image** dialog box.
9. Choose the **Browse** button; the **Open a File** dialog box is displayed.

10. Browse and select the image file of the logo you want to use, refer to Figure 11-32.
11. Set the transparency option for the logo, if required.
12. After selecting the file, choose the **OK** button from the **Insert Image** dialog box; the image will be placed on the sheet.
13. Drag the image and place it, as shown in Figure 11-32. The size of the image can be modified using the handles available on the image, when it is selected.

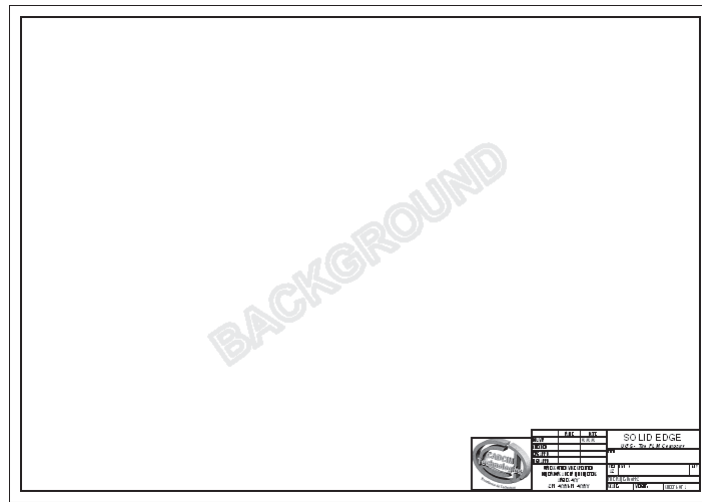


Figure 11-32 Title block with the image

14. After placing the image, choose **Tools > Options** from the menu bar to display the **Options** dialog box.
15. From the **Drawing Standards** tab, select the **Third** radio button to set the current projection to the third angle projection. Choose **OK** to exit the dialog box.
16. Choose **View > Working Sheets** from the menu bar and then choose **View > Background Sheet** to clear this option. Next, the drawing views that you generate will be placed on the worksheet and not on the background sheet.
17. Save the drawing file as **Template.dft** in *C:\Program Files\Solid Edge V18\Program\Template*

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




Note

The unit system in this template file is set to mm.

Generating the Drawing Views

The **Drawing View Orientation** page of the **Drawing View Creation Wizard** dialog box enables you to generate multiple views in a single attempt. All views required in this tutorial will be generated together using this option.

1. Choose the **Drawing View Wizard** button from the **Drawing View** toolbar to display the **Select Model** dialog box. 
2. Select the part created in Exercise 1 of Chapter 7 and choose the **Open** button to display the **Drawing View Creation Wizard** dialog box.
3. Accept the default options and choose the **Next** button from the **Part/Sheet Metal Drawing View Options** page.
4. From the **Drawing View Orientation** page, select **front** and choose the **Next** button.
5. Select the views shown in Figure 11-33 from the **Drawing View Layout** page and then choose the **Finish** button.

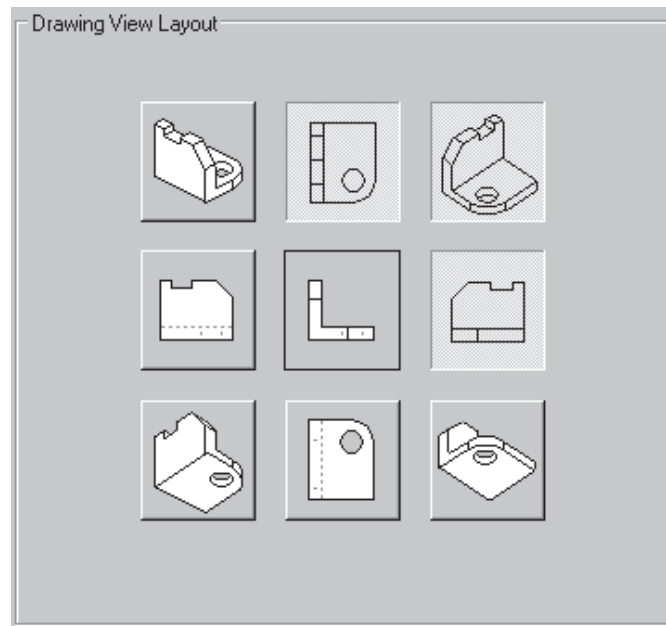


Figure 11-33 The *Drawing View Layout* area of the *Drawing View Creation Wizard* dialog box

6. Place the drawing views on the sheet.

Notice that you need to increase the scale of the views. The scale of any one orthographic view, when modified, changes the scale of the other two orthographic views also.

7. Select the isometric view and right-click to invoke the shortcut menu.
8. Choose the **Properties** option from the shortcut menu to display the **Drawing View Properties** dialog box.
9. In the **General** tab, modify the value of the scale to **1.2**. Choose **OK** to exit the dialog box.
10. Similarly, select any one of the orthographic views and right-click to invoke the shortcut menu. Modify the value of the scale to **1.2**.
11. You can move the views on the sheet by dragging them. Arrange all views, as shown in Figure 11-34.

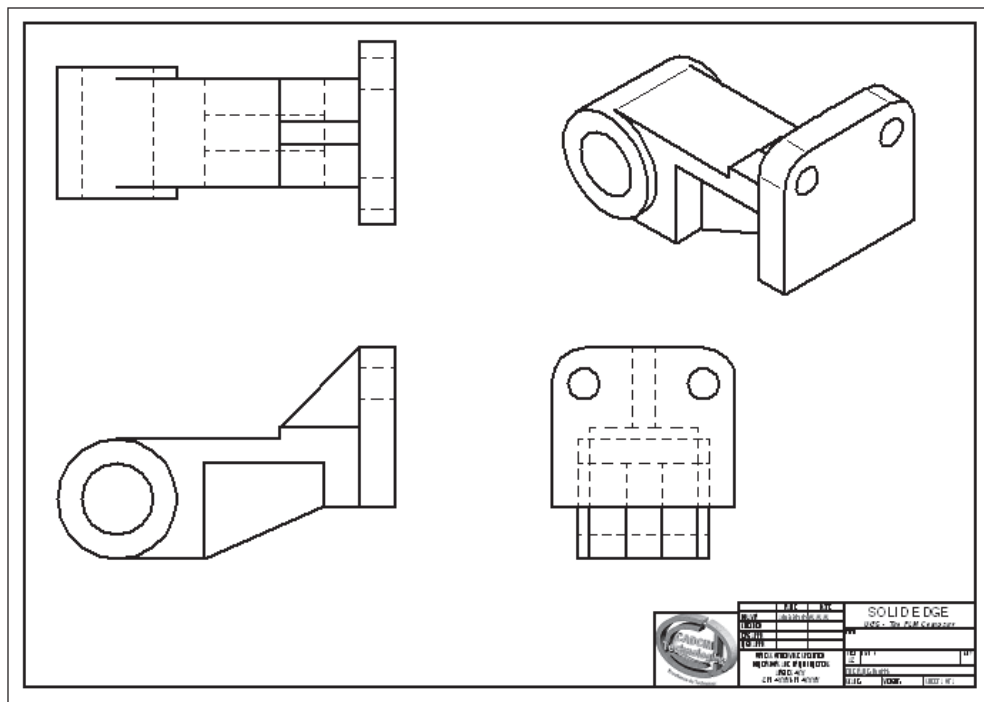


Figure 11-34 Sheet after generating all the drawing views

Saving the File

Remember that you will have to use the **Save As** option to save the file because this file is already saved with the name *Template*.

1. Choose **File > Save As** from the menu bar to display the **Save As** dialog box. Save the file with the name *c11tut1.dft*.
2. Choose **File > 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 that was created in Exercise 1 of Chapter 6. You will also generate the dimensions, as shown in Figure 11-35. Use the template that was created in Tutorial 1.

(Expected time: 30 min)

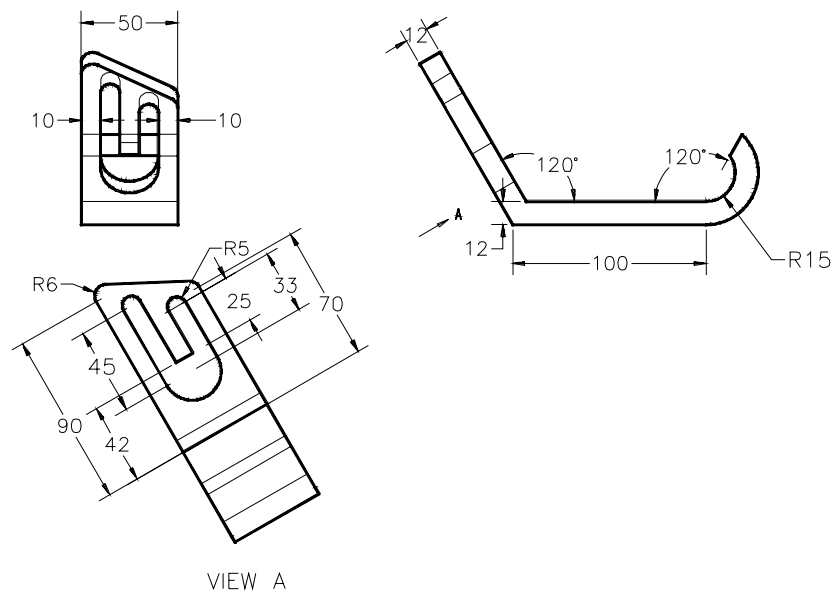


Figure 11-35 Left-side view, auxiliary view, and the front view of the model

The following steps are required to complete this tutorial:

- Start a new draft file.
- Generate the drawing views, refer to Figures 11-37 and 11-39.
- Generate the dimensions.
- Create the remaining dimensions that are not generated, refer to Figure 11-43.
- 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 **Main** 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. Here, you will use the *Template.dft* file as the template.

Generating the Base Drawing Views

As mentioned earlier, the drawing views are generated from their parent part. The following are the steps for generating the drawing views:

1. Choose the **Drawing View Wizard** button from the **Drawing View** toolbar to display the **Select Model** dialog box. 
2. Select the part created in Exercise 1 of Chapter 6 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.

Notice that you need to scale down the views. The scale of any one orthographic view, when modified, changes the scale of the other orthographic views also.

7. Select one of the views and then invoke the shortcut menu. Choose the **Properties** option to display the **Properties** dialog box.
8. Modify the value of the scale to **1.2**.
9. You can move the views on the sheet by dragging them. Arrange the views, as shown in Figure 11-36.

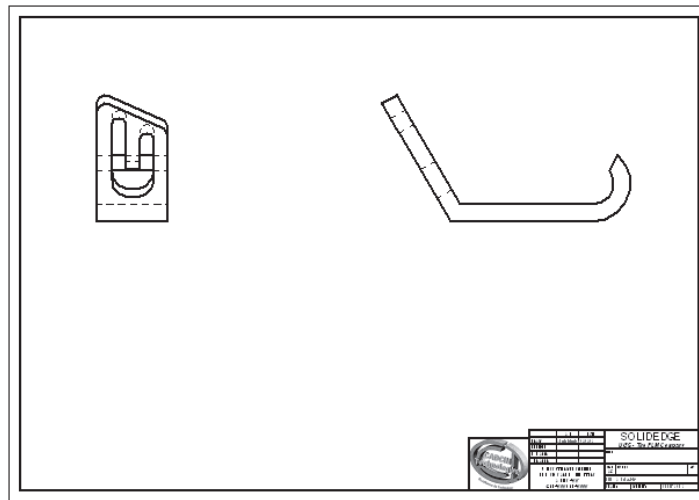



Figure 11-36 Drawing views after reducing the scale

Generating the Auxiliary View

The auxiliary view will be generated 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 View** button from the **Drawing Views** toolbar; 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 view will be projected about this imaginary line. 
2. Select the edge, as shown in Figure 11-37. An imaginary fold line is formed and the auxiliary view is projected normal to this fold line. Move the cursor and to place the view, see Figure 11-38. After you place the view, the arrow pointing in the direction normal to the fold line is displayed, as shown in Figure 11-39.

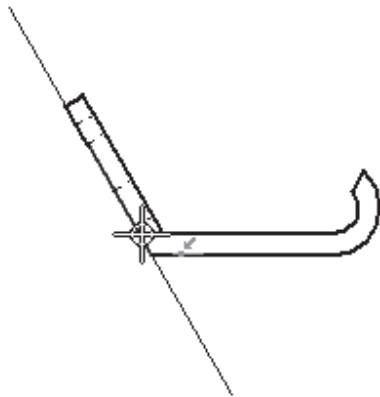


Figure 11-37 Edge to be selected

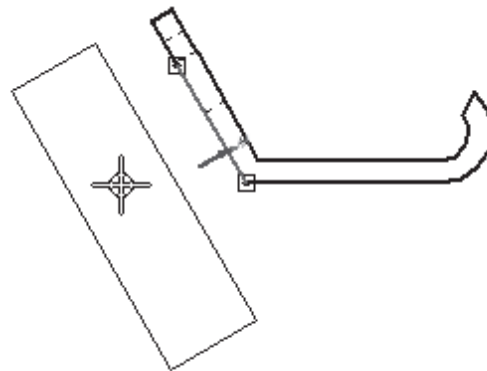


Figure 11-38 Moving the cursor to place the view

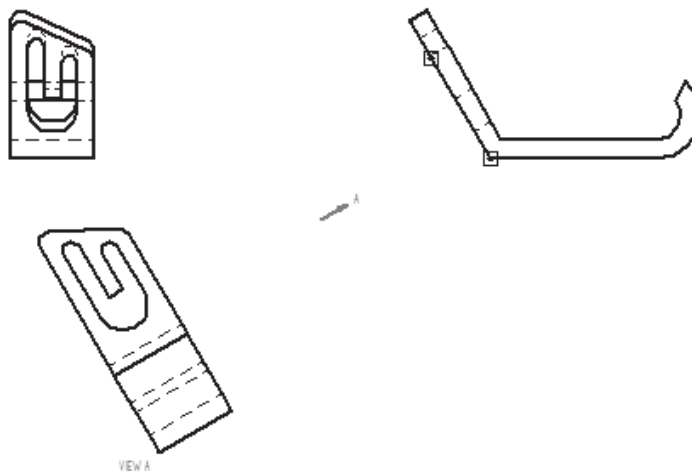





Figure 11-39 Drawing sheet after generating the auxiliary view

Dimensioning the Views

First the dimensions will be generated from the model and then the remaining dimensions will be created.

1. Choose the **Retrieve Dimensions** button from the **Drawing Views** toolbar; you are prompted to select the drawing view. 
2. Select the front view; the dimensions are displayed on it. You will notice that the display of the dimensions is very small. You need to 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 ribbon 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. Now, choose **Format > Style** from the menu bar 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 font size to **3.5** in the **Font size** edit box. Choose **OK** and then choose **Apply** from the **Style** dialog box.
6. Place the missing dimensions, as shown in Figure 11-40, using the dimensioning tools.
7. Choose the **Distance Between** button from the **Drawing Views** toolbar and choose the **2 Points** option from the **Orientation** drop-down list. Dimension the auxiliary view with dimensions 90, 42, and 70, as shown in Figure 11-41. 
8. To dimension the auxiliary view with dimensions 45, 25, and 33, draw an axis passing through the center of the arc, as shown in Figure 11-41.
9. Next, select the center of the arc and then the axis, and place the dimension with a value of 33, see Figure 11-42.
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. 

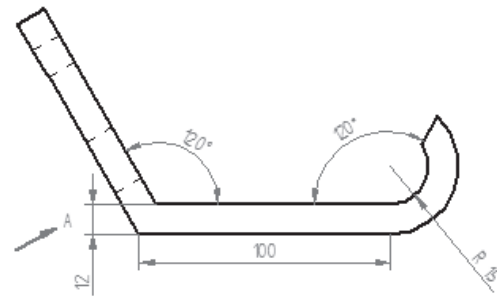


Figure 11-40 Front view after placing the angular dimension

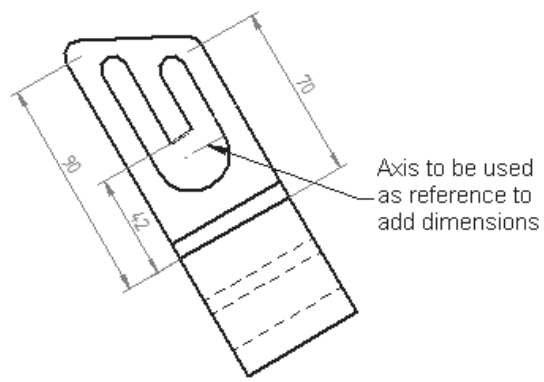


Figure 11-41 Axis drawn to dimension

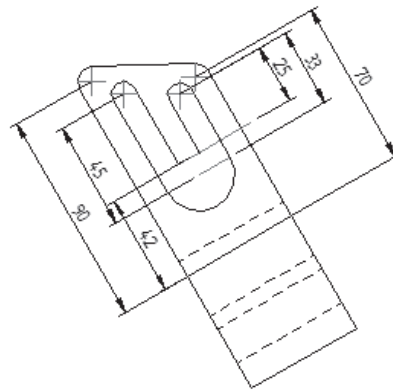


Figure 11-42 Dimensioning the auxiliary view

The drawing sheet, after dimensioning the drawing views, is shown in Figure 11-43.

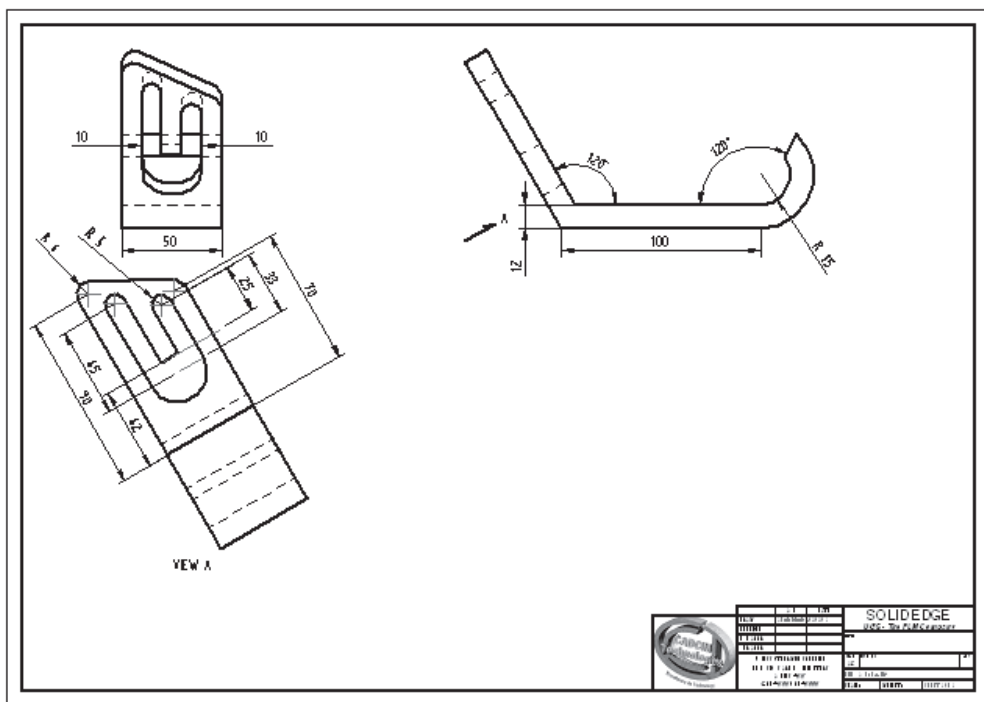


Figure 11-43 Drawing sheet after dimensioning the views

Saving the File

1. Save the file with the name *c11tut2.dft*.
2. Choose **File > Close** to close the file.

Tutorial 3

In this tutorial, you will generate an exploded drawing view of the assembly created in Chapter 9. You will also add the parts list and balloons to the assembly, as shown in Figure 11-44.

(Expected time: 30 min)

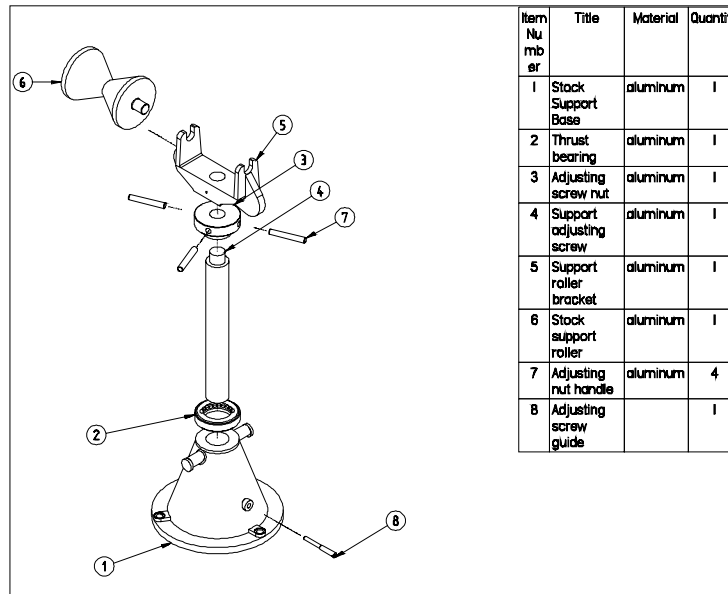


Figure 11-44 Parts list and balloons in exploded drawing view

The following steps are required to complete this tutorial:

- Start a new draft file.
- Generate the exploded drawing view.
- Generate the parts list and balloons.
- Edit balloons.
- Save the drawing file and close the window.

Starting a New File in the Draft Environment

- Choose the **New** button from the **Main** toolbar to display the **New** dialog box.
- 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


- Choose the **Drawing View Wizard** button from the **Drawing View** toolbar to display the **Select Model** dialog box.



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

Generating the Parts List and Balloons

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

1. Choose the **Parts List** button from the **Drawing Views** toolbar; the **Parts List** ribbon bar is displayed and you are prompted to select a view. 
2. Select the exploded drawing view.
3. Choose the **Properties** button from the ribbon bar to display the **List Properties** dialog box.
4. In the **Text** area of the **General** tab, modify the value in the **Size** edit box to **7**.
5. Select the **Columns** tab. In the **Columns used** display box, select the **Document Number** option and choose the **Remove** button to remove it.
6. Select the **Material** option from the **Columns used** display box.
7. Next, in the **Column Format** area, change the value in the **Column width** edit box to **42**. Similarly, change the column width value of **Quantity** to **42**.

By default, the balloons that you place 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 using the **Balloon** tab.

8. Choose the **Balloon** tab to display the options related to balloons.
9. Modify the value in the **Size** edit box to **7**.
10. Next, clear the **Item count** check box to make sure the item counts are not displayed.
11. Choose the **OK** button to exit the dialog box. Choose **Finish** from the ribbon bar.
12. Drag the parts list table to the desired location on the drawing sheet and place it.

The drawing sheet after placing parts list and balloons is shown in Figure 11-45.

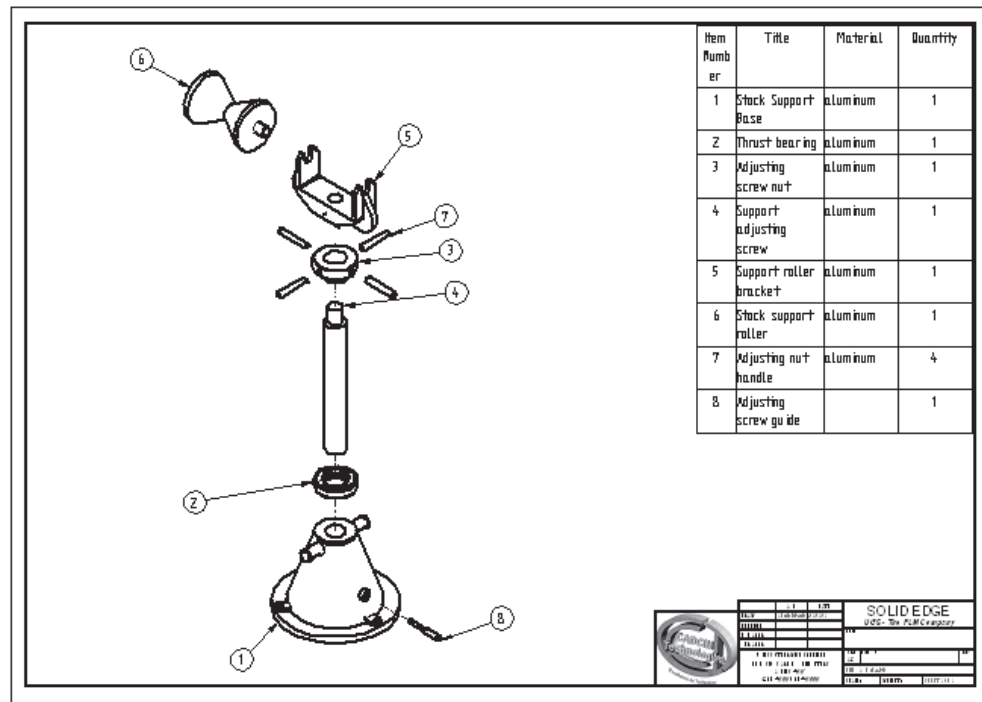


Figure 11-45 Exploded drawing view with the parts list and

Saving the File

1. Choose **File > Save** from the menu bar to display the **Save As** dialog box.
2. Save the file with the name *c11tut3.dft*.
3. Choose **File > Close** to close the file.

Self-Evaluation Test

Answer the following questions and then compare your answers with 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. In Solid Edge, you can also draft a drawing view. (T/F)
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 **Drawing View Wizard** button is used to generate the base view. (T/F)
7. The _____ is the file extension of the files created in the **Draft** environment of Solid Edge.
8. The _____ view is used for parts that have a high length to width ratio.
9. A cutting plane can be edited by _____ or by choosing the _____ button from the ribbon bar.
10. To generate a section drawing view of a part, you need a _____.

Review Questions

Answer the following questions:

1. The dimensions cannot be generated from the part in which of the following views?
(a) Front (b) Right-side
(c) Top (d) None of the above
2. Which of the following buttons in the **Drawing Views** toolbar is used to generate a BOM?
(a) **SmartDimension** (b) **Parts List**
(c) **Draft View** (d) None of the above
3. What is the minimum number of sections required for a blend feature?
(a) one (b) two
(c) three (d) None of the above

4. Which of the following dialog boxes is displayed when you choose the **Drawing View Wizard** button from the **Drawing Views** toolbar?
 - (a) **Properties**
 - (b) **Select**
 - (c) **Drawing View Properties**
 - (d) None of the above
5. Before placing the BOM on the drawing sheet when you choose the **Properties** button from the ribbon 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 9, see Figure 11-46. Generate the BOM and balloons. **(Expected time: 30 min)**

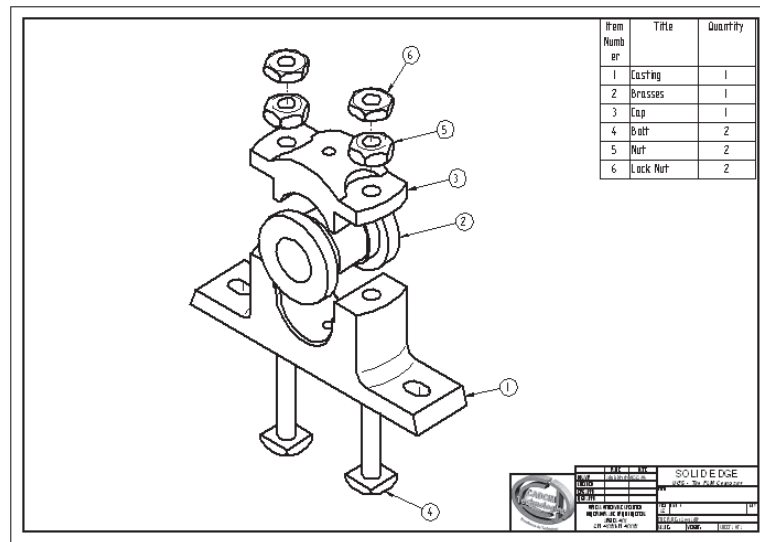


Figure 11-46 Exploded drawing view with the BOM and balloons

Exercise 2

Create the model whose drawing views are shown in Figure 11-47 and then generate the drawing views of the model. Dimension the drawing views, as shown in Figure 11-47.

(Expected time: 45 min)

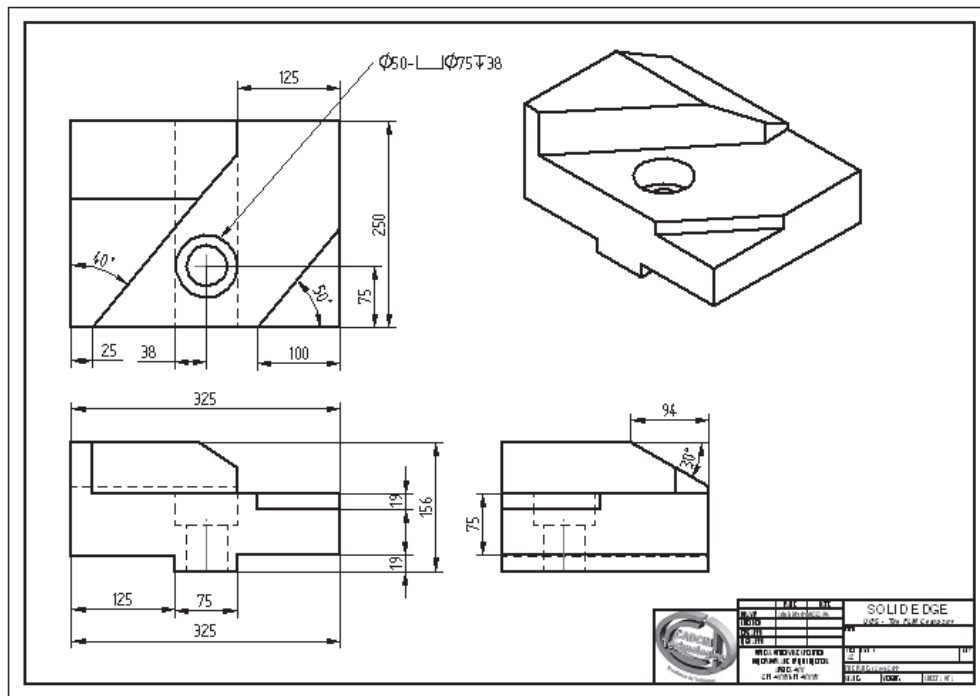


Figure 11-47 Top, front, right-side, and isometric views of the model

Answers to Self-Evaluation Test

1. F, 2. F, 3. T, 4. T, 5. T, 6. T, 7. *.dft*, 8. Broken, 9. double-clicking, **Edit**, 10. Cutting plane