# Chapter 12

# Working With Drawing Views-I

## Learning Objectives

***After completing this chapter you will be able to:***

- *Generate standard three views.*
- *Generate Named Views.*
- *Generate Relative Views.*
- *Generate Predefined Views.*
- *Generate Empty Views.*
- *Generate Projected Views.*
- *Generate Section Views.*
- *Generate Aligned Section Views.*
- *Generate Broken-out Section Views.*
- *Generate Auxiliary Views.*
- *Generate Detail Views.*
- *Generate Crop Views.*
- *Generate Broken Views.*
- *Generate Alternate Position Views.*
- *Generate the View of an Assembly in Exploded State.*
- *Work with Interactive Drafting.*
- *Edit the Drawing Views.*
- *Change the Scale of the Drawing Views.*
- *Delete Drawing Views*
- *Modify the Hatch Pattern of the Section Views.*
- *Apply Hatch Patterns to the Section Views.*

# THE DRAWING MODE

After you have created the solid models of the parts, or an assembly, you will have to generate the drawing views. A 2D drawing is the life line of all the manufacturing systems because at the shop floor or machine floor, the machinist mostly needs the 2D drawing for manufacturing. Therefore, SolidWorks has provided a specialized environment known as the **Drawing** mode. The **Drawing** mode has all the tools that are required to generate the drawing views, modify the drawing views, and add dimensions and annotations to the drawing views. In other words, you can get the final shop floor drawing using this mode of SolidWorks. You can also create the 2D drawings in the **Drawing** mode of SolidWorks using the sketching tools provided in this mode. In other words, there are two types of drafting methods available in SolidWorks: Generative drafting and Interactive drafting. Generative drafting is a technique of generating the drawing views using a solid model or an assembly. Interactive drafting is a technique in which you use the sketching tools to sketch a drawing view in the **Drawing** mode. In this chapter, you will learn about generating the drawing views of parts or assemblies. One of the major advantage of working in SolidWorks is that this software is bidirectionally associative in nature. This property ensures that if the modifications are made in a model in the **Part** mode, the same modification will be reflected in the **Assembly** mode and the **Drawing** mode, and vice versa.

For creating a new document in the **Drawing** mode, invoke the **New SolidWorks Document** dialog box. Choose the **Drawing** template from the **Templates** tab as shown in Figure 12-1 and choose the **OK** button.

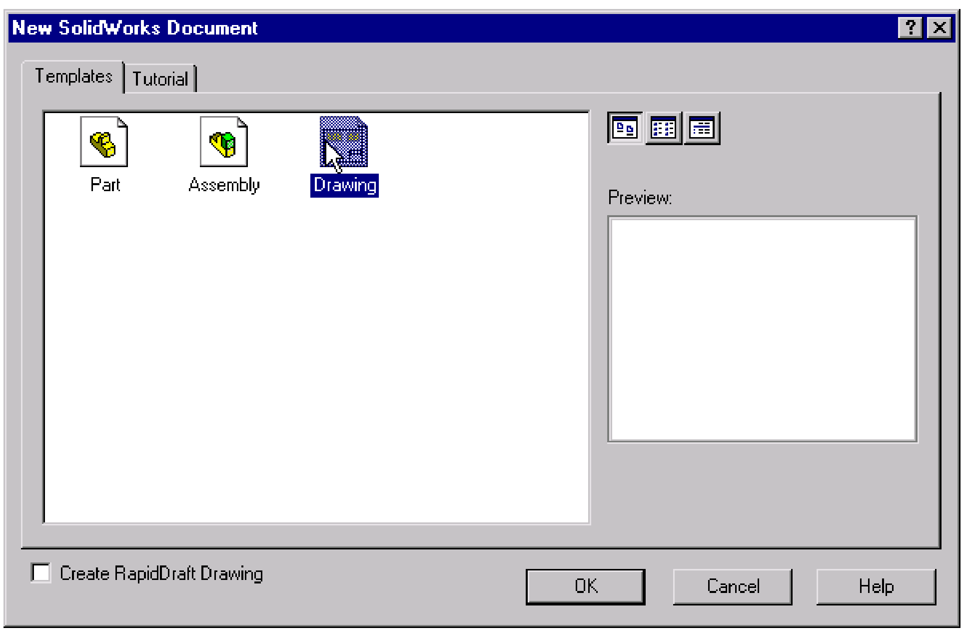



***Figure 12-1** The **New SolidWorks Document** dialog box*

**Tip**. *When you select the **Drawing** template from the **Templates** tab, the **Create RapidDraft Drawing** check box is displayed at the lower right corner of the dialog box. You will learn more about rapid draft drawings later.*

When you choose the **OK** button from the **New SolidWorks Document** dialog box, a new **Drawing** document is invoked. The **Sheet Format To Use** dialog box is also displayed. Figure 12-2 shows the initial screen of the drawing document with the **Sheet Format to Use** dialog box.

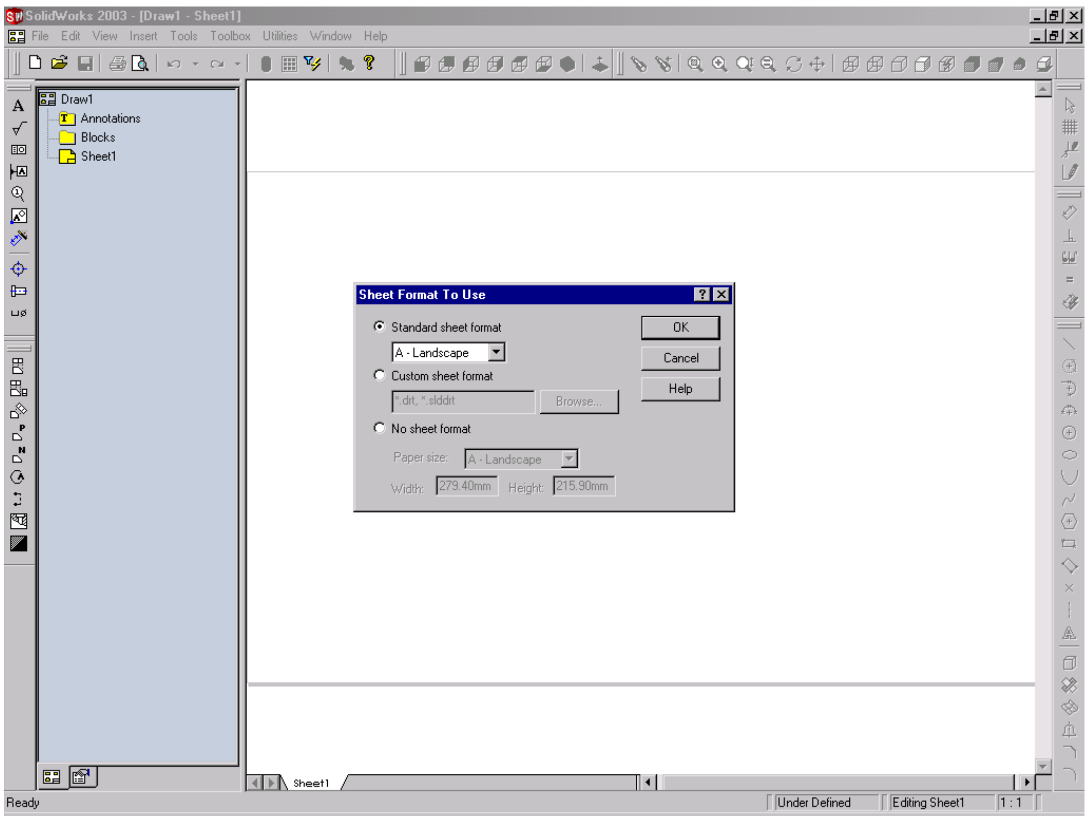



***Figure 12-2*** *The initial screen of the drawing document with the* ***Sheet Format To Use*** *dialog box*

The **Sheet Format To Use** dialog box is used to specify the sheet and the format of sheet to be used. The options available in the this dialog box are discussed next.

## Standard sheet format

The **Standard sheet format** radio button is selected by default. Using this option you can select the predefined standard sheet formats available in SolidWorks. You can select the sheet size from the drop-down list available below the **Standard sheet format** radio button.

## Custom sheet format

The **Custom sheet format** radio button is used to add a user-defined sheet format to the drawing sheet. Select the **Custom sheet format** radio button and choose the **Browse** button to invoke the **Open** dialog box. From the **Open** dialog box, you can select and open a user-defined sheet format. You will learn more about creating a user-defined sheet format in later chapters.

## No sheet format

The **No sheet format** radio button is selected if you want to use an empty sheet, without any margin lines or title block. Select this radio button and select the size of sheet from the drop-down list available below the **No sheet format** radio button and choose the **OK** button.

Figure 12-3 shows a drawing document created using **A4** standard sheet format from the drop-down list available below the **Standard sheet format** radio button.

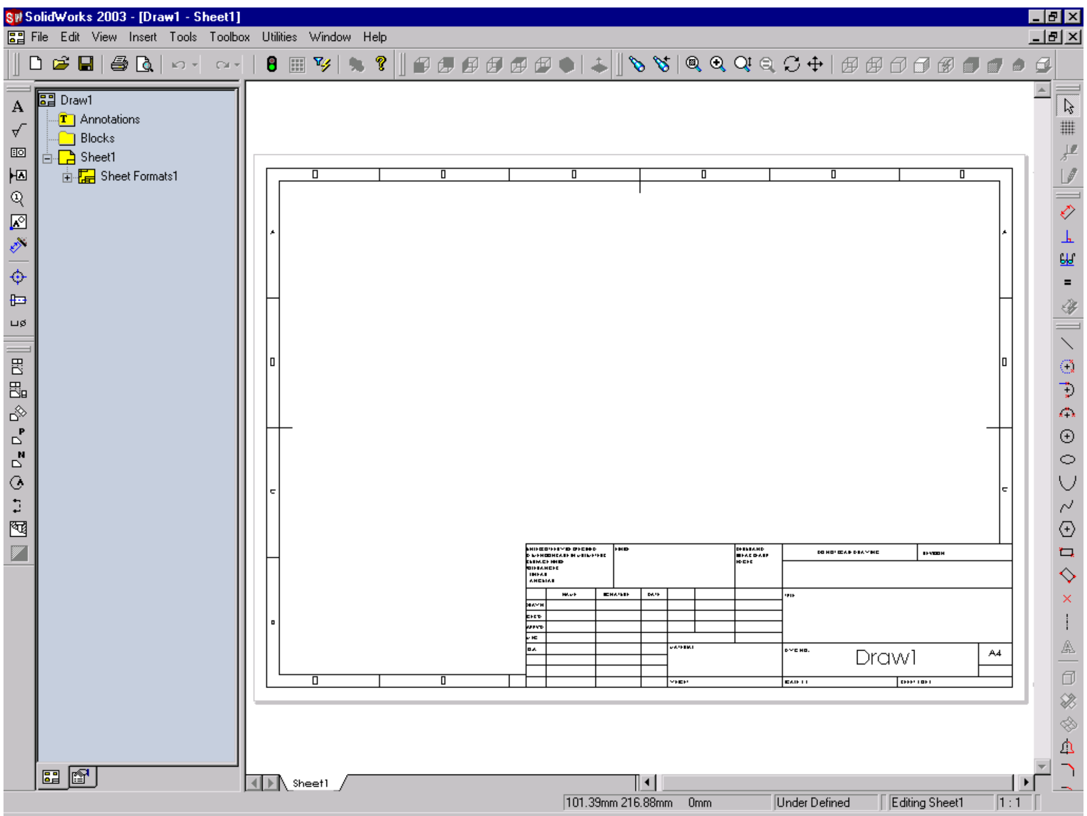
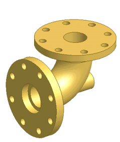



***Figure 12-3** Drawing document with **A4** standard sheet format*



**Tip**. *If you choose the **Cancel** button from the **Sheet Format To Use** dialog box, a blank sheet of **B** size in landscape orientation will be inserted in the drawing document.*

# TYPE OF VIEWS

In SolidWorks, you can generate nine types of views. Generally, you first need to generate a standard view such as the top view or the front view and then use this view to derive the remaining views. After generating a standard view, you can generate or derive the following views from the standard view(s).

## Projected View

The projected view is generated by taking an existing view as the parent view. This view is generated by projecting the lines normal to the parent view. The resultant view will be an orthographic view.

## Section View

A section view is generated by chopping a part of an existing view using a plane and then viewing the parent view from a direction normal to the section plane.

## Aligned Section View

An aligned section view is used to section those features that are created at a certain angle to the main section planes. Align sections straighten these features by revolving them about an axis that is normal to the view plane. Remember that the axis about which the feature is straightened should lie on the cutting planes.

## Auxiliary View

An auxiliary view is generated by projecting the lines normal to a specified edge of an existing view.

## Detail View

A detail view is used to display the details of a portion of an existing view. You can select the portion whose detailing has to be shown in the parent view. The portion that you have selected will be magnified and will be placed as a separate view. You can control the magnification of the detail view.

## Broken View

A broken view is used to display a component by removing a portion of it from between, keeping the ends of the drawing view intact. This type of view is used to display the components whose length to width ratio is very high. This means that either the length is very large as compared to the width or the width is very large as compared to the length. The broken view will break the view along the horizontal or vertical direction such that the drawing view fits the area you require.

## Broken-out Section

A broken-out section view is used to remove a part of the existing view and display the area of the model or the assembly that lies behind the removed portion. This type of view is generated using a closed sketch that is associated with the parent view.

## Crop View

A crop view is used to crop an existing view enclosed in a closed sketch associated to that view. The portion of the view that lies inside the associated sketch is retained and the remaining portion is removed.

## Alternate Position View

The alternate position view is used to create a view in which you can show the maximum and minimum range of motion of the assembly. The main position is displayed in the drawing view in continuous lines and the alternate position of the assembly is shown in the same view in dashed lines (phantom lines).

# GENERATING THE DRAWING VIEWS

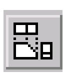
The methods of generating various types of drawing views are discussed next.

## Generating the Standard Drawing Views

The various options of generating the standard views are discussed next.

### Generating the Three Standard Views

| | |
|---|---|
| **Toolbar:** | Drawing > Standard 3 View |
| **Menu:** | Insert > Drawing View > Standard 3 View |



Using the **Standard 3 View** option, you can generate three default orthographic views of the part or the assembly. For creating the three standard views, choose the **Standard 3 View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Standard 3 View** from the menu bar. The **Standard View PropertyManager** will be displayed as shown in Figure 12-4 and the select cursor is replaced by the part selection cursor. The **Message** rollout available in the **Standard View PropertyManager** lists the various options that you can use to select the model whose views are to be generated. The various options of selecting the model whose views are to be generated are discussed next.

#### From File

After invoking the **Standard View PropertyManager**, right-click in the drawing area to invoke the shortcut menu. Choose the **Insert From File** option from the shortcut menu. The **Open** dialog box will be displayed. Browse and select the part or assembly document to generate the drawing views and choose the **Open** button from the **Open** dialog box. The **Tangent Edge Display** dialog box is displayed. Choose **OK** from this. You will learn more about this dialog box later. The three standard orthographic views of the selected models are generated.

#### Selecting the model from the graphics area of another window

To select the model for generating the three drawing views using this option, you first need to open the part document from which you need to generate the views. Choose **Window > Tile Horizontally/Tile Vertically** to tile the part and the drawing documents. If the drawing document is not active, select the title bar of the drawing window once to activate it. Now, choose the **Standard 3 View** button from the **Drawing** toolbar, and

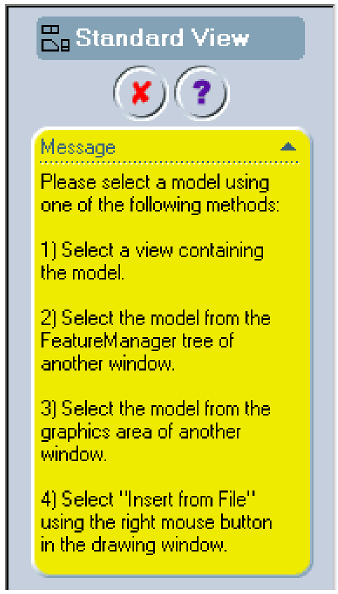



*Figure 12-4 The **Standard View PropertyManager***

move the cursor in the drawing area of the part document and pick a point. Choose **OK** from the **Tangent Edge Display** dialog box that is displayed. The three standard views of the selected model are generated in the drawing document. You can also generate the three standard views of an assembly using the same procedure.

#### Selecting the model from the FeatureManager Design Tree of another window

Open the part document whose drawing views are to be generated and tile the window horizontally or vertically. Now, choose the **Standard 3 View** button from the **Drawing** toolbar, and move the cursor to the **FeatureManager Design Tree** of the part window. Select the name of the part displayed on the top of the **FeatureManager Design Tree** and choose **OK** from the **Tangent Edge Display** dialog box. The three standard orthographic views are generated in the drawing document.

#### Selecting a view containing the model

This option is used only if another view is generated on the drawing view. If one view is available in the drawing view, choose the **Standard 3 View** button from the **Drawing** toolbar and select the view from the drawing area. The three standard orthographic views will be generated in the drawing document.

Figure 12-5 shows the three standard views of a model created using the **Standard 3 View** tool.

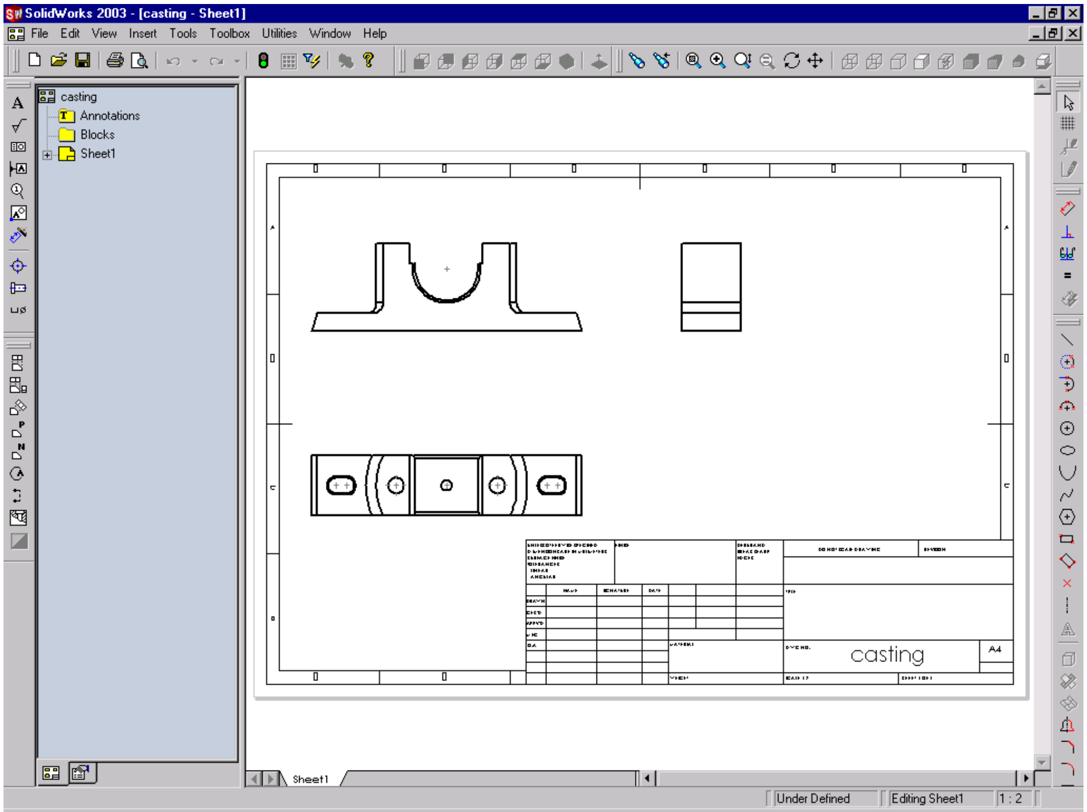
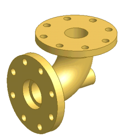
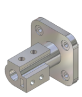



***Figure 12-5*** *Three standard views created using the* ***Standard 3 View*** *tool*



**Tip**. *If the generated view overlaps the title block, you need to move it. To move a view, move the cursor over it. The bounding box of the view is displayed in gray color. At this point, left-click to select the view. Now, move the cursor to the boundary of the selected view; the cursor is replaced by the move cursor. Press and hold down the left mouse button and drag the cursor to move the view. Remember that if you move the front view, all the views are moved.*

*If the bounding box of the view appears in red dashed lines, you need to set the dynamic view activation option from the* ***System Options*** *dialog box. Choose* ***Tools > Options*** *from the menu bar to display the* ***System Options - General*** *dialog box. Select the* ***Drawing*** *option to display the settings related to the* ***Drawing*** *mode. Select the* ***Dynamic drawing view activation*** *check box.*



**Note**

*You will observe that the name of the part document whose drawing views are generated is displayed in the* ***DWG NO.*** *text box of the title block. The size of the sheet is also displayed at the lower right corner of the title block. You will also learn to generate all the information like draw by, check by, and so on as you generate the drawing views later.*

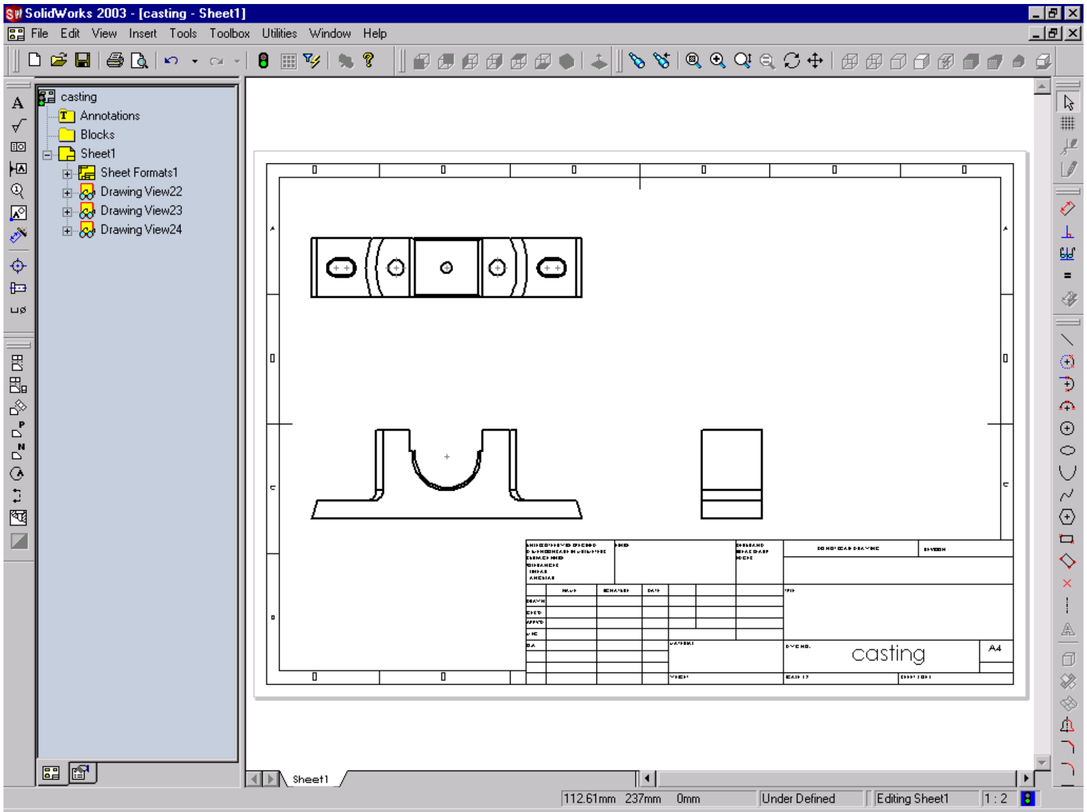
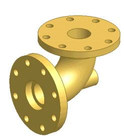


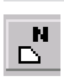

*Figure 12-6 Standard views generated in third angle projection*



**Tip**. *By default, the views in the drawing document of SolidWorks are generated using the **First Angle** projection. If you need to create the views in the **Third Angle** projection, then before generating the views select **Sheet 1** from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Properties** option; the **Sheet Setup** dialog box is displayed. Select the **Third angle** radio button from the **Type of projection** area and choose **OK**. You will learn more about the other options available in the **Sheet Setup** dialog box later. Figure 12-6 shows the standard orthographic views in the **Third Angle** projection.*

## Generating the Standard View Using the Named View Tool

| | |
|---|---|
| **Toolbar:** | Drawing > Named View |
| **Menu:** | Insert > Drawing View > Named View |



The **Named View** tool is used to create a standard view, such as front, right, top, bottom, isometric, and so on. The type of view is defined while placing the view in the drawing document. To create a named view, open the part or assembly document and drawing document and tile the windows horizontally or vertically. Choose the **Named View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Named View** from the menu bar. The **Named View PropertyManager** is invoked, which lists the various methods of selecting the part, and the select cursor is replaced by the part selection cursor. Select the part or the assembly from the document window. Select the **Maximize** button from the drawing document to maximize the drawing document. The cursor is replaced by the place view cursor. The **View Orientation** and the **Custom Scale** rollouts are displayed in the **Named View PropertyManager** as shown in Figure 12-7.

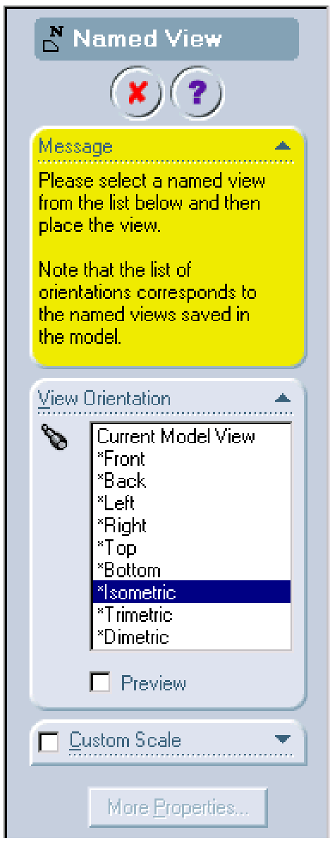
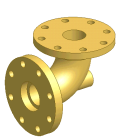



***Figure 12-7** The **Named PropertyManager***

By default, the **Isometric** option is selected in the **View Orientation** area. Therefore, if you place the view in the drawing document, the isometric view will be generated. If you need the front view of the model, select the **Front** option from **View Orientation** area and place the view in the drawing view. The other options available in this **PropertyManager** will be discussed later.



**Tip**. *If you suppress the features of a model whose drawing views are generated, the suppressed features will not be displayed in the drawing views. As you unsuppress the feature, it will be displayed in the drawing views.*

*When you hide or suppress the components of an assembly, the hidden or suppressed components are not displayed in the drawing views.*

**Note**

*If you select the **Preview** button available in the **View Orientation** rollout, the preview of the view to be placed will be displayed in the drawing area.*

*If you choose the **Current Model View** option from the **View Orientation** area, the view of the current orientation of the model in the part document will be placed in the drawing document. When you place the view using the **Current Model View** option, the **SolidWorks** dialog box will be displayed. This dialog box prompts you that this view may need Isometric (True) dimensions instead of standard Projected dimensions. Do you want to switch the view to use Isometric dimensions? It is always recommended to use isometric dimensions in a 3D view. You will learn more about dimensions in the later chapters.*

## Generating the Standard View Using the Relative View Tool

| | |
|---|---|
| **Toolbar:** | Drawing > Relative View (*Customize to Add*) |
| **Menu:** | Insert > Drawing View > Relative To Model |

The **Relative View** tool is used to generate an orthographic view. The orientation of the view is defined by selecting the reference planes or the planar faces of the model. This option is very useful if you need the orientation of the parent view other than the default orientations. To create a relative view open the documents and tile them vertically or horizontally. Choose the **Relative** button from the **Drawing** toolbar or choose **Insert > Drawing View > Relative To Model** from the menu bar. If the **Relative** button is not available in the **Drawing** toolbar, you need to customize the toolbar. The **Relative View PropertyManager** is displayed, which prompts you to select the model.

Select the model or the assembly using the face that you want to use to orient the resultant view. The **Drawing View Orientation** dialog box is displayed as shown in Figure 12-8.

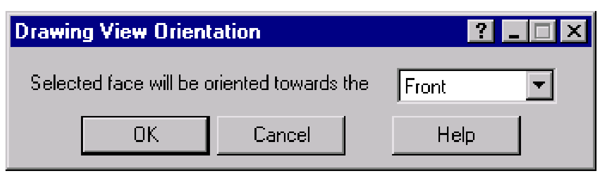



***Figure 12-8** The **Drawing View Orientation** dialog box*

Now from the drop-down list in this dialog box select the option toward which the selected face will be oriented. Choose the **OK** button. Next, select another face and then select the option toward which the second selected face will be oriented. Move the cursor in the drawing document; the cursor will be replaced by the place view cursor. The preview of the view of is also displayed in the drawing document. Place the view at an appropriate place in the drawing document.

Figure 12-9 shows the faces of the model selected to generate a standard view and Figure 12-10 shows the resultant view.

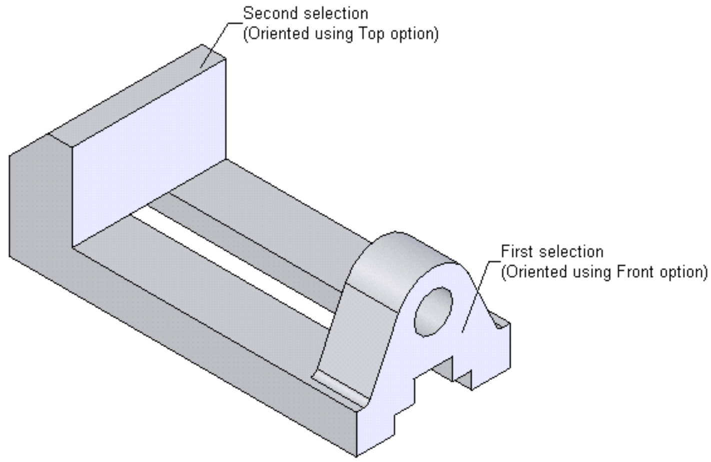



*Figure 12-9* *Faces to be selected*

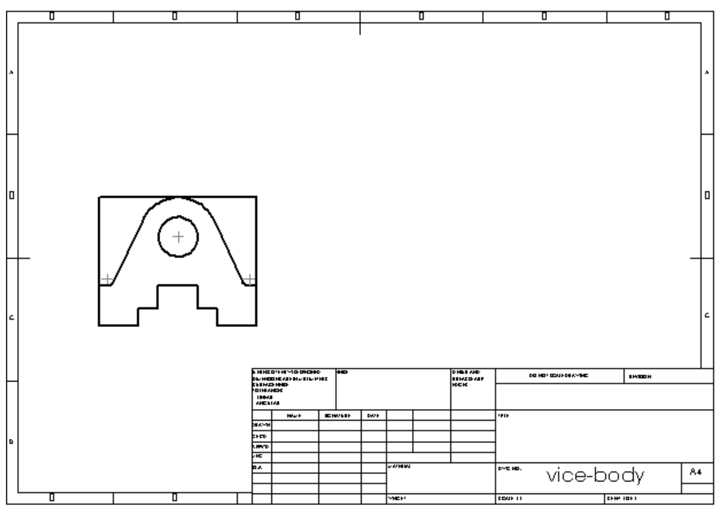
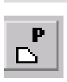



*Figure 12-10* *Resultant view*

## Generating the Standard View Using the Predefined View Tool

**Toolbar:** Drawing > Predefined View
**Menu:** Insert > Drawing View > Predefined



The **Predefined View** tool is used to create empty views with the specified orientation. After creating all the predefined views, populate them by dragging the part from the other window. All the predefined views will be populated. This option is basically used to add the empty views in the drawing document and then save it as a drawing template. Next time when you open a new drawing document using that template, you just need to drag and drop the part from other window; all the predefined views will be populated. You will learn more about creating a drawing template in the later chapters. To create the predefined views, choose the **Predefined View** button from the **Drawing** toolbar or choose **Insert > Drawing > Predefined** from the menu bar. An empty view is attached to the cursor. Specify a point in the drawing document to place the predefined view. The view will be placed in the drawing document and the **Predefined View PropertyManager** is displayed as shown in Figure 12-11.

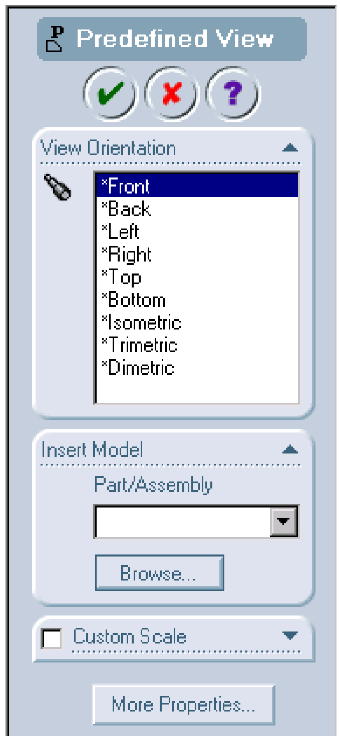



***Figure 12-11*** *The* ***Predefined View PropertyManager***

Select the view orientation from the **View Orientation** area of the **View Orientation** rollout and choose the **OK** button from the **Predefined View PropertyManager**. To add the next predefined view, you need to define the alignment option while placing the view. To create additional predefined view, invoke the **Predefined View** tool and place the view in the drawing sheet. The bounding box of the view is displayed. Right-click inside the bounding box and choose **Alignment > Align Horizontal by Center/Align Vertical by Center**. Now select the previous predefined view to align the corresponding view. Similarly, add other predefined views using this tool.

After creating all the predefined views open a part or an assembly document and tile the documents windows horizontally or vertically. Now, drag the component from the part document and drop in the drawing document; all the predefined views will be populated.

Figure 12-12 shows the selected predefined views with the orientation in which the views are created. Figure 12-13 shows the drawing document after drooping the part in the drawing document. Note that in these figures, the views are not aligned to each other.

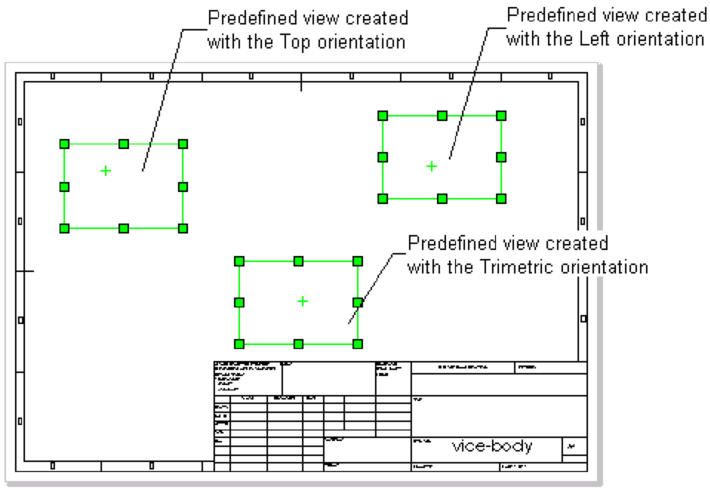



***Figure 12-12*** *Different predefined views*

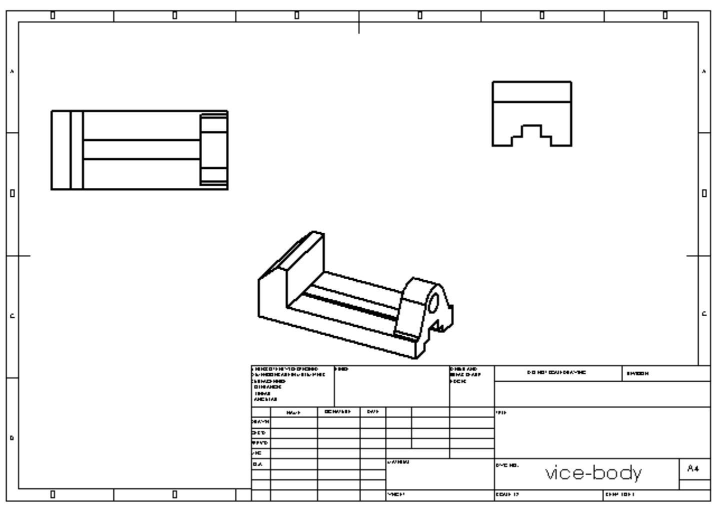
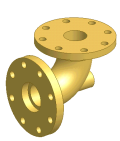
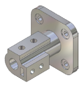



***Figure 12-13*** *Views created after populating the predefined views*



**Tip**. *You can add the views of different parts and assemblies in each predefined view. In this way, you can have the dimensions of different parts and assemblies in a single drawing document. For adding different parts in each predefined view, select the predefined view and choose the* ***Browse*** *button from the* ***Insert Model*** *rollout of the* ***Predefined View PropertyManager****. The* ***Open*** *dialog box is displayed and you can select the part or the assembly to be inserted in the selected predefined view.*



**Note**

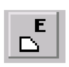
*When you generate the views in the **Drawing** mode of SolidWorks, the views are scaled automatically depending upon the size of the sheet.*

*In case of predefined view, if the drawing contains more than one predefined view, the views will be scaled automatically.*

*If only one predefined view is placed in the drawing document, the view will be scaled with respect to the **Custom Scale** value if specified. Otherwise, the view will be scaled with the default scale factor of the drawing sheet. You can change the view scale using the **Sheet Setup** dialog box. You will learn more about scaling the views in later chapters.*

### Empty View

| | |
|---|---|
| **Toolbar:** | Drawing > Empty View (*Customize to Add*) |
| **Menu:** | Insert > Drawing View > Empty |



The **Empty View** tool is used to create an empty view. The empty views are used to create the sketches in the drawing document. This option is used in interactive drafting. Interactive drafting is discussed later in this chapter. To create an empty view, choose the **Empty View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Empty** option from the menubar. If the **Empty View** button is not available in the **Drawing** toolbar, then you need to customize the toolbar. An empty view is attached to the cursor. Select a point at the desired location in the drawing document to place an empty view.

## Generating the Derived Views

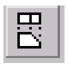
All the views that are generated from a view already placed in the drawing document are known as derived views. The various types of derived views that are generated from the standard views are

1. Projected view
2. Section view
3. Aligned Section view
4. Broken-out Section view
5. Auxiliary view
6. Detail view
7. Crop view
8. Broken view
9. Alternate Position view

The methods of generating the various derived drawing views are discussed next.

## Generating Projected Views

| | |
|---|---|
| **Toolbar:** | Drawing > Projected View |
| **Menu:** | Insert > Drawing View > Projected |



As mentioned earlier, the projected views are generated by projecting the normal lines from an existing view. To generate a projected view, choose the **Projected View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Projected** from the menu bar. The **Projected View PropertyManager** is displayed and it prompts you to select a drawing view from which you need to project the normal lines. The select cursor is replaced by the view cursor. Select the parent view and move the cursor vertically to generate a top view or a bottom view or move the cursor toward the left or right to create a right view or left view. Specify a point on the drawing sheet to place the view. For creating more than one projected view, choose the **Keep Visible** button to pin the **Projected View PropertyManager**. Figure 12-14 shows the front view generated from the top view.

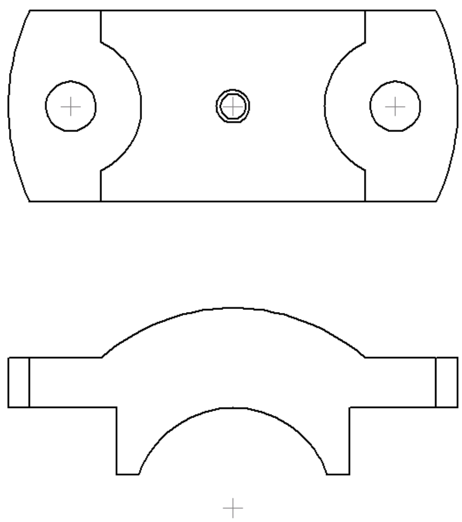
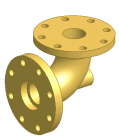


***Figure 12-14*** *Front view generated from the top view*



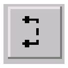
**Tip**. *When you generate a projected drawing view, the drawing view is aligned to the parent view. To place the projected view that is not in alignment with the parent view, press and hold down the CTRL key before placing the view. Now move the cursor to the desired location and place the view.*

*All the standard views and derived views that include projected views, section view, detailed view, and so on are linked to their parent view by a Parent-Child relationship. If you select the child view, the boundaries of the parent view will be displayed in yellow.*

*Select the child view and invoke the shortcut menu and choose the* ***Jump to Parent View*** *option from the shortcut menu. The parent view will be selected automatically.*

## Generating Section Views

| | |
|---|---|
| **Toolbar:** | Drawing > Section View |
| **Menu:** | Insert > Drawing View > Section |



As mentioned earlier, section views are generated by chopping a portion of an existing view using a cutting plane and then viewing the parent view from the direction normal to the cutting plane. To create a section view, you first need to activate the view in which you need to create the section line or cutting plane. To invoke the drawing view, move the cursor to the drawing view. The view symbol will be displayed below the cursor and the bounding box of the view is displayed. Left-click to activate the drawing view. Now, choose the **Section View** button from the **Drawing** toolbar or choose **Insert** > **Drawing** > **Section** from the menu bar. The **Section View PropertyManager** is displayed and it prompts you to sketch a line to continue view creation. You can zoom using the **Zoom** tool to increase the display area of the activated view. Draw a line that will define the section plane. As soon as you specify the endpoint of the section line the **Section View PropertyManager** is displayed. The section view is also displayed in the drawing area and the name of the section view is displayed on the section line. Move the cursor and specify a point on the  drawing sheet to place the section view. The name and the scale factor of the drawing view is displayed below the section view. The **Section View PropertyManager** is still available in the drawing document as shown in Figure 12-15.

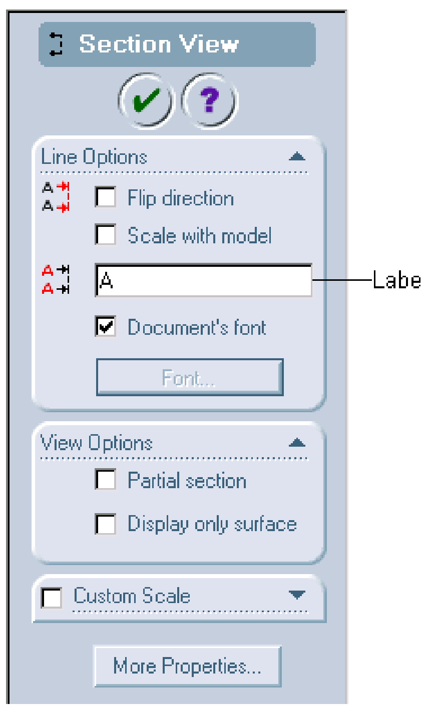



*Figure 12-15  The* ***Section View PropertyManager***

You can use the **Flip direction** check box to flip the direction of the section view. The view will be automatically modified in the drawing sheet. The **Scale with model** check box is used to scale the drawing view if the model is scaled in the part document. You will learn more about scaling the model in the later chapters. Choose **OK** from the **Section View PropertyManager**. Figure 12-16 shows the top view and the section view of a model.

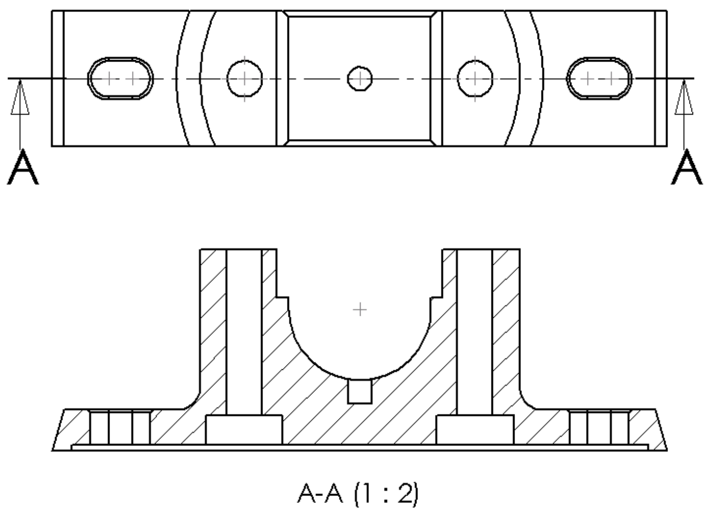



***Figure 12-16*** *The section view*

To create a half section view, make the section lines extended beyond the parent view, see Figure 12-17.

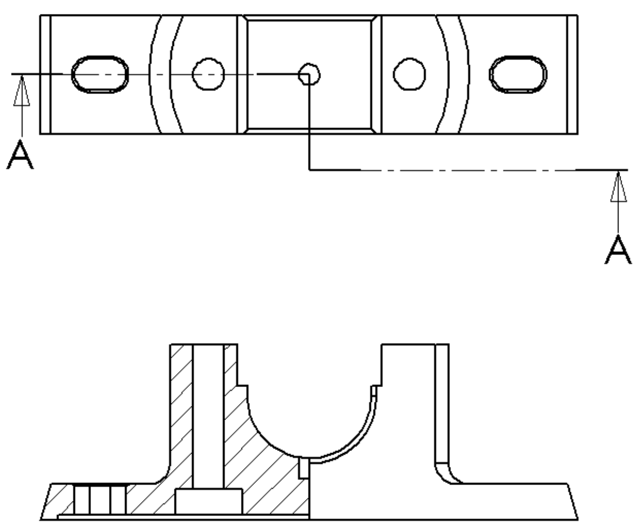
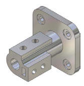
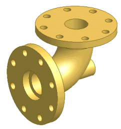



***Figure 12-17*** *The half section view*



**Note**

*You will observe that by default the spacing of the hatch is not what is required. Therefore, you may need to increase the spacing of the hatch pattern. You will learn more about editing the hatch pattern later in this chapters.*



**Tip**. *When you create a section view and move the cursor to place the section view, you will observe that the view is aligned to direction of arrows on the section line. If you need to remove this alignment to place the section view, press and hold down the CTRL key and move the view to the desired location. Select a point in the drawing sheet to place the view.*

There are some other options available in the **Section View PropertyManager** using which you can create partial section view and the surface section view. The options are discussed next

### Creating the Partial Section View

If the section line does not cut through the model, the **SolidWorks** information box will be displayed. This dialog box prompts you that the section line does not completely cut through the bounding box of the model in this view. Do you want this to be a partial section cut? For creating the partial section view, choose the **Yes** button from this dialog box. If you choose the **No** button from this dialog box, then the complete section view will be created. Figure 12-18 shows a partial section view generated from the top view.

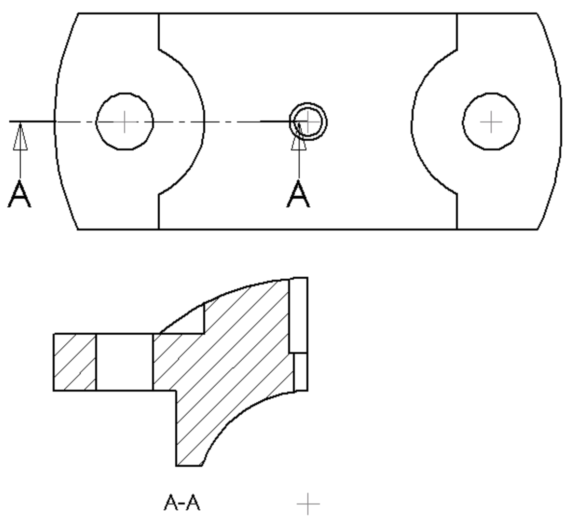



***Figure 12-18*** *A partial section view*

### Creating the Surface Section View

A surface section view is the one in which only the sectioned surface is displayed in the section view. All the other edges or faces are not displayed in the surface section view. For creating a surface section view first you need to create the section view and then choose the **Display only surface** check box from the **Section View PropertyManager**. Choose the **OK** button from the **PropertyManager**. You can also replace the section view by a surface section view by selecting the view to invoke the **Section View PropertyManager**. Select the **Display only surface** check box from the **Section View PropertyManager**. Figure 12-19 shows a surface section view.

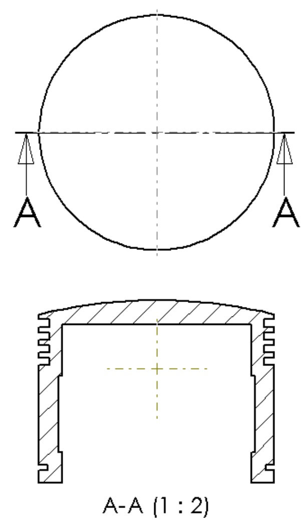



***Figure 12-19*** *A surface section view*

## Generating the Section View of an Assembly

According to the drawing standards when you create the section view of an assembly, some components such as fasteners, shafts, keys, and so on should not be sectioned. Therefore, when you create the section view of an assembly, the **Section View** dialog box is displayed as shown in Figure 12-20. Select the components that should not be sectioned from the parent view. You can also select the component by invoking the **FeatureManager Design Tree** flyout and expanding the parent drawing view. Then expand the assembly to display all the components of the assembly. The name of the selected component is displayed in the **Excluded components** display area. The **Auto hatching** check box is used to automatically define the hatch patterns. You can even change the hatch pattern. Applying the hatch pattern and changing the hatch pattern is discussed later.

If you have more than one instance of the component in the assembly, and you need to exclude all the instances of the components from the section view, select the component from the drawing sheet and select the name of the component from the **Exclude components** display area. Click two times the **Don't cut all instances** check box from the **Section Scope** tab. All the instances of the selected component will be excluded from the section view, see Figure 12-21.

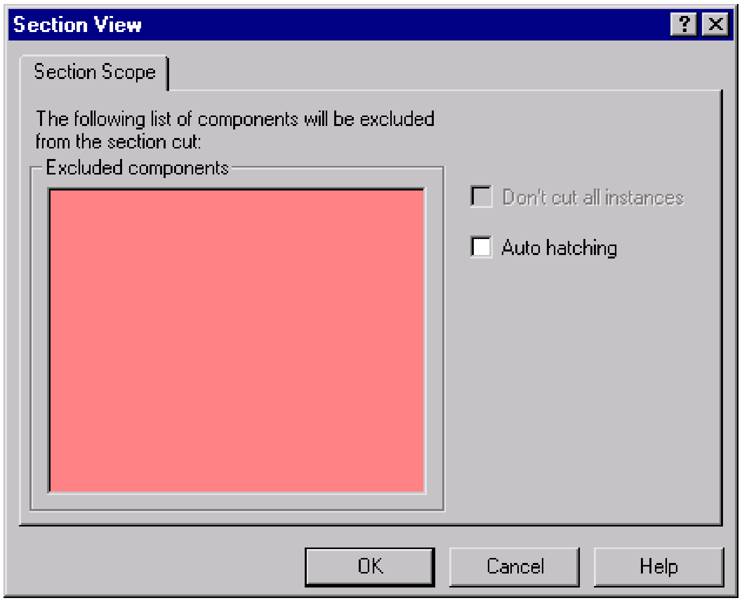



***Figure 12-20*** *The* ***Section View*** *dialog box*

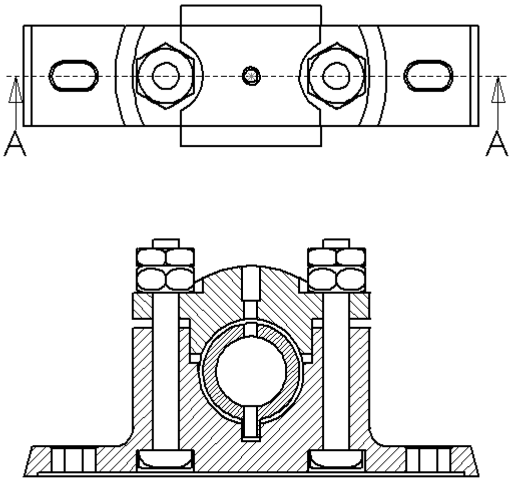
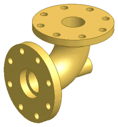


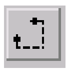

***Figure 12-21*** *Section view of an assembly with some of the components excluded from the section scope*



**Tip**. *You can add or remove the components that are sectioned by right-clicking the drawing view and choosing* ***Properties*** *from the shortcut menu. Now, choose the* ***Section Scope*** *tab and add or remove the components.*

## Generating Aligned Section Views

| | |
|---|---|
| **Toolbar:** | Drawing > Aligned View (*Customize to Add*) |
| **Menu:** | Insert > Drawing View > Aligned Section |



This tool is used to generate a section view of the component in which at least one of the feature is at an angle. In the aligned section view, the section portion revolves about an axis normal to the viewing plane such that it is straightened. For example, refer to Figure 12-22. This figure explains the concept of an aligned section view of a model. Notice that the inclined feature that is sectioned in this view is straightened. As a result, the section view is longer than the parent view. To create the aligned section view, activate the view. Choose the **Aligned Section** button from the **Drawing** toolbar or choose **Insert > Drawing View > Aligned Section** from the menu bar. Draw the sketch that defines the section line. The aligned section view will be attached to the cursor; place the view at an appropriate location in the drawing sheet. Note that the resultant view will be projected normal to the line that is drawn in the end in the section sketch. Therefore, to get the aligned section view similar to that shown in Figure 12-22, the inclined line in the section sketch should be drawn first than the vertical line. Figure 12-23a shows the aligned section view in which the vertical line in the section sketch is drawn first. This is the reason the section view is projected normal to the inclined line that is drawn last. On the other hand, Figure 12-23b shows the views in which the inclined line is drawn first.

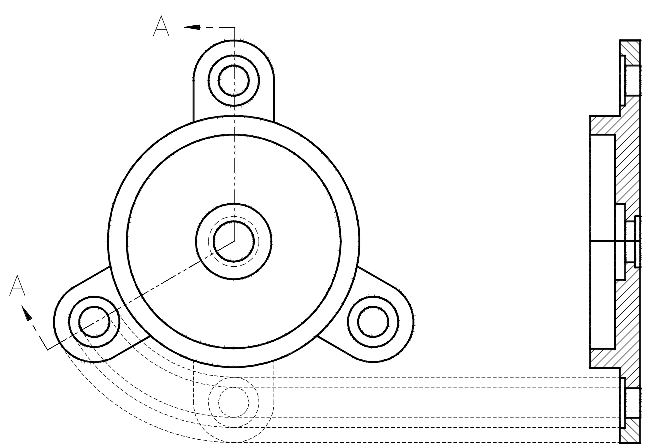



***Figure 12-22*** *Aligned section view*

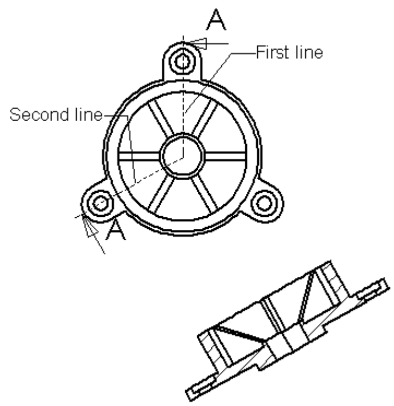



*Figure 12-23a Aligned section view*

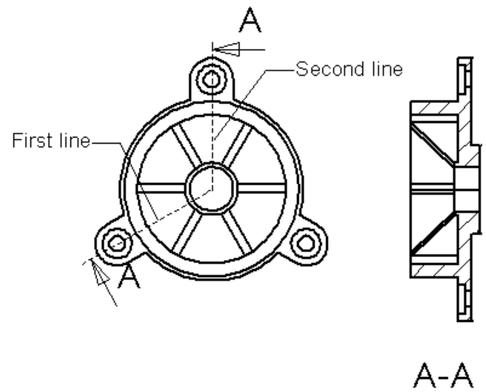
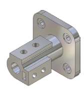



*Figure 12-23b Aligned section view*



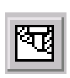
**Note**

*You can also create a sketch associated to a view. This sketch can be selected as the section plane for generating the section view. To create an associated sketch, active the view and draw the sketch that defines the section plane using the **Line** tool.*

*If you create a sketch to define the section plane for the aligned section view before invoking the **Aligned Section View** tool, the view will be projected normal to the line that you select last. However, if you select the sketch to define section plane by dragging a window around it, the view will be projected normal to the line that was drawn last.*

## Generating Broken-out Section Views

| | |
|---|---|
| **Toolbar:** | Drawing > Broken-out Section |
| **Menu:** | Insert > Drawing View > Broken-out Section |



This tool is used to create a broken-out section view. A broken-out section view is used to remove a part of the existing view and display the area of the model or the assembly behind the removed portion. This type of view is generated using a closed sketch that is associated with the parent view. To create a broken-out section view, activate the view on which you need to create the broken out section view. Choose the **Broken-out Section** button from the **Drawing** toolbar or choose **Insert** > **Drawing View** > **Broken-out Section** from the menu bar. The **Broken-out Section PropertyManager** is displayed and it prompts you to create the closed spline to continue section creation. The cursor will be replaced by the spline cursor. Create a closed sketch using the spline cursor. If you do not want a spline profile, select a closed profile before choosing the **Broken-out Section** button. Figure 12-24 shows an associated sketch created for creating a broken-out section view.

When you create a closed sketch, some options are displayed in the **Broken-out Section PropertyManager** as shown in Figure 12-25 and it prompts you to specify the depth of the broken-out section. Choose the **Preview** check box to preview the broken-out section view.

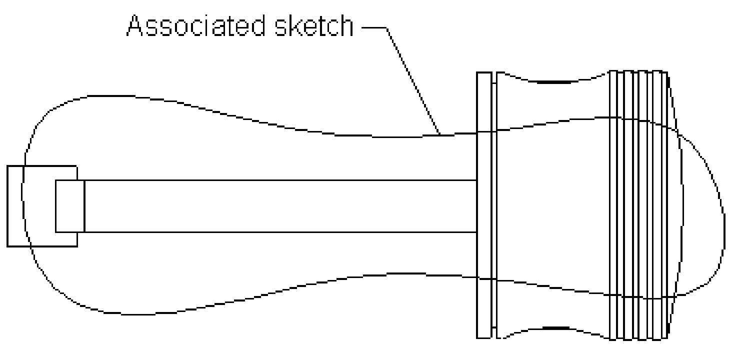



*Figure 12-24 Sketch for creating the broken-out section view*

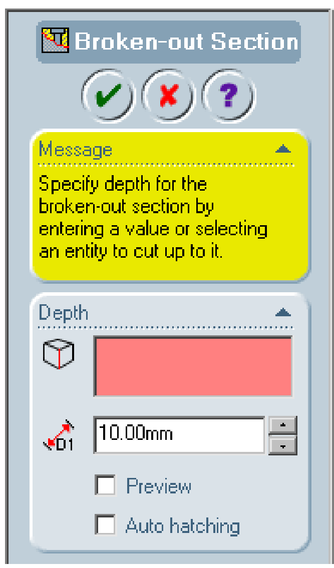



*Figure 12-25 The **Broken-out Section PropertyManager***

The **Auto hatching** check box is used to define the hatch pattern automatically to the section drawing view of the assembly. If you are creating the broken-out section view of a part, then the **Auto hatching** check is not available in the **Broken-out Section PropertyManager**. Figure 12-26 shows the preview of the broken-out section view.

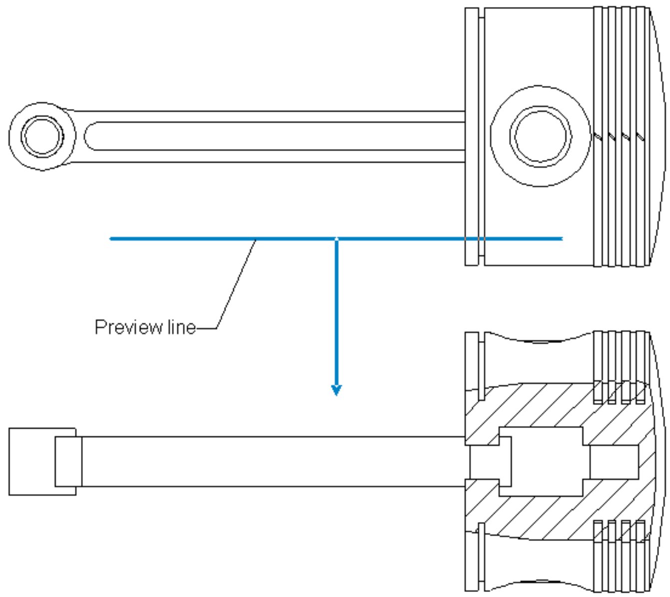



***Figure 12-26*** *Preview of the broken-out section view*

Set the value of the depth of the broken-out section in the **Depth** spinner. The preview of the section will be modified dynamically in the drawing view. After setting the value of the depth of the broken-out section, choose the **OK** button from the **Broken-out Section PropertyManager**. Figure 12-27 shows a broken-out section view.

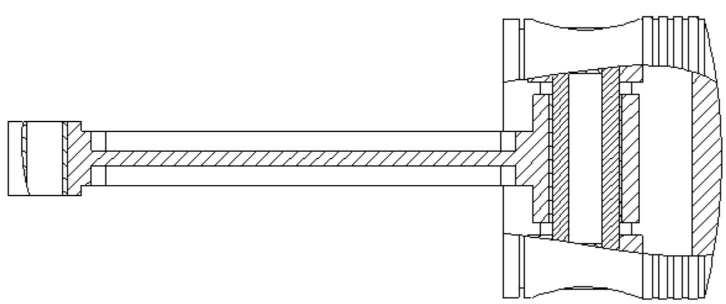
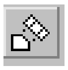


***Figure 12-27*** *Broken-out section view*

## Generating Auxiliary Views

| | |
|---|---|
| **Toolbar:** | Drawing > Auxiliary View |
| **Menu:** | Insert > Drawing View > Auxiliary |



This tool is used to generate an auxiliary view. An auxiliary view is a drawing view that is generated by projecting the lines normal to a specified edge of an existing view. To create an auxiliary view, choose the **Auxiliary View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Auxiliary** from the menu bar. The **Auxiliary View PropertyManager** is displayed and it prompts you to select a reference edge to continue. Select the edge that will be the reference to generate the auxiliary view. A view will be attached to the cursor and some options are displayed in the **Auxiliary View PropertyManager** as shown in Figure 12-28. Now, the **Auxiliary PropertyManager** prompts you to select a location of new view. Select a point on the drawing sheet to place the auxiliary view.

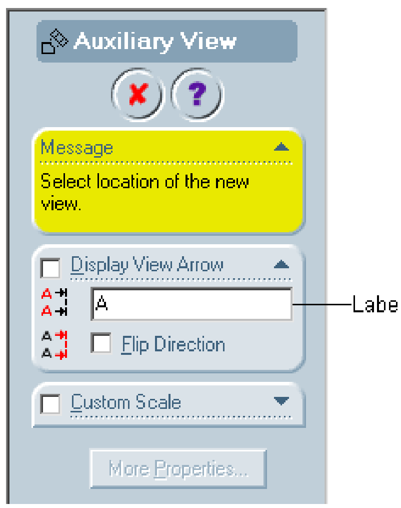
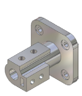



*Figure 12-28 The **Auxiliary View PropertyManager***

The **Display View Arrow** check box available in the **Display View Arrow** rollout is used to display the arrow of the viewing plane in the drawing views. The name of the auxiliary view is specified in the **Label** edit box. Using the **Flip Direction** check box you can flip the viewing direction for creating the auxiliary view. Figure 12-29 shows the reference edge to be selected to create the auxiliary view. Figure 12-30 shows the auxiliary view created with the default viewing direction. Figure 12-31 shows the auxiliary view created with the **Flip Direction** check box selected.



**Note**

*You can also create a sketch using the sketch tools available in the **Sketch Tools** toolbar to select as a reference edge for generating the auxiliary view.*

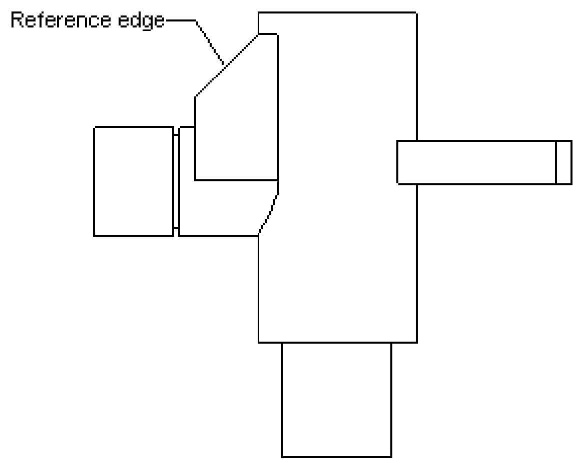



***Figure 12-29*** *Reference edge to be selected to create the auxiliary view*

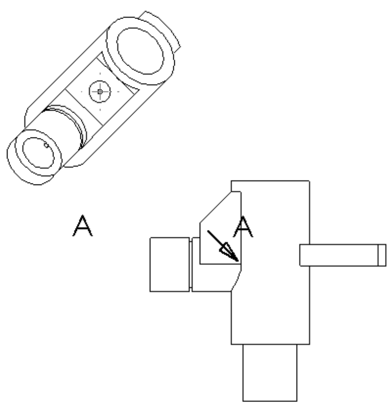



***Figure 12-30*** *Auxiliary view created with the* ***Flip Direction*** *check box cleared*

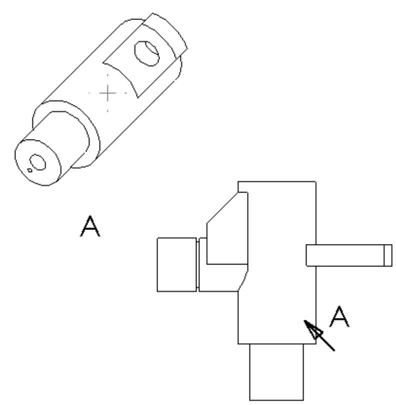
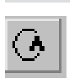


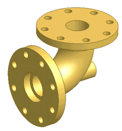

***Figure 12-31*** *Auxiliary view created with the* ***Flip Direction*** *check box selected*

## Generating Detail Views

| | |
|---|---|
| **Toolbar:** | Drawing > Detail View |
| **Menu:** | Insert > Drawing View > Detail |



This tool is used to generate the detail view. A detail view is used to display the details of a portion of an existing view. You can select the portion whose detailing has to be shown in the parent view. The portion that you select will be magnified and placed as a separate view. You can control the magnification of the detail view. To create a detail view, you first need to activate the view from which you will generate the detail view. Choose the **Detail View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Detail**



> **Tip**. *You can also create closed profile other than circle for creating the detail view. For this you need to create the closed profile in the current active view before invoking the **Detail View** cursor. After creating the closed profile, select the profile and invoke the **Detail View** tool.*

from the menu bar. The **Detail View PropertyManager** is displayed and it prompts you to sketch a circle to continue view creation. The cursor is replaced by a circle cursor. Create the circle on the portion of the view that is to be displayed in the detail view.

As soon as you draw the circle, the detail view is attached to the cursor and some options are displayed in the **Detail View PropertyManager** as shown in Figure 12-32. You are also prompted to select a location for the new view. Specify a point on the drawing sheet to place the view. The options available in the **Detail View PropertyManager** are discussed next.

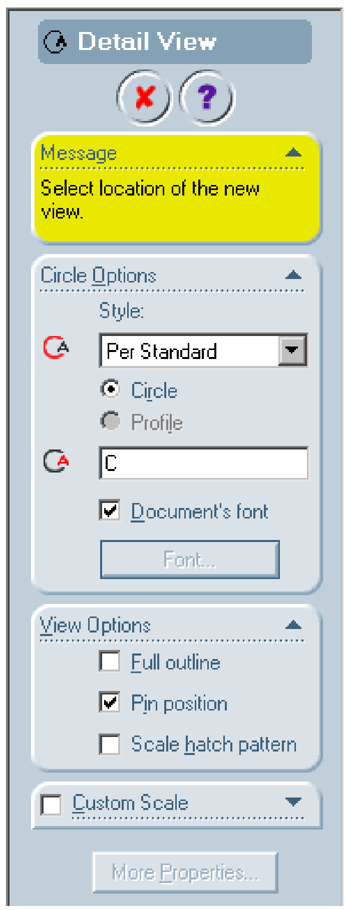



***Figure 12-32** The **Detail View PropertyManager***

### Circle Options

The **Circle Options** rollout is used to define the options to display the circle of the detail view. Using the options available in the rollout, you can also apply the leader to the detail view. The options available in this rollout are discussed next.

#### Style

The **Style** area has the **Style** drop-down list to specify the style of the closed profile. By default, the **Circle** radio button is selected below the **Style** drop-down list. Therefore, the portion of the parent view that is shown in the detail view is highlighted in circle. If you have created a closed profile for defining the portion to be shown in the detail view, select the **Profile** radio button. The options available in the **Style** drop-down list are discussed next.

**Per Standard.** The **Per Standard** option is used to create the detail view as per default standards.

**Broken Circle.** The **Broken Circle** option is used to display the area of the parent view to be displayed in the detailed view in a broken circle.

**With Leader.** The **With Leader** option is used to add the leader to the callout of the detail view.

**No Leader.** The **No Leader** option is used to remove the leader from the callout of the detail view.

**Connected.** This option is used to create a line that connects the detail view with the closed profile in the parent view.

### View Options

The **View Options** rollout is used to set the parameters of the detail view. The various options available in this rollout are discussed next.

#### Full outline

The **Full outline** check box is used to display the complete outline of the closed profile in the detail view.

#### Pin position

The **Pin position** check box is used to pin the position of the detail view.

#### Scale hatch pattern

The **Scale hatch pattern** check box is used to scale the hatch pattern with respect to the scale factor of the detail view when you create a detail view of a section view.

Figure 12-33 shows the detail view created using the **Detail View** tool.

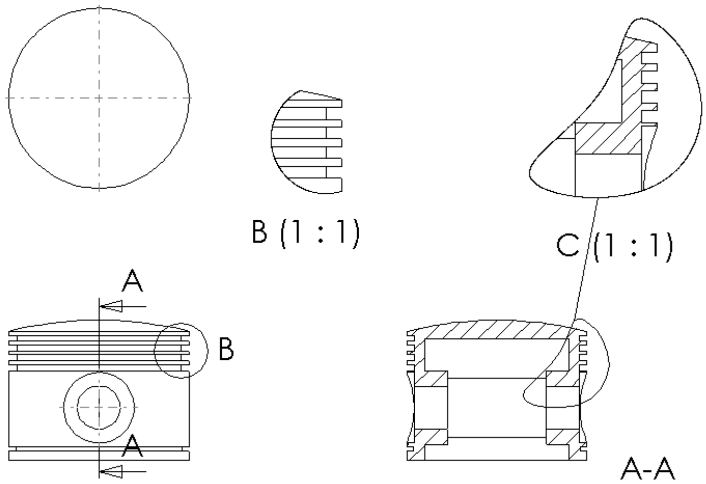
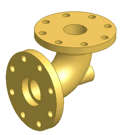



***Figure 12-33** Detail views*

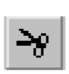


**Tip**. *When you create a detail view, by default the detail view is scaled as 1:1. You can define the scale factor in the **System Options** dialog box so that whenever you create a detail view, it will be created with the scaling factor provided by you. To specify the scale factor for the detail view, invoke the **System Options** dialog box and select the **Drawings** option from the left of this dialog box. Set the value of the scale factor of the detail view in the **Detail View Scaling** edit box and choose the **OK** button. Hence forth, the detail view will be created of the scale factor defined in the **System Options** dialog box.*

*If you need to scale a detail view in the current drawing sheet, select the view and select the **Custom Scale** check box. Specify the scale factor in the **Custom Scale** rollout and choose the **OK** button from the **Detail View PropertyManager**. The scaling of other views is discussed later in this chapter.*

## Cropping Drawing Views

| **Toolbar:** | Drawing > Crop View | (*Customize to Add*) |
|---|---|---|



This tool is used to crop an existing view using a closed sketch associated to that view. The portion of the view that lies inside the associated sketch is retained and the remaining portion is removed. To crop the view, you first need to create a closed profile associated to the view that defines the area of the view that will be displayed. The area of the view outside this closed profile will not be displayed when you crop the view. Select the closed profile and choose the **Crop View** (Customize to Add) button from the **Drawing** toolbar. Figure 12-34 shows the closed profile used to crop the view. Figure 12-35 shows a crop view.

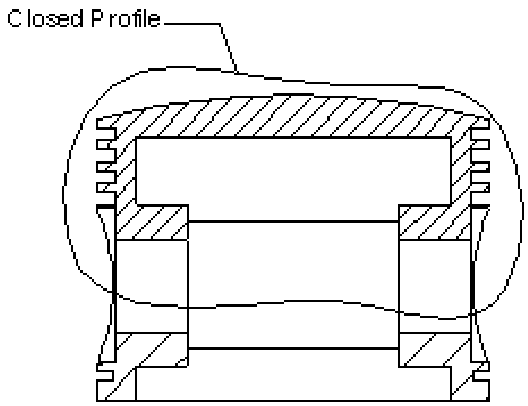



*Figure 12-34* Closed profile to crop the view

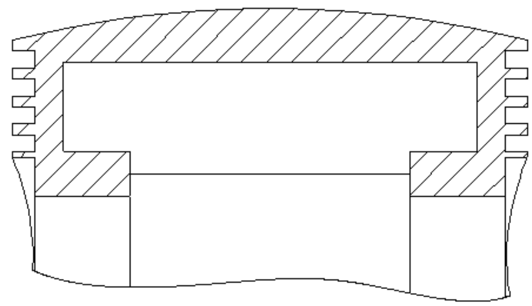
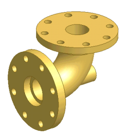


*Figure 12-35* Resultant crop view



**Tip**. *You can remove the cropping of view by selecting the crop view and invoking the shortcut menu. Choose* ***Crop View > Remove Crop*** *from the shortcut menu. The initial view will be displayed in the drawing sheet.*

*If you need to edit the closed profile of the crop view, select the crop view and invoke the shortcut menu. Choose* ***Crop View > Edit Crop*** *from the shortcut menu. The sketch of the closed profile and the complete view is displayed in the drawing sheet. Edit the closed profile and choose the* ***Rebuild*** *button from the* ***Standard*** *toolbar or use CTRL+B from the keyboard.*

## Broken View

A broken view is used to display a component by removing a portion of it from between, keeping the ends of the drawing view intact. This type of view is used for displaying the components whose length to width ratio is very high. This means that either the length is very large as compared to the width or the width is very large as compared to the length. The broken view will break the view along the horizontal or vertical direction such that the drawing view fits the area you require. To create a broken view, you first need to define the break line. Select the view you need to break and choose **Insert > Horizontal Break/Vertical Break** depending on the direction in which you need to break the component. Two break lines will be displayed on the selected view as shown in Figure 12-36.

After adding the break lines, you need to move the break lines to define the gap in the broken view. Select the break lines and move them away from each other as shown in Figure 12-37. Now, select the view and invoke the shortcut menu. Choose the **Break View** option from the shortcut menu. The broken view will be created as shown in Figure 12-38.

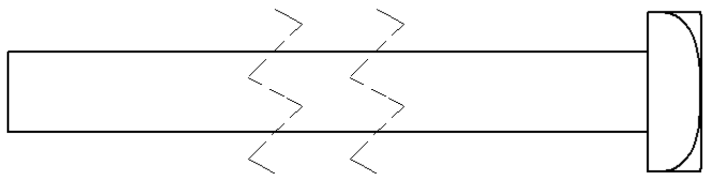
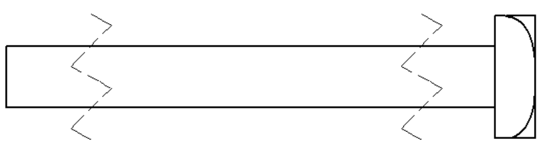


*Figure 12-36* Break lines added to the view



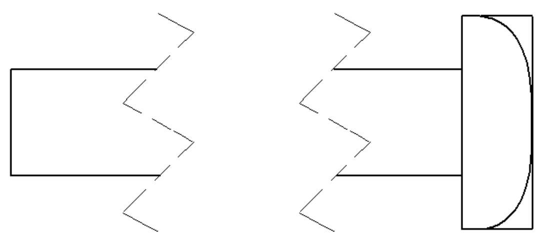


***Figure 12-37*** *Extended gap between break lines*

***Figure 12-38*** *Resultant broken view*

**Note**

*Select the break line to increase or decrease the gap between the broken view. As you move the break line, the broken view is modified dynamically.*

*If you generate a projected view from a broken view, the resultant projected view is also a broken view.*

*You can also break an isometric view; the procedure of breaking an isometric view or any 3D view is the same as discussed earlier. Figure 12-39 shows a broken isometric view.*

*If you break a 3D view placed horizontally, the two parts of the view as a result of the* ***Broken View*** *tool will lose their alignment.*

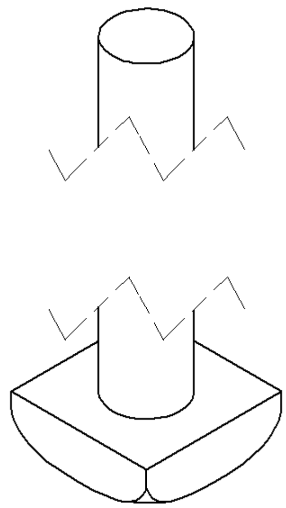
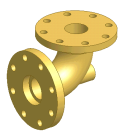


***Figure 12-39*** *A broken isometric view*



**Tip**. *You can change the style of the break line by selecting the break line and invoking the shortcut menu. The various break line styles available are straight cut, curve cut, zig zag cut, and small zig zag cut.*

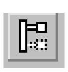
*To unbreak the broken view, select the view and invoke the shortcut menu. Choose the **Un-Break View** option from the shortcut menu.*

*Select the view and invoke the shortcut menu. Choose the **Break View** option to again break the view.*

*If you select the break lines and press the DELETE key from the keyboard, the broken view will be replaced by the parent view.*

## Alternate Position View

| | | |
|---|---|---|
| **Toolbar:** | Drawing > Alternate Position View | *(Customize to Add)* |
| **Menu:** | Insert > Drawing View > Alternate Position | |



The alternate position view is used to create a view in which you can show the maximum and minimum range of motion of an assembly. The main position is displayed with continuous lines in the drawing view and the alternate position of the assembly is shown in the same view with dashed (phantom) lines. To create an alternate position view, first you need to activate and select the view of the assembly drawing on which you need to create the alternate position view. Choose the **Alternate Position View** (Customize to Add) button from the **Drawing** toolbar or choose **Insert** > **Drawing View** > **Alternate Position** from the menu bar. The **Alternate Position PropertyManager** is displayed as shown in Figure 12-40.

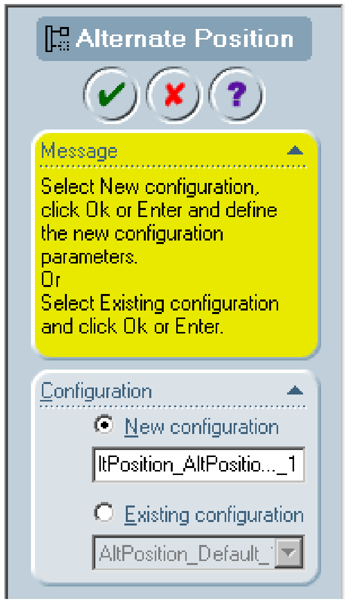



*Figure 12-40 The **Alternate Position PropertyManager***

The **Alternate Position PropertyManager** prompts you to select New configuration, click OK or Enter and define the new configuration parameters. Or Select Existing configuration and click OK or Enter. But you have not created any configurations yet. Therefore, the **New Configuration** radio button is automatically selected to create a new configuration. Enter the name of the configuration in the edit box given below. Choose the **OK** button from the **Alternate Position PropertyManager** to create a new configuration. Choose **OK** from the **Tangent Edge Display** dialog box

The assembly document is invoked and the **Move Component PropertyManager** is displayed in the assembly document. The **Move Component PropertyManager** prompts you to move the desired components to the position to be shown in the alternate view. Note that the component or components that you need to move to show in the alternate view should have that particular degree of freedom free to move. These components should not be fully defined in the assembly. Select and drag the cursor to move the components to the desired location. After defining the alternate position of the components, choose the **OK** button from the **Move Component PropertyManager**. You will return to the drawing document automatically. The alternate position of the components that are moved will be displayed in phantom lines in the drawing view, see Figure 12-41.

You can also create the alternate position view of an isometric view or of any 3D view. The procedure of creating the alternate position view of a 3D view is the same as that discussed earlier. Figure 12-42 shows the alternate position view of an isometric view.

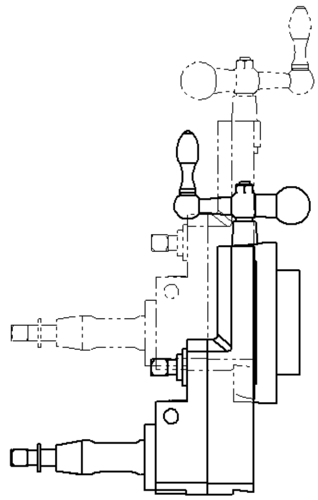


***Figure 12-41*** *Alternate position view*

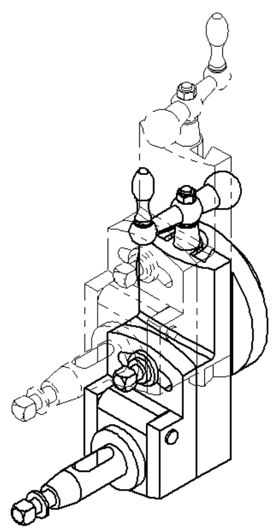


***Figure 12-42*** *Alternate view of an isometric view.*

**Note**

*When you create an alternate view of an assembly, a new configuration is created inside the assembly document with the same name that is specified to the configuration while creating the alternate position view. Open the assembly document and invoke the* ***ConfigurationManager****; you will observe that a new configuration is created along with the default configuration. By default, the newly created configuration is selected. Therefore, the assembly is displayed with the component moved to their extreme positions. To switch back to the default configuration, select* ***Default*** *from the* ***ConfigurationManager*** *and invoke the shortcut menu. Choose the* ***Show Configuration*** *option from the shortcut menu. You will observe that the assembly will be displayed with the moved components back at their original positions. If the* ***Show Configuration*** *option is not available in the shortcut menu, the assembly is at the default configuration.*

### Creating the Drawing view of the Exploded State of the Assembly

You can create the drawing view of the exploded state of the assembly. To generate the view of the exploded state of an assembly, you need to have an exploded state defined in the assembly document. Generate the isometric view of the assembly on the drawing sheet. Select the view and invoke the shortcut menu. Choose the **Properties** option from the shortcut menu. The **View Properties** tab of the **Drawing View Properties** dialog box is displayed. Select the **Show in exploded state** check box from the **Configuration information** area and choose the **OK** button from the **Drawing View Properties** dialog box. Figure 12-43 shows the drawing view of the exploded state of assembly with explode lines.

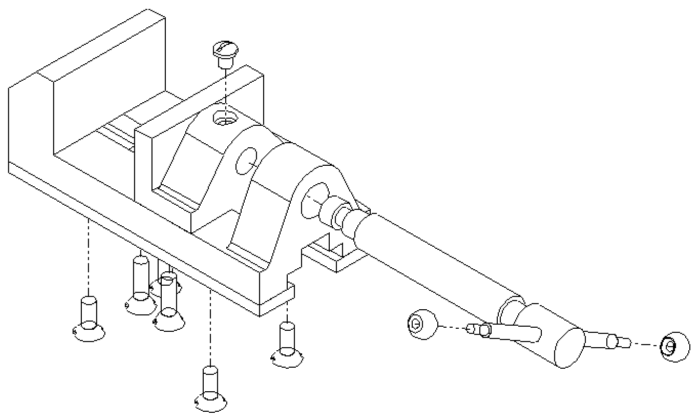
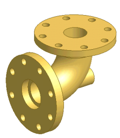


***Figure 12-43*** *Drawing view of the exploded state with explode lines*



**Tip**. *If currently the assembly in the assembly document is in the exploded state, then if you drag and drop the assembly to generate the drawing views, all the views of the assembly will be generated in the exploded state.*

*If you need to switch to the collapse state in the drawing view, select the model and clear the **Show in exploded state** check box from the **Drawing View Properties** dialog box.*

## WORKING WITH INTERACTIVE DRAFTING IN SOLIDWORKS

As mentioned earlier, you can also sketch the 2D drawings in the drawing document of SolidWorks. In technical terms, sketching 2D drawings is known as interactive drafting. Before starting the drawing, it is recommended that you insert an empty view and start creating the drawing in the empty view after activating that view. The 2D drawings are sketched using the standard sketching tools available in the **Sketch Tools** toolbar.

# EDITING THE DRAWING VIEWS

In SolidWorks, you can edit the drawing views using the **PropertyManager**. You can also change the orientation of the generated views using the **Named Views** option. To change the orientation, select the parent view that was created using the **Named** View option; the **Named View PropertyManager** is displayed. Double-click the view orientation that you want to be made current from the **View Orientation** rollout as shown in Figure 12-44. The orientation of the selected view will be modified. Choose the **OK** button from the **Named View PropertyManager**. You will notice that all the views derived from the parent view will also change their orientation when you change the orientation of the parent view.

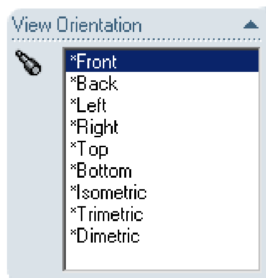



***Figure 12-44*** *The **View Orientation** rollout*

# CHANGING THE SCALE OF THE DRAWING VIEWS

In SolidWorks, you can also change the scale of the drawing views. To change the scale of the drawing views, select the drawing view and then select the **Custom Scale** check box to invoke this rollout available in the **PropertyManager**. The **Custom Scale** rollout is displayed in Figure 12-45. Set the scale of the drawing view in the **Custom Scale** rollout and choose the ENTER key. The view will be scaled using the current scale factor.

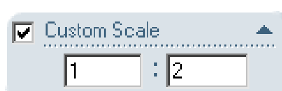



***Figure 12-45*** *The **Custom Scale** rollout*

**Note**

*When you scale a view, the view is scaled independently. This is the reason that if you scale the parent view, the views derived from the parent view will not be scaled. You can also change the scale factor of the derived view independent of its parent view.*

# DELETING THE VIEWS

The unwanted views are deleted from the drawing sheet using the **FeatureManager Design Tree** or directly from the drawing sheet. Select the view to be deleted from the **FeatureManager Design Tree** and invoke the shortcut menu. Choose the **Delete** option from the shortcut menu as shown in Figure 12-46. The **Confirm Delete** dialog box will be displayed. Choose the **Yes** button from this dialog box. You can also delete a view by selecting it directly from the

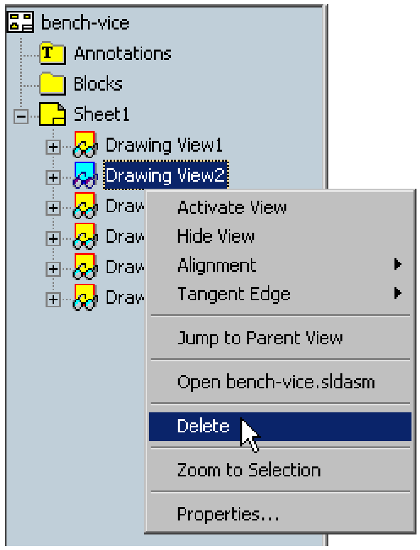
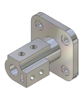


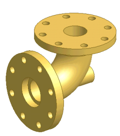

***Figure 12-46*** *Selecting the **Delete** option from the shortcut menu*

drawing sheet and pressing the DELETE key from the keyboard. The **Confirm Delete** dialog box is displayed; choose the **Yes** button from this dialog box.



**Note**

*When you delete a parent view, the projected views are not deleted. If you delete the parent view from which a section view or a detail view is generated, the name of the dependent view is also displayed in the **Confirm Delete** dialog box. If you choose **Yes**, the dependent views are also deleted.*



**Tip**. *You can also rotate a drawing view in the 2D plane. To rotate a drawing view, select the view and choose the **Rotate View** button from the **View** toolbar. The **Rotate Drawing View** dialog box is displayed. You can enter the value or rotation angle in this dialog box or you can also dynamically rotate the drawing view. If you select the **Dependent views update to change in orientation** check box, the views dependent on the rotated view will also change their orientation.*

*You can also copy and paste the drawing view in the drawing sheet. Select the view to copy and press CTRL+C from the keyboard. Now, select any where in the drawing sheet to select the sheet and press CTRL+V from the keyboard to paste the drawing view.*

## MODIFY THE HATCH PATTERN OF THE SECTION VIEW

As discussed earlier, when you generate a section view of an assembly, a default hatch pattern is automatically defined in the section view. If you need to modify the default hatch pattern, select the hatch pattern from the section view and invoke the shortcut menu. Choose the **Properties** option from the shortcut menu; the **Area Hatch/Fill** dialog box is displayed as shown in Figure 12-47. The various options available in this dialog box are discussed next.

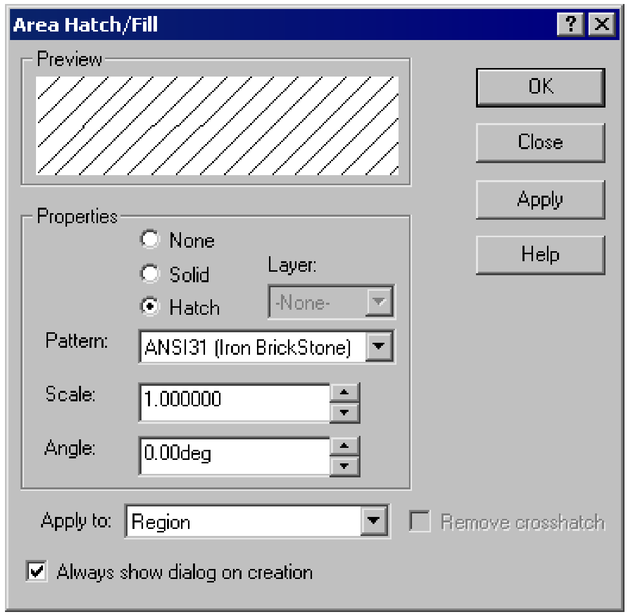



*Figure 12-47 The **Area Hatch/Fill** dialog box*

## Preview

The **Preview** area is used to preview the hatch pattern with the current setting of the hatch pattern.

## Properties

The **Properties** area is used to define the type of hatch pattern and the properties of the hatch pattern. The options available in this area are discussed next.

### None

The **None** radio button is selected if you donot need to apply any hatch pattern in the section view.

### Solid

The **Solid** radio button is used to apply the solid filled hatch pattern to the section view. By default, black color is applied as solid filled hatch pattern.

### Hatch

The **Hatch** radio button is selected to apply the standard hatch patterns to the section view. When you select this button, some options available in this dialog box are invoked to define the properties of the hatch pattern. The options available to define the properties of the hatch pattern are discussed next.

### Pattern

The **Pattern** drop-down list is used to define the style of the standard hatch pattern you need to apply to the section view. Some of the standard hatch patterns available in this drop-down list are shown in Figure 12-48. The preview of the hatch pattern selected from this drop-down list is displayed in the **Preview** area of the **Area Hatch/Fill** dialog box.

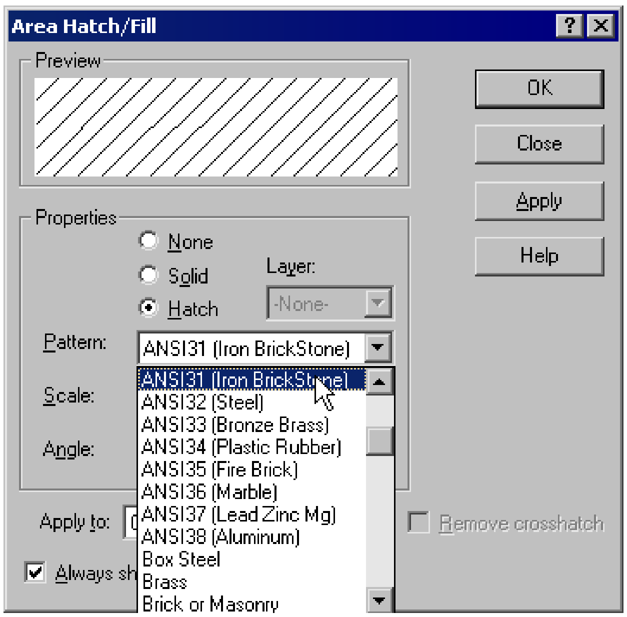



***Figure 12-48*** *Hatch patterns available in the* ***Pattern*** *drop-down list*

### Scale

The **Scale** spinner is used to specify the scale factor of the standard hatch pattern selected from the **Pattern** drop-down list. When you change the scale factor using this spinner, the preview displayed in the **Preview** area updates dynamically.

### Angle

The **Angle** spinner is used to define the angle to the selected hatch pattern.

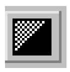
## Apply to

The **Apply to** drop-down list is used to define whether you need to apply this hatch pattern to the selected region or to the entire view. If you are editing the hatch pattern of a section view of an assembly, then you can also specify if you need to apply the hatch pattern to the component.

The **Always show dialog on creation** check box is used to invoke this dialog box when you apply the hatch pattern next time.

# APPLYING A HATCH PATTERN

| | |
|---|---|
| **Toolbar:** | Drawing > Area Hatch/Fill |
| **Menu:** | Insert > Area Hatch/Fill |



You can also apply the hatch pattern to a 2D sketch created using interactive drafting. To apply the hatch pattern to a closed sketch you just need to select any one entity of the closed sketch and then choose the **Area Hatch/Fill** button from the **Drawing** toolbar or choose **Insert > Area Hatch/Fill** from the menu bar. The **Area Hatch/Fill** dialog box will be displayed. Set the properties of the hatch pattern in this dialog box and choose the **OK** button from the **Area Hatch/Fill** dialog box.

You can also apply the hatch pattern to a face of the model in the drawing view. To apply the hatch pattern to the face of the model in the drawing view, activate the drawing view and select the face on which you need to apply the hatch pattern. Invoke the **Area Hatch/Fill** dialog box to apply the hatch pattern. Figure 12-49 shows the hatch pattern applied to the planar face of the model in the drawing view and a hatch pattern applied to a closed sketch created in the drawing view.

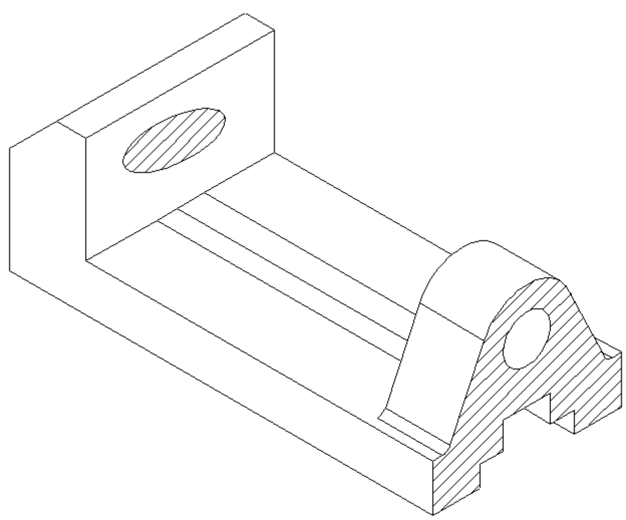
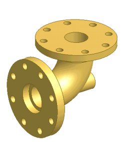


***Figure 12-49*** *Hatch pattern applied to the 2D sketch and the planar face*



**Tip**. *To change the color of the hatch or solid, select it and then choose the* ***Line Color*** *button from the* ***Line Format*** *toolbar. Select the required color from the* ***Edit Line Color*** *dialog box. It should be noted that you cannot change the color of the hatch patterns of the generated section view.*

# TUTORIALS

## Tutorial 1

In this tutorial you will generate the top view, front view, right view, aligned section view, detail view, and isometric view of the model created in the Tutorial 2 of Chapter 7. Use the Standard A4 Landscape sheet format for generating the views. **(Expected time: 30 min)**

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

a. Copy the model whose drawing views you want to generate in the current directory.
b. Create a new drawing document in the standard A4 Landscape sheet format and generate the parent view using the named view tool, refer to Figures 11-50 and 11-51.
c. Generate the projected views using the **Projected View** tool, refer to Figure 11-52.
d. Generate the aligned section view using the **Aligned Section View** tool, refer to Figures 11-53 through 11-56.
e. Generate the detail view and the isometric view, refer to Figure 11-57.

### Copying the Model in the Current Directory

First, you need to copy the model whose drawing views are to be generated in the current directory.

1. Create a directory with the name *c12* in the *SolidWorks* directory and copy *c07tut02.sldprt* from the */My Document/SolidWorks/c07* directory to this directory.

### Opening the New Drawing Document

As mentioned earlier in the description, you need to create a new drawing document with standard A4 sheet.

1. Create a new SolidWorks document in the drawing mode.

   The **Sheet Format To Use** dialog box is displayed. The **Standard sheet format** radio button is selected by default.

2. Choose the **A4 Landscape** sheet from the **Standard sheet format** drop-down list and choose the **OK** button from the **Sheet Format To Use** dialog box.

   The new drawing document is created with standard A4 sheet size as shown in Figure 12-50.

   As discussed earlier, by default the new drawing document starts with first angle projection. But you need to generate the drawing views in the third angle projection. Therefore, you need to change the type of projection from first angle to third angle.

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

   The **Sheet Setup** dialog box is displayed.

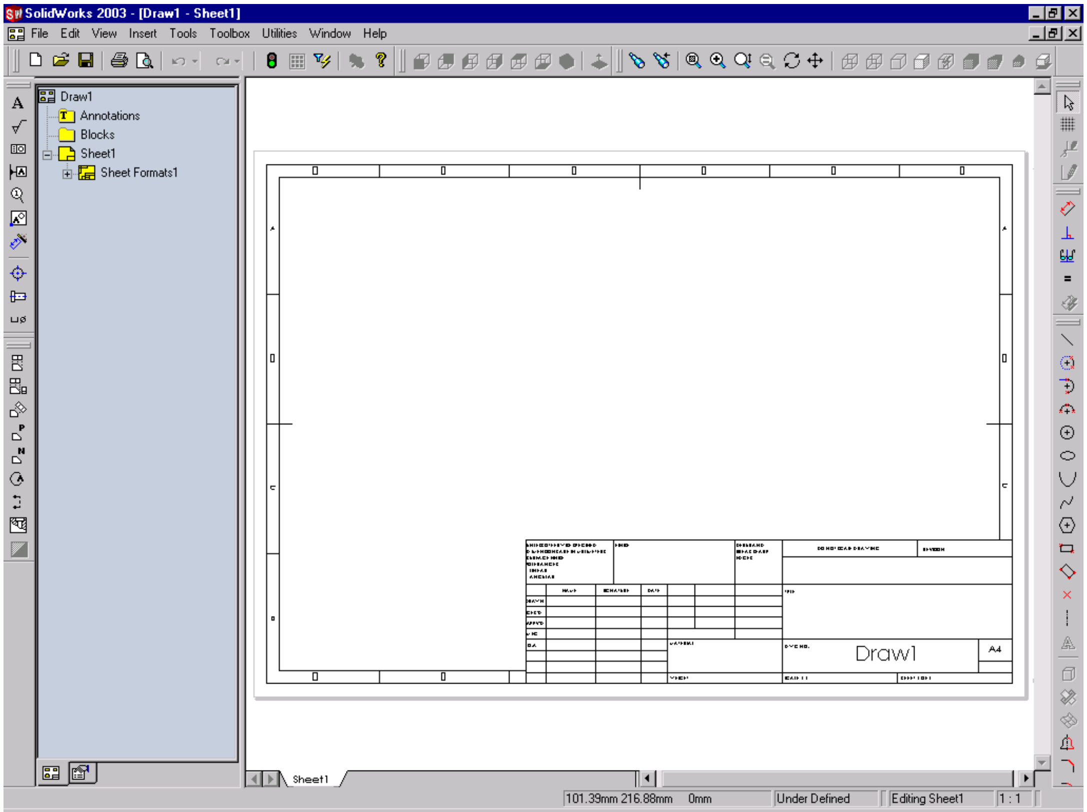


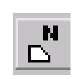

*Figure 12-50 New drawing document created with* ***A4*** *standard sheet format*

4. Select the **Third angle** radio button from the **Type of projection** area of the **Sheet Setup** dialog box and choose the **OK** button to exit the dialog box.

## Generating the Parent View and the Projected Views

First, you will generate the parent view in the drawing sheet. After generating the base view, you can use it to generate the other views such as the projected view, detail view, section view, and so on. The parent view will be generated using the **Named View** tool.

1. Choose the **Named View** button from the **Drawing** toolbar and right-click in the drawing sheet to invoke the shortcut menu.



2. Choose **Insert From File**; the **Open** dialog box is displayed.

3. Select *c07tut01.sldprt* and choose the **Open** button from the **Open** dialog box.

   The cursor is replaced by the view placement cursor and the **Named View PropertyManager** is displayed. The **Front** option is selected in the **View Orientation** rollout. Therefore, if you place the view, it will generate the front view. But the parent view that you need to generate is the top view.

4. Select the **Top** option from the **View Orientation** rollout.

5. Move the cursor close to the upper left corner of the drawing sheet and specify a point at this location to place the view.

6. If the **Tangent Edge Display** dialog box is displayed, select the **Don't ask me again** check box and choose the **OK** button from the **Tangent Edge Display** dialog box.

   Figure 12-51 shows the top view generated using the **Named View** tool bar. This view is generated at 1:1 scale.

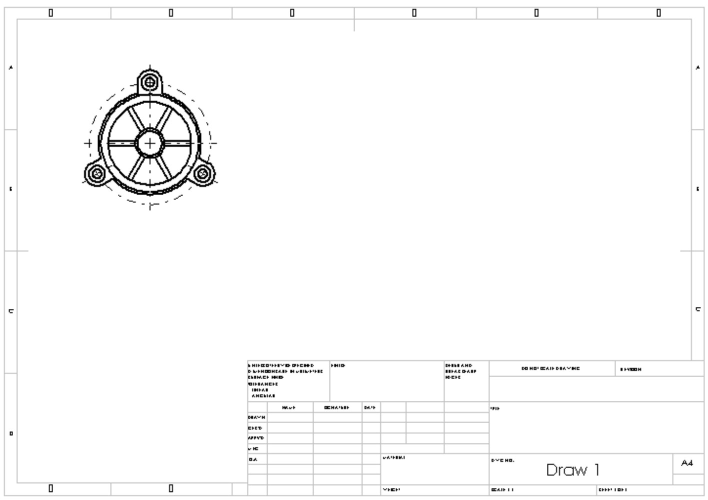
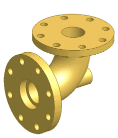


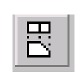

***Figure 12-51*** *Top view generated using the* ***Named View*** *tool*



**Tip**. *When you generate the drawing views in SolidWorks, if there is any cylindrical feature in the drawing view then the centermarks are automatically displayed in the drawing view containing the cylindrical feature.*

Next, you need to generate the projected view from the parent view generated earlier, which is the top view.

7. Select the parent view and choose the **Projected View** button from the **Drawing** toolbar.



   The cursor replaces the place view cursor.

8. Move the cursor below the parent view and specify a point to place the projected view.

9. Next, select the newly generated view and choose the **Projected View** cursor from the **Drawing** toolbar.

10. Move the cursor horizontally toward the right and specify a point on the screen to place the view. Figure 12-52 shows the views generated from the parent view using the **Projected View** tool.

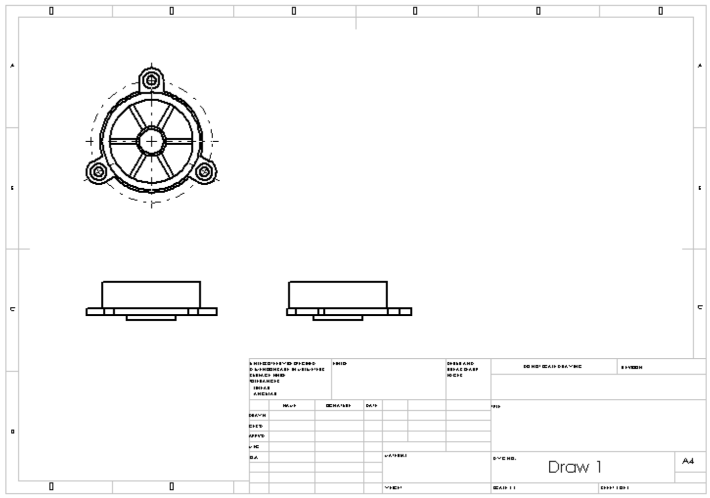



***Figure 12-52*** *Projected views derived from the top view*

## Generating the Aligned Section View

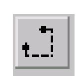
Next, you need to create the aligned section view. But before creating the aligned section view, you need to create a sketch that will define the section sketch for creating the aligned section view. Remember that the view is projected normal to the line sketched last.

1. Click on the top view to activate the view.

2. Using the **Line** tool draw the sketch and apply the relations and dimensions to the sketch as shown in Figure 12-53.

3. Select the dimension and invoke the shortcut menu. Choose the **Hide** option from the shortcut menu.

4. Now, select the inclined line and then the vertical line. Note that if you select the inclined line in the end, the view is generated normal to the inclined line. Now, choose the **Aligned Section View** button from the **Drawing** toolbar or choose **Insert > Drawing View > Aligned Section** from the menu bar.



   The **Section View PropertyManager** is displayed and you will observe that the aligned section view will be attached to the cursor as you move the cursor on the drawing sheet. The view generated is normal to the vertical line of the section sketch and the direction of viewing the section creation is the reverse to the required direction. Therefore, first you need to flip the viewing direction of section view.

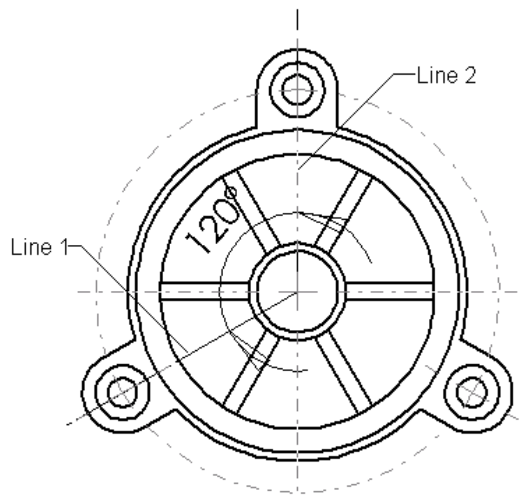



*Figure 12-53* *Sketch to be used as section sketch for the aligned section view*

5. Select the **Flip direction** check box from the **Line Options** rollout of the **Section View PropertyManager**.

6. Place the aligned section view on the drawing sheet as shown in Figure 12-54.

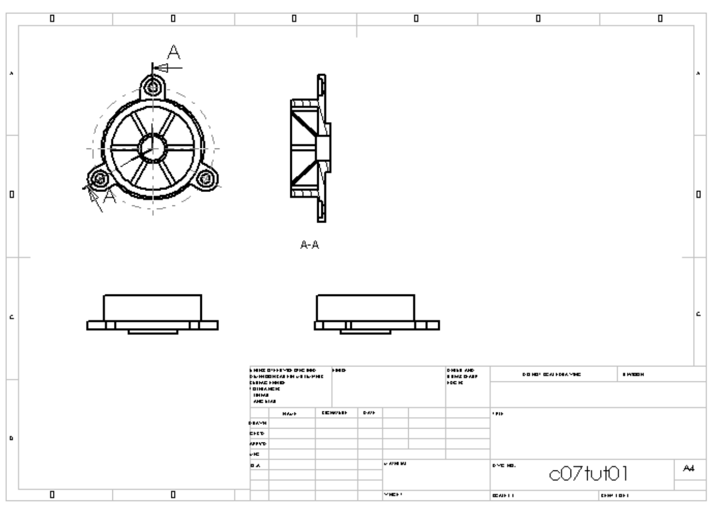



*Figure 12-54* *Sheet after generating the aligned section view*

## Modifying the Hatch Pattern of the Aligned Section View

The gap between the hatching line in the aligned section view is large; therefore, you need to modify the spacing.

1. Select the hatch pattern and right-click to invoke the shortcut menu. Choose **Properties** to display the **Area Hatch/Fill** dialog box.

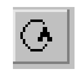
2. Set the value of the **Scale** spinner to **2** and select **View** from the **Apply to** drop-down list. Choose **OK** to close the dialog box.

## Generating the Detail View

Next, you need to generate the detail view of the right circular feature of the model. Before invoking the **Detail View** tool to generate the detail view, you need to activate a view from which you will drive the detail view.

1. Activate the top view.

2. Choose the **Detail View** button from the **Drawing** toolbar. The **Detail View PropertyManager** is displayed and you are prompted to sketch a circle to continue the view creation.



   The cursor will be replaced by the circle cursor.

3. Create a small circle on the right circular feature of the model in the top view, refer to Figure 12-55.

   As you create the circle, the detail view is attached to the cursor; place the view on the right of the drawing sheet.

4. Set the value of the scale factor of the detail view to **3:1** and choose the **OK** button from the **Detail View PropertyManager**.

   Figure 12-55 shows the detail view derived from the top view.

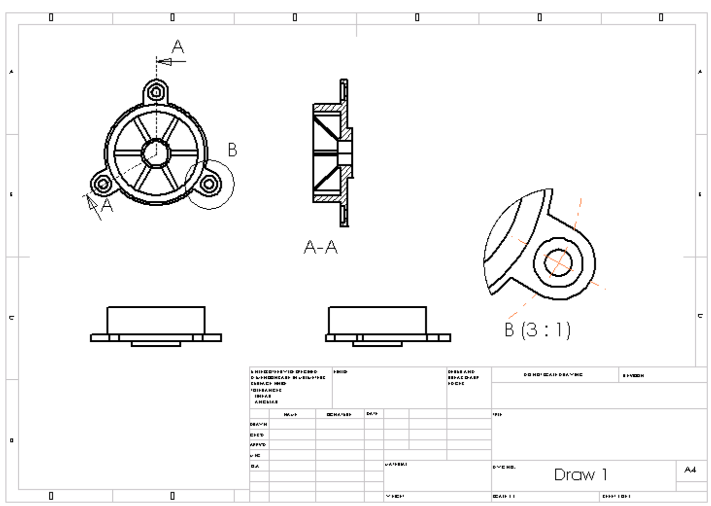
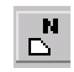



***Figure 12-55*** *The detail view derived from the top view*

## Generating the Isometric View

The last view that you need to generate is the isometric view. You will generate the isometric view using the **Named View** tool.

1. Choose the **Named View** button from the **Drawing** toolbar. The **Named View PropertyManager** is displayed and it lists various options of selecting the model to generate the drawing view.



   The cursor will be replaced by the select model cursor.

2. Select any of the view from the drawing sheet to select the model.

   A view is attached to the cursor.

3. The **Isometric** option is selected from the **Orientation View** rollout. Place the view close to the top right corner of the drawing sheet and choose the **OK** button from the **Named View PropertyManager**.

   Figure 12-56 shows the final drawing sheet after generating all the models.

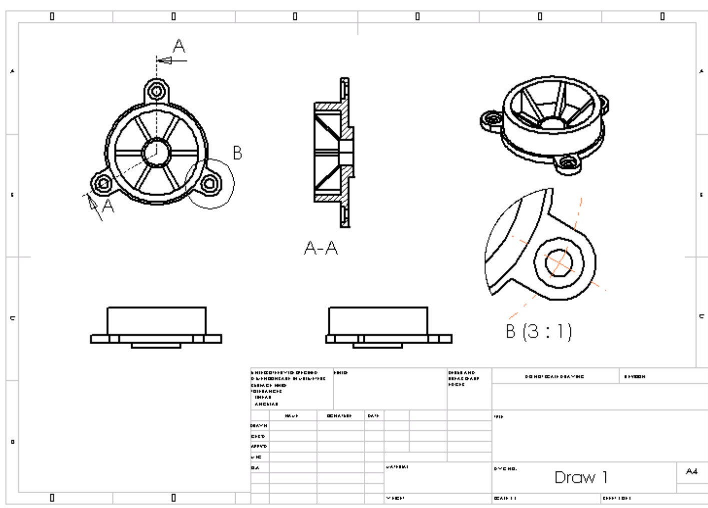



***Figure 12-56** The detail view derived from the top view*

## Saving the Drawing

Next, you need to save the drawing.

1. Choose the **Save** button from the **Standard** toolbar and save the drawing with the name given below and close the file.

   *\My Documents\SolidWorks\c12\c12-tut01.SLDDRW*.

## Tutorial 2

In this tutorial you will generate the drawing view of the Bench Vice assembly created in Chapter 10. You will generate the top view, section front view, right view, and isometric view of the assembly in the exploded state. **(Expected time: 45 min.)**

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

a. Copy the Bench Vice directory from Chapter 10 to the current directory.
b. Create the exploded state of the Bench vice assembly, refer to Figure 12-57.
c. Create the drawing document in the standard A4 Landscape sheet format and generate the parent view using the named view tool, refer to Figure 12-58.
d. Generate the section views using the **Section View** tool, refer to Figure 12-59.
e. Generate the right projection view using the **Projected View** tool, refer to Figure 12-60.
f. Generate the isometric view and change the state of the isometric view to the exploded state, refer to Figure 12-61.

### Copying the Model in the Current Directory

First, you need to copy the Bench Vice assembly and its parts to the current directory.

1. Copy the Bench Vice directory from the */My Document/SolidWorks/c10* directory to the current directory.

### Creating the Exploded View of the Assembly

Before proceeding further to create the drawing views of the assembly, you need to create the exploded state of the assembly in the assembly mode.

1. Open the Bench Vice assembly and create the exploded state and the explode lines as shown in Figure 12-57. It is recommended that whenever you create an exploded state of

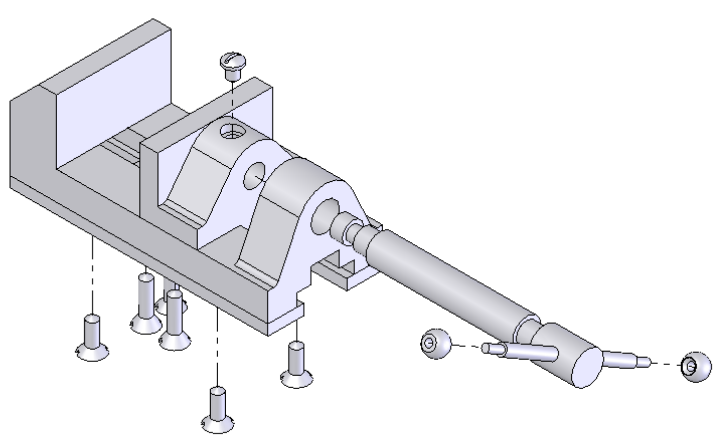


***Figure 12-57*** *Exploded view of the assembly with explode lines*

an assembly, you must revert to the collapse state. If you save the assembly in the exploded state, everytime you generate the drawing views of the assembly, it will generate views with the exploded state.

2. Right-click **Bench Vice Configutarion(s)** > **Default** in the **ConfigurationManager** and choose **Collapse** to unexplode the assembly.

3. Save and close the assembly.

## Opening the New Drawing Document

As mentioned earlier in the description, you need to create a new drawing document with standard A4 sheet.

1. Create a new SolidWorks document in the drawing mode.

   The **Sheet Format To Use** dialog box will be displayed. The **Standard sheet format** radio button is selected by default.

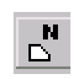
2. Select the **A4 Landscape** sheet from the **Standard sheet format** drop-down list and choose the **OK** button from the **Sheet Format To Use** dialog box.

3. Change the projection type from first angle to third angle.

## Creating the Parent View

The parent view generated in this tutorial is the top view. This view will be generated using the **Named View** tool.

1. Choose the **Named View** tool and invoke the **Open** dialog box.



2. Open the Bench vice assembly.

3. Select the **Top** option from the **View Orientation** rollout and place the view close to the upper left corner of the drawing sheet.

   When you place the view, you will notice that the view placed is larger in size. The size of the view is not that is required. Therefore, you need to scale the view.

4. Select the **Custom Scale** check box to invoke this rollout.

5. Set the value of the scale factor to **1:2** and choose the **OK** button from the **Named View PropertyManager**.

   You may also need to move the view. Figure 12-58 shows the parent view placed in the drawing sheet.

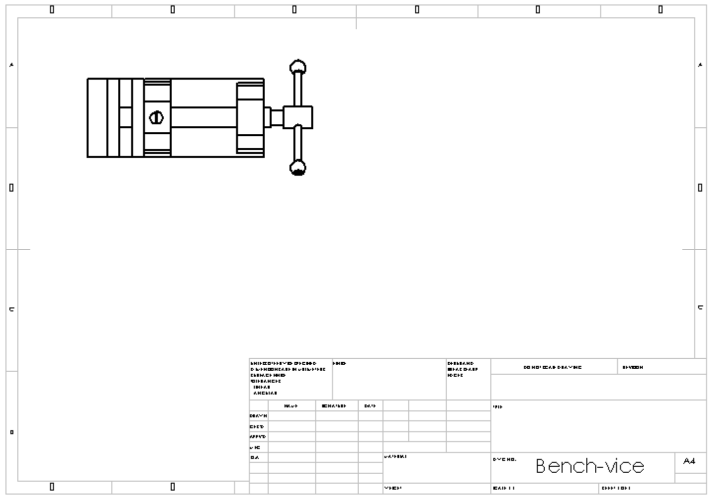
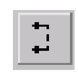



*Figure 12-58 Top view generated using the **Named View** tool*

## Creating the Sectioned Front View

The next view that you need to generate is the sectioned front view that is derived from the parent view.

1. Activate the top view and choose the **Section View** button from the **Drawing** toolbar.



   The **Section View PropertyManager** is displayed and it prompts you to sketch a line to continue view creation. The cursor will be replaced by the line cursor.

2. Create a horizontal line such that it passes through the center of the Bench vice assembly.

   As soon as you create the line, the **Section View** dialog box is displayed. This dialog box is used to exclude the components from the section cut.

3. Invoke the **FeatureManager Design Tree** flyout and expand **Drawing View1** from the flyout.

4. Now, expand the Bench vice assembly from the flyout.

5. Select the component that will be excluded from the section cut. The components that will be excluded from the section cut are Screw Bar, Bar Globes, Jaw Screw, Oval Fillister, Set Screw1, and Set Screw2.

6. Select the **Auto hatching** check box and choose the **OK** button from the **Section View** dialog box.

The preview of the section view is displayed on the drawing sheet as you move the cursor up and down. The direction of viewing of section view is the same as required. Therefore, you need to flip the viewing direction of section view.

7. Select the **Flip direction** check box and place the section view below the parent view. Choose the **OK** button from the **Section View PropertyManager**.

   Figure 12-59 shows the section view generated using the **Section View** tool.

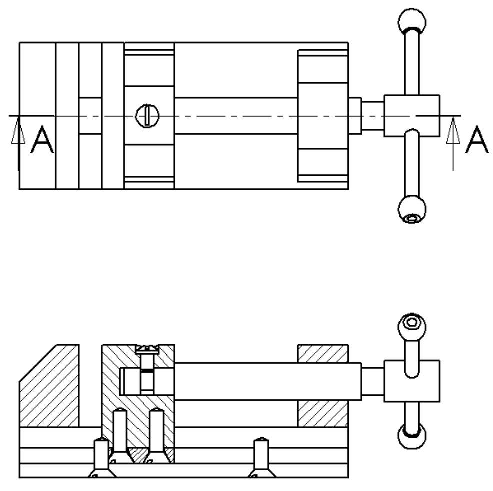



***Figure 12-59*** *Section view generated using the* ***Section View*** *tool*

## Generating the Right-Side View

The next view that you need to generate is the right-side view derived from the front section view and will be generated using the **Projected View** tool.

1. Select the sectioned front view and invoke the **Projected View** tool.

2. Move the cursor to the right of the sectioned front view and place the view on the right of the sectioned front view as shown in Figure 12-60.

## Creating the Isometric View in the Exploded State

The last view that you need to generate is the isometric view in the exploded state.

1. Using the **Named View** tool, generate the isometric view and place the view near the upper right corner of the drawing sheet.

2. Set the scale factor of the drawing view to **1:2**.

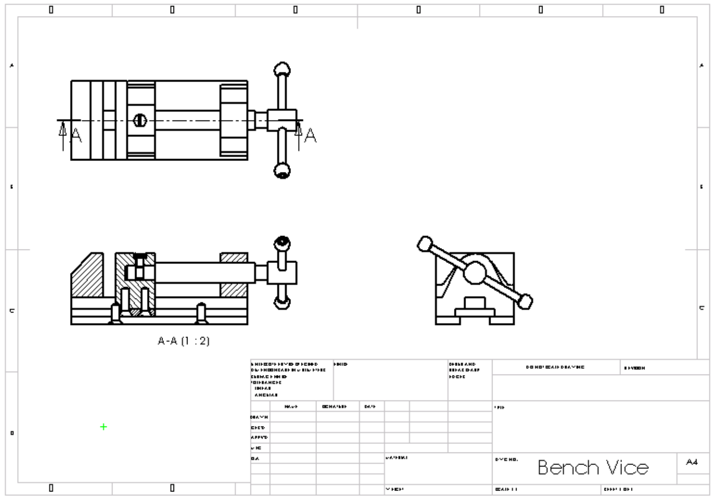



***Figure 12-60*** *Right view generated using the* ***Projected View*** *tool*

3. Select the view and invoke the shortcut menu. Choose the **Properties** option from the shortcut menu.

   The **Drawing View Properties** dialog box is displayed.

4. Select the **Show in exploded state** check box from the **Drawing View Properties** dialog box and choose the **OK** button.

   You may need to move the view. Figure 12-61 shows the final drawing sheet after generating all the drawing views.

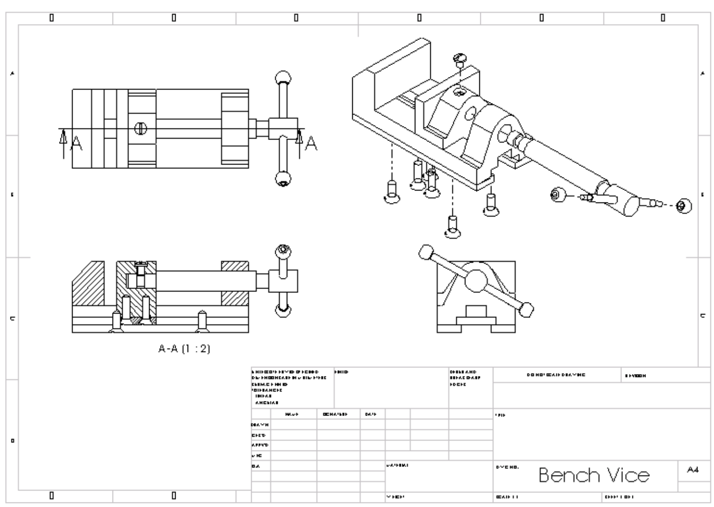



***Figure 12-61*** *Final drawing sheet*

### Saving the Drawing

Next, you need to save the drawing.

1. Choose the **Save** button from the **Standard** toolbar and save the drawing with the name given below and close the file.

   \My Documents\SolidWorks\c12*c12-tut02.SLDDRW*.

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## SELF-EVALUATION TEST

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

1. The **Standard sheet format** radio button is selected by default in the **Sheet Format To Use** dialog box. (T/F)

2. The **No sheet format** radio button is selected if you want to use the empty sheet, without any margin lines or title block. (T/F)

3. The **Relative View** tool is used to generate an orthographic view; the orientation of the view is defined by selecting the reference planes or the planar faces of the model. (T/F)

4. An auxiliary view is a drawing view that is generated by projecting the lines normal to a specified edge of an existing view. (T/F)

5. You cannot change the style of the break line in a broken view. (T/F)

6. In technical terms, creating a 2D drawing in the drawing document is known as __________.

7. To create the predefined views choose the __________ button from the **Assembly** toolbar.

8. The __________ check box available in the **View Options** rollout is used to display the complete outline of the closed profile in the detail view.

9. For changing the scale of the drawing views select the drawing view and select the __________ check box to invoke this rollout available in the **PropertyManager**.

10. For rotating a drawing view, select the view and choose the __________ button from the **View** toolbar.

# REVIEW QUESTIONS

**Answer the following questions:**

1. Choose the __________ button from the **Drawing** toolbar to create an alternate position view.

2. The __________ dialog box is used to apply the hatch pattern to a closed profile.

3. The __________ check box is used to scale the hatch pattern.

4. A __________ view is a section view in which only the sectioned surface is displayed in the section view.

5. The __________ dialog box is displayed to confirm the deletion of the views.

6. The views that are generated from a view already placed in the drawing sheet are known as?

   (a) Child views (b) Derived views
   (c) Predefined views (d) Empty views

7. Which rollout is used to set the parameters of the detail view in the **Detail View PropertyManager**?

   (a) **View Options** (b) **View Parameters**
   (c) **Parameters** (d) **Options**

8. In which edit box the name of the auxiliary view is specified?

   (a) **Label** (b) **View name**
   (c) **Name** (d) **Detail view label**

9. Which drop-down list is used to define whether you need to apply this hatch pattern to the selected region or to the entire view?

   (a) **Define** (b) **Apply to**
   (c) **Name view** (d) None of these

10. From which rollout can you select the view orientation in the **Named View PropertyManager**?

    (a) **View Orientation** (b) **Define View**
    (c) **Specify View** (d) **Scale View**

# EXERCISE

## Exercise 1

In this exercise you will generate the front view, section right view, isometric view, and the alternate position view on the isometric view of Exercise 1 of Chapter 11. You need to scale the parent view to the scale factor of **1:3**. The views that you need to generate are shown in Figure 12-62. **(Expected time: 30 min.)**

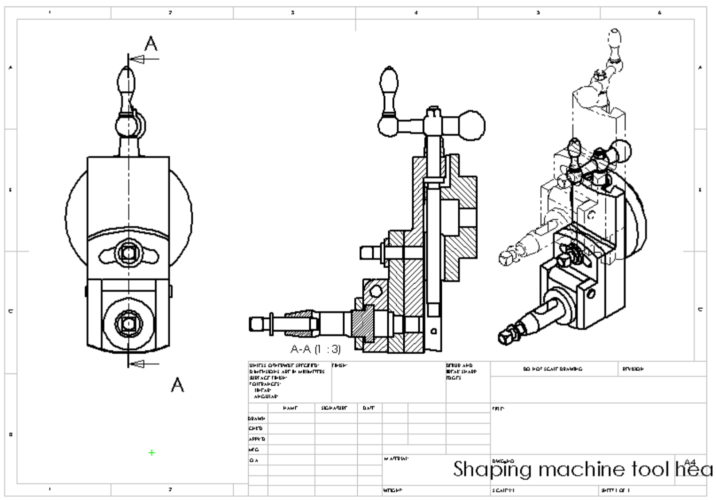


***Figure 12-62*** *Views of Exercise 1*

### Answers to Self-Evaluation Test

**1.** T, **2.** T, **3.** T, **4.** T, **5.** F, **6.** Interactive drafting, **7. Predefined View**, **8. Full outline**, **9. Custom Scale**, **10. Rotate View**