



# Chapter 12

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## Working With Drawing Views-I

### Learning Objectives

After completing this chapter you will be able to:

- *Generate standard three views.*
- *Generate Model views.*
- *Generate Relative views.*
- *Generate Predefined 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 the 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.*

## THE DRAWING MODE

After creating the solid models of the parts, or assemblies, you need to generate the two-dimensional (2D) drawing views. These views are the lifeline of all the manufacturing systems because at the shop floor or machine floor, the machinist mostly needs the 2D drawing for manufacturing. SolidWorks has provides a specialized environment, known as the **Drawing** mode, that has all the tools required to generate and modify the drawing views, and add dimensions and annotations to them. In other words, you can get the final shop floor drawing using this mode of SolidWorks. You can also sketch 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 of using 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 advantages of working in SolidWorks is that this software is bidirectionally associative in nature. This property ensures that the modifications made in a model in the **Part** mode are reflected in the **Assembly** mode and the **Drawing** mode, and vice versa.

## STARTING A DRAWING DOCUMENT

To generate the drawing views, you need to start a new drawing document. There are two methods of starting a drawing document in SolidWorks 2005. You can use the **New SolidWorks Document** dialog box or the option available in the part of assembly document to start a drawing document. Both these methods are discussed next.

### Starting a New Drawing Document Using the New SolidWorks Document Dialog box

To start a new drawing document for generating the drawing views, invoke the **New SolidWorks Document** dialog box. Choose the **Drawing** button, as shown in Figure 12-1, and choose the **OK** button. A new drawing document is started and the **Sheet Format/Size** dialog box is also displayed. Figure 12-2 shows the initial screen of the drawing document with the **Sheet Format/Size** dialog box. Select the available drawing template file from this dialog box. A new drawing document will be started.

The **Model View PropertyManager** is invoked automatically when you start a new drawing document. Its appearance will depend on whether any part or assembly document was opened or not when you started the new drawing document.



**Tip.** If you start a new drawing document by choosing the **drawing** template from the **Tutorial** tab of the **New SolidWorks Document** dialog box, when used in the **Advanced** mode, the **Sheet Format/Size** dialog box is not displayed.

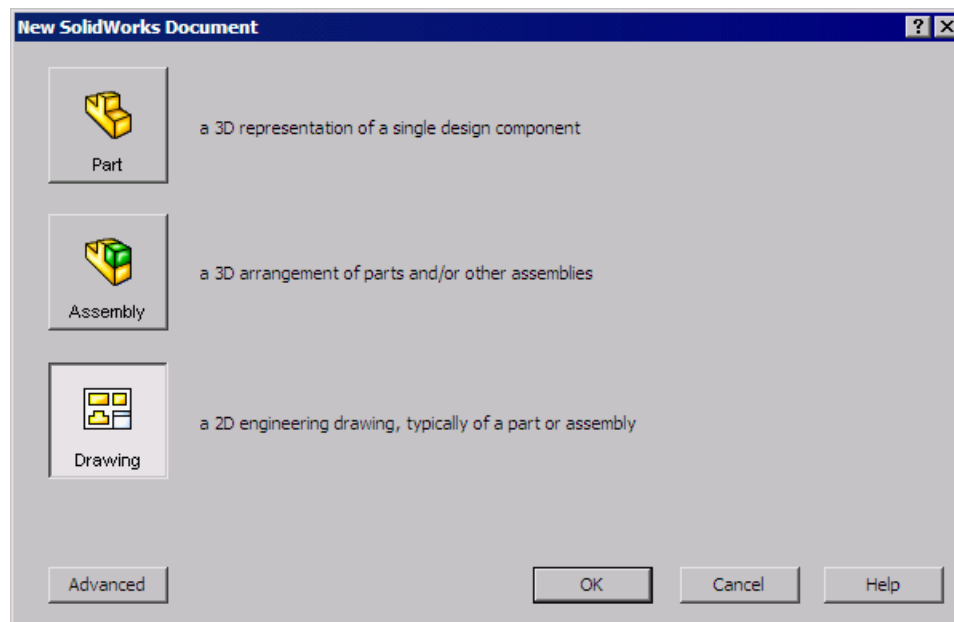


Figure 12-1 The New SolidWorks Document dialog box

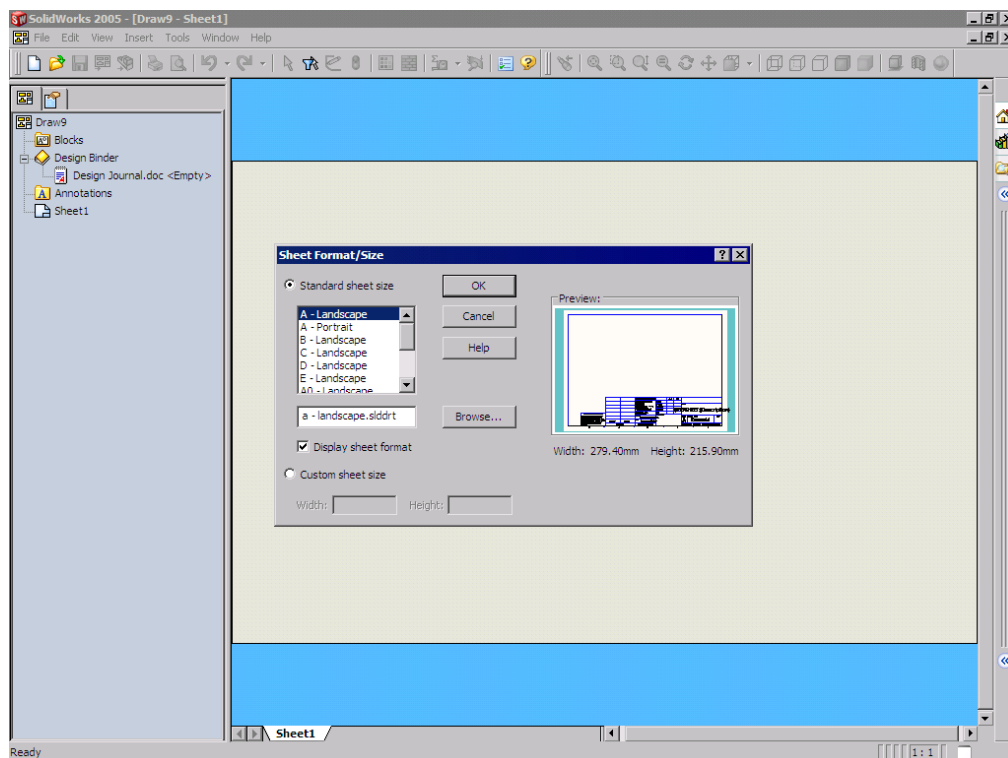


Figure 12-2 The initial screen of the drawing document with the Sheet Format/Size dialog box

**Evaluation chapter. Logon to [www.cadcim.com](http://www.cadcim.com) for more details**

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**Figure 12-3** A new drawing document with the **Model View PropertyManager**



**Tip.** If you choose the **Cancel** button from the **Sheet Format/Size** dialog box, a blank custom sheet of size 431.80 mm x 279.40 mm will be inserted in the drawing document.

## TYPE OF VIEWS

You can generate nine type of views in SolidWorks. Generally, you first need to generate a standard view, such as the top view or the front view, and then use it to derive the remaining generate or derive the following views from the standard view.

### Model View

The model view is used to create the base view in the drawing sheet. You can generate orthogonal views such as the front, top, left, and so on as the model view. You can also generate isometric, trimetric, or dimetric views as the model view.

### Projected View

The projected view is generated by taking an existing view as the parent view. It is generated by projecting the lines normal from the parent view or at an angle. The resulting view will be an orthographic view or a 3D 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. In SolidWorks, the section plane is defined using one or more sketched line segments.

### Aligned Section View

An aligned section view is used to section the 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 placed as a separate view. You can control the magnification of the detail view.

### Broken View

A broken view is the one in which a portion of the drawing view is removed from in 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 required area.

### 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 that lies behind the removed portion. This type of view is generated using a closed sketch 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 both 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 STANDARD DRAWING VIEWS

A standard view is generally the first view that you generate in the current drawing sheet. There are a number of methods of generating the standard drawing views. All these methods are discussed next.

## Generating Model Views

<b>Command Manager:</b>	Drawing > Model View
<b>Menu:</b>	Insert > Drawing View > Model
<b>Toolbar:</b>	Drawing > Model View



As mentioned earlier, the model views can be used to generate the base view in the drawing sheet. Whenever you start a new drawing document, the **Model View PropertyManager** is automatically invoked.

If you start the new drawing document from within the part or the assembly document, the part or the assembly is automatically selected and you can place the view. However, if you start the new drawing document using the **New SolidWorks Document** dialog box, a message will be displayed in the **Model View PropertyManager** and you will be prompted to select a part or assembly to generate the drawing view. If any part or assembly document is opened, it will be displayed in the selection area of the **Part/Assembly to Insert** rollout. You can preview the part or the assembly document by opening the **Thumbnail Preview** rollout, as shown in Figure 12-4.

You can also choose the **Browse** button and use the **Open** dialog box to select the document. After selecting the required part or assembly document, double-click on it in the **Part/Assembly to Insert** rollout. You can also choose the **Next** button available above the **Message** rollout in the **Model View PropertyManager**.

The **Model View PropertyManager** is automatically modified and now provides the options related to generating the standard views, as shown in Figure 12-5. The various options available in this **PropertyManager** are discussed next.

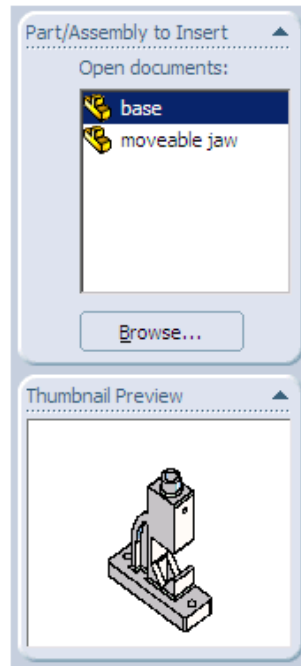


Figure 12-4 Selecting the document to generate the model views

### Orientation Rollout

The list box available in the **Orientation** rollout is used to specify the orientation of the view. The selection of the **Preview** check box allows you to preview the drawing view before it is placed.



#### Note

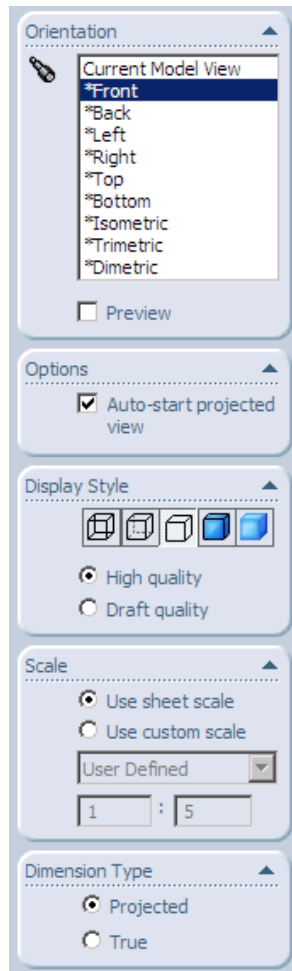
*You can change its orientation of the model view, even after placing it. To do this, place the view and make sure that it is still selected. This can be confirmed by the box that appears around the model view. The view is selected, if the box is displayed in green with grip point. Now, double-click on the required view orientation from the **Orientation** rollout in the **Model View PropertyManager**. The orientation of the view is automatically modified.*

### Options Rollout

The **Auto-start projected view** check box available in this rollout is used to automatically invoke the **Projected View** tool to generate the projected view immediately after placing the model view. The model view that you generate will be automatically taken as the parent view to generate the projected view.

### Display Style Rollout

The options available in this rollout are used to specify the display styles for the model view. These display styles are similar to those available in the **View** toolbar to display parts or assemblies in the part or assembly documents.



*Figure 12-5 Partial view of the **Model View PropertyManager** after selecting the document to generate the drawing views*

### Scale Rollout

A default scale, used to generate the drawing views is automatically defined, when you select a template to generate the drawing views. This is the reason the **Use sheet scale** radio button is selected in the **Scale** rollout. If you want to define a custom scale for the model view, select the **Use custom scale** radio button and specify the scale factor in the edit boxes available below this radio button.

### Dimension Type Rollout

The radio buttons available in this rollout are used to specify whether the model view will have true dimensions or projected dimensions. The true dimensions are the exact model dimensions that were specified while creating the model. The projected dimensions are reduced dimensions that are used in case of isometric, dimetric, or trimetric view. Generally, the value of the projected dimension is about 81.6% of the value of true dimension.





**Tip.** The drawing views will be generated depending on the default projection type of the current sheet. If the sheet is configured for first angle projection, the drawing views will be generated according to that. If the drawing sheet is configured for third angle projection, the views will be generated accordingly.

To change the projection type of the current sheet, right-click on **Sheet Format1** in the **FeatureManager Design Tree** and choose **Properties** from the shortcut menu to display the **Sheet Properties** dialog box. Set the required projection type using the **Type of projection** area.

## Generating the Three Standard Views

**CommandManager:** Drawing > Standard 3 View  
**Menu:** Insert > Drawing View > Standard 3 View  
**Toolbar:** Drawing > Standard 3 View



You can generate three default orthographic views of the specified part or assembly by using the **Standard 3 View** option. To create the three standard views, choose the **Standard 3 View** button from the **Drawing CommandManager**; the **Standard 3 View PropertyManager** will be displayed. If any part or assembly document is opened in the current session of SolidWorks, it will be displayed in the list box available in the **Part/Assembly to Insert** rollout, as shown in Figure 12-6.

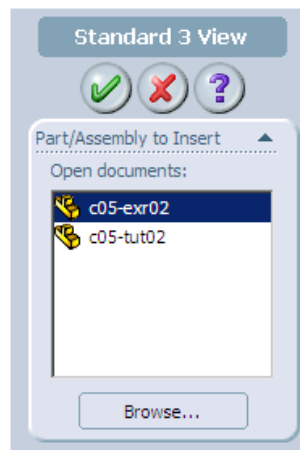


Figure 12-6 Standard 3 View PropertyManager

You can select the document from this list box. You can use the **Browse** button to select the part or assembly document, if no documents are opened. As soon as you select a document, three standard views are generated based on the default scale of the current sheet. Figure 12-7 shows the three standard views of a model generated in the third angle projection using the **Standard 3 View** tool.

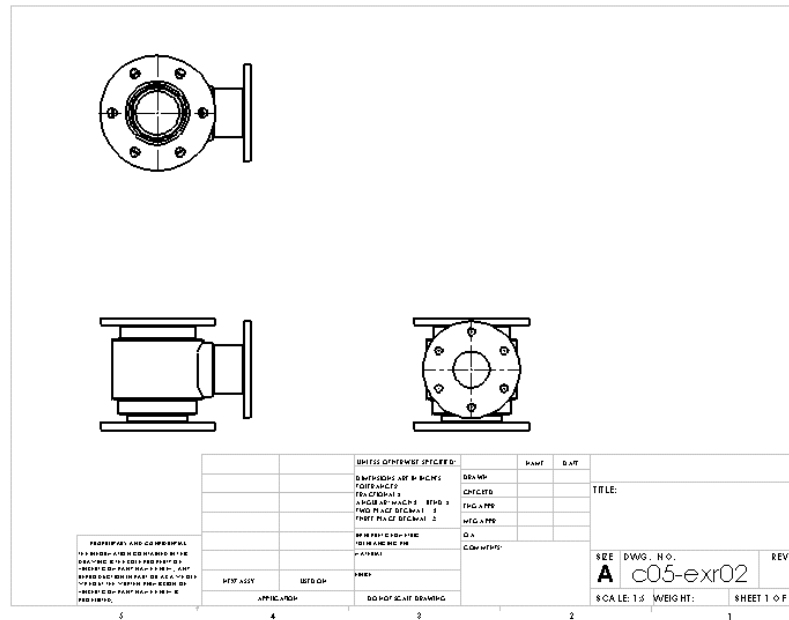


Figure 12-7 Three standard views generated using the **Standard 3 View** tool



#### 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. Try changing the sheet format, if these parameters are not displayed.

You will observe that center marks are automatically created on generating the drawing views. Otherwise, you can set the option to automatically create the center marks. To do this, invoke the **System Options** dialog box and choose the **Document Properties** tab. The **Detailing** tab is chosen by default. Select the **Center marks** check box from the **Auto insert on view creation** area. You can also set auto-insertion of centerlines, balloons, and drawing using the options provided in this dialog box.



**Tip.** You need to move the generated view if it overlaps the title block. Place the cursor over the view to move it; the bounding box of the view is displayed in dashed red lines. At this point, click to select the view. Next, 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 on moving the parent view, all the views generated using it are also moved.

## Generating Standard Views Using the Relative View Tool

<b>CommandManager:</b>	Drawing > Relative View (Customize to Add)
<b>Menu:</b>	Insert > Drawing View > Relative To Model
<b>Toolbar:</b>	Drawing > Relative View (Customize to Add)



The **Relative View** tool is used to generate an orthographic view. The orientation of the view is defined by selecting two 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.

Open the part or the assembly document and tile it vertically or horizontally with the drawing document in order to create a relative view. Invoke the **Relative View** tool to display the **Relative View PropertyManager**. You are prompted to select a planar face of the model. Click in the window of the part or assembly document; the **Relative View PropertyManager** will be displayed, as shown in Figure 12-8.

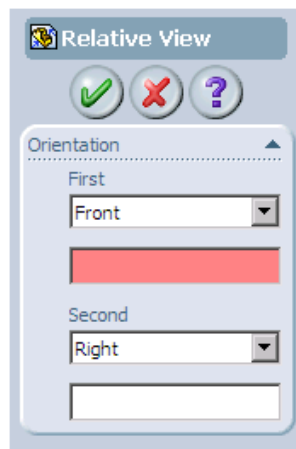


Figure 12-8 Relative View PropertyManager

Select the orientation for the first plane or planar face from the drop-down list available in the **First** area. Then select the plane or planar face of the model to be oriented in that direction. For example, you can select the **Top** option from this drop-down list and then select the top planar face of the model.

The selection area in the **Second** area is highlighted on specifying the first reference. Select the orientation for the second reference from the drop-down list and then select a plane or planar face. Next, choose the **OK** button from the **Relative View PropertyManager**. You will return to the drawing document. Place the view at the required location. Figure 12-9 shows the faces of the model selected to generate a standard view and Figure 12-10 shows the resulting view.

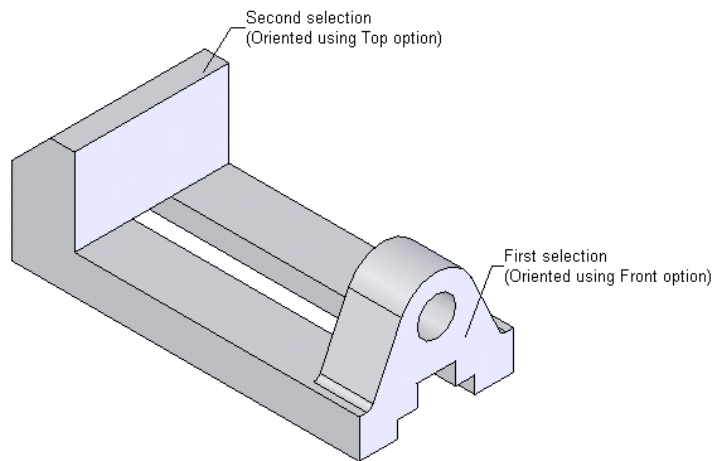


Figure 12-9 Faces to be selected

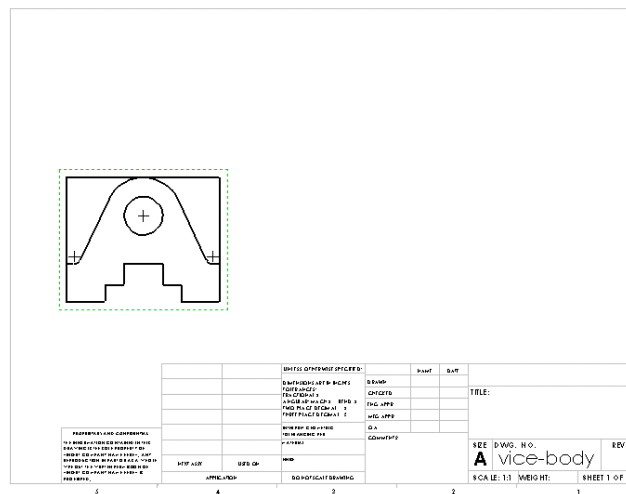


Figure 12-10 Resulting view

## Generating Standard Views Using the Predefined View Tool

**Toolbar:** Drawing > Predefined View (Customize to Add)  
**Menu:** Insert > Drawing View > Predefined  
**Toolbar:** Drawing > Predefined View (Customize to Add)



The **Predefined View** tool is used to create empty views with the predefined orientation. After their creation, you can populate them by dragging a part from the other window by holding the name of the part document in the **FeatureManager Design Tree**. All the predefined views will be populated. This option adds the empty views in the drawing document and then saves it as a drawing template. You just need to

drag and drop a part from another window, in the drawing document opened using the template, to populate it with all the predefined views. To create predefined views, choose the **Predefined View** button from the **Drawing CommandManager**. An empty view will be 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 **Drawing View PropertyManager** will be displayed, as shown in Figure 12-11.

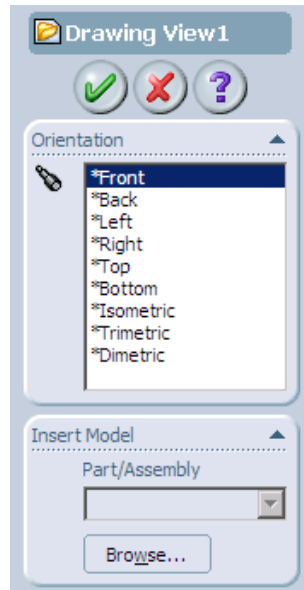


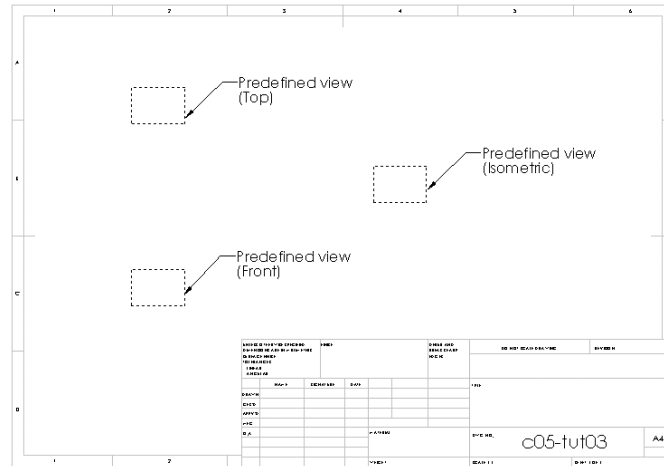
Figure 12-11 The Drawing View PropertyManager

Select the view orientation from the **View Orientation** area of the **View Orientation** rollout and choose the **OK** button from the **Drawing View PropertyManager**.

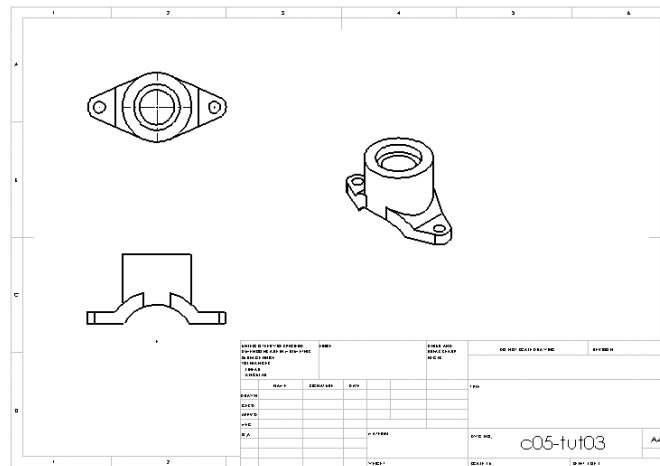
You need to define the alignment option while placing the next predefined view. To create additional predefined views, invoke the **Predefined View** tool and place the view in the drawing sheet. The bounding box of the view is displayed. Right-click inside the bounding box and choose **Alignment > Align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

After creating all the predefined views, open the part or assembly document and tile the document windows horizontally or vertically. Next, press and hold the left mouse button down on the name of the part or assembly in the **PropertyManager** and drag the component or the assembly in the drawing document; all the predefined views will be populated. You can also populate individual predefined views by selecting them and choosing the **Browse** button from the **Insert Model** rollout of the **Drawing View PropertyManager**.

Figure 12-12 shows the selected predefined views with the orientation in which the views are created, and Figure 12-13 shows the drawing document after populating the drawing views.



*Figure 12-12 Various predefined views*



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

**Note**

The views generated in the **Drawing** mode of SolidWorks, are automatically scaled, depending on the size of the sheet.

The views will also be scaled automatically, if the drawing contains more than one predefined view.

One predefined view, placed in the drawing document, will be scaled with respect to the **Custom Scale** value, if specified. Otherwise, it will be scaled with the default scale factor of the drawing sheet. You can change the view scale using the **Sheet Properties** dialog box. You will learn more about scaling the views in the next chapter.

## GENERATING THE DERIVED VIEWS

All views, generated from a view already placed in the drawing document, are known as derived views. These include:

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.

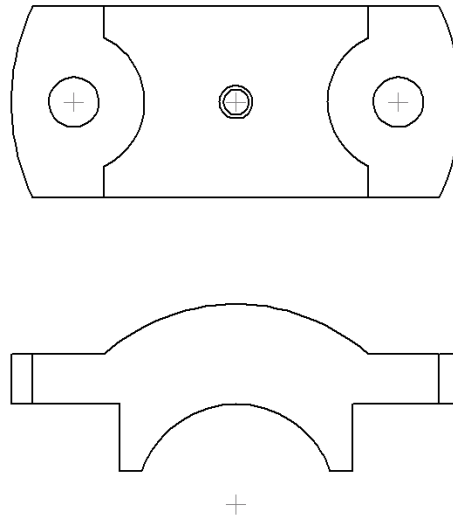
### Generating Projected Views

<b>CommandManager:</b>	Drawing > Projected View
<b>Menu:</b>	Insert > Drawing View > Projected
<b>Toolbar:</b>	Drawing > Projected View



As mentioned earlier, the projected views are generated by projecting the normal lines from an existing view, or at an angle from an existing view. To generate a projected view, choose the **Projected View** button from the **Drawing CommandManager**. The **Projected View PropertyManager** is displayed and you are prompted to select a drawing view 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 the top view or the bottom view, or move the cursor horizontally to create right or left view. If you move the cursor at an angle, a 3D view will be created. Specify a point on the drawing sheet to place the view. For generating more than one projected views, 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.** The drawing view is aligned to the parent view, when you generate a projected drawing view. To place the projected view that is not in alignment with the parent view, press and hold down the CTRL key before placing it. Next, move the cursor to the desired location and place the view.

All the standard and derived views such as 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 bounding box of the parent view will also be displayed.

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

## Generating Section Views

**CommandManager:** Drawing > Section View  
**Menu:** Insert > Drawing View > Section  
**Toolbar:** Drawing > Section View

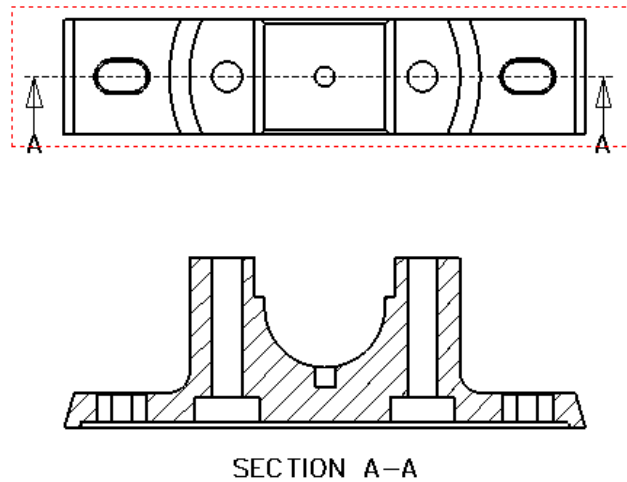


As mentioned earlier, section views are generated by chopping a portion of an existing view, using a cutting plane (defined by sketched lines), and then viewing the parent view from a direction normal to the cutting plane.

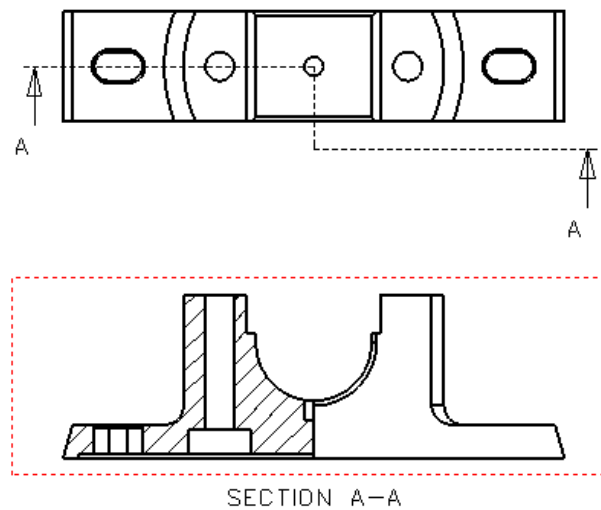
In SolidWorks, you can use the **Section View** tool to create a full section view, or a half section view, as shown in Figure 12-15 and Figure 12-16 respectively. A full section view is defined using a single line segment, but a half section view is defined using three line segments. Note that the section plane for a full section view can be defined after invoking the **Section View** tool. But to generate a half section view, you need to draw the line segments to define the section plane before invoking the **Section View** tool. To do this, select the drawing view that you want to use as







*Figure 12-15 Full section view*



*Figure 12-16 Half section view*

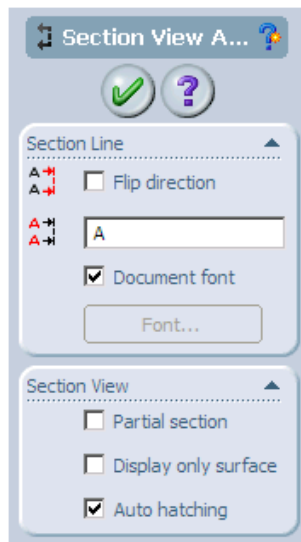
the parent view and choose the **Sketch** button from the **CommandManager**; the sketching environment will be invoked. You can use the inferencing lines to draw the lines for the section plane.

To create a full section view, activate the view in which you need to draw the section line. The view symbol will be displayed below the cursor and the bounding box of the view will also be displayed. Now, choose the **Section View** button from the **Drawing CommandManager**. The **Section View PropertyManager** is displayed and you will be prompted to sketch a line to continue view creation.

After activating the view, draw a line that will define the section plane. On specifying the endpoint of the section line, the view is defined and is attached to the cursor. Also, the other options in the **Section View PropertyManager** are displayed.

To generate a half section view, select all the line segments that define the section plane and then invoke the **Section View** tool. The section view is defined and is attached to the cursor.

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 and the **Section View PropertyManager** is invoked, as shown in Figure 12-17.



**Figure 12-17** Partial view of 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. You can also flip the direction of the viewing section view using the TAB key on the keyboard before placing the section view. 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 next chapter.



**Tip.** On creating a section view and moving the cursor to place the section view, you will observe that the view is aligned to the direction of arrows on the section line. In order 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.



**Note**

The default hatch pattern in the section view depends on the material assigned to the model. Also, you may need to increase the spacing of the hatch pattern, if it is not correct. You will learn more about editing the hatch pattern later in this chapter.

Some other options available in the **Section View PropertyManager** to create partial section view and the surface section view, 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.

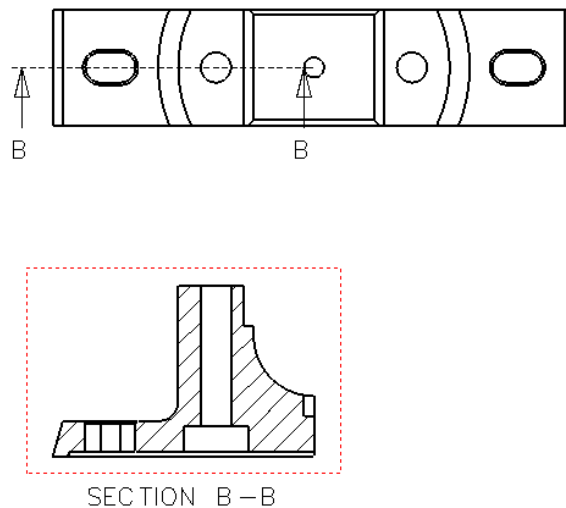


Figure 12-18 A partial section view

### Creating the Surface Section View

A surface section view is the one in which only the sectioned surface is displayed in the section view. To create a surface section view, you first need to create the section view and then select the **Display only surface** check box from the **Section View PropertyManager**. Figure 12-19 shows a surface section view.



**Tip.** Sometimes, the sectioned view is generated upside down, even if you have set the projection type to third angle. In such cases, you need to flip the direction of the section line from the **Section View PropertyManager**.

### 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.

This dialog box allows you to select the components that will be excluded from the section cut. You can also select the components from the parent view. But if the components are not visible

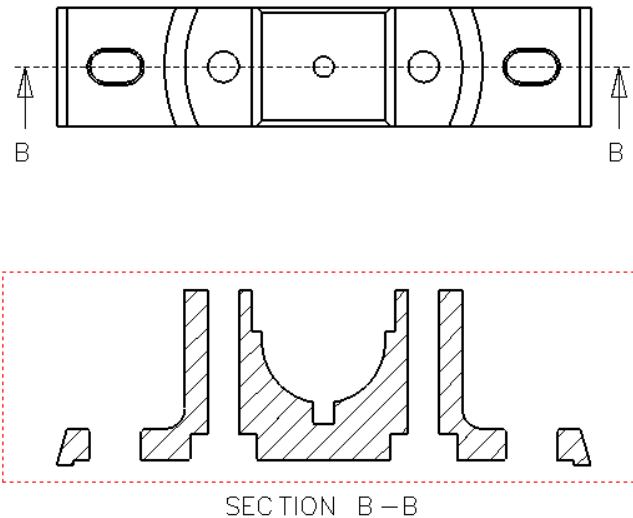


Figure 12-19 A surface section view

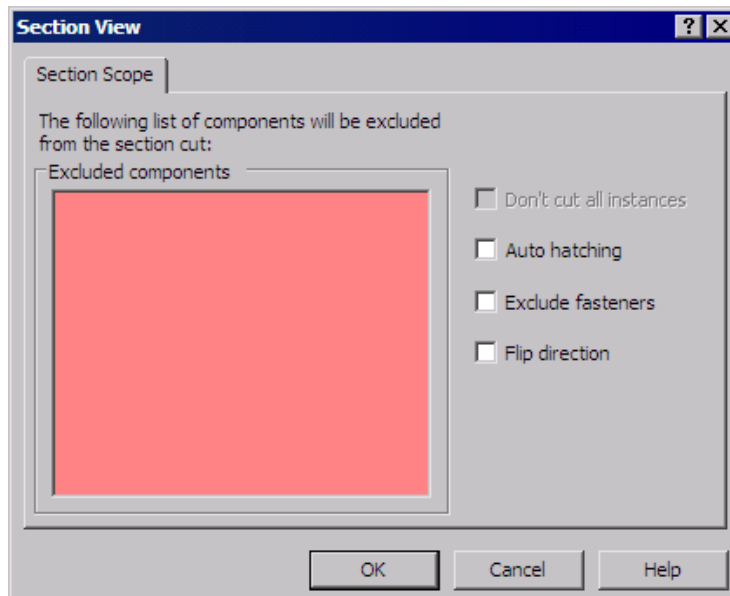


Figure 12-20 The Section View dialog box

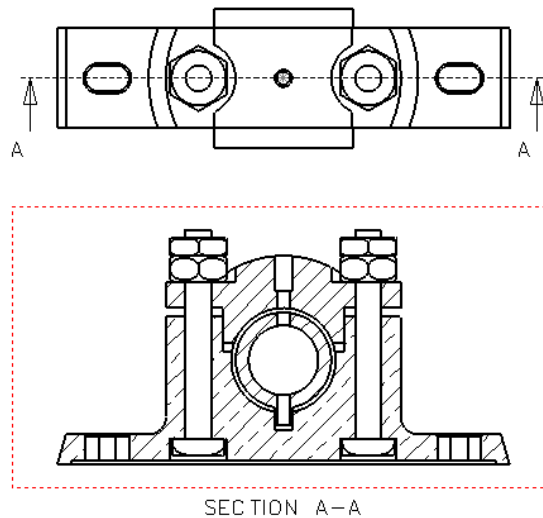
in the parent view, you can invoke the **FeatureManager Design Tree** and expand the parent drawing view. Next, expand the assembly tree view to display all the components of the assembly. Select the components that are not required to be sectioned. The name of the selected component is displayed in the **Excluded components** selection area.

The **Auto hatching** check box is used to automatically define the hatch patterns. You can even change them if required. The method of changing hatch patterns is discussed later. This release of SolidWorks provides you with an option to exclude the fasteners that are inserted in the

assembly, using the Toolbox application. Toolbox, one of the add-ins of SolidWorks, is used to insert standard fasteners to the assembly. To exclude the fasteners that are inserted using this option, select the **Exclude fasteners** check box from the **Drawing View Properties** dialog box.

The **Flip direction** check box is used to flip the direction of viewing of the section view.

In case you have more than one instance of the component in the assembly, and you need to exclude all the instances of the component from the section view, select the component from the drawing sheet and also the name of the component from the **Exclude components** selection area. Select the **Don't cut all instances** check box from the **Section Scope** dialog box. All instances of the selected component will be excluded from the section view. Figure 12-21 shows an assembly section view with, fasteners excluded from the cut.



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



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

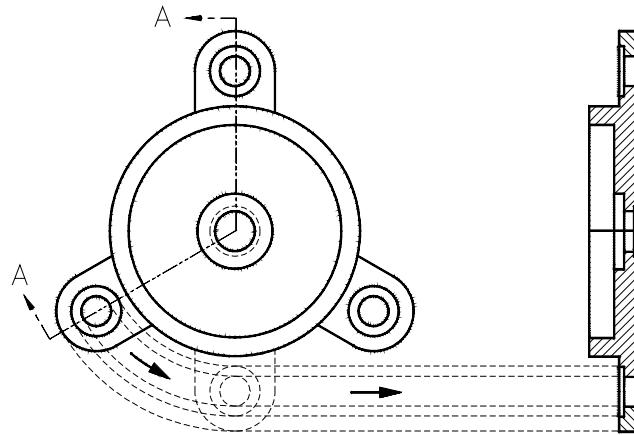
## Generating Aligned Section Views

<b>CommandManager:</b>	Drawing > Aligned Section View
<b>Menu:</b>	Insert > Drawing View > Aligned Section
<b>Toolbar:</b>	Drawing > Aligned Section View



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 sectioned portion revolves about an axis normal to the view such that it is straightened. For an example, refer to Figure 12-22.





**Figure 12-22** Aligned section view

This figure explains the concept of an aligned section view of a model. Notice that the inclined feature, sectioned in this view is straightened. As a result, the section view is longer than the parent view. Activate the view to create the aligned section view. Choose the **Aligned Section View** button from the **Drawing CommandManager**. Draw the sketch that defines the section plane. The aligned section view will be attached to the cursor; place the view at an appropriate location in the drawing sheet. Note that the resulting view will be projected normal to the line drawn at 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, followed by the vertical line. Figure 12-23 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 12-24 shows the view in which the inclined line is drawn first.

With this release of SolidWorks, you can also create a section view and aligned section views from a crop view, detail view, and an orthogonal exploded views.



**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, activate 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, by dragging a window around it, the view will be projected normal to the line that was drawn last.*

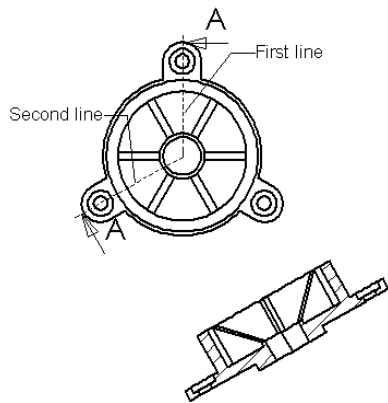


Figure 12-23 Aligned section view

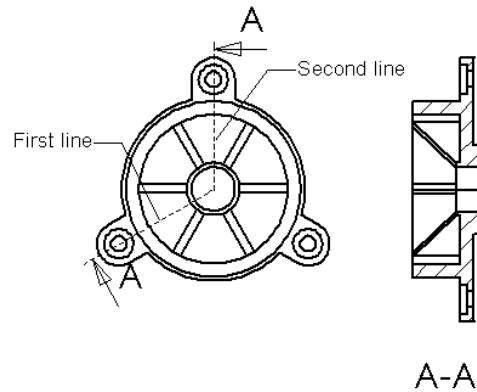


Figure 12-24 Aligned section view



**Tip.** With this release of SolidWorks you can also use more than two lines to create an aligned section view. To do this, you need to draw the lines prior to invoke the **Aligned Section View** tool.

## Generating Broken-out Section Views

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



This tool is used to create a broken-out section view, which 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 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 CommandManager**. The **Broken-out Section PropertyManager** is displayed and it prompts you to create a closed spline to continue the section creation. The cursor will be replaced by the spline cursor. Draw 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 12-25 shows an associated sketch created for creating a broken-out section view.

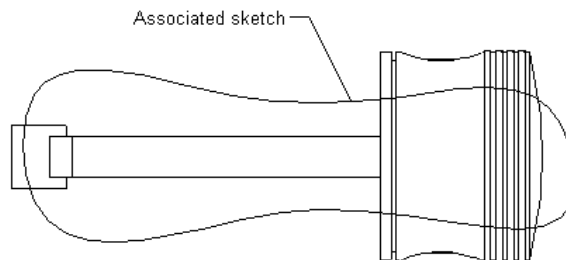
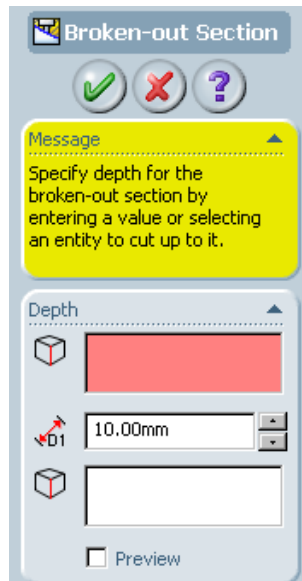


Figure 12-25 Sketch for creating a broken-out section view

When you draw a closed sketch, the options are displayed in the **Broken-out Section PropertyManager**, as shown in Figure 12-26, and you are prompted to specify the depth of the broken-out section.



*Figure 12-26 The Broken-out Section PropertyManager*

Select the **Preview** check box to preview the broken-out section view. The **Auto hatching** check box is used to automatically define the hatch pattern to the section drawing view of the assembly. Note that the **Auto hatching** check box is not available in the **Broken-out Section PropertyManager**, if you are creating the broken-out section view of a part. Figure 12-27 shows the preview of the broken-out section view.



*Figure 12-27 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 12-28 shows a broken-out section view with a different depth value.



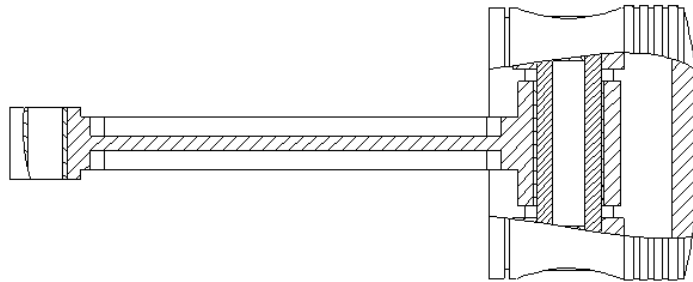


Figure 12-28 Broken-out section view

## Generating Auxiliary Views

**CommandManager:** Drawing > Auxiliary View  
**Menu:** Insert > Drawing View > Auxiliary  
**Toolbar:** Drawing > Auxiliary View



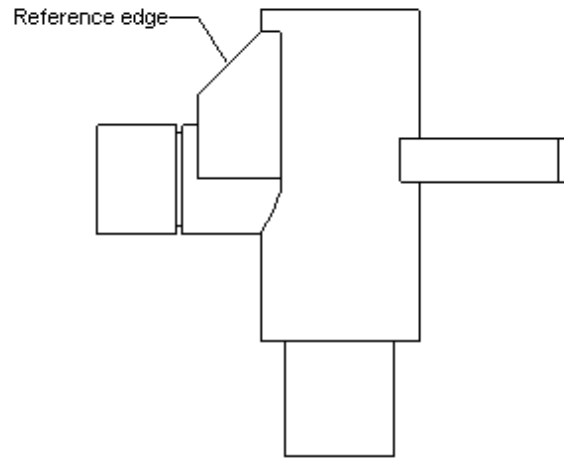
An auxiliary view is a drawing view that is generated by projecting the lines normal to a specified edge of an existing view. SolidWorks also allows you to create a line segment associated to the view that can be used to generate the auxiliary view. For this, the associated line segment needs to be created before invoking this tool.

To create an auxiliary view, choose the **Auxiliary View** button from the **Drawing CommandManager**. The **Auxiliary View PropertyManager** is displayed and you are prompted to select a reference edge to continue. Select the edge or the associated sketch; a view will be attached to the cursor and some options will be displayed in the **Auxiliary View PropertyManager**, as shown in Figure 12-29. Also, you will be prompted to specify the location to place the view.



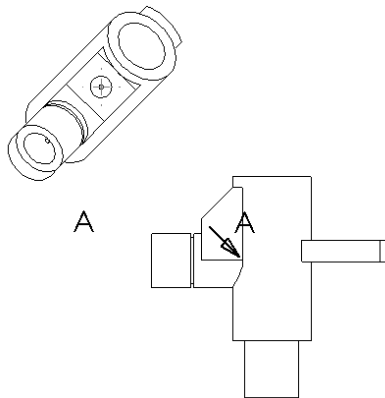
Figure 12-29 Partial view of the Auxiliary View PropertyManager

The check box available in the **Arrow** rollout is used to display the arrow of the viewing direction 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 12-30 shows the reference edge to be selected to create the auxiliary view.

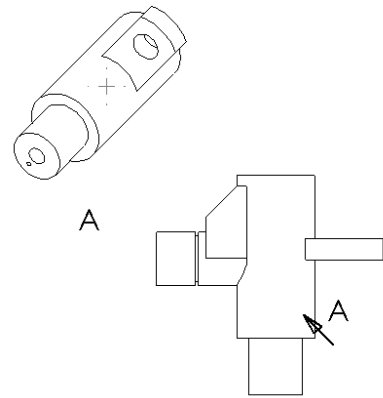


*Figure 12-30 Reference edge to be selected to create the auxiliary view*

Figure 12-31 shows the auxiliary view created with the default viewing direction. Figure 12-32 shows the auxiliary view created with the **Flip Direction** check box selected.



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



*Figure 12-32 Auxiliary view created with the **Flip Direction** check box selected*

## Generating Detail Views

**CommandManager:** Drawing > Detail View  
**Menu:** Insert > Drawing View > Detail  
**Toolbar:** Drawing > 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, activate the view from which you will generate the detail view. Next, choose the **Detail View** button from the **Drawing CommandManager**; the **Detail View PropertyManager** is displayed and you are prompted 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. If you want to use a profile other than the circle, you need to create it associated to the sketch before invoking the **Detail View** tool.

As soon as you draw the circle, the detail view is attached to the cursor and the options are displayed in the **Detail View PropertyManager**, as shown in Figure 12-33. 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.

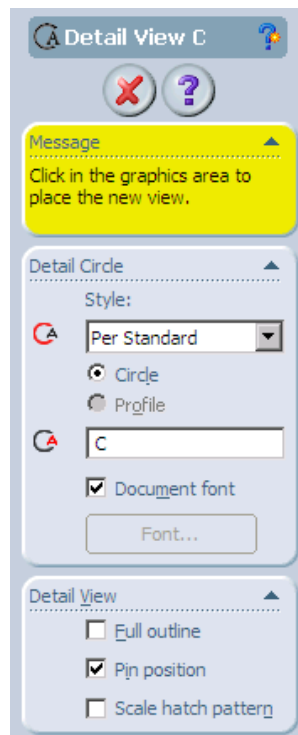


Figure 12-33 The Detail View PropertyManager

## Detail Circle

This rollout is used to define the options to display the circle of the detail view. You can also apply the leader to the detail view, using the options available in the rollout, which are discussed next.

### Style

The **Style** area has the **Style** drop-down list to specify the style of a 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 the circle. Select the **Profile** radio button, if you have already created a closed profile for defining the portion to be shown in the detail view. 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 the 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.

## Detail View

This 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.

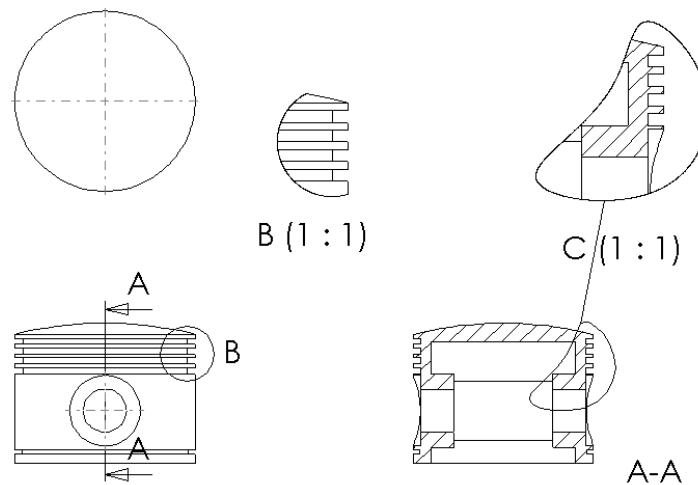
### Pin position

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

### Scale hatch pattern

While creating a detail view of a section view, the **Scale hatch pattern** check box is used to scale the hatch pattern with respect to the scale factor of the detail view.

If you create a detail view with another detail view or a crop view as the parent view, the default scale factor of the resulting detail view is twice of the immediate parent view. Figure 12-34 shows the detail view created using the **Detail View** tool.



**Figure 12-34** Detail views generated using the existing views



**Tip.** When you create a detail view, by default it is scaled as 1:1. You can define the default scale factor in the **System Options** dialog box so that the 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 its left. 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.

## Cropping Drawing Views

**CommandManager:** Drawing > Crop View  
**Menu:** Insert > Drawing View > Crop  
**Toolbar:** Drawing > Crop View



This tool is used to crop an existing view using a closed sketch associated to it. 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 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** button from the **Drawing CommandManager**. Figure 12-35 shows the closed profile used to crop the view and Figure 12-36 shows the cropped view.

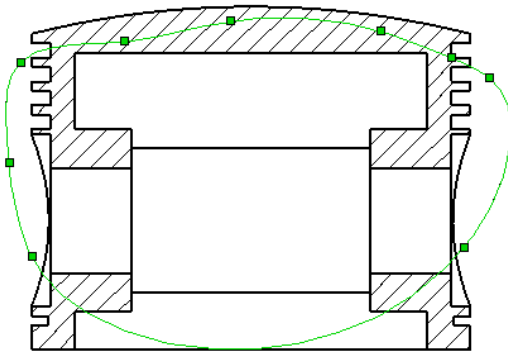


Figure 12-35 Closed profile to crop the view

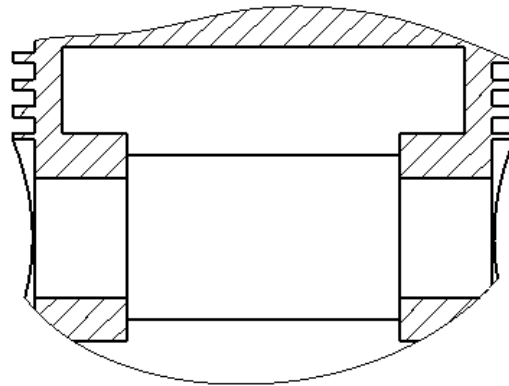


Figure 12-36 Resulting crop view



**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.

To edit the closed profile of the crop view, select the crop view and invoke the shortcut menu and choose **Crop View > Edit Crop** from it. 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** on the keyboard.

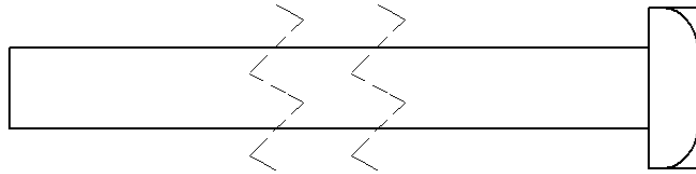
## Generating Broken View

A broken view is the one in which a portion of the drawing view is removed from in between, keeping the ends of the drawing view intact. This 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 the **Horizontal Break/Vertical Break** buttons from the **Drawings CommandManager**, 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 12-37.



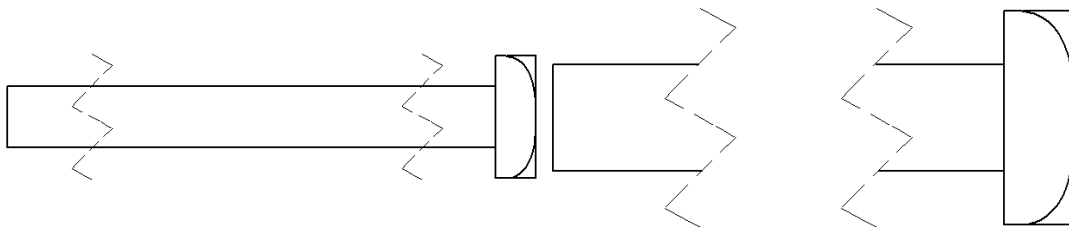
**Tip.** If you suppress the features of a model whose drawing views have been generated, the suppressed features will not be displayed in the drawing views. The feature, on suppressing, will be displayed in the drawing views.

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



**Figure 12-37** Break lines added to the view

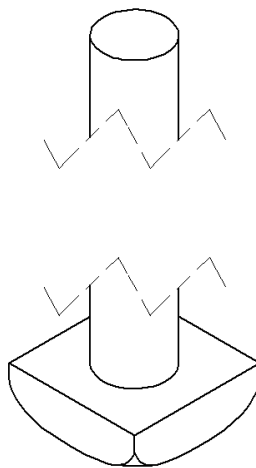
After adding the break lines, you need to move the break lines to define the gap in the broken view. Select the break lines and move them away from each other, as shown in Figure 12-38. Now, select the view and invoke the shortcut menu. Choose the **Break View** option from the shortcut menu. The broken view will be created, as shown in Figure 12-39.



**Figure 12-38** Extended gap between break lines

**Figure 12-39** Resulting 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 12-40 shows a broken isometric view.



**Figure 12-40** A broken isometric view

**Note**

Select the break line to increase or decrease the gap between the broken view. On moving the break line, the broken view is modified dynamically.

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



**Tip.** You can change the style of the break line by selecting it 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, invoke the shortcut menu and choose the **Un-Break View** option from it.

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

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

## Alternate Position View

<b>CommandManager:</b>	Drawing > Alternate Position View
<b>Menu:</b>	Insert > Drawing View > Alternate Position
<b>Toolbar:</b>	Drawing > Alternate Position View

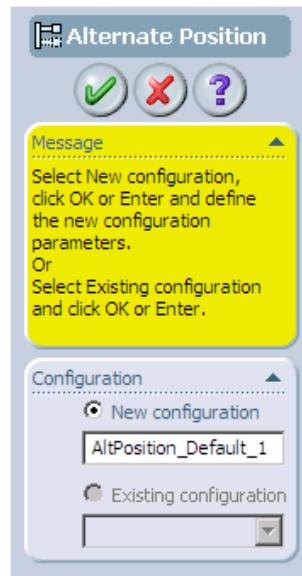


The alternate position view is used to create a view in which you can show the maximum and minimum range of the motion of an assembly. The main position is displayed with continuous lines in the drawing view, while the alternate position of the assembly is shown in the same view with dashed (phantom) lines. To create an alternate position view, activate and select the view of the assembly drawing on which you need to create the alternate position view. Choose the **Alternate Position View** button from the **Drawing CommandManager**. The **Alternate Position PropertyManager** is displayed, as shown in Figure 12-41.

The **Alternate Position PropertyManager** prompts you to select a new configuration, click OK or enter and define the new configuration parameters. If you have not created any configurations, the **New Configuration** radio button will be automatically selected to create one. Enter the name of the configuration in the edit box given below and choose the **OK** button from the **Alternate Position PropertyManager**. Choose **OK** from the **Tangent Edge Display** dialog box, if displayed.

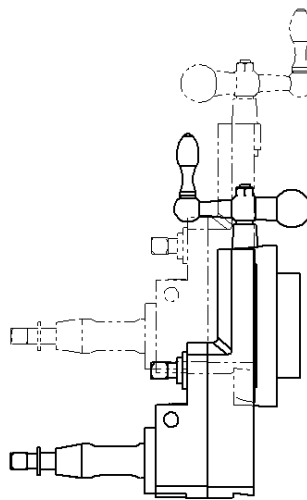
The assembly document is opened and the **Move Component PropertyManager** is displayed in the assembly document. The **Move Component PropertyManager** prompts you to move the desired components to the position to be shown in the alternate view. Note that the component





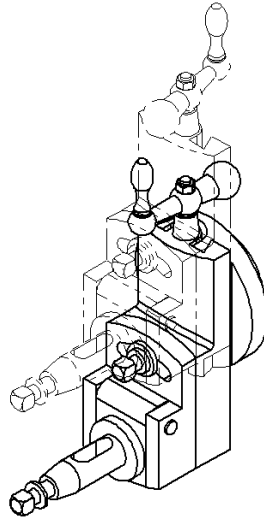
**Figure 12-41** The *Alternate Position PropertyManager*

or components that you need to move should have that particular degree of freedom free. 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 the phantom lines in the drawing view, as shown in Figure 12-42.



**Figure 12-42** Alternate position view

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 12-43 shows the alternate position view of an isometric view.



**Figure 12-43** Alternate view of an isometric view.

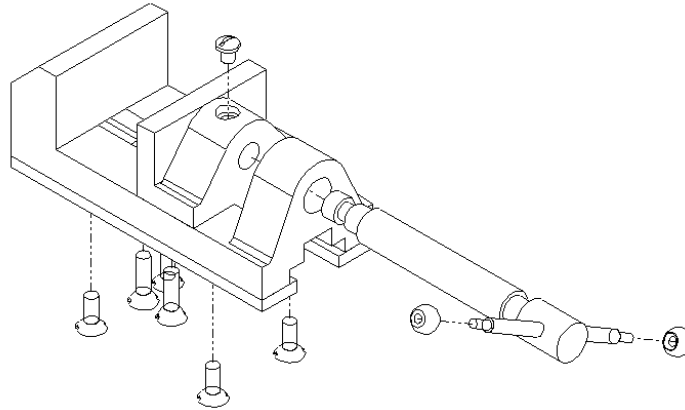


#### Note

On creating 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 has been created along with the default configuration. By default, the newly created configuration is selected. Therefore, the assembly is displayed with the components moved to their extreme positions. To switch back to the default configuration, select **Default** from the **ConfigurationManager**, invoke the shortcut menu and choose the **Show Configuration** option from it. 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 an Assembly

You can create the drawing view of the exploded state of the assembly. For this, 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, invoke the shortcut menu, and choose the **Properties** option from it. 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. Figure 12-44 shows the drawing view of the exploded state of assembly with explode lines.



**Figure 12-44** Drawing view of the exploded state with explode lines



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

To the collapse state in the drawing view, select the view and clear the **Show in exploded state** check box from the **View Properties** tab of 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. To create an empty view, choose **Insert > Drawing View > Empty** from the menu bar. An empty view is attached to the cursor. Select a point at the desired location to place an empty view. Now, select the empty view to activate and use the tools in the **Sketch CommandManager** to sketch the view.

The sketched entities will also move if you move the empty view by selecting and dragging it. This is because the sketch that you draw is associated to the empty view.

## EDITING AND MODIFYING THE DRAWING VIEWS

In SolidWorks, you can perform various kinds of editing operations and modifications on the drawing views. For example, you can change the orientation of the view, or the view scale, or you can delete the view. All these operations are discussed next.

### Changing the View Orientation

You can change the orientation of the views generated using the **Model View** or the **Predefined View** option. To change the orientation, select the view; the **Drawing View PropertyManager** is

displayed in both the cases. Double-click on the view orientation that you want as the current one, from the **View Orientation** rollout. The orientation of the selected view will be modified. Choose the **OK** button from the **Drawing View PropertyManager**.

All the derived views will also change their orientation, when you change the orientation of the parent view.

## Changing the Scale of Drawing Views

In SolidWorks, you can also change the scale of the drawing views. For doing so, select the drawing view and then select the **Use custom scale** radio button from the **Scale** rollout. Set the new scale of the drawing view in the edit boxes available below this radio button. You can also change the scale of the derived views. However, the scale of the parent view will not be changed if you change the scale of a derived view.

## Deleting Drawing 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**, invoke the shortcut menu and choose the **Delete** option from it. 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 drawing sheet and pressing the **DELETE** key on the keyboard. The **Confirm Delete** dialog box is displayed; choose the **Yes** button from this dialog box. On deleting a parent view, the projected views are not deleted. However, if you delete the parent view from which a section, auxiliary, or a detail view is generated, the name of the dependent view will also be displayed in the **Confirm Delete** dialog box. If you choose **Yes**, the dependent views will also be deleted.

## Rotating Drawing Views

SolidWorks allows you to rotate a drawing view in the 2D plane. Select the view and choose the **Rotate View** button from the View toolbar. The **Rotate Drawing View** dialog box is displayed, as shown in Figure 12-45. You can enter the value or rotation angle in this dialog box or you can also dynamically rotate the drawing view by dragging the mouse. 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.

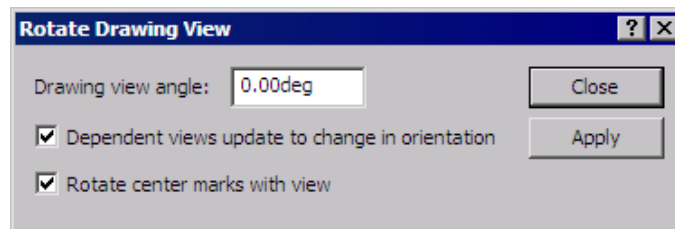


Figure 12-45 Rotate Drawing View dialog box



**Tip.** You can also copy and paste the drawing view in the drawing sheet. To do so, select the view to copy and press **CTRL+C** on the keyboard. Now, click anywhere on the drawing sheet to select the sheet and press **CTRL+V** on the keyboard to paste the drawing view.

## MODIFY HATCH PATTERN IN SECTION VIEWS

As discussed earlier, when you generate a section view of an assembly or a component, a hatch pattern is applied to the component or components. This hatch pattern is based on the material assigned to the components in the part document. If you need to modify the default hatch pattern, select the hatch pattern from the section view; the **Area Hatch/Fill PropertyManager** is displayed, as shown in Figure 12-46. The options available in this dialog box are discussed next.

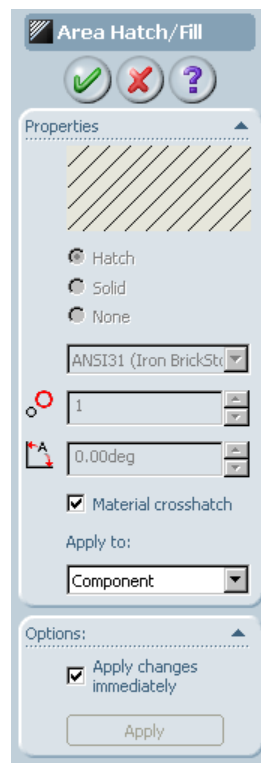


Figure 12-46 The Area Hatch/Fill PropertyManager

### Properties Rollout

The **Properties** rollout is used to define the type of hatch pattern and its properties. Some of the options in this area are not available by default. This is because by default, the material-dependent hatch pattern is applied to the component. If you want to make the other options also available, clear the **Material crosshatch** check box. The options available in this area are discussed next.

## Preview

The **Preview** area displays the preview of the hatch pattern with the current setting.

## Hatch

The **Hatch** radio button is selected to apply the standard hatch patterns to the section view. On selecting this button, some options available in the 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.

## Solid

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

## None

Select the **None** radio button, if you do not need to apply any hatch pattern in the section view.

## Hatch Pattern

The **Hatch Pattern** drop-down list is used to define the style of the standard hatch pattern you need to apply to the section view. 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.

## Hatch Pattern Scale

The **Hatch Pattern 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.

## Hatch Pattern Angle

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

## Material crosshatch

The **Material crosshatch** check box is selected to apply the hatch pattern based on the material assigned to the model. Clear this check box to change the type of the hatch pattern.

## Apply to

The **Apply to** drop-down list is used to specify whether you need to apply this hatch pattern to the selected component, region, body, or to the entire view. Note that some of these options are available only while modifying the hatch pattern of an assembly section view.

## Options Rollout

The **Options** rollout provides the **Apply changes immediately** check box, which when selected applies the changes immediately on the view and the preview is modified dynamically. If you clear this check box, you need to choose the **Apply** button, after making the changes to reflect the changes in the preview.

## TUTORIALS

### Tutorial 1

In this tutorial, you will generate the front view, top view, right view, aligned section view, detail view, and isometric view of the model created in 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 listed next.

- Copy the part document of Tutorial 2 of Chapter 7 in the folder of the current chapter.
- Open the copied part document and start a new drawing document from within the part document.
- Select the standard A4 landscape sheet format and generate the parent view using the **Model View** tool, refer to Figure 12-47.
- Generate the projected views using the **Projected View** tool, refer to Figure 12-47.
- Generate the aligned section view using the **Aligned Section View** tool, refer to Figures 12-48 and 12-49.
- Generate the detail view, refer to Figure 12-50.
- Save and close the drawing document.

### Copying and Opening the Part Document

- Create a folder with the name *c12* in the *SolidWorks* directory and copy *c07tut2.sldprt* from the *\My Document\SolidWorks\c07* folder to this folder.
- Start SolidWorks and open the part document of Tutorial 2 of Chapter 7 that you copied in the folder of the current chapter.

### Starting the New Drawing Document

As mentioned earlier, one of the latest enhancements of SolidWorks 2005 is its ability to let you start a new drawing document from within the part document. This way, the model in the part document is automatically selected and you can generate its drawing views.

- Choose the **Make Drawing from Part/Assembly** button from the **Standard** toolbar.



If you are in the practice of using the advanced form of the **New SolidWorks Document** dialog box, this dialog box will be displayed every time you choose the **Make Drawing from Part/Assembly** button.

- From the **New SolidWorks Document** dialog box, select the drawing template from the **Template** tab and choose **OK**.

Remember that if you are not using the advanced form of the **New SolidWorks Document** dialog box, this dialog box is not displayed. Instead, a new drawing document is started directly and the **Sheet Format/Size** dialog box is displayed.

- Select the **A4 - Landscape** sheet from the list box available in this dialog box and choose the

**OK** button. The new drawing document is started with the standard A4 sheet and the **Model View** tool is invoked automatically. The model of Tutorial 2 of Chapter 7 is selected by default for generating the drawing views. A view is also attached to the cursor.

### Generating the Parent View and the Projected Views

Before you proceed with generating the drawing views, you need to confirm whether the projection type for the current sheet is set to the third angle.

1. Press the ESC key to exit the **Model View** tool. Select **Sheet1** from the **FeatureManager Design Tree** and then right-click on it. Choose the **Properties** option from the shortcut menu to display the **Sheet Properties** dialog box.
2. Select the **Third angle** radio button from the **Type of projection** area, if it is not already selected. Close this dialog box.



#### Note

*This option can also be selected in the **Sheet Format/Size** dialog box, which is displayed when you start a new drawing document.*

3. Now, choose the **Model View** button from the **CommandManager** to invoke the **Model View PropertyManager**.
4. Select the **Front** option from the list box available in the **Orientation** rollout of the **Model View PropertyManager**, if not already selected. Also, select the **Preview** check box; the preview of the front view of the model is displayed.
5. Select the **Auto-start projected view** check box in the **Options** rollout, if it is not selected. Move the cursor to the middle left of the drawing sheet just above the title block and specify a point at this location to place the front view, refer to Figure 12-47.

Specify the location of the view; the front view will be generated and placed at this location. This view is generated at a 1:1 scale. Note that because you selected the option to start projected views immediately after generating the front view, the **Projected View PropertyManager** is invoked and the preview of the projected view is attached to the cursor. This view is being generated by referencing the front view as the parent view.

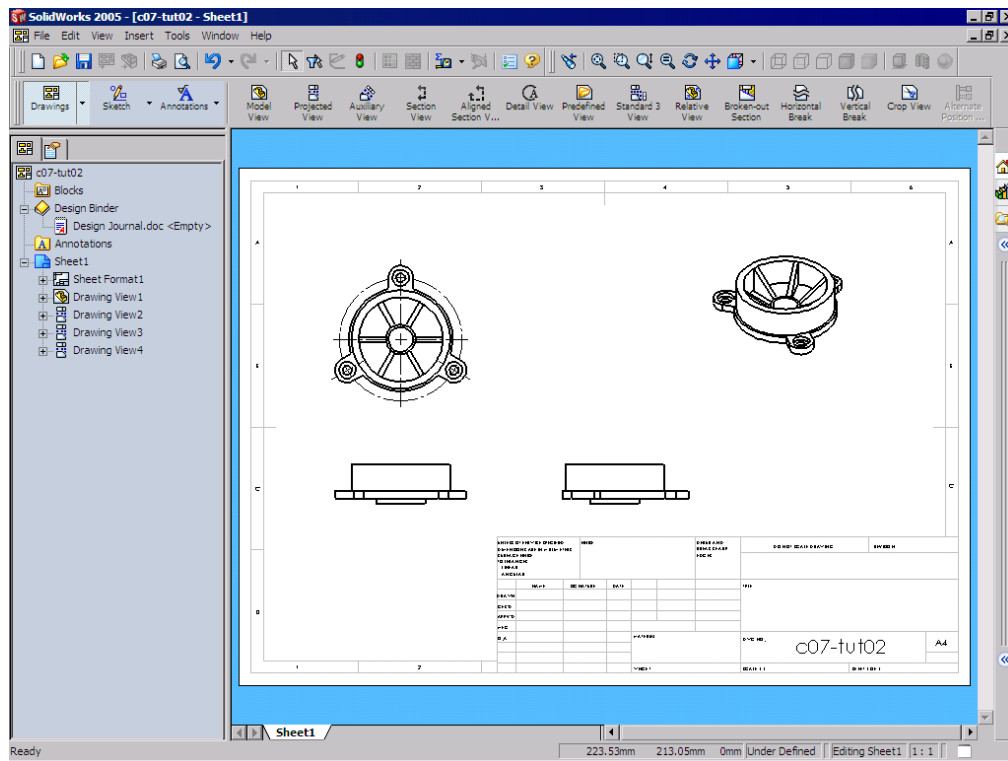
6. Move the cursor above the front view and specify a point to place the top view, refer to Figure 12-47. The top view of the model is generated and the preview of another projected view with the front view as the base view, is attached to the cursor.
7. Move the cursor to the right of the front view and place the right view, refer to Figure 12-47.
8. Similarly, move the cursor horizontally toward the right and then move it upward, the preview of the isometric view is displayed. Specify a point to place the isometric view. Right-click to exit the **Projected View PropertyManager**.

The current location of the isometric is such that it will interfere with the aligned section



view that you need to place next. Therefore, you need to move the isometric view close to the top right corner of the drawing sheet.

9. Move the cursor over the isometric view; the bounding box of the view is displayed in red.
10. Click to select the view; the border of the view is displayed in green.
11. Move the cursor on one of the border lines of the view; the cursor changes to the move cursor.
12. Press and hold the left mouse button and drag the view close to the upper right corner of the drawing sheet. The drawing sheet, after generating and moving the drawing view, is shown in Figure 12-47.



**Figure 12-47** Drawing sheet after generating the front, top, right, and isometric views



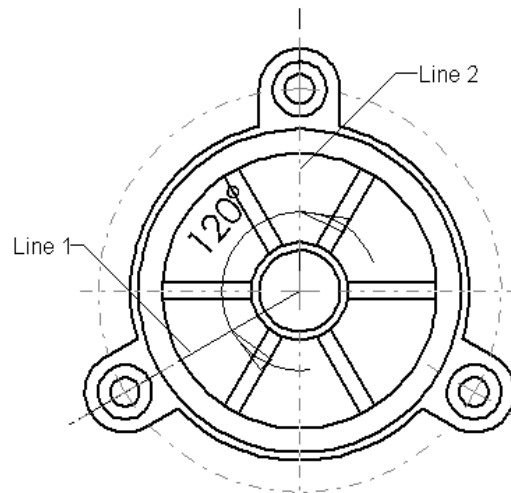
**Tip.** You can turn off the origins displayed in the drawing views by using the **View** menu.

Center marks are automatically created in the drawing views of the circular features in the model.

### Generating the Aligned Section View

Next, you need to generate the aligned section view. The line segments, used to generate this view, will be drawn before invoking the **Aligned Section View** tool. Remember that the view is projected normal to the line selected last, irrespective of the last line drawn. This means that you can draw any line first. In this tutorial, you will first select the inclined line and then the vertical line.

1. Click on the top view to activate the view.
2. Choose the **Sketch** button from the **CommandManager** to invoke the sketching tools. Using the **Line** tool, draw the sketch and apply the relations and dimensions to the sketch, as shown in Figure 12-48.



**Figure 12-48** Sketch to be used as section sketch for the aligned section view

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

Next, you need to select the lines to generate the aligned section view. Note that the vertical line should be selected last to generate the view normal to this line. It will be difficult to select the line because the vertical line coincides with the center marks. You will use the **Select Other** option to select this line.

4. Select the inclined line. Make sure you do not select any segment of the center mark.
5. Next, press and hold the CTRL key down and move the cursor to the vertical line. Right-click on the vertical line and choose the **Select Other** option from the shortcut menu. The **Select Other** list box is displayed.

6. Select the **Line** option from the **Select Other** list box. Also, because the CTRL key was pressed, the inclined line will still be in the current selection set.
7. Now, choose the **Aligned Section View** button from the **Drawing CommandManager**. The aligned section view will be attached to the cursor.



The view generated is normal to the vertical line. If the direction of viewing the aligned section view is reversed, you need to flip it after placing the view.

8. Move the cursor to the right of the top view and place the aligned section view. The **Section View PropertyManager** is displayed. Select the **Flip direction** check, if the direction of viewing is not as required. Click anywhere on the sheet to exit the **PropertyManager**. The sheet, after generating the aligned section view, is shown in Figure 12-49.

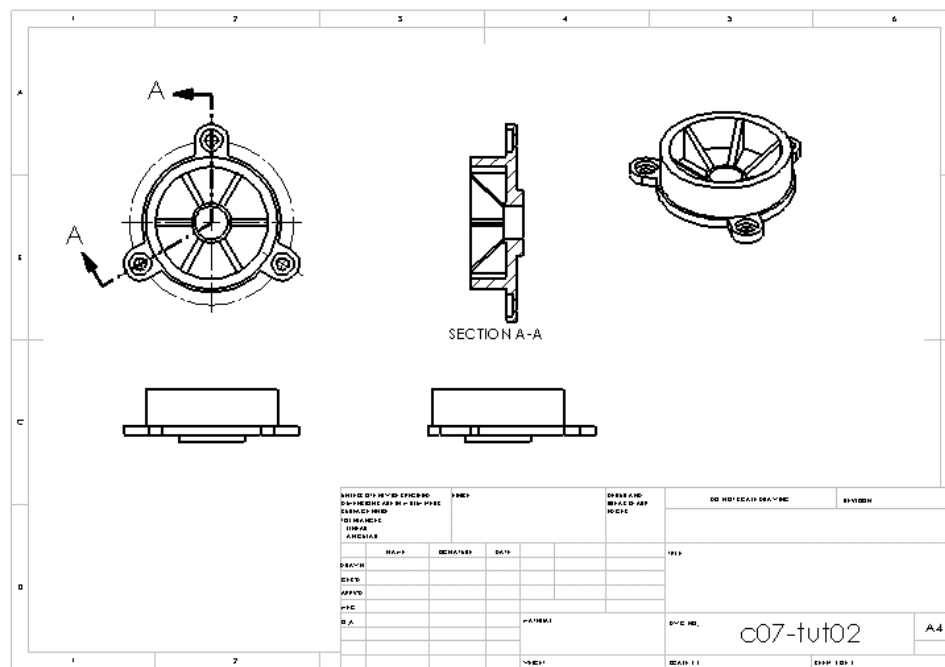


Figure 12-49 Sheet after generating the aligned section view

### Modifying the Hatch Pattern of the Aligned Section View

The gap between the hatching lines in the aligned section view is large. Therefore, you need to modify the spacing.

1. Select the hatch pattern from one of the sections in the aligned section view; the **Area Hatch/Fill PropertyManager** is displayed.
2. Clear the **Material crosshatch** check box; the **Hatch Pattern**, **Hatch Pattern Scale**, and

**Hatch Pattern Angle** options are available. Set the value of the **Hatch Pattern 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 doing so, you need to activate the view from which you will drive the detail view.

1. Activate the top view and choose the **Detail View** button from the **Drawing CommandManager**; the **Detail View PropertyManager** is displayed and you are prompted to sketch a circle to continue to view the creation. Also, the cursor is replaced by the circle cursor.
2. Draw a small circle on the right circular feature of the model in the top view, refer to Figure 12-50. As you draw the circle, the detail view is attached to the cursor.
4. Place the view on the right side of the drawing sheet above the title block, refer to Figure 12-50.
5. Set the value of the scale factor of the detail view to **3:1** and choose the **OK** button from the **Detail View PropertyManager**.



You may need to move the drawing view and its label so that the view is not overlapping the title block. Figure 12-50 shows the final drawing sheet, after generating the detail view from the top view.

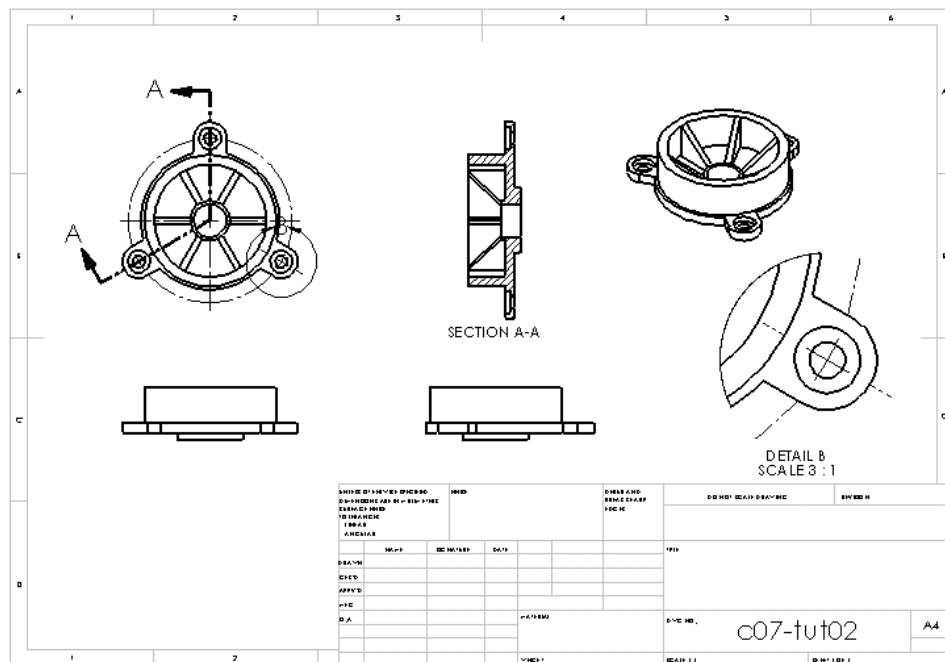


Figure 12-50 The detail view derived from the top view

### Saving the Drawing

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

*\My Documents\SolidWorks\c12\c12tut1.slddraw*

2. Choose **File > Close** to close this document. Also, close the part document of Tutorial 2.

## 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, sectioned 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 listed next.

- a. Copy the folder of the Bench Vice assembly from Chapter 10 to the *c12* folder.
- b. Create the exploded state of the Bench Vice assembly, refer to Figure 12-51.
- c. Start a new drawing document from within the assembly document using A4 landscape sheet format. Generate the top view using the **Model View** tool, refer to Figure 12-52.
- d. Generate the section view using the **Section View** tool, refer to Figure 12-53.
- e. Generate the right view using the **Projected View** tool, refer to Figure 12-54.
- f. Generate the isometric view and change the state of the isometric view to the exploded state, refer to Figure 12-55.
- g. Save and close the drawing and assembly documents.

### Copying the Folder of the Bench Vice Assembly

First, you need to copy the folder of the Bench Vice assembly to *c12* folder.

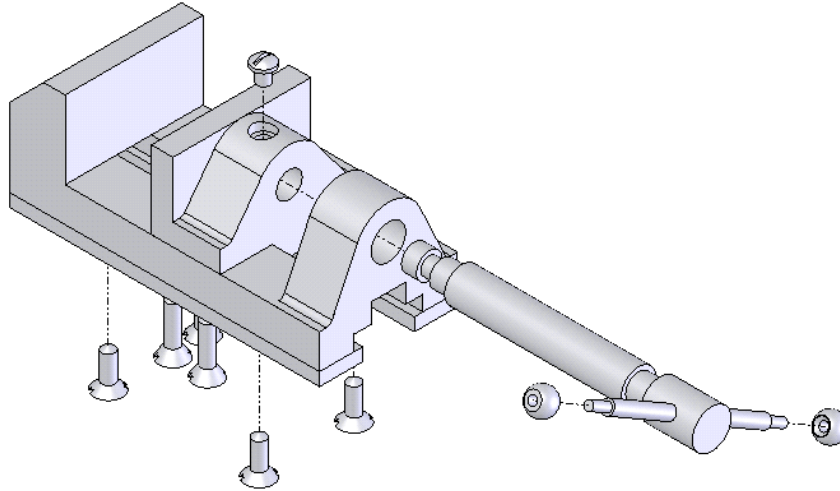
1. Copy the folder of the Bench Vice assembly from the *\My Document\SolidWorks\c10* folder to the *c12* folder.

### Creating the Exploded View of the Assembly

Before proceeding further to generate 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 12-51.

It is recommended that whenever you create an exploded state of an assembly, you must revert to the collapsed state. On saving the assembly in the exploded state, whenever you generate the drawing views of the assembly, it will generate views with the exploded state.



*Figure 12-51 Exploded view of the assembly with explode lines*

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

### Starting a New Drawing Document From Within the Assembly Document

As mentioned earlier, you can also start a drawing document from within the assembly document.

1. Choose the **Make Drawing from Part/Assembly** button from the **Standard** toolbar. Start a new drawing document from the **Templates** tab of the **New SolidWorks Document** dialog box, if it appears.



A new SolidWorks drawing document is started and the **Sheet Format/Size** dialog box is displayed.

2. Select the **A4 - Landscape** sheet and make sure that the projection type for the current sheet is set to the third angle. Choose the **OK** button to close this dialog box.

The **Model View PropertyManager** is automatically invoked and you are prompted to select a named view and place the drawing view.

### Generating the Top View

1. Select the **Top** option from the list box available in the **Orientation** rollout of the **Model View PropertyManager**.

2. Select the **Use custom scale** radio button from the **Scale** rollout.
3. Set the value of the scale factor to **1:2** and then select the **Preview** check box from the **Orientation** rollout. The preview of the top view of assembly is displayed.
4. Make sure the **Auto-start projected view** check box in the **Options** rollout is cleared.
5. Place the view close to the top left corner of the drawing sheet, refer to Figure 12-52. Click anywhere on the sheet to exit the **PropertyManager**.

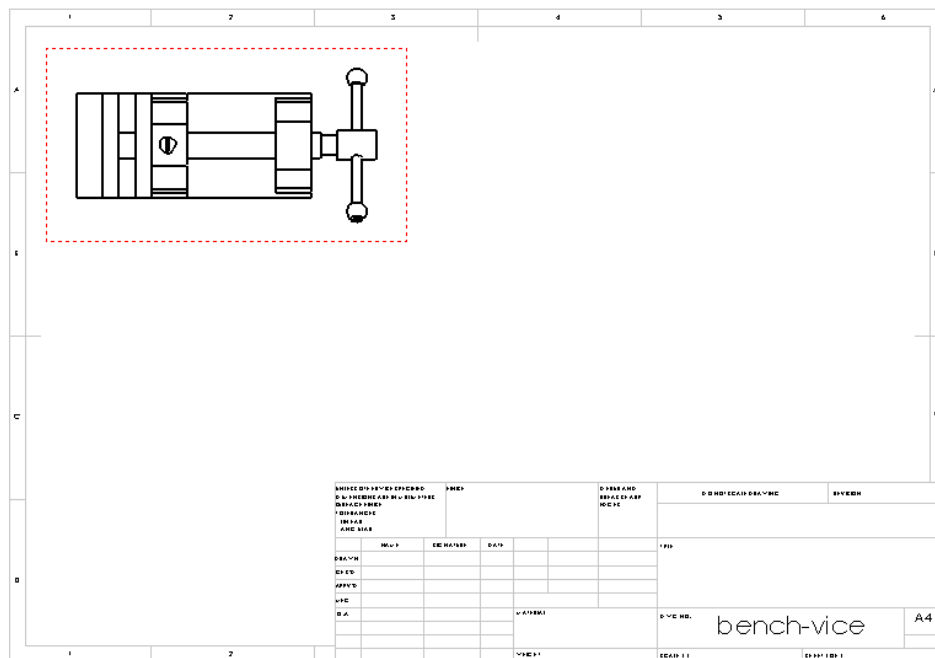


Figure 12-52 Top view generated using the **Model View** tool

## Creating the Sectioned Front View

Next, you need to generate the sectioned front view that is derived from the top view.

1. Activate the top view and choose the **Section View** button from the **Drawings CommandManager**.



The **Section View PropertyManager** is displayed and you are prompted to sketch a line in order to continue to view the creation. The cursor will be replaced by the line cursor.

2. Draw a horizontal line such that it passes through the center of the Bench Vice assembly.

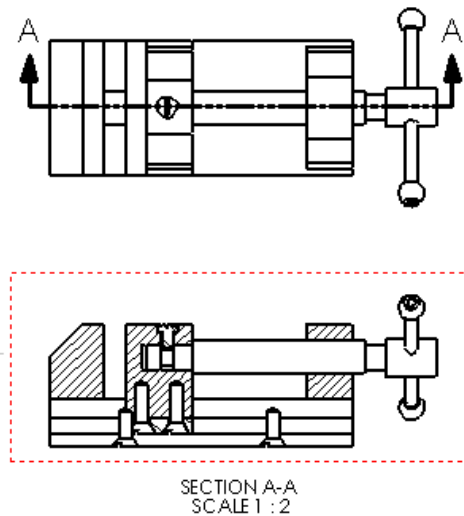
On specifying the endpoint of the line, the **Section View** dialog box is displayed. This dialog box is used to exclude the components from the section cut.

3. Click on the + sign located on the left of the **FeatureManager Design Tree** and continue expanding it until Bench Vice assembly name is no longer displayed. Next, expand the assembly.
4. Select Screw Bar, Bar Globes, Jaw Screw, Oval Fillister, Set Screw1, and Set Screw2. Next, activate the **Section View** dialog box and select the **Don't cut all instances** check box individually for all the selected items.
5. 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 in the drawing sheet as you move the cursor up and down.

If the direction of viewing of the section view is not what is required, flip the direction by selecting the **Flip direction** check box from the **Section Line** rollout.

6. Place the section view below the top view. Click anywhere on the sheet to exit the **PropertyManager**.
7. Modify the hatch scale for the components. Figure 12-53 shows the section view generated using the **Section View** tool, after modifying the hatch scale.

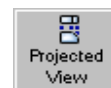


*Figure 12-53 Section view generated using the Section View tool*

### Generating the Right-Side View

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

1. Select the sectioned front view and invoke the **Projected View** tool. The **Projected View PropertyManager** is displayed and a projected view is attached to the cursor.





2. Move the cursor to the right of the sectioned front view and place the view on the right of the sectioned front view.
3. Choose **OK** from the **PropertyManager**. The sheet after generating the projected view is shown in Figure 12-54.

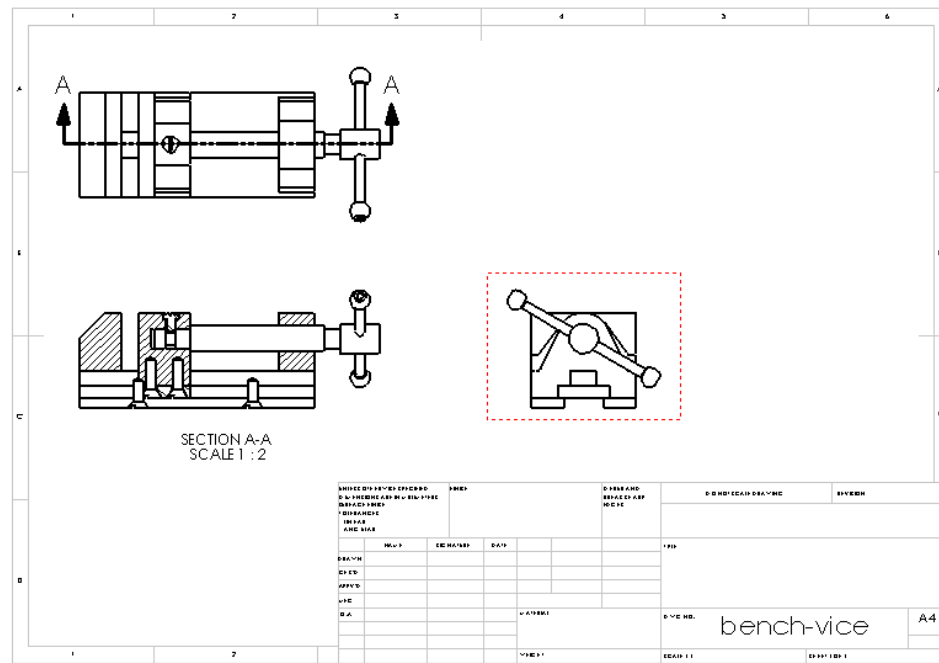
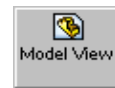


Figure 12-54 Right-side view generated using the *Projected View* tool

### Creating the Isometric View of the Assembly in the Exploded State

The last view to be generated is the isometric view of the assembly in the exploded state.

1. Using the **Model View** tool, generate the isometric view and place the view close to the upper right corner of the drawing sheet. Choose **Yes** from the **SolidWorks** information box.
2. Set the scale factor of the drawing view to **1:2**.
3. Right-click on the view to 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 **View Properties** tab of the **Drawing View Properties** dialog box and choose the **OK** button.
5. Move the views to place all of them in the drawing sheet. Figure 12-55 shows the final drawing sheet after generating all the drawing views.



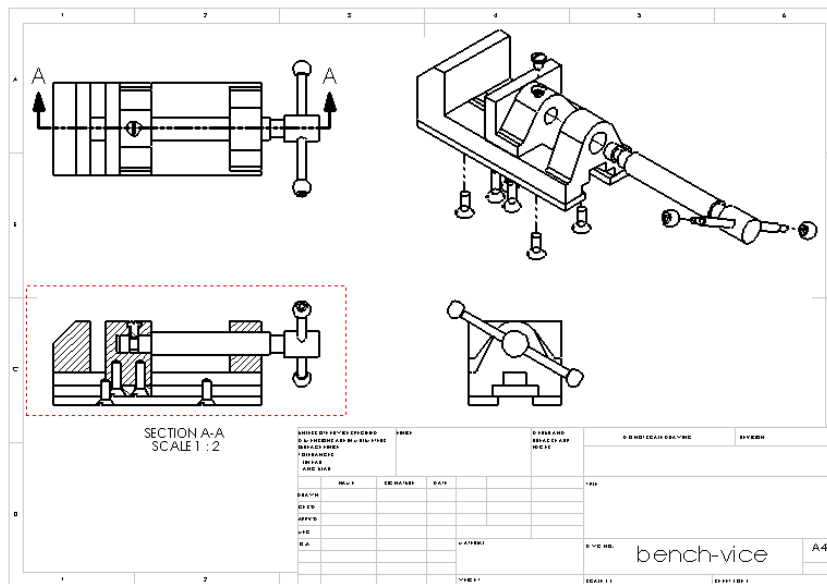


Figure 12-55 Drawing sheet after generating all the views

### Saving the Drawing

Next, you need to save the drawing document.

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

\\My Documents\\SolidWorks\\c12\\c12tut2.slddrw

2. Close the drawing and assembly documents.

## SELF-EVALUATION TEST

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

1. The **Standard sheet size** radio button is selected by default in the **Sheet Format/Size** dialog box. (T/F)
2. The **Display sheet format** check box is selected if you want to use the empty sheet without any margin lines or a title block. (T/F)
3. The **Relative View** tool is used to generate an orthographic view by defining the orientation of the view, using reference planes or 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 start a new drawing document from within the part document, choose the \_\_\_\_\_ button from the **Standard** toolbar.
8. The \_\_\_\_\_ check box available in the **Detail View** rollout of the **Detail View PropertyManager** is used to display the complete outline of the closed profile in the detail view.
9. To change the scale of the drawing views, select the drawing view and select the \_\_\_\_\_ radio button from the **Scale** rollout.
10. To rotate 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 **Drawings CommandManager** to create an alternate position view.
2. The \_\_\_\_\_ dialog box is used to modify the hatch pattern of a section view.
3. The \_\_\_\_\_ check box needs to be cleared to modify the scale of 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. By default, the detail view boundary is in which shape?
  - (a) Circle
  - (b) Ellipse
  - (c) Rectangle
  - (d) None

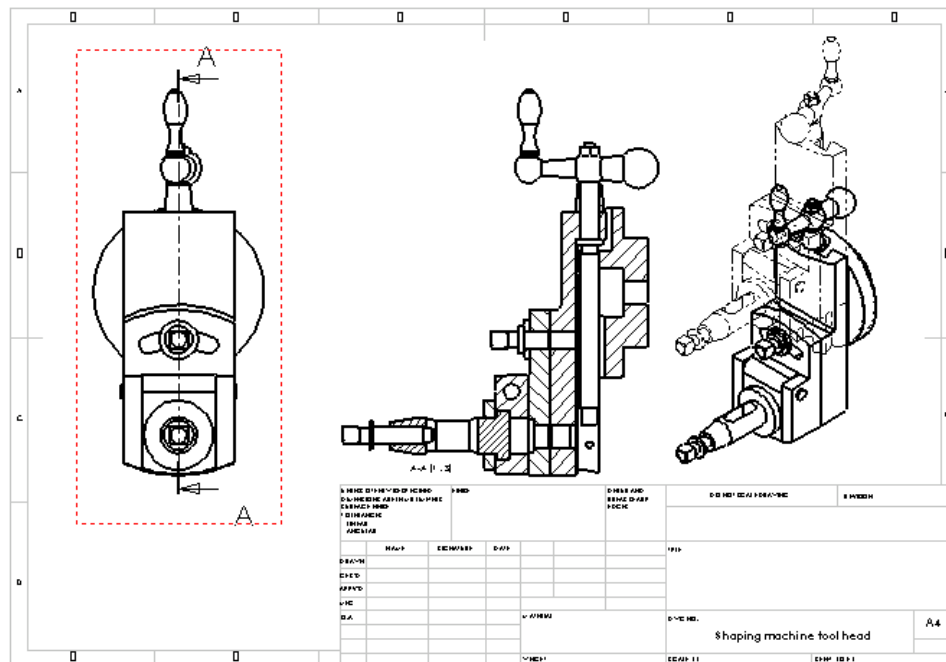
8. In which edit box is the name of the auxiliary view specified?
- (a) **Label** (b) **Arrow**  
(c) **Name** (d) **Detail view label**
9. From which rollout can you select the view orientation in the **Named View PropertyManager**?
- (a) **View Orientation** (b) **Define View**  
(c) **Specify View** (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 12-56.

**(Expected time: 30 min)**



**Figure 12-56** Views of Exercise 1

## Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. T, 5. F, 6. Interactive drafting, 7. Make Drawing from Model/Assembly, 8. Full outline, 9. Use custom scale, 10. Rotate View