

Chapter 12

Generating, Editing, and Modifying the Drawing Views

Learning Objectives

After completing this chapter, you will be able to:

- Create and retrieve the drawing sheet formats.
- Generate different drawing views of an existing part.
- Edit the existing drawing views and parameters associated with the views.
- Modify the existing drawing views.

THE DRAWING MODE

In the earlier chapters, you learned about creating the parts in the **Part** mode and assembling different parts in the **Assembly** mode. In this chapter, you will learn to generate the drawing views of the parts and assemblies created earlier. Drawing views are generated in the **Drawing** mode. One of the major advantages of working with this software package is its bidirectional associative nature. This is the property that ensures that any modifications made in the model in the **Part** mode updates its drawing views and vice-versa. In Pro/ENGINEER Wildfire 4.0, there are two types of drafting methods: Interactive drafting and Generative drafting. In this chapter, you will learn Generative drafting. To generate the drawing views, first you need to start a new file in the **Drawing** mode of Pro/ENGINEER Wildfire 4.0.

Choose the **New** button from the **File** toolbar to display the **New** dialog box. Select the **Drawing** radio button in the **New** dialog box, as shown in Figure 12-1.

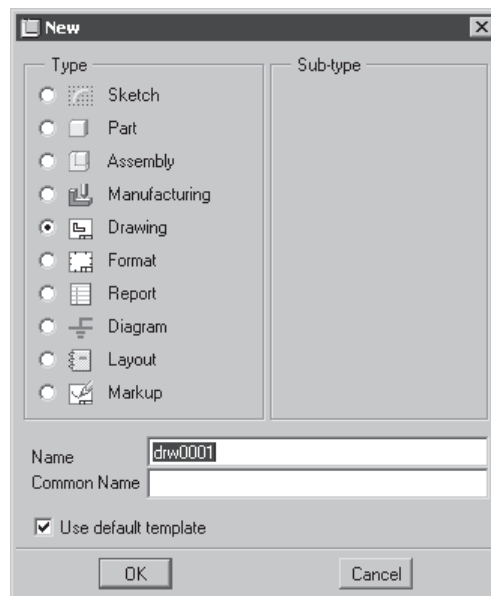


Figure 12-1 The New dialog box

Specify the name of the drawing in the **Name** edit box and then choose **OK** to display the **New Drawing** dialog box, as shown in Figure 12-2.

New Drawing Dialog Box

The **New Drawing** dialog box is used to specify the template that will be used while starting a new file in the **Drawing** mode. The options available in this dialog box are discussed next.

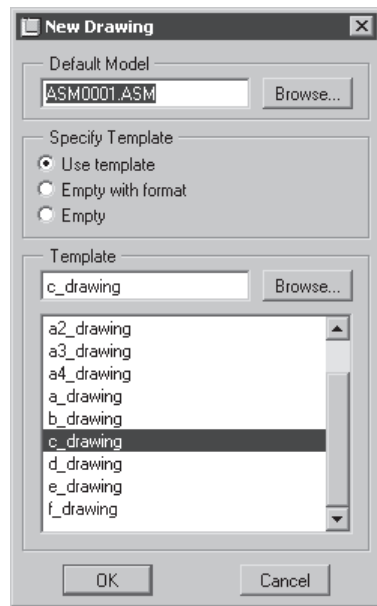
Default Model Area

The **Default Model** area is used to specify the name of the model whose drawing views you want to generate. You can specify the name of the model in the **Name** edit box or select the model using the **Open** dialog box that will be displayed when you choose the **Browse** button.

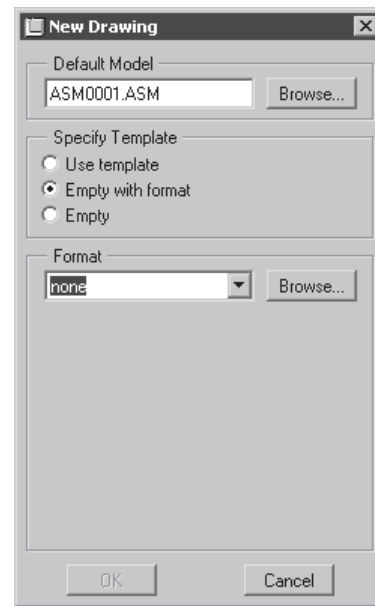
If a part or assembly file is already opened in the current session then the name of that model will be displayed by default in the **Name** edit box of the **Default Model** area.

Specify Template Area

The **Specify Template** area is used to specify whether you want to use an empty sheet, predefined formats, or the default template available in Pro/ENGINEER. There are three radio buttons in this area. The **Use template** radio button is selected by default. When the **Empty with format** radio button is selected, the dialog box get changed, as shown in Figure 12-3.



*Figure 12-2 The New Drawing dialog box with the **Use template** radio button selected*



*Figure 12-3 The New Drawing dialog box with the **Empty with format** radio button selected*

When you select the **Empty** radio button, the **New Drawing** dialog box is modified, as shown in Figure 12-4.

Orientation Area

The **Orientation** area is available only when you select the **Empty** radio button from the **Specify Template** area. The buttons in this area are used to specify the orientation of the sheet. You can select a standard size sheet with a portrait or a landscape orientation using the **Portrait** or the **Landscape** button. You can also specify a sheet with the user-defined size by choosing the **Variable** button.

Size Area

The options in the **Size** area are used to set the size and the units of the sheet. The **Size** area is available only when you select the **Empty** radio button from the **Specify Template** area. The options in this area are discussed next.

Standard Size

The **Standard Size** drop-down list is used to select a drawing sheet of standard size. This drop-down list is available only when you choose the **Portrait** or **Landscape** button from the **Orientation** area.

Inches and Millimeters Radio buttons

These radio buttons are selected to set the standards for the user-defined sheets. You can set the size of the sheet in inches or in millimeters. These buttons are available only when you choose the **Variable** button from the **Orientation** area.

Width and Height edit boxes

These edit boxes are used to specify the width and the height of the user-defined drawing sheets. These edit boxes are available only when you choose the **Variable** button from the **Orientation** area.

Format Area

The **Format** area is available only when you select the **Empty with format** radio button from the **Specify Template** area.

Format

The **Format** drop-down list is used to select the available formats.

Browse

The **Browse** button is chosen to display the **Open** dialog box for retrieving the drawing formats. By default, there are only eight standard system formats that can be retrieved. However, you can create your own user-defined formats that can be retrieved later.

Choose **OK** from the **New Drawing** dialog box to proceed to the **Drawing** mode. A drawing sheet of the specified size and orientation will be placed on which you can now generate the drawing views.

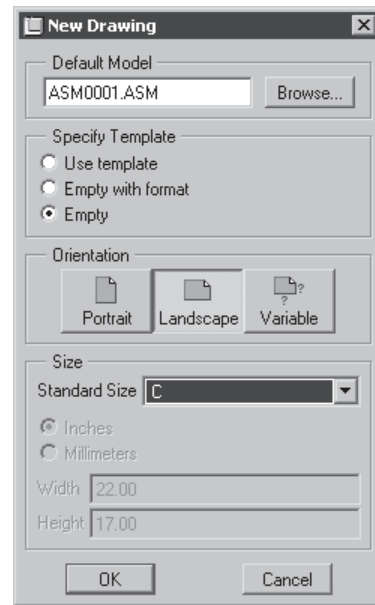


Figure 12-4 The **New Drawing** dialog box with the **Empty** radio button selected

GENERATING DRAWING VIEWS

In Pro/ENGINEER Wildfire 4.0, the first view that you need to generate is the general view. This view mostly acts as the parent view for the remaining views. The method of generating various views in Pro/E Wildfire 4.0 is discussed next.

Generating the General View

The **General** view is the first view that is generated on the drawing sheet. This view can be the top, front, right-side, left-side, bottom, back, trimetric, isometric view, or any user-defined view of the model. To generate the general view, choose the **General**



button from the **Top Toolchest**; the **Select Combined State** dialog box will be displayed, as shown in Figure 12-5. This dialog box will be displayed only if the selected model is an assembly model. If the selected model is a part model, then this dialog box will not be displayed and directly the **Drawing View** dialog box will be displayed, as shown in Figure 12-6. In the **Select Combined State** dialog box, if you select the **No Combined State** option from the **Combined state names** collector and then choose the **OK** button, you will be prompted to select a center point to place the drawing view in the drawing sheet. Select the center point; the preview of the drawing with unexploded view in the default orientation will be displayed in the drawing sheet and also the **Drawing View** dialog box will be displayed, as shown in Figure 12-6.

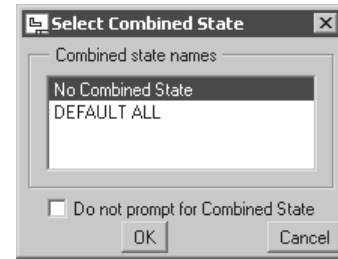


Figure 12-5 The *Select Combined State* dialog box

If you select the **DEFAULT ALL** option from the **Combined state names** collector of the **Select Combined State** dialog box and choose the **OK** button, you will be prompted to select a center point to place the drawing view in the drawing sheet. Select the center point; the preview of the drawing, with the exploded view in the default orientation will be displayed in the drawing sheet and also the **Drawing View** dialog box will be displayed, as shown in Figure 12-6.

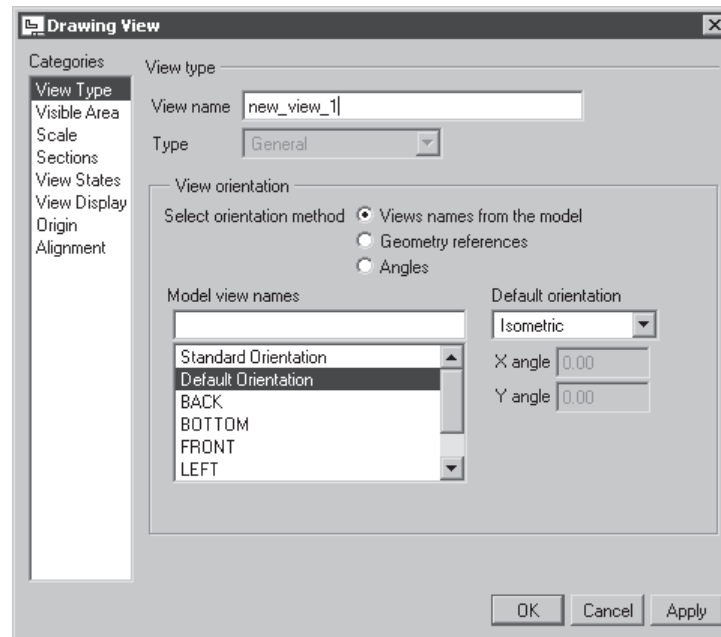


Figure 12-6 The *Drawing View* dialog box



Note

If you do not want to be prompted to select a combination state, select the **Do not prompt for Combined State** check box from the **Select Combined State** dialog box.



Tip: If you have not specified any model in the **Default Model** area of the **New Drawing** dialog box and choose the **General** button from the **Top Toolchest**, the **Open** dialog box will be displayed. You can select the model from the **Open** dialog box.

Pro/ENGINEER automatically selects the model in the **Default Area** of the **New Drawing** dialog box. This model is selected from the present session of Pro/ENGINEER. If the model selected automatically is not the required model, then change the model by choosing the **Browse** button in the **New Drawing** dialog box.

The **Drawing View** dialog box contains the options that are required to generate a general view. The **Categories** list box in the dialog box lists all parameters that are required to generate a drawing view in Pro/ENGINEER Wildfire 4.0. The **View Type** option is selected by default in the **Categories** list box and its related options are displayed on the right side of the **Categories** list box. These parameters/options are discussed later in this chapter.

The procedure to generate a general view is given below:

1. Choose the **General** button from the **Top Toolchest** or choose **Insert > Drawing View > General** from the menu bar; you will be prompted to select a center point.



Note

If the model, that you have selected, is an assembly model, then in such cases, after choosing the **General** button from the **Top Toolchest**, the **Select Combined State** dialog box will be displayed, as discussed earlier.

2. As soon as you select the center point, the **Drawing View** dialog box will be displayed and the default view of the model will be displayed on the drawing sheet. In the **Type** drop-down list, the **General** option is selected by default and in the **View orientation** area, the **Views names from the model** radio button is selected by default. This radio button allows you to select the standard orientations of the model from the **Model view names** list box. If you select the **Geometry references** radio button, you can select the planar faces or the datum planes to orient in a particular orientation.
3. Select the view that you need to generate as the general view from the list provided below the **Model view names** list box.
4. After selecting the required options, choose the **Apply** button and then choose the **Close** button from the **Drawing View** dialog box.



Tip: To set the model orientation using the **Geometry references** radio button, select an option from the **Reference 1** drop-down list and then from the model, select the face or the datum plane that you need to see from that selected view. Similarly, select an option from the **Reference 2** drop-down list and then select a datum plane or a face from the model. For example, if you select **Top** from the **Reference 1** drop-down list, then from the model, you need to select the face or the datum plane that you need to keep in the top orientation in the drawing view.

Generating the Projection View

The projection views are the orthographic views generated by projecting lines normal from an existing view, as shown in Figure 12-7. Before generating a projected view, you need to make sure that at least one parent view is already present on the drawing sheet. To generate a projected view, choose **Insert > Drawing View > Projection** from the menu bar. Now, move the cursor to a location where you need to place the view and specify a point on this location. Depending on the point selected to place the view, the resulting view will be top, front, right-side, or left-side view.

The scale factor of these views will be the same as that of the parent view from which they are projected. If there exists more than one view that can be the parent view of the projection view, then you will be prompted to select the parent view for the new view.

Generating the Detailed View

Detailed views are used to provide the enlarged view of a particular portion of an existing view. To generate a detailed view, choose **Insert > Drawing View > Detailed** from the menu bar. You are prompted to specify the center point for detail on an existing view. Define a center point on the view whose detail needs to be generated. Next, you are prompted to sketch a spline without intersecting other splines to define an outline. Draw a closed spline to define the outline of the detail view. Next, you need to specify the placement point for placing the detail view. You can change the name, scale, or the reference point on the parent view and can also select the boundary type of the detailed view using the options from the **Drawing View** dialog box, which will be displayed when you double-click on the detailed view.

Figure 12-7 shows a drawing sheet with various drawing views.

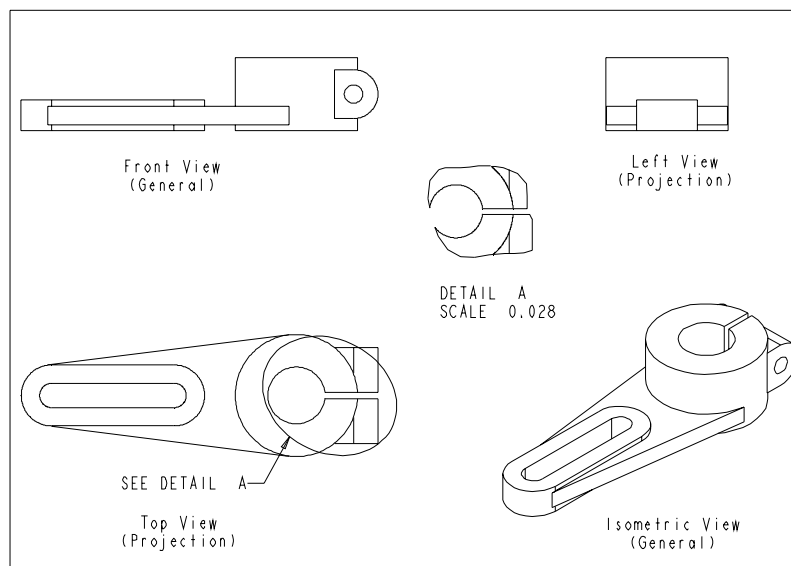


Figure 12-7 Sheet with various drawing views

**Note**

After generating a view, you need to choose the **Current Sheet** button from the **Top Toolchest**. This is because sometimes the scale or the other information related to the generated view is not displayed along with it.

Generating the Auxiliary View

The auxiliary views are generated by projecting normal lines from a specified edge, axis, or datum plane of an existing view. The view scale will be the same as that of the parent view. To generate an auxiliary view, choose **Insert > Drawing View > Auxiliary** from the menu bar; you will be prompted to select an edge, axis, or datum plane from the front surface of the main view. Select the edge or surface normal to which you need to place the generated view. The preview of the drawing view is attached to the cursor. Select a point on the drawing sheet to place the view on the drawing sheet. Figure 12-8 shows the edge to be selected as reference to generate the drawing view and the resulting auxiliary view.

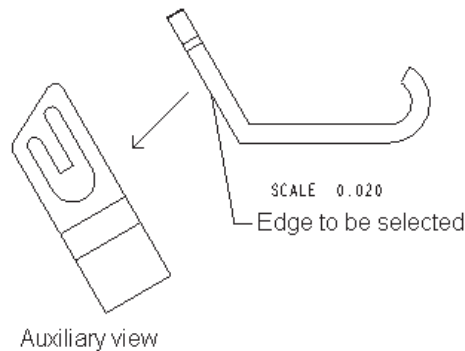


Figure 12-8 Resulting Auxiliary view

Generating the Revolved Section View

A revolved section view is the view that is generated from an existing view by revolving the section through an angle of 90-degree about the cutting plane and then projecting it along the length. Remember that the cutting plane of the revolved section view is normal to the parent view. The procedure to generate a revolved section view is discussed next.

1. Choose **Insert > Drawing View > Revolved** from the menu bar; you are prompted to select a parent view for the revolved section.
2. Select the view from the drawing sheet that will be defined as the parent view for generating the revolved view.
3. Next, you need to select a center point on the drawing sheet to place the view. Select a point anywhere on the drawing sheet but the resulting view will be placed in-line with the section plane. The **Drawing View** dialog box will be displayed. The **Create New** option is selected by default in the **Revolved view properties** area of the **Drawing View**

dialog box. The **XSEC CREATE** menu is also displayed on the lower right corner of the screen.

**Note**

The **Create New** option is selected by default in the **Revolved view properties** area of the **Drawing View** dialog box, only if there is no previously created revolved section present.

4. Choose **Planar > Single > Done** from the **XSEC CREATE** menu. You need to specify the name of the cross-section in the **Message Input Window** that will be displayed. The **SETUP PLANE** menu will be displayed and you are prompted to select a planar surface or datum plane.
5. Select the plane along which the view will be sectioned, as shown in Figure 12-9. The resulting section view will be placed on the point specified earlier.
6. Choose the **Apply** button and then choose the **Close** button to exit the **Drawing View** dialog box. The resulting sectioned view is shown in Figure 12-10.

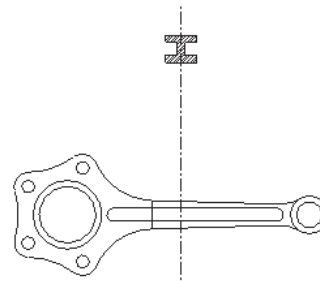
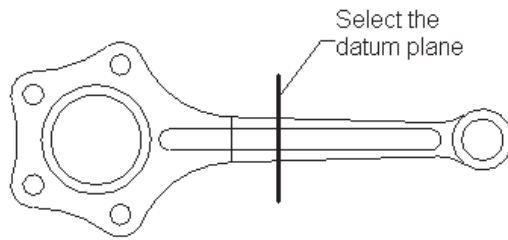


Figure 12-9 Datum plane for the revolved section view **Figure 12-10** Resulting revolved section view

**Note**

The copy and align view will be discussed later in this chapter.

Drawing View Dialog Box Options

The remaining types of drawing views that you can generate in Pro/ENGINEER Wildfire 4.0 require the options available in the **Drawing View** dialog box. Therefore, it is important for you to understand these options before proceeding further.

View Type Option

When you invoke the **Drawing View** dialog box, the **View Type** option is automatically selected in the **Categories** list box. The **View name** edit box is used to enter the name of the drawing view. The **View orientation** area lists the options to orient the drawing view. These options are the standard options of orienting the model.

Visible Area Option

The **Visible Area** option is used to set the display of the view. You can display a drawing view as a full view, partial view, a half view, and so on. The **Drawing View** dialog box, after choosing the **Visible Area** option, is displayed, as shown in Figure 12-11. The options in the **View visibility** drop-down list of this dialog box are discussed next.

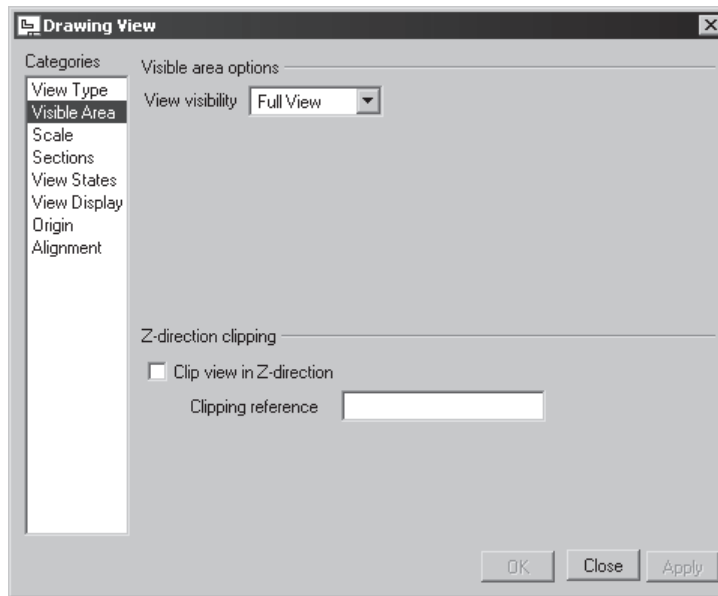


Figure 12-11 The **Drawing View** dialog box with the **Visible Area** option chosen

Full View

The **Full View** option is selected by default in the **View visibility** drop-down list. This option can be combined with any of the view types to generate a drawing view displaying the complete view.

Half View

The **Half View** option can be used on the projection, auxiliary, or general views to generate a drawing view that displays only half view of the part. Generally, an existing view is selected to display the half view. For generating a half view, double click on an existing view to display the **Drawing View** dialog box. From the **Categories** list box, choose the **Visible Area** option in the **Categories** to invoke the **Visible Area** options and then select **Half View** from **View visibility** drop-down list. You are prompted to select a reference plane that will be used to remove half of the drawing view. The reference plane can be a datum plane or a planar surface, and must be normal to the screen in the selected view. Select the reference plane; an arrow will be displayed attached to the selected plane. This arrow indicates the portion of the view to be removed. You can also flip the direction of the arrow using the **Side to keep** button available below the **Half view reference plane** collector in the dialog box. Generally, this type of view is generated for symmetric parts. Therefore, you can specify the type of symmetry line using the **Symmetry line standard** drop-down list. Figure 12-12 shows the reference plane to be selected and Figure 12-13 shows the resulting half view.

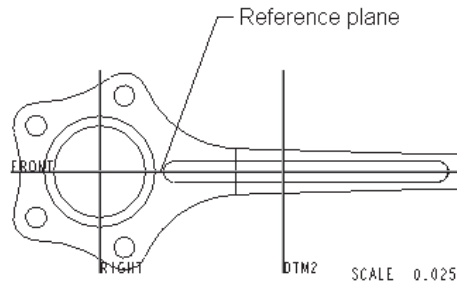


Figure 12-12 Reference plane to be selected to generate a half view

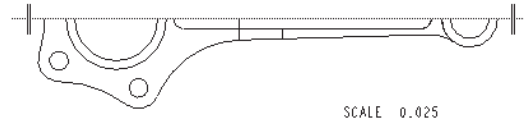


Figure 12-13 Resulting Half view

Partial View

The **Partial View** option can be used on the projection, auxiliary, revolved, or general views to generate a view that displays a specified portion of the view. To convert an existing view to a partial view, double-click on that view. Choose the **Visible Area** option from the **Categories** list box and select the **Partial View** option from the **View visibility** drop-down list; you are prompted to draw a spline that will be the boundary of the portion of the view you want to display, see Figure 12-14. Select the center point on the view and then draw the spline. Now, choose the **Apply** button from the **Drawing View** dialog box. You will notice that only the area of the view inside the spline is retained and the rest of the portion of the drawing view is cropped, see Figure 12-15.

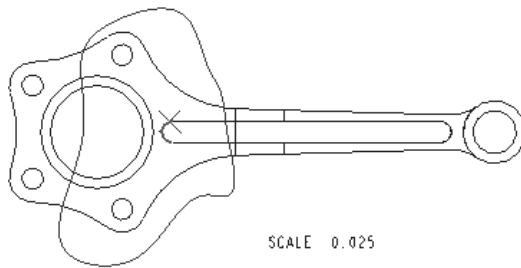


Figure 12-14 Spline drawn on the view

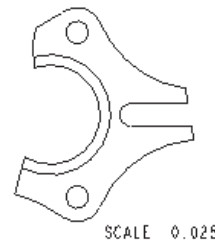


Figure 12-15 Partial view

Broken View

This type of view is used for the parts having a high length to width ratio. The **Broken View** option can be used on the projection or the general views to generate a view that is broken along the horizontal or the vertical direction using horizontal or vertical lines. To generate a broken view, select the **Broken View** option from the **View visibility** drop-down list. Select the **Add break** button in the **Drawing View** dialog box to add break lines. Select any one edge from the view and move the cursor vertically and select a point on the screen to draw the first vertical break line. Move the cursor and select a point up to

which you want to break the view, see Figure 12-16. You can also specify the style of the break line using the **Break Line Style** drop-down list available in the list box. Now, choose the **Apply** button from the **Drawing View** dialog box to create the resulting broken view and then choose the **Close** button to exit it. Figure 12-17 shows the resulting broken view.

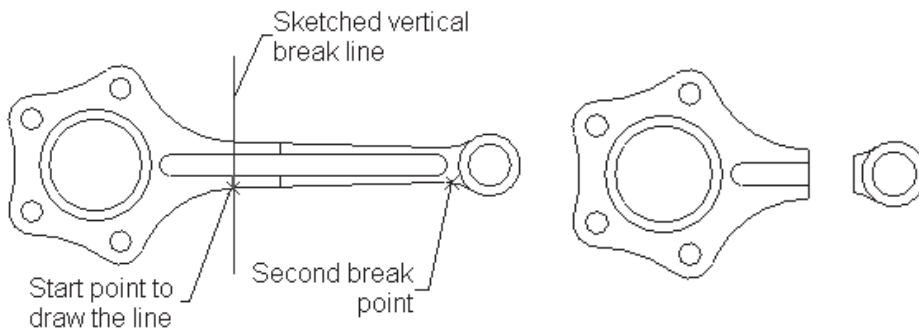


Figure 12-16 References to break the view **Figure 12-17** Resultant Broken view

Figure 12-18 shows various views that have already been discussed in this section.

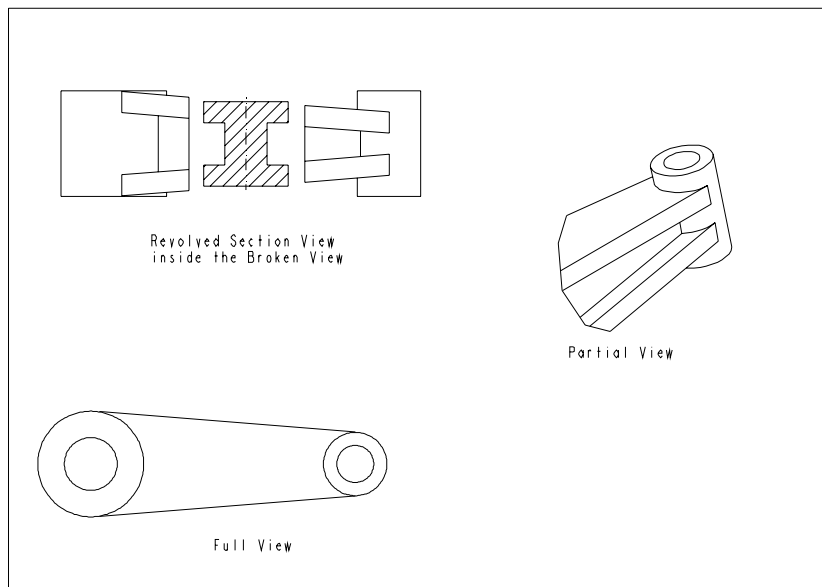


Figure 12-18 The different drawing views

Scale Option

When you select the **Scale** option, the **Scale and perspective options** area will be displayed on the right of the **Categories** list box. The **Default scale for sheet** radio button is selected by

default. Therefore, all generated views are scaled with the default sheet scale. The second radio button is **Custom scale**. This radio button, when selected, allows you to enter a scale for the drawing view. When you modify the scale factor of a view, the view associated with this parent view is also scaled.

The third radio button is **Perspective**. This radio button is selected to generate a perspective view. After selecting the **Perspective** radio button, you can specify the distance from the eye and the diameter of the view circle. Figure 12-19 shows a perspective view.

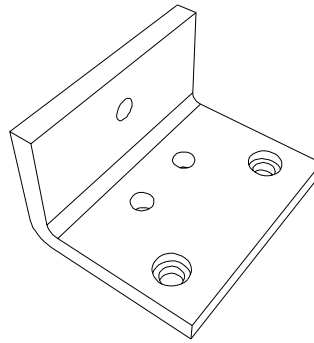


Figure 12-19 A general perspective view

Sections Option

The section views are generally used for the views that are complicated from inside. As it is not possible to display the inside of the part using the conventional views, therefore, these views are cut (sectioned) using a datum plane or a planar surface and the resulting section view is displayed. When you choose the **Sections** option from the **Categories** list box in the **Drawing View** dialog box, the **Section options** area will be displayed. The **No section** radio button is selected by default. Select the **2D cross-section** radio button and then choose the **Add cross-section to view** button to activate the options to create a section. The options in the **Section options** area and the methods of section plane creation are discussed next.

Full Section View

Consider a part that is cut throughout its length, width, or height and the cut portion is removed from the display. The remaining portion, when projected normal to the cutting plane, displays the full section view. The **Full** option in the **Sectioned area** drop-down list allows you to create a full section view.

To generate a full section view, you first need to generate a projected view from one of the views existing on the drawing sheet. After generating the projected view, double-click on it to invoke the **Drawing View** dialog box. Choose the **Sections** option from the **Categories** list box. The options available for generating the section view are displayed on the right of the **Drawing View** dialog box. Select the **2D cross-section** radio button to define a section plane. Now, choose the **Add cross-section to view** button to add a section plane. Some options in the drop-down lists are invoked in the area provided below the **Add cross-section to view** button.

When you choose the **Add cross-section to view** button; the **XSEC CREATE** menu will be displayed. Choose **Planar > Single > Done** from the **XSEC CREATE** menu; you are prompted to specify the name of the cross-section. Specify the name of the cross-section in the **Message Input Window**. The **SETUP PLANE** menu will be displayed and you will be prompted to select a planar surface or a datum plane. Select the datum plane from the other view; the name of the section view will be displayed in the **Name** drop-down list. Scroll to the right in the dialog box and click once in the **Arrow Display** collector to invoke the selection mode and select the view to display the section view arrows. You can also flip the material side of the section view using the **Flip** button. Choose the **Apply** button from the **Drawing View** dialog box. Figure 12-20 shows the top view and section view.

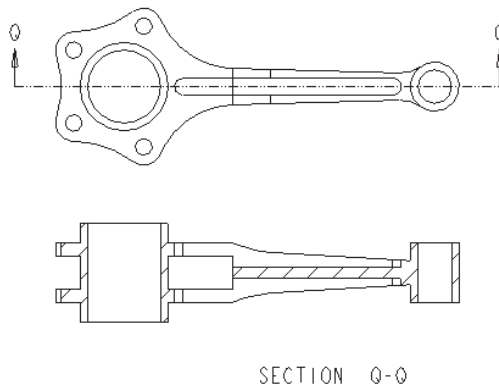


Figure 12-20 Top view and section front view

In the above section, you have learned to generate a full section view by defining a planar section plane. Now, you will learn to generate a full section view by defining an offset section plane. You need to make sure that before generating a section view by defining an offset section plane, the part file of the current model should be closed.

Invoke the **Drawing View** dialog box by double-clicking on an already generated projected view. Now, choose the **Sections** option from the **Categories** list box on the left of this dialog box. Select the **2D cross-section** radio button and choose the **Add cross-section to view** button to define the section plane. If there are some existing section planes, select the **Create New** option from the **Name** drop-down list; the **XSEC CREATE** menu will be displayed. Choose **Offset > Both Sides > Single > Done** from the **XSEC CREATE** menu; the **Message Input Window** will be displayed. Specify the name of the cross-section.

The model will be displayed in a subwindow on the left of the screen. The **SETUP SK PLN** menu will be displayed and you need to select a sketching plane for drawing the sketch that represents an offset section plane. Select the sketching plane from the subwindow. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu will be displayed; choose **Default** from this submenu. Draw the sketch of the offset section using the sketch tools available in the **Sketch** menu of the subwindow, refer to Figure 12-21. After drawing the sketch, choose **Sketch > Done** from the menu bar to exit the sketcher environment.

Click once in the **Arrow Display** collector to invoke the selection mode and then select the view in which the section arrows will be displayed. Choose the **Apply** button from the **Drawing View** dialog box. Figure 12-21 shows the top view, offset sectioned front view, and sectioned general view of a part.

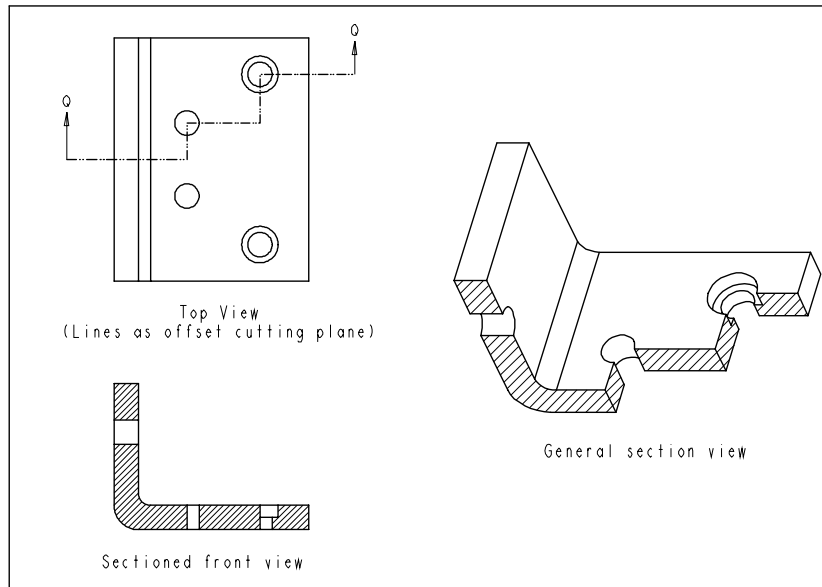


Figure 12-21 The section views created using the offset section plane

To generate an isometric section view, generate the general view and switch the orientation of the general view to isometric. Now, select the **Sections** option from the **Categories** list box and sketch a section plane using the same subwindow method as discussed earlier. Note that you cannot select a datum plane to define the section plane for an isometric section view. Figure 12-22 shows the top, full section front, and full section isometric views.

Half Section View

Consider a part that is cut half way through the length, width, or height and the front cut portion (front quarter) is removed from the display. This part when projected is called the half section view. In this projected view, only half of the part will be displayed sectioned and the other half of the part will be displayed as it is. To generate the half section view, you need to select the **Half** option from the **Sectioned Area** drop-down list while specifying the parameters for sectioning the view. After selecting this option, you are prompted to select the reference plane for the half section creation. Select the plane from the drawing view; you are prompted to pick the side to keep. Select a point on the side of the section view that you need to keep. Select the parent view where the section arrow will be placed and choose the **Apply** button from the **Drawing View** dialog box. Figure 12-23 shows the resulting half section view.

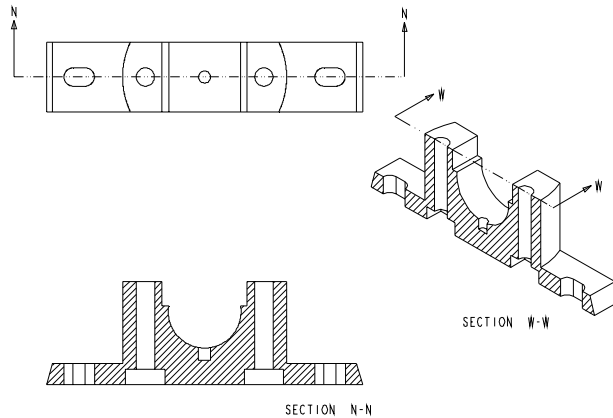


Figure 12-22 Top view, full section front view, and full section isometric view

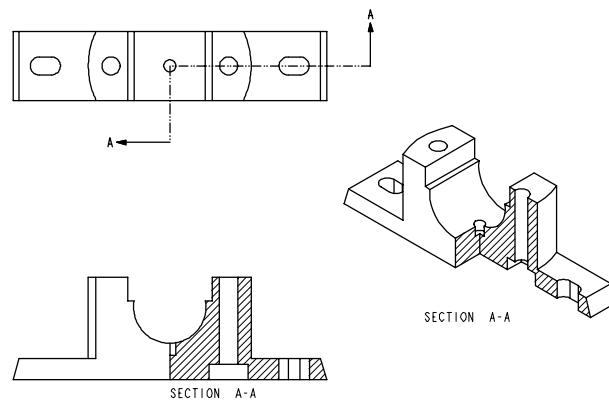


Figure 12-23 Top view, half section front view, and half section isometric view

Local Section View

The **Local** section view is used when you want to show a particular portion of the view in the section, and at the same time not section the remaining view. The local section area is specified by drawing a spline around it. To generate the local section view, you first generate the full section view, as discussed earlier. After generating the full section view, select the **Local** option from the **Sectioned Area** drop-down list. Next, you need to draw a spline that will define the area of the drawing that needs to be sectioned.

To draw a local section isometric view, you need to draw a section plane and then choose the **Local** option. Define a center point for the local section and sketch the spline. This method of generating the local section isometric view is similar to generating the other

section isometric views. Figure 12-24 shows the local section front view and the local section isometric view.

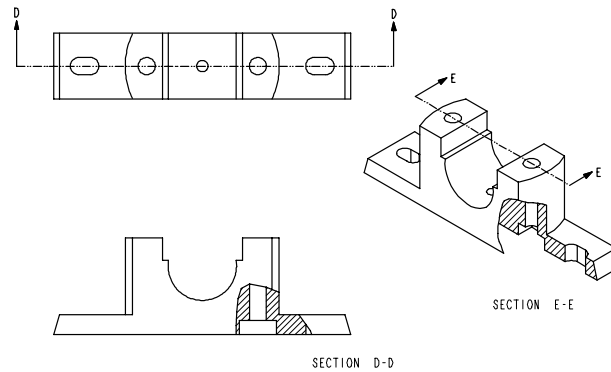


Figure 12-24 Top view, local section front view, and local section isometric view

Model edge visibility

Pro/ENGINEER Wildfire 4.0 provides you with two options for displaying the edges of the section drawing views. The **Total** radio button when selected in the **Section options** area of the **Drawing View** dialog box creates a section view that displays all visible edges of the section view, in addition to the section area. This option can be combined with the full, half, local, full (unfold) and full (aligned) views. Figure 12-25 shows the front and total sectioned left view.

The **Area** radio button, when selected, displays only that area of the section view that is sectioned. No other edges of the view are displayed in the area cross-section view, as shown in Figure 12-26. This option can be combined with the full, half, local, full (unfold) and full (aligned).

Full(Aligned)

This 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 parent view. Generally for this section view, the section plane is sketched. Remember that the axis about which the feature is straightened should lie on all cutting planes. This means that the lines that are used to sketch the section plane should pass through the axis about which the feature will be revolved. Figures 12-27 and 12-28 show these views when the **Area** radio button is selected. Figures 12-29 and 12-30 shows these views when the **Total** radio button is selected.

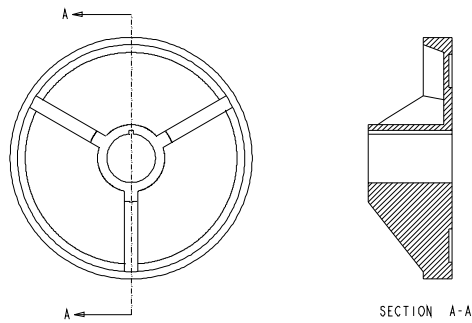


Figure 12-25 The front view and the total cross-section view

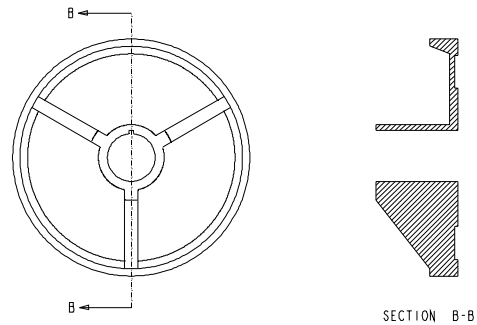


Figure 12-26 The front view and the area cross-section view

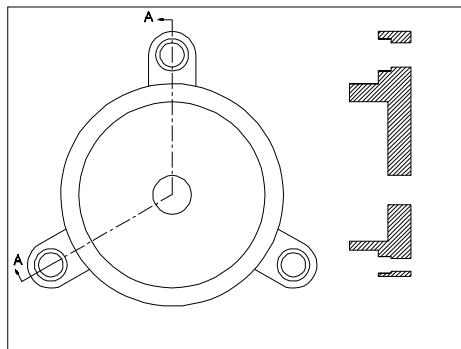


Figure 12-27 The area cross-section view with normal lines of projection

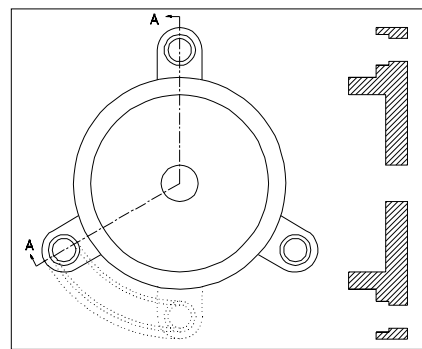


Figure 12-28 The area cross-section view with aligned lines of projection

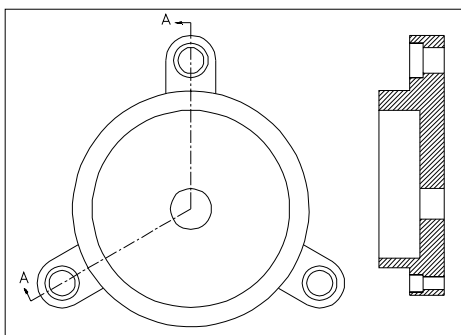


Figure 12-29 The total cross-section view with normal lines of projection

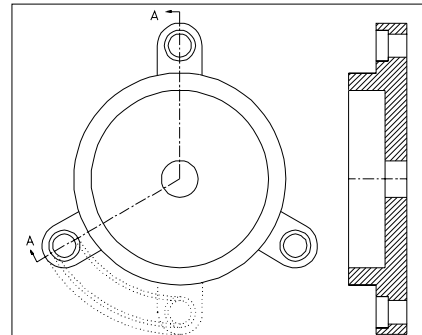


Figure 12-30 The total cross-section view with aligned lines of projection

Full(Unfold)

The **Full(Unfold)** option is used to generate the section view by unfolding the section surface of an offset section view. This type of view is only generated using a general view. To generate this type of view, first you need to generate a general view in the required orientation. You can place this general view anywhere on the drawing sheet. Now, choose the **Sections** option from the **Categories** on the left of the **Drawing View** dialog box. Now, define the section plane using the offset option and specify the view in which you need to display the arrows. Now, choose the **Full(Unfold)** option from the **Sectioned Area** drop-down list and complete the creation of the full unfold section view. Figure 12-31 shows the model and the section plane. Figure 12-32 shows an offset section view and a full unfold section view.

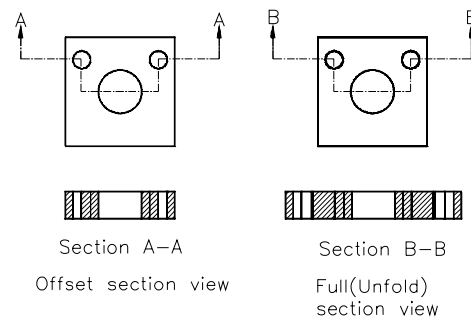
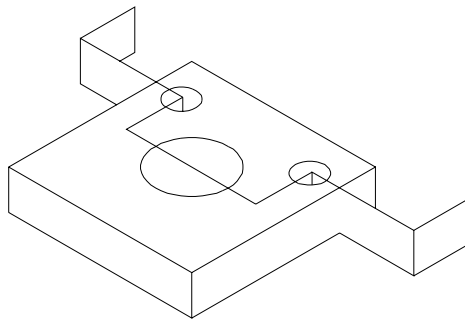


Figure 12-31 The model and the section plane

Figure 12-32 Offset section view and full unfold section view

Once you have understood the main options of the **Drawing View** dialog box, you can generate the copy and align the view. This view is discussed next.

Generating the Copy and Align View

These are the aligned views that are generated from an existing partial view. Therefore, to generate this type of drawing view, it is necessary that first a partial view should be generated. You will learn more about the partial views later in this chapter.

The procedure to generate a **Copy and Align** view is discussed next.

1. To generate the **Copy and Align** view, choose **Insert > Drawing View > Copy and Align** from the menu bar. You will be prompted to select an existing partial view to be aligned with. Once you have selected a partial view, you are prompted to select a center point for the drawing view.
2. Select a point anywhere on the drawing sheet to place the view. The view is placed at the selected location. You are prompted to specify a center point for detail on the current view and to sketch a spline to define the outline of the **Copy and Align** view.
3. Specify the center point of the spline and draw the spline. Press the middle mouse button

to finish sketching the spline. The drawing view will be cropped along the spline drawn. You are prompted to select a straight line (axis, segment, datum curve) alignment on the current view.

4. Select an edge from the current view, as shown in Figure 12-33, to align it with the partial view. The copied view is aligned with its parent partial view, as shown in Figure 12-34.

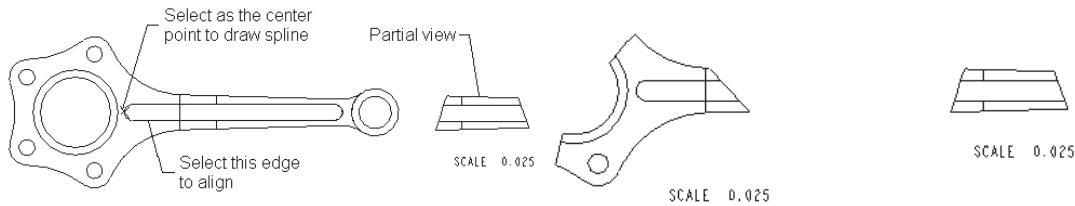


Figure 12-33 Selecting an edge from the current view to align with the partial view

Figure 12-34 Resulting copy and align view

Generating the 3D Cross-Section View

The **3D Cross-section** allows you to display 3D sections in the drawing views. This type of sectioning may be required to show the complex cross-sections of a model. Note that the 3D cross-sections are created using zones. Therefore, you first need to create zones and then use them to create 3D cross-sections. The procedure to create zones and 3D cross-sections is discussed next.

1. Open the part file of the model and choose **View > View Manager** from the menu bar or choose the **View Manager** button from the **Top Toolchest**; the **View Manager** dialog box will be displayed.
2. Choose the **XSec** tab and then choose the **New** tab to define a new cross-section in the model.
3. Enter the name for the new cross-section and press ENTER; the **XSEC CREATE** menu will be displayed, as shown in Figure 12-35.
4. Select the **Zone** option from the menu; a dialog box with the name you have specified for the new cross-section will be displayed, as shown in Figure 12-36.



Note

In the example discussed above, the name for the new 3D cross-section is given as **Xsec0001**. Therefore, the dialog box shown in Figure 12-36 is named as **Xsec0001**.

5. Now, you need to define the planes for the zone creation. Select the datum plane through which you want to create a zone.



Figure 12-35 The XSEC CREATE menu

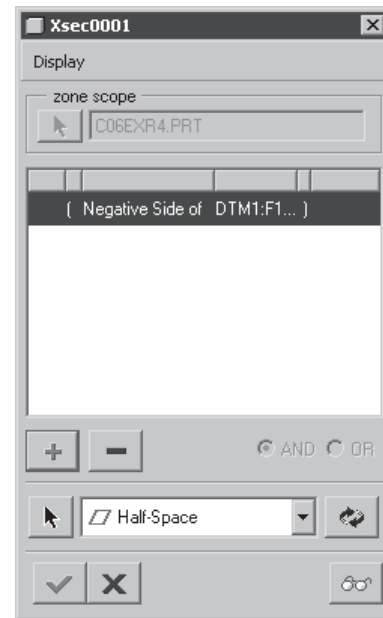





Figure 12-36 The Xsec0001 dialog box

6. You can add multiple reference entities or datum planes for defining the zone by choosing the **Add a reference to the zone** button from the **Xsec0001** dialog box. You can also remove references specified for the zone by selecting the reference from the **zone scope** area and then choosing the **Remove a reference from the zone** button from the **Xsec0001** dialog box.  
7. By default, the direction for creation of the zone is chosen as the negative side of the selected reference entity. You can change the orientation by choosing the **Change orientation** button from the **Xsec0001** dialog box. 
8. After specifying all the references for the zone, choose the **Accept settings** button to finish the zone creation.

**Note**

*If you want to make the 3D cross-section active in the drawing area, select it from the **View Manager** dialog box and right-click to invoke the shortcut menu. Choose the **Set Active** option from the shortcut menu; the 3D cross-section will be activated in the drawing area.*

9. Start a new drawing file and insert a general view in the drawing sheet by choosing the **General** button from the **Top Toolchest**.
10. Choose the **Sections** option from the **Categories** list of the **Drawing View** dialog box. Next, select the **3D cross-section** radio button from the **Section options** area; the 3D cross-section with the name **Xsec0001** will be listed in the drop-down list.

11. The **Show X-Hatching** check box is not selected by default. If you need to display the hatching lines, then select the **Show X-Hatching** check box and choose the **Apply** button from the **Drawing View** dialog box. The 3D cross-section will be displayed on the drawing sheet.

Figure 12-37 shows the trimetric view and the 3D cross-section of a part model, inserted in an A4 drawing sheet.

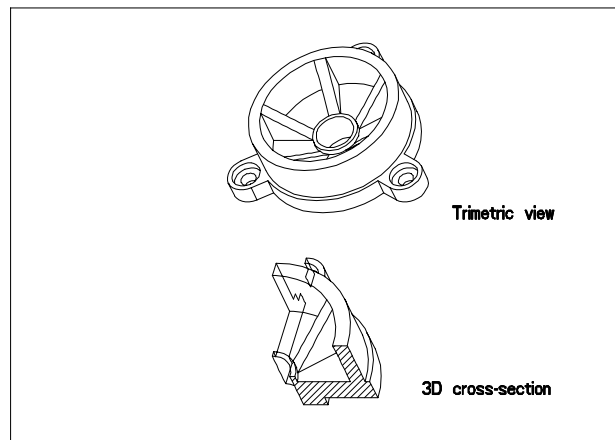


Figure 12-37 The trimetric view and the 3D cross-section in an A4 drawing sheet



Note

The model created in Exercise 4 of chapter 7 is used to generate a 3D cross-section, as shown in Figure 12-37. The references for creating the zone are the **FRONT** and the **RIGHT** datum planes.

You can also display the 3D cross-section in the **Shading** mode.

EDITING THE DRAWING VIEWS

With this release of Pro/ENGINEER, it has become very easy to edit a drawing view and the items in the drawing views. All options of modifying the drawing views or items related to it can be chosen from the shortcut menu that will be displayed when you right-click after selecting the view or the item. However, these options can also be invoked from the menus available in the menu bar. Pro/ENGINEER allows you to perform the following types of editing operations on the drawing views.

Moving the Drawing View



When a view is generated, its movement is locked by default. To unlock the drawing view, choose the **Lock View Movement** button from the **Top Toolchest**. When you select the drawing view, it will be displayed inside a boundary. To move the drawing view, select it and then drag it to the required location on the drawing sheet. Remember that

if you select the view that has some child views, the child views will also move along with the parent view in order to maintain their alignment with the parent view. Also, the projected views can be moved only in the direction of projection.

**Note**

*The **General** and **Detailed** views can be moved to any new location because they are not the projections of any view.*

Erasing the Drawing View

Choose **View > Drawing Display > Drawing View Visibility** from the menu bar to invoke the **VIEWS** menu. The **Erase View** option in the **VIEWS** menu is used to temporarily remove the selected drawing view from the sheet. However, the view still remains in the memory of the drawing and can be resumed at any point of time. As the view is not completely removed from the memory, you can also erase a view that has some child views associated to it and it will not affect the child view. Once a view is erased, a box will be displayed in place of the view displaying the name of the view. To resume the view, choose the **Resume View** option from the **VIEWS** menu.

**Note**

When you erase a view, the leaders and dimensions that were attached with the view are also erased. When you resume an erased view, the leaders and dimension values are displayed again.

Deleting the Drawing View

To delete a drawing view from the drawing sheet, select the view and hold down the right mouse button to invoke a shortcut menu. Choose the **Delete** option from the shortcut menu. Alternatively, you can select the drawing view to be deleted and then choose the **Delete** button in the **Top Toolchest**. Once the view is deleted, no information related to the deleted view remains in the memory of the drawing. Remember that if a view that has some child views associated with it is deleted, then before deletion Pro/ENGINEER informs you that views associated with this view will also be deleted. You can use the **Undo** button from the **Top Toolchest** for restoring the deleted views.

Adding New Parts or Assemblies to the Current Drawing

You can also add more parts or assemblies, in addition to the default part or assembly, for generating the drawing views. This is done by choosing **File > Properties** from the menu bar; the **FILE PROPERTIES** menu will be displayed, as shown in Figure 12-38. Choose the **Drawing Models** option; the **DWG MODELS** menu will be displayed, as shown in Figure 12-39.

Choose the **Add Model** option to add a model to the current drawing. When you choose this option, the **Open** dialog box will be displayed. You can select the new model to be added using this dialog box. Remember that the latest model added will be the current model and the drawing views generated will be of this model. You can change the current model by choosing the **Set Model** option from the **DWG MODELS** menu. Similarly, you can also delete a model from the current drawing. However, only the model that does not have any view generated from it can be deleted.

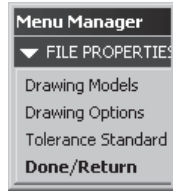


Figure 12-38 The **FILE PROPERTIES** menu

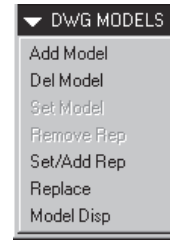


Figure 12-39 The **DWG MODELS** menu

MODIFYING THE DRAWING VIEWS

You can also make the following modifications in the existing drawing views.

Changing the View Type

Select the drawing view that needs to be modified and hold down the right mouse button to invoke the shortcut menu. Choose **Properties** from the shortcut menu; the **Drawing View** dialog box will be displayed, as shown in Figure 12-40. The **Drawing View** dialog box can also be invoked by double-clicking on the view to be modified. The **View Type** option in the **Categories** area is chosen by default. You can modify the view type by selecting one of the types from the list under **Model view names** collector. Remember that only the general, projection, and auxiliary view types can be modified using this option.

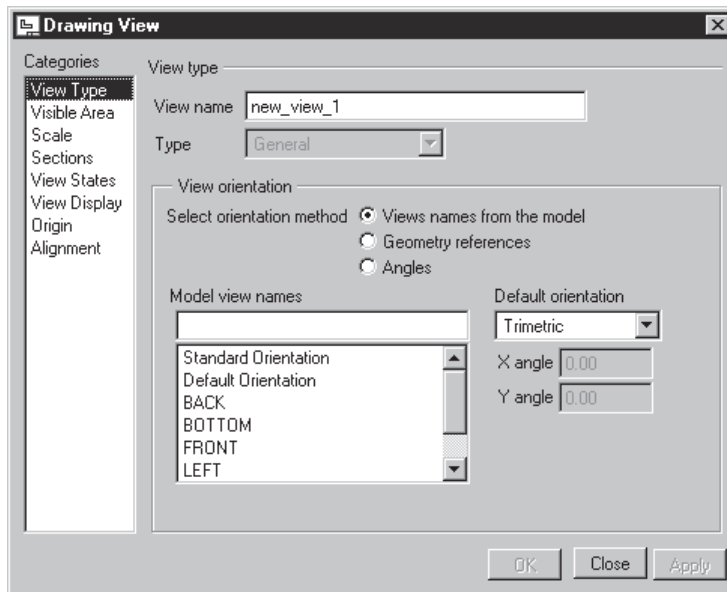


Figure 12-40 The **Drawing View** dialog box

Changing the View Scale

The scale of a view can be modified using the **Scale** option from the **Categories** list box of the **Drawing View** dialog box. You can modify the scale factor of the views that are generated using the general view and detailed view options.



Note

*The scale factor represents the scaling of the drawing model. For example, if the scale set is **0.25**, Pro/ENGINEER scales the drawing views to one-quarter (1/4) of the actual size of the model.*

*You can also modify the scale factor by selecting it from the drawing sheet and holding down the right mouse button. From the shortcut menu, choose the **Edit Value** option. Next, specify the scale factor in the **Message Input Window** displayed.*

Reorienting the Views

Double click on the general view to be reoriented to display the **Drawing View** dialog box. You can use the options from this dialog box to change the orientation of the selected view. However, if some child view is associated with the general view, then the child view will also be reoriented accordingly.

Modifying the Cross-sections

You can flip the side of sectioning and replace, delete, or rename the sections in the views using the **Section** option from the **Categories** list in the **Drawing View** dialog box.

Modifying Boundaries of Views

You can modify the boundaries of the detailed or partial views using the option in the **Drawing View** dialog box. To modify the boundary of a detailed view, double-click on it to invoke the **Drawing View** dialog box. The selection mode is invoked by default in the **Reference point on parent view** selection area. Therefore, you can specify the center point for the boundary of the detailed view. To sketch the boundary of the detailed view again, click once in the **Spline boundary on parent view** selection area and draw the boundary on the parent view. To modify the boundary of a partial view, invoke the **Drawing View** dialog box and choose the **Visible Area** option from the **Categories** list box. Using the options available on the right of this dialog box, you can modify the boundary of the detailed view.

Adding or Removing the Cross-section Arrows

If you have not specified the cross-section arrows while generating the section views, you can specify them by selecting the section view and holding down the right mouse button to invoke the shortcut menu. Choose the **Add Arrows** option from the shortcut menu; you will be prompted to select the view where the arrows should be displayed. Select the view in which you need to display the arrows.

To remove arrows, select the arrows and choose the **Delete** button from the **Top Toolchest**.

Modifying the Perspective Views

Double-click on the perspective view to display the **Drawing View** dialog box and choose the

Scale option from the **Categories** list box. The options related to the perspective are available. You can modify the eye point distance or the view diameter using these options.

MODIFYING OTHER PARAMETERS

Apart from modifying the drawing views, you can also modify other parameters related to the drawing views. For example, you can modify the size and the style of the text, scale factor of all drawing views, cross-section hatching, and so on. All this is done by selecting the item. When the item is highlighted in red color, hold down the right mouse button to invoke the shortcut menu. You can select the options from the shortcut menu to modify that item.

You can modify any parameter associated with the drawing views. Depending on the item selected to modify, the options related to it vary. The options displayed in the shortcut menu vary from item to item.

Editing the Cross-section Hatching

Select the hatching from a drawing view. When the hatching turns red in color, hold down the right mouse button to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu; the **MOD XHATCH** menu will be displayed, as shown in Figure 12-41.

The parameters related to cross-section hatching that can be modified are: the spacing of the hatching, angle of the hatching lines, offset value, and the line style of the hatching lines. There are also some standard hatch patterns that are available in Pro/ENGINEER. You can retrieve these standard patterns by using the **Open** dialog box displayed upon choosing **Retrieve** from the **MOD XHATCH** menu.

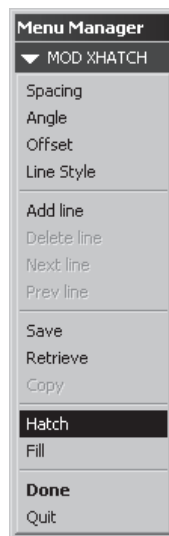


Figure 12-41 The MOD XHATCH menu

TUTORIALS

Tutorial 1

In this tutorial, you will generate the drawing views of the model created in Exercise 4 of Chapter 7, as shown in Figure 12-42. Select the A4 size drawing sheet and generate all drawing views shown in Figure 12-43. **(Expected time: 45 min)**

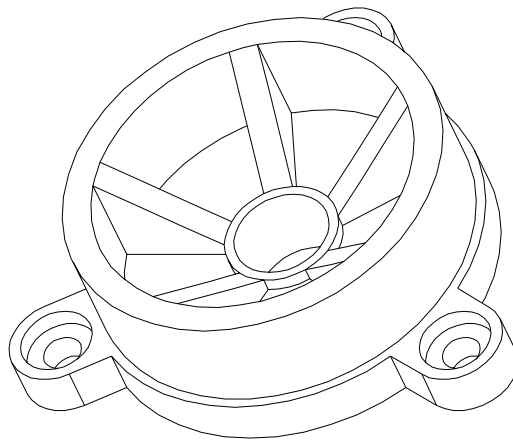


Figure 12-42 Part for generating the drawing views

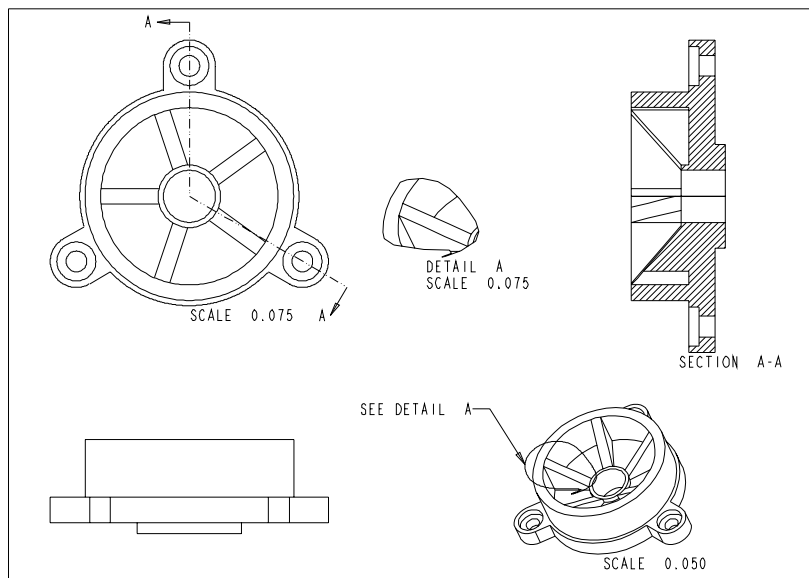


Figure 12-43 The drawing views to be generated

The following steps are required to complete this tutorial:

- Start a new drawing file and select the required drawing sheet.
- Generate the top view, refer to Figure 12-45.
- Generate the front view taking the top view as the parent view, refer to Figure 12-45.
- Generate the sectioned right view by defining a plane on the top view of the model, refer to Figures 12-46 and 12-47.
- Generate the default 3D view of the model, see Figure 12-47.
- Generate the detail view from the 3D view, see Figure 12-48.

Before you start generating the drawing views, set the working directory to *C:\ProE-WF-4.0\c12*. Copy and paste the model *c07exr4.prt* file in the folder named *c12* from *c07* folder. The drawing that you will generate will be saved in the *c12* directory with an extension *.dwt*. The part file (*.prt*) and the drawing file (*.dwt*) should lie in the same directory or folder.



Tip: If you delete the part file from which the drawing views are generated, the drawing file will not open. Also, if the part file is removed from the folder where its drawing file is placed or if you rename the part file after generating the drawing views, the drawing file will not open.

Starting a New Drawing File

To generate the drawing views in Pro/ENGINEER Wildfire 4.0, you first need to start a new file in the **Drawing** mode.

- Choose the **New** button from the **File** toolbar to display the **New** dialog box.
- Select the **Drawing** radio button and then enter the name of the file as *c12tut1*.
- Choose **OK** from the **New** dialog box to display the **New Drawing** dialog box.
- Choose the **Browse** button to select *c07exr4.prt* for generating the drawing views.
- Select the **Empty** radio button from the **Specify Template** area.
- Choose the **Landscape** button from the **Orientation** area, if it is not chosen by default, as shown in Figure 12-44.
- Select **A4** from the **Standard Size** drop-down list. Choose **OK** to proceed to the **Drawing** mode.

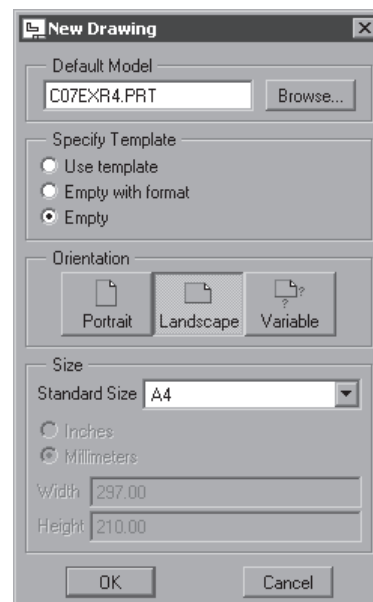


Figure 12-44 The **New Drawing** dialog box with the **Landscape** button selected

An A4 size sheet with the landscape orientation is displayed in the drawing area.

Generating the Top View

Generally, you can generate any view as the first view on the drawing sheet. However, in this tutorial, you will generate the top view as the first view.

1. Choose the **General** button from the **Top Toolchest**.

You are prompted to specify the center point for the drawing view.

2. Specify the center point for the first view, which is closer to the top left corner of the sheet, as shown in Figure 12-45. As soon as you specify the center point, the **Drawing View** dialog box is displayed along with the preview of the drawing view in the default orientation. From the **Model view names** list box, select the **TOP** option and choose the **Apply** button. The default view is oriented as the top view.



Note

Although the views once placed can be moved on the sheet yet while specifying the placement point, try to place the drawing view inside the sheet and also leave space for the other views to be generated.

3. Choose the **Scale** option from the **Categories** list box in the **Drawing View** dialog box; the **Scale and perspective options** area is displayed. Select the **Custom scale** radio button and enter the scale factor for the view as **0.075**. Choose the **Apply** button; the scale of the view will be modified.
4. Choose the **Close** button from the **Drawing View** dialog box to complete the placing of the top view.
5. Turn off the display of datum planes, axes, points, and coordinate system by choosing their respective buttons from the **Datum Display** toolbar.
6. Also, change the model display to no hidden by choosing the **No hidden** button from the **Model Display** toolbar. Now, choose the **Repaint** button from the **Top Toolchest** to repaint the screen.



Tip: If the view that you have placed on the drawing sheet is not at the proper location, you need to move the drawing view. To unlock the movement, choose the **Lock View Movement** button from the **Top Toolchest**. Select the drawing view; the selected drawing view is enclosed in a red box. Now, press the left mouse button inside the box and drag it to place it at the required location.

Generating the Front View

The front view of the model is generated from the top view, which is already placed on the drawing sheet.

1. Choose **Insert > Drawing View > Projection** from the menu bar; you will be prompted to specify a center point for drawing view.

2. Specify the center point for the front view below the top view, as shown in Figure 12-45.
3. Click anywhere on the drawing sheet to clear the current selection set.

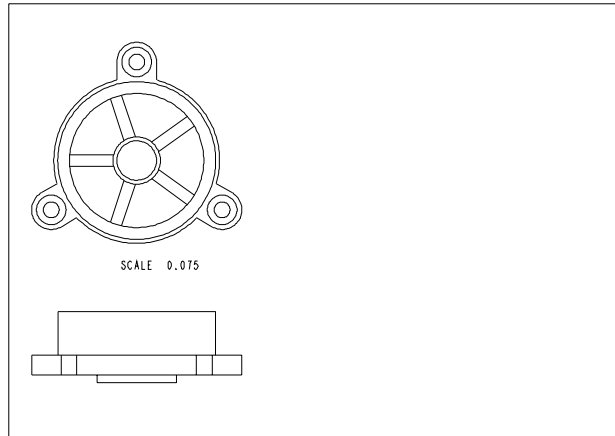


Figure 12-45 The top and front views



Tip: It is recommended that you use the **Repaint** button available in the **View** toolbar to remove any temporary information in the drawing area and to refresh the screen.

Generating the Section View

To generate a section view, a section must be defined on the model. You will sketch this section to generate the section view for this tutorial.

1. Choose **Insert > Drawing View > Projection** from the menu bar; you are prompted to select a projection parent view.
2. Select the top view as the projection parent view. You are again prompted to specify a center point for drawing view.
3. Specify the center point for placement of the view on the right side of the top view.
4. Once the view is placed, double-click on it to display the **Drawing View** dialog box.
5. Select **Sections** from the **Categories** list box in the **Drawing View** dialog box; the section options are invoked in **Drawing View** dialog box.
6. Select the **2D cross-section** radio button and choose the **Add cross-section to view** button; the window area below is activated. Select the **Create New** from the **Name** drop-down list; the **XSEC CREATE** menu is displayed.
7. Choose **Offset > Both Sides > Single > Done** from the **XSEC CREATE** menu.
8. Specify the name of the cross-section as **X** in the **Message Input Window** and press

ENTER. Once you have specified the name of the cross-section, a separate window will appear displaying the part. You are prompted to select a sketching plane.

**Note**

*If the model in the **Part** mode is opened in another window, the subwindow will not appear and you have to manually change the window. Choose the **Window** menu, and select the part file that is opened. Now, you can continue with the sketcher environment.*

9. Choose the **Plane Display** button from the **Datum Display** toolbar in the original window to display the datum planes. You need to repaint the screen in the subwindow by choosing **View > Repaint** from the menu bar of the subwindow to view the datum planes.
10. Select the **TOP** datum plane from the subwindow. Choose **Okay** from the **DIRECTION** submenu; the **SKET VIEW** submenu is displayed.
11. Choose the **Bottom** option from the **SKET VIEW** submenu and select the **FRONT** datum plane from the subwindow.
12. Choose the **Sketch > Line > Line** from the menu bar and sketch the line, as shown in Figure 12-46. These lines define the section plane.
13. Align the lines and modify the angular dimension to **120**, as shown in Figure 12-46.

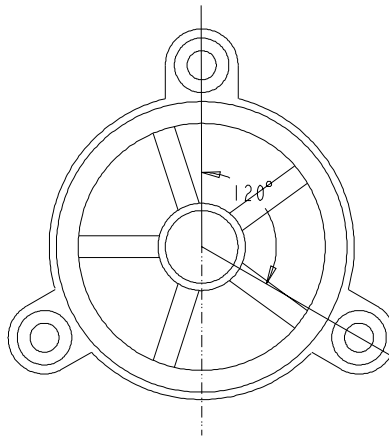


Figure 12-46 Sketch for generating the total aligned section view

14. Choose **Sketch > Done** from the menu bar of the subwindow.
15. Select the **Full(Aligned)** option from the **Sectioned Area** drop-down list; you are prompted to select an axis.
16. Select the central axis of the model from the front drawing view. You may need to turn on the display of the central axis if its display is turned off.

17. Scroll the bar in the **Drawing View** dialog box to the right edge. Click once in the field below the **Arrow Display** column; you will be prompted to pick a view for arrows where the section is perpendicular. Select the top view to display the arrows.
18. Choose the **Apply** button from the **Drawing View** dialog box and then exit the dialog box; the section view is displayed.
19. Turn off the display of the datum planes and the datum axes.



Tip: If you do not want to display the section arrows in any view, you can use the middle mouse button (in case of three button mouse) to abort the creation of arrows.

Modifying the Hatching

You need to modify the spacing between the hatch lines in order to make the hatching dense. In this tutorial, you will select the filter that is available to select the hatch lines. This filter is available from the **Filter** drop-down list.

1. Select the **X-Section** filter from the **Filter** drop-down list that is present in the **Status Bar** below the drawing area. Select the hatching from the drawing sheet; the hatching lines turn red.
2. Press the right mouse button to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu; the **MOD XHATCH** menu is displayed. Alternatively, you can invoke the **MOD XHATCH** menu by double-clicking on the hatching lines.
3. From the **MOD XHATCH** menu, choose the **Spacing** option to display the **MODIFY MODE** submenu.
4. The spacing between the hatching lines has to be reduced. Choose the **Half** option twice and then choose **Done** in the **MOD XHATCH** menu. Now, the hatching appears to be more dense, as shown in Figure 12-47.
5. Set the filter back to the **Drawing Item and View** option using the **Filter** drop-down list.

Generating the General View

The **General** view is generated to show a 3D view of the model, which is the trimetric view.

1. Choose the **General** button from the **Top Toolchest**.
2. Specify the center point for the placement of the general view below the section view, as shown in Figure 12-47; the **Drawing View** dialog box is displayed.
3. Set the value of the scale for the new view to **0.05** using the **Scale** option from the **Drawing View** dialog box.
4. Choose the **Apply** button and then exit the dialog box.

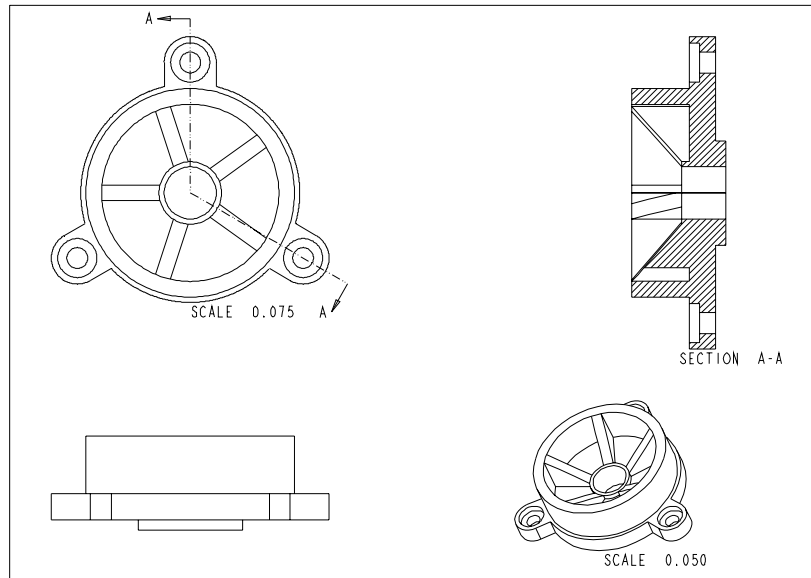


Figure 12-47 Different drawing views

Generating the Detail View

As mentioned earlier, the detail view is required to provide details of a particular portion of the drawing view. In this tutorial, you need to give the details of one of the ribs of the model.

1. If the general view is selected (highlighted in red box), then you need to click once on the drawing sheet to exit the current selection set. Choose the **Insert > Drawing View > Detailed** from the menu bar.

You will be prompted to select a center point for detail on an existing view. This center point will be the center of the detailed view.

2. Select the center point for the detail on one of the ribs in the trimetric view, refer to Figure 12-47.

You will be prompted to sketch a spline about the center point that you selected.

3. Draw the spline. After the spline is drawn, press the middle mouse button to quit the spline tool.
4. Select the center point for the placement of the drawing view. The detailed view is generated and is selected by default.
5. Press and hold the right mouse button on the detail view to invoke the shortcut menu and choose the **Properties** option; the **Drawing View** dialog box will be displayed.

6. Choose **Spline** from the **Boundary type on parent view** drop down list.
7. Choose the **Scale** option from the **Categories** list box. Set the value of the custom scale to 0.075. Choose the **Apply** button and exit the dialog box.

A note is also displayed with an arrow attached to the trimetric view. You may need to move the note to a suitable position by selecting it and then dragging the cursor. The final sheet after generating all views is shown in Figure 12-48.

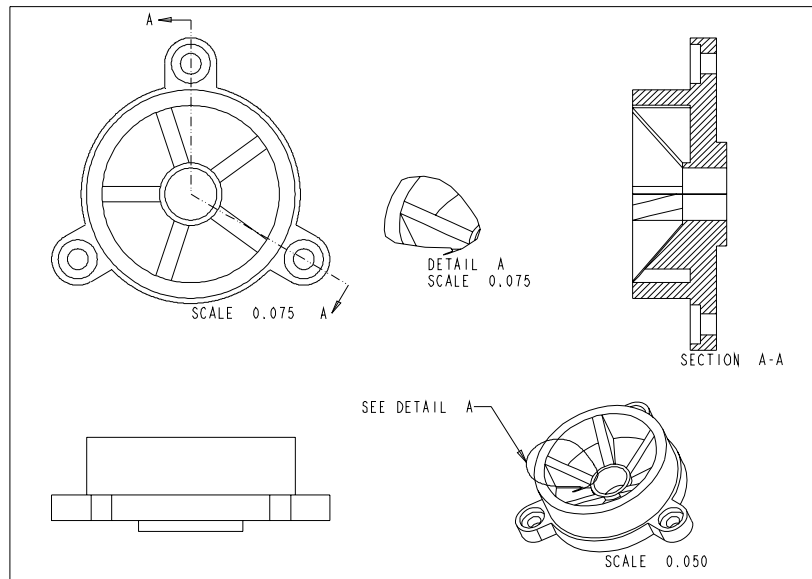


Figure 12-48 The drawing views for Tutorial 1



Tip: If any of the views or the text on the drawing is overlapping or is not at the desired place on the sheet, select it and then move it by dragging the mouse.



Note

To select any item or drawing view from the drawing sheet, you need to invoke the selection mode by choosing the **Entity select** button from the **Right Toolchest**.

Saving the Drawing File

You need to save the drawing file that you have created as you may need it later.

1. Choose the **Save** button from the **Top Toolchest**; the **Save Object** dialog box is displayed with the name of the drawing file that you entered earlier.
2. Press ENTER to confirm the saving of the file.

Closing the Drawing File

After you have saved the drawing file that you have created, you need to close the drawing file.

1. Choose **Window > Close** from the menu bar to close the drawing window.

Tutorial 2

In this tutorial, you will generate the drawing views of the part created in Tutorial 1 of Chapter 8. The part is shown in Figure 12-49. The drawing views that need to be generated are shown in Figure 12-50. **(Expected time: 45 min)**

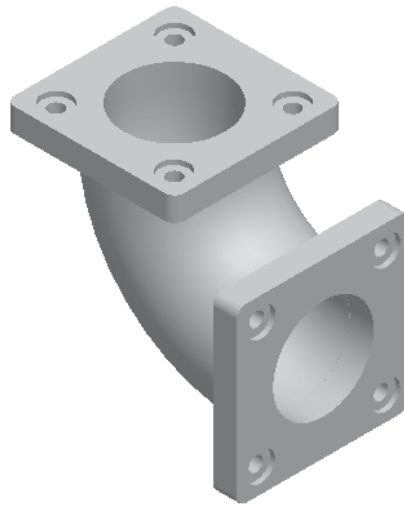


Figure 12-49 Model for generating drawing views

Before you start generating the drawing views, copy the file *c08tut1.prt* from *c08* folder in the current directory.

The following steps are required to complete this tutorial:

- a. Start a new drawing file and select the size of the drawing sheet.
- b. Generate the top view, refer to Figure 12-51.
- c. Generate the sectioned front view by defining the **FRONT** datum plane as the section plane, refer to Figure 12-51.
- d. Generate the right-side view of the sectioned front view, refer to Figure 12-52.
- e. Generate the isometric sectioned view. The section will be defined by drawing a line on the **TOP** datum plane, refer to Figures 12-53 and 12-54.

Before starting to generate the drawing views, set the working directory to *C:\ProE-WF-4.0\c12*. The part (*.prt*) file and the drawing (*.dru*) file should lie in the same directory or folder.

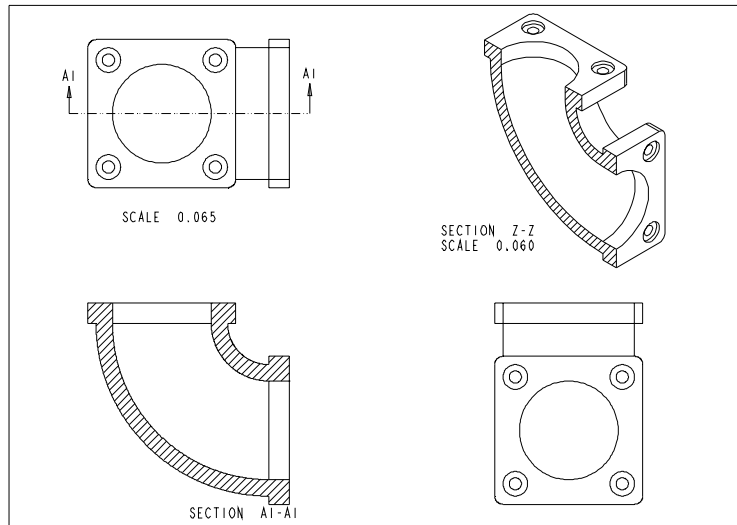


Figure 12-50 Drawing views to be generated

Starting a New Drawing File

To generate the drawing views, you first need to start a new drawing file.

1. Choose the **New** button from the **File** toolbar to display the **New** dialog box.
2. Select the **Drawing** radio button and then enter the name of the file as *c12tut2*.
3. Choose **OK** from the **New** dialog box to display the **New Drawing** dialog box.
4. Choose the **Browse** button to select *c08tut1.prt* from *c08* for generating drawing views.
5. Select the **Empty** radio button from the **Specify Template** area.
6. Choose the **Landscape** button from the **Orientation** area.
7. Select **A4** from the **Standard Size** drop-down list. Choose **OK** from the **New Drawing** dialog box to proceed to the **Drawing** mode.

Generating the Top View

First the top view will be generated. All other views, except the sectioned isometric view, will be the child views of the top view. You need to generate the top view first because the required sectioned front view can be generated only from the top view. The right-side view can also be generated independently, but then this view will not help to generate any other required view.

1. Choose the **General** button from the **Top Toolchest**.

2. Specify the center point for the placement of the top view close to the upper left corner of the drawing sheet; the **Drawing View** dialog is displayed.
3. Select the **TOP** option from the **Model view names** list box and choose the **Apply** button.
4. Choose the **Scale** option from the **Categories** list box. Select the **Custom Scale** radio button and enter **0.065** in the edit box.
5. Choose **OK** to exit the **Drawing View** dialog box.

If necessary, move the view, as shown in Figure 12-51, and choose **No Hidden** from the **Model Display** toolbar. You may also need to repaint the screen.

Generating the Sectioned Front View

The sectioned front view of the model is generated from the top view. Before proceeding further, use the **Plane Display** button from the **Datum Display** toolbar to turn on the display of datum planes and repaint the screen.

1. Choose the **Insert > Drawing View > Projection** from the menu bar.
2. Specify the center point for the placement of the front view below the top view, as shown in Figure 12-51.
3. Select the newly generated view and invoke the shortcut menu. Choose the **Properties** option to display the **Drawing View** dialog box.
4. Select the **Sections** option from the **Categories** list box to display the section related options in the dialog box.
5. Select the **2D cross-section** radio button and choose the **Add cross-section to view** button; the window area below it gets activated. Select the **Create New** from the **Name** drop-down list, if it is not selected; the **XSEC CREATE** menu is displayed.
6. Choose **Planar > Single > Done** from the **XSEC CREATE** menu.
7. Enter the name of the cross-section in the **Message Input Window** as **A1** and press ENTER to display the **SETUP PLANE** submenu. You are prompted to select a planar surface or a datum plane.
8. Select the **FRONT** datum plane (the plane that cuts the part horizontally from the center of the cylindrical feature in the top view) from the drawing area.
9. Scroll the bar in the **Drawing View** dialog box to the right edge. Click on the field below the **Arrow Display** column. You will be prompted to pick a view for arrows where the section is perpendicular. Select the top view to display the arrows.
10. Choose the **Apply** button and then exit the dialog box.

Modifying the Hatching

The offset distance between the hatching lines in the front sectioned view is large. You need to reduce the distance between the hatching lines.

1. Select the **X-Section** filter from the **Filter** drop-down list in the **Status Bar**. Select the hatching lines from the sectioned front view in the drawing sheet; the hatching lines turn red. Hold the right mouse button to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu; the **MOD XHATCH** menu is displayed.
2. From the **MOD XHATCH** menu, choose the **Spacing** option to display the **MODIFY MODE** submenu.
3. The spacing between the hatching lines has to be reduced. Choose the **Half** option twice and then choose **Done** in the **MOD XHATCH** menu. Now, the hatching appears to be more dense.
4. Click once in the drawing area to remove the X-hatch from the current selection set. The sheet, after placing these two views, should look similar to the one shown in Figure 12-51.
5. Set the selection filter back to **Drawing Item and View**.

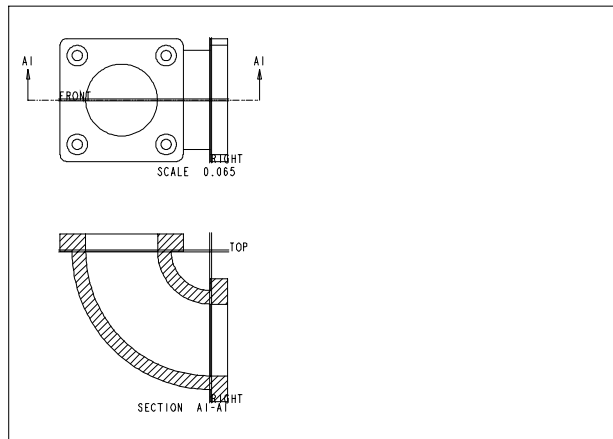


Figure 12-51 Drawing sheet after generating the top view and the sectioned front view with the display of datum planes turned on

Generating the Right-Side View

The right-side view is the projection of the sectioned front view. Before proceeding further, turn off the display of the datum planes.

1. Choose **Insert > Drawing View > Projection** from the menu bar; you will be prompted to select the projection parent view.

2. Select the sectioned front view as the parent view.
3. Specify the center point for the placement of the drawing view on the right of the sectioned front view. The right-side view of the model is placed, as shown in Figure 12-52.

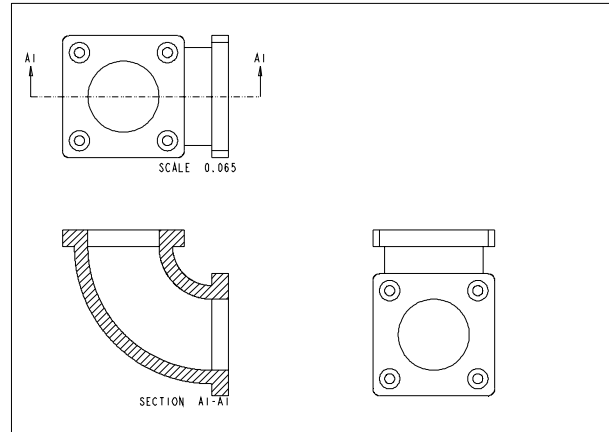


Figure 12-52 The drawing sheet after generating the top, sectioned front, and right-side views

Generating the Isometric Section View

The isometric section view is an independent view and it will be generated by using the **General** option.

1. Choose the **General** button from the **Top Toolchest**.
2. Specify the center point for the placement of the view close to the upper right corner of the drawing sheet; the **Drawing View** dialog is displayed.

The default view is a trimetric view but you need the isometric view to be displayed. Therefore, you need to change the orientation.

3. Select the **Isometric** option from the **Default orientation** drop-down list; the isometric view of the model is displayed on the drawing sheet.
4. Choose the **Scale** option from the **Categories** list box. Enter scale factor value as **0.06** by selecting the **Custom scale** radio button and click the **Apply** button.
5. Select the **Sections** option from the **Categories** list box to display the section options in the dialog box.
6. Select the **2D cross-section** radio button and then choose the **Add cross-section to view** button. The collector below it gets activated. Select **Create New** from the name drop-down list; the **XSEC CREATE** menu manager is displayed.

7. Choose **Offset > Both Sides > Single > Done** from the **XSEC CREATE** menu.
8. Enter the name of the section as **Z** in the **Message Input Window** that appears and press ENTER. Once you have specified the name of the section, a separate window is displayed with the model.
9. Select the **TOP** datum plane from the subwindow. You may need to turn on the display of the datum planes. Choose **Okay** from the **DIRECTION** submenu; the **SKET VIEW** submenu is displayed.
10. Choose the **Right** option from the **SKET VIEW** submenu and select the **RIGHT** datum plane from the subwindow.
11. Choose the **Sketch > Line > Line** from the menu bar and draw the line, as shown in Figure 12-53. This line creates a section plane.
12. Choose the **Constrain** option from the **Sketch** menu; the **Constraints** dialog box is displayed.
13. Choose the **Create same points, points on entity or collinear constraint** button from the **Constraints** dialog box. Align both the ends of the line to the edges.
14. Choose **Sketch > Done** from the menu bar.
15. Choose the **Apply** button to place the view and then exit the dialog box.

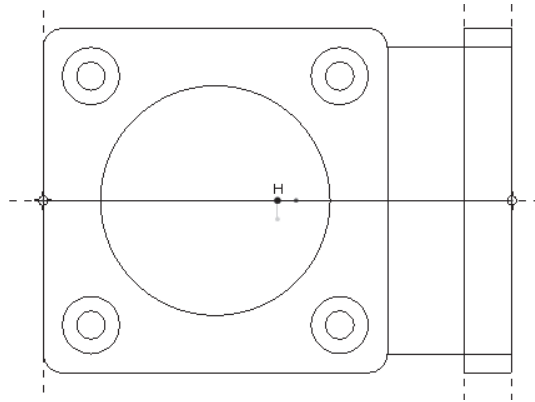


Figure 12-53 The sketch for the section line with the constraint



Note

If the drawing view is placed on the sheet such that it overlaps the boundary of the drawing sheet, then you can move the drawing view. Select the drawing view; the selected drawing view is enclosed in an red box. Now, press the left mouse button inside the box and drag it to place it at the desired location.

Modifying the Hatching

The spacing between the hatching lines in the sectioned isometric view is large. Therefore, you need to reduce the distance between the hatching lines. Use the same procedure that was discussed earlier in this tutorial to modify the spacing between the hatch lines. The sheet, after placing all views, should look similar to the one shown in Figure 12-54.

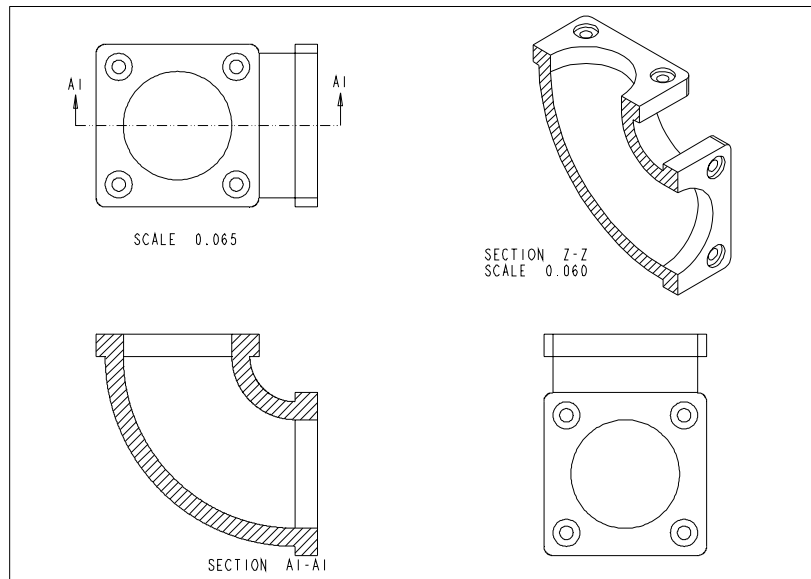


Figure 12-54 Different drawing views generated

Saving the Drawing File

Save the drawing file that you have created.

1. Choose the **Save** button from the **Top Toolchest**; the **Save Object** dialog box is displayed with the name of the drawing file that you entered earlier.
2. Press ENTER to confirm the saving of the file.

Closing the Drawing File

After you have saved the drawing file, close the drawing file.

1. Choose **Window > Close** from the menu bar; the drawing window is closed.

Self-Evaluation Test

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

1. The bidirectional associative nature of a software package means when one file related to the part model is modified, the corresponding modification can be seen in other related files. (T/F)
2. General view is the first view that is generated in the sheet. (T/F)

3. The **Full** section view is the most widely used section view. (T/F)
4. Section views are generally created for the models having features that are not clearly visible in the standard view. (T/F)
5. Broken views are used for the parts that have a high length to width ratio. (T/F)
6. The _____ view is used when you want to show a particular portion of the view in a section and at the same time, do not section the remaining view.
7. The _____ option allows you to temporarily remove the selected drawing view from the sheet.
8. The _____ option is used to redisplay the drawing views that are erased using the **Erase View** option.
9. Using the _____ dialog box, you can reorient the general view.
10. The cross-section hatching on the sectioned portion can be modified using the _____ menu.

Review Questions

Answer the following questions:

1. Which of the following options, when chosen, displays all the erased drawing views on the sheet?
 - (a) **Delete**
 - (b) **Resume View**
 - (c) **Move**
 - (d) **Erase**
2. Which of the following buttons in the **View** toolbar is used to refresh the screen?
 - (a) **Zoom Out**
 - (b) **Zoom In**
 - (c) **Redraw the current view**
 - (d) **None**
3. Which of the following options, when selected, displays only the area of the section view that is sectioned?
 - (a) **Total**
 - (b) **Area**
 - (c) **Align**
 - (d) **None**
4. Which of the following options is used to permanently remove a drawing view from the sheet?
 - (a) **Resume**
 - (b) **Erase**
 - (c) **Delete**
 - (d) **Move**

5. Which of the following options is used to modify the numeric value associated with the drawing views?
- (a) **Xhatching** (b) **Any Item**
(c) **Value** (d) **None**
6. The view type of an existing view can be changed. (T/F)
7. You can reorient only the general views. (T/F)
8. You can flip the side of the cross-section views. (T/F)
9. The **Orientation** area in the **New Drawing** dialog box is available only when you select the **Empty** radio button from the **Specify Template** area. (T/F)
10. The orientation of a model saved in the **Part** mode can be used to orient the drawing view in the **Drawing** mode. (T/F)

Exercise

Exercise 1

Generate the drawing views of the model created in Tutorial 3 of Chapter 8 on an A4 size sheet. The model is shown in Figure 12-55. The drawing views to be generated are shown in Figure 12-56. (Expected time: 45 min)

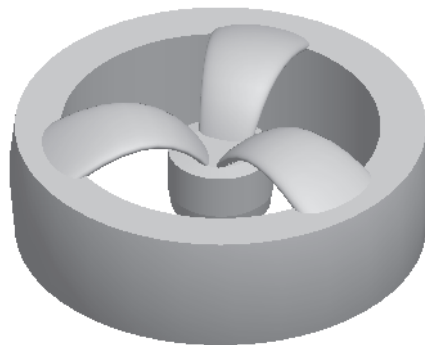


Figure 12-55 Part for generating the drawing views

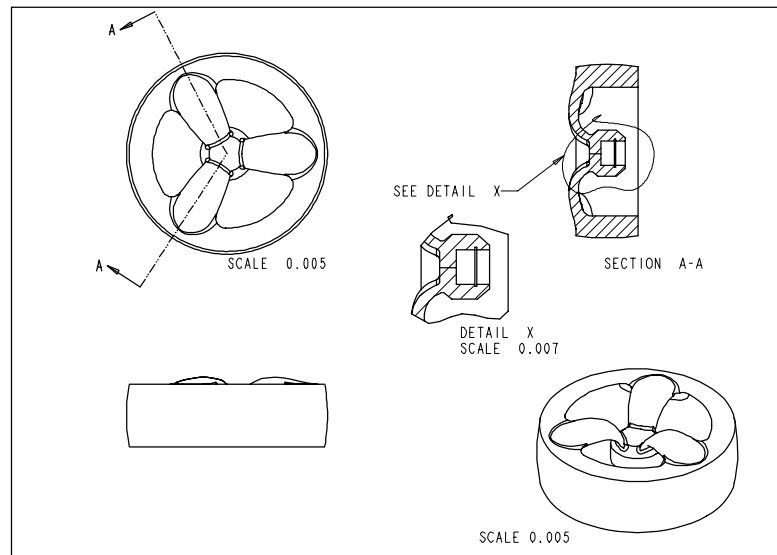


Figure 12-56 Drawing views to be generated in Exercise 1

Answers to Self-Evaluation Test

1. T, 2. T, 3. T, 4. T, 5. T, 6. Local Section, 7. Erase, 8. Resume View, 9. Drawing View, 10. MOD XHATCH