

# **Chapter 10**

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## **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 previous chapters you have learned about creating the parts in the Part mode and assembling different parts in the Assembly mode. One of the major advantages of working with this software package is its bidirectional associative nature. This is the property that ensures that if any modifications are made in the model in the Part mode, its drawing views are updated automatically and if any modifications are made in the dimensions of the drawing in the drawing views, the model gets updated automatically. In Pro/ENGINEER there are two types of drafting methods: Interactive drafting and Generative drafting. In this chapter you will learn about generating the drawing views of the parts or the assemblies that you have created. In Pro/ENGINEER, the drawing views are generated in the Drawing mode. Choose the **Create a new object** button from the **File** toolbar to display the **New** dialog box as shown in Figure 10-1.

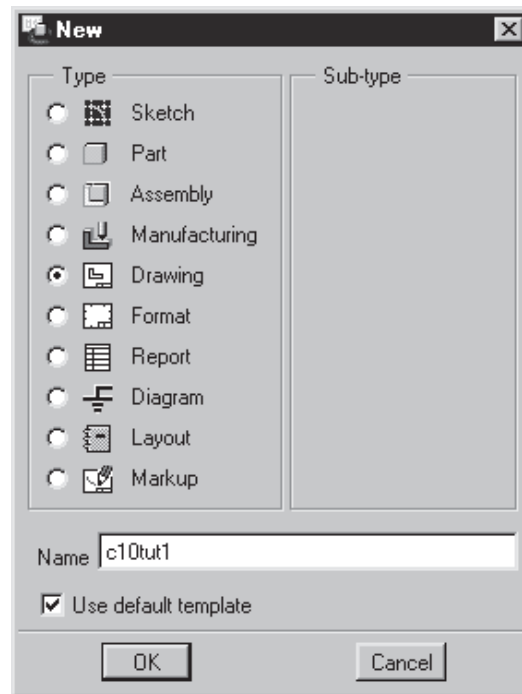


Figure 10-1 New dialog box

Select the **Drawing** radio button in 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 shown in Figure 10-2.

### New Drawing Dialog Box Options

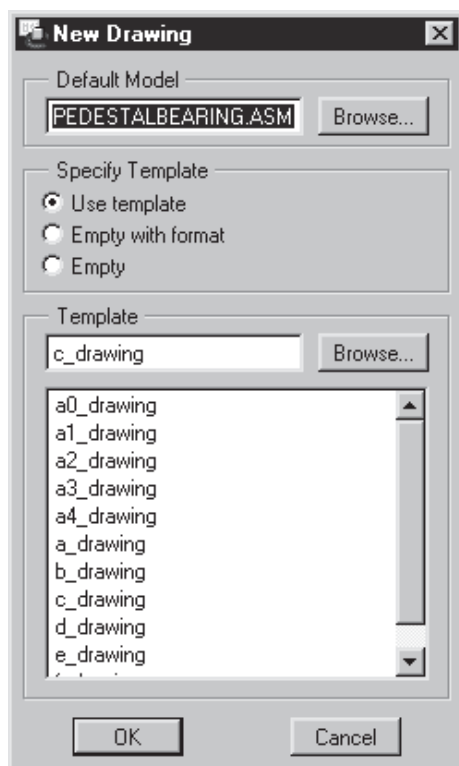
#### Default Model Area

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 is displayed when you choose the **Browse** button. If a

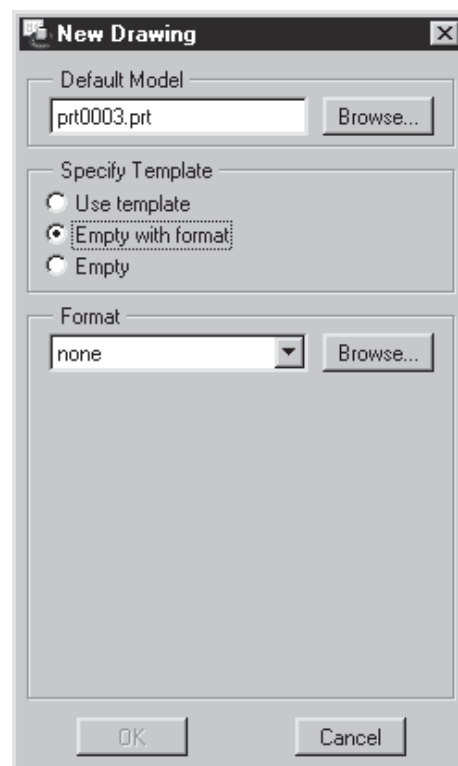
model exists in another window then the name of the model is displayed by default in the **Name** edit box of the **Default Model** area.

### Specify Template Area

The **Specify Template** area is used to specify that weather you want to use the default templates available in Pro\ENGINEER, use predefined formats, or use an empty sheet. There are three radio buttons in this area. The **Use template** radio button is selected by default. When you select the **Empty with format** radio button, the dialog box changes as shown in Figure 10-3. When you select the **Empty** radio button, the **New Drawing** dialog box is modified as shown in Figure 10-4.



*Figure 10-2 New Drawing dialog box with the Use template radio button selected*



*Figure 10-3 New Drawing dialog box with the Empty with format radio button selected*

### Orientation Area

The **Orientation** area is available only when you select the **Empty** radio button from the **Specify Template** area. The buttons provided 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. The size of the sheet and its units can be set using the **Size** area.

## Size Area

The **Size** area is available only when you select the **Empty** radio button from the **Specify Template** area. The drop-down list in this area is used to specify the sheet size.

### Standard Size

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

### Inches/Millimeters

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 select the **Variable** button from the **Orientation** area.

### Width/Height

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 select the **Variable** button from the **Orientation** area.

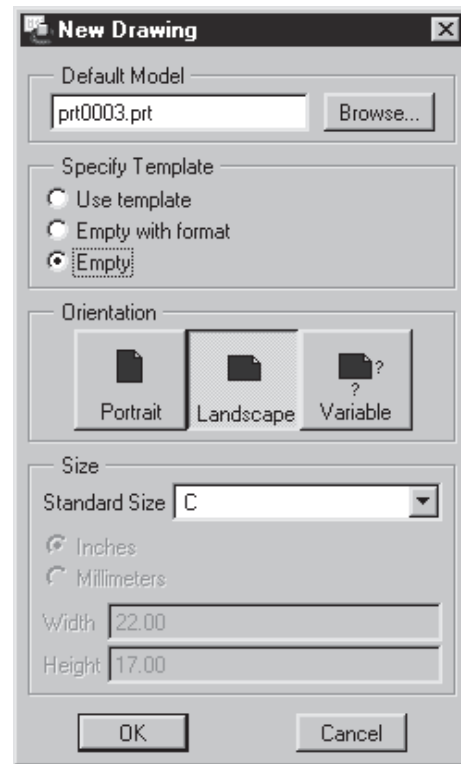


Figure 10-4 New Drawing dialog box with the **Empty** radio button selected

## Format Area

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

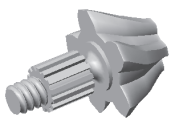
### Format

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 five 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.



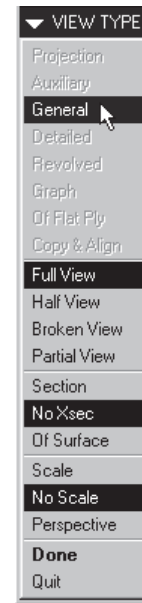
**Tip:** If you had not specified any model in the **Default Model** area of the **New Drawing** dialog box and choose **OK**. When you choose the **Views** option from the **DRAWING** menu in the **Menu Manager**, the **Open** dialog box is displayed and you can select the model from the **Open** dialog box.

## GENERATING THE DRAWING VIEWS

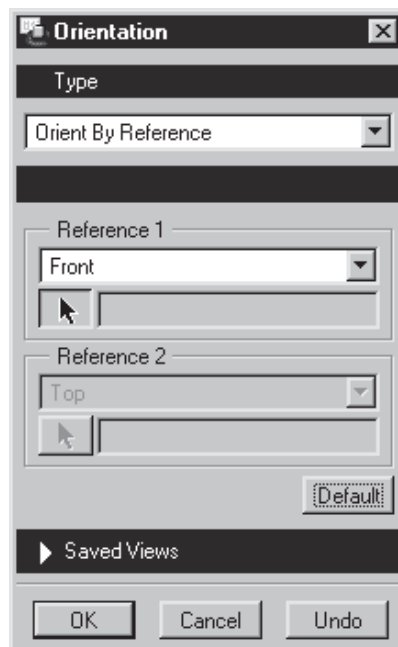
To generate drawing views in Pro/ENGINEER, choose the **Views** option from the **DRAWING** menu in the **Menu Manager**. The **VIEW TYPE** submenu is displayed as shown in Figure 10-5. All the views that can be generated by using the options from the **VIEW TYPE** submenu are discussed next.

### General View

The **General** view is generated with the help of two references and is similar to orienting the model in the Part mode. This view is also used as the parent view for generating the other views. You can specify any desired scale factor for these views. The **General** option in the **VIEW TYPE** submenu and some other options are selected by default as shown in the Figure 10-5. The other options are discussed later in the chapter. Accept the default options and choose **Done**. You are prompted to specify the center point for the drawing view. After specifying the center point of the view on the graphics screen, the view is first placed with the default orientation and then the **Orientation** dialog box is displayed as shown in Figure 10-6. You can orient the view using datum planes or the planar faces of the part or use a view saved in the Part mode.



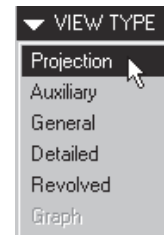
*Figure 10-5 Selecting the General option from the VIEW TYPE submenu*



*Figure 10-6 Orientation dialog box to orient the General view*

## Projection View

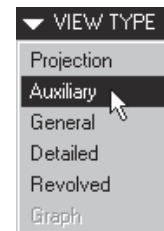
The projection views are the orthographic views generated by projecting the normal lines from an existing general view. The **Projection** option shown in Figure 10-7 is available only after you have generated a general view. The scale factor of these views will be the same as that of the parent view from which they are generated. Depending upon the center point of the projection view with respect to the parent view, the resultant view will be the top, front, left, or the right view. If there exists more than one view that can be the parent view of the projection view, then you will be prompted to specify the parent view for the new view. These views move along with the parent view if their parent view is moved from its original location.



*Figure 10-7 Selecting the **Projection** option from the **VIEW TYPE** submenu*

## Auxiliary View

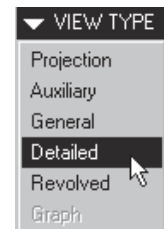
Auxiliary views are mainly used where you want to show the true length of an inclined surface. The auxiliary views are created by projecting normal lines from a specified edge, an axis, or a datum plane in an existing view. The view scale will be the same as that of the parent view. The **Auxiliary** view option is available in the **VIEW TYPE** menu as shown in Figure 10-8.



*Figure 10-8 Selecting the **Auxiliary** option from the **VIEW TYPE** submenu*

## Detailed View

Detailed views are used to provide the enlarged view of a particular portion of an existing view. When you choose the **Detailed** option from the **VIEW TYPE** menu as shown in Figure 10-9, you will be prompted to specify the scale for the detailed view. You will also be prompted to specify the center point and sketch a spline in the parent view to define the outline of the detailed view. You are also prompted to specify the name of the detailed view, the boundary type, and a point to specify the location for the placement of note.



*Figure 10-9 Selecting the **Detailed** option from the **VIEW TYPE** submenu*

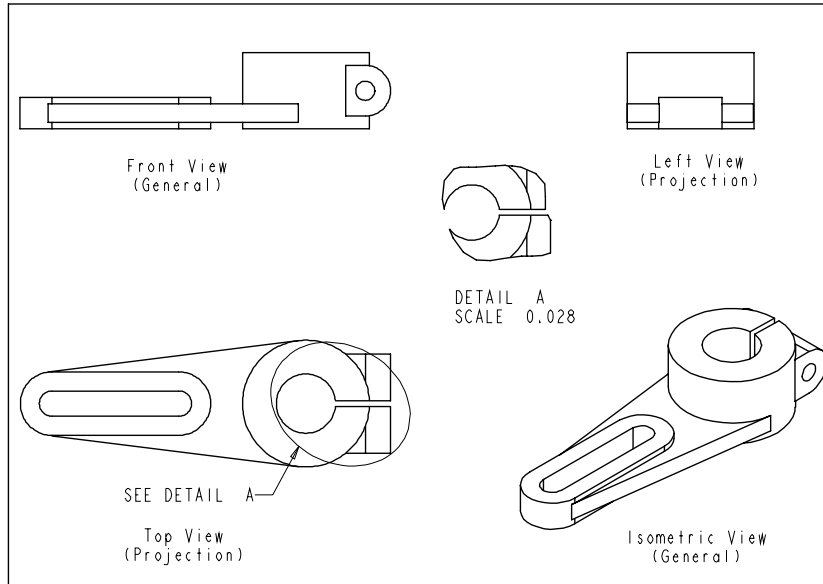
Figure 10-10 shows the different drawing views.

## Revolved View

Figure 10-11 shows the **Revolved** option available in the **VIEW TYPE** menu. These are the section views that are 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 views is normal to the viewing plane.

## Graph View

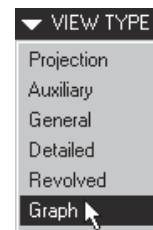
Figure 10-12 shows the **Graph** option that is available only when you have created a datum graph in the Part mode of the current drawing. This view is used to display the datum graphs in the drawing views.



*Figure 10-10 The different drawing views*



*Figure 10-11 Selecting the **Revolved** option from the **VIEW TYPE** submenu*



*Figure 10-12 Selecting the **Graph** option from the **VIEW TYPE** submenu*

You can also combine these standard view types with some more options to generate some other views. You can use the following options and combine them with the standard view types to control their visibility. The options are discussed next.

### Full View

The **Full View** option can be combined with any of the view type to generate a drawing view displaying the complete part.

### Half View

The **Half View** option can be combined with the **Projection**, **Auxiliary**, or the **General** view type to create a drawing view that displays only half the part. When you place this view, 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 perpendicular to the screen in the new view. This option is available only for **Projection**, **Auxiliary**, and **General**

views. This type of view is generated for symmetric parts.

### Broken View

The **Broken View** option can be combined with the **Projection** or the **General** view type to generate a view that is broken along the horizontal or the vertical direction using the horizontal or vertical lines. This type of view is used for the parts have high length to width ratio. Remember that if the view is projected vertically, the number of the vertical break lines should be zero in the view and similarly, if the view is projected horizontally, the number of horizontal break lines should be zero in the view.



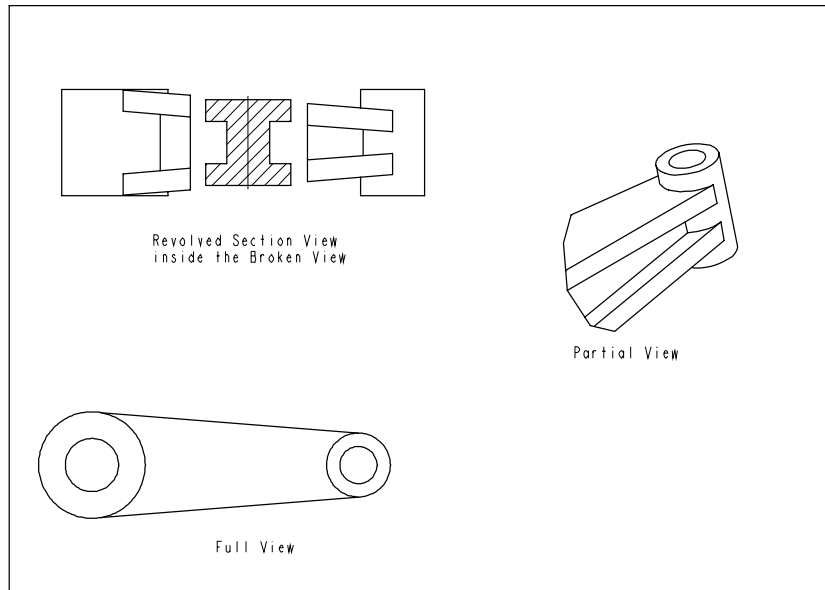
#### Note

*Remember that while creating the **Broken** view, you cannot zoom or pan even if the size of the object requires it. Therefore, it is recommended to fit the drawing size on the graphics screen as required before creating the **Broken** view.*

### Partial View

The **Partial View** option can be combined with the **Projection**, **Auxiliary**, **Revolved**, or **General** view type to generate the view that displays a specified portion of the view. When you choose to place this view, you are prompted to draw a spline that will be the boundary of the portion of the view you want to display.

Figure 10-13 shows the different views.



*Figure 10-13 The different drawing views*

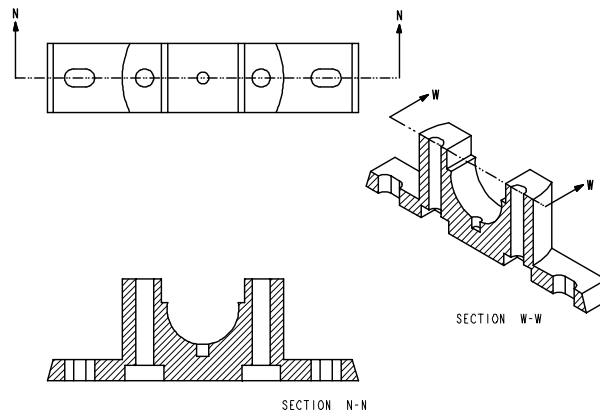
## SECTIONAL VIEWS

The sectional 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 resultant section view is displayed. Pro/ENGINEER allows you to generate the following types of section views by choosing them from the **XSEC TYPE** menu.

### Full Section View

Consider a part that is cut throughout its length, width, or height and the front cut portion (front half) is removed from the display. The remaining half portion when projected is called a full section view. When you choose to place a full section view, you are prompted to create a new cross section or retrieve an existing cross section. This is the most widely used type of section views. Figure 10-14 shows the full section view.



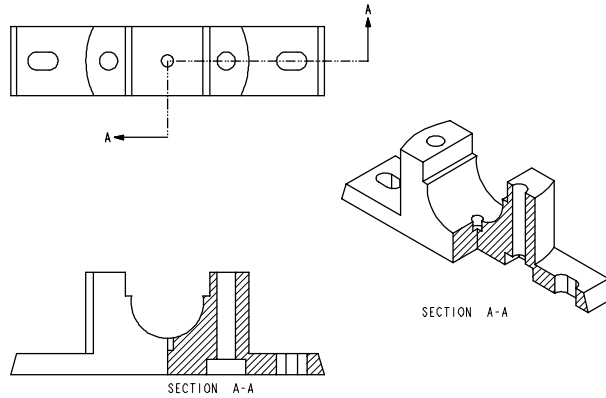
**Figure 10-14** Top view, Full section front view, and Full section isometric view

### Half Section View

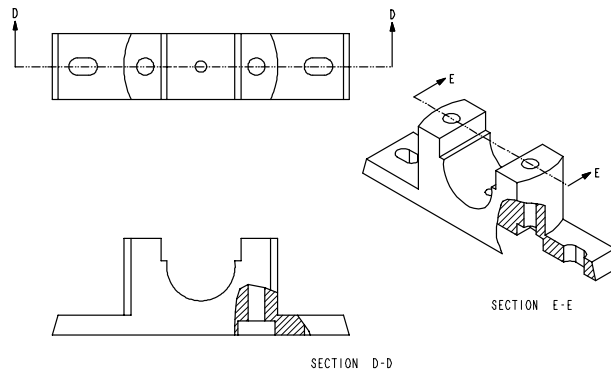
Consider a part that is cut half way through the length, width, or height and the front cut portion (front quarter) is removed from the display. This part when projected is called the half section view. In this view, only half of the part is sectioned and the other half of the part is displayed as it is. You need to specify the half side of the view that has to be sectioned. Remember that this option is available only when you select **Full View** from the **VIEW TYPE** submenu. Figure 10-15 shows the half section view.

### Local Section View

The **Local Section** view is used when you want to show a particular portion of the view in section and at the same time not section the remaining view. The local section area is specified by drawing a spline around it. Figure 10-16 shows the local section view.



**Figure 10-15** Top view, Half section front view, and Half section isometric view



**Figure 10-16** Top view, Local section front view, and Local section isometric view



#### Note

As evident from Figure 10-14 and Figure 10-16, to create a sectioned isometric view you have to define a section plane other than that used to generate the front section view.

### Full & Local

The **Full & Local** view is the combination of the full and local section views. This option allows you to generate a full and a local section view in the same view.

### © Advanced Sectioning Options

You can combine the above-mentioned section types with other advanced sectioning options

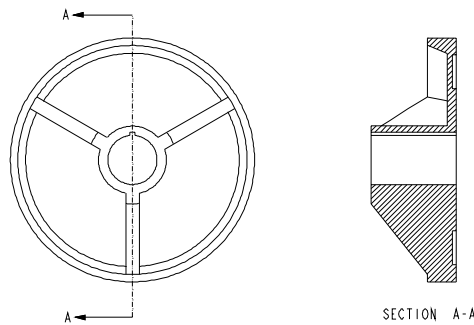
to generate the section views. The advanced sectioning options that can be used are discussed next.

### Total Xsec

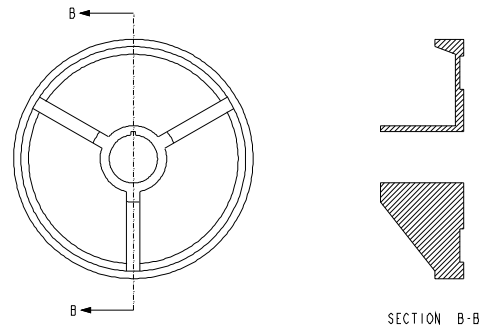
The **Total Xsec** option when selected creates a section view that displays all the visible edges of the section view in addition to the section area. This option can be combined with the full, half, local, or the full and local Xsec types. Figure 10-17 shows the front and total sectioned left view.

### Area Xsec

This option 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 shown in Figure 10-18. This option can be combined with only the full Xsec type.



**Figure 10-17** The front view and the Total cross section view



**Figure 10-18** The front view and the Area cross section view

### Align Xsec

These views are 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 parallel to the view plane. Remember that the axis about which the feature is straightened, should lie on all the cutting planes. These views are similar to the area section views in which only the portion that is sectioned is displayed. Figure 10-19 and Figure 10-20 show these views.

### Total Align

These views are similar to the align section views with the only difference being that these views display all the visible edges of the section view in addition to the section portion as shown in Figure 10-21 and Figure 10-22.

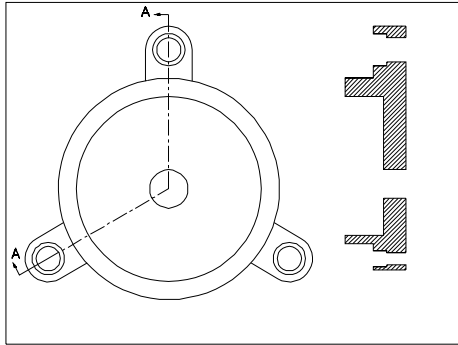


#### Note

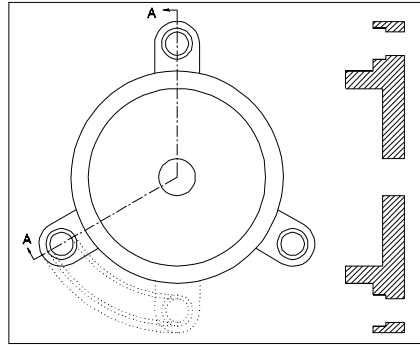
The **Align Xsec** and the **Total Align** options are available only when you choose the **Full** option from the **XSEC TYPE** menu.

### Unfold Xsec

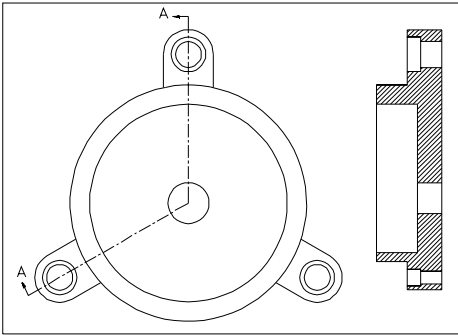
The **Unfold Xsec** option is also used to generate the align section view but only for the **General**



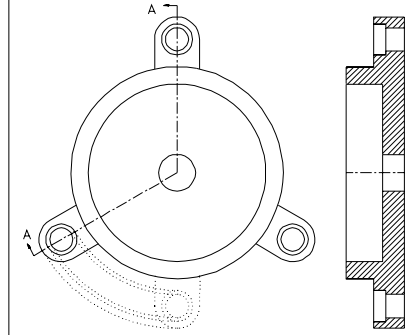
**Figure 10-19** The Area cross section view with normal lines of projection



**Figure 10-20** The Align cross section view with aligned lines of projection



**Figure 10-21** The Total cross section view with normal lines of projection



**Figure 10-22** The Total align cross section view with aligned lines of projection

view type. Once you have placed the view and specified the cutting plane, you will be prompted to specify the orientation of the section view.

### Total Unfold

The **Total Unfold** option is also used to generate the total align section views but only for the **General** view type.

## CREATING THE SECTION PLANES

Once you have selected all the options for generating the section views and specified the location of the section view, you need to specify the cutting plane for the section view. You can either retrieve the cross section created in the Part mode or create the cross section in the Drawing mode by specifying the cutting plane. Pro/ENGINEER allows you to create two types of section planes. They are discussed next.

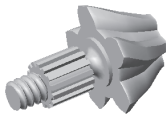
### Planar

The **Planar** option is used to select an existing planar face or the datum plane as the cutting

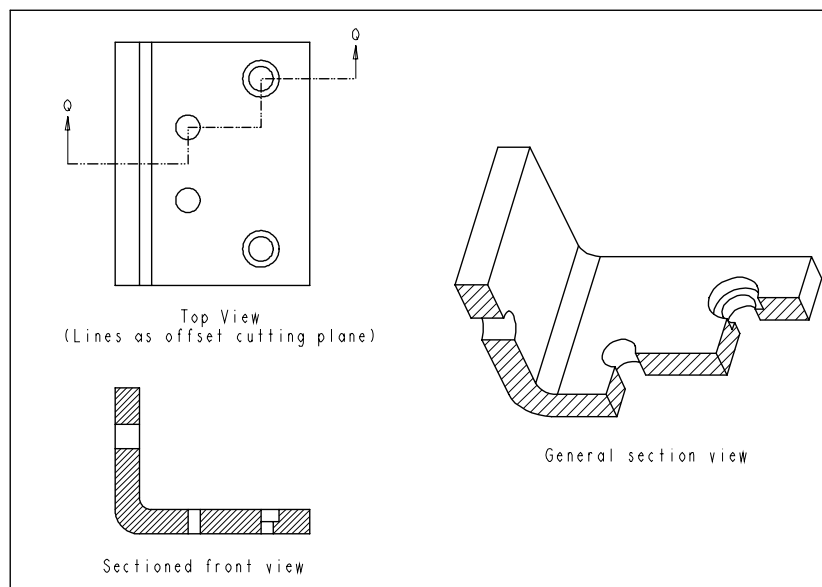
plane. You can also make a datum plane that can be used as the cutting plane. However, remember that you cannot draw a cutting plane using this option.

## Offset

The **Offset** option is used to sketch an offset cutting plane using the sketcher options. When you invoke this option, you are prompted to specify the name of the cross section. Once you have specified the name of the cross section, another window appears displaying the part. You can then follow the Part mode options to draw a line that will act as the cutting plane for generating the section views. This option is generally used for creating the aligned section views. Figure 10-23 shows the top view, offset sectioned front view, and the sectioned general view of a part.



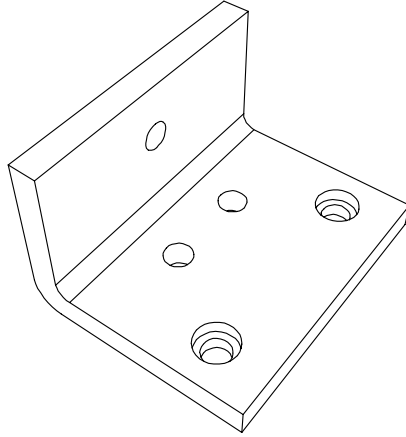
**Tip:** You can also create the cross sections in the Part mode and later retrieve those sections in the Drawing mode and use them for generating the section views.



**Figure 10-23** The section views created using the offset section plane

## PERSPECTIVE VIEWS

Pro/ENGINEER provides you with an option of creating the perspective drawing views of the parts or assemblies. The perspective views are generated using two parameters. They are the eye distance for the view in the model space and the diameter of the view in the paper space. The eye distance in the model space here refers to the distance of the viewer from the actual model in the Part mode and the diameter of the view in paper space refers to the size of the view in the Drawing mode. Figure 10-24 displays a general perspective view.



**Figure 10-24** *A general perspective view*

The following points should be remembered while creating the perspective views:

1. You cannot use perspective views to generate projection or detailed views.
2. Perspective views do not have dimensions, datum planes, and axes.
3. Perspective views cannot have cross sections.
4. Only general views can be displayed in perspective.

## EDITING THE DRAWING VIEWS

Pro/ENGINEER allows you to perform the following types of editing operations on the drawing views.

### Moving the Drawing View

#### **DRAWING > Views > VIEWS > Move View**

When you invoke the **Move View** option, all the drawing views on the sheet are displayed inside a boundary and you are prompted to select the drawing view to be moved. Remember that if you select the view that has some child views then 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

### **DRAWING > Views > VIEWS > Erase View**

The **Erase View** option 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, therefore, you can also erase a view that has some child views associated to it and it will not effect the child view. Once a view is erased, a box is displayed in place of the view displaying the name of the view.



#### **Note**

*When you erase a view, the leaders and dimension values that were attached with the view are also erased. When you select the erased view to resume, the leaders and dimension values are redisplayed.*

## Resuming the Drawing Views

### **DRAWING > Views > VIEWS > Resume View**

The **Resume View** option is used to redisplay the drawing views erased using the **Erase** option. This option is available only when you have erased at least one drawing view.

## Deleting the Drawing Views

### **DRAWING > Views > VIEWS > Delete View**

The **Delete View** option is used to permanently remove a selected drawing view from the sheet. Once the view is deleted, no information related to the deleted view remains in the memory of the drawing. Remember that the view that has some child views associated with it cannot be deleted unless all the child views are deleted first. When you select the drawing view to be deleted, you are prompted to confirm the deletion process.

## Adding New Parts or Assemblies to the Current Drawing

### **DRAWING > Views > VIEWS > Drawing Models > Add Model**

You can also add more parts or assemblies in addition to the default part or assembly for generating the drawing views. This is done using the **Add Model** option. When you invoke this option, the **Open** dialog box is displayed. You can select the new model to be added using this option. Remember that the latest model added will be the current model and the drawing views that will be generated now will be of this model. You can change the current model by choosing **DRAWING > Views > VIEWS > Drawing Models > Set Model**. 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.

## MODIFYING THE DRAWING VIEWS

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

### Changing the View Type

#### **DRAWING > Views > VIEWS > Modify View > View Type**

When you invoke the **View Type** option, you will be prompted to select the drawing view to be

modified. However, this should be noted that only the general, projection, and auxiliary view types can be modified using this option.

## Changing the View Scale

### **DRAWING > Views > VIEWS > Modify View > Change Scale**

The **Change Scale** option is used to change the scale factor of the selected view. You can only modify the scale factor of the views in which the scale factor has been assigned during generation.



#### **Note**

*The scale factor represents the drawing to model scaling. 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.*

## Changing the View Name

### **DRAWING > Views > VIEWS > Modify View > View Name**

The **View Name** option is used to change the name of the selected view. View name appears inside a box when the view is erased using the **Erase** option. You can change this name by using this option. When you invoke this option, you will be prompted to select the view whose name has to be changed.

## Reorienting the Views

### **DRAWING > Views > VIEWS > Modify View > Reorient**

The **Reorient** option is used to reorient the general view. However, if some child view is associated with the general view then the child view will also be reoriented accordingly. The **Orientation** dialog box will be displayed when you select the general view to reorient. You can use the options from this dialog box to change the orientation of the selected view.

## Modifying the Cross Sections

### **DRAWING > Views > VIEWS > Modify View > X-Section**

You can flip the side of sectioning, replace, delete, or rename the X sections in the views using this option.

## Modifying the Boundaries of Views

### **DRAWING > Views > VIEWS > Modify View > Boundary**

The **Boundary** option is used to modify or resketch the boundary of the broken views, partial views, or the detailed views.

## Adding or Removing the Cross Section Arrows

### **DRAWING > Views > VIEWS > Modify View > Add Arrows/Del Arrows**

The **Add Arrows/Del Arrows** options are used to add or delete the cross section arrows from the views. In case the arrows are not displayed, then you will be prompted to select the section views whose arrows are to be displayed and then the view where the arrows should be displayed.

In case the arrows are to be removed, you will be prompted to select the arrows to be deleted.

## Modifying the Perspective Views

### **DRAWING > Views > VIEWS > Modify View > Perspective**

The **Perspective** option is used to modify the perspective views. You can modify the eyepoint distance or the view diameter using this option.

## 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 the drawing views, cross section hatching, and so on. All this is done by selecting the item and right-clicking on it. The shortcut menu is displayed. You can select the options from the shortcut menu to modify that item.

You can modify any parameter associated with the drawing views. Also the selected drawing view can be moved by selecting the view and then moving the mouse. Depending upon the item selected to modify, the options related to it varies. The options in the shortcut menu that are displayed varies from item to item. For example, if you select any text item associated with the drawing views then you can move it from its original location, edit the text, edit text style, and so on. If you select a drawing view and right-click then the modifications that are possible are; changing the location of the view, changing the view scale, renaming the view, and reversing the direction of the sectioning or replacing the cross sectioning (in case of section views).

## Modifying the Values

The **Value** option is used to modify any numeric value associated with the drawing views. The numeric value can be the view scale in case it is provided or changing the dimension value. You can also change the scale of all the views in case it is not available by modifying the scale value provided near the lower left corner of the screen. Choose the **Value** option from the **Edit** menu in the menu bar to change the scale factor for all the views in the current sheet.

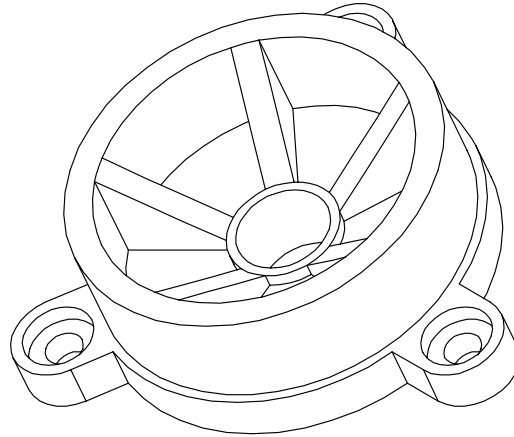
## Editing the Cross Section Hatching

Select the hatching from a drawing view. When the hatching turns red in color, choose the **Properties** option from the **Edit** menu in the menu bar. The **MOD XHATCH** submenu is displayed. 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** submenu.

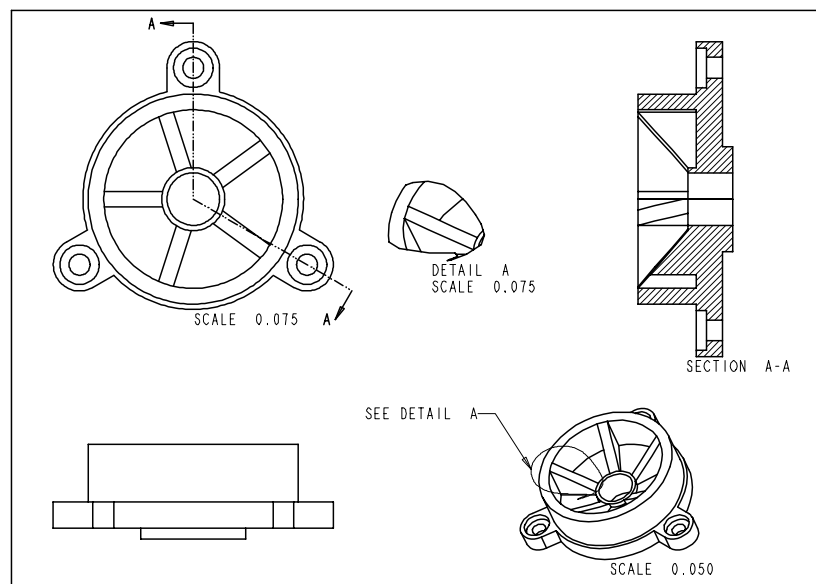
## TUTORIALS

### **Tutorial 1**

In this tutorial you will generate the drawing views of the model created in Exercise 4 of Chapter 6 shown in Figure 10-25. Select the A4 size drawing sheet and generate all the drawing views shown in Figure 10-26. (Estimated time: 40 min)



**Figure 10-25** Part for generating the drawing views



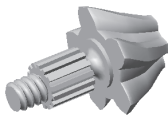
**Figure 10-26** The drawing views to be generated

The following steps outline the procedure to generate the drawing views of the model:

- a. Create a new drawing file and select the required drawing sheet.
- b. Generate the top view.

- c. Generate the front view from the top view.
- d. Generate the sectioned right view of the top view by defining a plane in the model.
- e. Generate the default 3D view of the model.
- f. Generate the detail view from the 3D view.

Before you start generating the drawing views, set the working directory to **C:\ProE\c10**. Copy and paste the model **c06exr4.prt** file in the folder named **c10** from **c06** folder. The drawing that you will generate will be saved in the **c10** directory with an extension **.drw**. The **.prt** file and the **.drw** file should lie in the same directory or folder.



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

### Creating New Drawing File

1. Choose the **Create a new object** 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 **c10tut1**, see Figure 10-27.
3. Choose **OK** from the **New** dialog box to display the **New Drawing** dialog box.
4. Choose the **Browse** button to select **c06exr4.prt** for generating the drawing views.
5. Select the **Empty** radio button from the **Specify Template** area.
6. Choose the **Landscape** button from the **Orientation** area if it is not chosen by default as shown in Figure 10-28.
7. Select **A4** from the **Standard Size** drop-down list. Choose **OK** to proceed to the Drawing mode.

An A4 size sheet with landscape orientation is displayed on the graphics screen.

### 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 front view.

1. Choose the **Views** option from the **DRAWING** menu in the **Menu Manager** to display the **VIEW TYPE** submenu.
2. Choose **Add View > General > Full View > No Xsec > Scale > Done**. You will be

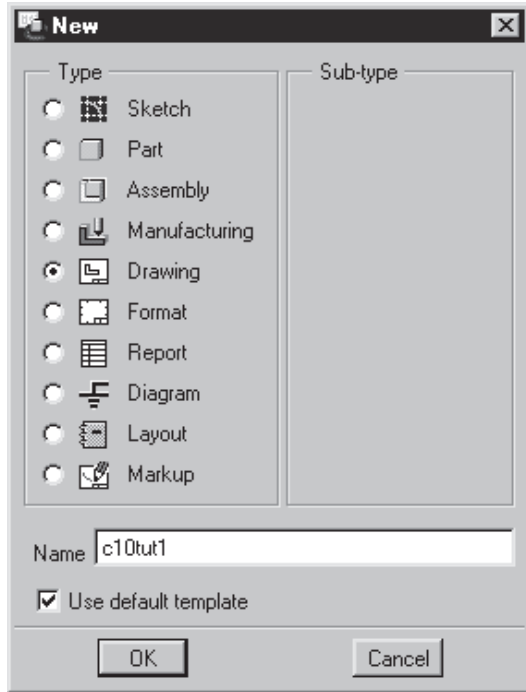
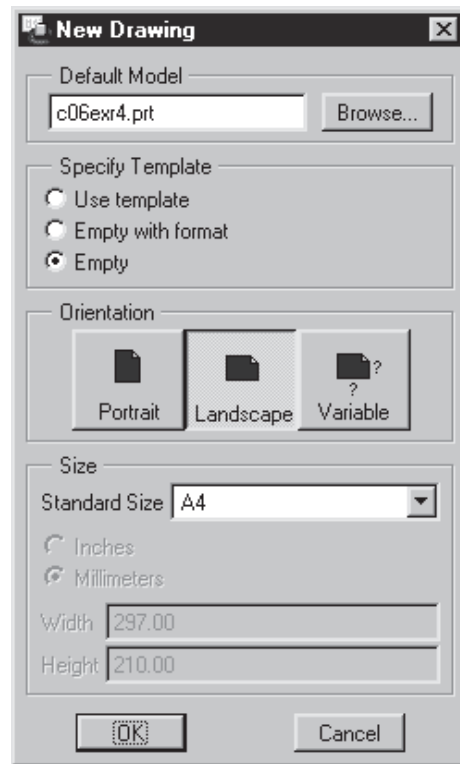


Figure 10-27 New dialog box

Figure 10-28 New Drawing dialog box with the **Empty** radio button selected

prompted to specify the center point for the drawing view.

- Specify the placement point for the first view close to the top left corner of the sheet as shown in Figure 10-29. As soon as you specify the point, the **Message Input Window** is displayed. Enter the scale factor for the view as **0.075** and press ENTER. The default view is placed on the selected point

The **Orientation** dialog box is displayed.



#### 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 created.*

- From the **Saved Views** drop-down list, select the **TOP** option and choose the **Set** button. The default view is oriented as top view.
- Choose the **OK** button from the **Orientation** dialog box.

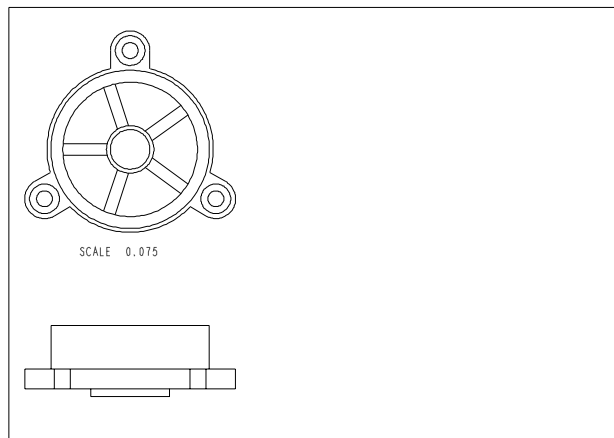
6. Turn off the display of datum planes, axes, points, and coordinate system by choosing their respective buttons from the **Datum Display** toolbar. Also, change the model display to no hidden by choosing the **No Hidden** button from the **Model Display** toolbar.

If in case the view that you have placed on the drawing sheet is not at the proper location on the drawing sheet then you need to move the drawing view. Select the drawing view using the left mouse button. The selected drawing view is enclosed in a red box. Select the view again and move the mouse to place it at the desired location.

### Generating the Front View

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

1. Choose the **Add View** option from the **VIEWS** submenu in the **Menu Manager** to display the **VIEW TYPE** submenu.
2. The **Projection** > **Full View** > **No Xsec** > **No Scale** is selected by default. Choose **Done**.
3. Specify the placement point for the front view below the top view as shown in Figure 10-29.



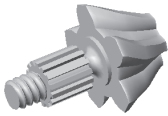
*Figure 10-29 The top and front view*

### Generating the Section View

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

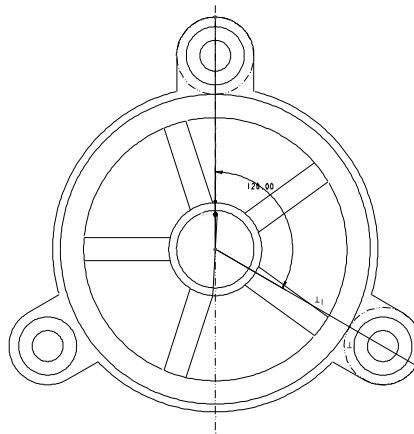
1. Choose the **Add View** option from the **VIEWS** submenu to display the **VIEW TYPE** submenu.
2. Choose **Projection** > **Full View** > **Section** > **No Scale** > **Done** to display the **XSEC TYPE** submenu.

3. Choose **Full > Total Align > Done** from the **XSEC TYPE** submenu.
4. Specify the placement point on the right side of the top view.
5. Choose the **Create** option from the **XSEC ENTER** submenu to display the **XSEC CREATE** submenu.
6. Choose **Offset > Both Sides > Single > Done**.
7. Enter 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.
8. Choose the **Datum planes on/off** 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 to view the datum planes.



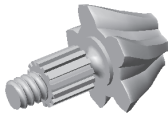
**Tip:** It is recommended to use the **Redraw the current view** button available in the **View** toolbar to remove any temporary information on the graphics screen and to refresh the screen.

9. Select the **TOP** datum plane from the subwindow. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.
10. Select the **Bottom** option from the **SKET VIEW** submenu and choose the **FRONT** datum plane from the subwindow. The **References** dialog box is displayed. Choose the **Close** button to exit the **References** dialog box.
11. Choose the **Create lines.** button from the **Right Toolchest** and draw the lines as shown in Figure 10-30. These lines create a section plane.



**Figure 10-30** Sketch for the Total Align section

12. Align the lines and modify the angular dimension to **120** as shown in Figure 10-30. To modify the dimension value, press the middle mouse button in the subwindow and then double-click on the dimension value.
13. Choose **Sketch > Done** from the menu bar of the subwindow. You are prompted to select the central axis on the new view about which the section will be aligned. You can turn off the display of datum planes.
14. Select the central axis on the new view that you have created. Now, you are prompted to select a view where the section arrows will be placed.
15. Select the top view to display the arrows. You will be prompted to choose a viewing direction. Choose **Okay**. The section view is created.
16. Choose **Done/Return** from the **VIEWS** menu.



**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. If you have two button mouse, use **SHIFT+left** mouse button.

You will notice that the spacing between the hatch lines is large. Therefore, you need to modify the scale of hatching.

### Modifying the Hatching

1. Select the hatching from the drawing sheet. The hatching lines turn red in color. Press and hold the right mouse button to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu.

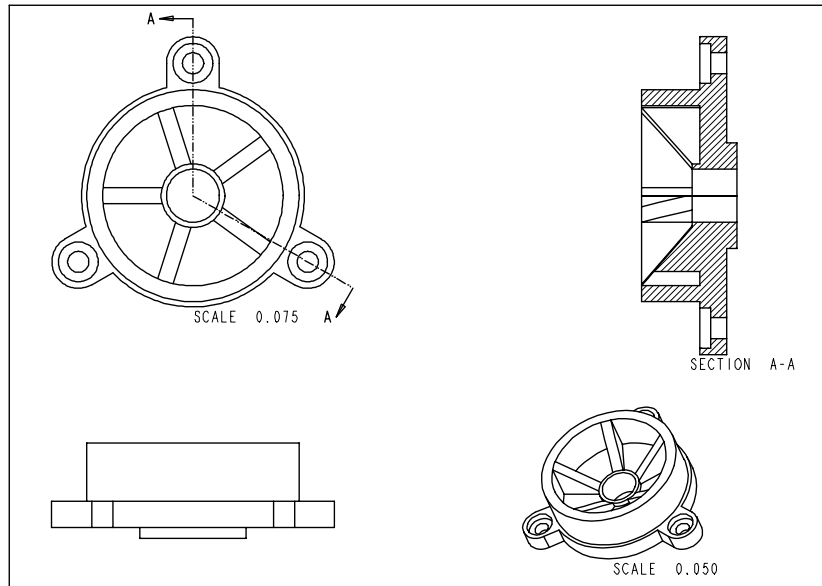
The **MOD XHATCH** submenu is displayed.

2. From the **MOD XHATCH** submenu, 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** submenu. Now, the hatching appears to be more dense as shown in Figure 10-31.

### Generating the General View

The **General** view is created to show a three-dimensional view of the model.

1. Choose **Views** from the **DRAWING** menu in the **Menu Manager**.
2. Choose **Add View > General > Full View > No Xsec > Scale > Done** from the **VIEW TYPE** submenu.
3. Specify the placement point for the general view below the section view as shown in Figure 10-31.



*Figure 10-31 Different drawing views*

4. Enter the scale for the new view in the **Message Input Window** as **0.05** and press ENTER.
5. Choose **Quit Sel** in the **GET SELECT** menu and then choose **OK** from the **Orientation** dialog box.

### 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. Choose **Add View** from the **VIEWS** submenu in the **Menu Manager** to display the **VIEW TYPE** submenu.
2. Choose **Detailed > Full View > No Xsec > Scale > Done**.
3. You will be prompted to specify the placement point for the new view. Select a point between the top view and the sectioned side view using the left mouse button.
4. Enter the value of the view scale as **0.075** in the **Message Input Window** and press ENTER.

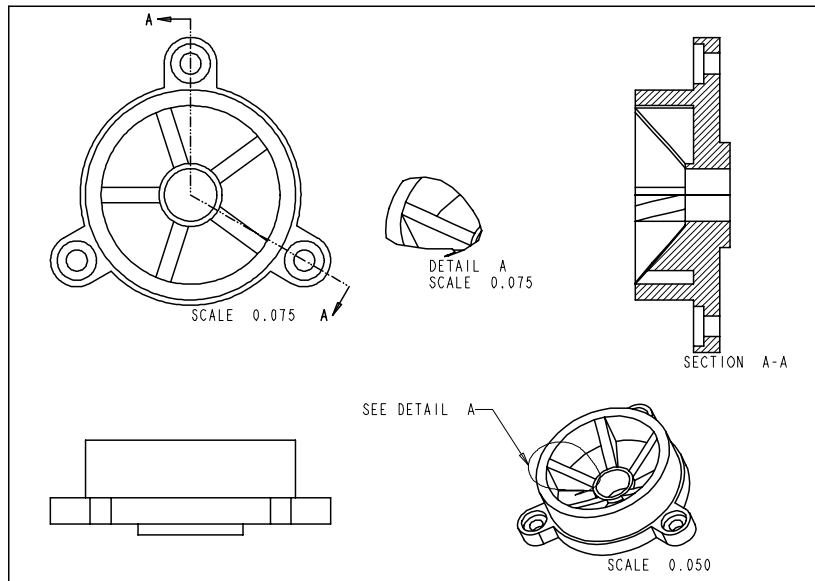
Now, you will be prompted to select a center point. This center point will be the center of the detailed view. Around this center point you will specify the boundary of the detail view.

5. Select the center point for the detail on one of the ribs in the trimetric view, refer Figure 10-32.

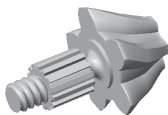
6. Sketch a spline about the center point that you had previously selected. After the spline is drawn, press the middle mouse button to quit the spline creation.
7. The **Message Input Window** is displayed. Enter the name of the detail view as **A** in this window and press ENTER.
8. Choose **Spline** from the **BOUNDARY TYPE** submenu that appears.

Now, you will be prompted to select an attachment point on the spline where the note will be attached and displayed.

9. Select a point on the sketched spline as the attachment point. You are prompted to select a location for note.
10. Select the location of the note somewhere near the spline. You may need to move the note by selecting it and then dynamically moving it. The final sheet after generating all the views should look similar to the one shown in Figure 10-32.



**Figure 10-32** 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 the view using the left mouse button and move it by moving the mouse.

11. Choose **Done/Return** from the **VIEWS** submenu.

### 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 the active object** button from the **Top Toolchest**. The **Message Input Window** is displayed with the name of the drawing file that you had entered earlier.
2. Press ENTER to confirm the saving of the file.

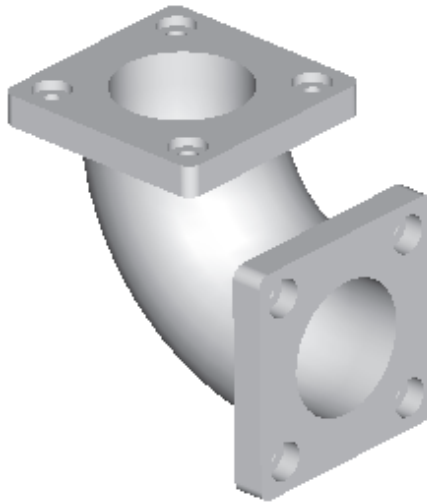
### Closing the Drawing File

After you have saved the drawing file that you have created, you need to close the drawing file and exit the Drawing mode.

1. Choose the **Close** option from the **Window** menu in the menu bar. The drawing window is closed and you have exited the Drawing mode.

## Tutorial 2

In this tutorial you will generate the drawing views of the part created in Tutorial 1 of Chapter 7 shown in Figure 10-33. The drawing views that need to be generated are shown in Figure 10-34. (Estimated time: 40 min)

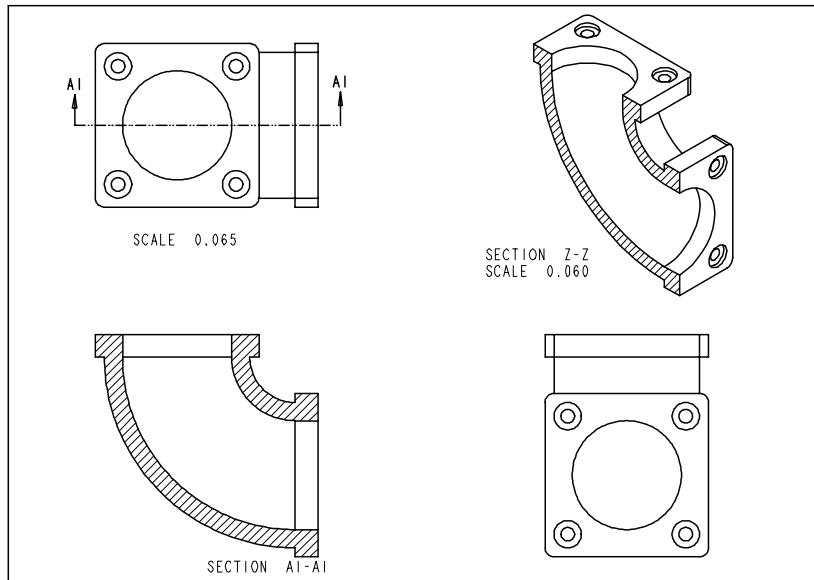


*Figure 10-33 Model for generating the drawing views*

Before you start generating the drawing views, copy the file **c07tut1.prt** from **c07** folder in the current directory.

The following steps outline the procedure to generate the drawing views of the model:

- a. Create a new drawing file and select the size of the drawing sheet.



*Figure 10-34 The different drawing views to be generated*

- b. Generate the top view.
- c. Generate the sectioned front view by defining the **FRONT** datum plane as the section plane.
- d. Generate the right-side view of the sectioned front view by using the **No Xsec** option.
- e. Generate the isometric sectioned view. The section will be defined by drawing a line on the **TOP** datum plane.

Before starting to generate the drawing views, set the working directory to **C:\ProE\c10**. The .prt file and the .drw file should lie in the same directory or folder.

### Creating New Drawing File

Create a new drawing file for generating the drawing views.

1. Choose the **Create a new object** 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 **c10tut2**.
3. Choose **OK** from the **New** dialog box to display the **New Drawing** dialog box.
4. Choose the **Browse** button to select **c07tut1.prt** from **c10** for generating the 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 and all the 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 section front view can only be generated from the top view. The right-side view can be generated independently but then this view will not help to generate any other required view.

1. Choose **Views** from the **DRAWING** menu in the **Menu Manager** to display the **VIEW TYPE** submenu.
2. Choose **General > Full View > No Xsec > Scale > Done**.
3. Specify the placement point for the first view as shown in Figure 10-35. As soon as you specify the point, the **Message Input Window** is displayed. Enter the scale factor for the view as **0.065** and press ENTER. The default view is placed and the **Orientation** dialog box is displayed.
4. Choose the **TOP** option from the **Saved Views** drop-down list in the **Orientation** dialog box.
5. Choose the **Set** button. The default view is oriented as top view.
6. Choose **OK** to exit the **Orientation** dialog box.

If it is necessary move the view as shown in Figure 10-35 and choose **No Hidden** from the **Model Display** toolbar.

### Generating the Sectioned Front View

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

1. Choose **Add View** from the **VIEWS** submenu in the **Menu Manager** to display the **VIEW TYPE** submenu.
2. Choose **Projection > Full View > Section > No Scale > Done** to display the **XSEC TYPE** submenu.
3. Choose **Full > Total Xsec > Done** in the **XSEC TYPE** submenu.

4. Specify the placement point for the front view below the top view as shown in Figure 10-35.
5. When you place the view, the **XSEC ENTER** submenu will be displayed. Choose **Create** to display the **XSEC CREATE** submenu.
6. Choose **Planar > Single > Done**.
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 graphics screen. You are prompted to select a view where you want the arrows to be displayed.
9. Select the top view to display the arrows.
10. Choose **Done/Return** from the **VIEWS** submenu.
11. Turn the datum display off and repaint the screen.

### 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 hatching from the sectioned front view in the drawing sheet. The hatching lines turn red in color. Hold down the right mouse button to invoke the shortcut menu. Choose the **Properties** option from the shortcut menu.

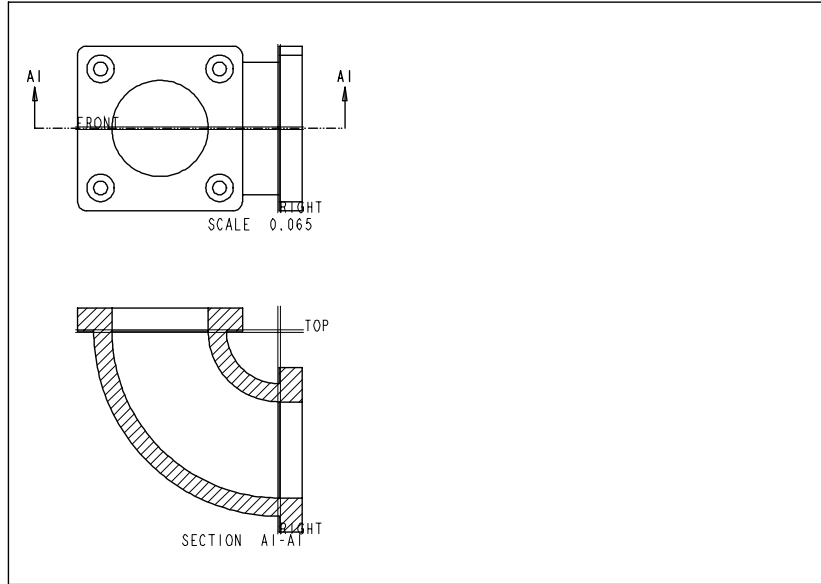
The **MOD XHATCH** submenu is displayed.

2. From the **MOD XHATCH** submenu, 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** submenu. Now, the hatching appears to be more dense.
4. Choose **Done** from the **MODIFY XHATCH** submenu and once click on the graphics screen to unselect the X-hatch.

The sheet after placing these two views should look similar to the one shown in Figure 10-35.

### Generating the Right-Side View

The right-side view is the projection of the front sectioned view. However, you will use the **No Xsec** option to create a view that is not sectioned. Turn off the display of datum planes.



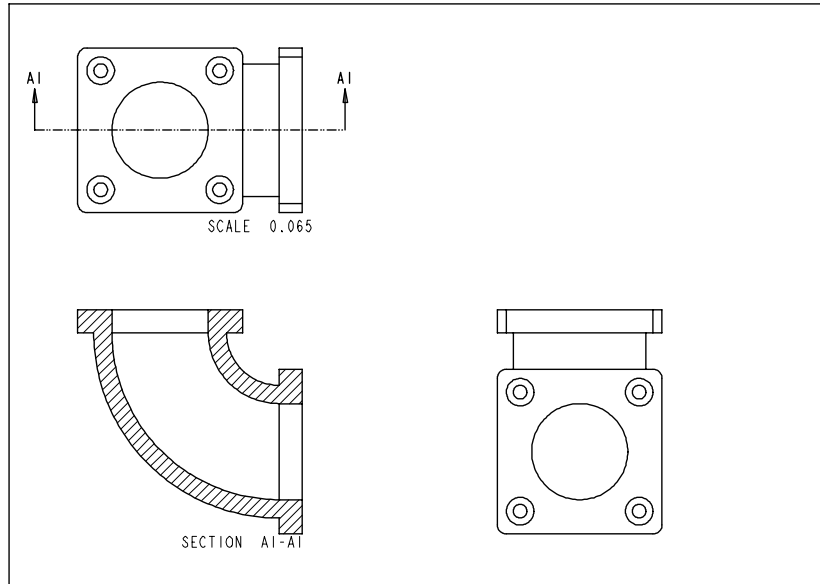
*Figure 10-35 Drawing sheet after generating the top view and the sectioned front view with the display of datum planes*

1. Choose **Views** from the **DRAWING** menu to display the **VIEW TYPE** submenu.
2. Choose **Projection** > **Full View** > **No Xsec** > **No Scale** > **Done**. You are prompted to specify the placement point for the drawing view.
3. Specify the placement point for the view on the right side of the sectioned front view. The right-side view of the model is placed as shown in Figure 10-36.

### Generating the Isometric Section View

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

1. Choose the **Add View** option from the **VIEWS** submenu in the **Menu Manager**. The **VIEW TYPE** submenu is displayed.
2. Choose **General** > **Full View** > **Section** > **Scale** > **Done** from the **VIEW TYPE** submenu. The **XSEC TYPE** submenu is displayed.
3. Choose **Full** > **Total Xsec** > **Done** from the **XSEC TYPE** submenu.
4. Specify a point to the right of the top view and the **Message Input Window** is displayed and you are prompted to specify the scale for the drawing. Enter **0.06** in this window and press ENTER. The default trimetric view of the model is placed and the **Orientation**



**Figure 10-36** Drawing sheet after generating the top view, the sectioned front view, and the right-side view

dialog box is displayed.

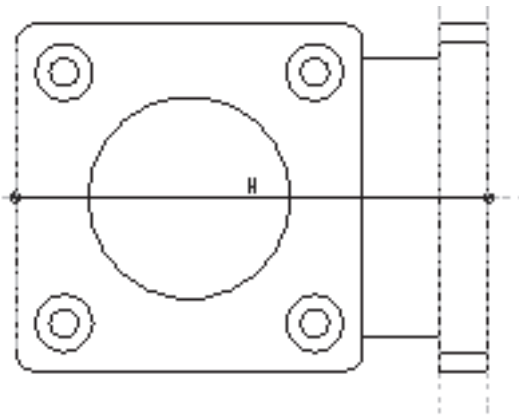
The default view is a trimetric view but you need the isometric view to be displayed. Therefore, you need to change the preferences of the view.

5. Choose the **Preferences** option from the **Type** drop-down list.
6. Choose the **Isometric** option from the **Default Orientation** drop-down list. The isometric view of the model is displayed on the drawing sheet.
7. Choose **OK** in the **Orientation** dialog box to exit it. The **XSEC ENTER** submenu is displayed.
8. Choose the **Create** option from the **XSEC ENTER** submenu to display the **XSEC CREATE** submenu.
9. Choose **Offset > Both Sides > Single > Done** from the **XSEC CREATE** menu.
10. 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 will appear displaying the model.

**Note**

If the model in the Part mode is opened in another window, then 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.

11. Select the **TOP** datum plane from the subwindow. Choose **Okay** from the **DIRECTION** submenu. The **SKET VIEW** submenu is displayed.
12. Select the **Right** option from the **SKET VIEW** submenu and choose the **RIGHT** datum plane from the subwindow. The **References** dialog box is displayed. Choose the **Close** button to exit the **References** dialog box.
13. Choose the **Create lines.** button from the **Right Toolchest** and draw the line as shown in Figure 10-37. This line create a section plane.
14. Choose the **Constrain** option from the **Sketch** menu. The **Constraints** dialog box is displayed.
15. 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.
16. Choose **Sketch > Done** from the **Sketch** menu in the menu bar. Now, you are prompted to select a view where the section arrows will be placed.
17. Press the middle mouse button. The section view is generated but the direction arrows are not displayed.
18. Choose **Done/Return** from the **VIEWS** submenu.



**Figure 10-37** Sketch for the section line with 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. You can move the drawing view by double-clicking on it and then moving the mouse. As you move the mouse the drawing also moves.

### Modifying the Hatching

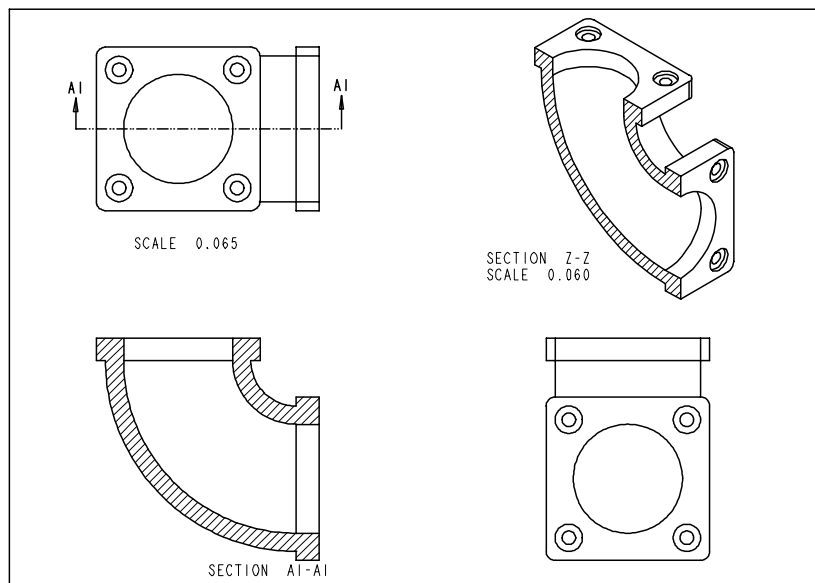
The spacing between the hatching lines in the sectioned isometric view is large. Therefore, you need to reduce the distance between the hatching lines.

1. Select the hatching from the drawing sheet. The hatching lines turn 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** submenu is displayed.

2. From the **MOD XHATCH** submenu, 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** submenu. Now, the hatching appears to be more dense.
4. Choose **Done** from the **MODIFY XHATCH** submenu.

The sheet after placing all the views should look similar to the one shown in Figure 10-38.



*Figure 10-38 The different drawing views to be generated*

### **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 the active object** button from the **Top Toolchest**. The **Message Input Window** is displayed with the name of the drawing file that you had 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 and exit the Drawing mode.

1. Choose the **Close** option from the **Window** menu in the menu bar. The drawing window is closed and you have exited the Drawing mode.

### Self-Evaluation Test

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

1. The bidirectional associative nature of a software package means that 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 type of view. (T/F)
4. The sectional views are generally created for the models that have features that are not clearly visible from standard view. (T/F)
5. Broken views are used for parts that have high length to width ratio. (T/F)
6. The \_\_\_\_\_ view is used when you want to show a particular portion of the view in section and at the same time 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. The \_\_\_\_\_ option is used to reorient the general view.
10. The cross section hatching on the sectioned portion can be modified using the \_\_\_\_\_ submenu.

### Review Questions

Answer the following questions:

1. Which of the following options when selected displays all the erased drawing views on the sheet?
 

(a) <b>Delete View</b>	(b) <b>Resume View</b>
(c) <b>Move View</b>	(d) <b>Erase View</b>

2. Which of the following buttons on 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 that area of the section view that is sectioned?
  - (a) **Total Xsec**
  - (b) **Area Xsec**
  - (c) **Align Xsec**
  - (d) **Total Align**
4. Which of the following options is used to permanently remove a drawing view from the sheet?
  - (a) **Resume View**
  - (b) **Erase View**
  - (c) **Delete View**
  - (d) **Move View**
5. Which of the following options is used to modify any 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 shown in Figure 10-39. Generate the drawing views that are shown in Figure 10-40 on an A4 size sheet.  
(Estimated time: 40 min)

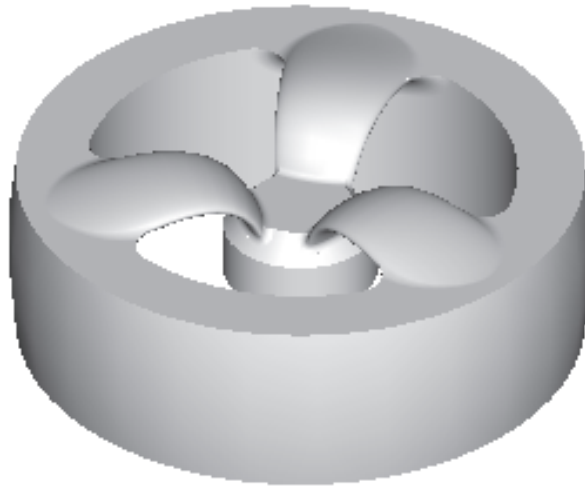


Figure 10-39 Part for generating the drawing views

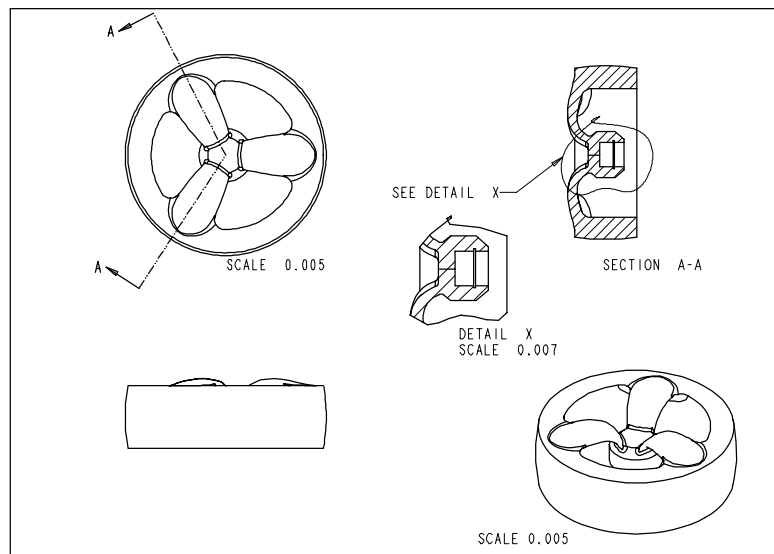


Figure 10-40 Drawing views to be generated in Exercise 1

#### Answers to the Self-Evaluation Test

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