Pro/ENGINEER: Design Documentation and Detailing

Wildfire 5

Student Guide

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Wildfire 5, Revision 1.0

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Chapter 1

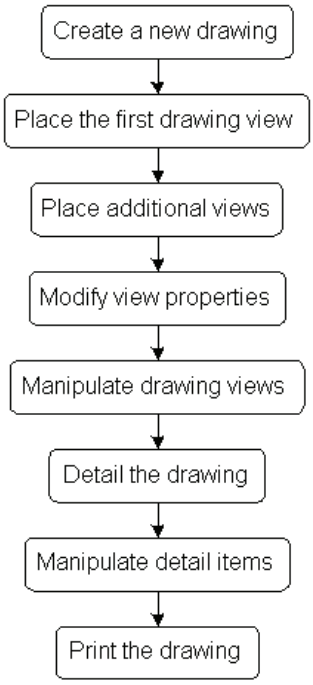
Views

Drawings are created to communicate information on how the model is to be built. Accurate representation of the model is key for manufacturing. In this chapter, you learn how to add the basic model views to a new drawing.

This chapter introduces:

- ✓**General Steps for Drawings**
- ✓**Create a New Drawing**
- ✓**Drawing Mode Interface**
- ✓**Place the First Drawing View**
- ✓**Place Additional Views**
- ✓**Modify View Properties**
- ✓**Drawing Environment Options**

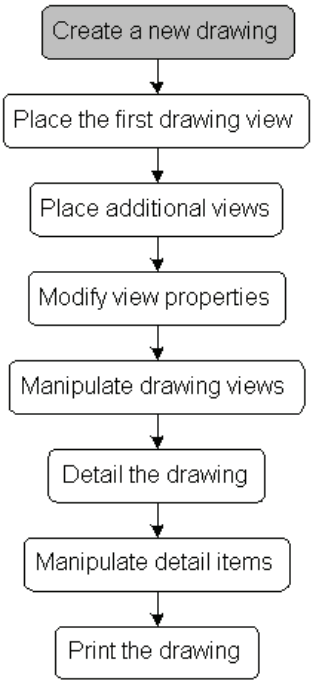
1.1 General Steps for Drawings



In this course, you learn how to create a production drawing of a part or assembly model by progressing through the following steps:

1. Create a drawing.
2. Place the first drawing view.
3. Place additional views.
4. Modify view properties.
5. Manipulate drawing views, as necessary.
6. Detail the drawing (e.g., dimensions, notes, tolerances etc.).
7. Manipulate detail items, as necessary.
8. Print (or Plot) the drawing.

1.2 Create a New Drawing



As with creating a new part or assembly file, you must go through a series of steps.

Complete the following steps to create a new drawing file:

1. Create a new file.
2. Define the default model and the template.
3. Complete the creation of the drawing file.

Step 1 - Create a new file

To create a new drawing, select **File > New** or click  in the toolbar. The New dialog box appears. Select the **Drawing** option in the New dialog box, as shown in Figure 1–1, and enter a name for the drawing.

Click OK.

The Common Name field is optional and enables you to assign a common (user-friendly) name to a new model for use with Windchill.

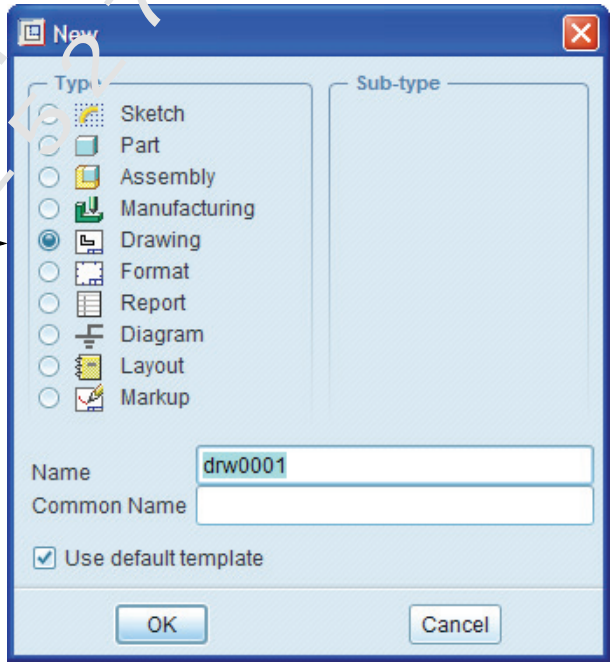


Figure 1–1

Step 2 - Define the default model and the template.

The next dialog box that appears is the New Drawing dialog box, as shown in Figure 1–2. The Default Model field defines the drawing model to be represented in the drawing. To assign the model, enter the model name or click Browse... . By default, only one model can be selected; additional models can be added once the drawing has been created.

You define the template to be used in the drawing once the default model has been defined. The template options include the following:

- **Use Template**
- **Empty with Format**
- **Empty**

Use Template

To use a predefined drawing template, select the **Use template** option and select the template name in the Template section, as shown in Figure 1–2. Alternatively, you can click Browse... and browse to other templates that are stored in other directories.

*If the **Use default template** option is selected in the New dialog box, the New Drawing dialog box defaults to the **Use Template** option, as shown in Figure 1–2. A predefined template is selected for use.*

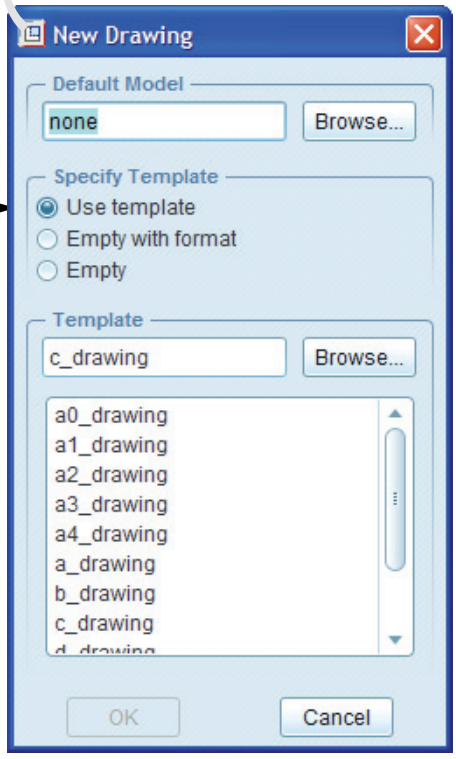


Figure 1–2

Empty with Format

A template can consist of placed views, pre-defined view displays, placed notes, defined tables, and shown dimensions. A format generally contains standard information, such as borders, title blocks, tables, and company information.

To use a predefined drawing format, select the **Empty with Format** option. Enter the name of the format or click Browse... to browse to a predefined format. The dialog box appears as shown in Figure 1–3.

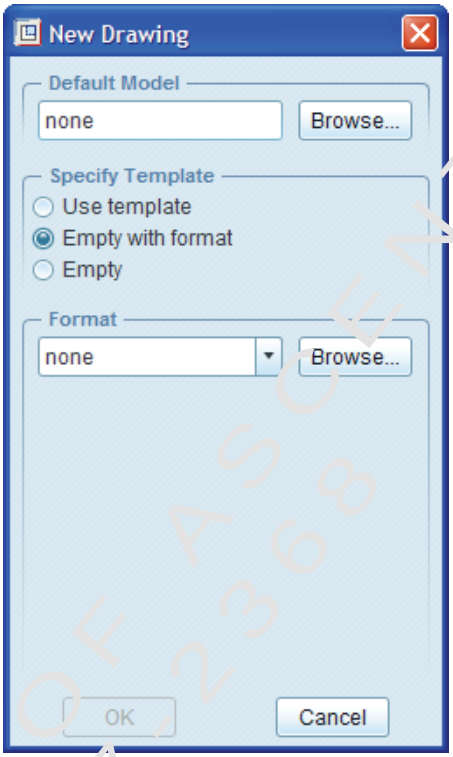


Figure 1–3

Empty

To create a drawing without a template or format, select the **Empty** option and define the drawing size and orientation, as shown in Figure 1–4.

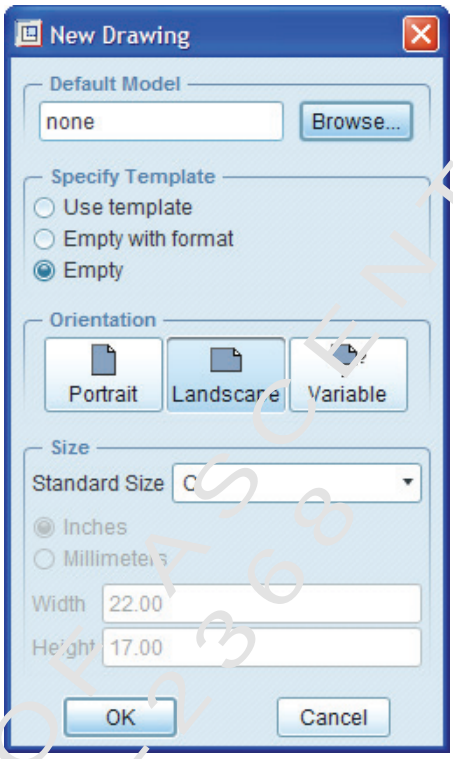


Figure 1–4

Step 3 - Complete the creation of the drawing file.

To create the new drawing, click OK in the New Drawing dialog box.

1.3 Drawing Mode Interface

Once the definition of the drawing and its templates have been completed, the Drawing mode user-interface appears. A variety of areas can be manipulated while working in the Drawing mode. Figure 1–5 illustrates the layout of the drawing environment.

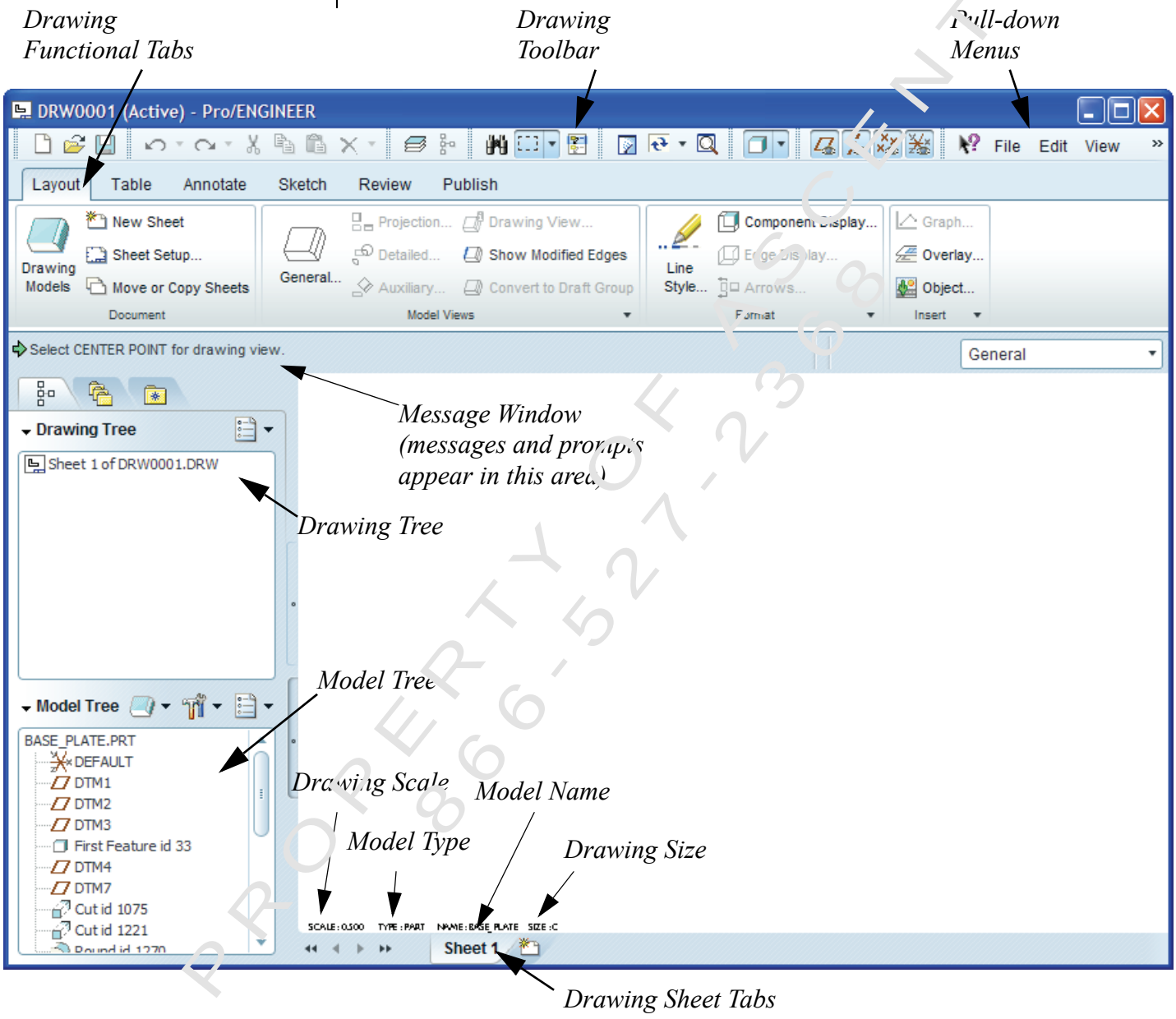


Figure 1–5

Drawing mode uses a ribbon style interface with tabs, as shown in Figure 1–6.

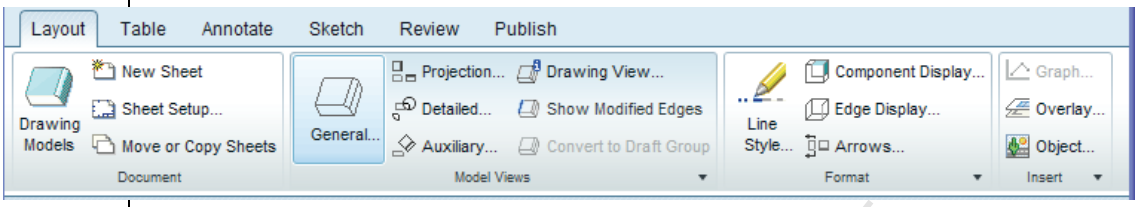


Figure 1–6

Tasks are grouped under tabs, and common icons related to the task are grouped under the tab. For example, all view icons are located in the Model View group under the Layout tab. Only commands that are appropriate for the current task are displayed at any given time. The items available in the selection filter automatically change to suit the current task. The functionality found under each tab is shown in Figure 1–7.

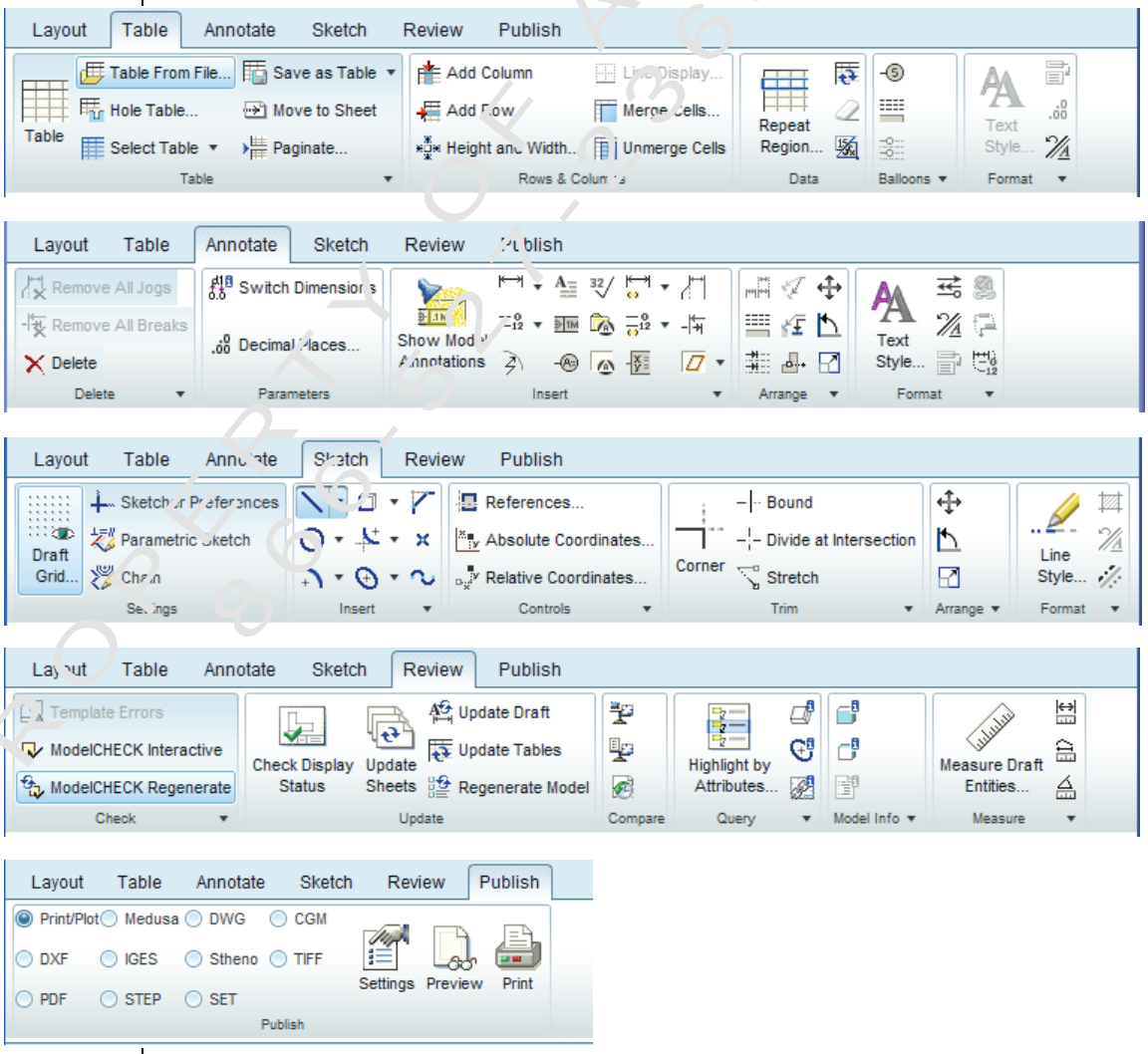


Figure 1–7

Commonly used icons are shown in Table 1–1 and discussed throughout this training guide.

Table 1–1

Icon	Description	Icon	Description
	Delete selected items		Create notes
	Set the active model		Edit hyperlinks
	Update the current sheet		Repeat last formatting
	Regenerate Model		Create geometric tolerance
	Move object to an exact location		Insert drawing symbol
	Create snaplines		Insert custom drawing symbol
	Open the Show/Erase dialog box		Insert general view
	Create dimensions		Insert a table
	Lineup dimensions		Update a table
	Cleanup dimensions		Text style

Pan, Zoom

If you have a scroll wheel, use the following methods to pan and zoom a drawing:

- Press down on the scroll wheel and drag the mouse to pan.
- Roll the scroll wheel to zoom.

If you do not have a scroll wheel, use the following methods to pan and zoom:

- Press the middle mouse button and drag the mouse to pan.
- Hold down <Ctrl> and press the middle mouse button to zoom.

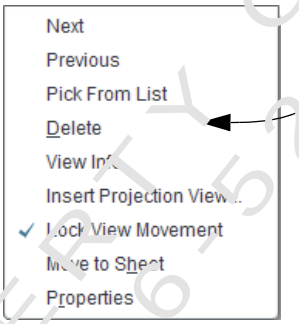
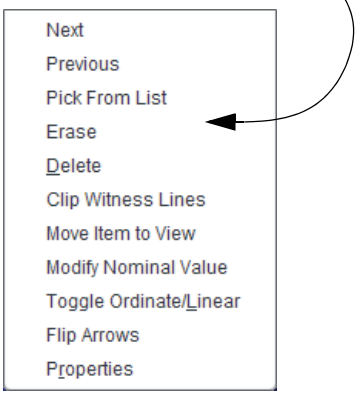
Selection Tool

 in the toolbar provides you with various options for selecting items in a drawing. By default, this option enables you to draw a rectangular box. All items specified by the filter that lie entirely within the rectangular box are selected. To select items that lie across the sketched boundary, click  in the pull-down menu.  enables you to sketch a polygon that defines the selection area. The remaining selection options in this pull-down are available for facet surfaces.

Pop-up Menus

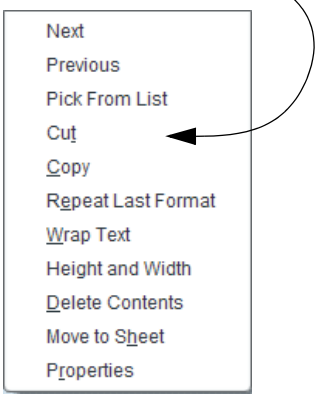
There are many shortcut options available in pop-up menus. To access these shortcuts, select an item and right-click. The options provided in the pop-up menu depend on what item is pre-selected. Table 1–2 shows several possible pop-up menus that appear when different drawing objects are selected.

Table 1–2

Pop-up Menus	Pop-up Menus
This pop-up menu is accessed when a drawing view is pre-selected. 	*This pop-up menu is accessed when a drawing dimension is pre-selected.* 

Pop-up Menus

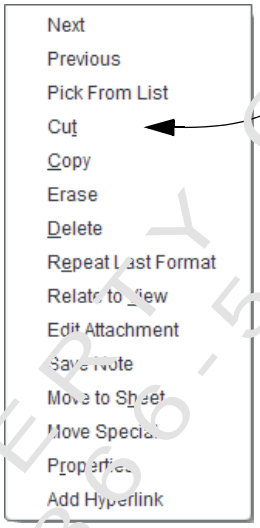
This pop-up menu is accessed when a drawing table cell is pre-selected.



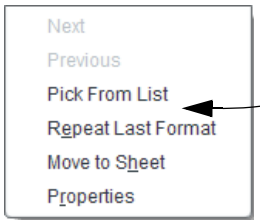
Pop-up Menus

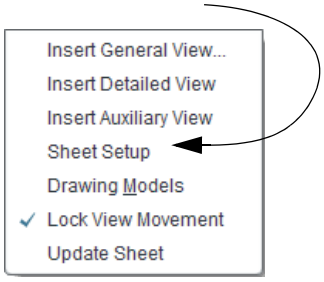
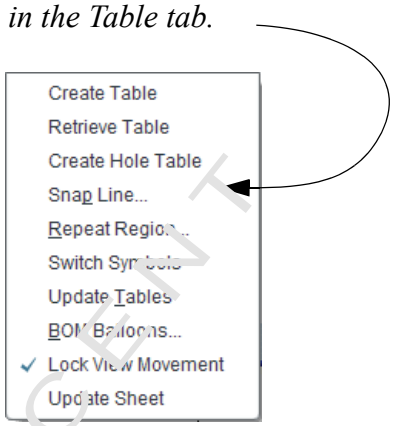
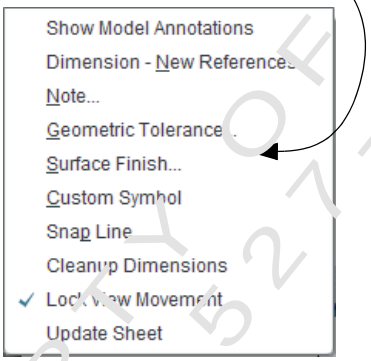
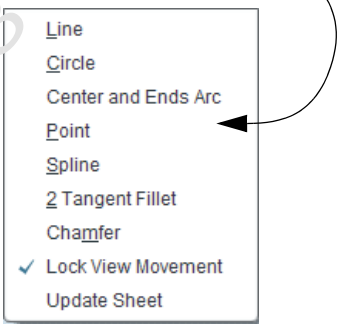
This pop-up menu is accessed when a view cross-section is pre-selected.

This pop-up menu is accessed when a drawing note is pre-selected.



This pop-up menu is accessed when an entire drawing table is pre-selected.



Pop-up Menus	Pop-up Menus
This pop-up menu is accessed when nothing is pre-selected in the Layout tab. 	*This pop-up menu is accessed when nothing is pre-selected in the Table tab.* 
This pop-up menu is accessed when nothing is pre-selected in the Annotate tab. 	*This pop-up menu is accessed when nothing is pre-selected in the Sketch tab.* 

Undo/Redo

The stack represents the list of operations that have been performed on the model.

The Undo/Redo functionality exists for drawings. The following types of operations enable undoing and redoing:

- Supported operations
- Unsupported operations that clear the stack
- Unsupported operations that do not clear the stack

The Undo/Redo functionality supports a limited number of operations. For a detailed list of the detailing operations and how they affect Undo/Redo functionality, go to the Help Center and search in the Detailing functional group for “Undo”. Three different articles that discuss each classification; Supported Undo/Redo Operations, Unsupported Undo/Redo Operations (Non-Clearing), and Unsupported Undo/Redo Operations (Stack Clearing).

By default, the stack limit is 50 operations. Once 50 operations have been stored, the first is removed so that only 50 operations are stored in memory. The stack limit is controlled using the *general_undo_stack_limit* config.pro option.

Drawing Tree

The Model Tree is divided into two parts, the Drawing Tree area and the Model Tree area, as shown in Figure 1–8.

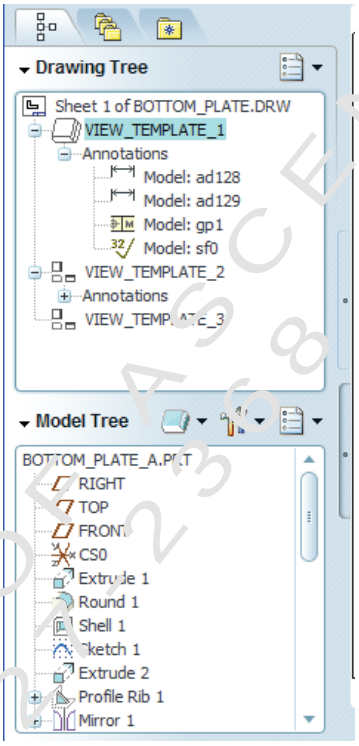
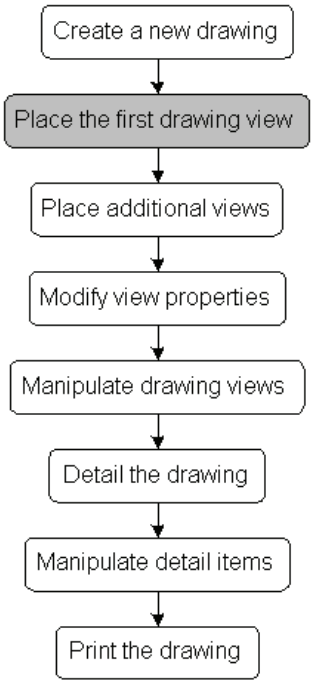


Figure 1–8

The content in the drawing tree changes depending on the tab that is currently active. Selected objects highlight in both the drawing tree and graphics window. Shortcut menus are available by right-clicking. The drawing tree displays the following drawing items:

- Sheets
- Views
- Tables
- Created/shown annotations
- Datums
- Draft entities
- Snap lines
- Sections
- Groups
- Overlays



1.4 Place the First Drawing View

A General view is always the first view that must be placed on a drawing. This view becomes the parent to other views in the drawing.

Complete the following steps to place the first (General) view:

1. Select the Layout tab.
2. Start the creation of the view.
3. Place the view on the drawing sheet.
4. Complete the creation of the drawing view.

Drawings created using a template could already contain certain views. In these situations, you might not need to create the first drawing view; however, you can create additional General views if they are required in the drawing.

Step 1 - Select the Layout tab.

The Layout tab only shows the commands relevant to that function. This tab must be active to create and modify views. The commands for the Layout tab are shown in Figure 1–9.

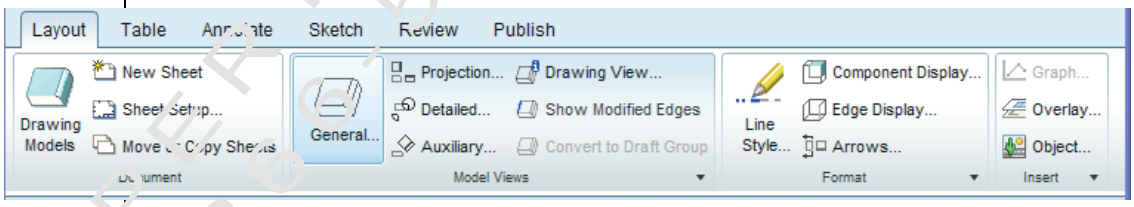


Figure 1–9

Step 2 - SStart the creation of the view.

Use one of the following methods to create a General view:

- Right-click and select **Insert General View**.
- Click 

in the Model Views group from the Layout tab.

Select a location on the drawing to place the view. The General view is initially placed on the drawing sheet in its default 3D orientation. Once the view is placed, the Drawing View dialog box appears as shown in Figure 1–10.

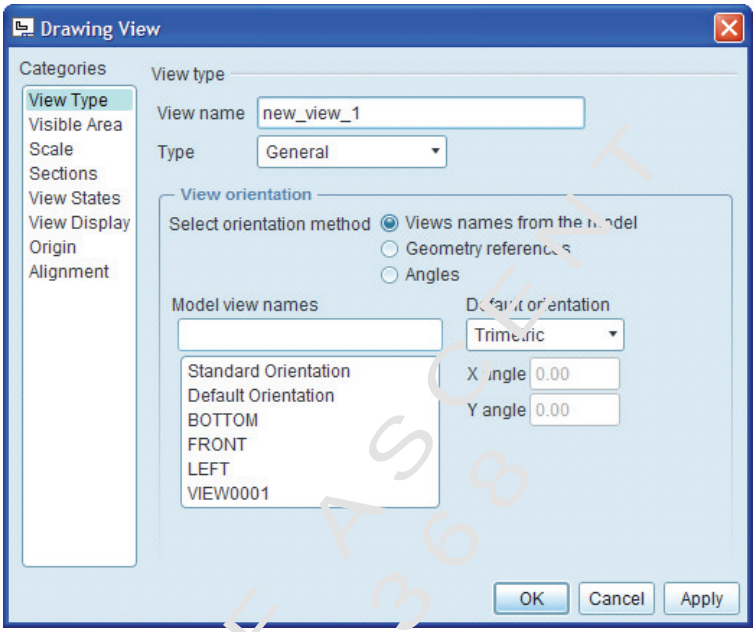


Figure 1–10

The remaining categories in the Drawing View dialog box are discussed later in this training manual.

By default, the View Type category settings are shown in the Drawing View dialog box. This section enables you to enter a name for the view as well as define its view orientation. To modify the view orientation, select one the following orientation options in the View orientation section.

- **View names from the model**
- **Geometry references**
- **Angles**

View Names from the Model

The **View names from the model** option enables you to orient the General view on the drawing using a predefined view saved in the model. The list of predefined views are provided in the Drawing View dialog box, as shown in Figure 1–11.

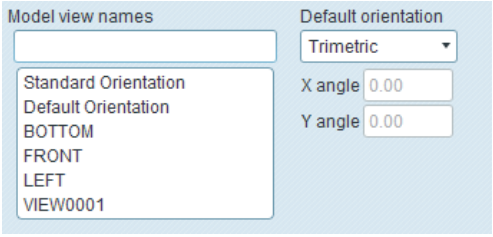


Figure 1–11

Geometry References

The **Geometry references** option enables you to orient the General view using the orientation tools that are used in other 3D models. You must select an orientation (e.g., Front, Top, Right, etc.), then select a planar surface or datum plane as its reference, as shown in Figure 1–12. The two references must be perpendicular to one another to orient the view into 2D.

Default datum planes are recommended to orient the model rather than planar surfaces; if the planar surfaces are deleted later, you lose orientation references.

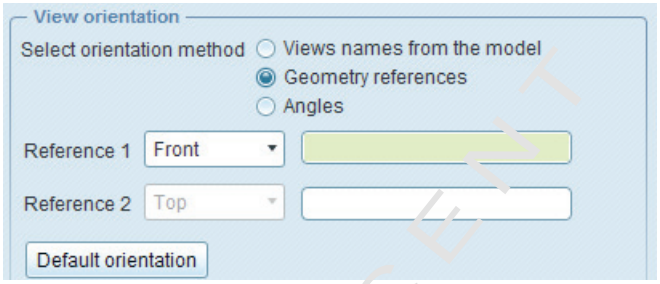


Figure 1–12

You can click Default orientation to return the view to the default orientation.

Angles

The **Angles** option enables you to orient the General view by selecting a direction and entering angular values to place the view. Directions available are Normal, Vertical, Horizontal, and Edge/Axis. The Normal, Vertical, Horizontal directions are relative to the drawing sheet (monitor) and the **Edge/Axis** option enables you to select a reference on the model to orient from. Figure 1–13 shows the Angles section. You can add and remove orientation angles as required using + and −.

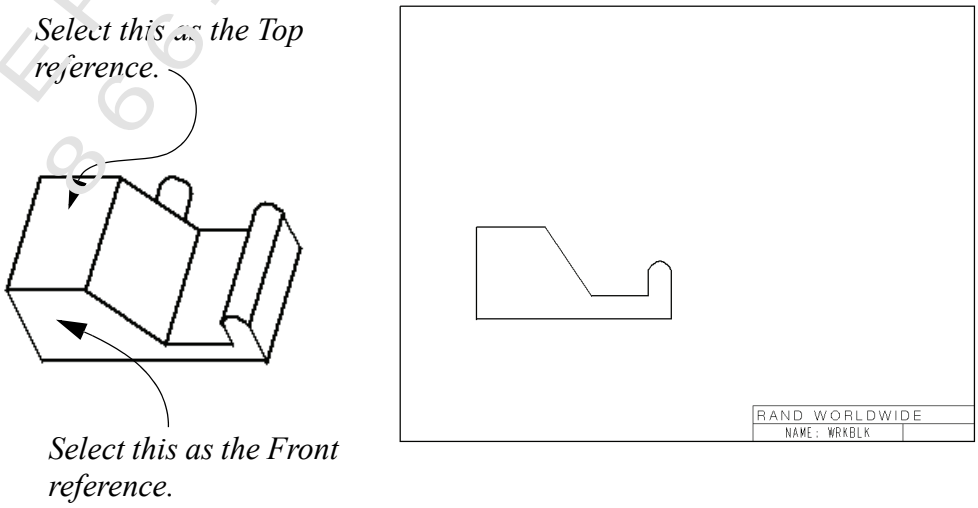


Figure 1–13

To apply new view orientation, select an option and click Apply.

For example, the model in Figure 1–14 is oriented into 2D using the **Geometry references** option and selecting references for the **Front** and **Top** reference options. As an alternative, you can select the **View names from the model** option with a predefined orientation to orient the model the same way.

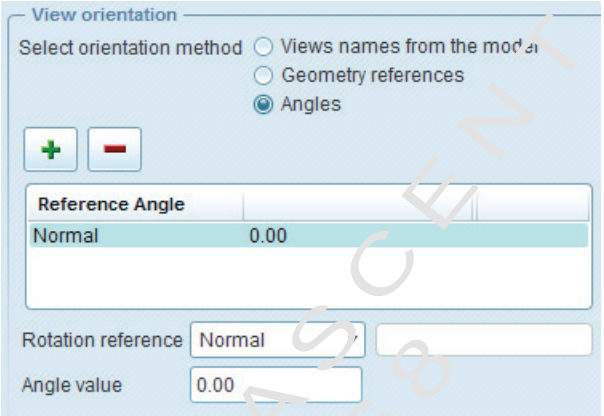
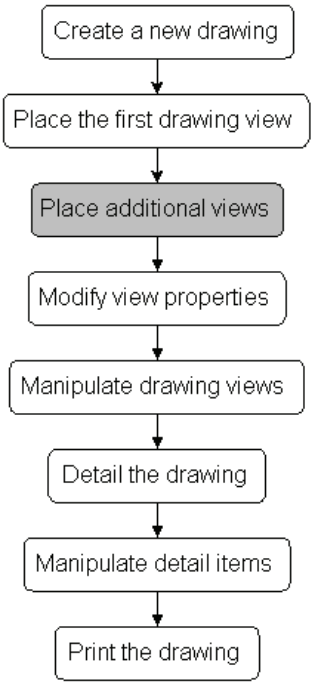


Figure 1–14

Step 3 - Place the view on the drawing sheet.

Once the orientation has been defined, click OK in the Drawing View dialog box to complete the view placement.

Additional options and categories are available in the Drawing View dialog box, which are discussed later in this training course.



1.5 Place Additional Views

Drawings usually consist of multiple views to accurately represent the drawing model.

Complete the following steps to place additional views in a drawing:

1. Select the Layout tab.
1. Start the creation of the view.
2. Place the view on the drawing sheet.
3. Complete the creation of the drawing view.

Step 1 - Select the Layout tab.

The Layout tab only shows the commands relevant to that function. This tab must be active to create and modify views.

Step 2 - Start the creation of the view.

Once a General view has been added to the drawing, you can create additional General views or other types of views that reference the General view. The view icons are found in the Layout tab in the Model Views group, as shown in Figure 1–15.

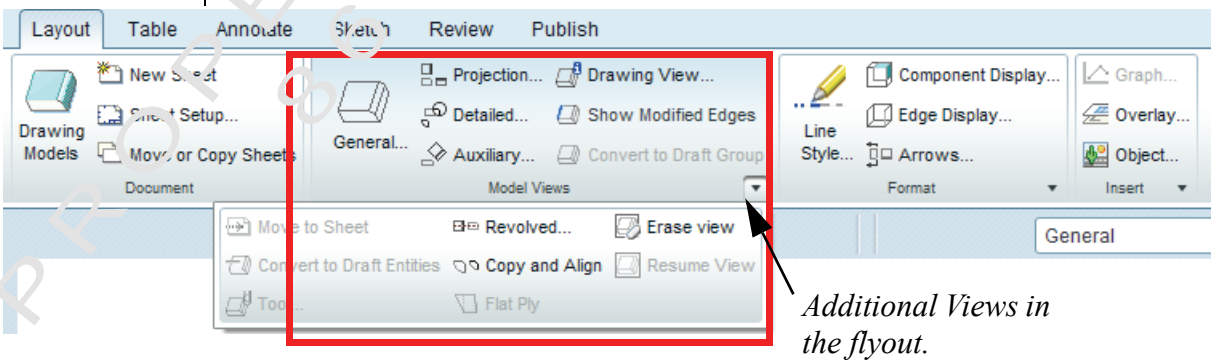
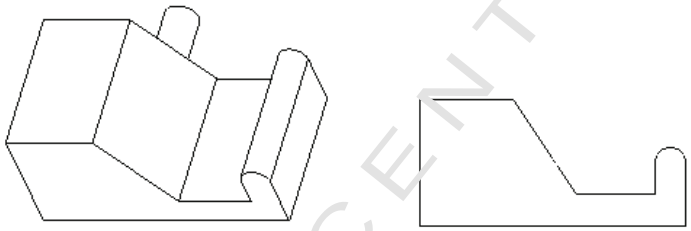
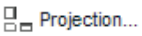
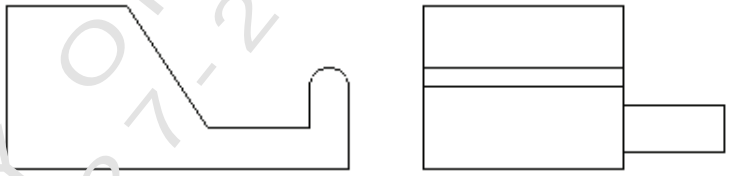
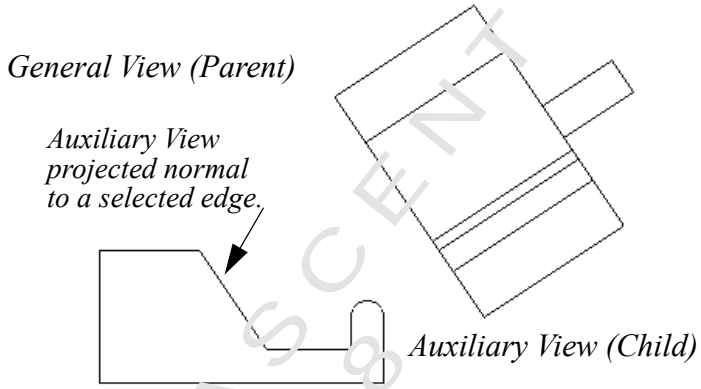
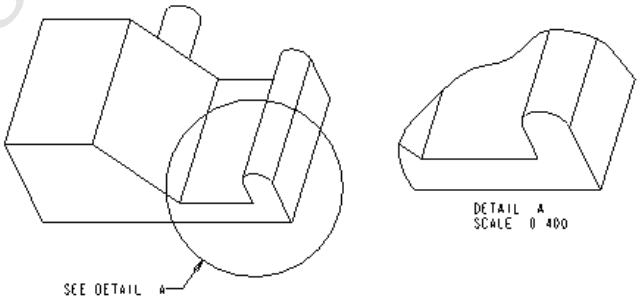


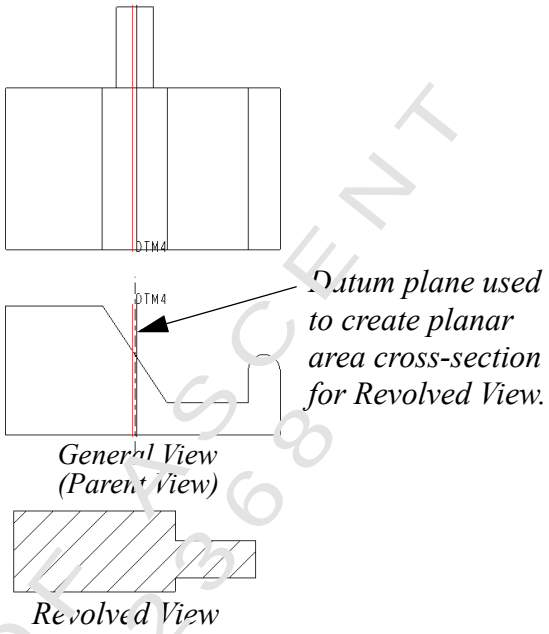
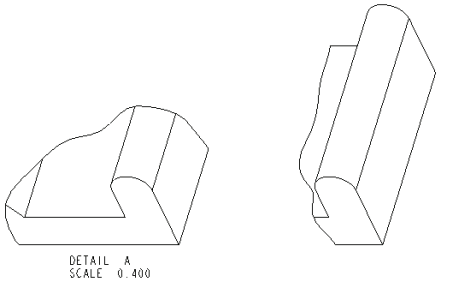
Figure 1–15

The additional view types are described in Table 1–3

Table 1–3

Option	Description
General 	This option creates an independent view of a model. The orientation of the general view is determined by using the **View Type** category in the Drawing View dialog box. It is possible to place several general views on a drawing, as shown in Figure 1–16.  *General View* *General (Oriented) View* **Figure 1–16**
Projection 	This option projects a view off an existing view. This creates orthogonal views, such as top, bottom, right, or left. A projection view maintains the scale of its parent view as shown in Figure 1–17.  *General View (Parent)* *Projection View* **Figure 1–17**

Option	Description
Auxiliary Auxiliary...	This option projects a view normal to a datum plane, edge, or axis of an existing view. This type of view can be used to see the actual size and form off of an angled surface. Auxiliary views can be created by referencing a datum plane, an edge or an axis of any other type of view as shown in Figure 1–18.  **Figure 1–18**
Detailed Detailed...	This option creates a scaled view focusing on a specific area of an existing view. The detailed view is created by sketching a spline on the parent view that encloses the area to be represented. To complete the view, assign a name, boundary type and note location. The detailed view is automatically labeled with its scale value and view name. Orientation of this view corresponds to the parent view as shown in Figure 1–19.  *General View (Parent View)* *Detailed View* **Figure 1–19**

Option	Description
Revolved Revolved...	This option creates a planar area cross-section that is revolved 90° around the projection of the cutting plane as shown in Figure 1–20. Revolved views can be translated in the projection direction.  **Figure 1–20**
Copy & Align Copy and Align	This option creates a duplicate of an existing Partial or Detailed view. A new boundary spline can be sketched and the view is aligned to the parent view as shown in Figure 1–21. The Copy & Align view maintains the same scale as its parent view.  **Figure 1–21**

The methods for creating a new view vary depending on the type of view that is required. The methods include the following:

Drawings created using a template could already contain certain views. Additional views can be added at any time.

- To create an additional General view, right-click and select **Insert General View** and place the view (explained in Section 1.4). As an alternative, you can click General... in the Layout tab.
- To create a Projected view, right-click on the parent view and select **Insert Projection View** or click Projection... in the Layout tab.
- If a parent view is pre-selected, only a Projection view can be created. To create a Detailed, Auxiliary, or Revolved view, you must clear all views and click the appropriate icon in the Model Views group from the Layout tab and then access these options.

Visibility area is a category that can be customized for a drawing view. This is discussed later in this training course.

- To create a Copy and Align view, pre-select a Detailed view or a partial visibility area view and click Copy and Align in the Model Views group from the Layout tab. This view type is not available unless a Detailed view or a partial visibility area view has been created in the drawing.

To obtain information about a view, you can click Drawing View... in the Model Views group from the Layout tab and select a view. An information window appears, as shown in Figure 1–22.

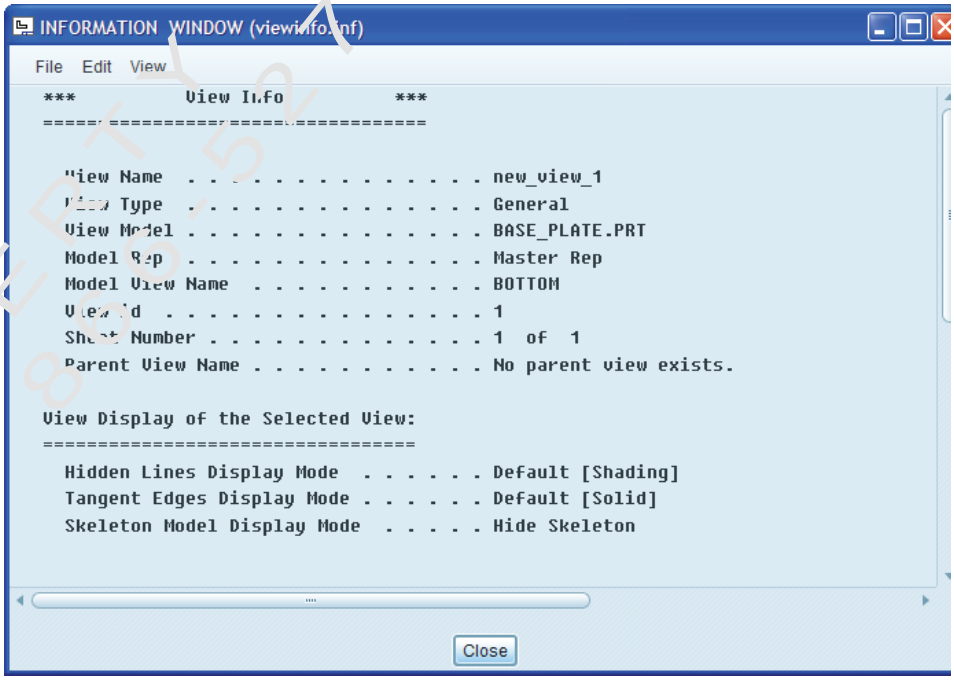


Figure 1–22

Step 3 - Place the view on the drawing sheet.

In general, views are placed using the left mouse button. Note the following additional information for placing specific view types:

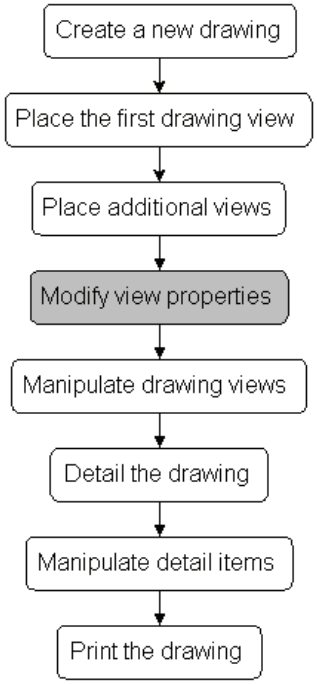
- General views are placed using the left mouse button. Initially, the view is placed in its default orientation; however, it can be oriented into 2D using any of the techniques discussed in Section 1.4.
- Projection views are placed using the left mouse button. They can only be placed horizontally or vertically relative to the parent view.
- Detailed views focus on a specific area; therefore, you must first select a point on an existing view. This point represents the centerpoint for the new view. Once selected, sketch a spline around this point to define the extent (boundary) of the view. To complete the spline, press the middle mouse button. Once the spline is complete, place the view using the left mouse button.
- Auxiliary views first require the selection of an edge, axis, or datum plane to represent the front surface of the new view. Once selected, use the left mouse button to place the view.
- Revolved views require the selection of a parent view followed by placement using the left mouse button. Once selected, the Drawing View dialog box appears. You can select the cross-section to be referenced or create a new one.

Step 4 - Complete the creation of the drawing view.

For many of the view types, the drawing view placement is complete upon placement. However, for General and Revolved views that use the Drawing View dialog box, you must click OK to complete the view placement.

Additional options and categories that are available in the Drawing View dialog box are discussed later in this training course.

1.6 Modify View Properties



All except General and Revolved views are placed without accessing the Drawing Views dialog box. To further refine any views that do not access this dialog box, you can modify the view properties. So far, you have learned about the View Type category. The remaining categories in the Drawing View dialog box can also be used.

Complete the following steps to modify view properties:

1. Select the Layout tab.
2. Open the Drawing View dialog box.
3. Modify the view properties.
4. Complete the modification of the drawing view

Step 1 - Select the Layout tab.

The Layout tab only shows the commands relevant to that function. This tab must be active to modify views.

Step 2 - Open the Drawing View dialog box.

Use one of the following techniques to open the Drawing View dialog box, if it is not already open:

- Double-click on the view that you are modifying.
- Select the view that you are modifying, right-click, and select **Properties**.

Step 3 - Modify the view properties.

To modify a view property, you must select a category in the left column of the Drawing View dialog box. Once selected, the dialog box updates and displays the available options for the selected category.

You learn about the following view property categories:

- Visible Area
- Scale
- Sections

The remaining categories are discussed in later chapters.

Visible Area

By default, views are created so that the entire model is displayed. You can customize how much of the model to show in a view using the Visible Area category. Once the Visible Area category is selected, the dialog box appears as shown in Figure 1–23.

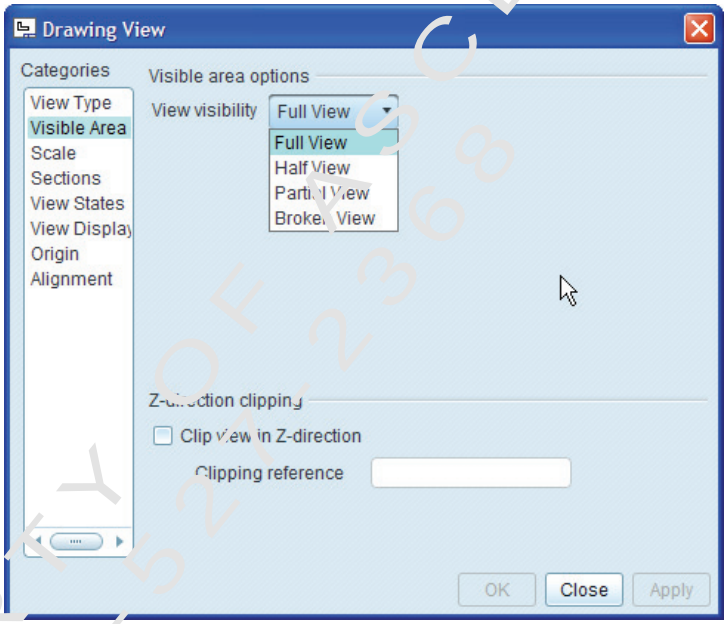
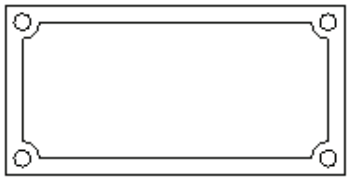
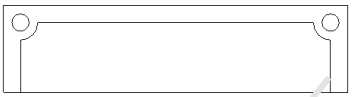
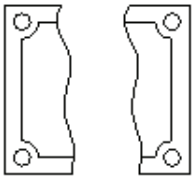


Figure 1–23

The **Visible area** options can be selected in the View Visibility pull-down menu. They are described in Table 1–4.

Table 1–4

Option	Description
Full View	Displays a view of the entire model as shown in Figure 1–24. This is the default option.  **Figure 1–24**

Option	Description
Half View	Displays a view of the model about a line of symmetry as shown in Figure 1–25. The cutting plane used to define the half view can be a planar surface or a datum plane. This option is available for Projection, Auxiliary, and General views.  **Figure 1–25**
Partial View	Displays a view focused on a specific location on the model. Bounding entities define the geometry that is represented as shown in Figure 1–26. This option is available for Projection, Auxiliary, General, and Revolved views.  **Figure 1–26**
Broken View	Displays a view in which break lines define sections to be removed. Break lines can be horizontal and/or vertical as shown in Figure 1–27. This option is available for Projection and General views. You cannot break a horizontally projected view horizontally and you cannot break a vertically projected view vertically.  **Figure 1–27**

Scale

The **Scale** category in the Drawing View dialog box enables you to assign scales to General and Detailed views that are independent of the drawing scale. The scale for all other views in the drawing are determined by the parent view. Figure 1–28 shows the **Scale category** options in the Drawing View dialog box.

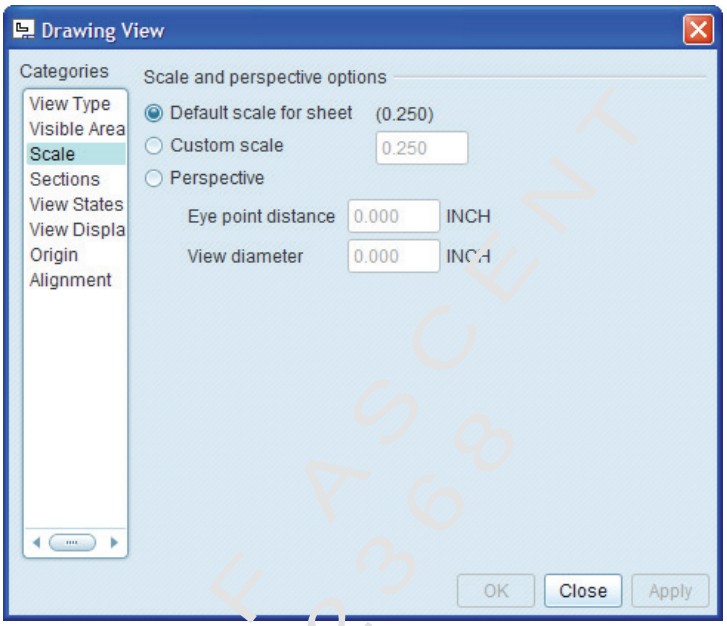


Figure 1–28

By default, views are created using the same scale that was assigned to the sheet (**Default scale for sheet**). Any changes made to the sheet scale updates these views. The **Custom scale** option enables you to assign an independent scale. Once assigned, the view scale is located directly under the view, as shown in Figure 1–29.

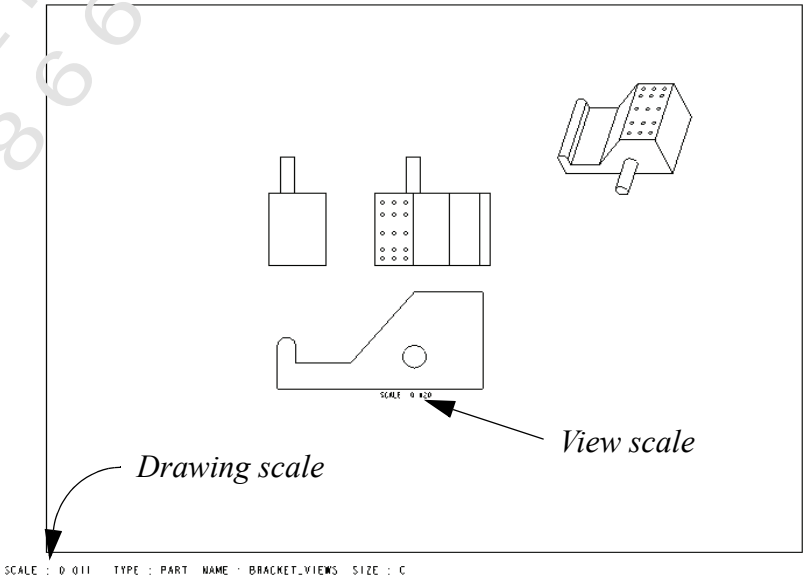


Figure 1–29

The **Perspective** option enables you to change the viewing distance and diameter parameters of a General perspective view. The viewing distance is the distance (in model units) between the object and the viewer.The viewing diameter determines the actual size of the view based on drawing units. Figure 1–30 shows a model with different viewing distance parameter values.

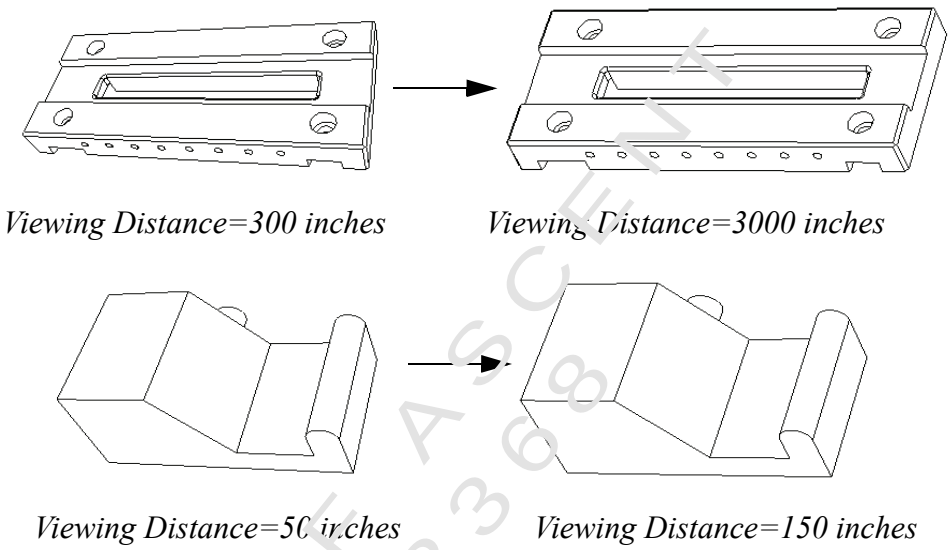


Figure 1–30

Sections

By default, a view is created with no sectioning. To create a cross-section view, select the **Sections** category in the Drawing View dialog box and select the **2D cross-section** option. The dialog box appears as shown in Figure 1–31.

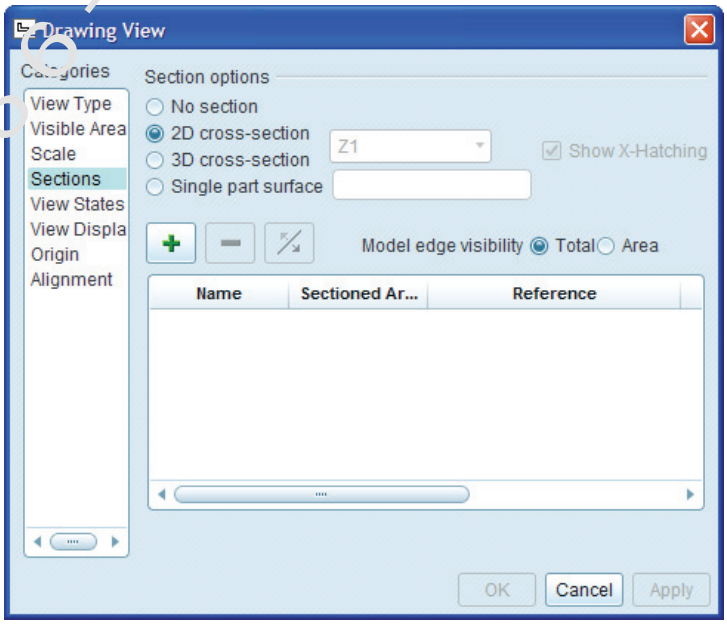


Figure 1–31

2D Cross-Sections

Click  to assign a cross-section to the view. The dialog box updates as shown in Figure 1–32. You can select the section name in the Name pull-down menu or use the **Create New** option in this menu to create a new section. The options for creating a new section are the same as in Part or Assembly mode.

For simplicity, create and save your section in the model, then open and place them in your drawing.

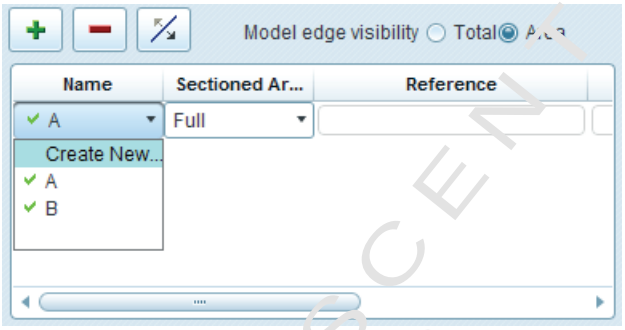
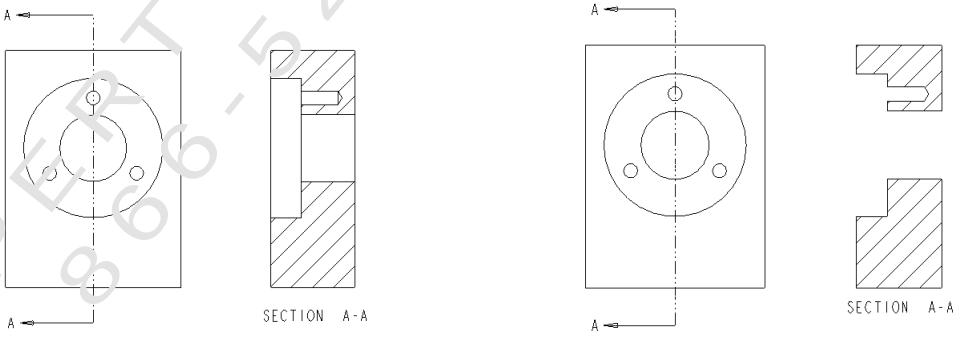


Figure 1–32

The edges that are displayed in the cross-section view can be customized by selecting an edge visibility method. The **Total** option creates a cross-section that displays all edges that are behind the cutting plane, as shown on the left side of Figure 1–33. The **Area** option creates a cross-section that displays only the material within the cutting plane, as shown on the right side of Figure 1–33. Edges behind the cutting plane are not visible.

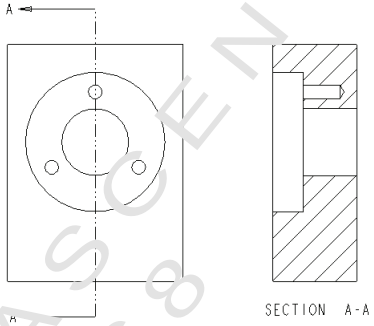
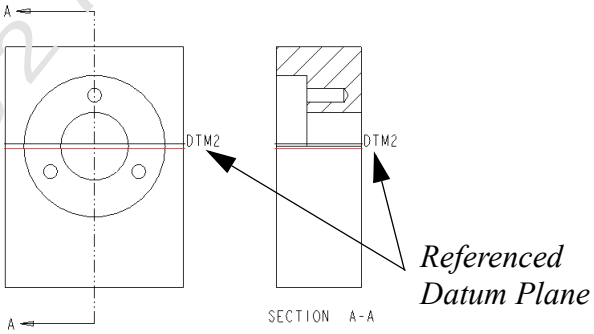


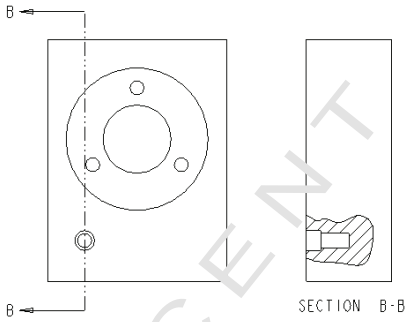
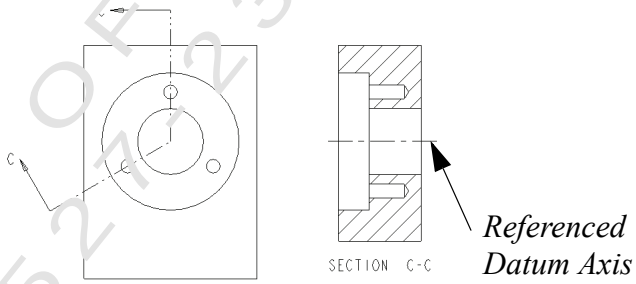
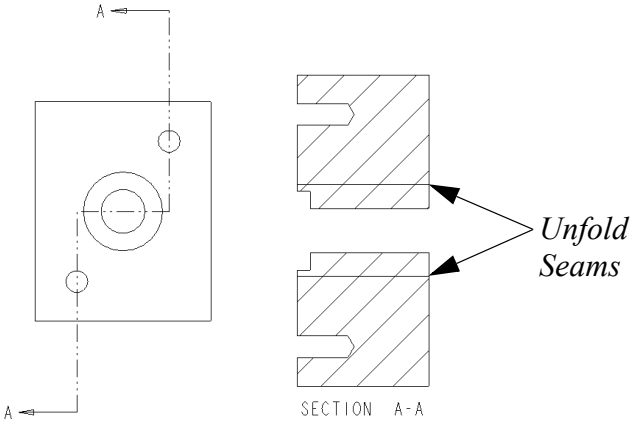
Total model edge visibility *Area model edge visibility*

Figure 1–33

Once the cross-section has been selected, you can select an option in the Sectioned Area pull-down menu. The options are described in Table 1–5.

Table 1–5

Option	Description
Full	Displays the cross-section on the entire view (default option) as shown in Figure 1–34.  **Figure 1–34**
Half	Displays the cross-section on one side of a boundary plane as shown in Figure 1–35. A datum plane is selected as the boundary for the cross-section. The material on the opposite side of the datum plane remains solid.  **Figure 1–35**

Option	Description
Local	Displays the cross-section focusing on a specific area(s). A sketched spline defines the boundary of the local view. The material outside the boundary remains solid as shown in Figure 1–36.  **Figure 1–36**
Full (Align)	This option creates an area cross-section through the cutting planes of an offset cross-section until parallel to the screen. The planes are unfolded about an existing axis in the view as shown in Figure 1–37.  **Figure 1–37**
Full (Unfold)	This option creates an area cross-section where the cutting planes of an offset cross-section unfold until parallel to the screen. The planes are unfolded about the seams of the offset cutting line as shown in Figure 1–38.  **Figure 1–38**

Once the **Sectioned Area** option has been selected, scroll to the right to access the remaining options for defining the section. These options are shown in Figure 1–39.

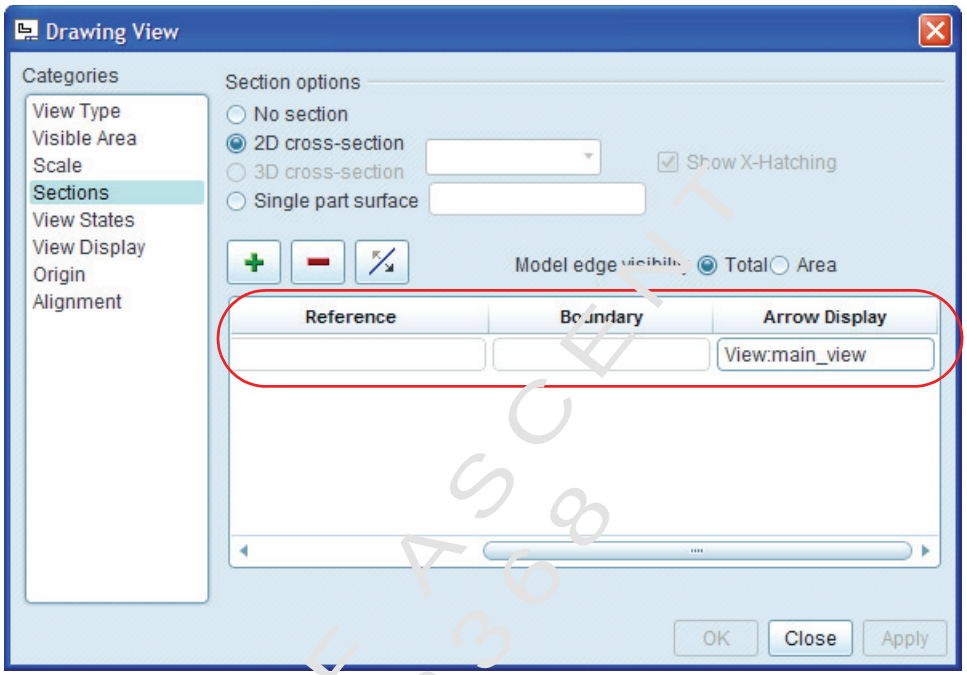


Figure 1–39

Once a section view has been created, cross-section arrows can be added by right-clicking on the sectioned view and then selecting ***Add Arrows or*** by clicking Arrows... .

The Arrow Display column enables you to add cross-section arrows to a view, as shown in Figure 1–40. To place the arrows, select this cell and select the view to place the arrows.

To clear the arrows from the display, right-click on the reference view that was assigned and select **Remove**.

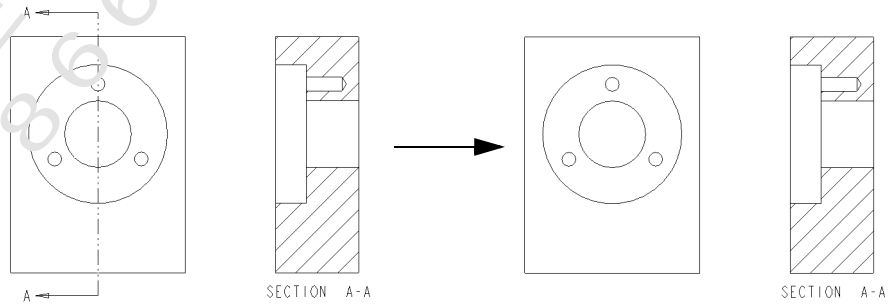


Figure 1–40

The Reference and Boundary columns are only available if you have selected a local cross-section. They enable you to define the center point for the breakout and its defining spline.

Multiple 2D Cross-Sections

Multiple cross-sections can be assigned to a view by clicking  to define another cross-section. Creating multiple cross-sections in one view can help reduce the number of views on a drawing sheet. For example, Figure 1–41 shows a view that has two cross-sections; one is set as Full and one as Local. To remove a cross-section from the view, select it and use  .

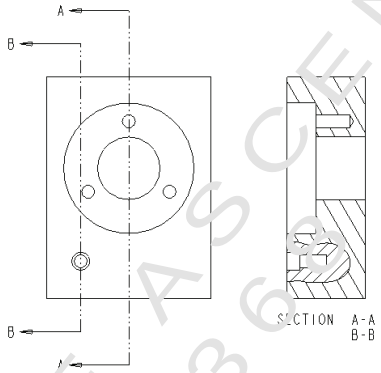


Figure 1–41

Single Part Surface

*A view to which the **Single part surface** option is applied cannot be converted back to a normal view.*

The **Single part surface** option in the Section options section of the dialog box provides an alternative to selecting or creating a cross-section. This option enables you to create a view of a single surface by projecting from the solid model surface or a datum quilt. Pro/ENGINEER deletes all other geometry, as shown in Figure 1–42. This option is not available for Detailed views.

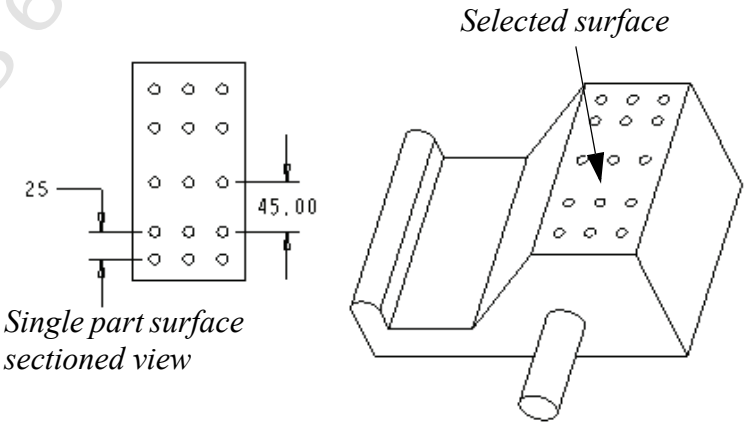


Figure 1–42

When you create a cross-section in Part mode or Assembly mode, the system automatically checks to see if the parts intersected by the cross-section have assigned materials. If the material name matches the name of the saved cross-hatching pattern file (e.g., copper.xch), the cross-hatching pattern is automatically applied. The *pro_crosshatch_dir* config.pro option is used to point to a directory containing user-defined cross-hatching pattern files.

In Figure 1–43, the assembly drawing contains a section view. The materials assigned to the two components (i.e., aluminum and steel) match the names of saved cross-hatching files (i.e., aluminum.xch and steel.xch). When the section view was created and placed, these cross-hatching patterns were automatically applied.

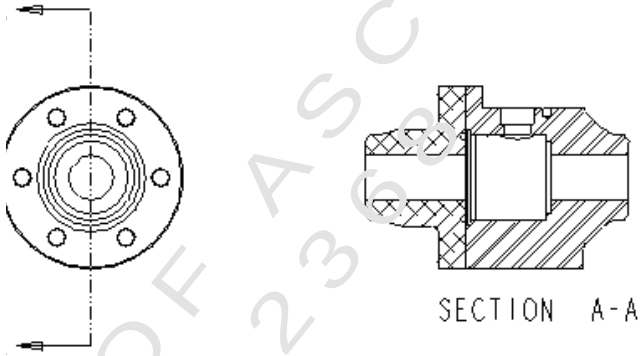


Figure 1–43

Cross-hatching

- You can hatch or fill flat surfaces by right-clicking on them and selecting **Hatch** or **Fill**.
- You can use the **X-Area** option on the MOD XHATCH menu to show or hide areas of component sections, as shown in Figure 1–44.

*To access the MOD XHATCH menu, right-click on the crosshatch/fill and select **Properties**.*

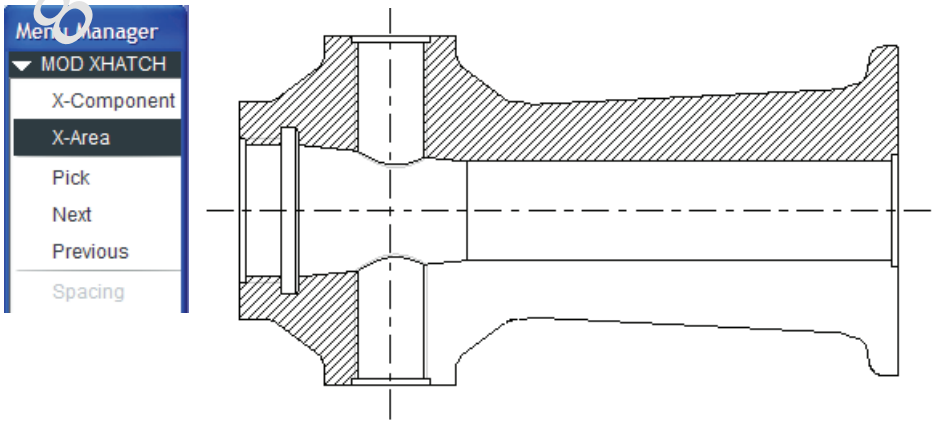


Figure 1–44

The configuration options listed in Table 1–6 are used in conjunction with cross-hatching in a drawing.

Table 1–6

Configuration Option	Value	Description
default_show_3d_ section_xhatch	**Yes**/No	Controls the default visibility of 3D sections in a drawing.
default_show_2d_ section_xhatch	**assembly_and_ part** assembly_only part_only No	Controls the default visibility of 2D sections in a drawing.

Shaded Views

Shaded views of models can be included in drawings. This capability improves the plotting of drawings with CLE objects and shaded views. Shaded drawing views display the color scheme from the part/assembly; this includes surface colors as well as textures. You can combine traditional views (such as wireframe, hidden line, etc.) and shaded views in one drawing, as shown in Figure 1–45.

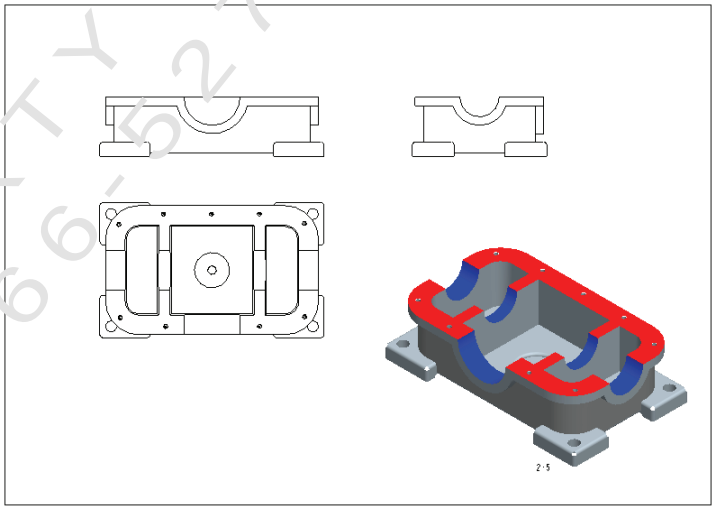


Figure 1–45

You define a view as shaded in the View Display category of the Drawing View properties dialog box. Select **Shading** in the Display style pull-down menu, as shown in Figure 1–46.

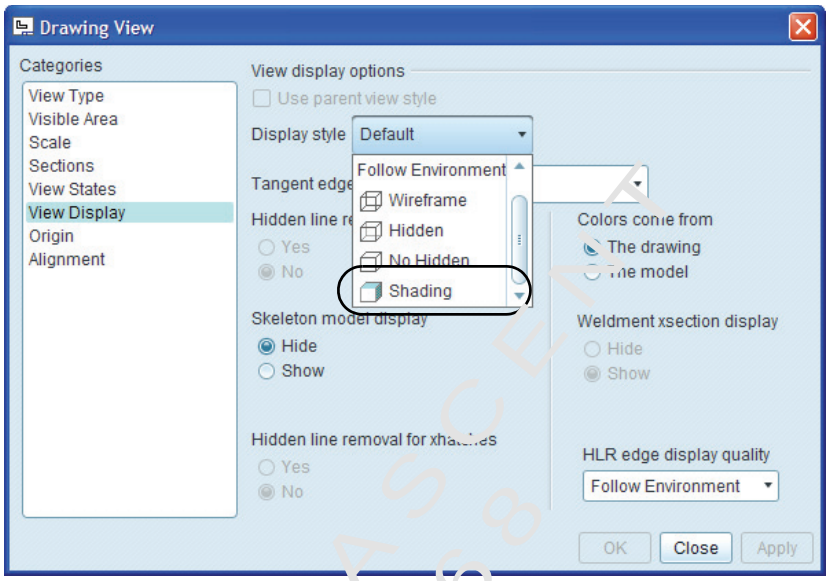


Figure 1–46

Displaying edges in shaded views

By default, edges are not displayed in the shaded view. You can display them by setting the *show_shaded_edges* configuration option to **Yes**. The model with edges displayed is shown in Figure 1–47.

The edges are displayed in all shaded views in the drawing.

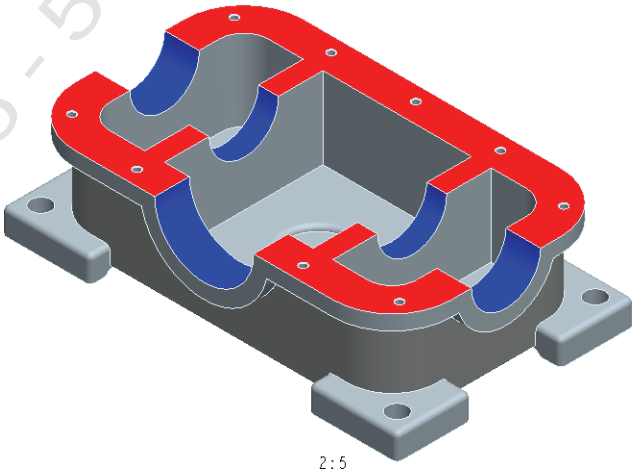


Figure 1–47

Step 4 - Complete the modification of the drawing view

To complete the modification to the drawing view, click OK in the Drawing View dialog box.

1.7 Drawing Environment Options

Environment Dialog Box

The Environment dialog box controls the display and default actions of your Pro/ENGINEER session. Select **Tools > Environment** to access the Environment dialog box, as shown in Figure 1–48.

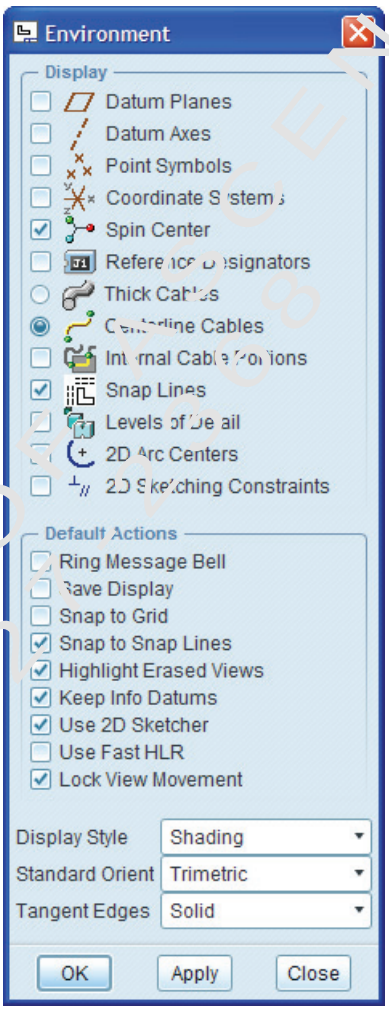


Figure 1–48

Many of the environment option values can be globally controlled in the configuration file (e.g., display_planes enables you to define whether by default datum planes are displayed or not when the model is originally opened).

Many options in the Environment dialog box enable you to control default settings in your drawing. These include the following:

- The **Snap Lines** option enables you to turn snap lines On or Off in your drawing.
- The **Snap to Grid** option enables you to snap your drawing views to a grid line to align selected views.
- The **Snap to Snap Lines** option enables you to locate dimensions, notes, geometric tolerances, and surface finishes.
- The **Highlight Erased Views** option enables you to easily identify the view outline of erased views.
- The **Lock View Movement** option enables you to automatically lock views to prevent them from being moved. Locking must be disabled to move a view.

Drawing Setup File

The drawing setup file is a file that gets stored with each drawing and controls certain characteristics of the drawing. For example, the drawing setup file values determine items such as projection type, view scale format, drawing units, dimension, and note text height.

All drawing setup file options can be reviewed using Pro/HELP. Refer to Appendix B for a description on how to use these options.

To access the drawing setup file, select **File > Drawing Options**. The Options dialog box appears as shown in Figure 1–49.

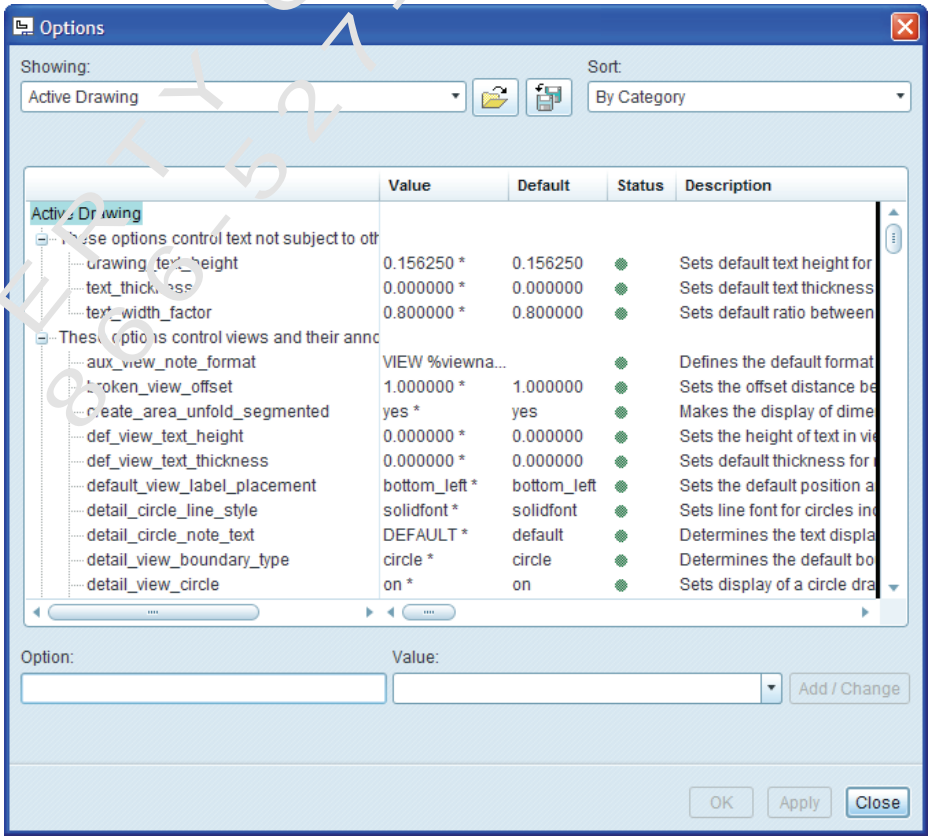


Figure 1–49

Drawing setup files are saved with a .dtl extension. The drawing_setup_file configuration file option enables you to specify a directory for storing all pre-defined company setup files.

Every Pro/ENGINEER drawing assigns default values to these setup file options. These values can be modified and saved for use in the same or other drawings. To save a drawing setup file for use in another drawing, click  . To open a pre-defined setup, click  and select the file.

Config.pro File

The config.pro is a configuration file that controls environment settings (e.g., setup file, drawing file editor) in Pro/ENGINEER. Select **Tools > Options** to access the config.pro file. The Options dialog box appears as shown in Figure 1–50.

*Select the **Show only options loaded from file** option to show the user-defined options. Clear this option to show all options.*

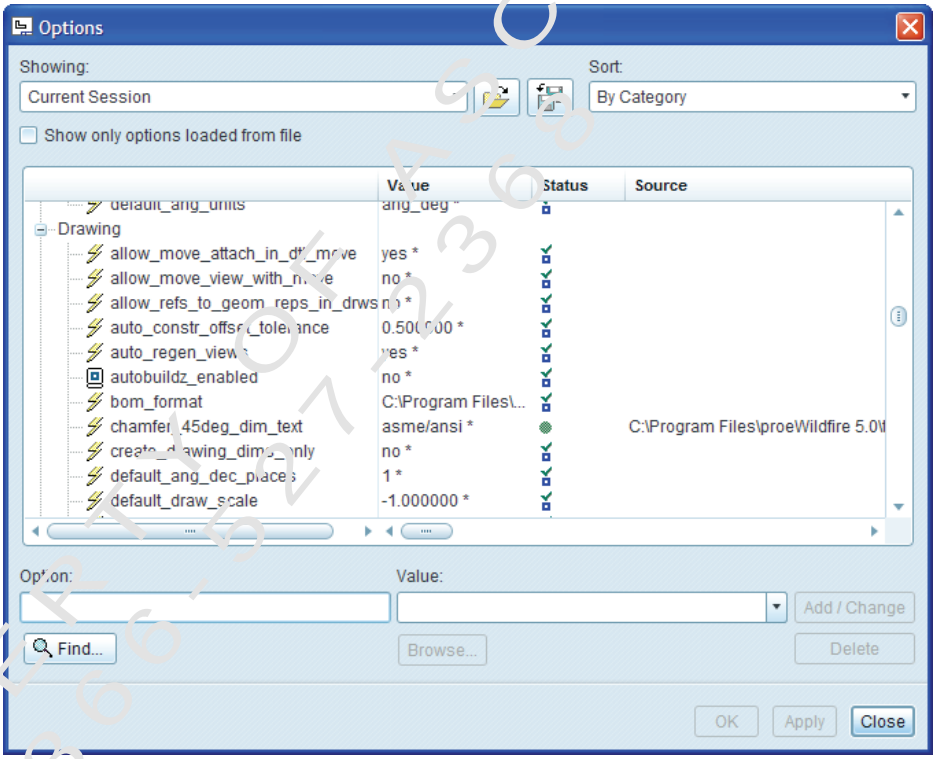


Figure 1–50

All configuration file options can be reviewed using Pro/HELP. Refer to Appendix B for a description on how to use Pro/HELP.

Many of the configuration file options are specific to Drawing mode. To access the specific drawing model options, select **By Category** in the Sort pull-down menu and scroll down to the Drawing list. All options in this section pertain to Drawing mode.

Unlike the drawing setup file, the config.pro is not automatically saved and stored with the drawing. Once changes are made, you must explicitly save these changes by clicking  . These changes are stored to the current configuration file.

Pro/ENGINEER reads the configuration file from several locations: the /loadpoint/text directory, the user's home directory, and the startup directory. Additional configuration files can be stored in other locations, but they must be explicitly loaded to affect the drawing. To open a config.pro, click  and select the file.

Exercise 1a

Create a New Drawing

In this exercise, you create drawing views from a Pro/ENGINEER solid part. The drawing model is shown in Figure 1–51.

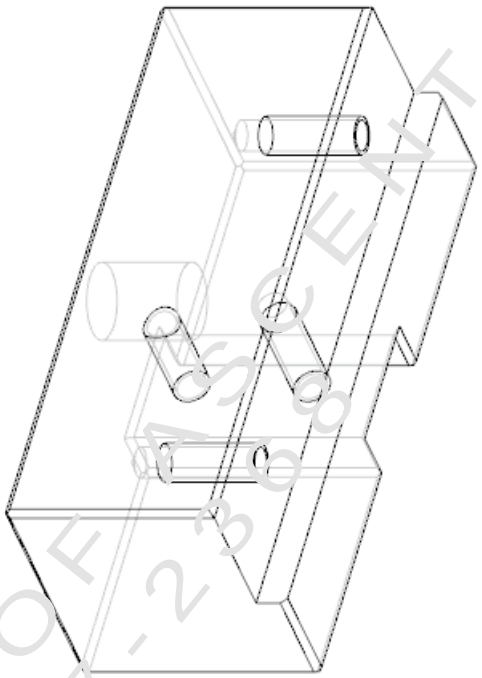


Figure 1–51

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a new drawing**
- ✓ **Assign a model to the drawing**
- ✓ **Select the sheet size**
- ✓ **Place views on the drawing**
- ✓ **Modify the drawing scale**

Task 1 - Create a new drawing and select the format and sheet size.

1. Click  in the toolbar to create a new drawing. The New dialog box appears.
2. Select the **Drawing** option.
3. Enter [support] for the name.

4. Clear the **Use Default Template** option to create an empty drawing. The New dialog box appears as shown in Figure 1–52.

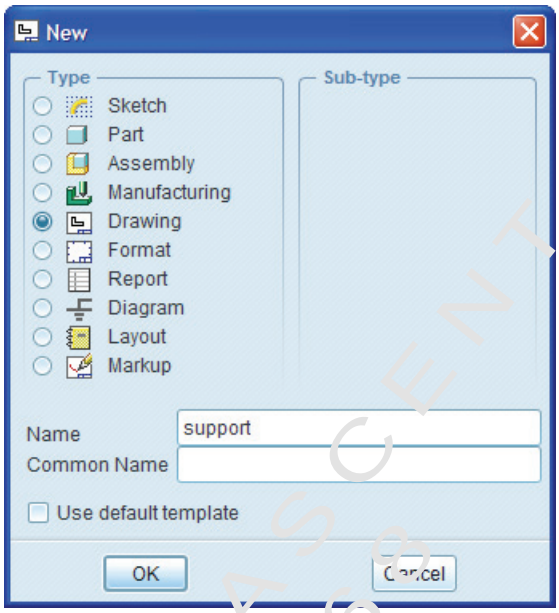


Figure 1–52

5. Click OK. The New Drawing dialog box appears.

6. Click Browse... in the Default Model section of the New Drawing dialog box.

7. Select **support.prt** in the current working directory and click Open.

8. Select the **Empty** option in the Specify Template section.

9. Select **A** in the Standard Size pull-down menu. The New Drawing dialog box appears as shown in Figure 1–53.

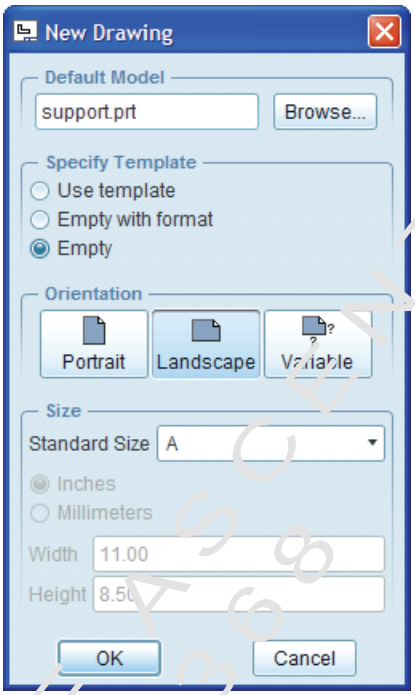


Figure 1–53

10. Click OK to create the drawing. The Drawing mode user-interface appears.

Task 2 - Define the drawing's projection type.

1. Select **File > Drawing Options** in the FILE PROPERTIES menu. The Options dialog box appears.
2. Select **By Category** in the Sort pull-down menu.

3. Select the *projection_type* option and ensure that it is set to **third_angle**. This is the default option. The Options dialog box appears as shown in Figure 1–54.

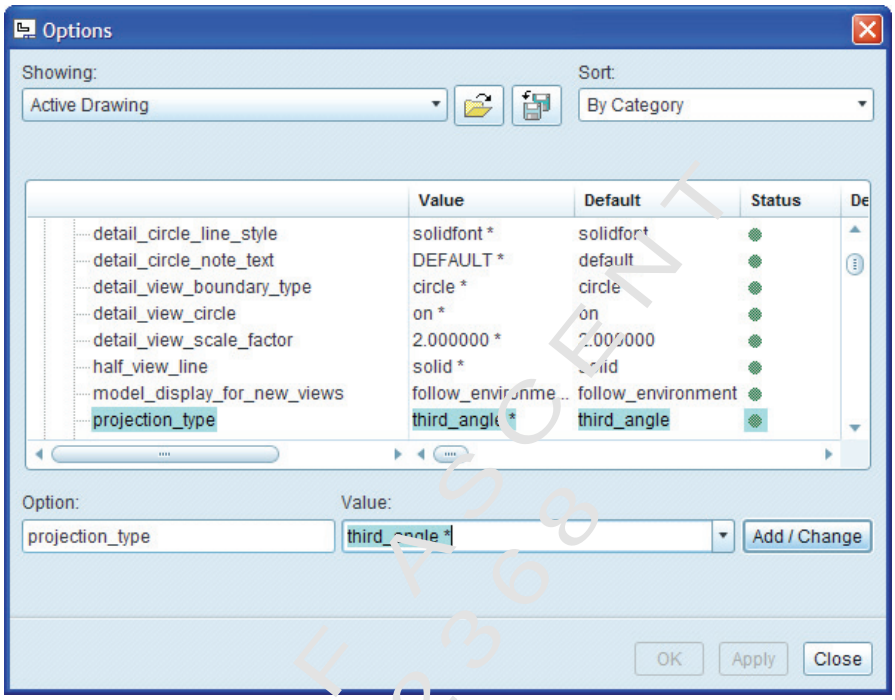


Figure 1–54

4. Close the Options dialog box.

Task 3 - Place a General view in the default orientation.

*You can also right-click and select **Insert General View**.*

1. Click from the flyout menu at the top.
2. Ensure the Layout tab is active.
3. Click General... in the Model Views group.

4. Select the top right corner of the drawing sheet to define the centerpoint for the drawing view. The Drawing View dialog box appears as shown in Figure 1–55.

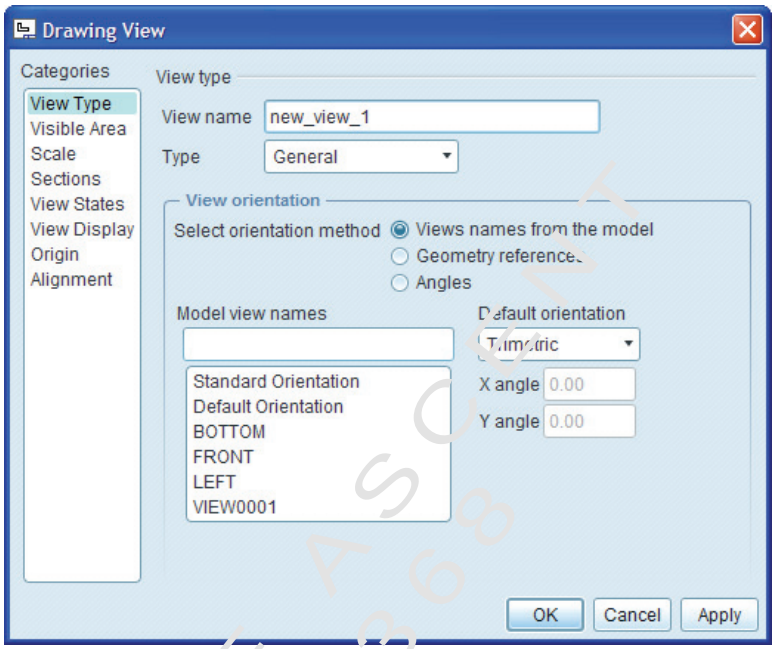


Figure 1–55

5. Select **Default Orientation** in the Model view names section and click OK. The drawing model appears as shown in Figure 1–56 in its default orientation.

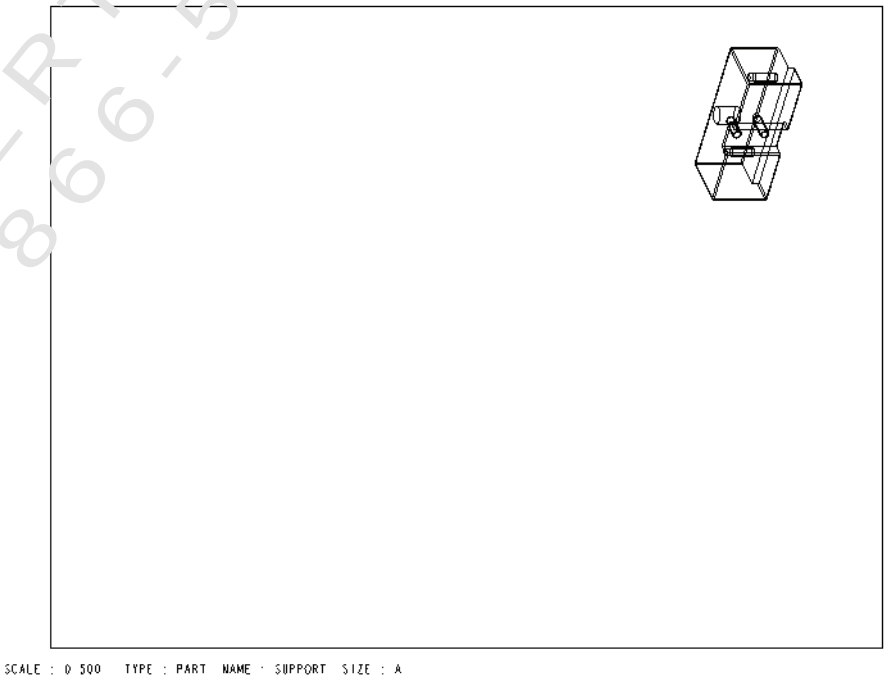


Figure 1–56

Task 4 - Place a General view.

1. Click 

in the Model Views group to create a new General view.
2. Select the bottom left corner of the drawing sheet using the left mouse button to define the centerpoint for the drawing view.
3. Select **Geometry references** as the orientation method in the Drawing View dialog box.
4. Select **Left** in the pull-down menu for Reference 1. Select datum plane DTM1 in the Model Tree.
5. Select **Top** for Reference 2 and select datum plane DTM2 in the Model Tree. The Drawing View dialog box appears as shown in Figure 1–57.

While orienting a view, the drawing can be dynamically zoomed and panned. Dynamic rotation is not available in a 2D drawing.

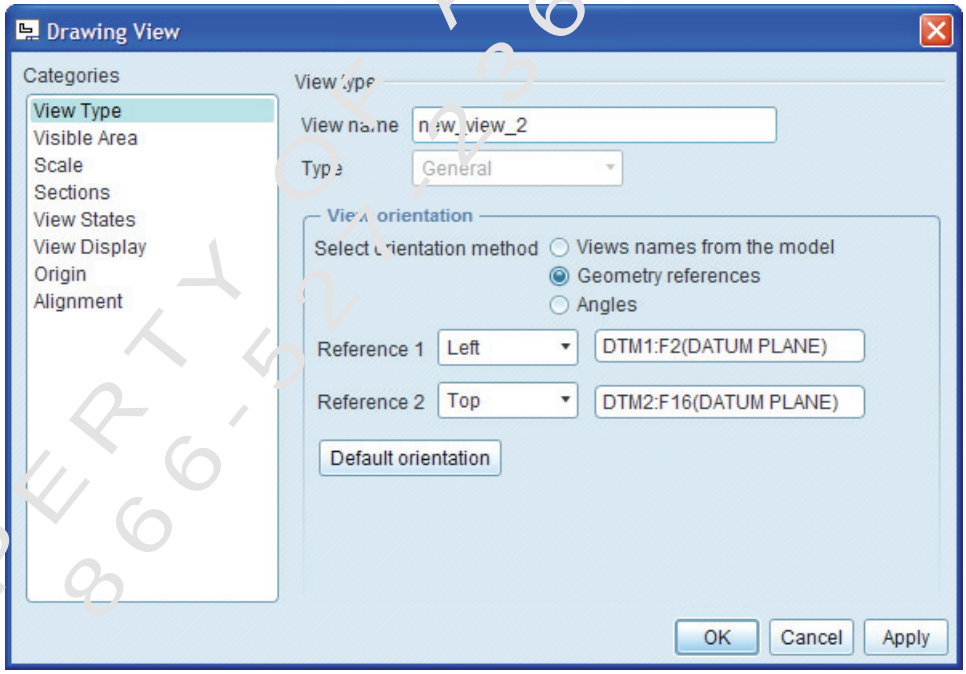


Figure 1–57

6. If the view orientation corresponds to the view shown in Figure 1–58, click OK. If not, click Default orientation and reorient the model as necessary.

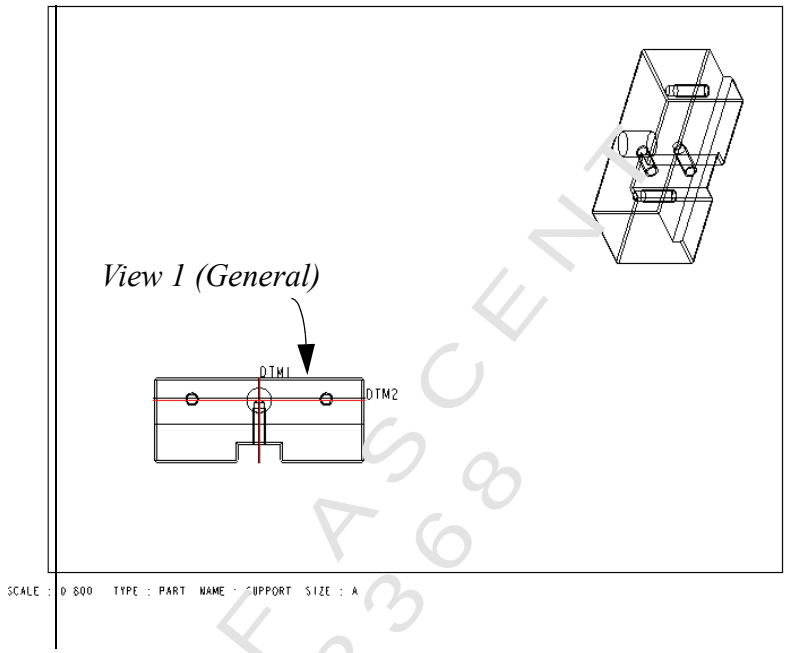


Figure 1–58

Task 5 - Modify the scale value for the drawing.

You can also double-click on the scale value to modify it.

1. Select **Edit > Value**. Select the scale value in the lower left corner of your drawing and modify it to [0.8]. The drawing and all of its views update, as shown in Figure 1–59.

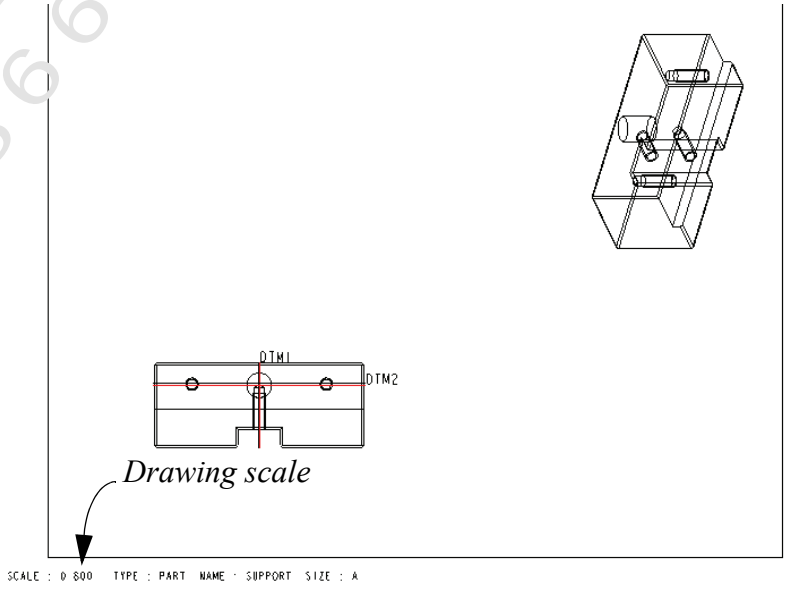


Figure 1–59

Task 6 - Create projection views in the drawing.

1. Select the second General view that was oriented into the 2D orientation. The view highlights in red. This is considered the parent view for the projection.
2. Right-click and select **Insert Projection View**.
3. Drag your mouse above its parent view.
4. Press the left mouse button to place the view. The drawing appears as shown in Figure 1–60.

Because of the location of where you are placing the projected view, you might experience a conflict in determining the parent. Select the oriented general view as the parent.

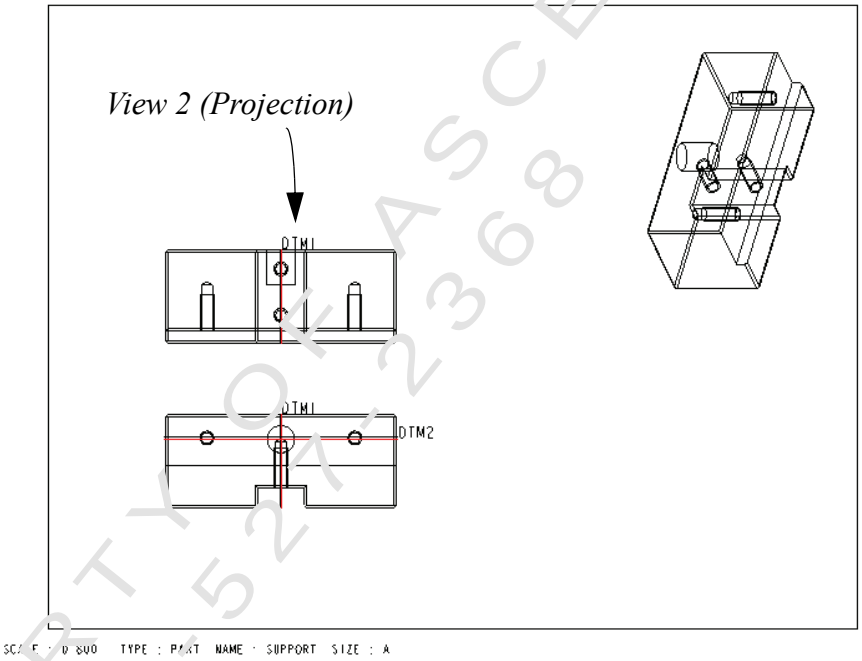


Figure 1–60

5. Repeat steps 1 and 4 to place a Projected view to the right of View 1. The drawing appears as shown in Figure 1–61.

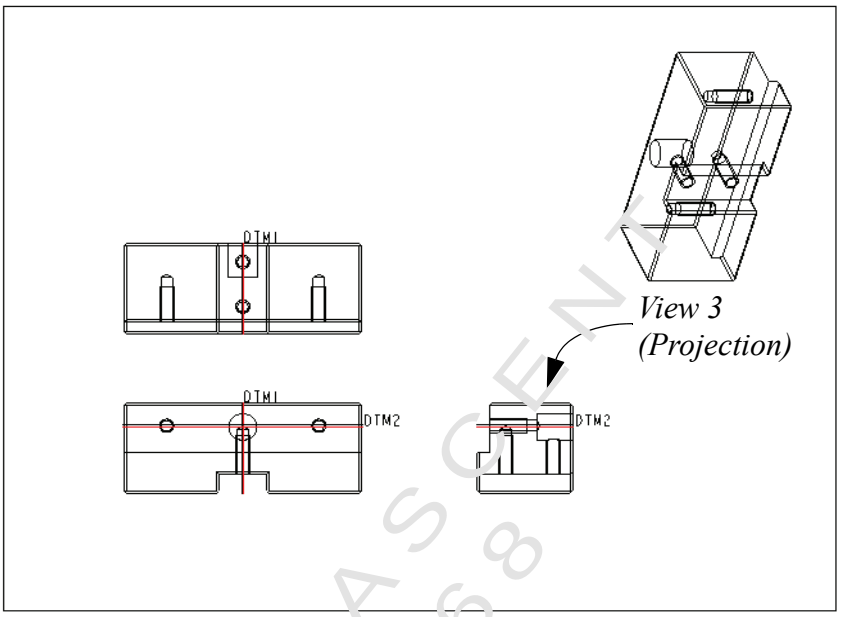


Figure 1–61

6. Click to save the drawing and press <Enter>.
7. Select **File > Close Window** to close the window.

Exercise 1b

Add a Drawing Format

In this exercise, you create drawing views from a Pro/ENGINEER solid part. You also assign a format to the drawing. The completed drawing is shown in Figure 1–62.

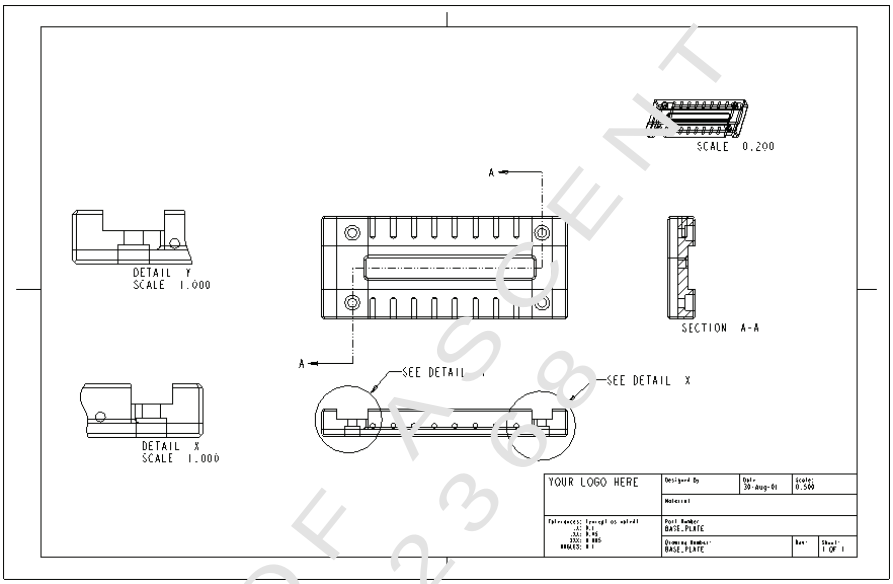


Figure 1–62

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a new drawing**
- ✓ **Add a drawing format**
- ✓ **Orient general views using saved views**
- ✓ **Create a cross-sectional view**
- ✓ **Create detailed views**

Task 1 - Create a new drawing.

1. Click  in the toolbar to create a new drawing. The New dialog box appears.
2. Select the **Drawing** option.
3. Enter [base_plate] as the name of the drawing.
4. Clear the **Use Default Template option** to create an empty drawing.

5. Click OK. The New Drawing dialog box appears.

6. Click Browse... in the Default Model section of the New Drawing dialog box.

7. Select **base_plate.prt** in the current working directory and click Open.

8. Select the **Empty with format** option.

9. Click Browse... in the Format section and double-click on **generic_b_1.frm** in the Formats directory. If the formats directory is not listed in the Open dialog box, click Working Directory to browse to the working directory and select the Format directory. The dialog box appears as shown in Figure 1–63.

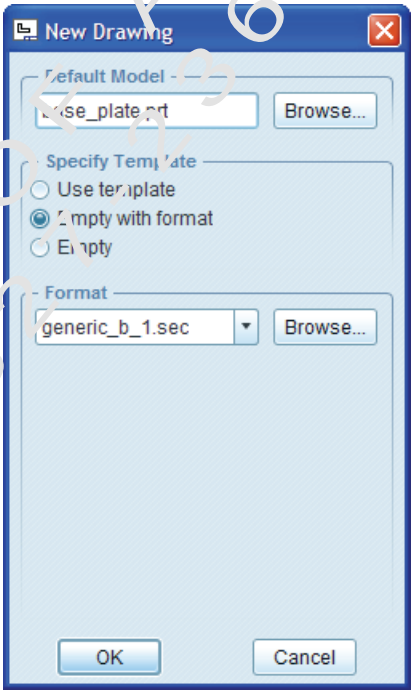


Figure 1–63

10. Click OK to create the drawing.

Task 2 - Create a General view and orient it using a saved view.

1. Select the Layout tab.

2. With your mouse on the drawing sheet, right-click and select **Insert General View**.
3. Select the center of the drawing sheet as the centerpoint of the drawing view. The Drawing View dialog box appears.
4. Click **BOTTOM** in the Model view names section of the dialog box.
5. Click Apply. The view automatically orients itself as shown in Figure 1–64.

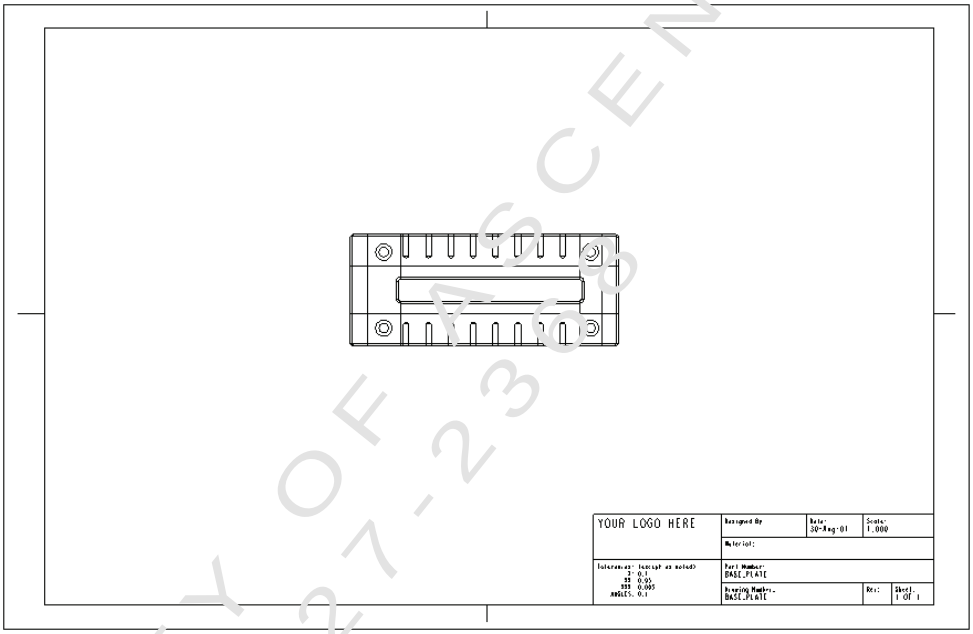


Figure 1–64

6. Click Close to complete view placement and close the Drawing View dialog box.

Task 3 - Create an offset cross-section view in the drawing.

1. Select the General view that was just added to the drawing. The view frame highlights in red. This is considered the parent view for the projection.
2. Right-click and select **Insert Projection View**.
3. Drag your mouse to the right of the parent view.
4. Press the left mouse button to place the view.

5. Select the Projection view, right-click, and select **Properties**. The Drawing View dialog box appears. This dialog box enables you to further customize the drawing view so that it can be displayed as a section view.
6. Select the **Sections** category in the left frame of the Drawing View dialog box.
7. Select **2D cross-section** to assign a cross-section to the view.
8. Keep the **Total** option selected in the Model edge visibility section to ensure that all edges are visible throughout the model, and not only for the 2D area of the section.
9. Click  to add a section to the view. The Drawing View dialog box appears as shown in Figure 1–65.

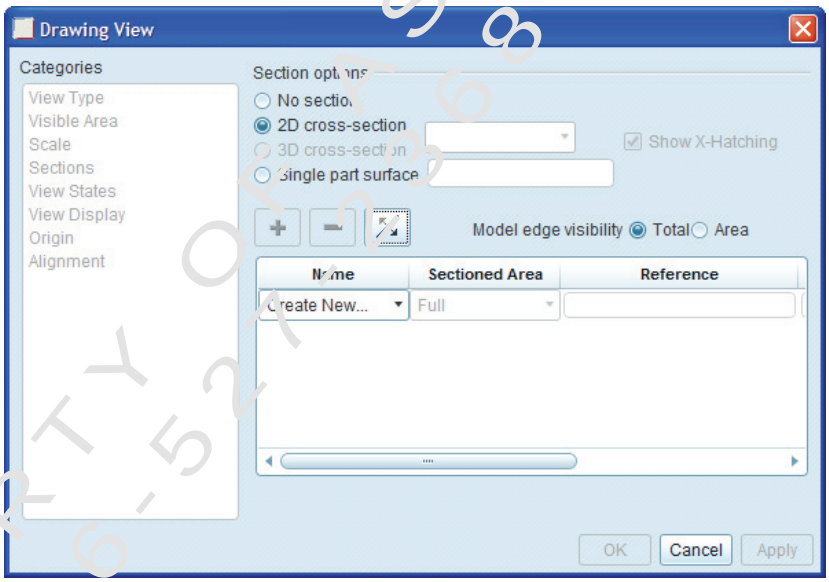


Figure 1–65

It is recommended to create cross-sections in Part or Assembly mode, because the Sketcher environment is more flexible when working in these modes.

10. Select **Offset > Both Sides > Single > Done** in the XSEC CREATE menu.

11. Enter [A] at the prompt for cross-section name. The model appears in Part mode, as shown in Figure 1–66.

Figure 1–66

The cross-section sketch is created using Sketcher.

12. Select **DTM2** as the sketching plane and flip the viewing direction.
13. Select **Top** and **DTM3** as the reference plane. The model is oriented into 2D for sketching.
14. Select **Sketch > References** from the part menu bar.

Click  *to display the datum axis.*

15. Select A_29 and A_20 and the top and bottom surfaces as additional sketcher references.
16. Select **Sketch > Line > Line** and sketch the cross-section shown in Figure 1–67.

*To turn off the grid, select **Sketch > Options** and clear the **Grid** option in the Display tab.*

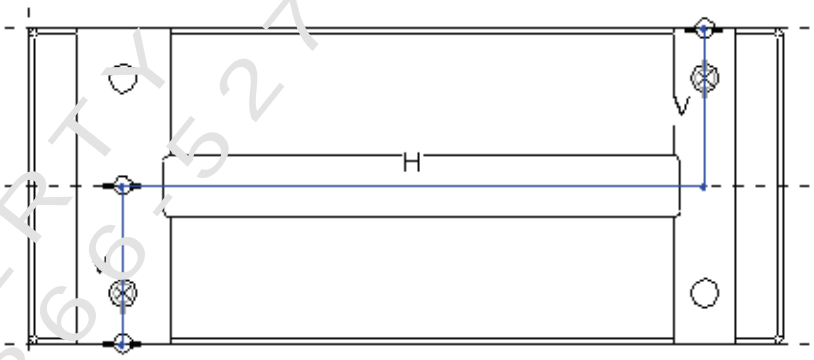


Figure 1–67

17. Select **Sketch > Constraints** and add coincidence constraints between the vertical sketched lines and the axis references, as required.
18. Select **Sketch > Done** to complete the sketch.
19. Ensure that **Full** is displayed in the Sectioned Area for the cross-section. This ensures that the full view is sectioned.

20. Scroll to the right of the information for section A and select in the Arrow Display column, as shown in Figure 1–68.

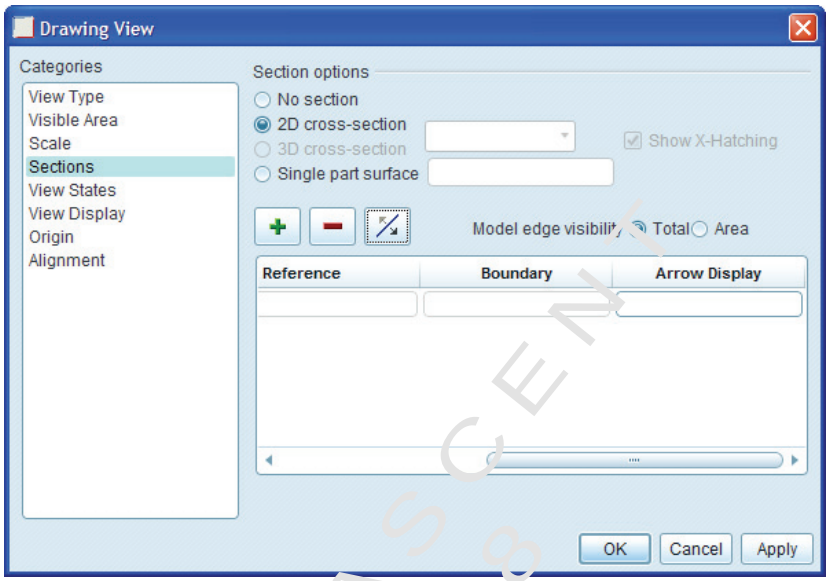


Figure 1–68

21. Select the General view in the drawing as the view to display the cutting plane arrows. Click Apply. The drawing appears as shown in Figure 1–69.

Flip the cutting plane arrows to point to the left if necessary.

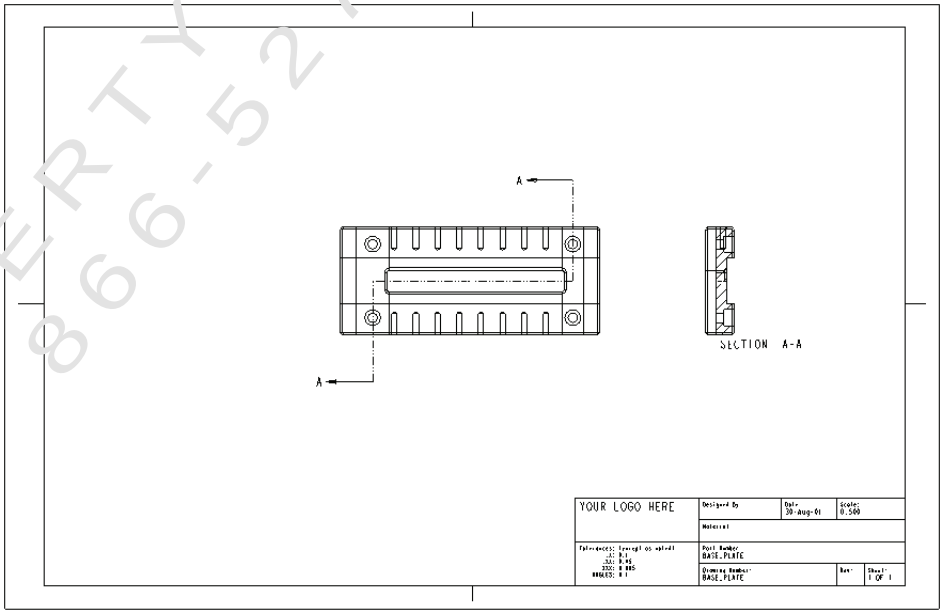


Figure 1–69

22. Click Close to complete view placement and close the Drawing View dialog box.

Task 4 - Create a Projection view in the drawing.

1. Create the Projection view shown in Figure 1–70.

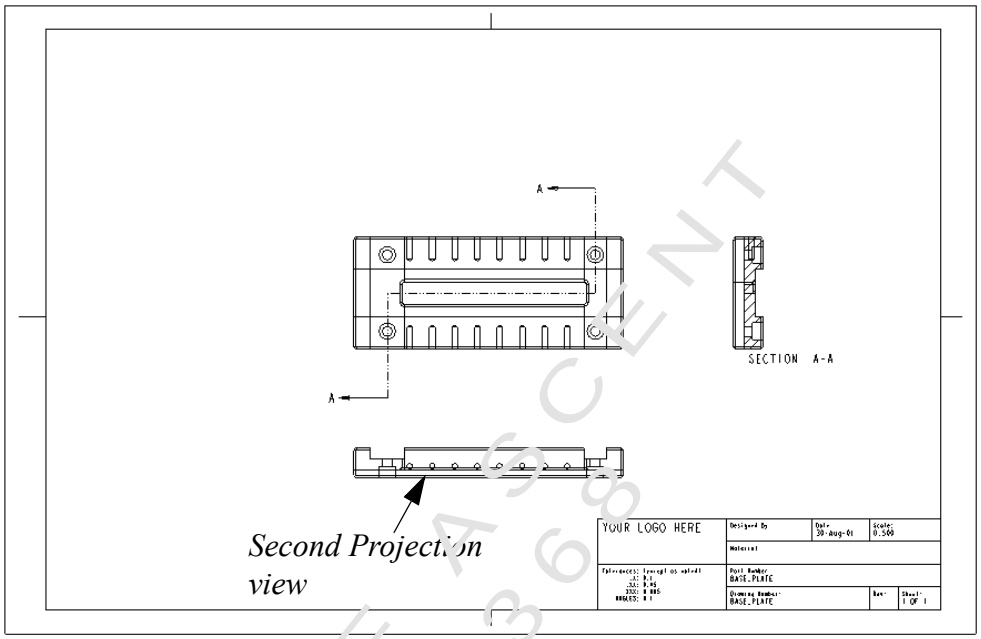


Figure 1–70

Task 5 - Create Detailed views in the drawing

In this task, you create the detailed view shown in Figure 1–71.

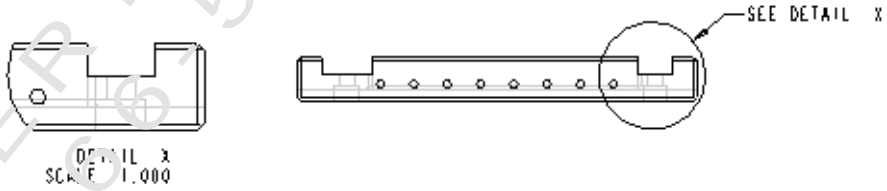


Figure 1–71

You can also click Detailed... *in the Model Views group from the Layout tab.*

1. Ensure that nothing is selected. Right-click and select **Insert Detailed View**.

2. To define a Detailed view, you must select a centerpoint in an existing view. This centerpoint must be on the model geometry. Select the second Projection view, as shown in Figure 1–72.

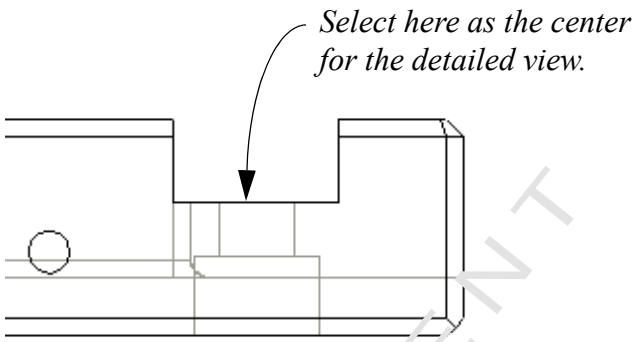


Figure 1–72

3. Sketch a closed spline around the centerpoint as shown in Figure 1–73 using the left mouse button. To complete the spline, press the middle mouse button. No entities should overlap.

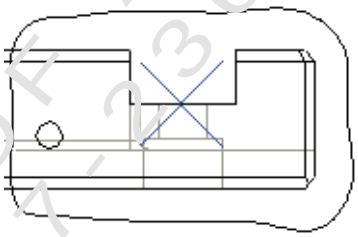


Figure 1–73

4. Select to the left side of the second Projection view as the centerpoint for placing this Detailed view. The drawing view is automatically placed.
5. Select the new Detailed view, right-click, and select **Properties**. The Drawing View dialog box appears.
6. Change the View name to [X] and click Apply.
7. Expand the Boundary type on parent view pull-down menu in the Detailed view properties section. These options enable you to change the shape that identifies the detailed area. Keep the **Circle** option selected.
8. Select the Scale category. The default value is 1.000. Notice how this value differs from the drawing scale. Maintain this value.
9. Click Close to complete the view.

10. The detailed view note is located by default. To change this location, select the Annotate tab. Select the note on the drawing and move it as necessary, as shown in Figure 1–74.

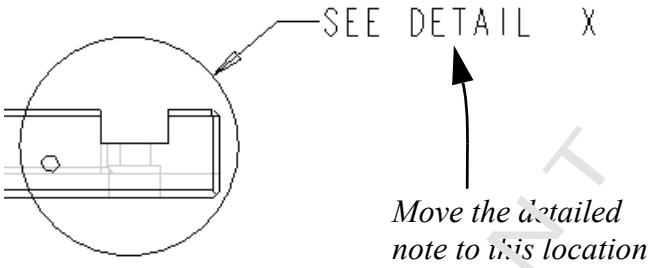


Figure 1–74

The drawing updates after the successful creation of the Detailed view, as shown in Figure 1–75.

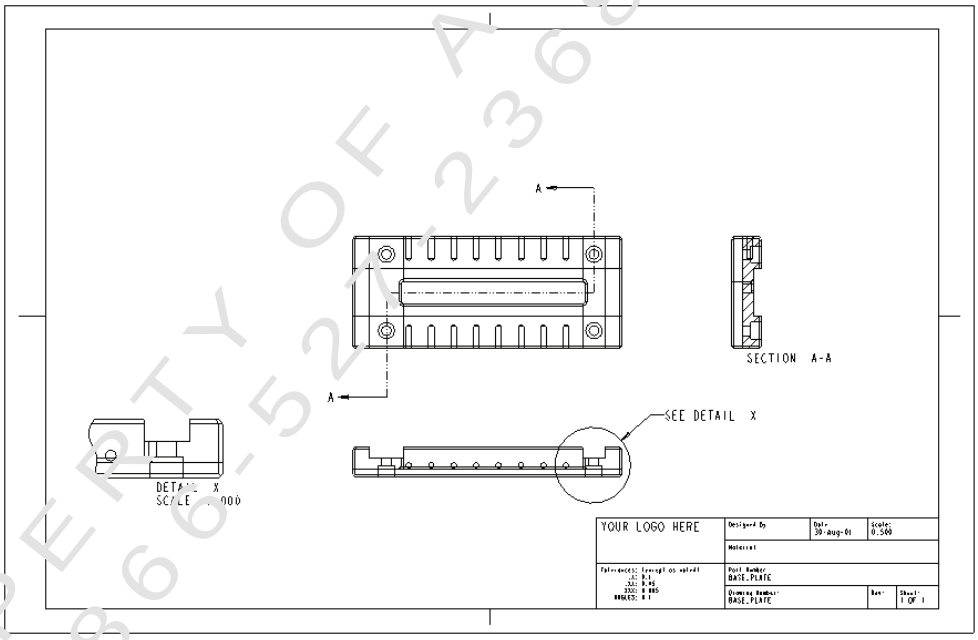


Figure 1–75

11. Select the Layout tab.

12. Repeat steps 1 to 10 to create the Detailed view called Y, shown in Figure 1–76.

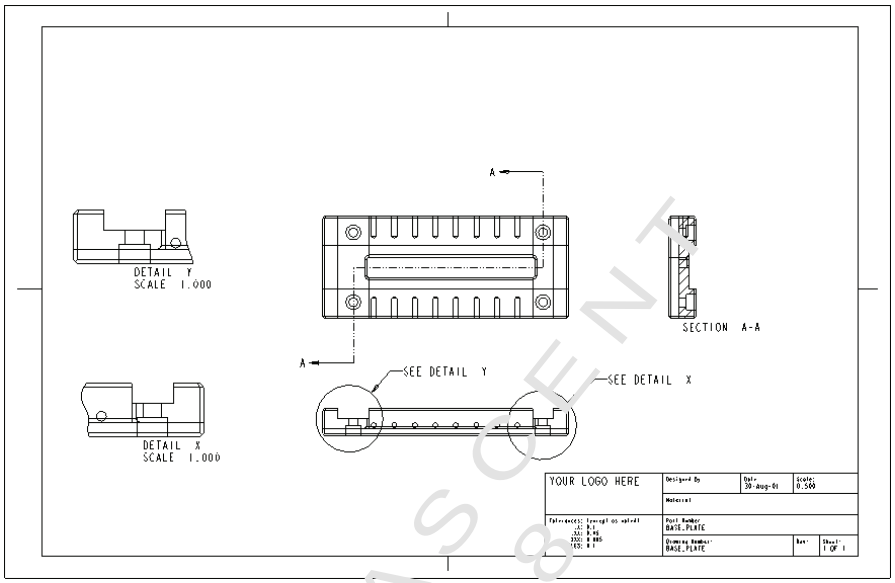


Figure 1–76

Task 6 - Create another General view using the default orientation.

1. Place an additional General view in the upper right corner of the drawing. Create the view with an independent scale value of 0.2. Use the model's default view to orient the model. The drawing updates as shown in Figure 1–77.

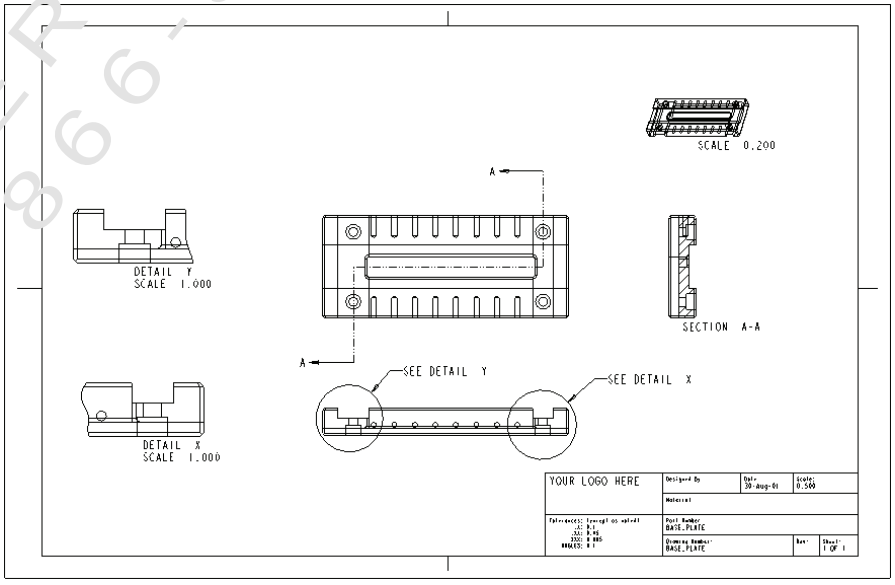


Figure 1–77

Task 7 - Move views as necessary.

1. By default, all views are locked. To unlock them, right-click in the view and clear **Lock view movement**, as shown in Figure 1–78.

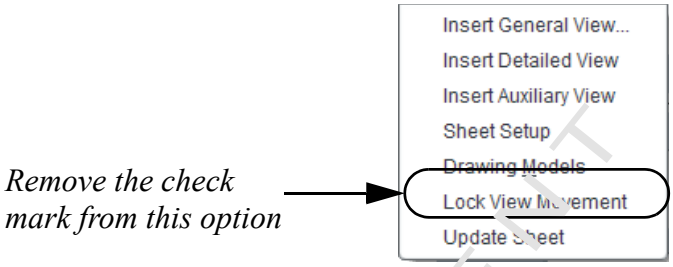


Figure 1–78

2. Once unlocked, you can select the view and drag it on the drawing sheet as required.
3. Relock the views once moved (recommended).
4. Save the drawing and close the window.

Exercise 1c

Create Views

In this exercise, you create partial, auxiliary and partial section views of the spindle part shown in Figure 1–79.

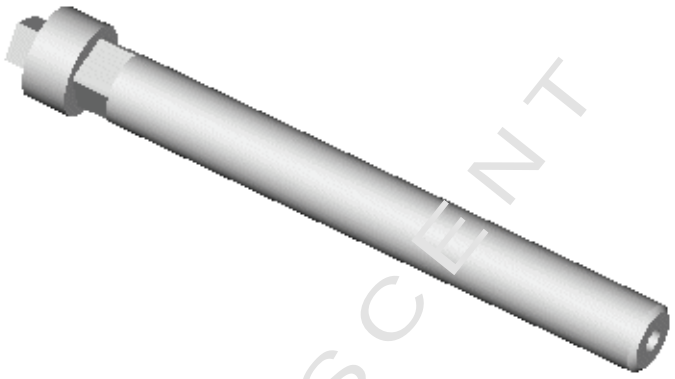


Figure 1–79

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a general view with partial sections**
- ✓ **Create a projection view**
- ✓ **Create a partial view**
- ✓ **Create an auxiliary view**

Task 1 - Create a new drawing.

1. Create a new drawing called [spindle] and clear the **Use Default Template** option.
2. Select **spindle.prt** as the default model.
3. Specify the template as **Empty with format** and select generic_b.frm.
4. Click OK.

Task 2 - Create a General view with a local cross-section.

Ensure the Layout tab is active.

1. Right-click and select **Insert General View**.
2. Place the view in the center of the drawing sheet. The Drawing View dialog box appears.

3. In the Drawing View dialog box, orient the view using the **Geometry references** option so that datum plane DTM1 faces Right and datum plane DTM2 faces Top. The view appears as shown in Figure 1–80.

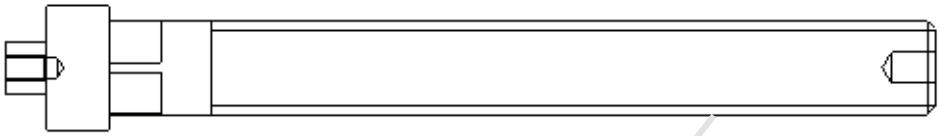


Figure 1–80

4. Select the **Sections** category in the left frame of the Drawing View dialog box.
5. Select **2D cross-section** to assign a cross-section to the view.
6. Maintain the **Total** option in the Model edge visibility section to ensure that all edges are visible throughout the model, and not only for the 2D area of the section.
7. Click  .
8. Select cross-section A to add a section to the view.

Design Considerations

The section is predefined in the model. A cross-section defines a slice through a model. Cross-sections are created in a part by selecting the Xsec tab in the View Manager dialog box. The planar cross-section type is created.

9. Expand the Sectioned Area pull-down menu adjacent to cross-section A in the dialog box.
10. Select **Local** as the sectioning area for this view. This enables you to define only a portion of the view that is displayed with cross-hatching.

11. Select the centerpoint for the Detailed view, as shown by the X in Figure 1–81. The reference appears in the Reference column.

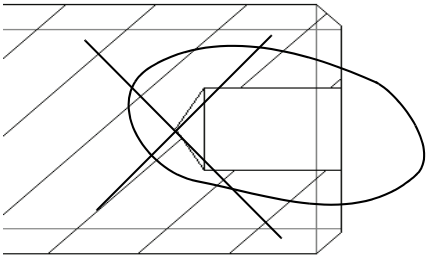


Figure 1–81

12. Sketch a closed spline, as shown in Figure 1–81. This area defines the boundary of the local section.

13. Click  . Select cross-section A to add another section to the view. This view is automatically assigned as **Local**. Keep this option selected.

14. Repeat Steps 12 and 13 to define the next local section on the opposite end of the spindle, as shown in Figure 1–82.

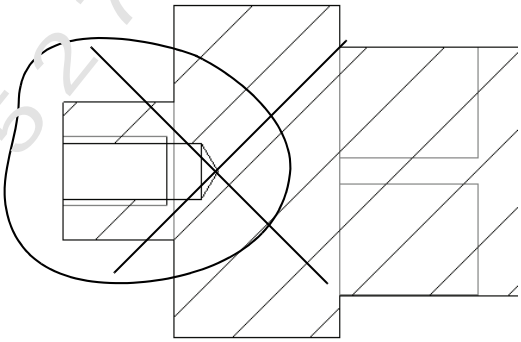


Figure 1–82

15. Click Apply to apply the changes. The Drawing View dialog box appears as shown in Figure 1–83.

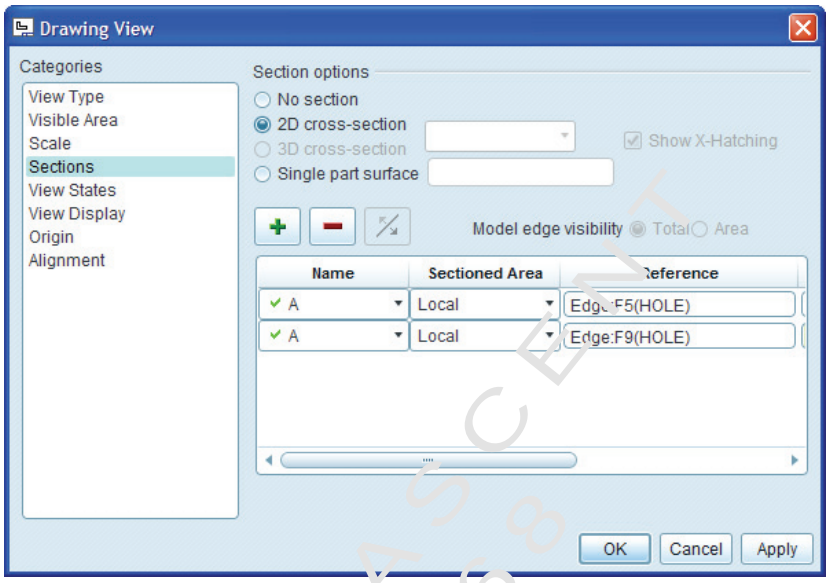


Figure 1–83

The partial sectioned view appears as shown in Figure 1–84.

Figure 1–84

16. Close the Drawing View dialog box to complete view creation.

Task 3 - Modify the scale of the drawing.

1. Modify the drawing scale to [1]. The drawing appears as shown in Figure 1–85.

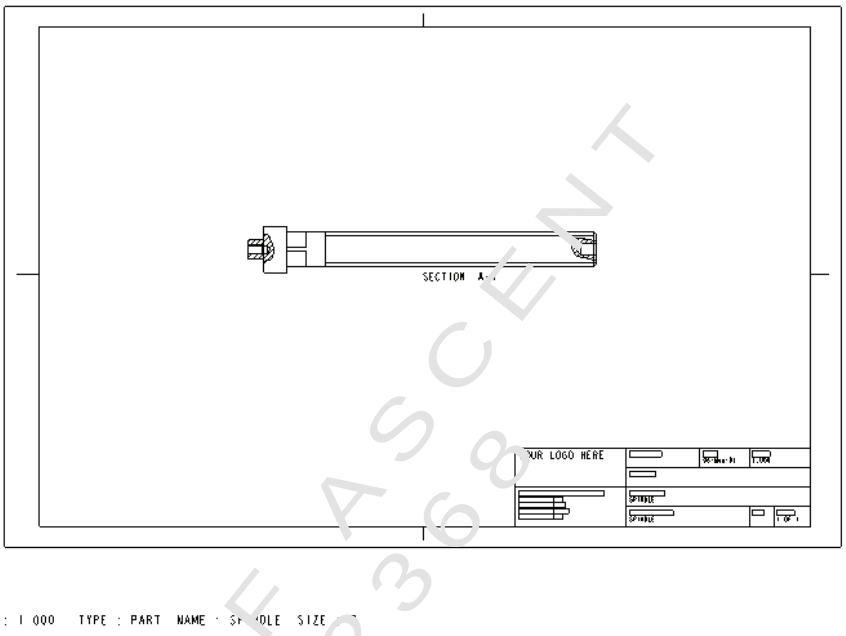


Figure 1–85

Task 4 - Create Projection views in the drawing.

1. Place a Projected view to the left and right sides of the General view, as shown in Figure 1–86.

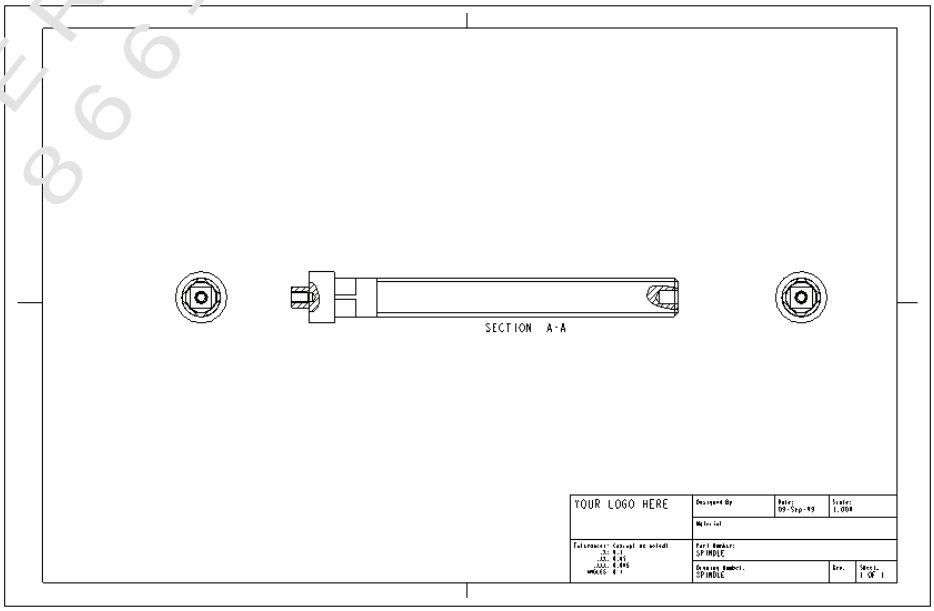


Figure 1–86

Task 5 - Create an Auxiliary partial view in the drawing.

1. Ensure that no views are currently selected in the drawing.
2. Right-click and select **Insert Auxiliary View**.
3. Read the message line. As a reference for the projection, select the hidden edge of the view shown in Figure 1–87.

When an edge, axis, surface, or datum plane is used as a reference, it must be parallel to the projection direction.

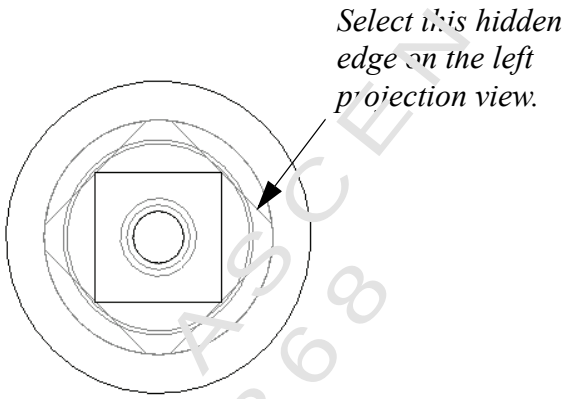


Figure 1–87

4. Select the centerpoint to place the view, as shown in Figure 1–88.

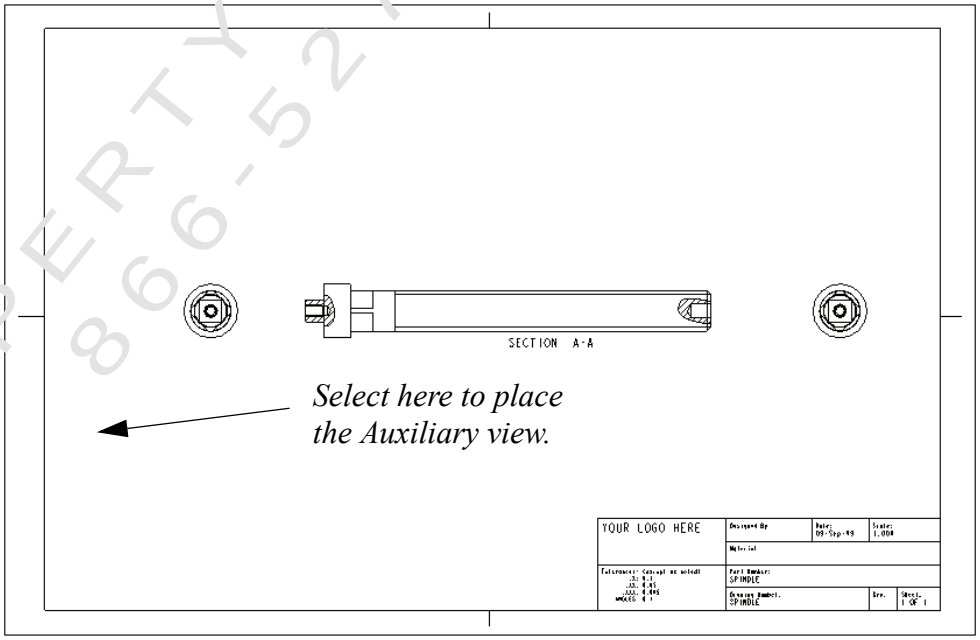


Figure 1–88

The auxiliary view appears as shown in Figure 1–89.

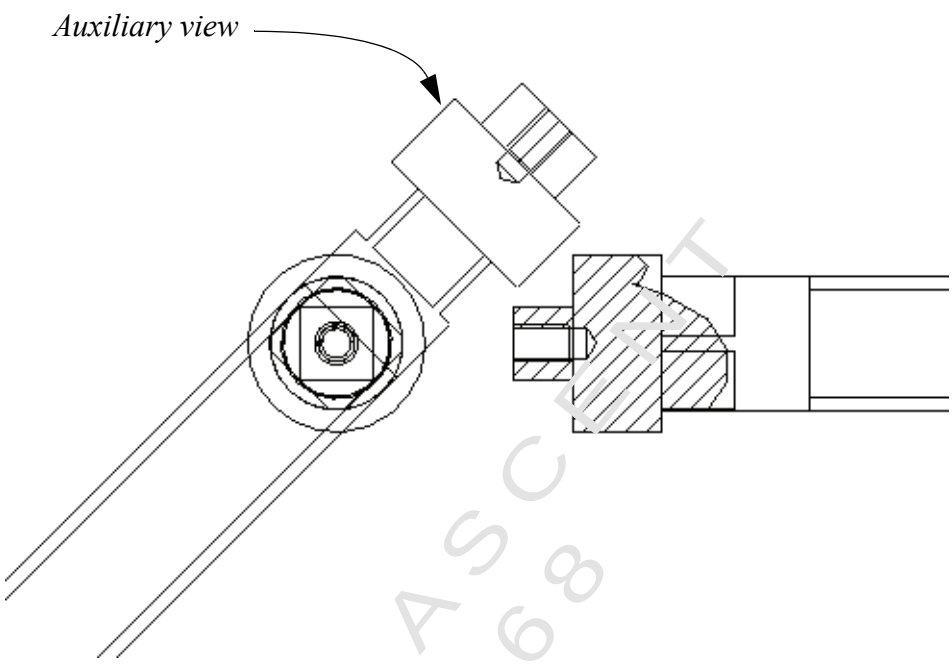


Figure 1–89

5. Select the new Auxiliary view and double-click it. The Drawing View dialog box appears.
6. Select the **Visible Area** category in the left column.
7. Select **Partial View** in the View Visibility pull-down menu.
8. Create the boundary for the Partial Auxiliary view by selecting the centerpoint and sketching a closed spline for the Partial view, as shown in Figure 1–90.

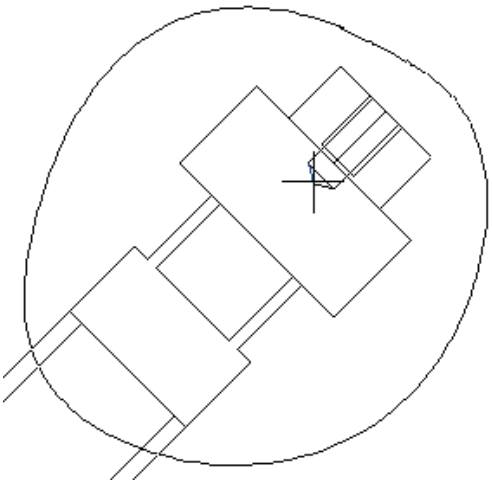


Figure 1–90

9. Click Apply to apply the changes. The Drawing View dialog box appears as shown in Figure 1–91.

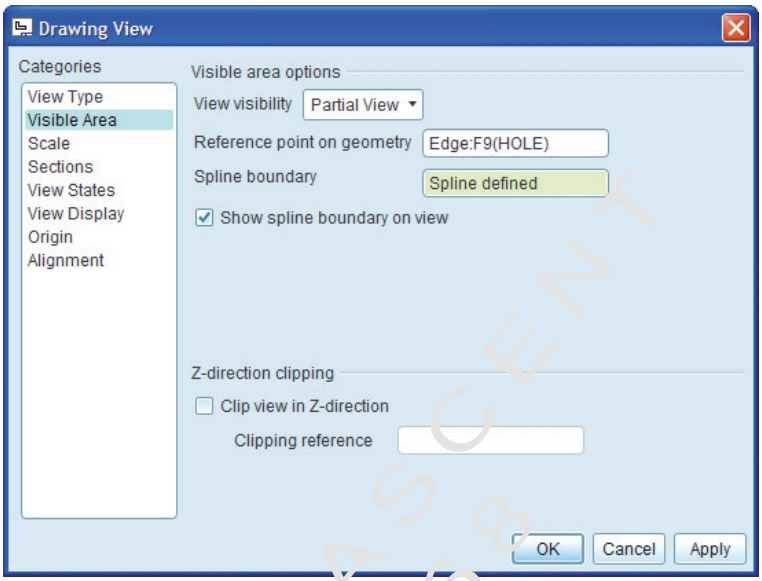


Figure 1–91

10. Close the Drawing View dialog box. The partial view appears as shown in Figure 1–92.

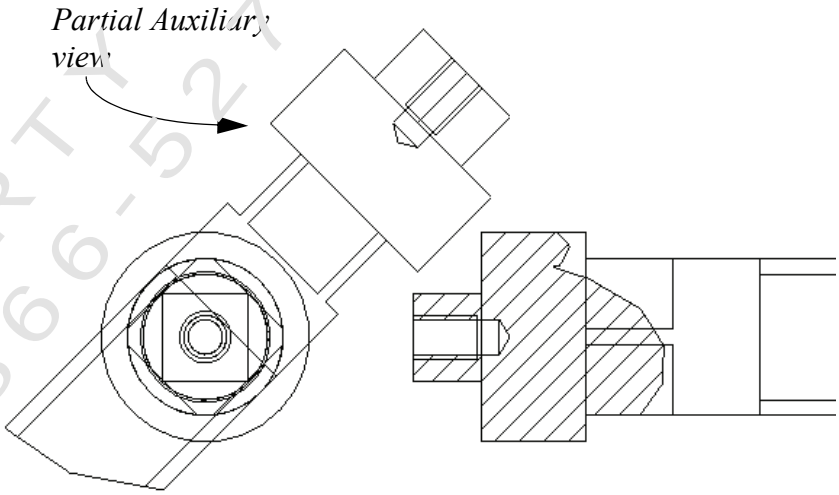


Figure 1–92

Task 6 - Move the Auxiliary partial view.

1. If the views are locked, right-click and clear the check mark beside **Lock View Movement** to unlock them.

2. Once unlocked, select the Auxiliary Partial view and drag the view to the location, as shown in Figure 1–93.

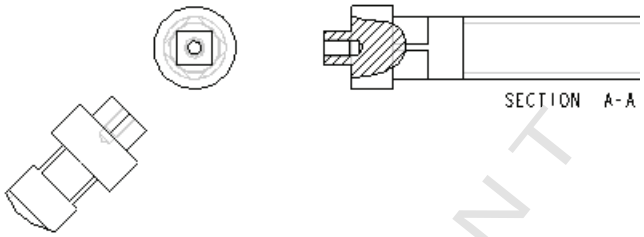


Figure 1–93

3. Relock the views once moved.

Task 7 - Add additional Detailed views in the drawing.

1. Create the detailed views named X and Y, as shown in Figure 1–94, using a scale of 2.0.

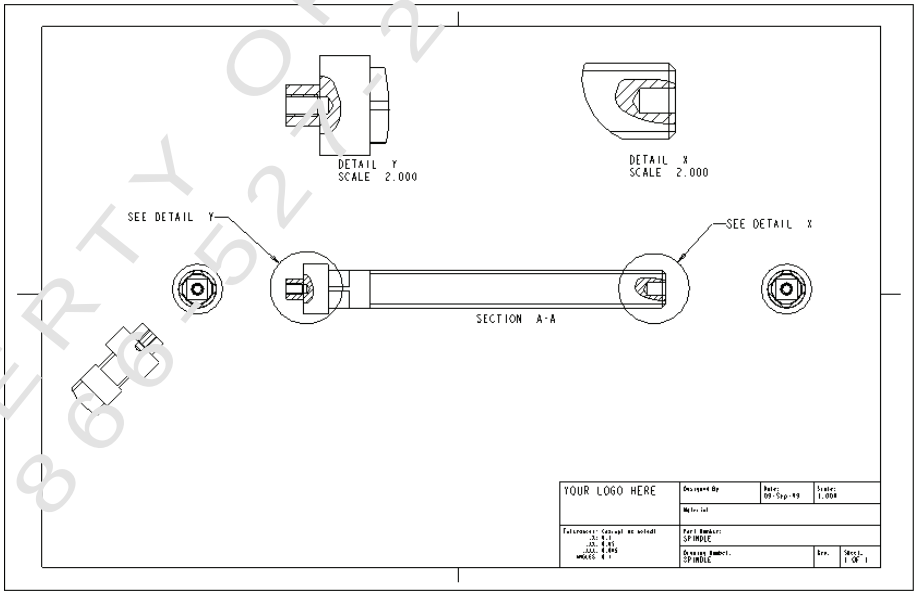


Figure 1–94

2. Save the drawing and close the working window.

Exercise 1d

Create Views with Break Lines

In this exercise, you create a broken view with vertical and horizontal break lines as shown in Figure 1–95.

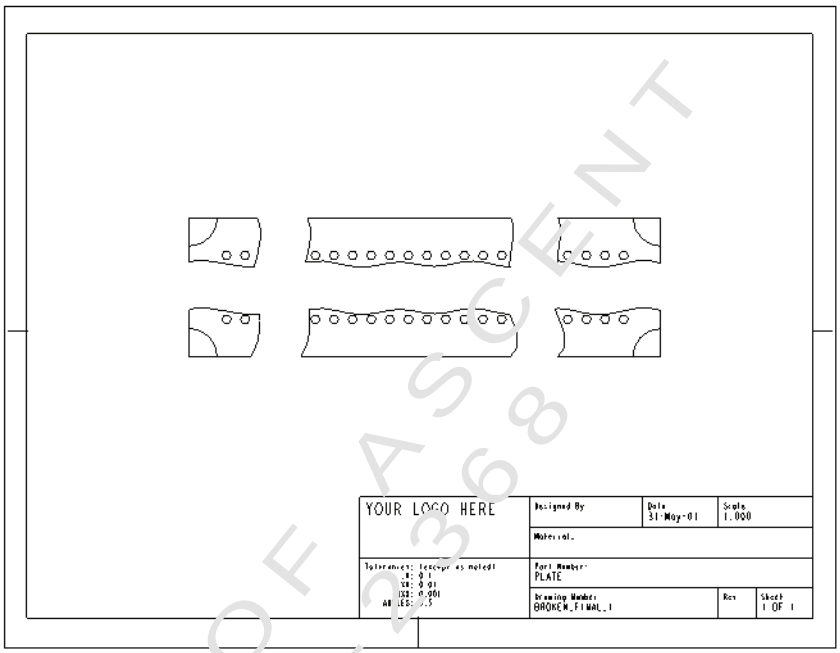


Figure 1–95

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a broken view with vertical and horizontal break lines**

Task 1 - Create a drawing.

1. Create a new drawing called [broken] and clear the **Use Default Template** option.
2. Select **plate.prt** as the default model, specify the template as **Empty with format**, and assign generic_b.frm.
3. Click OK to create the drawing.

Task 2 - Create a Broken view.

Ensure the Layout tab is active.

1. Right-click and select **Insert General View**.

2. Place the view in the center of the drawing sheet.
3. Orient the view using the **Views names from the model** option, and select the **FRONT** saved view.
4. Apply the orientation changes.

Task 3 - Define the break locations.

1. In the Drawing View dialog box, select the **Visible Area** category in the left column.
2. Select **Broken View** in the View Visibility pull-down menu.
3. Click  to add a breakout line to the view.
4. Select the vertical edge of the view in the location shown in Figure 1–96.

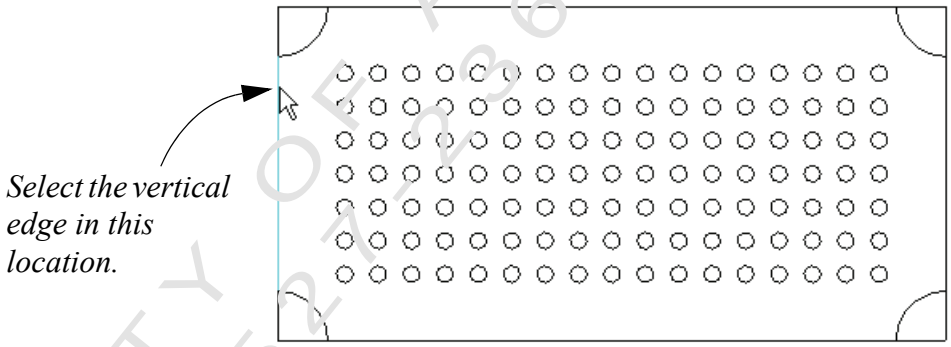


Figure 1–96

5. Move your mouse to the right side and select again to draw the first horizontal break line, as shown in Figure 1–97.

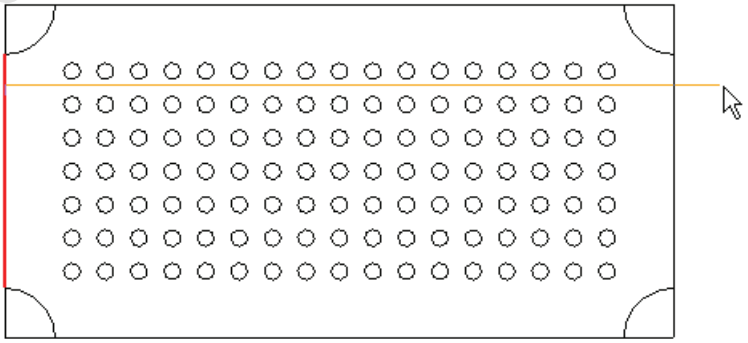


Figure 1–97

6. Select a second break line location, as shown in Figure 1–98.

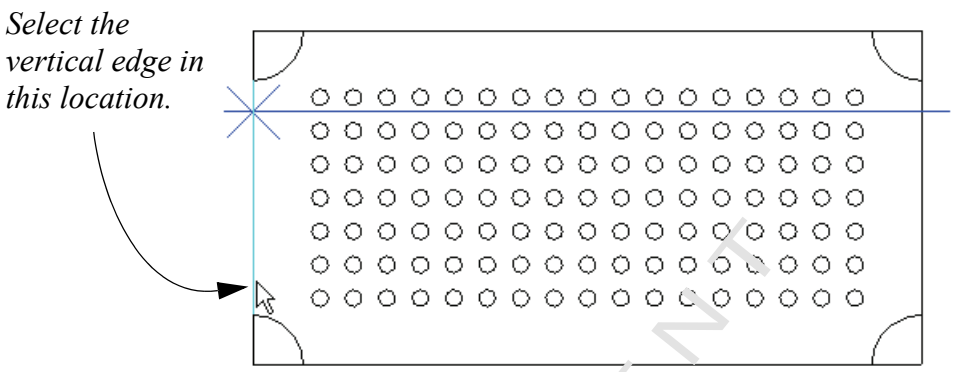


Figure 1–98

A second horizontal break line appears as shown in Figure 1–99.

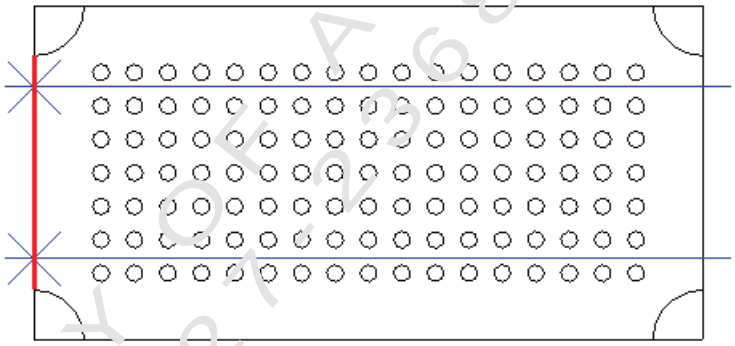


Figure 1–99

7. Scroll to the right of the horizontal break in the Drawing View dialog box.

8. Select **Sketch** from the pull-down in the Break Line Style cell, as shown in Figure 1–100.

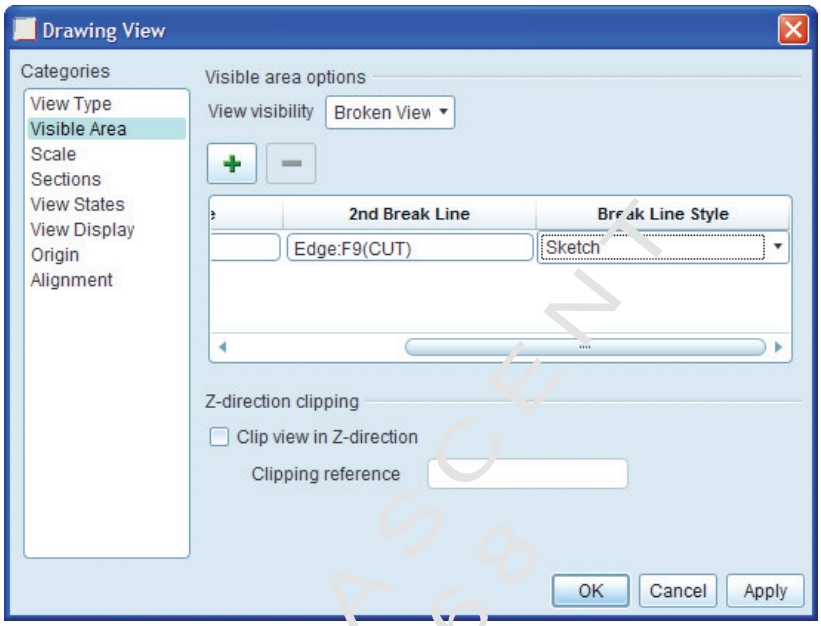


Figure 1–100

9. Sketch the spline shown in Figure 1–101 to represent the break line for the view. Press the middle mouse button to complete the spline.

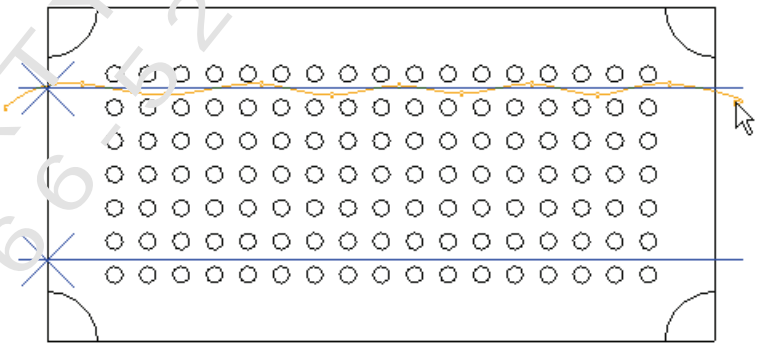


Figure 1–101

The break lines appear as shown in Figure 1–102.

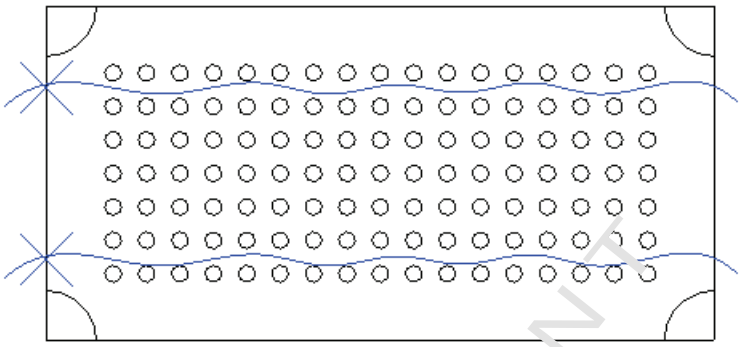


Figure 1–102

10. Click  to add a second set of break lines to the view.
11. Place the break lines, as shown in Figure 1–103. Refer to steps 4 through 6 to review how to place a break line.

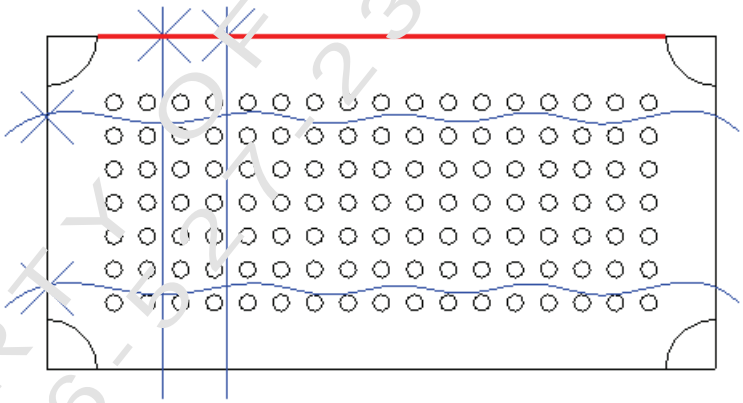


Figure 1–103

12. Scroll to the right of the horizontal break in the Drawing View dialog box.
13. Select **Sketch** in the Break Line Style pull-down menu. Sketch a spline vertically along the first vertical breakline.

14. Add another set of break lines, as shown in Figure 1–104, and sketch a spline in the same way as the others.

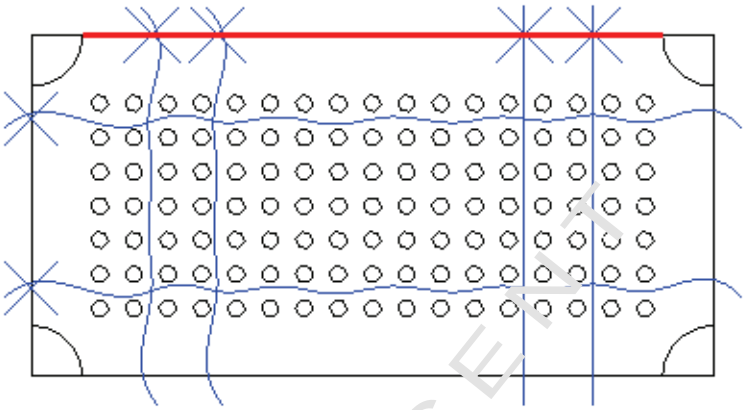


Figure 1–104

15. Apply the changes and close the Drawing View dialog box. The Broken view appears as shown in Figure 1–105.

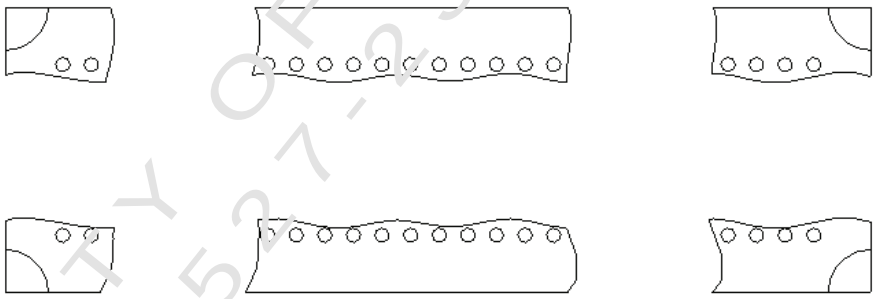


Figure 1–105

16. Modify the drawing scale to [1].

17. The view might not by centered on the drawing sheet. In the next chapter, you learn how to move a Broken view.

18. Save the drawing and close the window.

Exercise 1e

Create a Drawing

In this exercise, you create drawing views from a Pro/ENGINEER solid part. You also assign a format to the drawing. The drawing model is shown in Figure 1–106.

Figure 1–106

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a planar cross-section in part**
- ✓ **Create a new drawing**
- ✓ **Add a drawing format**
- ✓ **Orient general views using saved views**
- ✓ **Create a cross-sectional view**
- ✓ **Create detailed views**
- ✓ **Take specific measurements from a model**

Task 1 - Open a part file.

1. Open **screw.prt**. The model appears as shown in Figure 1–107.

Figure 1–107

Task 2 - Create the first planar cross-sections.

1. Ensure that datum planes are displayed and that Annotation Elements (3D notes) are not displayed, as shown in Figure 1–108.

Figure 1–108

2. Click  to open View Manager. Select the Xsec tab.
3. Create a new cross-section and enter [A] as its name.

4. Accept the defaults in the Menu Manager by selecting **Done** or clicking the middle mouse button.
5. Select datum plane **D3**. The cross-section is complete.
6. Double-click on cross-section A to make it active and to display it in the main window. One side of the cross-section is removed from the display.
7. Select **Display > Visibility** in the View Manager to see the cross-hatching for the section. The model is displayed as shown in Figure 1–109.

Figure 1–109

8. Right-click on the A cross-section in the View Manager and clear **Visibility**. Double-click on **No cross-section** to return the display back to normal.

Task 3 - Create the second planar cross-sections.

1. Create a new cross-section and enter [B] as its name.
2. Accept the defaults in the Menu Manager by selecting **Done** or clicking the middle mouse button.
3. Select datum plane **B**. The cross-section is complete.
4. Double-click on cross-section B to make it active and to display it in the main window. One side of the cross-section is removed from the display.
5. View the cross-hatching for the section by right-clicking on it and clicking **Visibility**. The model is displayed as shown in Figure 1–110.

Figure 1–110

6. Right-click on the cross-section and clear **Visibility**. Double-click on **No cross-section** to return the display back to normal.

Task 4 - Create the third planar cross-sections.

1. Create a new cross-section and enter [C] as its name.
2. Accept the defaults in the Menu Manager by selecting **Done** or clicking the middle mouse button.
3. Select datum plane **C**. The cross-section is complete.
4. Double-click on cross-section C to make it active and to display it in the main window. One side of the cross-section is removed from the display.
5. View the cross-hatching for the section by right-clicking on it and clicking **Visibility**. The model is displayed as shown in Figure 1–111.

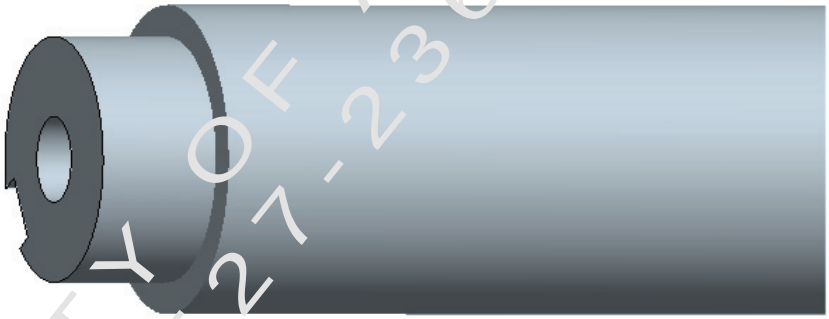


Figure 1–111

6. Right-click on the cross-section and clear **Visibility**. Double-click on **No cross-section** to return the display back to normal.
7. Save the part and close the window.

Task 5 - Create a new drawing called screw.

1. Click  in the toolbar to create a new drawing. The New dialog box appears.
2. Select the **Drawing** option.
3. Enter [screw] as the name of the drawing.
4. Clear the **Use Default Template option** to create an empty drawing.

5. Click OK. The New Drawing dialog box appears.

6. Click Browse... in the Default Model section of the New Drawing dialog box.

7. Select screw.prt in the current working directory and click Open.

8. Select the **Empty with format** option.

9. Click Browse... in the Format section and double-click on generic_b.frm in the formats directory. If the formats directory is not listed in the Open dialog box, click Working Directory to browse to the working directory and then select the format directory. The dialog box appears as shown in Figure 1–112.

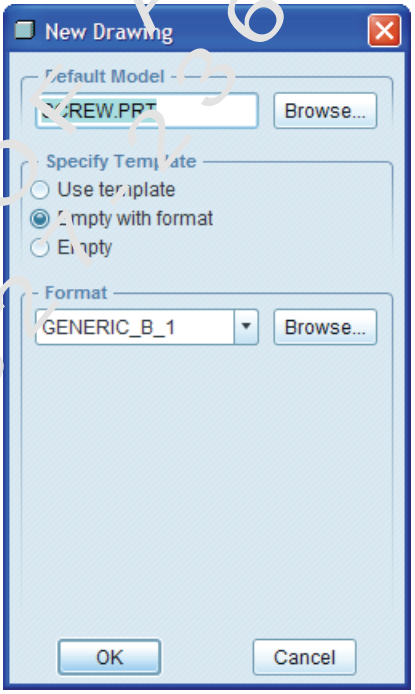


Figure 1–112

10. Click OK to create the drawing.

Task 6 - Create a Broken view and orient it using a saved view.

Ensure the Layout tab is active.

1. Right-click on the drawing sheet and select **Insert General View**.

2. Select the center of the drawing sheet as the centerpoint of the drawing view. The Drawing View dialog box appears.
3. Click **FRONT** in the Model view names section of the dialog box.
4. Click Apply. The view automatically orients itself as shown in Figure 1–113.

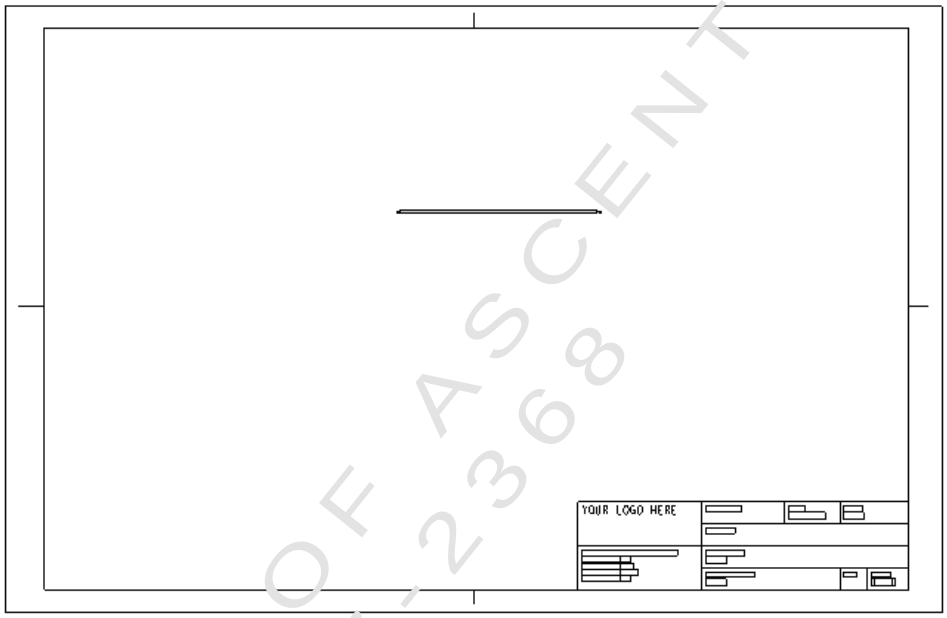


Figure 1–113

5. Click Close to complete the view placement and close the Drawing View dialog box.

Task 7 - Modify the scale value for the drawing.

You can also double-click on the scale value to modify it.

1. Select **Edit > Value**.

2. Select the scale value in the lower left corner of your drawing and modify it to [0.1]. The drawing and view update, as shown in Figure 1–114.

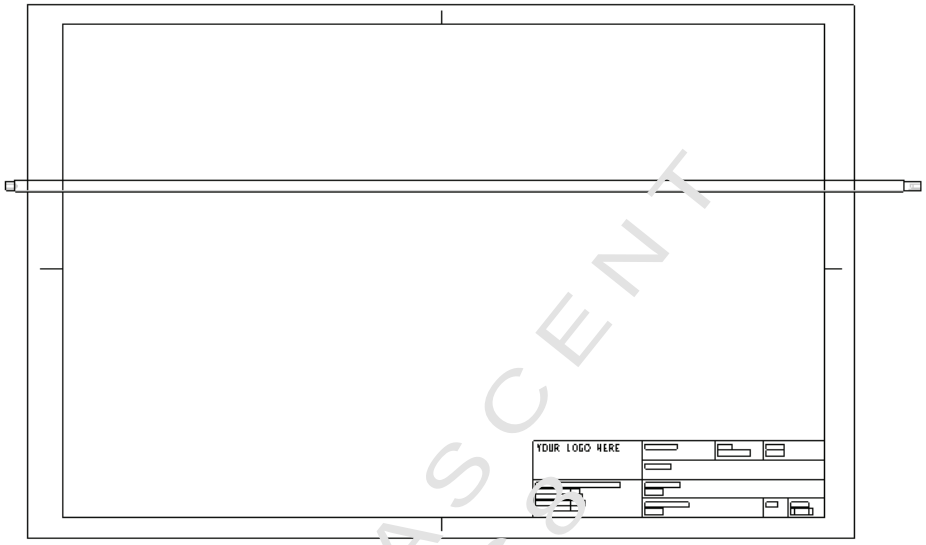


Figure 1-114

Task 8 - Modify a General view with a local cross-section.

1. Select the General view, right-click, and select **Properties**.The Drawing View dialog box appears.
2. In the Drawing View dialog box select the **Sections** category in the left frame of the Drawing View dialog box.
3. Select **2D cross-section** to assign a cross-section to the view.
4. Maintain the **Total** option in the Model edge visibility section to ensure that all edges are visible throughout the model and not only for the 2D area of the section.
5. Click  .
6. Select cross-section A to add a section to the view. The section was predefined in the model. A cross-section defines a slice through a model.
7. Expand the Sectioned Area pull-down menu adjacent to cross-section A in the dialog box.
8. Select **Local** as the sectioning area for the view. This enables you to define only a portion of the view to be displayed with cross-hatching.

9. Select the centerpoint for the Detailed view, as shown by the X in Figure 1–115. The reference appears in the Reference column.

10. Sketch a closed spline, as shown in Figure 1–115. This area defines the boundary of the local section.

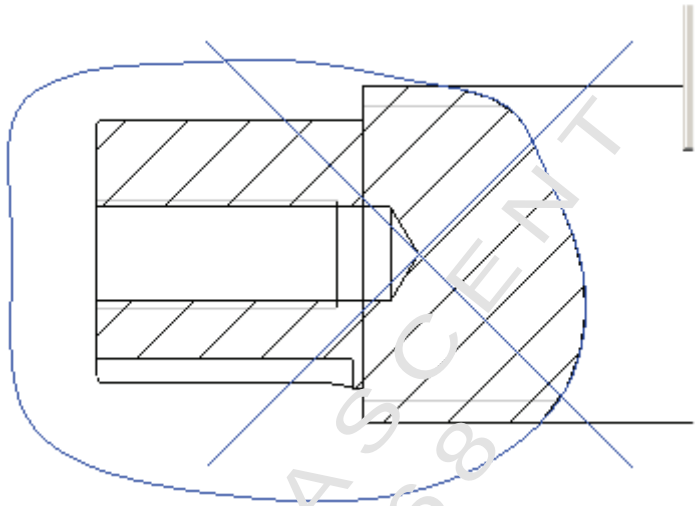


Figure 1–115

11. Click  . Select cross-section A to add another section to the view. This view is automatically assigned as **Local**. Keep this option selected.

12. Define the next local section on the opposite end of the screw, as shown in Figure 1–116.

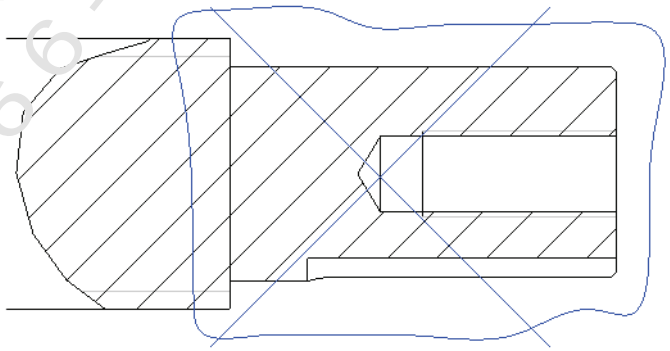


Figure 1–116

Task 9 - Define the break locations.

1. In the Drawing View dialog box, select the **Visible Area** category in the left column.

2. Select **Broken View** in the View Visibility pull-down menu.

3. Click  to add a breakout line to the view.

4. Select the horizontal edge of the view in the location shown in Figure 1–117.

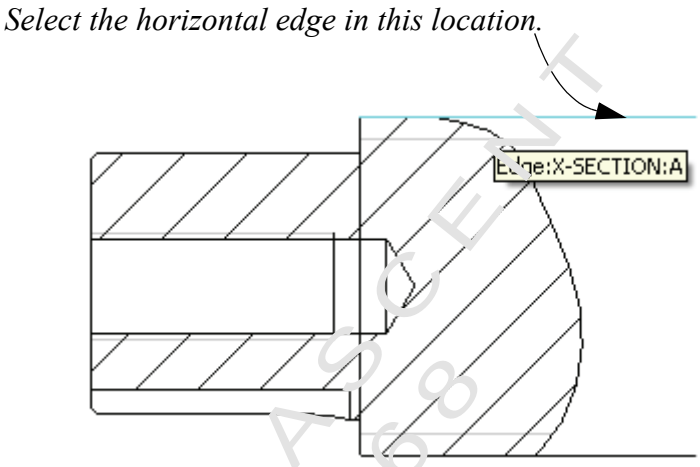


Figure 1–117

5. Move your mouse down and select again to draw the first vertical break line, as shown in Figure 1–118.

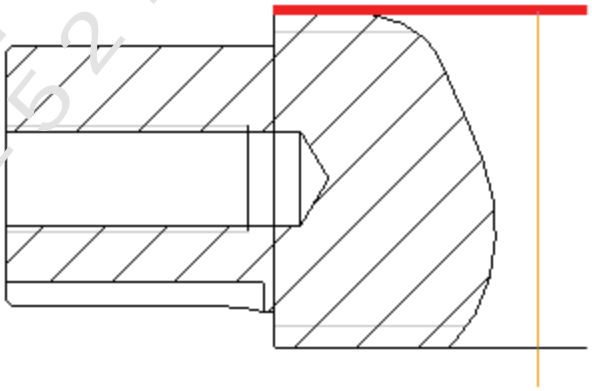


Figure 1–118

6. Select a second break line location, as shown in Figure 1–119.

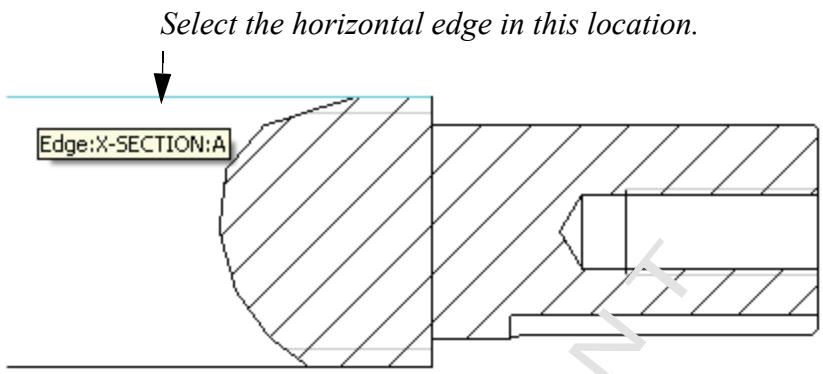


Figure 1–119

A second vertical break line appears as shown in Figure 1–120.

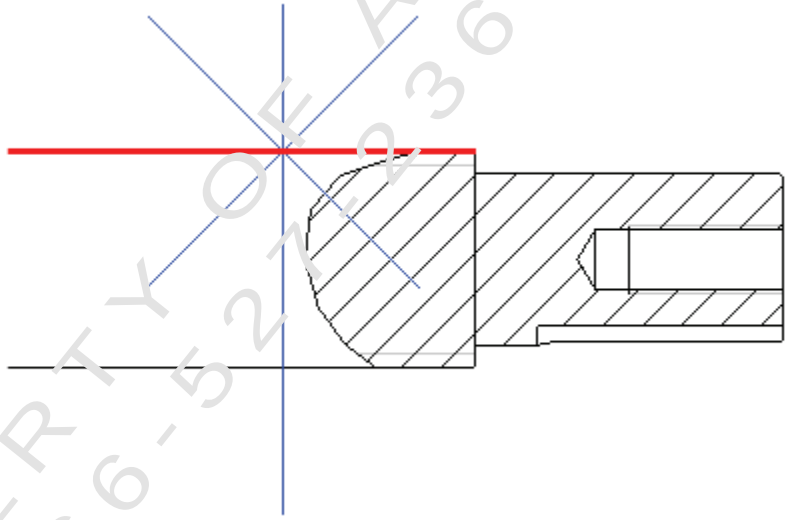


Figure 1–120

7. Scroll across to show the **Break Line Style** column in the Drawing View dialog box.

8. Select **Straight** in the Break Line Style cell to access the pull-down menu and select **Sketch**. The Drawing View dialog box appears as shown in Figure 1–121.

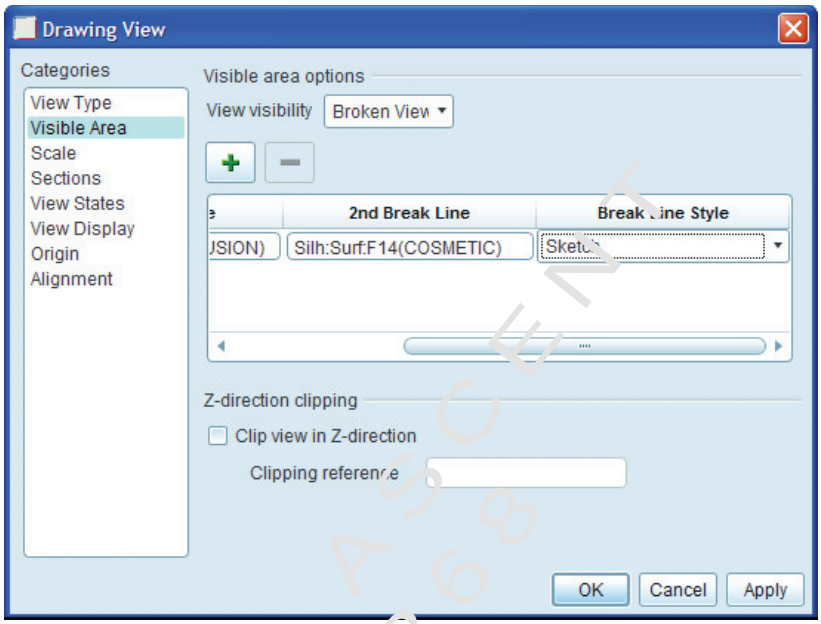


Figure 1–121

9. Sketch the spline shown in Figure 1–122 to represent the break line for the view. Press the middle mouse button to complete the spline.

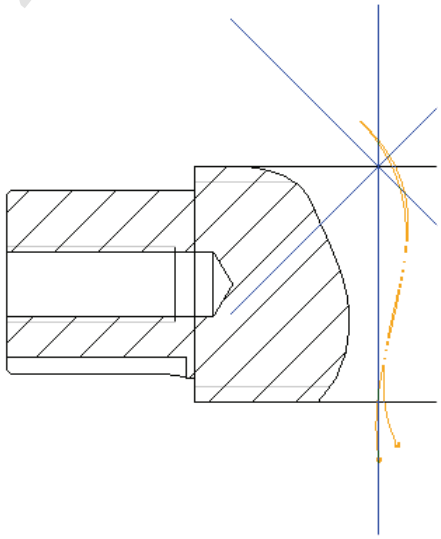


Figure 1–122

The break lines appear as shown in Figure 1–123.

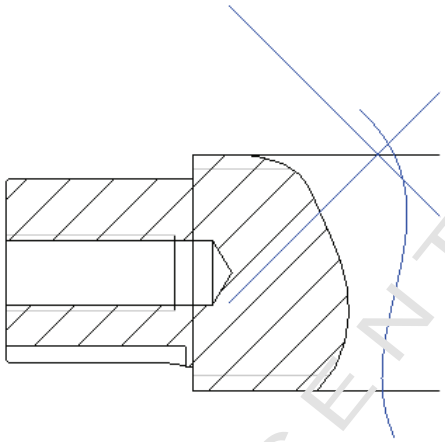


Figure 1–123

10. Apply the changes and close the Drawing View dialog box. The Broken view appears as shown in Figure 1–124.

Figure 1–124

11. The view cannot by centered in the drawing sheet. In the next chapter, you learn how to move a Broken view.

Task 10 - Modify the scale value for the drawing.

You can also double-click on the scale value to modify it.

1. Select **Edit > Value**.

2. Select the scale value in the lower left corner of your drawing and modify it to [0.5]. The drawing and view update, as shown in Figure 1–125.

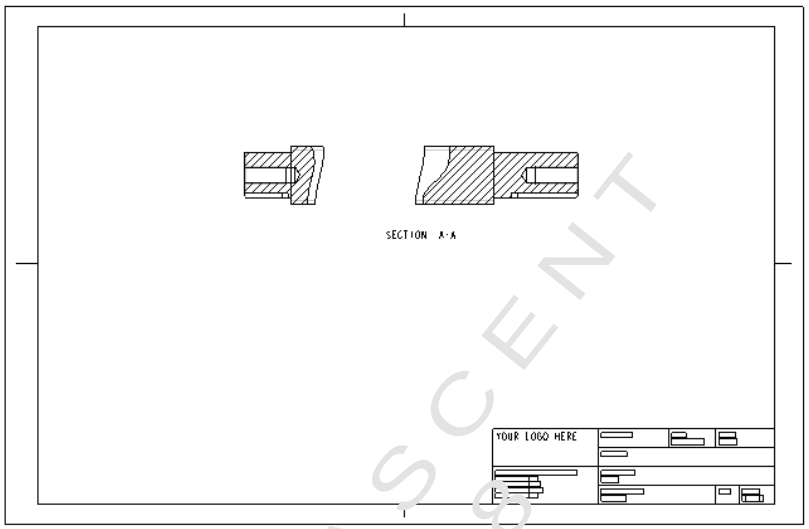


Figure 1–125

Task 11 - Create Projection views in the drawing.

1. Place a Projected view to the left and right sides of the General view with cross-sections B and C. The drawing appears as shown in Figure 1–126.

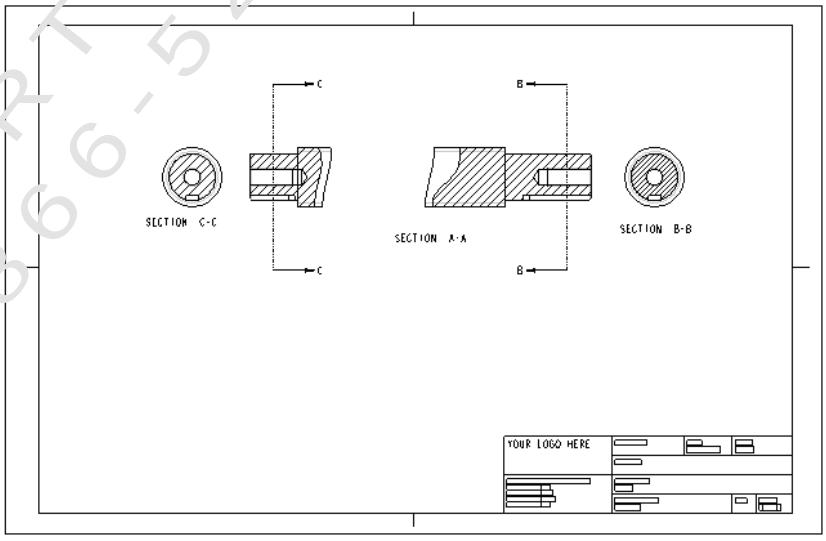


Figure 1–126

Task 12 - Create another General view using the default orientation.

1. Place an additional General view in the lower left corner of the drawing. Create the view with an independent scale value of 0.05. Use the model's default view to orient the model. The drawing updates as shown in Figure 1–127.

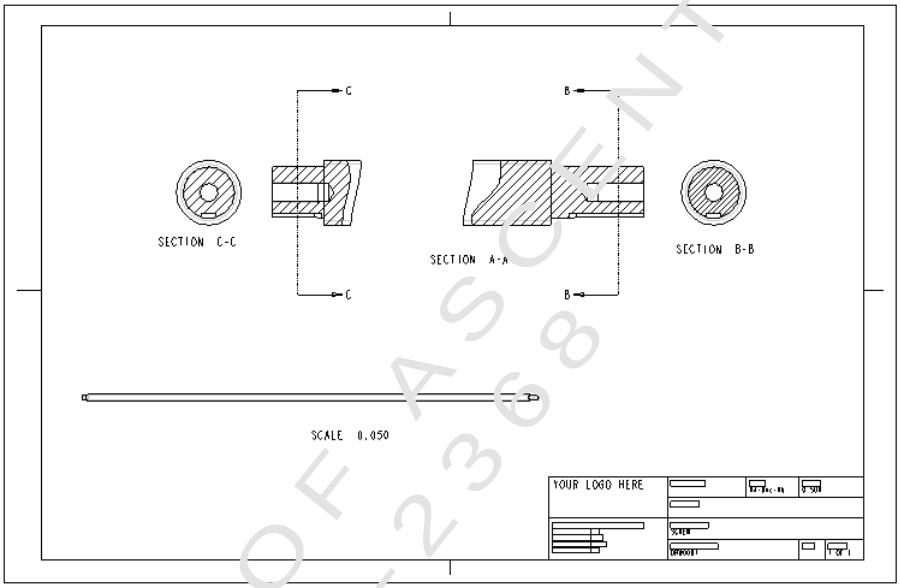


Figure 1–127

Task 13 - Move views as necessary.

1. By default, all views are locked. To unlock them, right-click and clear the check mark beside **Lock View Movement**.
2. Once unlocked, you can select the view and drag it on the drawing sheet as required.
3. It is recommended that you relock the views you have once moved them.
4. Save the drawing and close the window.