

Creation of drawings from PRO/Engineer models is a straightforward task. Completing a drawing can be broken down into two stages:-

1. Drawing Layout - Elements making up a drawing are brought together. These include selecting a drawing sheet, positioning views of your model on the sheet, adding cross-section or scaled views, etc.
2. Drawing Detail - Adding information such as dimensions, geometric tolerances and drawing notes to your drawing.

This document covers the first of these stages, drawing layout, the second stage is covered in a companion document ProTutor05.

This drawing layout tutorial covers the following procedures...

- Creating a drawing sheet and assigning a model to the drawing.
- Positioning drawing views onto the drawing sheet. The position of the first view of the specified model is important since it determines the layout of other views. Subsequent views are placed as projections of this view and PRO/Engineer automatically determines the view orientation based on the projection mode.
- Additional views can be placed which are not projections. For example it is often useful to add a 3D view (an isometric projection) to the drawing as this can aid visualisation of the part.
- Cross-sections are also a useful tool for communicating ideas. Cross-sections, either planar or dog legged, can be added and numbered quite easily in PRO/Engineer.

The drawing tutorial is based on the main housing of a valve. The model for this part can be found at the location where you found this document. Copy the model called *valve_housing* to your directory before you start.

Creating a Drawing

A new drawing is created using FILE > NEW choosing the type as DRAWING and giving a suitable name (*valve_housing* suggested). At this point the drawing format definition dialog appears, as shown in Figure 1. The default model will be set to none (unless you already have a model open). Use the browse button to locate the model you want to create a drawing of – in this case *valve_housing*.

Also from this dialog box the size of the drawing can be specified. When you are choosing the size bear in mind the size of printer or plotter that is available for the final output. If only an A4 printer is available then

choosing the A0 option is not sensible since by the time the page is shrunk to fit on an A4 sheet the text will be unreadable. For student work it is acceptable to choose an A3 format and plot this onto an A4 sheet as this gives more room for dimensions to be shown. Another way of specifying the size is to choose a Template or a Format. This is like starting with a pre-printed drawing sheet with boxes for drawing title and other information but we will show you how to add this in later. For this model choose Empty, Landscape and select the A3 standard size.

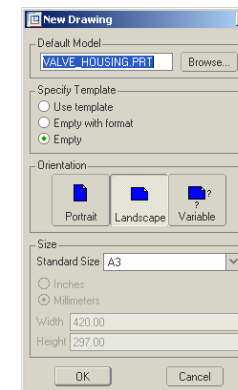


Figure 1: Creating a new Drawing

A new window will be displayed in which your drawing will be created with the file extension .DRW. If you have chosen one of the standard sheet sizes a rectangle will be displayed indicating the extents of the drawing sheet. All drawing should take place inside this rectangle. Figure 2 shows the new drawing sheet, as it should appear on your screen.

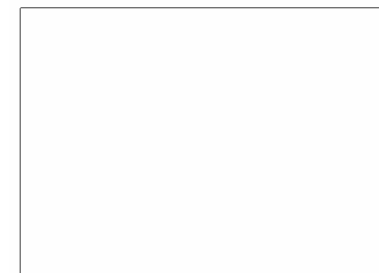


Figure 2: A Drawing Sheet

To position the first view of your model choose INSERT > DRAWING VIEW > GENERAL (if by mistake you left the model name as none in the original dialog box you will be asked to enter the name of the model that you want to detail - choose *valve_housing*). After the system has located the model you are asked to indicate the position of the view within the drawing. Click inside the drawing in the lower right-hand quadrant (see Figure 3) and a default view of the housing model will be shown.

Next the DRAWING VIEW dialog will appear. This dialog offers all of the options for setting up views.

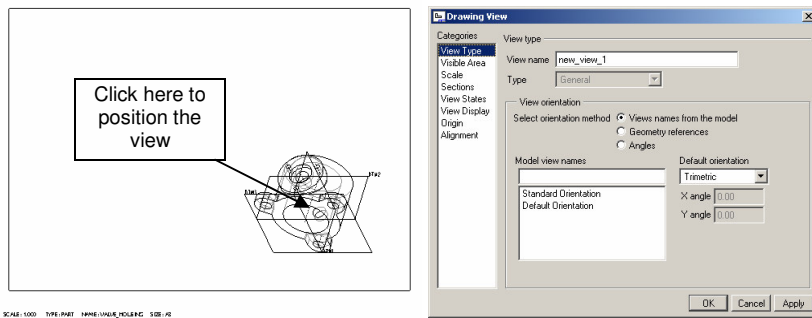


Figure 3 : The First View is Placed and Awaiting Orientation

Although the view is located on the drawing it is not correctly orientated. The VIEW ORIENTATION section of the dialog allows you to change this. If you click on GEOMETRY REFERENCES this works in the same way as orientating the display and setting up sketch planes. Choose *TOP* in reference 1 and click on DTM2, then choose *LEFT* in reference 2 and click on DTM3. The view should be orientated to show a side view as in Figure 4. Choose **OK** in the DRAWING VIEW dialog.

The projections from this first view can now be created. Choose INSERT > DRAWING VIEW > PROJECTION and click to the left of the first view.

The view is projected from the currently selected view so click on the first view you created (a red box should highlight around the view) then repeat the process to add a third view this time clicking above the first view. You should now have three projected views on the drawing.

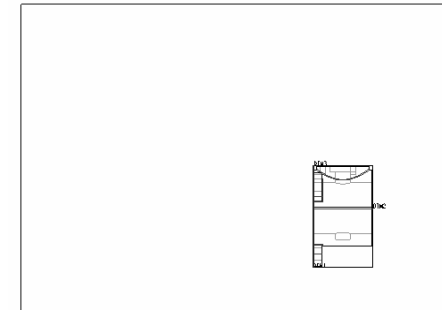


Figure 4 : The First View is Orientated

To complete the views choose INSERT > DRAWING VIEW > GENERAL again. Locate the view in the top-left quadrant and accept the default orientation by pressing **OK** in the DRAWING VIEW dialog.

Note : If you want some isometric view other than the default it is best to open the model and using the dynamic rotation option using the middle mouse button set up the view that you want and save it under a name using the ORIENT tab in the VIEW > VIEW MANAGER dialog.

The drawing should now look like Figure 5. If any of the views are incorrect click on the view to highlight it with a red outline then choose EDIT > DELETE. You will be asked to confirm removal then you can add it again.

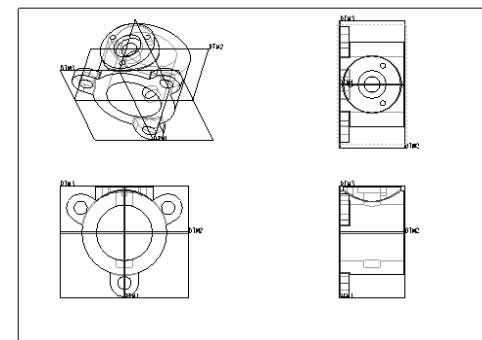
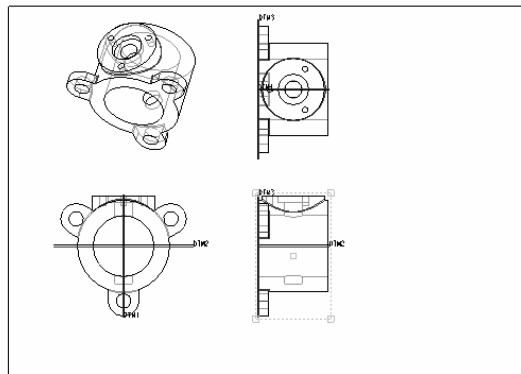


Figure 5: Four Views Positioned

The views are positioned but they can be moved if you wish. You may decide for example that an extra view is needed showing a cross-section. To accommodate this, the first view you placed needs to be moved to the

left. The movement of views is probably locked – check the icon on the toolbar is NOT pressed. Now click on the bottom right view (the one you placed first). A dotted box will be drawn around it with 'grab' handles at the corners (if you don't get the grab handles you need to press the icon). Now click and drag the view to the left. Note that the view above it also moves because it is a projection from the view that was moved. The first view placed can be moved freely. Projected views can only be moved along the projection. If you tried to move the top view to the left or right it would not move, it could only be moved up or down. Try it and see.



SCALE:1000 TYPE:PART NAME:VALUE_POLENG SIZE:A2

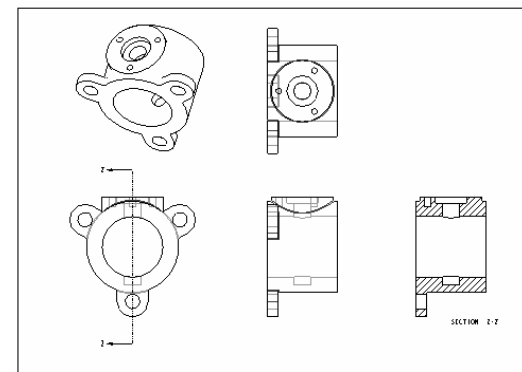
Figure 6: More Room Available

Having made room for an extra view to the right of the first view you can now add a cross-sectioned view. A cross-section is a view with some of the model cut away to reveal the inside. Choose INSERT > DRAWING VIEW > AUXILIARY. If this is greyed out it is because you have a view highlighted in – click in the graphics window away from any view to make sure none are highlighted in red. An auxiliary view is one that is aligned with the primary view, like a projection, but the view orientation can be chosen independently. To orient the view you will be asked to select a datum as the front surface for this view pick on DTM1 in the bottom left view. Click to the right of the first view to locate the view on the sheet.

The view is not sectioned yet so select it and choose EDIT > PROPERTIES (or double click on the view). The DRAWING VIEW dialog should appear. On the left of this dialog click on SECTIONS then click on 2D CROSS-SECTION and finally the icon. An 'old-fashioned' style menu will appear on which you should accept the defaults of a PLANNAR | SINGLE cross-section. After choosing DONE you will need to enter a name for the section. This is usually a single capital letter such as Z. Next choose DTM1 to indicate where the model is to be cut to create the section. Choose APPLY in the DRAWING VIEW dialog to see the section and if it is correct choose CLOSE.

If you feel the cross-section lines are not suitable, for example the spacing between the lines is too wide, then click on the cross hatching to select it and choose EDIT > PROPERTIES (or just double click on the hatching). On the 'old-fashioned' style menu to the right choose SPACING and OVERALL | HALF.

You may have noticed that hidden details (the lines showing what is going on inside the model) are shown on all views. It is not normal practice to show these lines on isometric or sectioned views. So to finally tidy up the drawing the cross-section (bottom right) and general view (top left) need to have hidden detail lines removed. Select one of these two views by clicking on it then choose EDIT > PROPERTIES. The familiar DRAWING VIEW dialog appears – you need to choose the VIEW DISPLAY option on the left. In the DISPLAY STYLE list box choose NO HIDDEN. APPLY and CLOSE the dialog. The finished drawing layout is shown in Figure 7.



SCALE:1000 TYPE:PART NAME:VALUE_POLENG SIZE:A2

Figure 7: Finished Layout

To fit the views onto the sheet Pro Engineer has chosen an overall scale. The text in the bottom left of the graphics window tells you what the scale is (probably 1.00 in this case). You can change this to increase or decrease the size of all of the views giving you more or less room for dimensions to be added. Simply double click on the scale text at the bottom left and type in a new value – try 0.5 in this case to make the views smaller.

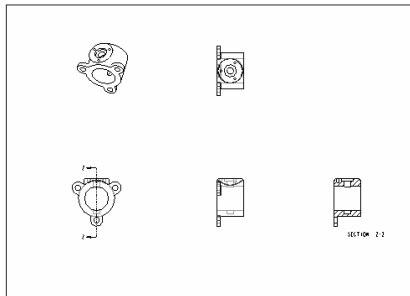


Figure 8: Resized Layout

At the start of the drawing process we said we would show you how to add a drawing format (information boxes etc.) to your drawing so here is how to do it. Choose FILE > PAGE SETUP and you will see the dialog in Figure 9. It shows you that you already have a format in the drawing called A3Size. This is the rectangle bounding the drawing area. Click on this and Browse to change it to b.frm (the standard formats distributed with Pro Engineer are only in American sizes where 'B' is the nearest to A3). Click OK and you should see the format sheet around your drawing.

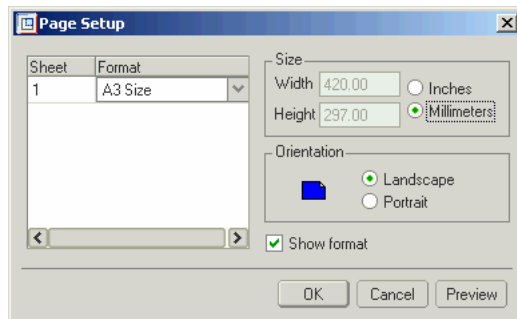


Figure 9 : Page Setup Dialog

The format has boxes where you can fill in your name and other information. To do this choose INSERT > NOTE choose the options from the menu and then choose MAKE NOTE and click to position your text then type in the text you want to appear. Press RETURN twice to end text entry.

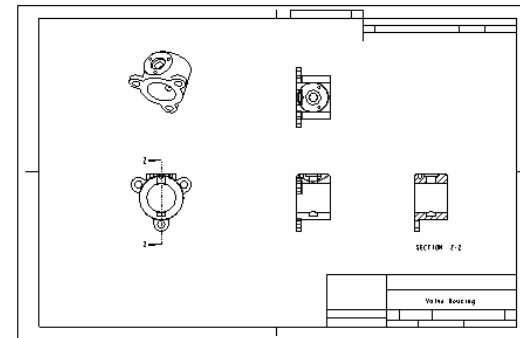


Figure 10 : Drawing with Format

Review

So what should you have learnt?

- How to create a drawing.
- How to create general, projection and sections views.
- How to reposition views.
- How to add a drawing format.

Any problems with these? Then you should go back through the tutorial – perhaps several times – until you can complete it without any help.

What Next?

You need to experiment with creating drawings of your own parts perhaps investigating how to do scaled views.

Now you know how to layout the drawing it would be good to move on to annotating the drawing with dimensions and other information. This is covered in the next tutorial.