

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

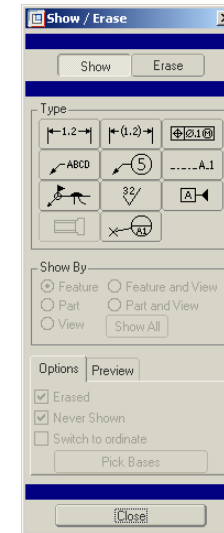
1. Drawing Layout – elements making up a drawing are brought together. These include selecting a drawing sheet, positioning views of your model on the sheet, adding cross-section or scaled views.
2. Drawing Detail – Adding annotation information such as dimensions, geometric tolerances and drawing notes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

The companion document ‘Drawing Layout’ covers the first stage of the process and should be worked through first, the second stage is covered here.

Since the model has already been built all of the dimensions to fully define the model have already been entered it would be stupid to have to enter all of these dimensions again on the drawing. ProEngineer is not stupid and does not expect you to be either! So the first stage of the detailing process allows you to show the existing dimensions on the views. Having done this you will probably find that the position of the dimensions on the drawing needs changing to make the drawing easier to read. Certain dimensions may not be displayed on the most suitable view so this can be changed too. Finally certain cosmetic features may not be to your liking so minor changes can be made to dimensions for example the arrow heads can be flipped around.

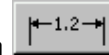
## Showing Dimensions

You should already have completed the companion tutorial so you should have a drawing called valve\_housing which can be retrieved using FILE > OPEN. Having retrieved the drawing lets take the sledge hammer approach and show on this drawing ALL of the dimensions which were used to create the model. Choose VIEW > SHOW AND ERASE and you should see the dialog in Figure 1.



**Figure 1 : Show and Erase Dialog**

In the dialog box which appears click on the button with a picture of a



dimension and then press SHOW ALL and confirm your action. The confirmation step is required because with a big drawing showing all of the dimensions can create a very confusing jumble. You don't always have to crack a nut with a sledge hammer – there are other choices available to you as you can see from the dialog. In this case the drawing is not too complex so this is probably the easiest way however it would be useful for you to investigate the other options. After showing the dimensions you are left in preview mode where you have the choice of picking dimensions to erase or as we require in this case simply pressing the OK menu option to keep all the dimensions shown. The result of this command is shown below. The drawing clearly needs tidying up!

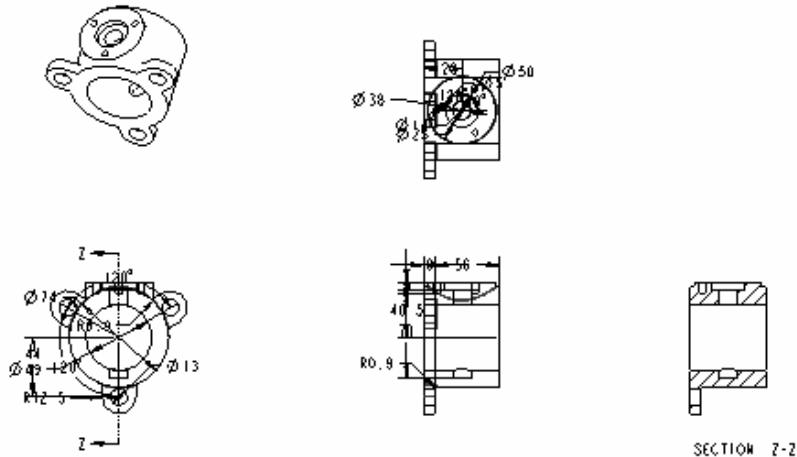



Figure 2 : All of the dimensions shown on a drawing.

## Positioning Dimensions

To see how to tidy up the drawing lets concentrate on the view in the top middle. Zoom in to this view to see it better. The dimensions are all overlapping the drawing so the first thing to do is to position them better.

Use the selection tool  and move each dimension in turn. Click on the dimension to select it, to start moving click again, drag the dimension then when the dimension is positioned better click a third time. Work on this part of the drawing now until the appearance is as shown in Figure 3.

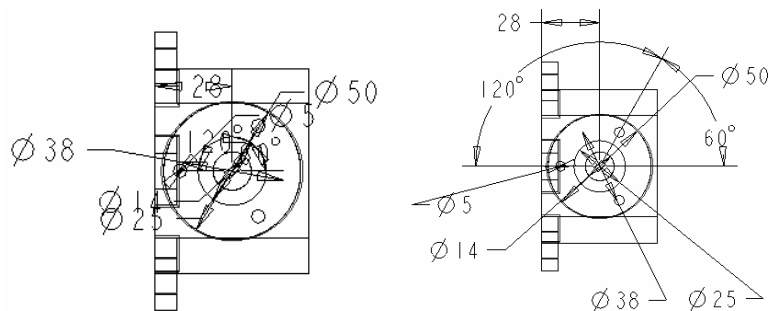



Figure 3 : Improved Positioning of Dimensions

Although this is better there is still work to do. The 28, Ø14 and Ø25 dimensions all refer to the boss and its central holes. Is this the best place to show these dimensions? This feature can be seen much better on the sectioned view. Why not add the dimensions here instead? You will have to zoom out a bit before you can do this so that you can see the section

view as well. Use the selection tool  and select these three dimensions now (hold the CTRL key to select multiple dimensions). Choose EDIT > MOVE ITEM TO VIEW and point to the view that you want them to be moved to – in this case the sectioned view.

Take a closer look at the section view and you will see that the switched dimensions need tidying again. Rather than moving them individually there is an automated way of tidying up the dimensions. You will need to ensure the three dimensions you just moved are selected then choose EDIT > CLEANUP > DIMENSIONS. Set the two tabs in the dialog box to the values shown in Figure 4 and then APPLY to see the changes.

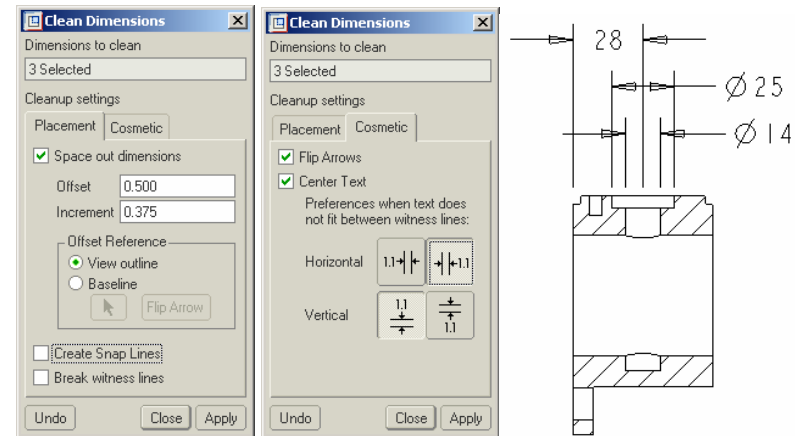
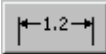


Figure 4 : Cleaning Up Dimensions

Now return to the original top view. If you look at this view you will probably decide that the 120° dimension is not required, since this information is inferred from the 60° dimension. We have shown a dimension that we don't need. The original command we used to show the dimensions can also be used to erase them again. From the pull down menu choose VIEW > SHOW AND ERASE. In the dialog box which appears (see Figure 1) click on the ERASE button and make sure the

button with a picture of a dimension  is pressed and then select the 120° dimension and finally OK and the dimension will disappear. CLOSE the dialog.

Now some of the dimensions have been moved or erased there is more room in this view so try to position the dimensions better. You will probably find that as you move the  $\varnothing 5$  dimension the text stays on the wrong side of the leader line. You will notice that when a dimension is selected small square boxes are drawn on key points of the dimension. These can be moved individually. The one on the end of the text allows you to move the text to the other side of the leader. Also whilst moving a dimension the right mouse button can be pressed to flip the arrows to a new position.

## Dimension Parameters

The  $\varnothing 5$  holes are probably going to be drilled together and so it is good practice to keep all of the information for these holes together as a single note. The actual text of a dimension can be changed too. Select the 5-diameter dimension and from the drop down menus use the command EDIT > PROPERTIES. This will bring up the properties dialog. On the DIMENSION TEXT tab there is a simple editor to modify the text of the dimension. You will see some unusual characters like  $\varnothing @D$ . These make up the existing dimension text – a diameter symbol followed by the value of the dimension represented by @D. Don't change these in any way but add your note to them as follows...

3 holes  $\varnothing @D \times 8$  deep.

This solution looks correct but really it is incomplete. The number 8 is the depth of the hole which appears elsewhere as a dimension. The beauty of Pro Engineer is that all occurrences of a model are linked or associated. You will see later that it is possible to change a dimension in a drawing and the model will be updated to reflect the change and vice-versa. Since the depth of the hole in this note is not a dimension, its just a text note, it will not be updated automatically. A better solution would be to include a reference to the actual dimension in the text note. Here is how to do that.

In Figure 5 you will notice that the highlighted 8 depth and the  $\varnothing 38$  both relate to the 3 holes. If you hover your cursor over each of these in turn you will notice that in the information line at the bottom of the screen it will say 'd49:F21(HOLE)' and 'd50:F21(HOLE)'. The reference d49 and d50 are the names that Pro Engineer uses for the dimensions. Take a note of the name of these i.e. 8 is called d50 and  $\varnothing 38$  is called d49. Having noted

these names, go back to the properties dialog for dimension we were editing earlier. Edit the note replacing the 8 with  $\&d50$  and adding 'on  $\&d49$  PCD' to the end (PCD stands for Pitch Circle Diameter meaning the holes are equally spaced around a circle). DO NOT FORGET TO PRECEED THE NAMES WITH &. The note should now be correct and will be automatically updated if any changes are made to the model. The original dimensions 8 and  $\varnothing 38$  can be ERASED to avoid duplication.

Continue to use the techniques you have learnt to tidy up the remaining views. Be prepared to switch dimensions to other views as you feel necessary. Make sure to use the handles on the end of the dimension extension lines to move them so that they do not overlap the model.

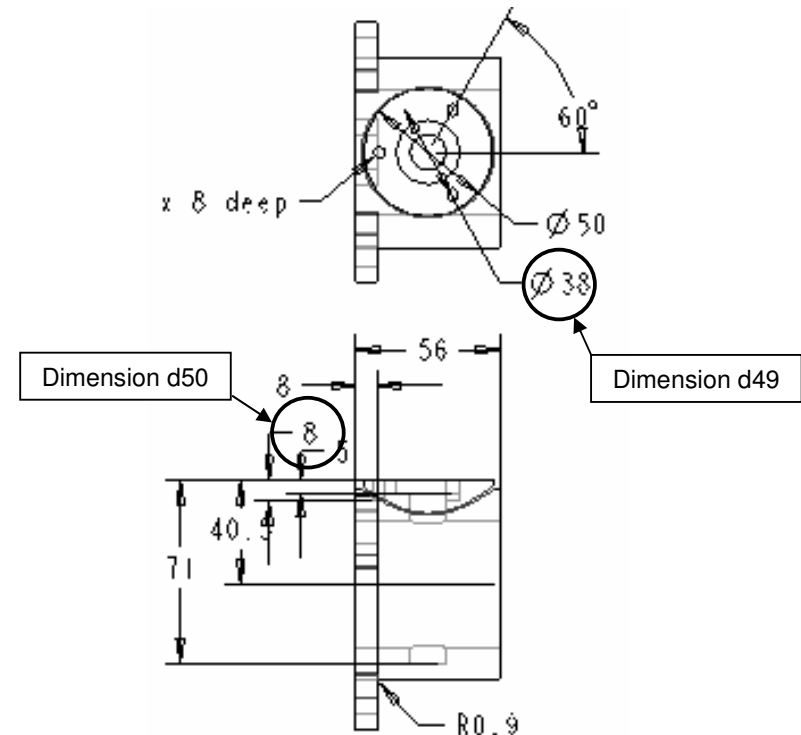
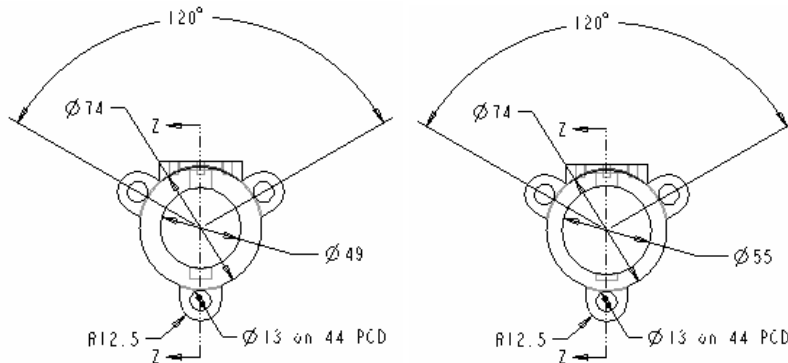


Figure 5 – Adding Parameters to Dimension Notes

## Modifying Dimension Values

Earlier in the tutorial it was mentioned that the links between models and drawings allows dimensional modifications to be made from within the drawing. You have no doubt already seen how modifications can be made within a model. The process from within a drawing is very similar. Just click on a dimension to select it THEN double click on it to edit it (NOTE: If you just double click with out first selecting you will get the Properties dialog you met earlier).

Try this now on the  $\varnothing 49$  for the central bore. Select, double click then type in a new value for this, say 55 and press RETURN. The colour of the dimension will change to show that it has been modified but the model will not change until you choose EDIT > REGENERATE > MODEL. The before and after of this exercise is shown in Figure 6.



**Figure 6 – Model Modifications Can Be Made From The Drawing**

## Printing a Drawing

Printing a drawing is straightforward using FILE > PRINT. One point worth noting is how do you print an A3 drawing on an A4 printer? Simple – in the Print dialog choose Configure – in the Page tab choose Size as A4 then in the Model tab choose Plot as FULL PLOT. The entire drawing should now be scaled to fit onto an A4 sheet.

This completes the drawing tutorial. The completed drawing is shown in Figure 7 at the end of the tutorial.

There are other additions that more advanced users may wish to include in their drawing such as geometric tolerances, machining and other symbols.

These are available from Pro Engineer and you should refer to the Help system under Detailing for more information.

## Review

So what should you have learnt?

- How to show and erase dimensions.
- How to reposition dimensions.
- How to maintain dimension parametrics.

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

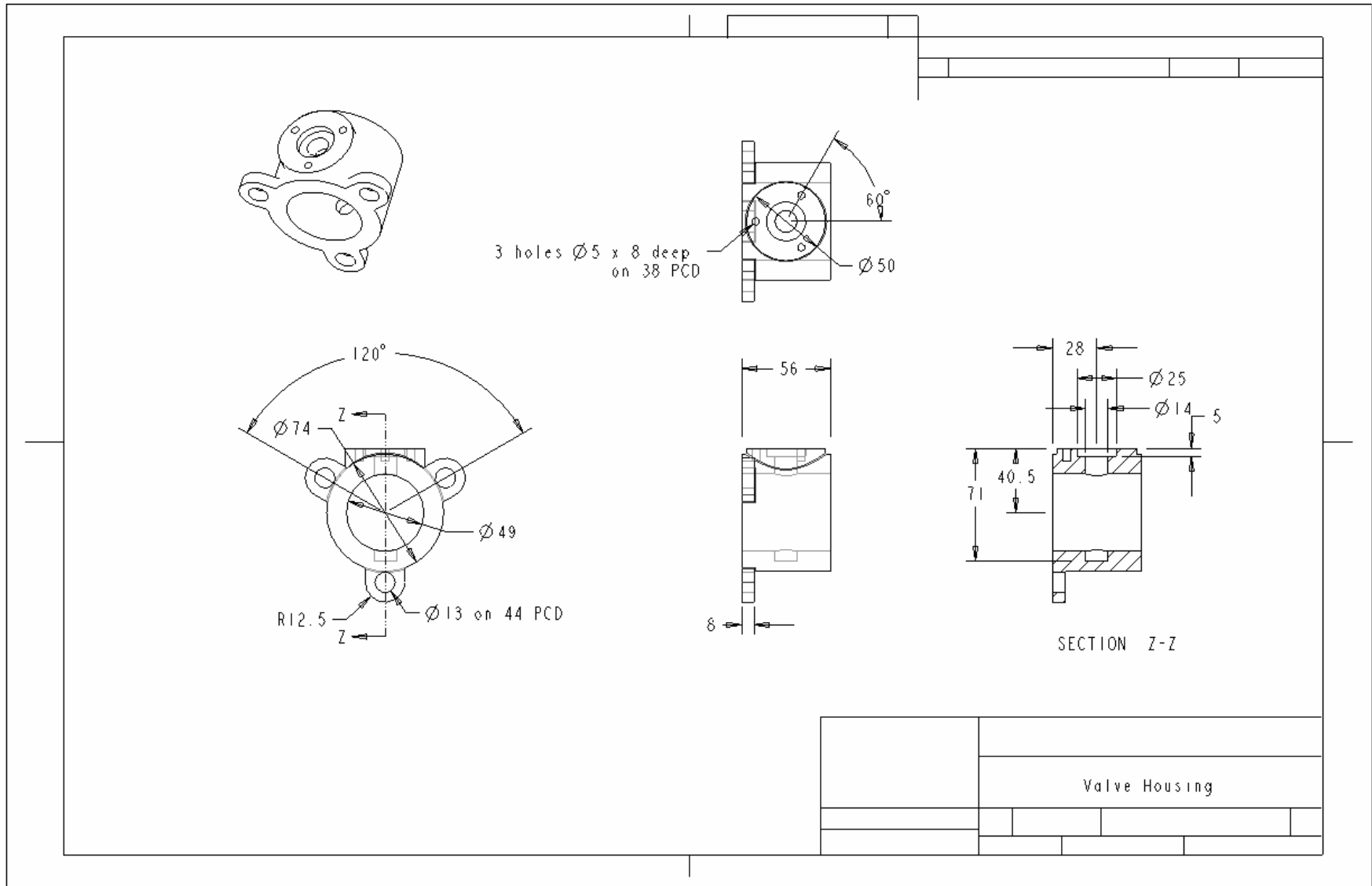


Figure 7 : The Completed Drawing