



Module 5: To create the 2-D Drawing from a 3-D Model

1. Start the program.
2. Select 'Open Document' .
3. Choose the **Housing Exercise** from the location where you have it saved.
 - i. Note the 'preview' window, this window permits you to look at the file before opening it.
 - ii. The 'Files of type' window. The down arrow allows you to display only your selected type of file from the drop down menu. In this case the type of file is a 'part file' and has the file extension (*.prt) or (*.sldprt).

Select 'open'.

4. With the Housing file open select 'New' and choose 'Draw'  from the tutorial window. See Figure 1.

Notice that a preview of a standard drawing border is shown.

Select 'open'.

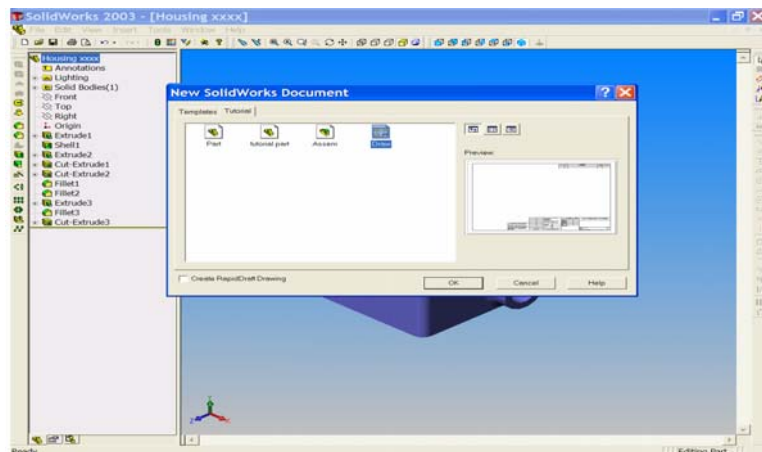


Figure 1

5. In the white drawing area 'Right click' once to open a short-cut window. Select 'Properties' to open the 'Sheet Setup' box. See Figure 2. In the sheet set-up box use the down arrow at the right side of the paper size box to choose the paper size, in this case, **A3-Landscape**.

In the Sheet Format window select **A3-Landscape** to match the paper size. Leave the scale at 1:2 for the time being. Ensure that the type of projection selected is '**Third angle**'. Click **OK** when these selections are made. See Figure 3.

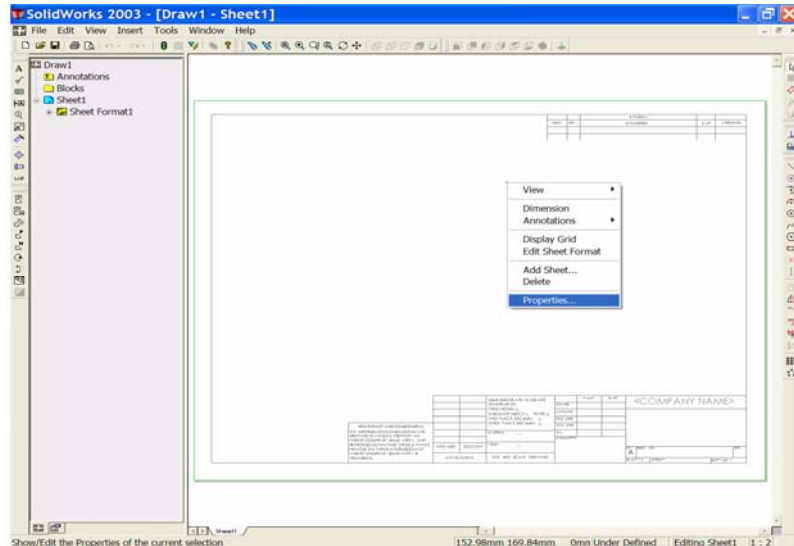


Figure 2

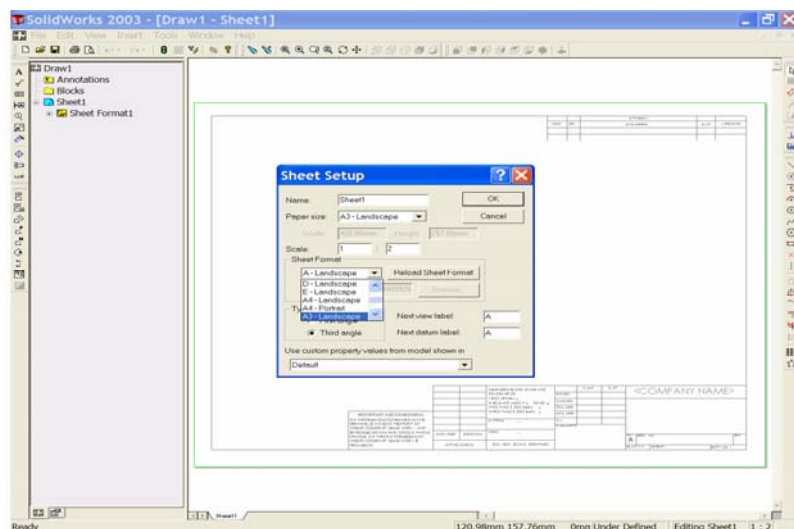



Figure 3

The default border is now replaced by your selected border.

- Use the '**Zoom to Area**'  button from the View Toolbar to examine the border contents. Left click the '**Zoom to Area**' button and draw a window around the '**Title block**' of the border. To do this, place the cursor symbol at the upper left corner of the Title Block, left click and **HOLD THE LEFT BUTTON DOWN**. Drag the cursor symbol to the lower right corner of the Title Block to draw a green window. Release the mouse button and the contents of the window are scaled to fill the screen. See Figure 4. Select the '**Zoom to Area**' button once again to end the command.

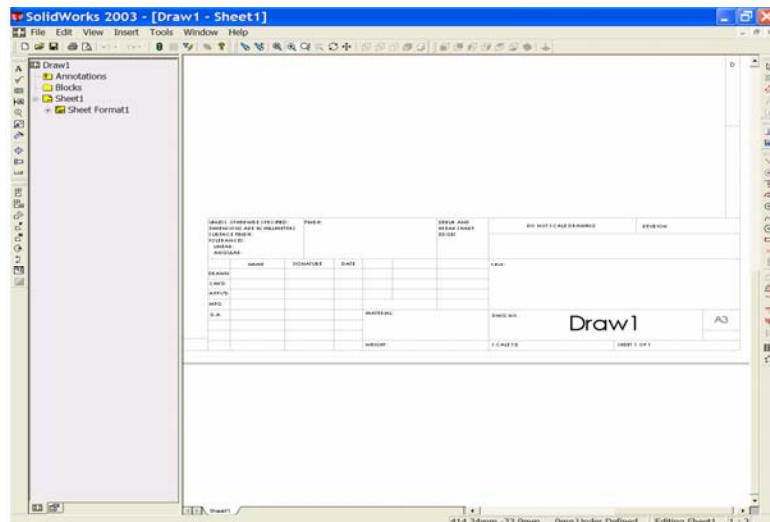


Figure 4

Notice that there are some new Toolbars available at the left side of the screen. The one at the top should be the 'Annotations Toolbar' and the one under it should be the 'Drawing Toolbar'.

7. From the 'Annotations Toolbar' choose 'Note'.

The Note Property Manager opens and a NOTE box is fixed to the cursor.

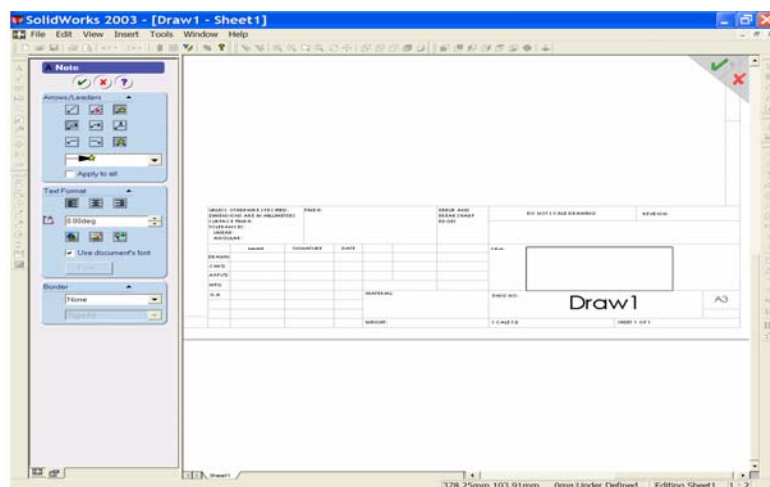


Figure 5

Place the Note box in the Title Area of the Title Block. See Figure 5.

This note box is now active, but the font size is too big.

In the Note Property Manager de-select 'Use document's font'.

By de-selecting this function we now have access to the font button.

Choose the font button to open the choose font box. In the 'Font', area scroll to find 'Book Antigua' and select. In the Height area, select 'Points' and scroll to select a value of 20. See Figure 6.

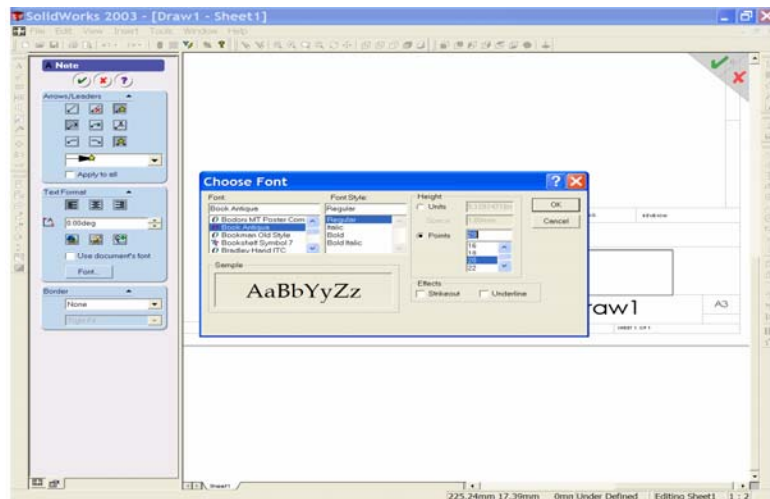


Figure 6

Note that while the units box is disabled (greyed out) its value still changes. Note also that a font sample is shown on screen.

When these changes are made, select **OK**.

8. Left click in the Note box and type the drawing title '**Housing Body**'. When complete, select the green tick in the Note Property Manager.
9. Add '**K.F.U.P.M. Mech. Eng. Dept.**' in the empty box above the title. Adjust the font size to make this new note fit the space available.
10. In the material box add '**Cast Iron**'.
11. Under the Name Box add your '**family name**'. Under signature add '**your ID Number**'. Under Date add the '**date the drawing was made**'. See Figure 7.

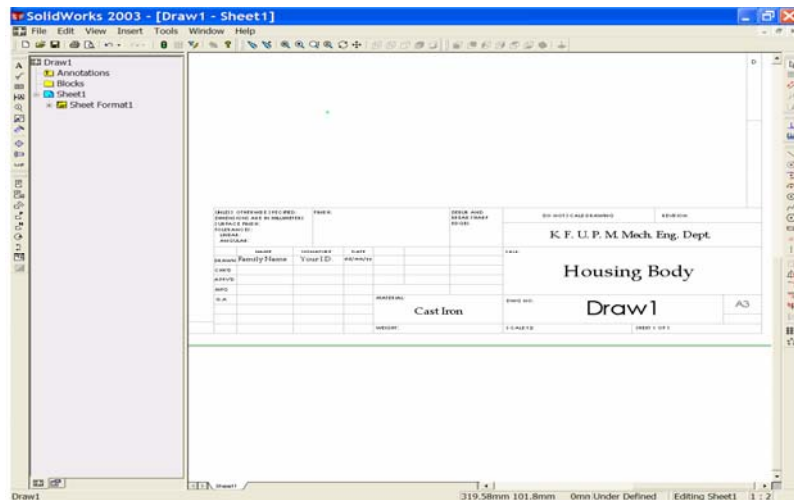


Figure 7

Note that a Dwg No (Drawing Number) is already in place. This drawing number will be the same as your file name.

12. From 'File' select 'save as' and save your work. See Figure 8.

Note that the file type extension is now (.dwt) or (*.slddwt).*

Use the 'rebuild' button  to update your drawing sheet.

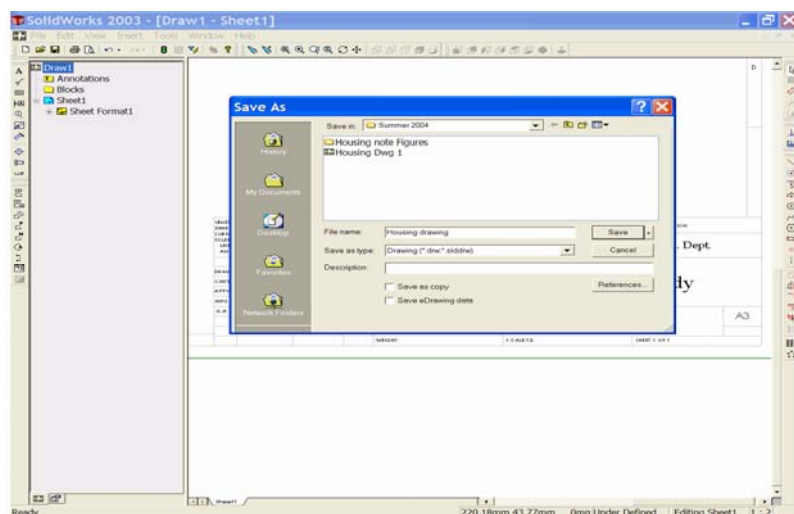


Figure 8

13. To modify the drawing number we must **right click in the drawing area** and select 'Edit Sheet Format' we can now select **the drawing number**. See Figure 9. **Right click on the drawing name** and select 'Edit text'. Choose 'Font' from the Note Property Manager and change the font to match the other notes that you have just added to the sheet. Choose **OK** to update your drawing sheet. See Figure 10.

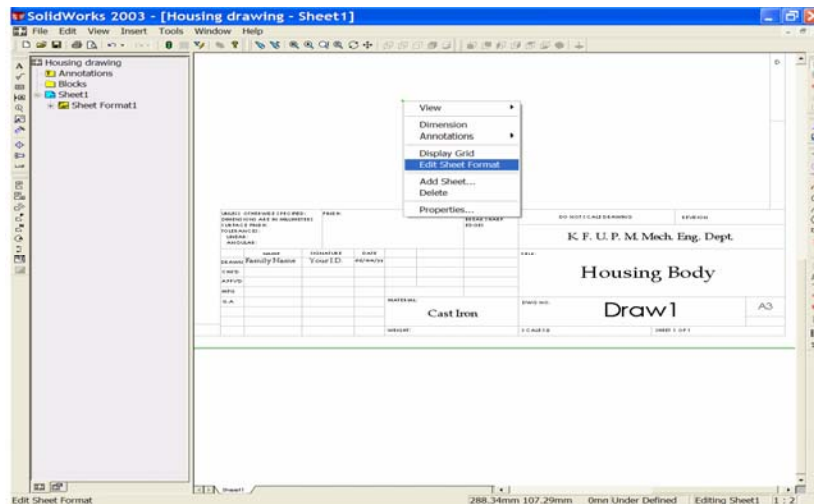


Figure 9

14. Right click in the drawing area and select 'Edit sheet' once again to exit the command.

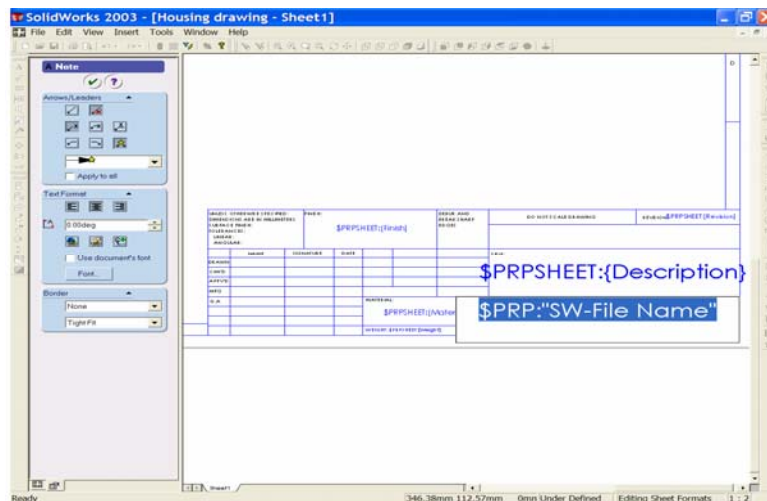



Figure 10

Save your work!

15. The basic preparation is complete so now choose 'Zoom to fit' . (At this stage the drawing border may be saved as standard A3-Landscape template for further use.)
16. *We are now ready to insert the drawing.*

Select 'Standard 3 View' from the 'Drawing Toolbar'. See Figure 11.

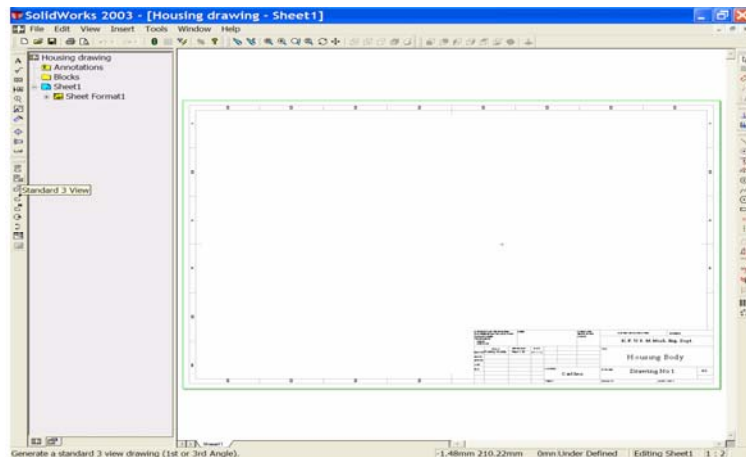


Figure 11

The 'Standard View Property Manager' opens and the cursor is accompanied by an isometric view symbol. The Standard View Property Manager lists four different methods you may use to select a model. In this exercise the model file is open, so we can use either choice 2 or 3. See Figure 12.

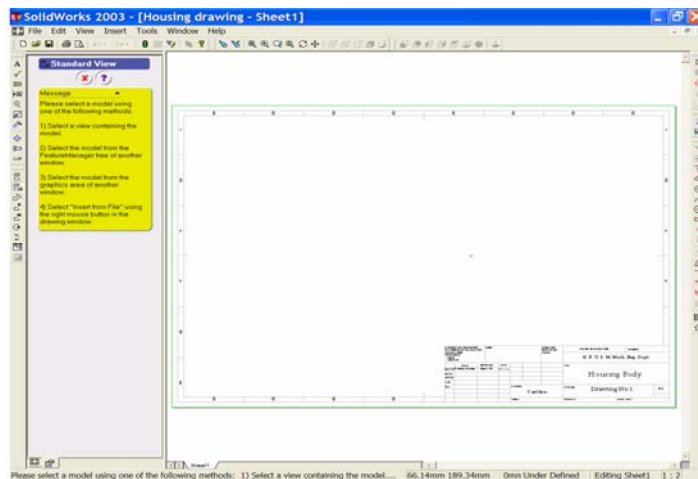


Figure 12

From the window drop down menu, select the Housing Model file. See Figure 13.

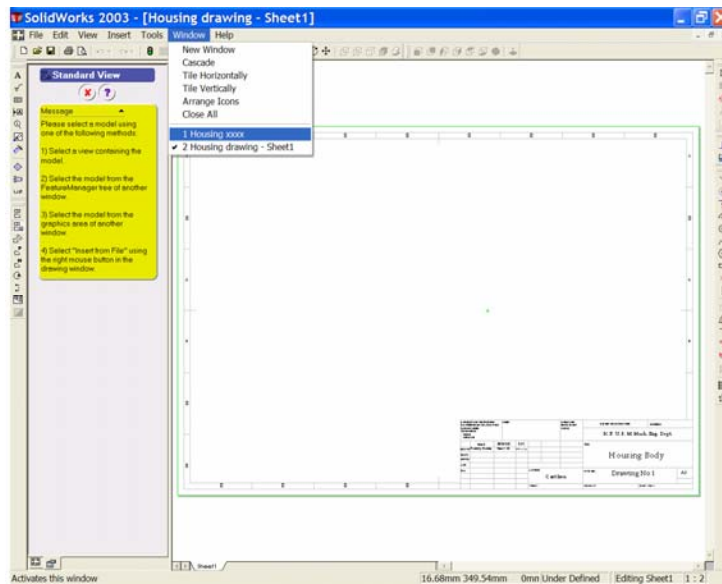


Figure 13

The model window opens with the cursor still indicating the draw symbol.

Left click anywhere in the graphics area of the screen.

The drawing border becomes visible again. We now have to decide if we wish to show 'tangent edges'.

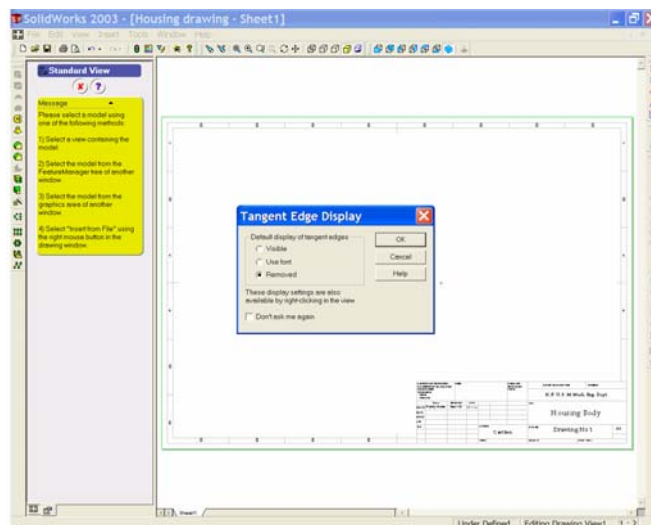


Figure 14

For most circumstances it is best to remove these tangent edges when working with orthographic drawings, so ensure that the '**removed**' option is selected and then select **OK**. See Figure 14.

The Three standard orthographic views of the model are inserted into the drawing sheet border. These views are added to the Feature Manager Design Tree. There will be one view highlighted at all times, and we can work on the highlighted view only. See Figure 15.

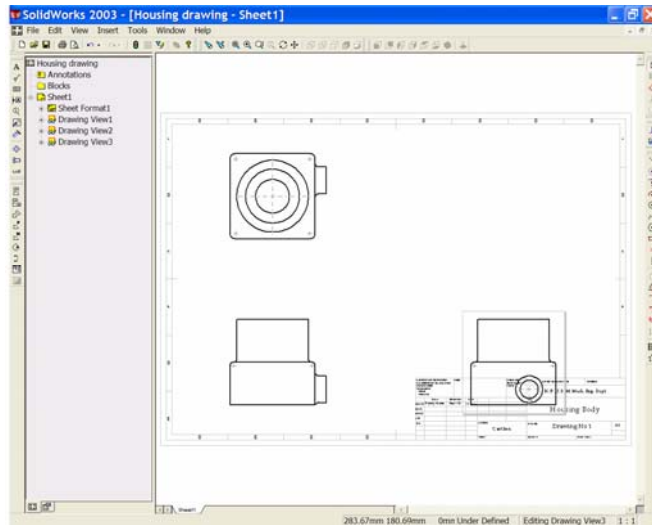


Figure 15

To highlight other views, simply move the cursor close to the view you wish to work with and left click.

17. Place the cursor close to the **PLAN** (the top view).

Notice that the view is framed in a grey box and that the cursor now has a view edit symbol next to it.

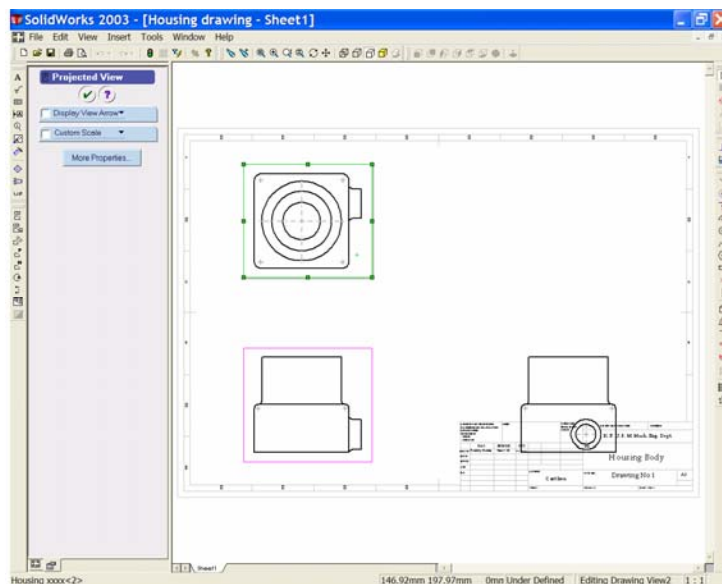


Figure 16

With the cursor inside this grey border left click once. **The Projected View Property Manager** opens and the view is now surrounded by a green window. See Figure 16. Place the cursor on the lower horizontal edge of this window. Note the arrow symbol that appears. **When the arrows are visible press and hold down the left button of the mouse.** We can now move this view 'up' or 'down' on the screen. Move the

PLAN View closer to the **ELEVATION** (front view). Note that while we can move this view vertically, we are unable to move it horizontally! See Figure 17.

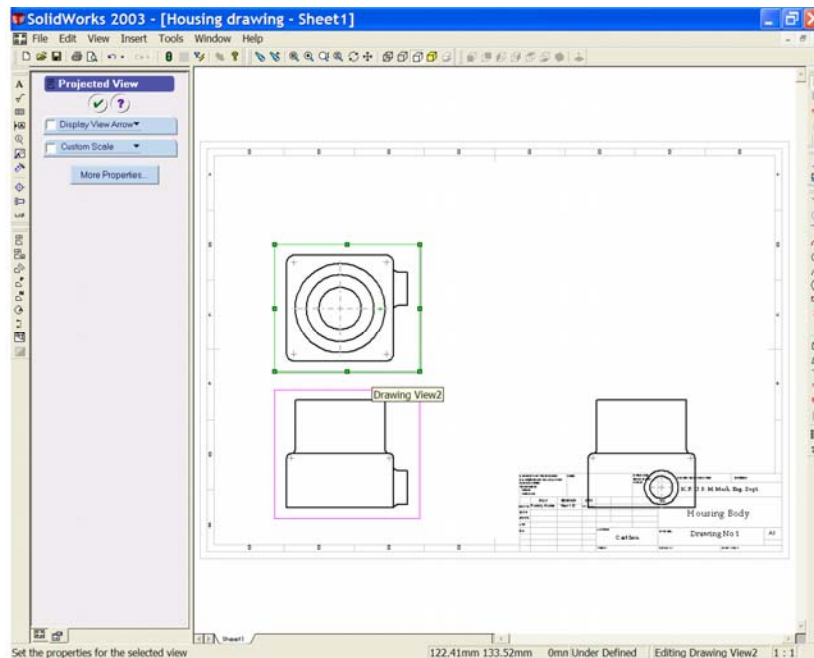


Figure 17

18. Place the cursor close to the **RIGHT END ELEVATION** (right end view) to highlight it. Left click once to activate the window. This time place the cursor on the left vertical edge of the green window. **Press and hold the left mouse button as before and move the view.** Notice that this view can only be moved from **'left to right'**. Place the view at a suitable location to the right of the **ELEVATION**.

19. Now bring the cursor close to the **ELEVATION** (the front view) and activate its window. Place the cursor on any part of the green window, **click and hold the left button once more.** See Figure 18. Now, when we move the **ELEVATION** all three views will move together. Note also that we can move the orthographic group of views to any location on the screen. Re-position the views so that they do not overlap the drawing border or the title block area. Left click anywhere in the drawing area to end the command.

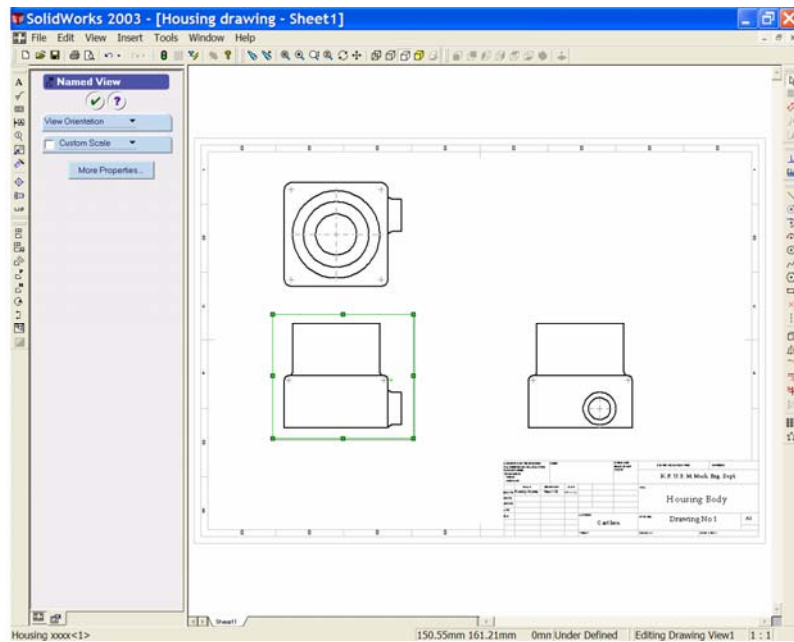


Figure 18

20. We need to show the hidden edges in these views. Place the cursor close to the **PLAN** and left click once. From the view Toolbar choose '**Hidden Lines Visible**'. The hidden edges of the **PLAN VIEW** are now switched on.
21. Place the cursor close to the **ELEVATION** to activate it. Hold down the control button and select the **RIGHT END ELEVATION**. Now switch on the hidden edges for these two views as before.

SAVE YOUR WORK!

Notice that where we can see circles (in the PLAN and the RIGHT END ELEVATION) the centre lines have been added automatically. However we MUST SHOW CENTRE LINES for all cylindrical features of the orthographic views.

22. To add centre lines to the cylindrical features of the drawing, first select '**Centrelines**' from the '**Annotations Toolbar**'. See Figure 19.

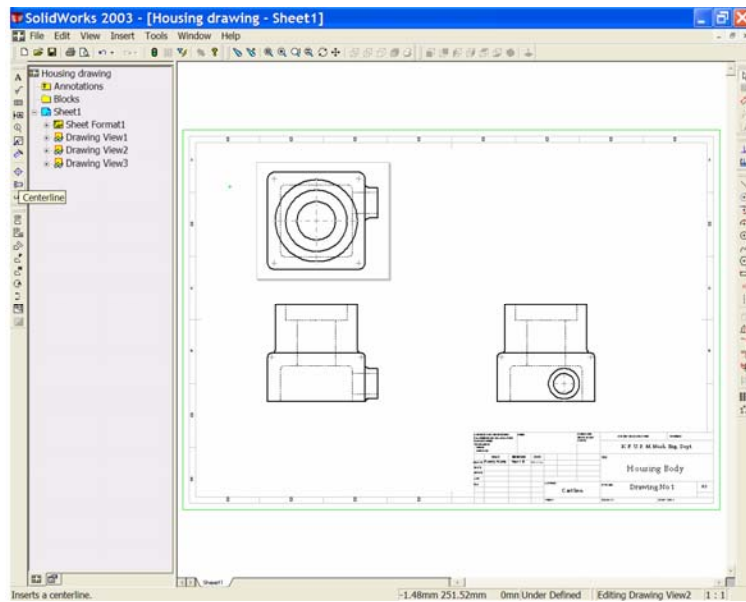


Figure 19

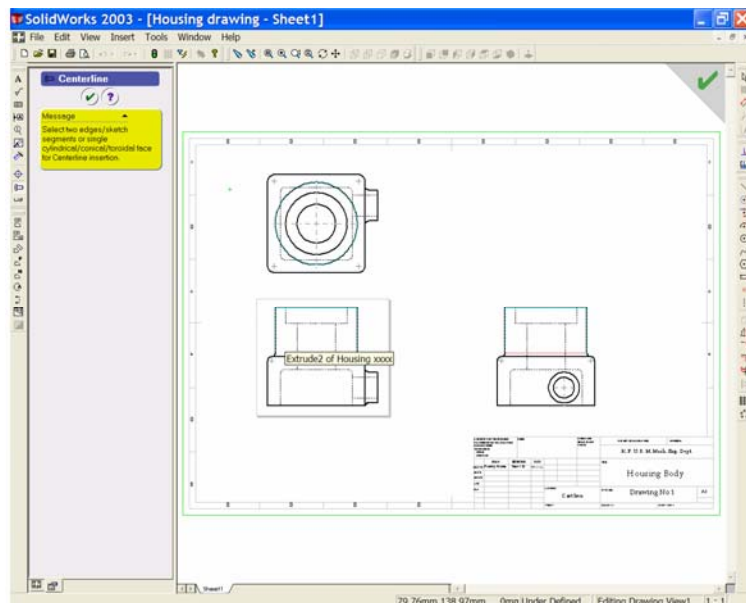


Figure 20

*Notice the message on the **Centreline Property Manager**. This message indicates how we may add centrelines to the drawing views.*

Place the cursor on the surface of the large boss in the **ELEVATION**. See Figure 20.

Notice the surface flag as the cylindrical surface is identified.

Left click once to add the centreline. This centreline should extend to the bottom of the hidden hole. To lengthen the centreline place the cursor on the centreline, note the colour change and the centreline symbol. Left click to select the centreline. The centreline is now active, is green and has two 'handles' one at each end.

Select the lower handle by pressing and holding the left button of the mouse. Still holding the left button, drag the centreline down to the bottom of the hole and release. See Figure 21.

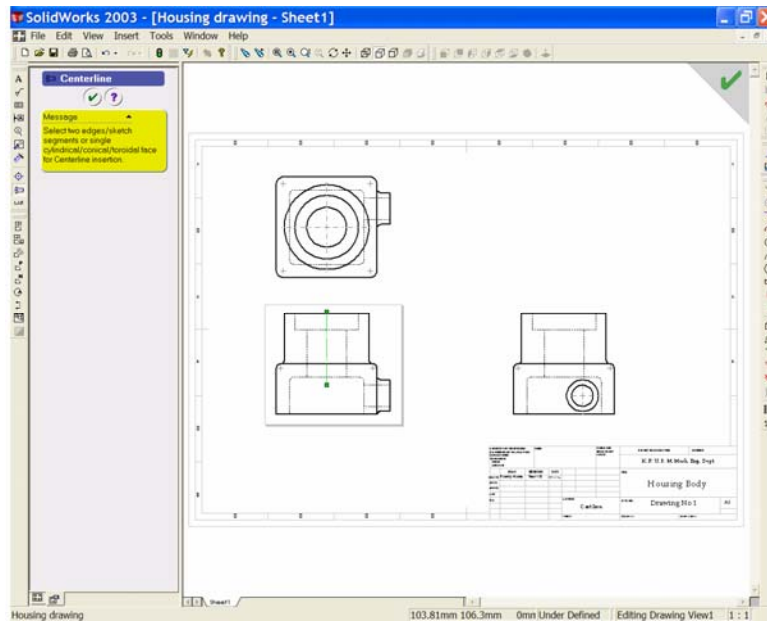


Figure 21

The centrelines of the circles (representing the bosses and bores) in the **PLAN** and **END ELEVATION** may be dragged in the same way.

23. We can place centrelines by identifying the '**edges**' of cylindrical features. Re-start the centreline command. This time select the two edges of the small bore in the **ELEVATION**.

*Note the symbol indicating that you have placed the cursor on the correct '**edge**'.*

Select the edge, it will turn green; (See Figure 22) continue to select the opposite edge of the bore. Once again the symbol indicates that the cursor is on the bore '**edge**'. Select this second edge to complete the command. The centreline is placed.

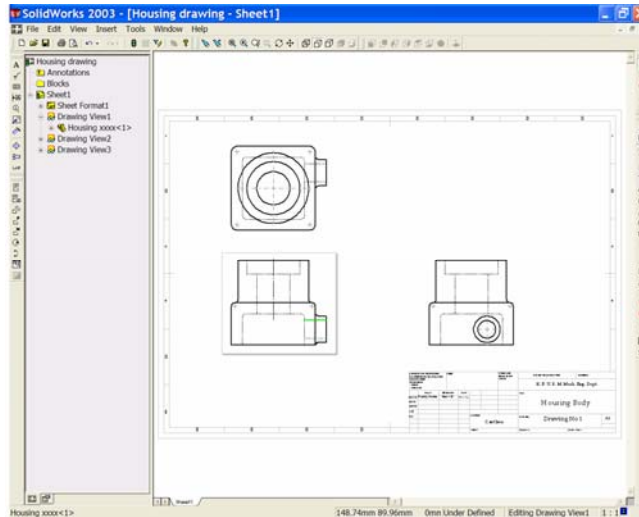




Figure 22

Repeat the above steps for the small bore in the **PLAN**. When you are finished, choose the green tick  to close the Centreline Property Manager.

24. Some section views would be useful for the drawing of this part. We will use the '**Broken-out**' section in two places.

From the '**Sketch Toolbar**' choose '**Circle**' . The pencil/circle symbol appears on screen and we will now draw a circle to establish the area for the broken-out section. Place the cursor pencil/circle symbol as indicated in Figure 23 and draw a circle as shown below.

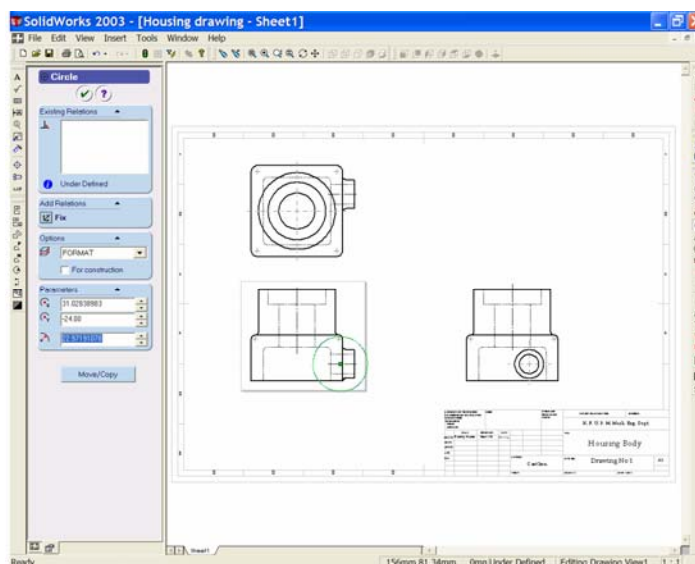


Figure 23

25. From the '**Drawing Toolbar**' select '**Broken-out Section**'. See Figure 24.

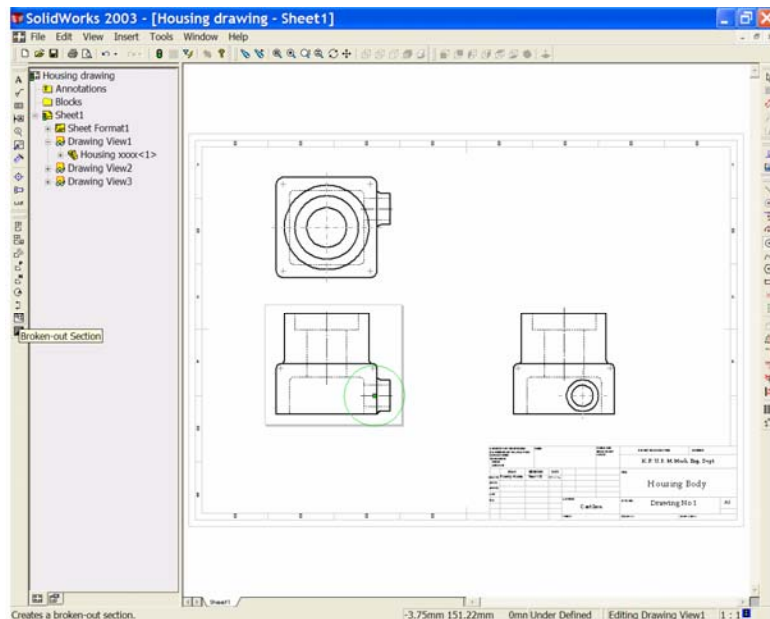


Figure 24

The Broken-out Section Property Manager opens.

Check the 'Preview' box. If a message box opens on your screen choose OK.

Notice that a yellow cutting plane is shown on the PLAN and RIGHT END ELEVATION. See Figure 25.

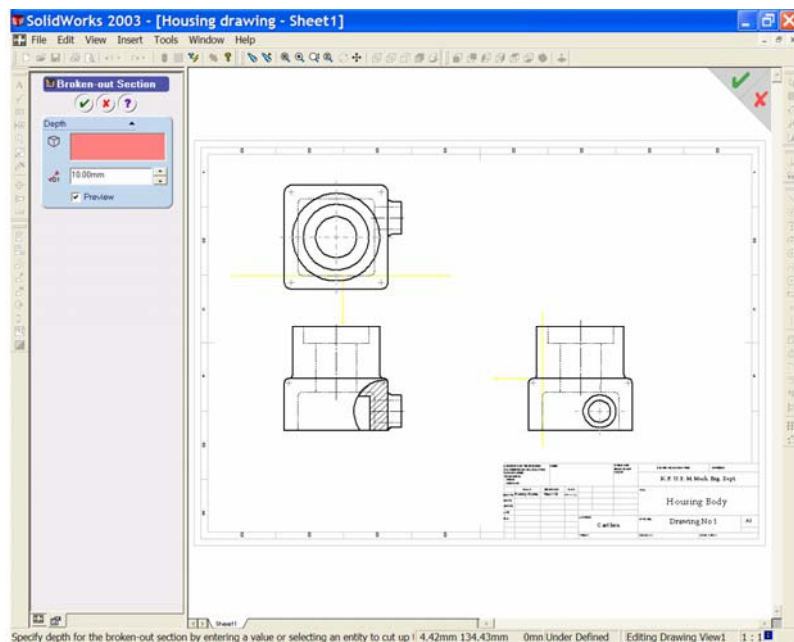


Figure 25

The Property Manager indicates a default depth of 10mm. Press the up arrow to increase the depth to 20mm.

Notice that the yellow cutting plan moves by 10mm and that the broken-out section now appears inside the circular area.

Press the up arrow and increase the depth to 50mm, watch the cutting plane move and the effect it has on the broken-out section. The cutting plane is now close to the centre of the small boss & bore. See Figure 26.

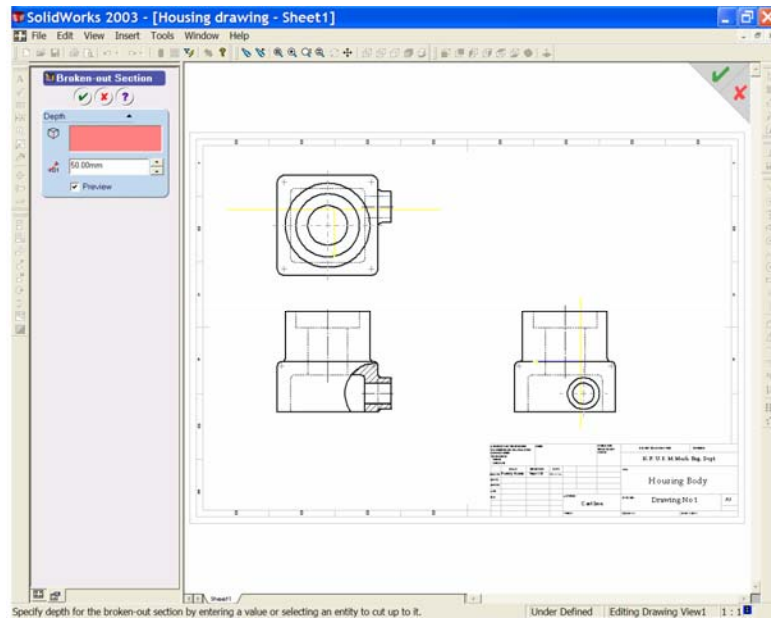


Figure 26

We want the cutting plane to pass through the boss & bore, exactly on its centre. To achieve this we may select the circle that represents the diameter of the boss. Place the cursor on the boss circle.

Notice that it is highlighted; there is an edge symbol next to the cursor and a note indicating the feature (Extrude 3 of Housing). See figure 27.

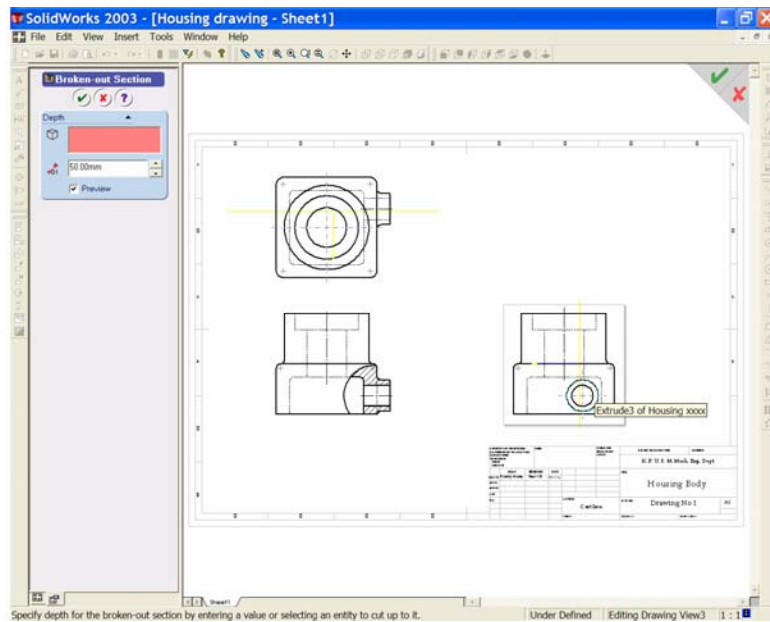


Figure 27

Left click to accept the boss feature. The yellow cutting plane is now exactly on the boss/bore centre, the broken-out section is complete and Edge<1> is listed in the depth window of the Broken-out Section Property Manager. See Figure 28.

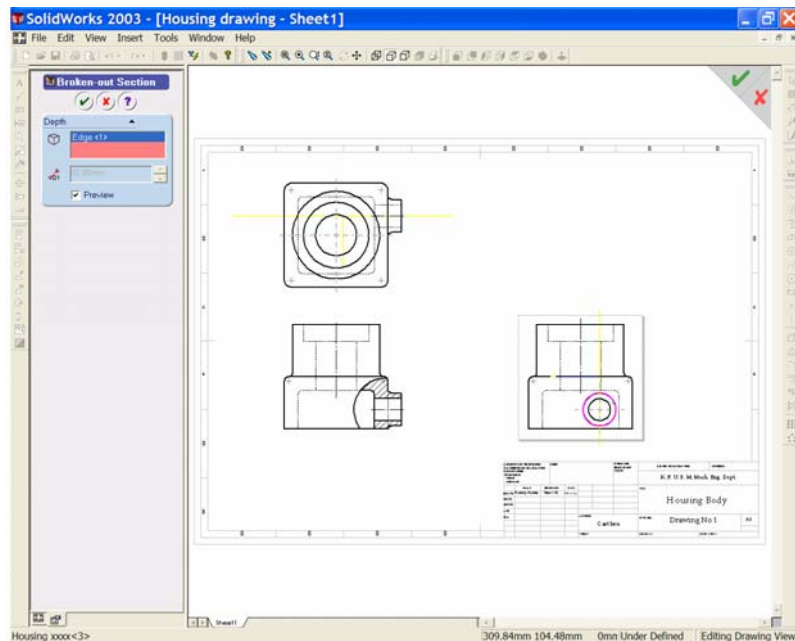


Figure 28

26. Select the green tick  to end the command.

27. On your own, use the same procedure to create a second Broken-out Section for the counterbore hole. See Figure 29.

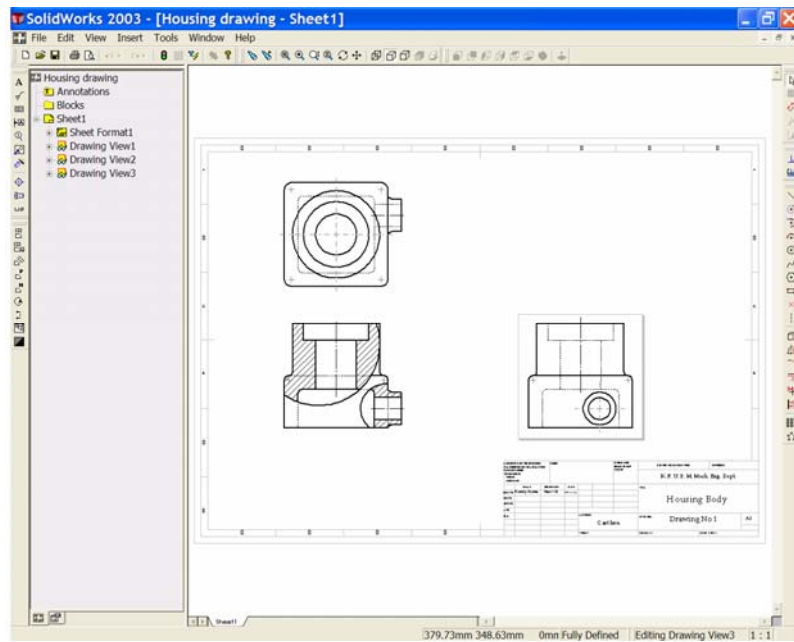


Figure 29

Notice that the broken-out sections are added to the design tree.

28. In the section view, normally hidden edges are not shown. Select the **ELEVATION** and switch off the hidden edges. See Figure 30.

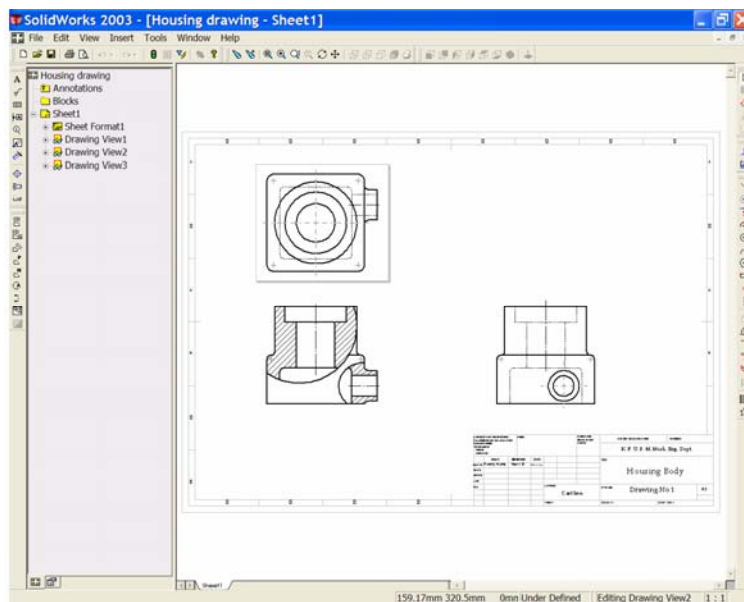


Figure 30

SAVE YOUR WORK!

29. We are now ready to insert the dimensions. Place the cursor over the **'Insert'** dropdown menu at the top of your screen and open. Choose **'Model Items...'** See Figure 31. **'The Insert Model Items'** dialog box opens. Ensure that the **'Dimensions'**, **'Import items into all views'**

and 'Eliminate duplicate model dimensions' boxes are checked. Choose **OK** to insert the dimensions. See Figure 32.

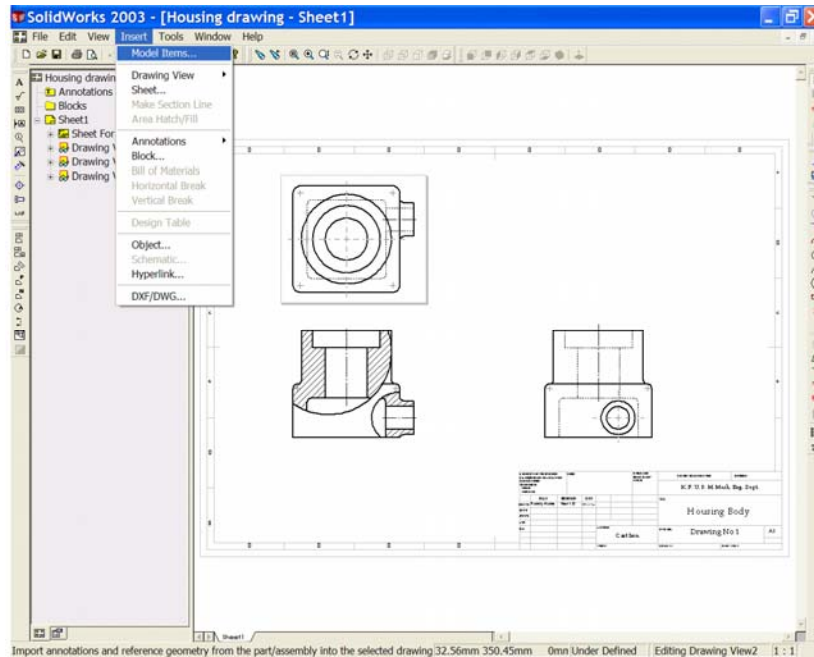


Figure 31

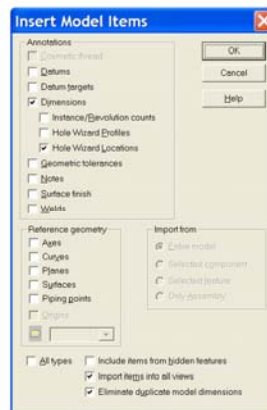


Figure 32

Note that the dimensions are not arranged well. The software simply dumps the dimensions from the model into the drawing file. See Figure 33. We must now re-arrange the dimensions to conform to International Standards.

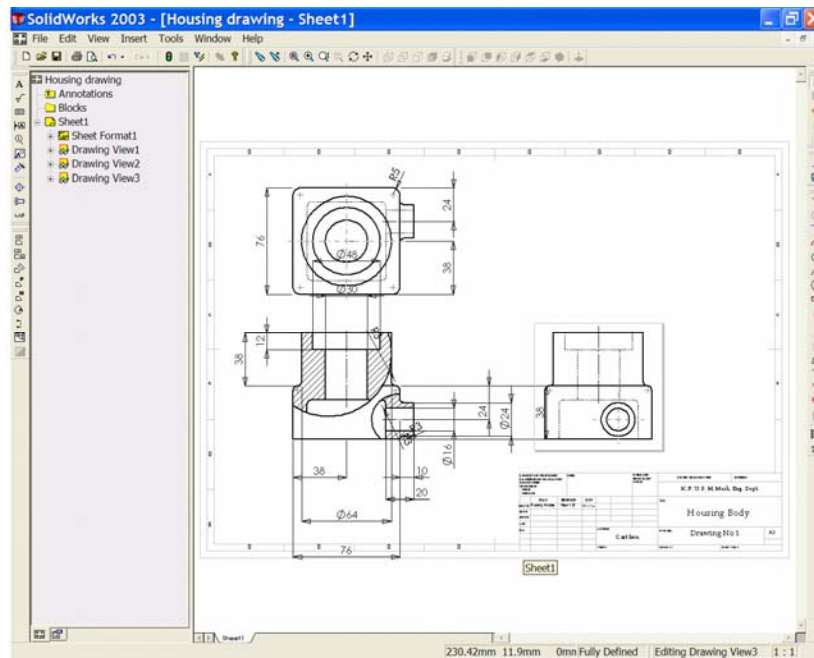


Figure 33

30. First, we should hide dimensions that may be given in a note. The most suitable note dimension is the R3 fillet. **Place the cursor over one of the R3 fillet dimensions and right click.** See Figure 34.

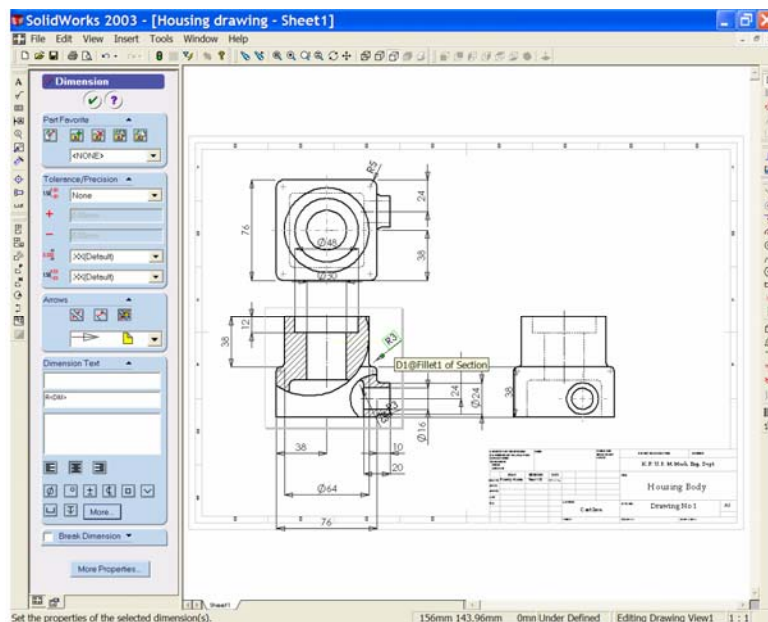


Figure 34

From the short-cut menu choose 'Hide', the dimension is hidden from view. Continue to hide all other R3 fillet dimensions. See Figure 35.

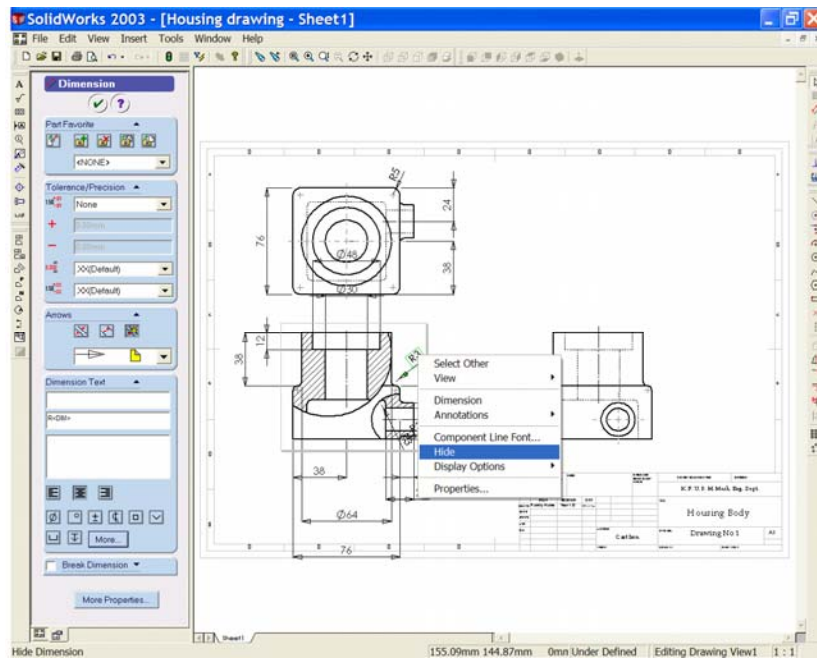


Figure 35

31. The location dimensions of the small boss/bore are best shown in the **RIGHT END ELEVATION**, therefore the dimensions should be moved from their present locations to this view. To do this hold down the control key as you select the dimension with the left button of the mouse, then drag the dimension to the **END ELEVATION**. Drop the dimension at the location shown in Figure 36.

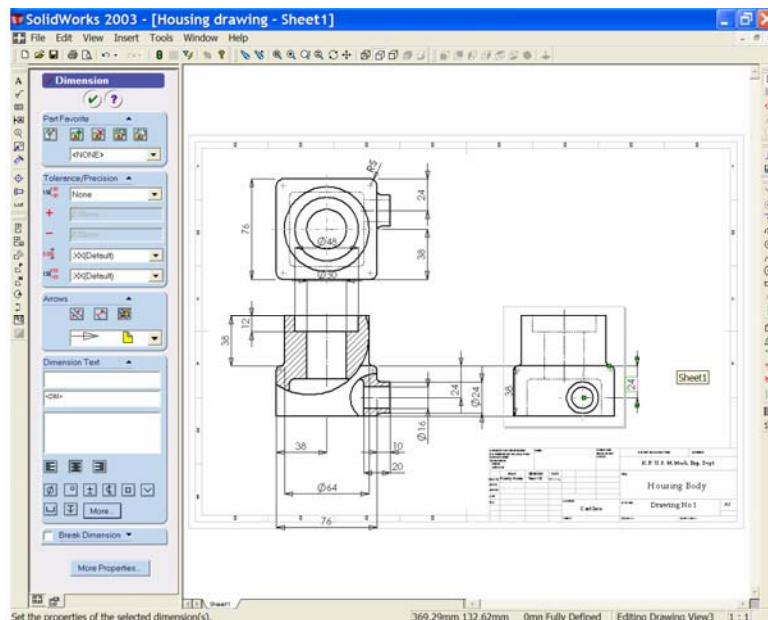


Figure 36

Notice the symbol change as the cursor is moved from the original view to the new location.

32. Other dimensions may be moved to better locations within a view by selecting them with the mouse, left clicking and hold as the dimension is dragged to a new position. When finished, your drawing should look similar to Figure 37.

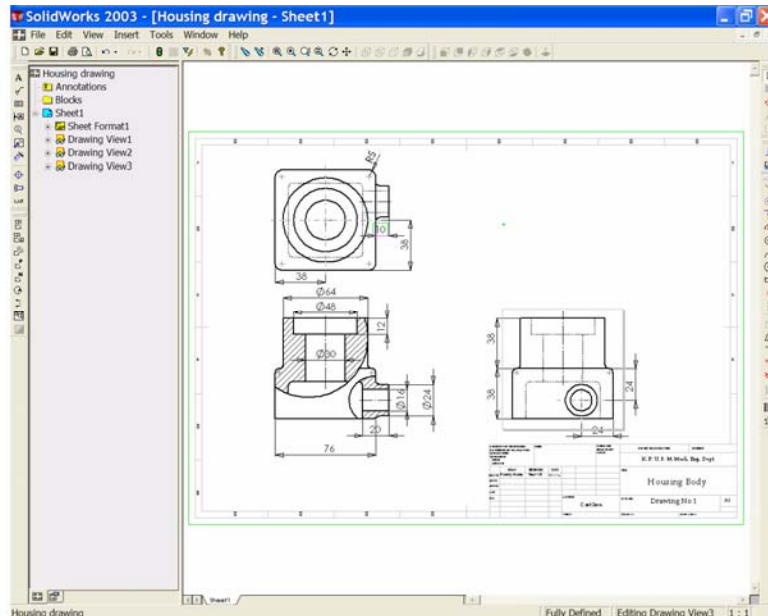


Figure 37

33. When modifying dimensions in the drawing it will be necessary to save both the drawing and its referenced model. When this must be done a dialog window will open asking if you wish to save the referenced models. When this happens it is best to answer 'yes'. Both the active drawing and its referenced model will be saved.

SAVE YOUR WORK!

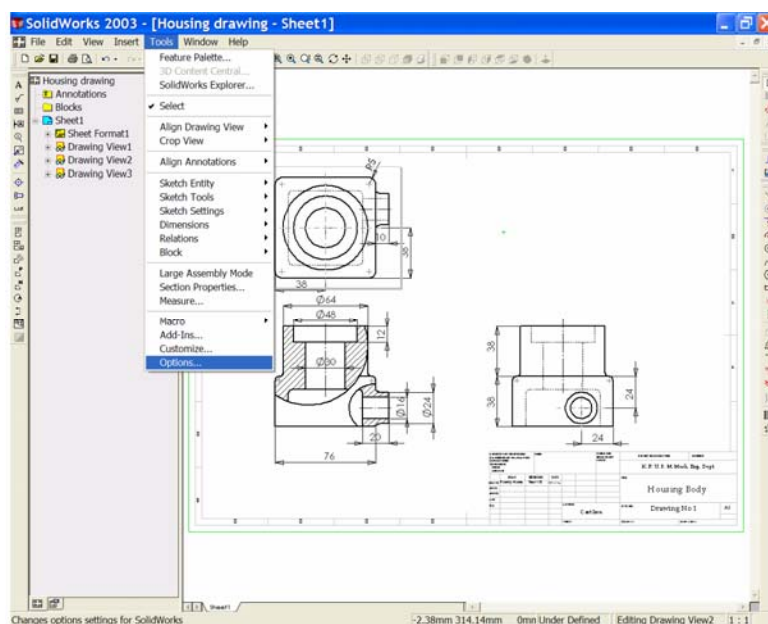


Figure 38

34. The dimension font, font size, arrow style and arrow size may be changed globally. From the **'Tools'** drop down menu choose **'Options...'**, See Figure 38. The System Options dialog box opens, usually at **'General'**. Select the **'Document Properties'** tab. Under **'Detailing'** select **'Dimensions'**. See Figure 39. In the **'Arrows'** area, choose the down arrow to view and select an arrow style.

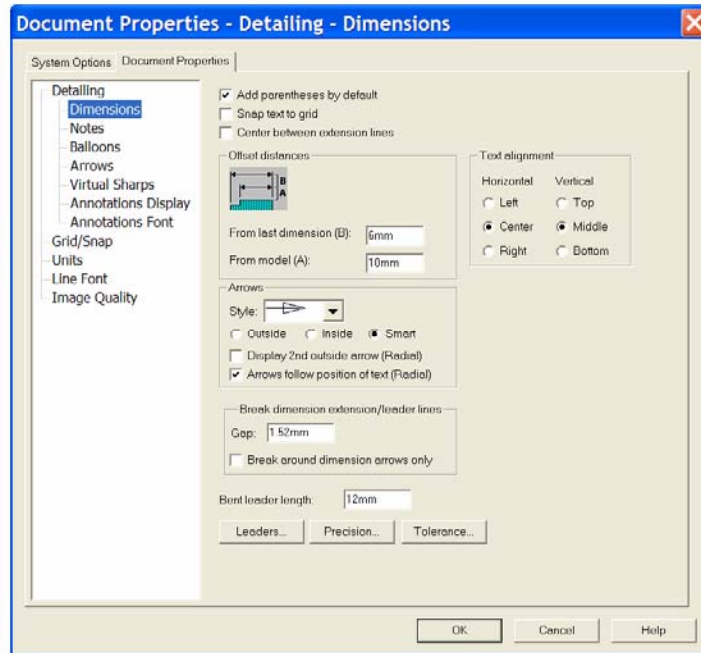


Figure 39

35. Choose **'Annotations Font'** under **'Detailing'** and from the annotations type window select **'Dimension'**. From the dialog box select **'Book Antigua'** and set the **Height to 12 points**. Select **OK** to close the box. See Figure 40.

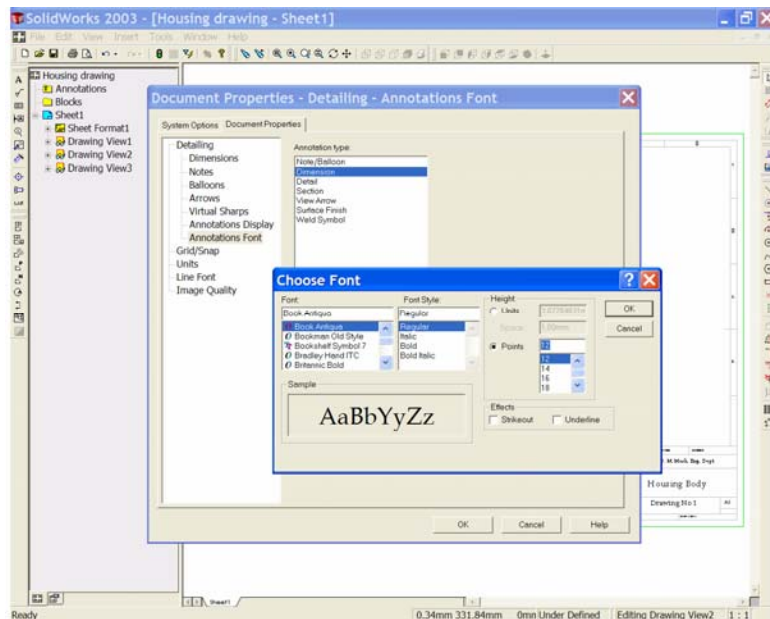


Figure 40

The drawing dimensions and arrows should now have the size and font selected. See Figure 41.

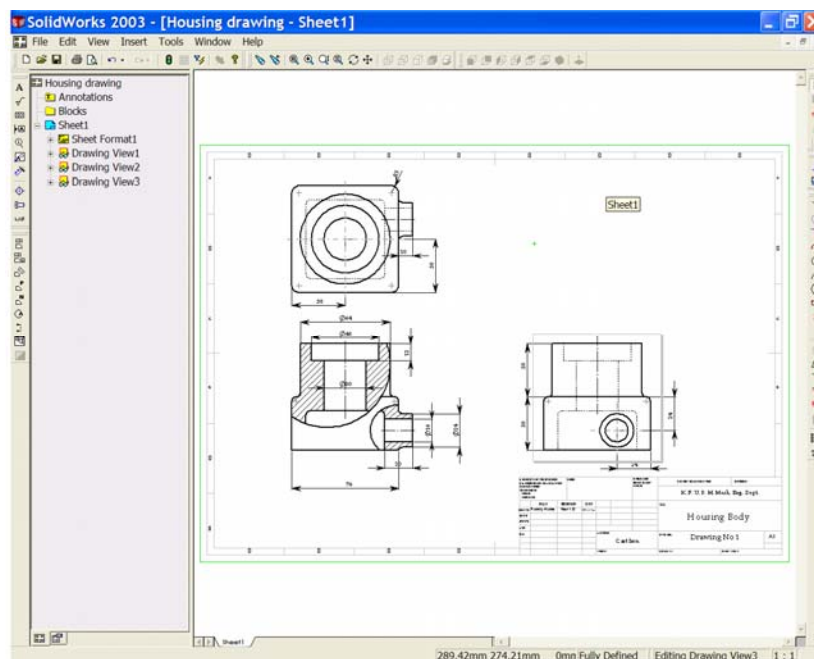


Figure 41

36. We should now add the note to give the dimension for the rounds & fillets that we have hidden. Choose 'Note' from the 'Annotations Toolbar' as before. Place the note box in a suitable location (See Figures 42 and 43.) and type **'Rounds & fillets to be R3 Unless otherwise stated'**

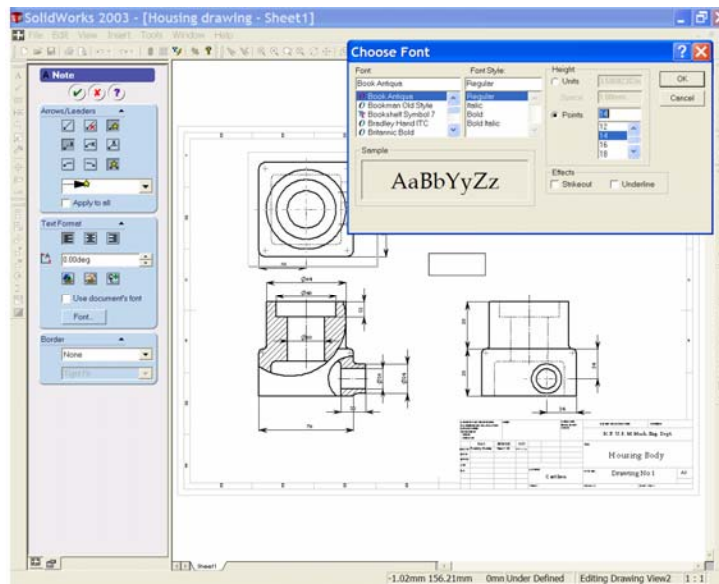


Figure 42

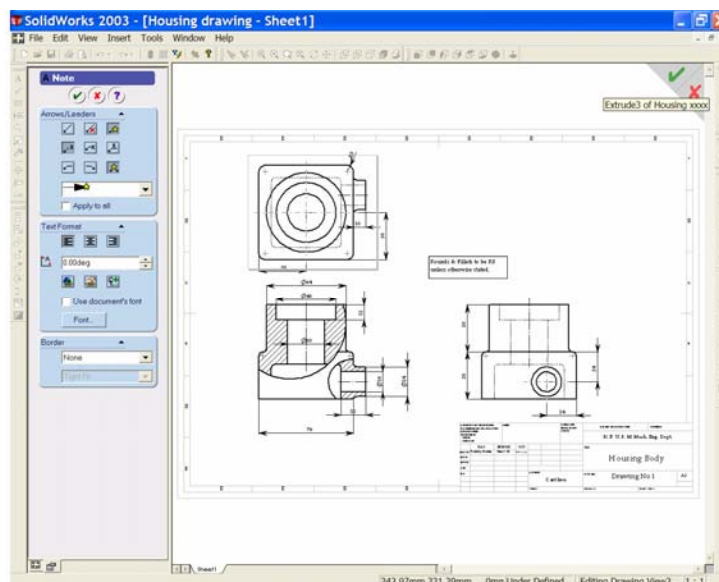


Figure 43

37. To finish we may now add a pictorial view of the model. From the 'Drawing Toolbar' select the 'Named View' button. See Figure 44. The Named View Property Manager opens and indicates four methods of selecting the model. See Figure 45. We will use method 1 for this drawing.

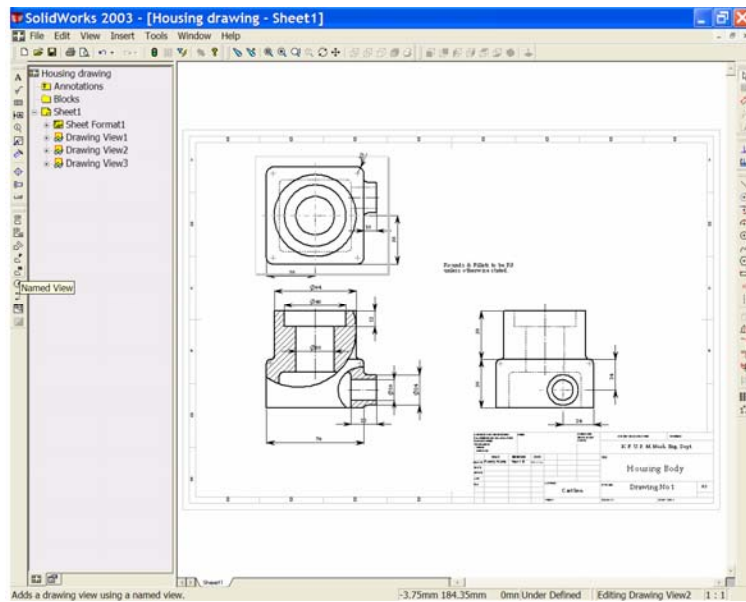


Figure 44

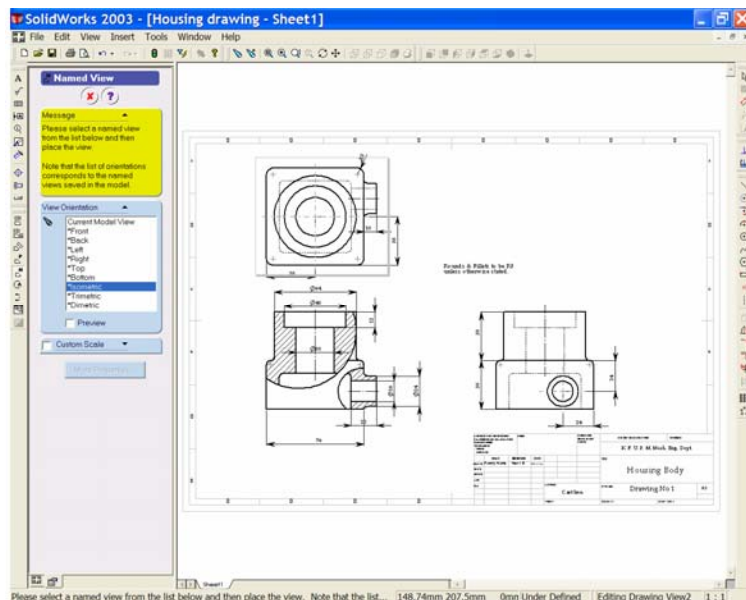



Figure 45

38. Place the cursor close to the **PLAN** view and left click once. A view orientation window opens in the Property Manager and the cursor changes to indicate that a named view may now be selected from the list offered. In the window ensure that '**ISOMETRIC**' is highlighted, you may also check the preview box. Place the isometric view in a suitable location at the top right corner of the border. (You may experiment with some other choices such as Trimetric or Dimetric.)

When you have placed the view choose the green tick  to complete the command. See Figure 45.

Notice that the 'tangent edges' are visible in the isometric view. These edges may be displayed with different types of lines.

39. Right click close to the **ISOMETRIC VIEW** and from the short-cut menu select '**Tangent Edge**'. A further menu slides out to the side and offers three choices; '**Tangent Edges Visible**', '**With font**' or '**Removed**'. Choose '**Tangent Edges With Font**'. Your model will now display the Isometric view tangent edges as phantom lines. See Figure 46.

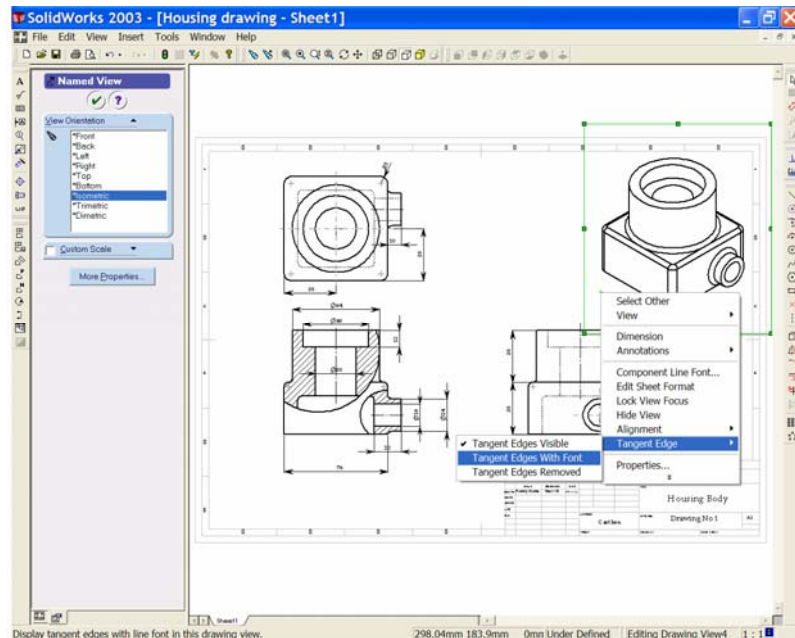



Figure 46

40. Finally, we may add colour to the Isometric view. Left click close to the isometric view and select '**Shaded**'  from the '**View Toolbar**'. You may need to left click anywhere in the graphics area to re-generate the drawing.

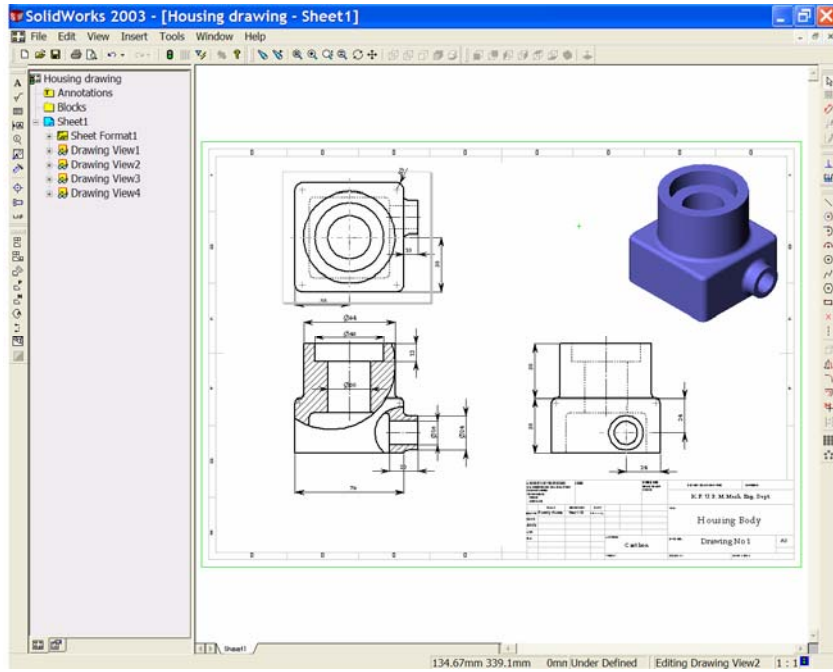


Figure 47 The complete Drawing