Module 4: To Construct Housing Body and Add Multiple Features

1. Start the Program **Desktop Symbol** OR **Start Menu**

2. Select '**New Document**' from the Welcome window OR Select '**New**' button OR Select File-new OR **Ctrl+N**.

3. Select '**Tutorial Tab**' and then '**Part**'

4. Select '**Top**' (plane) from the '**Feature Manager Design tree**'.

5. Select '**Normal to**' from the Standard Views Toolbar.

6. Select '**Sketch**' button OR Insert – Sketch from the dropdown menu bar at the top of your screen.

7. Select the '**Rectangle**' button . Notice that the cursor changes to a pencil symbol with a rectangle.

8. Place the Rectangle pencil at the lower left of the red origin and click the left button of the mouse **ONCE (left click)**.

9. Drag the mouse to the top right of the red origin. (Notice the x-y coordinates changing at the bottom of the screen) Left click once. The size of the rectangle is not important at this stage. The rectangle is now green and the symbol is disconnected from the sketch.

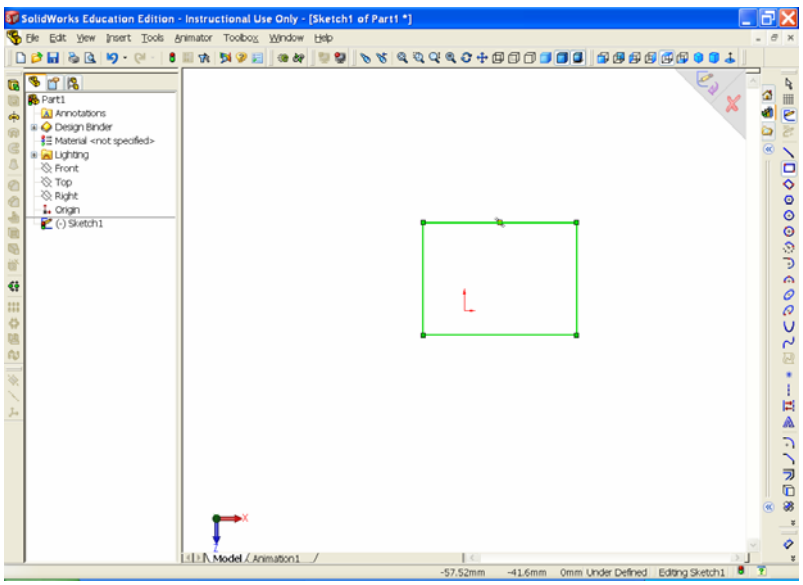

Figure 1

N.B. Look at the bottom right of the screen. As the cursor moves, the x & y coordinates change to indicate the location, 'z' is zero since this is a 2D sketch. The screen also indicates that we are 'Editing Sketch 1' and that the sketch is 'Under Defined'.

10. Choose '**Select**' from the Sketch Toolbar. The sketched rectangle is now blue and the Rectangle Command is complete.

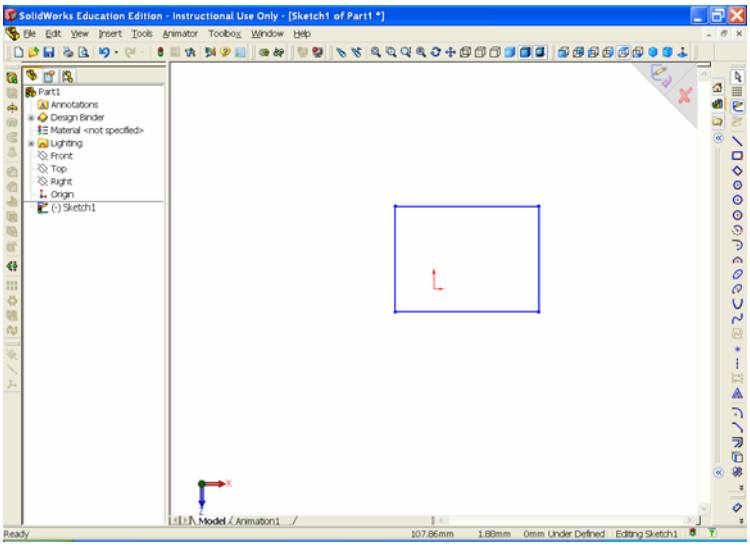

Figure 2

11. Choose '**Smart Dimension**' from the Sketch Toolbar. The cursor now indicates a dimension symbol.
12. Dimension the sketch as shown in the Figure 11.

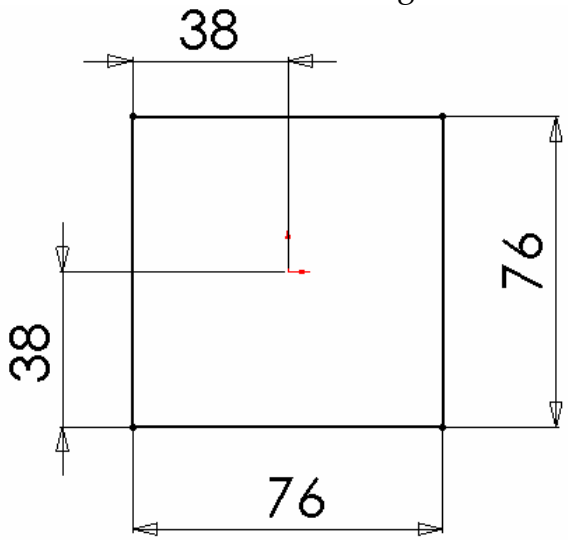

Figure 3

Notice that the sketch is now completely black, that it is 76mm square and that it is centred on the origin. Note also that the box at the bottom right of the screen indicates that the sketch is now fully defined.

13. Choose the select button to end the dimension command.

The sketch can now be transformed into a 3D solid object. Notice the Features Toolbar is now giving access to the Extruded Boss/Base Tool .

14. Choose the '**Extruded Boss/Base**' button .

The Extrude manager opens and the sketch is displayed on screen as an isometric view. A default distance of 10mm is added to the model in the direction of the arrow shown.

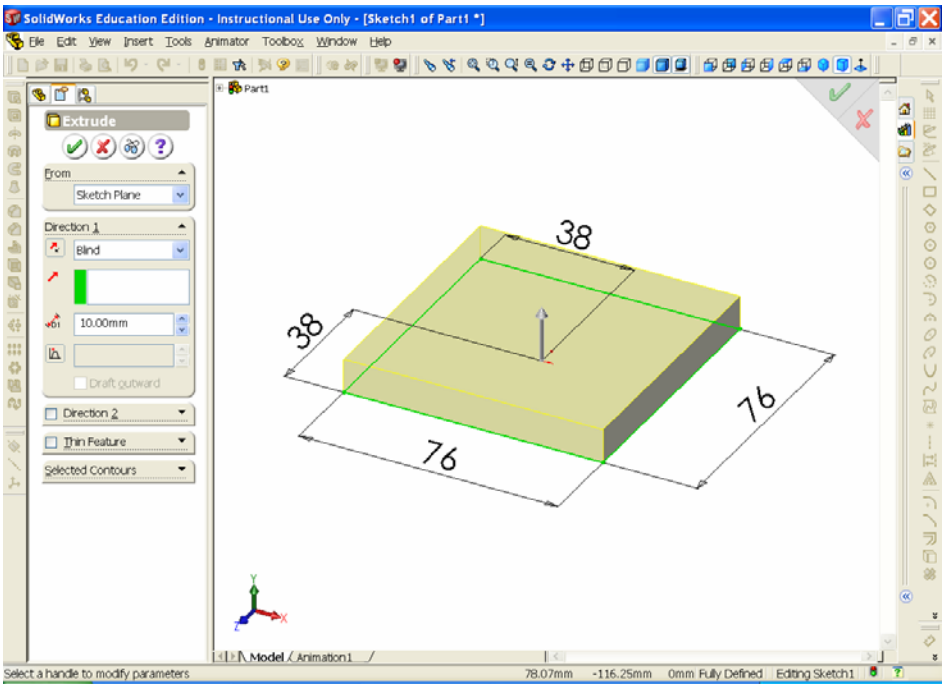

Figure 4

15. Leave the End Condition as '**Blind**'. Use the up arrow at the side of the depth box (d1) to increase the value to 20mm. Notice that the model increases by 10mm in the direction of the arrow on screen. Highlight this 20mm value and use the backspace key on the keyboard to remove.
Replace the 20mm with a new value of 38mm and press the enter key. Once again the model changes to the new value. Select the green tick to accept the model with this distance.

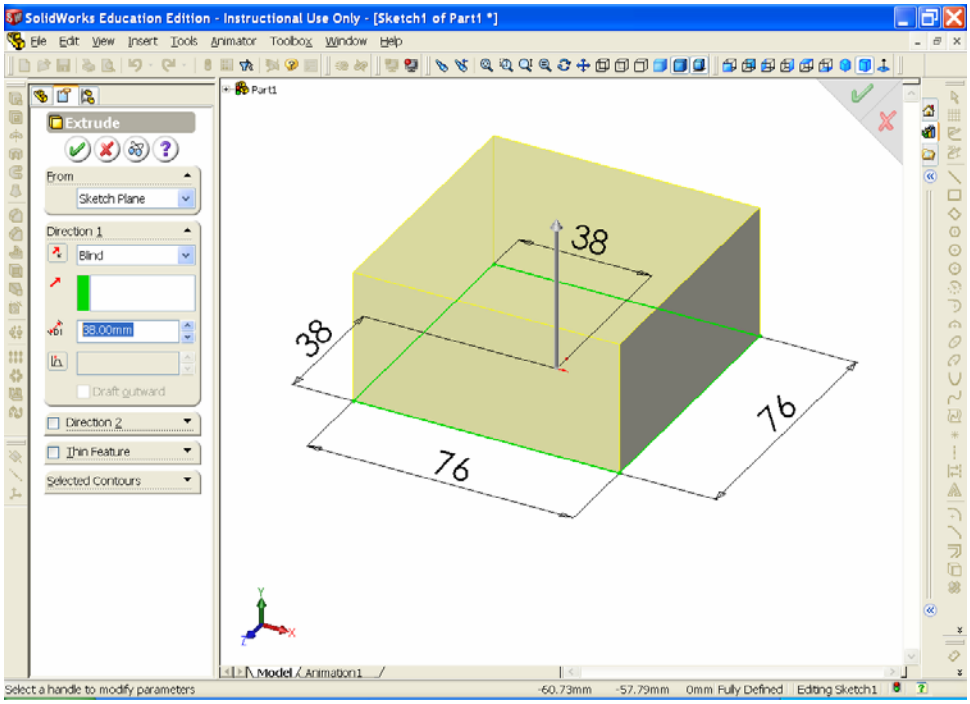

Figure 5

Notice that the Feature Manager Design tree now shows Extrude 1 with a '+' box to its left. Place the cursor on the '+' box and left click to open it. Notice that Sketch 1 is the base of the extruded model.

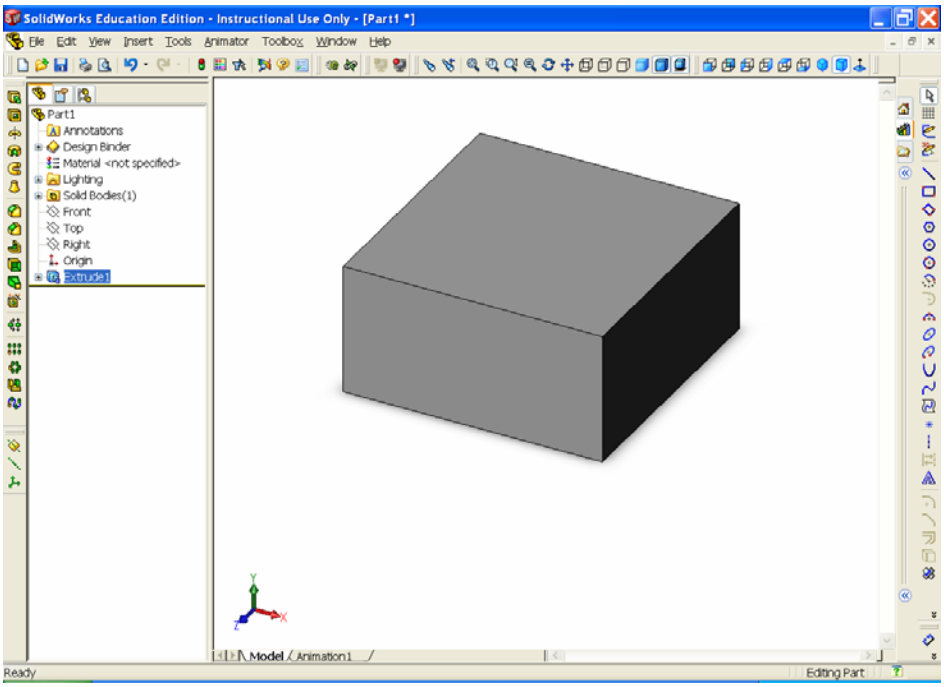

Figure 6

16. Right click on Sketch 1 and choose **Edit sketch**.

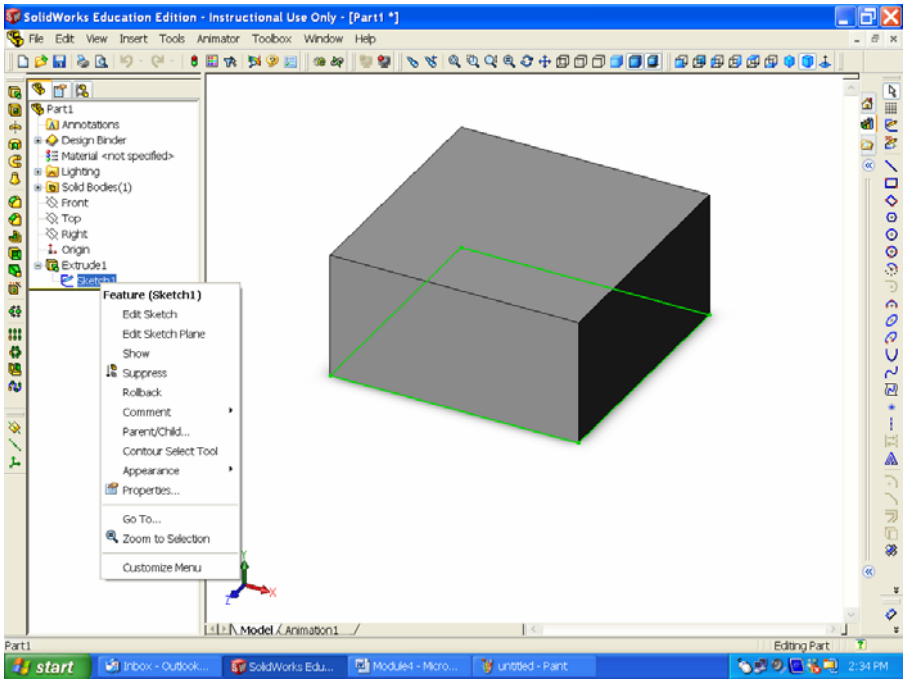

Figure 7

17. Choose '**Normal to**' from the Standard Views Menu.

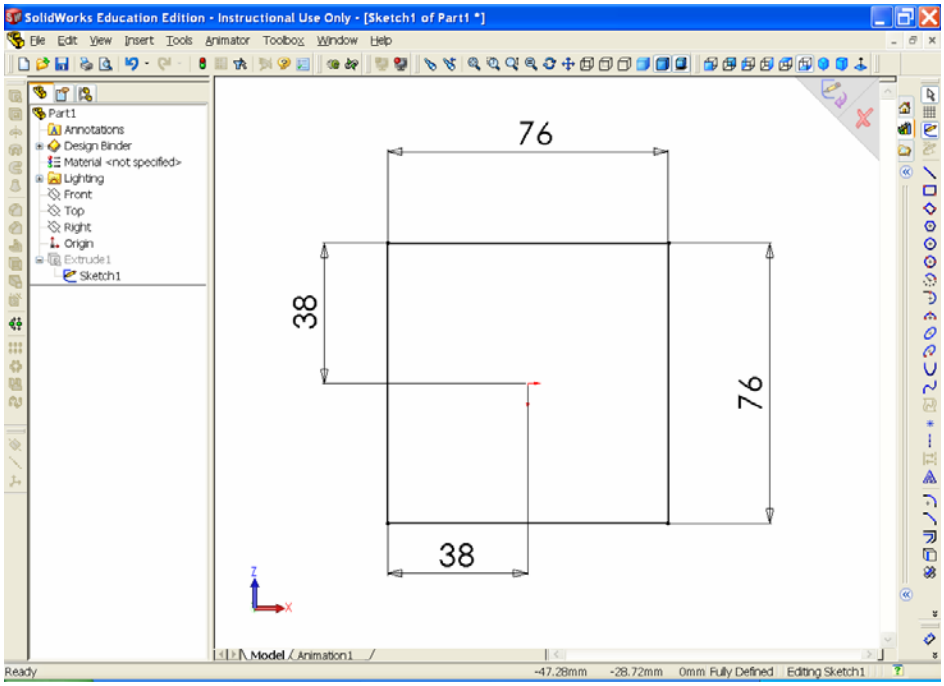

Figure 3

The now familiar returns to the screen and may be edited.

18. Choose '**Sketch fillet**' from the Sketch tools Toolbar.

The sketch fillet property manager opens with a default radius of 10mm.

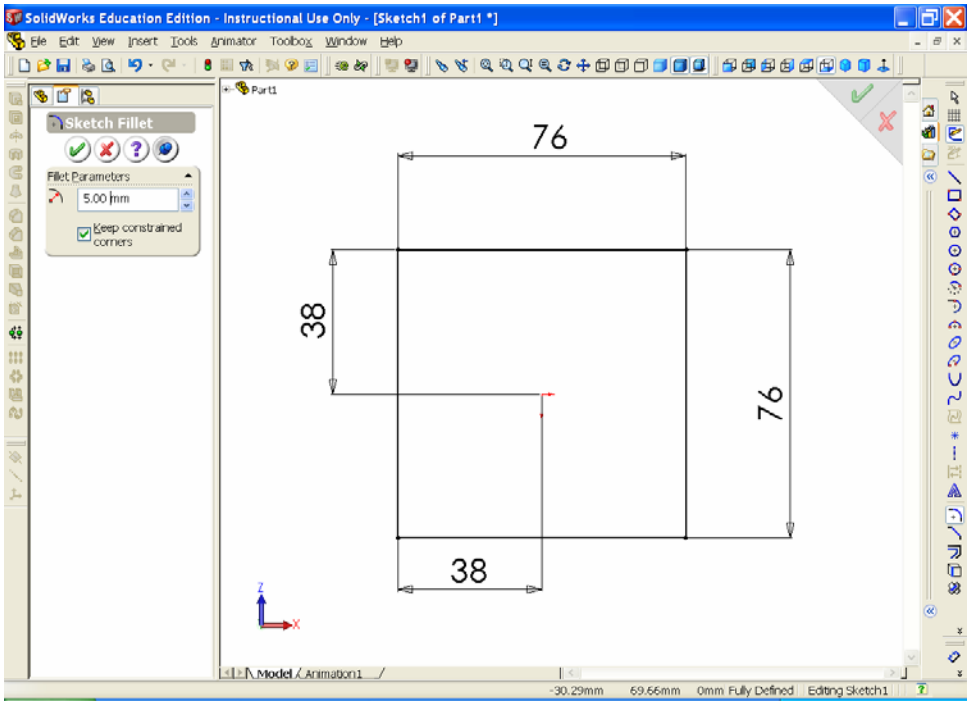

Figure 4

19. Change this radius to 5mm and select each corner of the sketch.

Notice that each corner is highlighted by a red dot as it is identified. Also note that the fillet dimension is added to the sketch on the first fillet only. Ensure that the 'Keep constrained corners' box is checked. Select the green tick (OK).

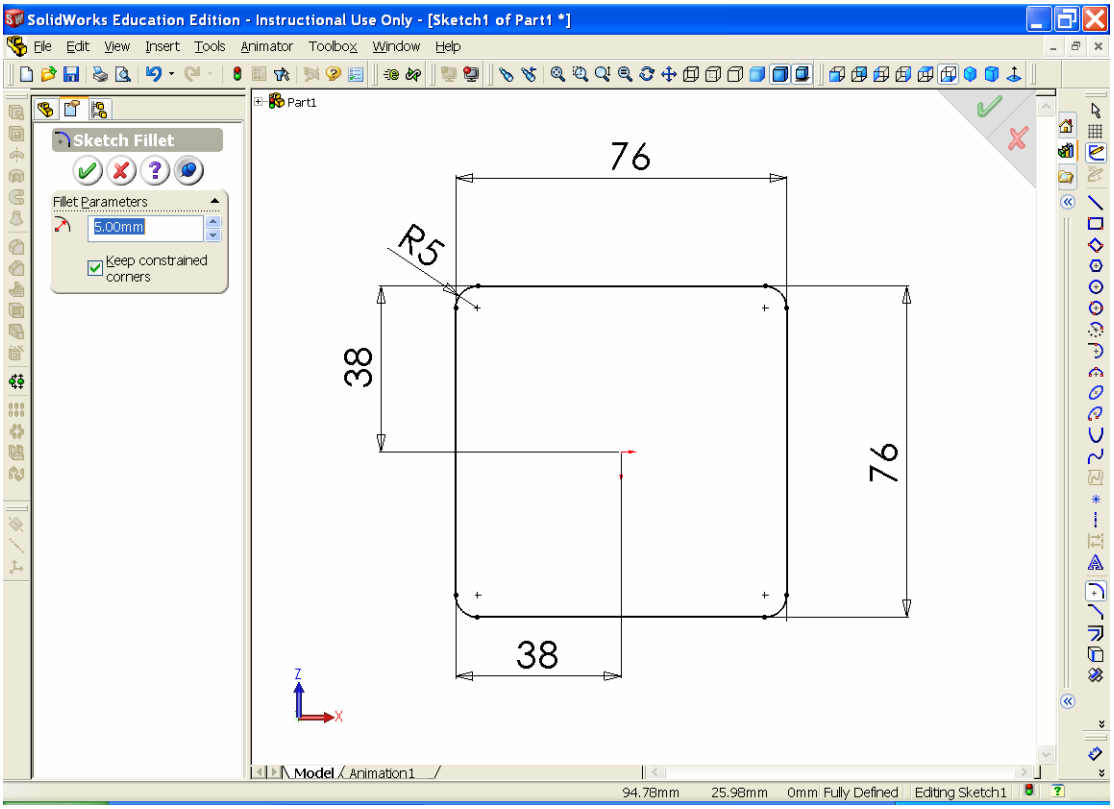

Figure 5

The Feature Manager Design tree now shows a rebuild required symbol to the left of Extrude 1.

20. Left click the '**Rebuild**' button and select '**Isometric**' from the Standard Views menu.

Figure 6

We now need to make the base of the model hollow. To view the base we must rotate the view to see the flat bottom.

21. Choose the '**Rotate view**' button . The cursor now appears like two semi circular arrows following each other. Press the left button of the mouse and hold it down. Still holding the left button down rotate the model so that the bottom surface is visible. Release the mouse button when you are finished rotating the model and press the rotate button once more to end the command.

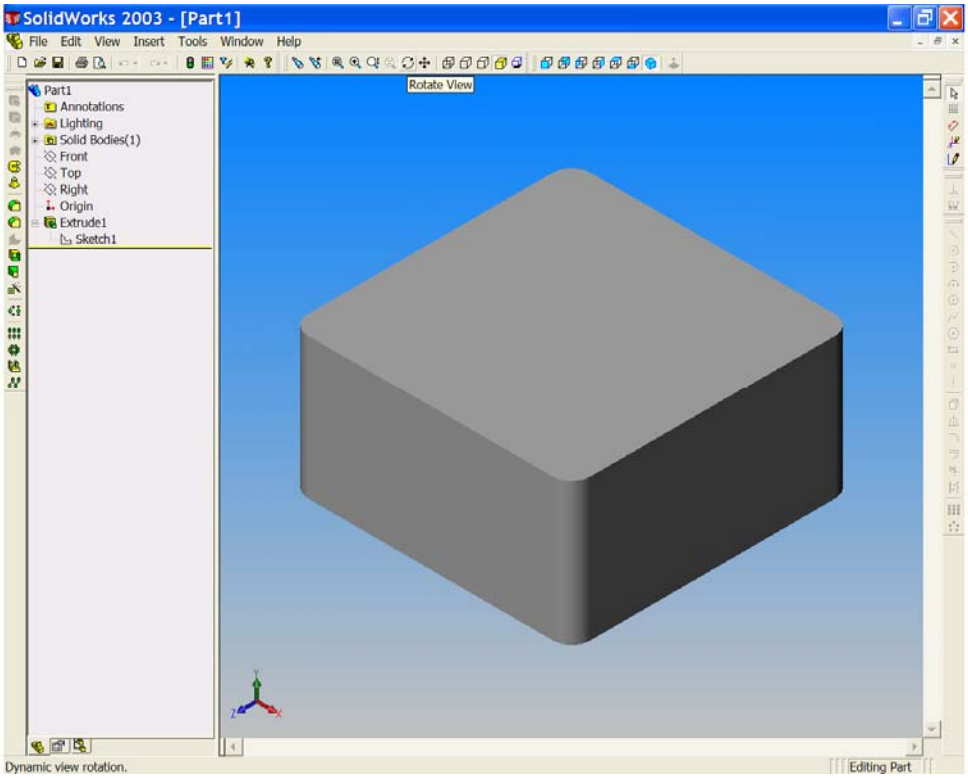

Figure 7

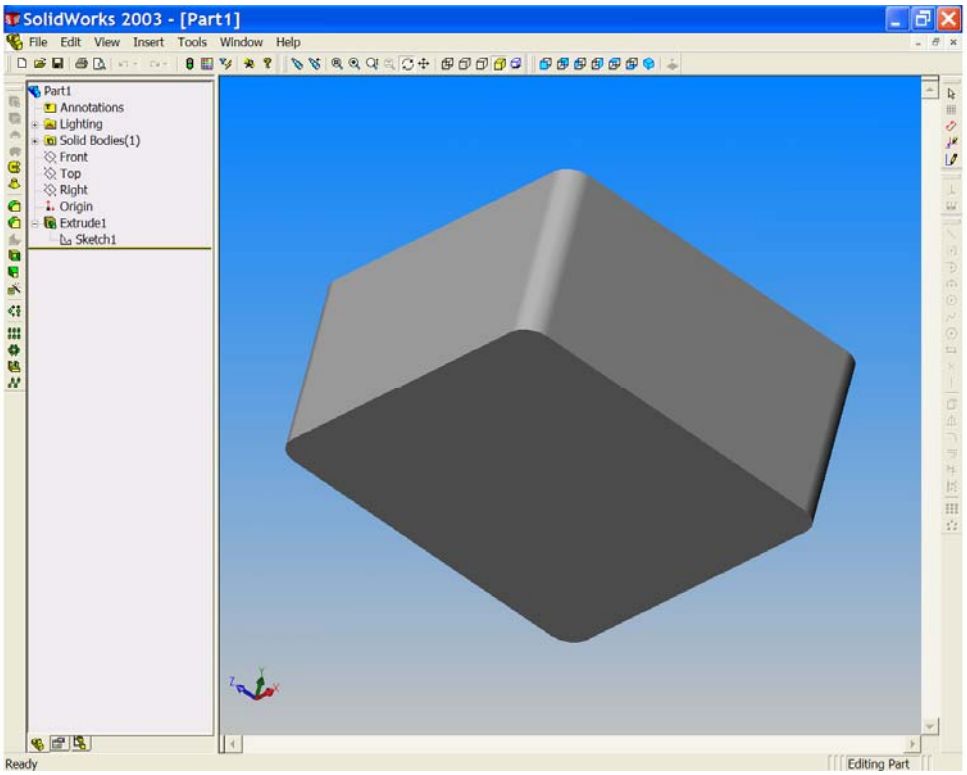

Figure 8

22. Select the bottom face of the model by placing the cursor over the face. Notice the flag appearing as the edges of the face are highlighted in red. Left click to select the face. The selected face is now green (active).

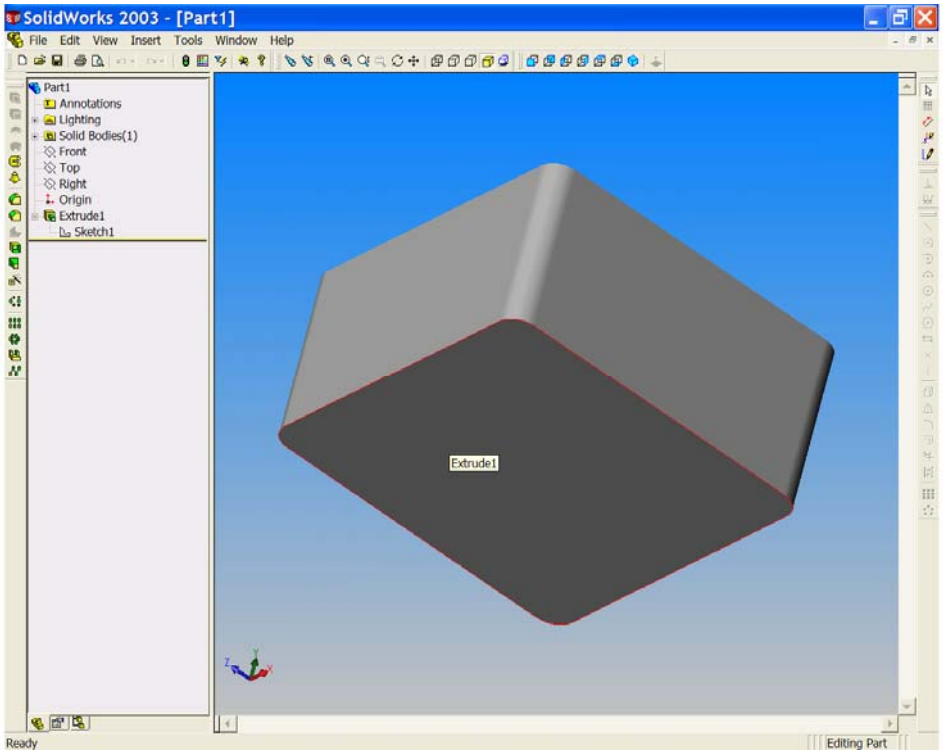

Figure 9

23. Choose '**Shell**' from the Feature menu. The shell property manager opens. Face<1> is already shown in the Faces to remove box and a default value of 10mm is indicated in the thickness window. This value is satisfactory for our model so there is no need to change it. Press the green tick for OK .

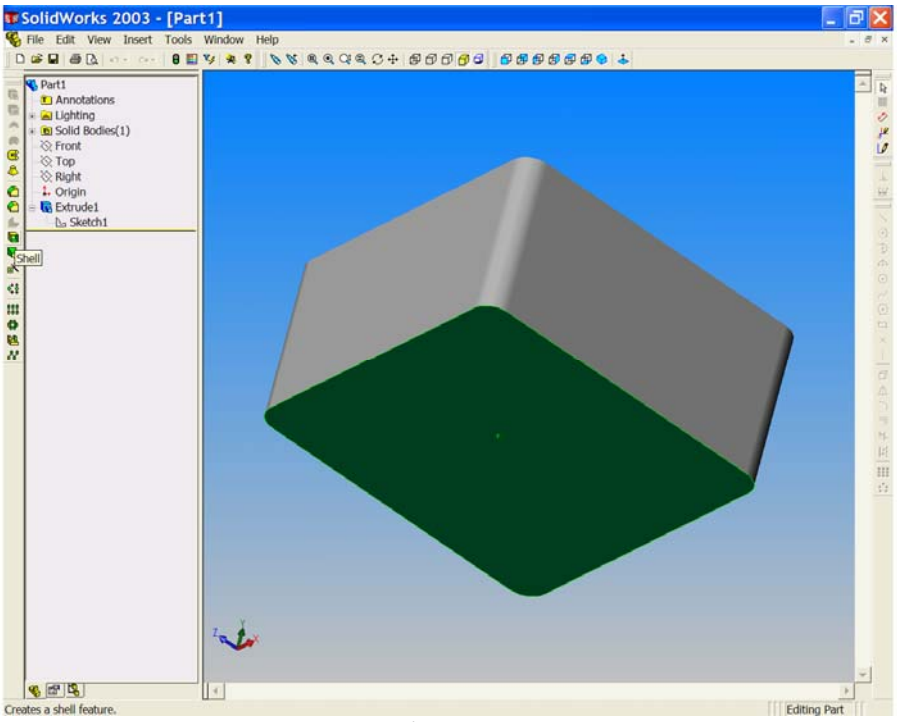

Figure 10

Notice that the base is now shelled or hollowed out with a thickness of 10mm. The Feature Manager Design tree indicates Shell 1.

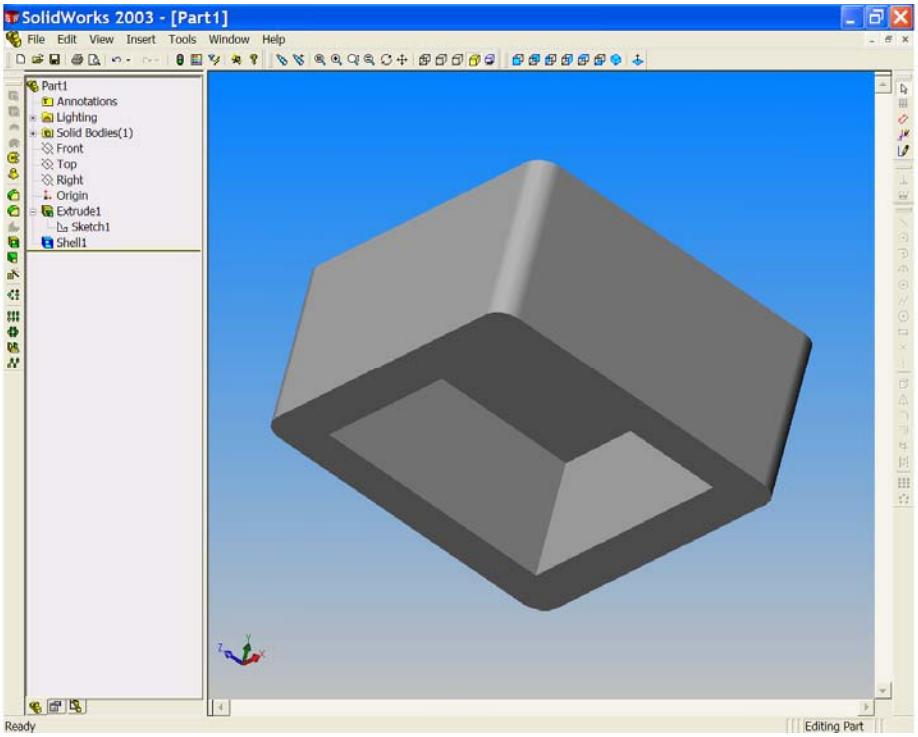

Figure 11

24. From the File menu select '**Save as**'. In the save as dialog box save the model as (give a file name here).

Figure 12

Notice that this new name appears at the top of the screen and at the top of the Feature Manager design tree.

25. Go to **Isometric** view and then select the top surface of the model and then choose '**Normal to**' from the View menu.

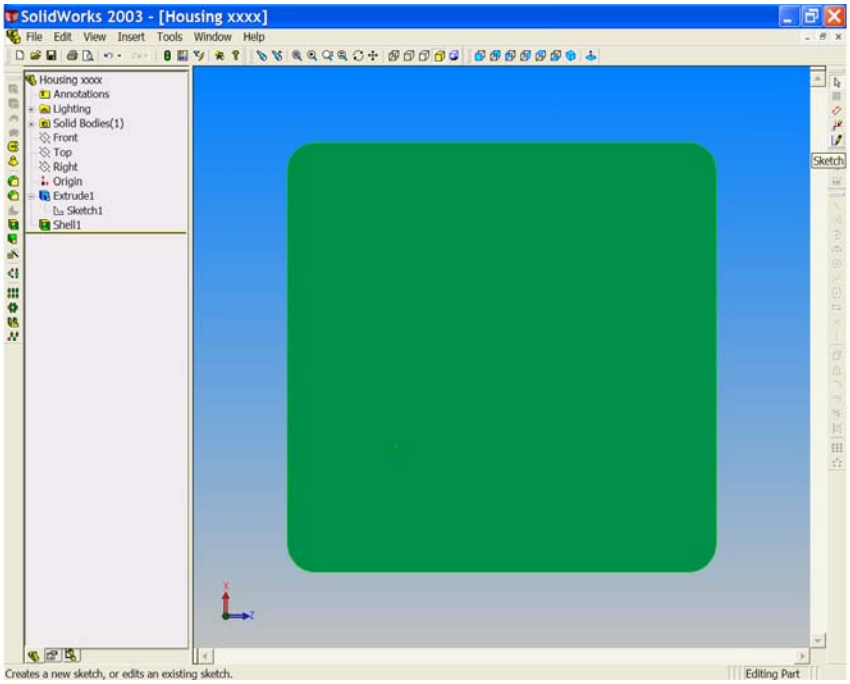

Figure 13

26. Open a new sketch on this surface. Select '**Sketch**'.

27. Choose '**Circle**' from the Sketch tools toolbar.

Notice that the pencil symbol is now accompanied by a circle symbol.

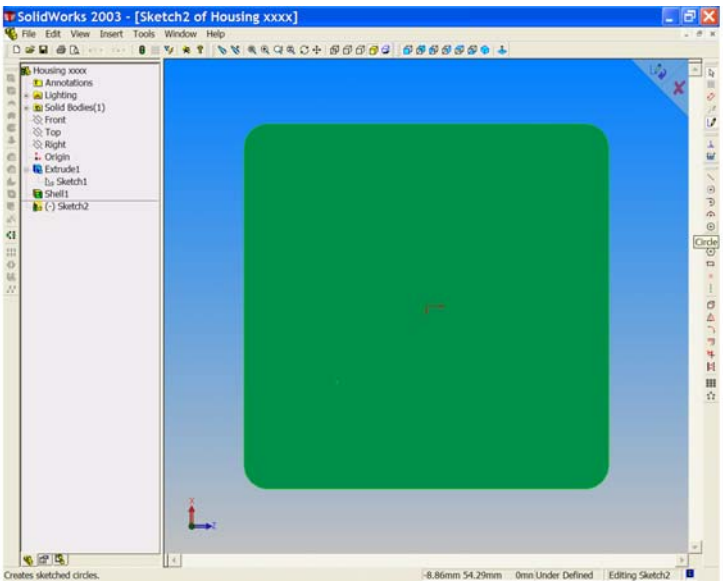

Figure 14

28. Place the pencil symbol on the model origin.

Notice that it changes colour (it becomes yellow and that the circle symbol is replaced by an orange square).

Left click once again to place the circle centre at the model origin. The circle property manager opens. Drag the mouse from the origin to draw the circle.

Notice that the circle centre is fixed at the origin and that the circumference is attached to the mouse.

The circle radius is shown beside the symbol. For this model the circle radius needs to be about 32mm. When the circle radius is about 32mm left click once, to place the circle and end the command.

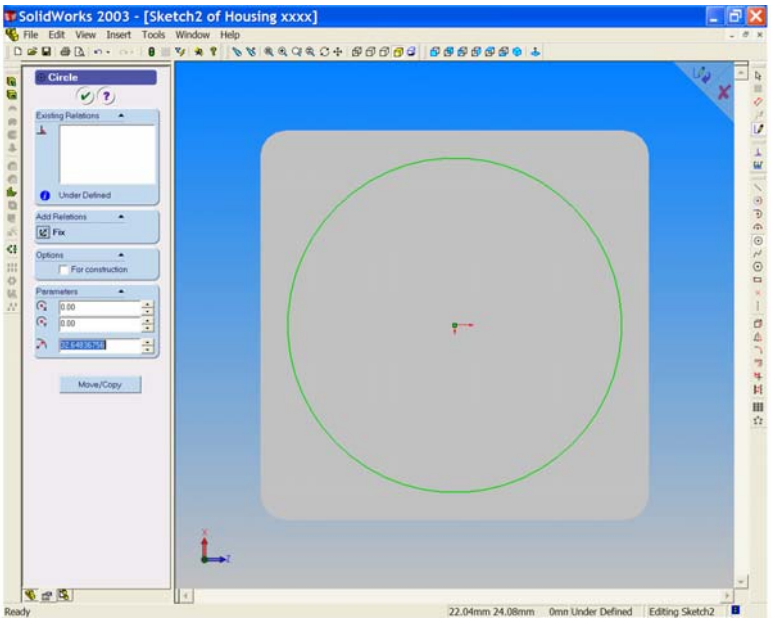

Figure 15

29. Right click once.

By right clicking we can open a shortcut menu to some commands, including the dimension command.

Choose Dimension from he shortcut menu by left clicking on the word '**Dimension**'

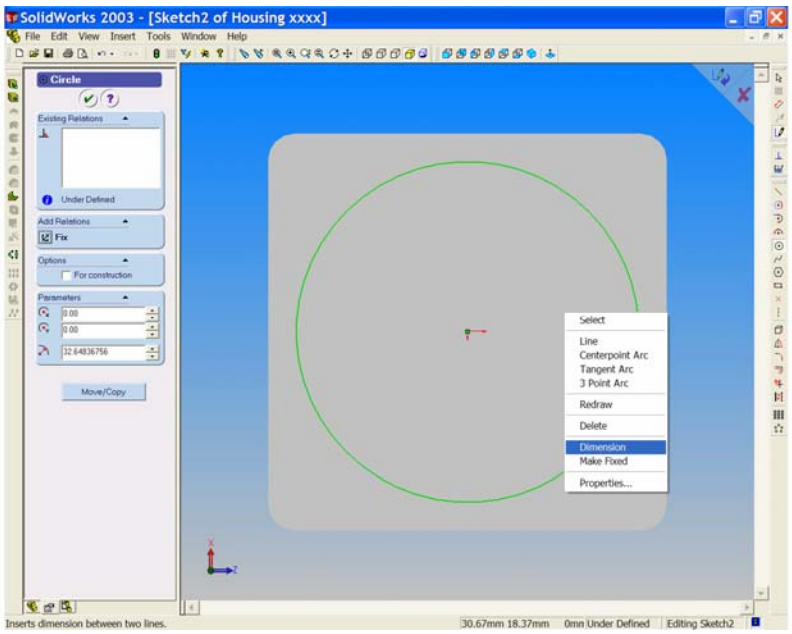

Figure 16

Notice that the circle command is now complete and the circle is blue in colour, fixed to the origin but still under defined. To fully define the circle we must add a dimension.

Select the circle at any point on its circumference and place the dimension at any convenient location. In the modify box delete the value and enter 64mm.

The Ø64 is added to the model and the circle re-sizes to the correct dimension. The circle sketch is now black and fully defined.

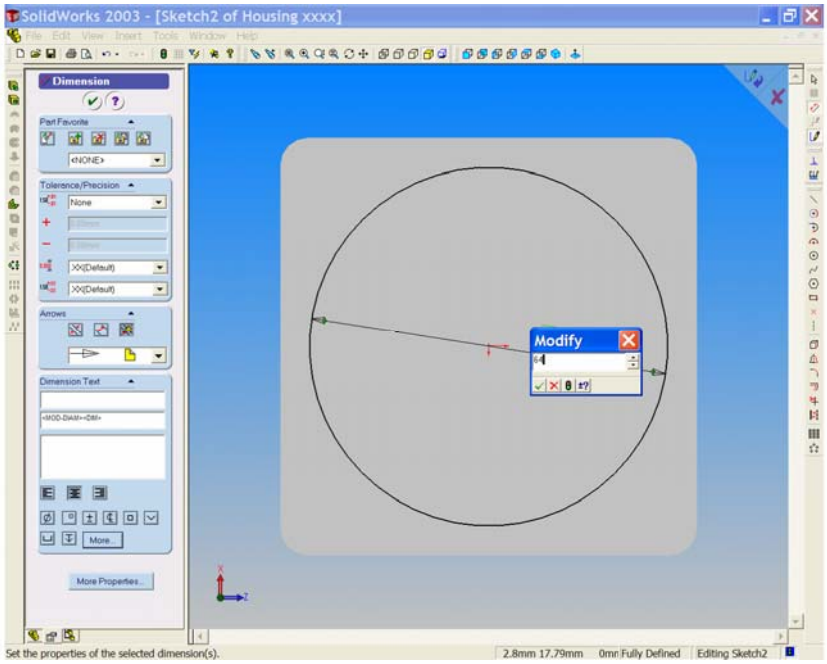

Figure 17

Right click once again and choose select to end the dimension command.

Sketch 2 is added to the Feature Manager Design tree.

30. Select '**Extruded Boss/Base**' from the Features Toolbar. The extrude feature property manager opens with a value of 10mm. Choose '**Isometric**' (from the Standard views menu) to see the model more clearly. The height of the boss should be 38mm, and we wish to merge the result. Enter the value 38 into the depth box and check the merge result box if needed.

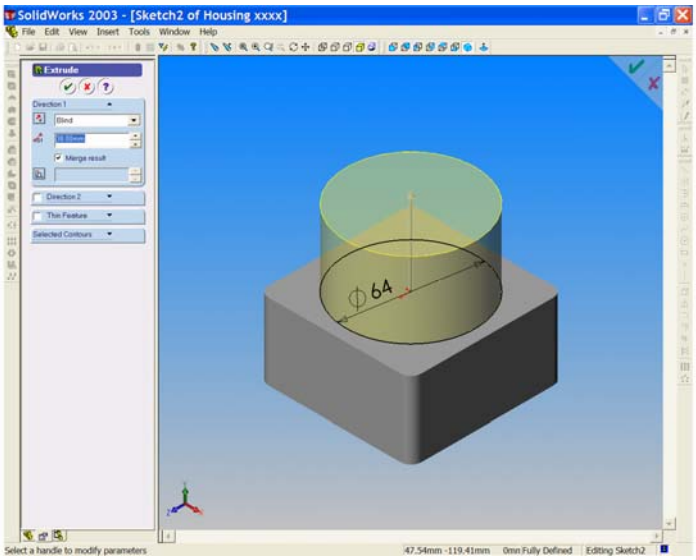

Figure 18

Notice that the circle sketch is extruded from the top surface of the base for a distance of 38mm in the direction of the arrow.

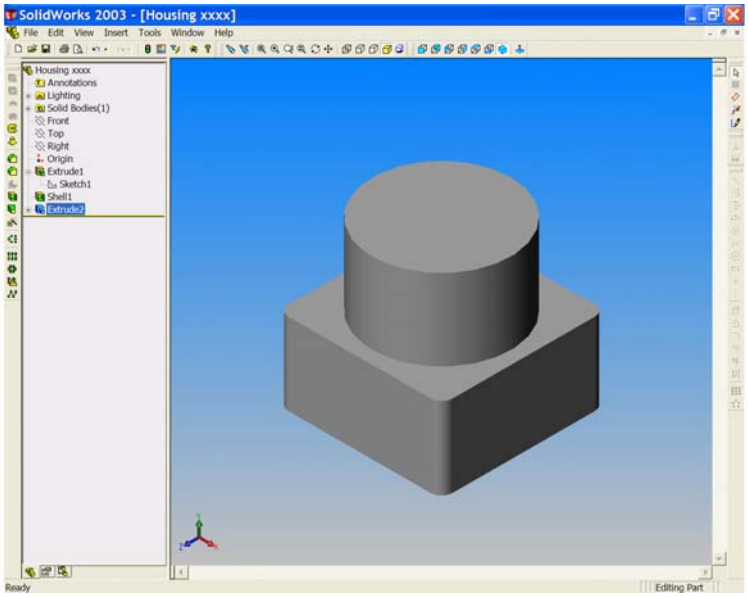

Figure 19

Left click the green tick to accept the extrusion.

Notice that Extrusion 2 is now added to the design tree. Open extrusion 2 as before and note that its base is sketch 2 (the circle).

31. Place the cursor on the top surface of the extruded boss. Notice that the surface is outlined in red and that a flag symbol appears with the cursor arrow. Left click to accept this surface; it turns green to indicate that it now active. Open a third sketch on this surface.

32. Choose '**Circle**' as before. Place the pencil symbol anywhere on the green surface **BUT NOT ON THE ORIGIN**. Draw a circle of any size.

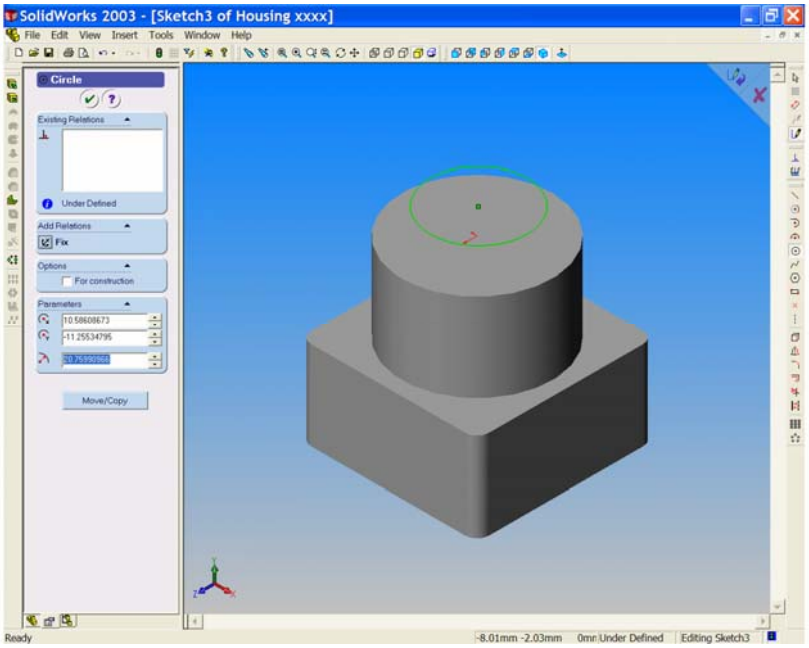

Figure 20

We now need to define this sketched circle.

33. Right click and choose '**Dimension**'. Dimension the circle to be Ø48mm.

To fully define sketch 3 we must now fix its centre. We can do this by using geometry.

34. From the 'Sketch relations' toolbar choose '**Add Relation**'. The Add Relation Property Manager opens.

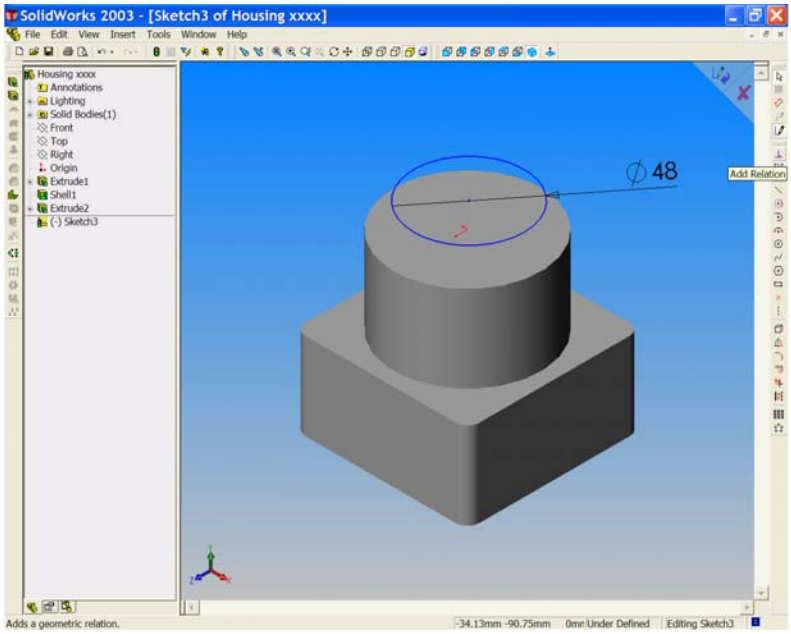

Figure 21

The selected entities box is pink indicating that it is active. Place the cursor on the blue circle and left click to select. The circle is added to the pink box as Arc 1 and the existing relation, its diameter, is listed in the white box.

Place the cursor on the top edge of the Boss. It is now red; left click to select this edge. The edge of the boss is added to the selected entities box (the pink box) as edge <1>.

The geometry relations that are now possible between these two features are now listed under 'Add relations'. We wish to make these two features concentric, i.e. to have the same centre.

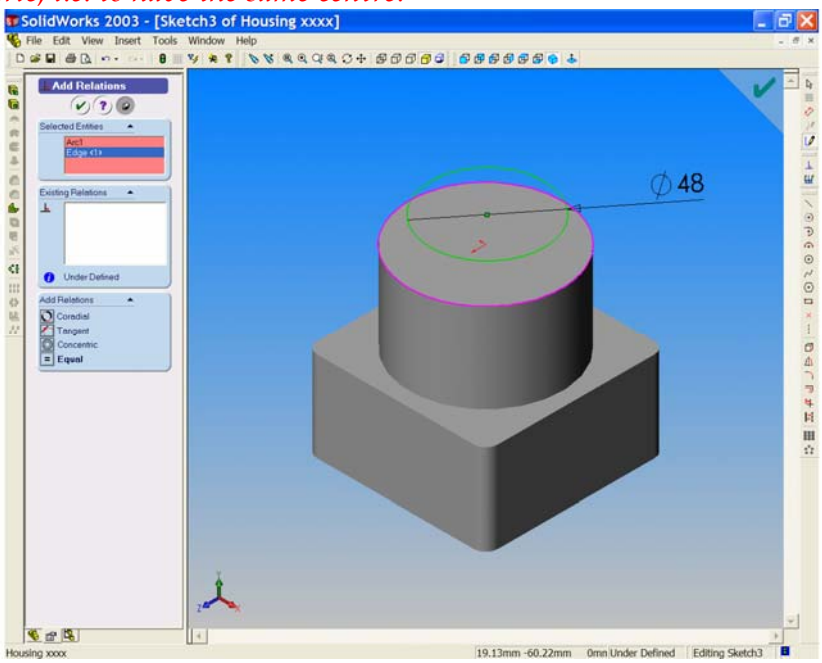

Figure 22

Choose '**Concentric**' by left clicking the mouse. Concentric 0 is added to the existing relations box and the green circle (sketch 3) moves so that its centre coincides with the boss centre. Choose the green tick to end the command.

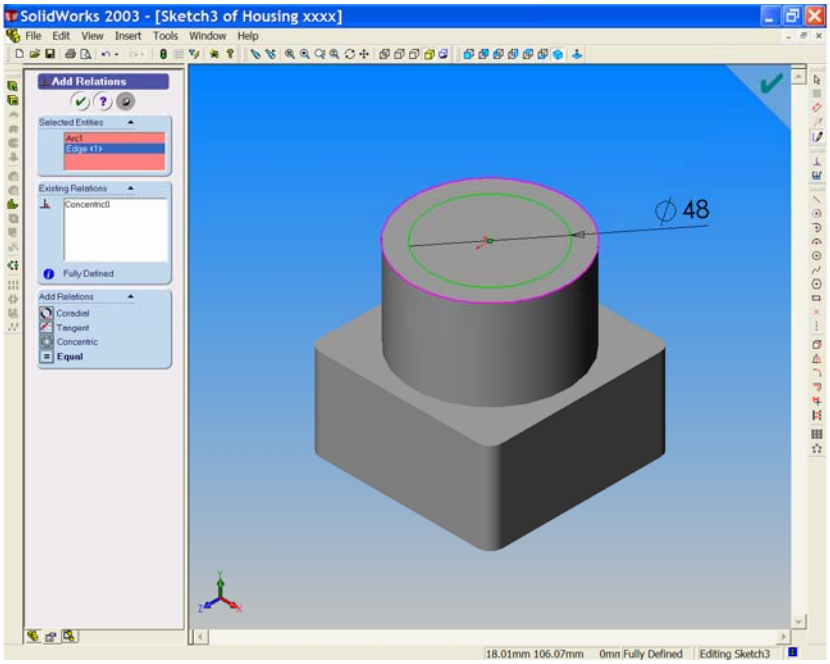

Figure 23

35. We now wish to cut a counterbore hole in the top of the Boss.

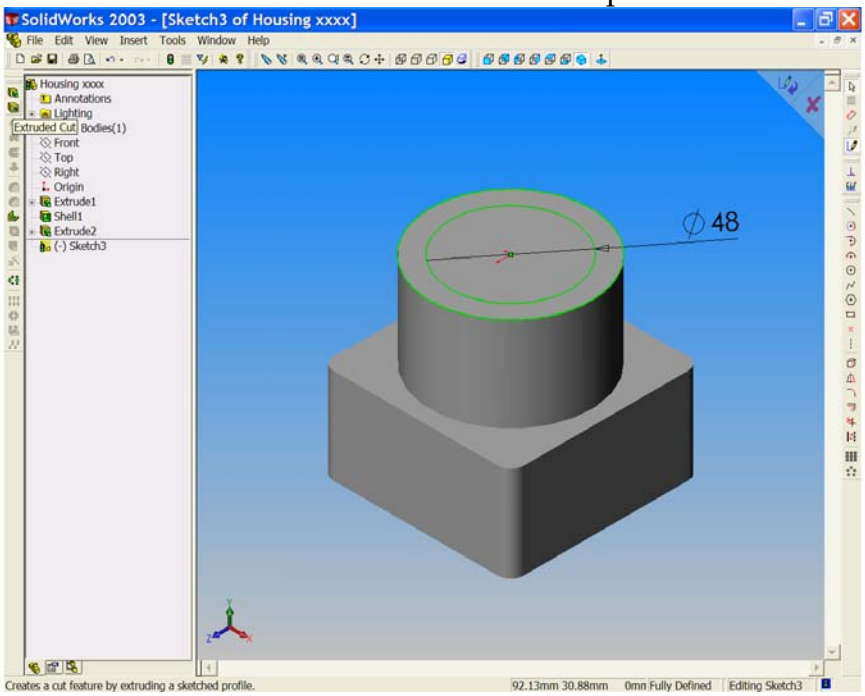

Figure 24

From the Features toolbar, select '**Extruded Cut**'. The Extruded cut Property Manager opens. In the depth window change the value to

12mm. Check that the end condition window shows 'Blind'. Select OK to cut the counterbore hole in the top pf the Boss.

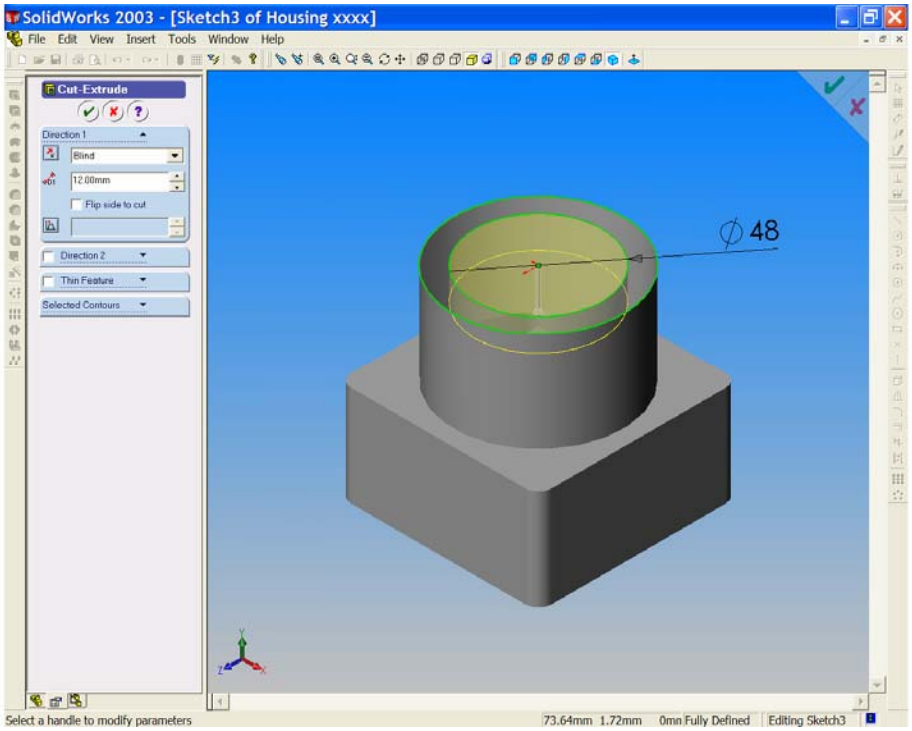

Figure 25

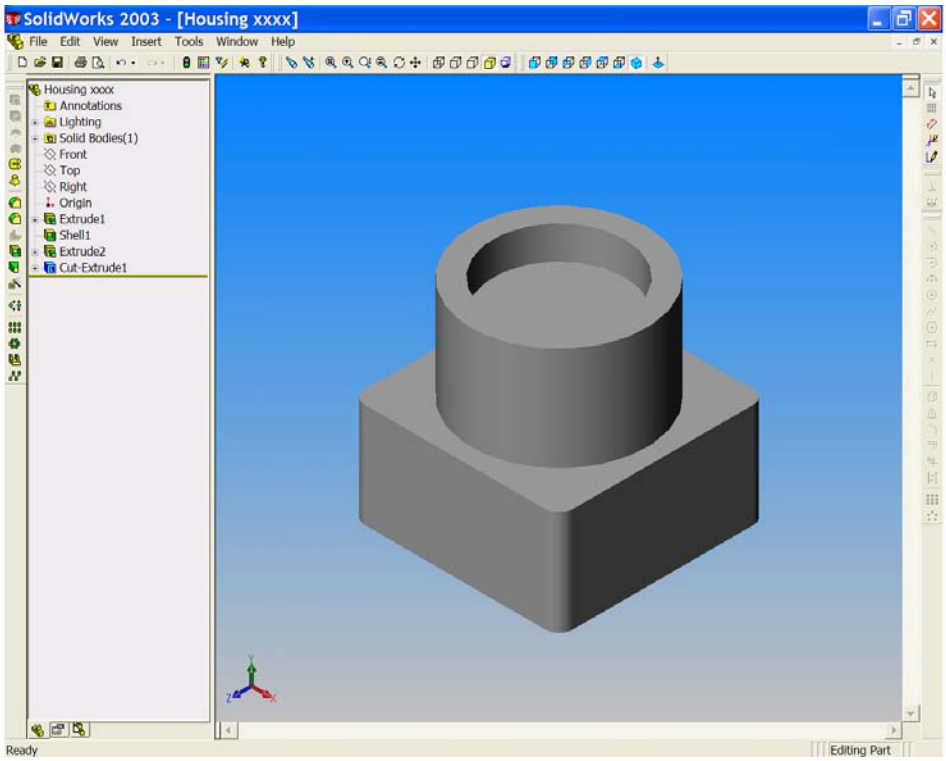

Figure 26

36. Save!!! (Regularly)

We now wish to cut a hole from the bottom of the counterbore all through the boss and into the shelled base.

37. Select the bottom of the counterbore hole. Open a new sketch on this surface and draw a Ø30mm circle concentric with the boss.

38. With sketch 4 active choose '**Extruded Cut**' . This time select the small down arrow at the right side of the '**End Condition**' window. The option to choose is '**Through All**'.

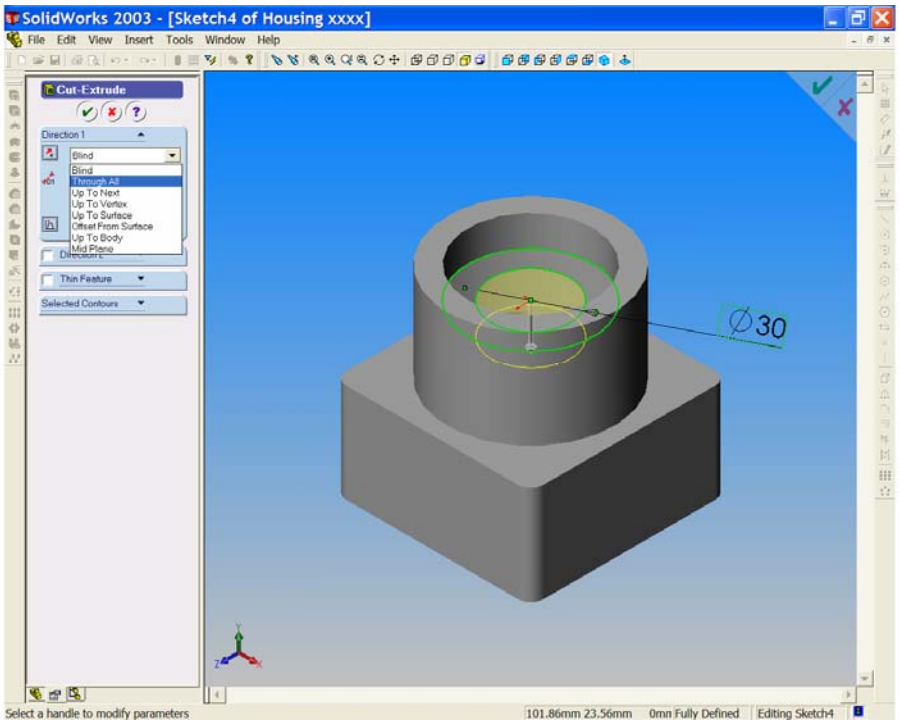

Figure 27

Notice that we do not need to give a depth when using this option.

Select OK to cut the hole.

Extrude-Cut 2 is added to the design tree. Use the rotate View button to look at the model from any direction. Return to the isometric view when you are satisfied that the hole is cut through.

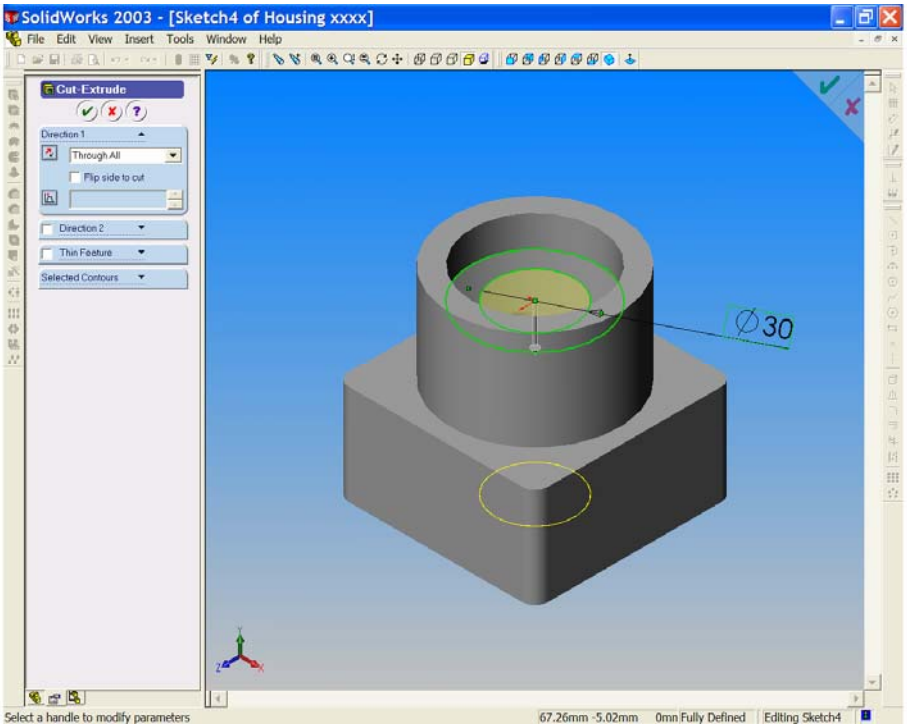

Figure 28

39. *We need to add a fillet to the bottom of the boss and the top surface of the square base. We can achieve this with the fillet command from the Feature Toolbar.*

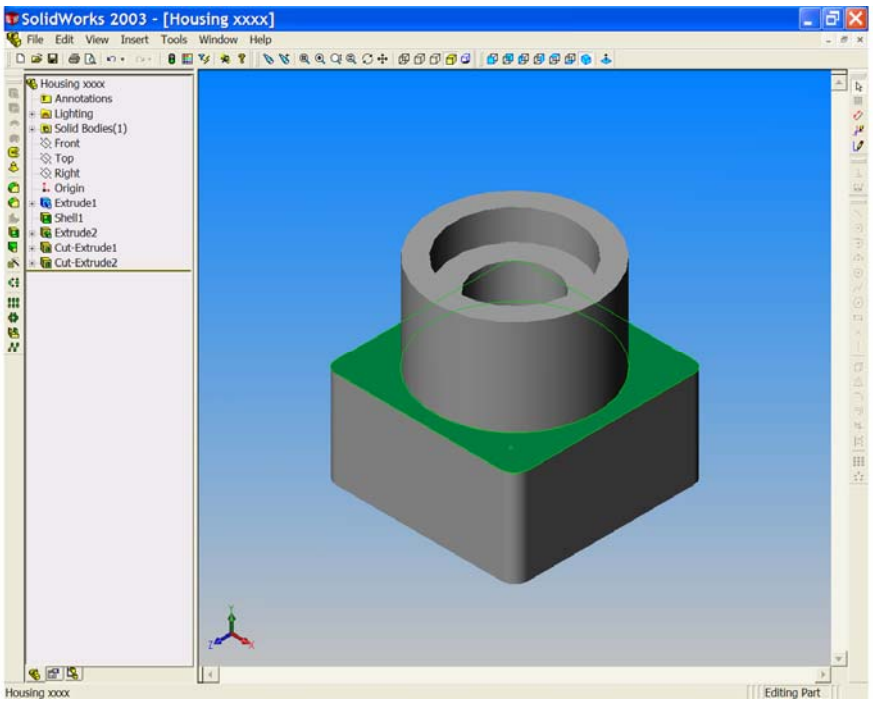

Figure 29

Select the top surface of the square base. It is now outlined and shaded green. Choose '**Fillet**' from the feature toolbar. Face <1> is already selected and shown in the '**Items to fillet**' box (the pink box). Change the fillet radius to 3mm.

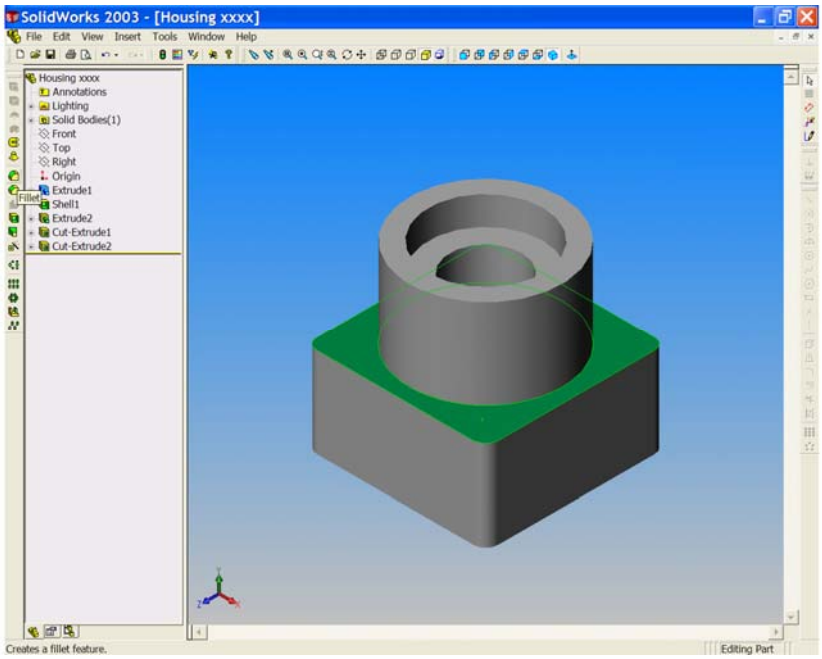

Figure 30

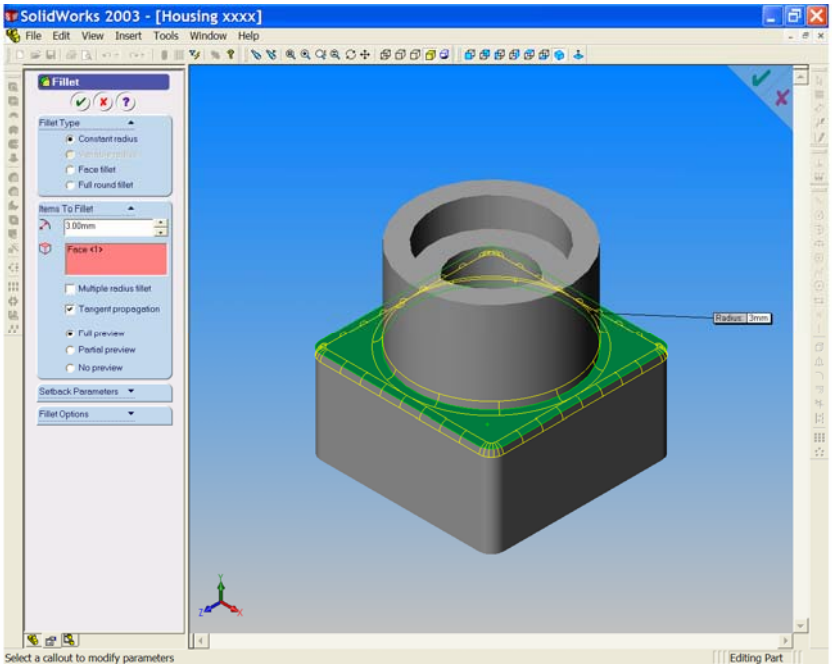

Figure 31

Notice the preview of the fillet appearing on the model, it will be shown in yellow.

Choose OK to accept the fillet.

40. Save your work !!!!

At this stage of the model construction we may add fillets to the hidden edges inside the square base.

41. From the view toolbar select '**Hidden lines visible**'. The sharp edges that need to be filleted can be seen and are shown as broken lines.

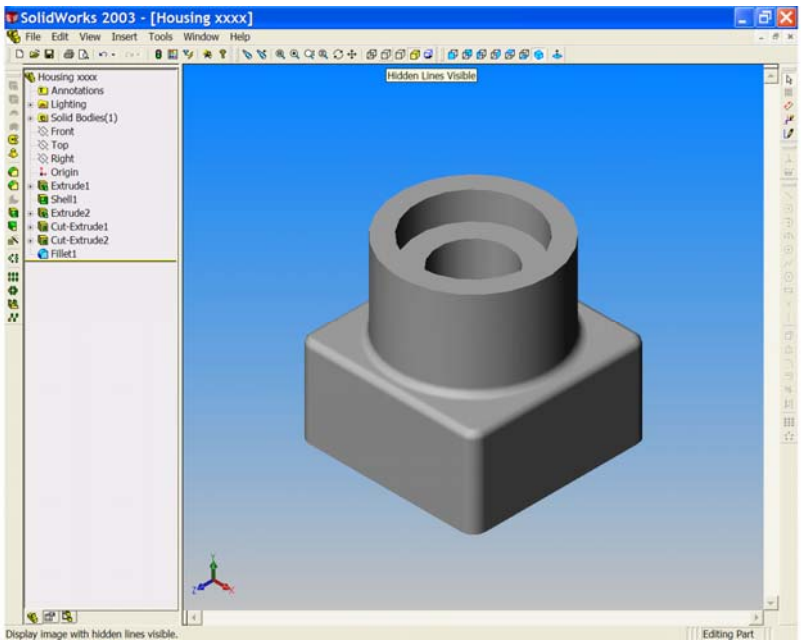

Figure 32

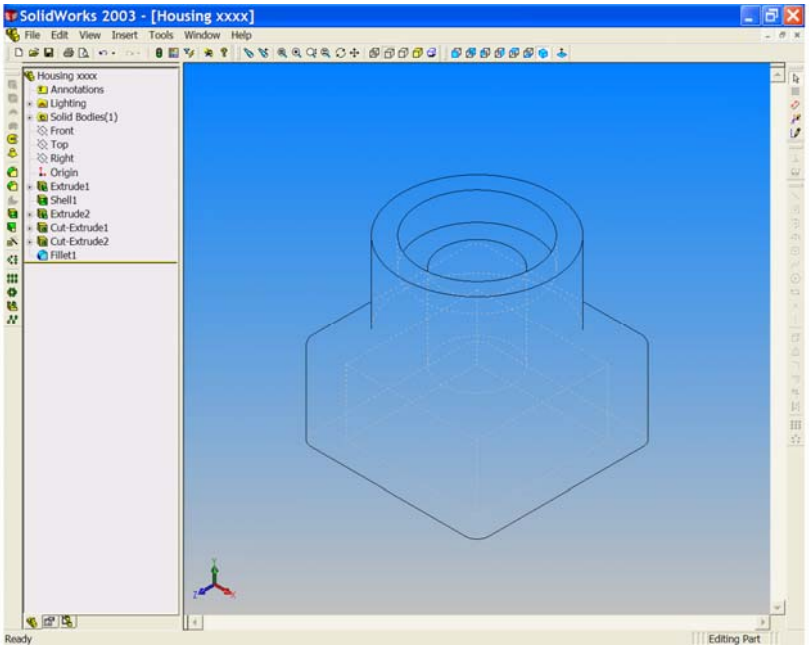

Figure 33

We need to select each of them one by one and to do this we will use the control key on the keyboard.

42. Press the control key on the keyboard and ***HOLD IT DOWN***. Select the four vertical edges. They will become red when your cursor is on them. As you select each one it will turn green.

Still holding the control key down, select the four horizontal edges in the same way. When all eight edges have been selected, release the control key.

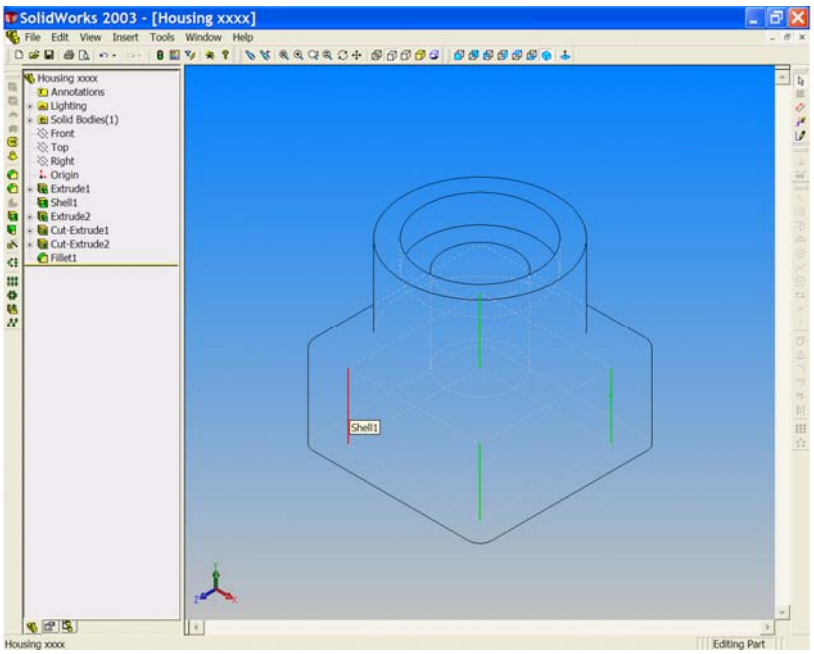

Figure 34

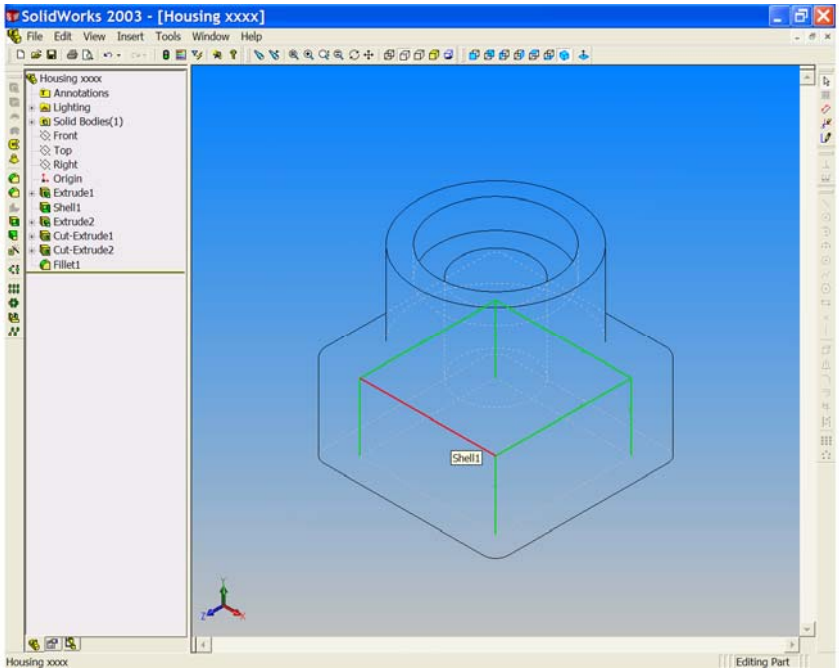

Figure 35

43. Choose the '**Fillet**' button from the Feature toolbar. Since the 3mm setting from the last command should still be active in the Fillet Property Manager, your model should show a yellow preview of the internal fillets.

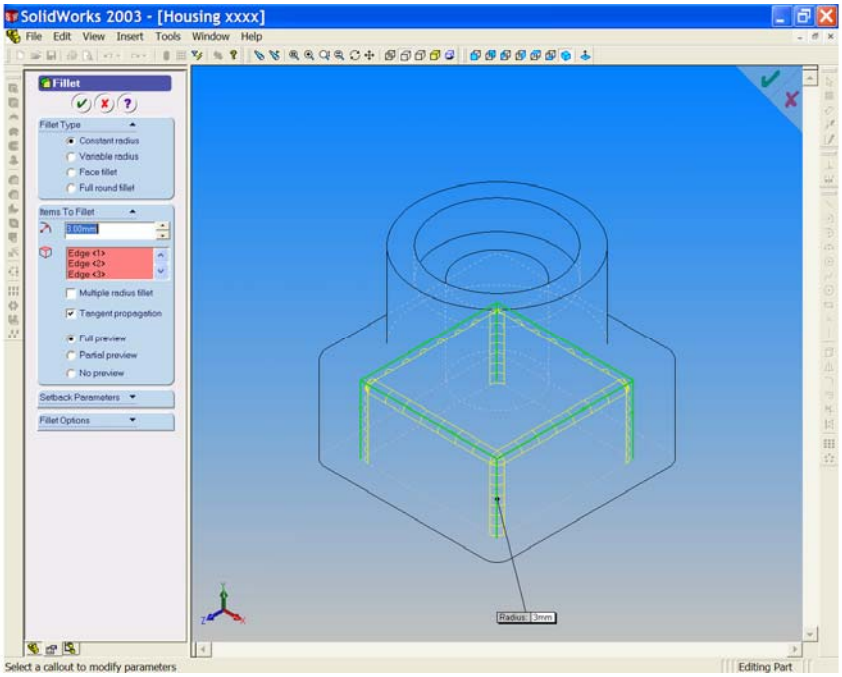

Figure 36

44. Choose **OK** to accept the fillets.

45. Sketch '**Shaded**' from the View toolbar and rotate the model to see your internal fillets. Use the '**Isometric'** button to return to the standard isometric view.

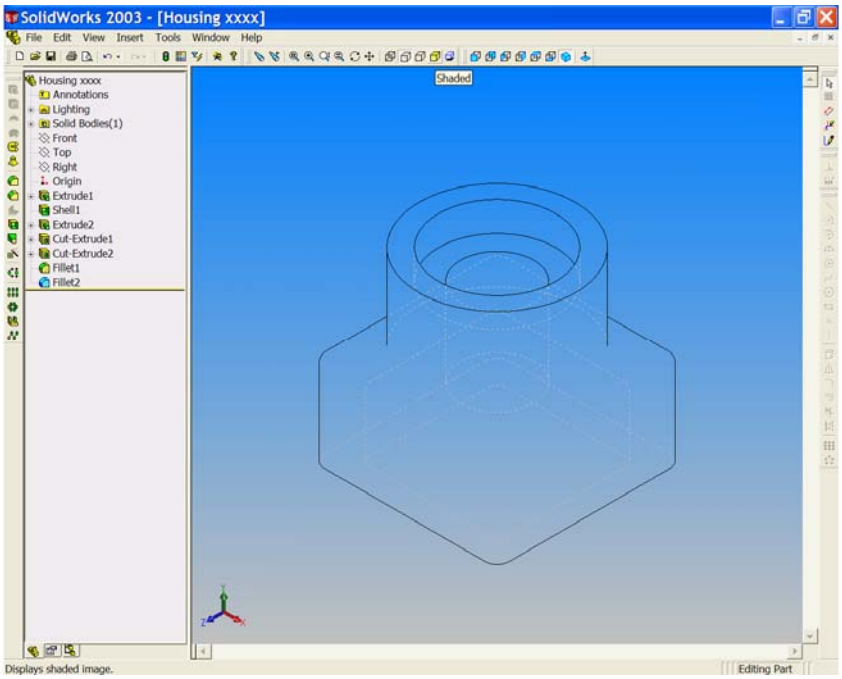

Figure 37

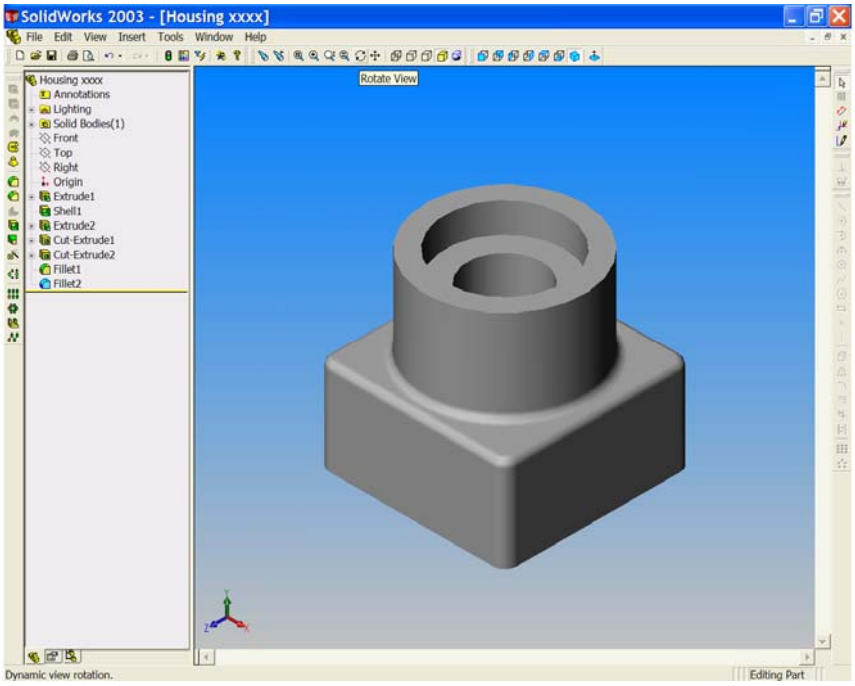

Figure 38

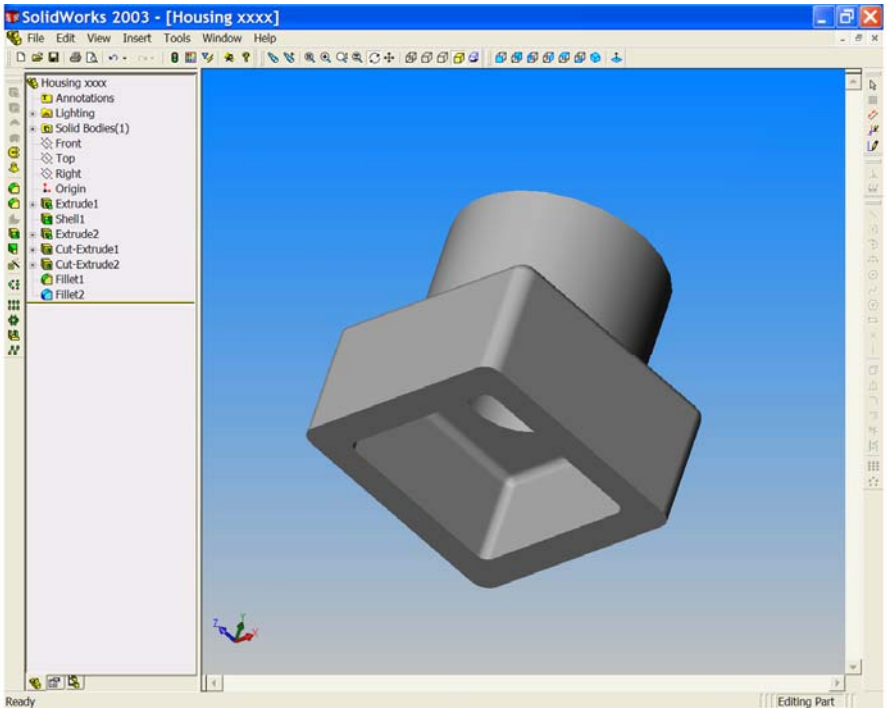

Figure 39

Save your work!

We need to add boss to the right face of the square base.

46. Select the right face of the square base, look for the flag and notice the selected surface colour change as before.

47. Choose '**Normal to**' and open a sketch on this surface.

48. Select '**Circle**' and draw a circle towards the right side of the surface.

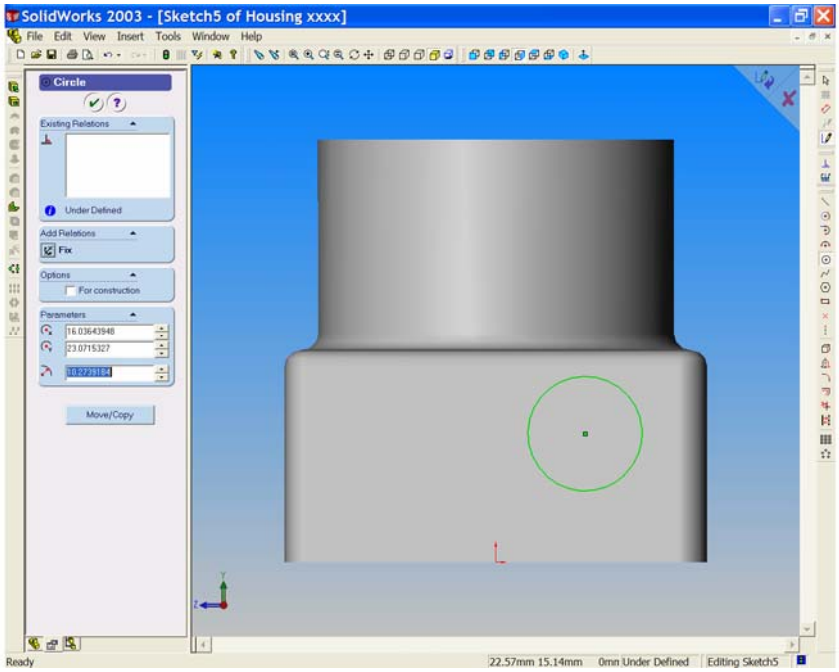

Figure 40

49. Right click and choose '**Dimension**' from the short-cut menu. Dimension the circle to have a Ø24mm.

50. Continue with the dimension tool to position the circle centre at 24mm from the right edge and 24mm from the top surface of the square base. Sketch 5 is now fully defined.

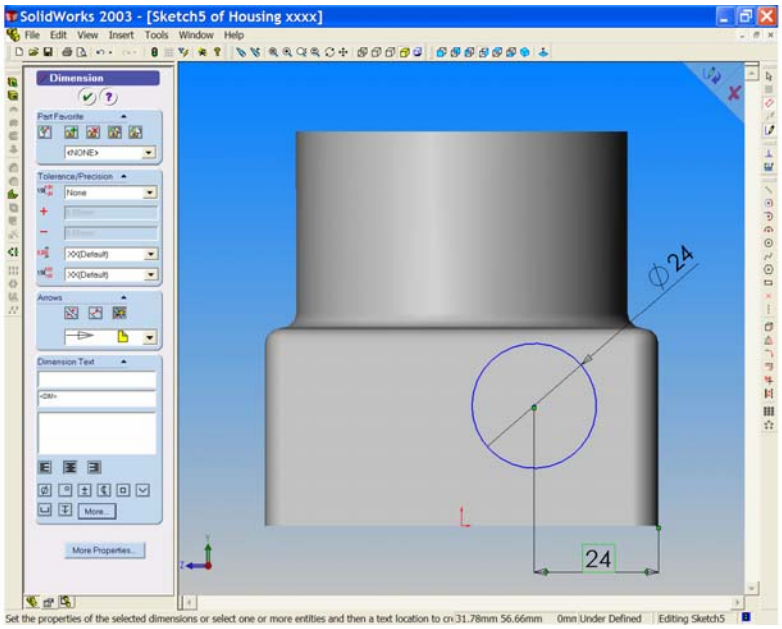

Figure 41

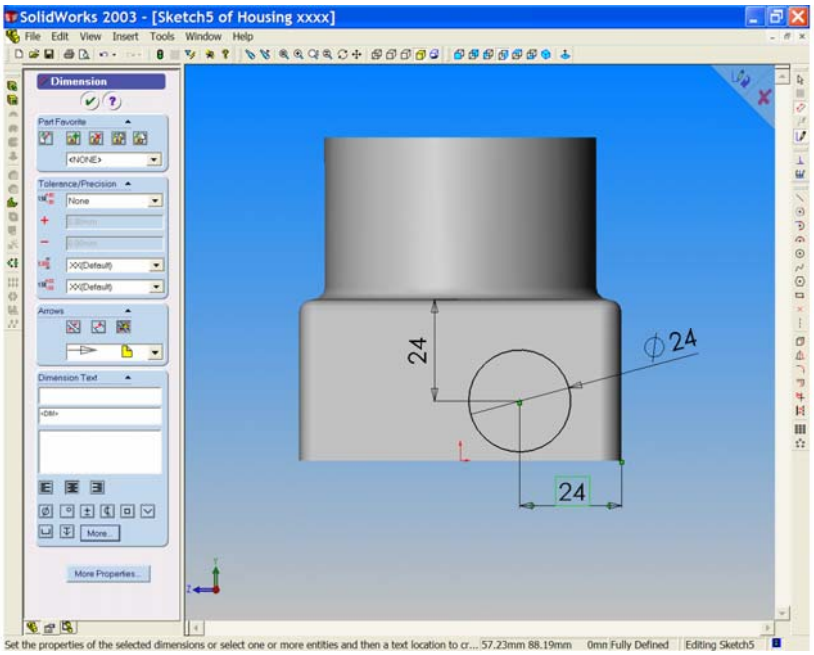

Figure 42

51. Use the '**Extrude Boss/Base**' tool to extrude the circle for a distance of 10mm. Use the isometric button to help you see what you are doing.

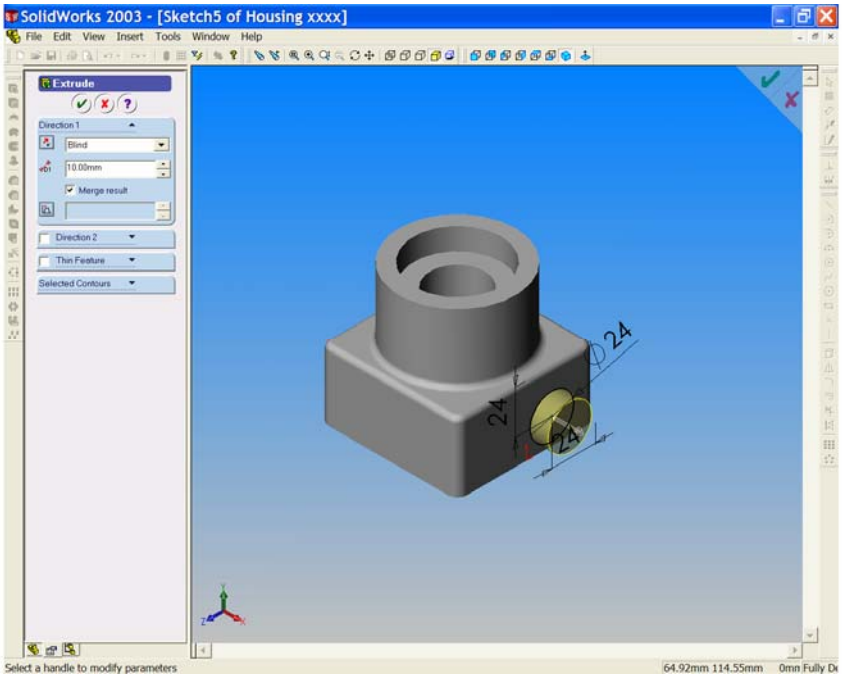

Figure 43

52. Add a 3mm fillet to the new boss feature. Select the circle at the base of the new boss and then choose the '**Fillet**' button from the features toolbar. Enter a value of 3mm into the radius window and then **OK**.

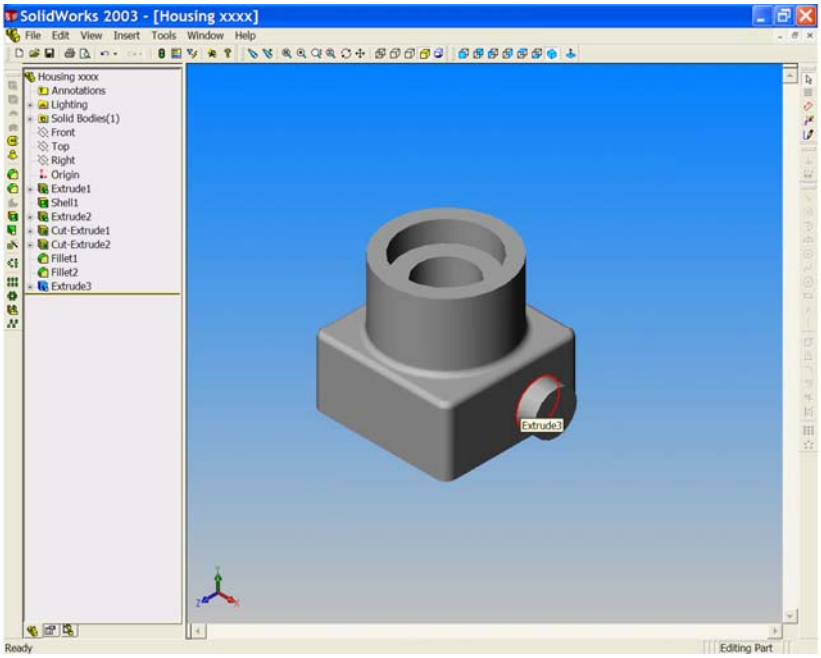

Figure 44

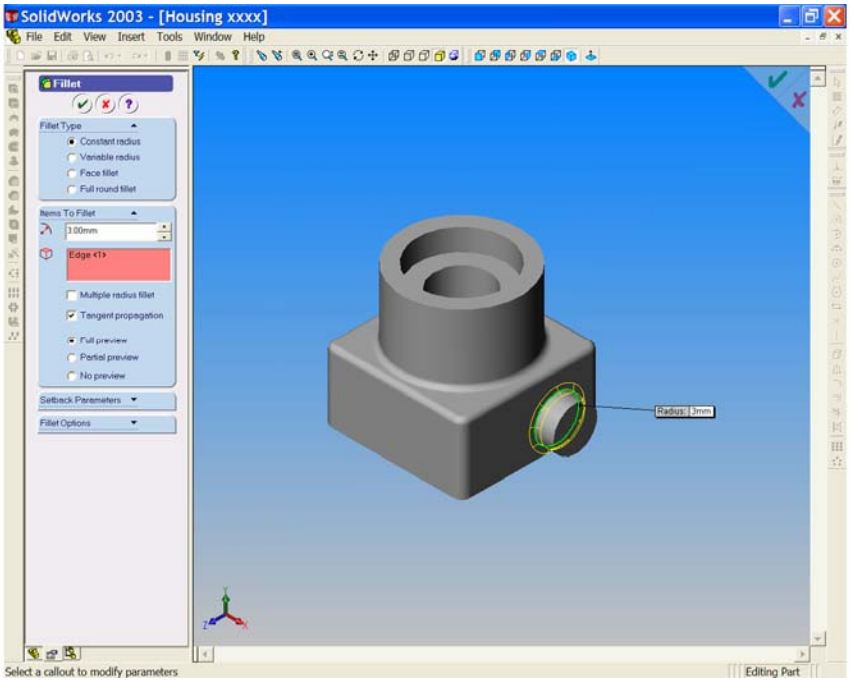

Figure 45

53. The last requirement for this model is a hole through this boss. Select the flat outer surface of the boss and open a '**Sketch**'. Choose '**Circle**' and draw a circle on this surface. Right click and dimension the circle to have a Ø16mm.

54. Now add a 'Concentric relation' to place the circle. Choose '**Add relation**', select the circle and then select the edge of the boss. Choose 'concentric' from the Add relations Property Manager.

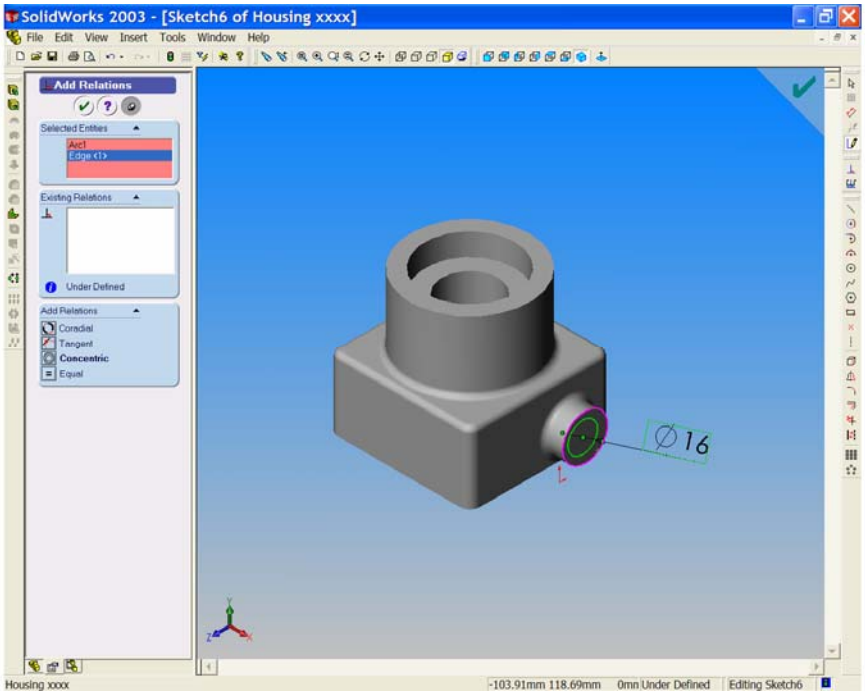

Figure 46

55. Use '**Extruded Cut**' to make the hole. Be sure to keep the end condition as '**Blind**'. Change the depth to 20mm and click **OK**.

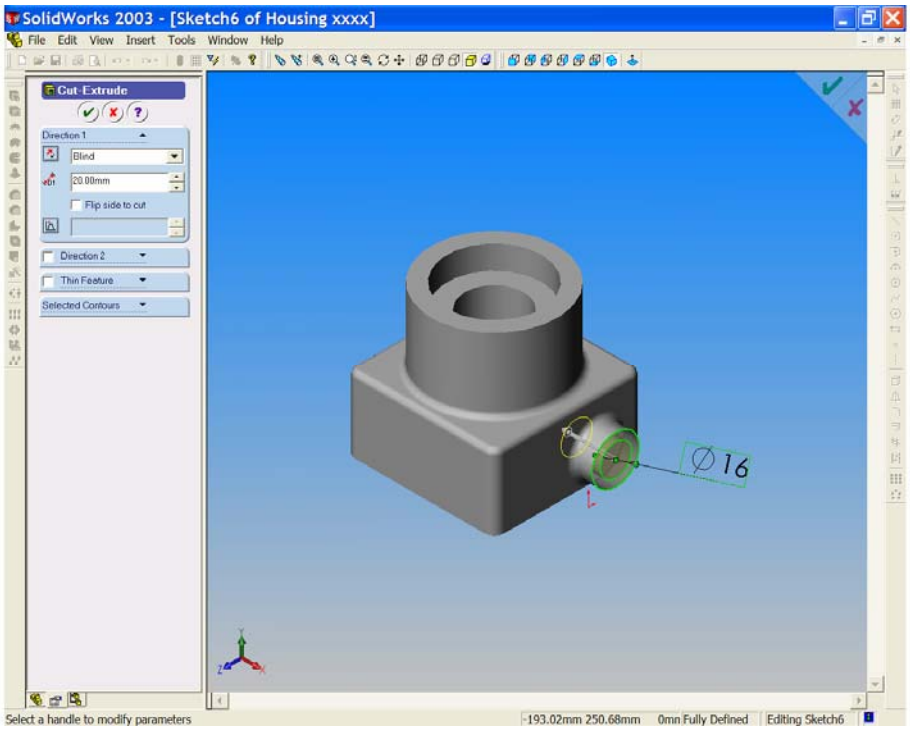

Figure 47

Rotate the model to see inside the base.

56. To change the colour of the model, first select the file name at the top of the feature manager design tree.

It is now highlighted, select '**Edit Colour**' from the Standard toolbar. You may now choose any colour you like! Try one of them or define your own colour. When you have selected a colour pick the 'Apply' button and then **OK**. Your model changes to your chosen colour!

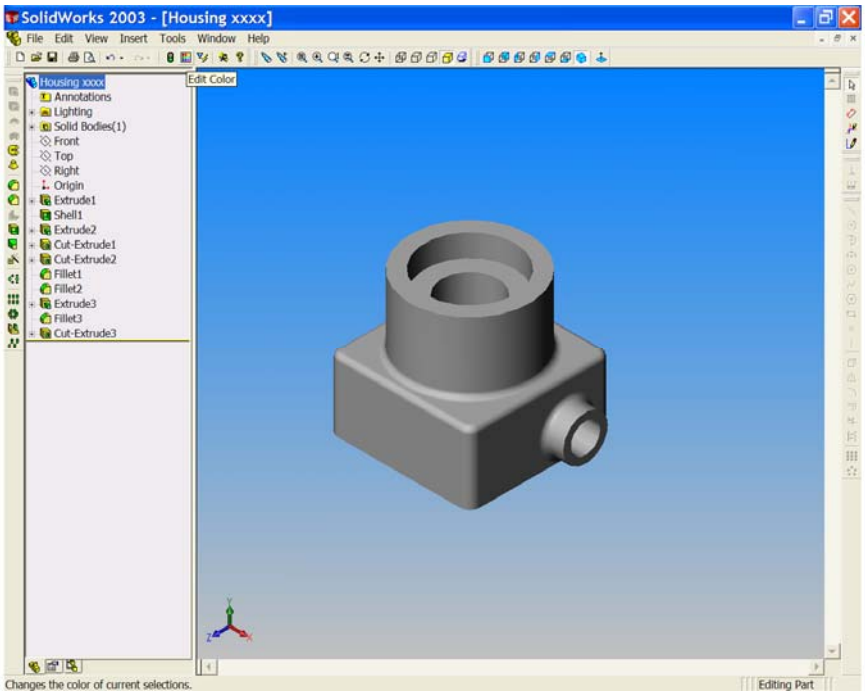

Figure 48

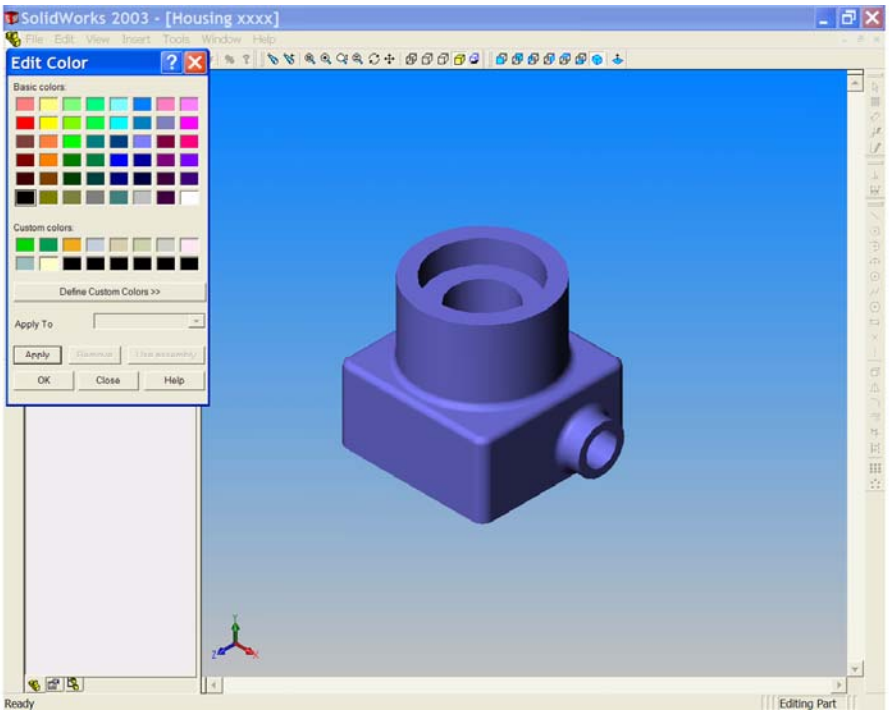

Figure 49

*Your model is complete so ……. **SAVE YOUR WORK*** *!*