Module 2a: Basic Sketching I

In this tutorial, you will draw the sketch of the model shown in Figure 1. The sketch is shown in Figure 2. You will not dimension the sketch. The solid model and the dimensions are given only for your reference.
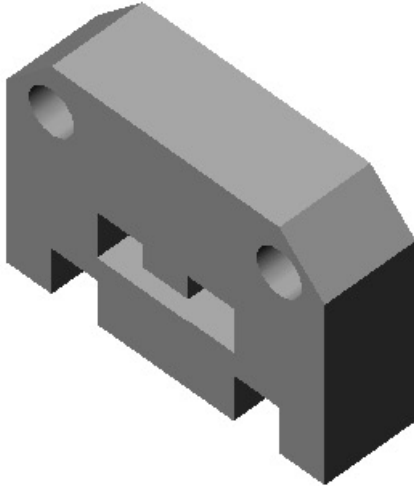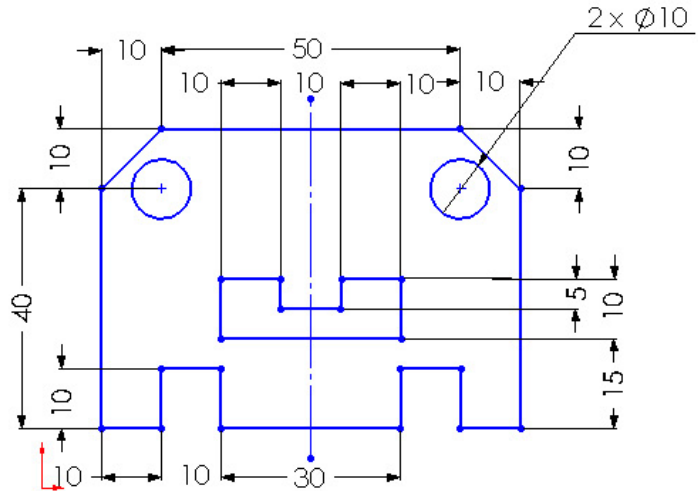

Figure 1 Solid model   **Figure 2 Sketch for the solid model**

The steps that will be followed to complete this tutorial are listed below:

1. Start SolidWorks and then start a new part document.

2. Maximize the part file document and then switch to the sketching environment.

3. Draw the sketch of the model using the Line and the Circle tools, refer to Figures 4 through 7.

4. Save the sketch and then close the document.

Starting SolidWorks and Starting a New Part Document

1. Start SolidWorks by choosing **Start> Programs> SolidWorks 2005 > SolidWorks 2005** or by double-clicking the shortcut icon of SolidWorks available on the desktop of your computer.

2. The SolidWorks 2005 window is displayed. Choose the New Document option from this window. The New SolidWorks Document dialog box is displayed.

3. The Part button is chosen by default. Choose the OK button from the New SolidWorks Document dialog box as shown in Figure 3. A new SolidWorks part document will be opened. But the part document window will not be maximized in the SolidWorks window.
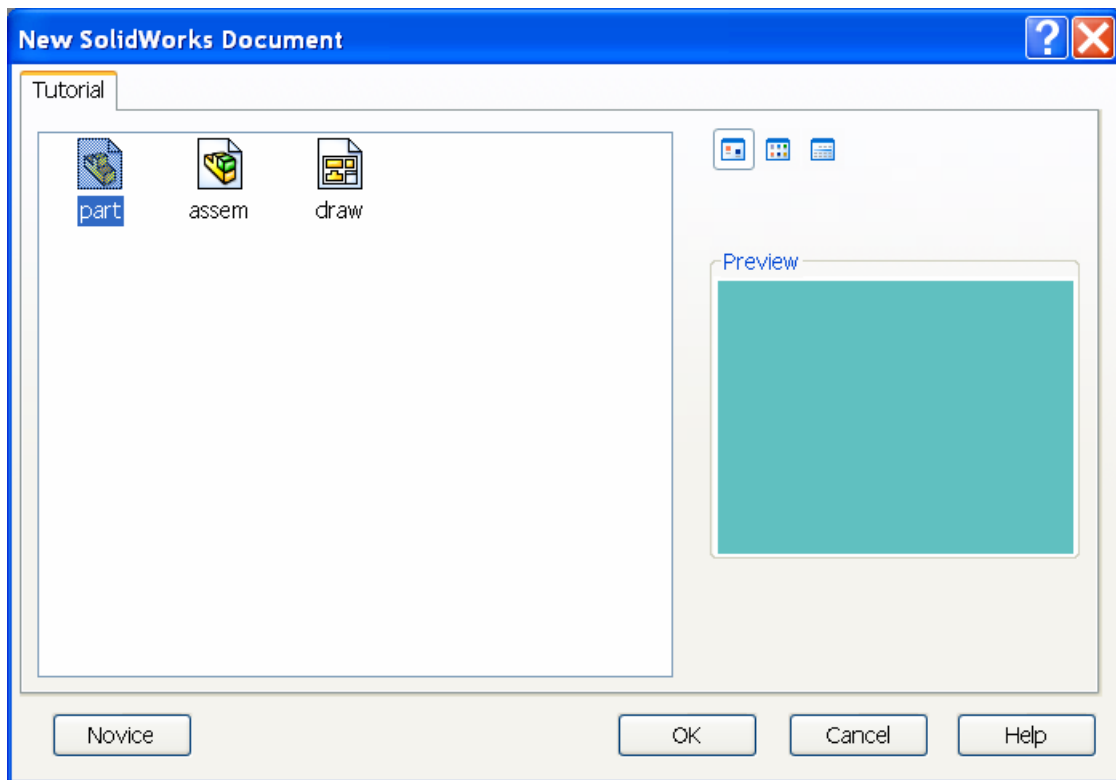
Figure 3 New SolidWorks document dialog box.

4. Choose the Maximize ⬜ button available on the upper right corner of the part document window to maximize the document window. When you open a new part document, the part modeling environment is active by default. As you first need to draw the sketch of the feature, you need to invoke the sketching environment.

5. Select the **Front** plane.

6. Choose the **Sketch** 📝 button from the Standard toolbar. The **Sketch tools** toolbar becomes highlighted. The sketching environment is invoked and the plane is oriented normal to the view i.e., parallel to screen. You will notice that a red color origin is displayed in the center of the screen, indicating the sketching environment. The default screen appearance of the sketching environment of SolidWorks is shown in Figure 4.
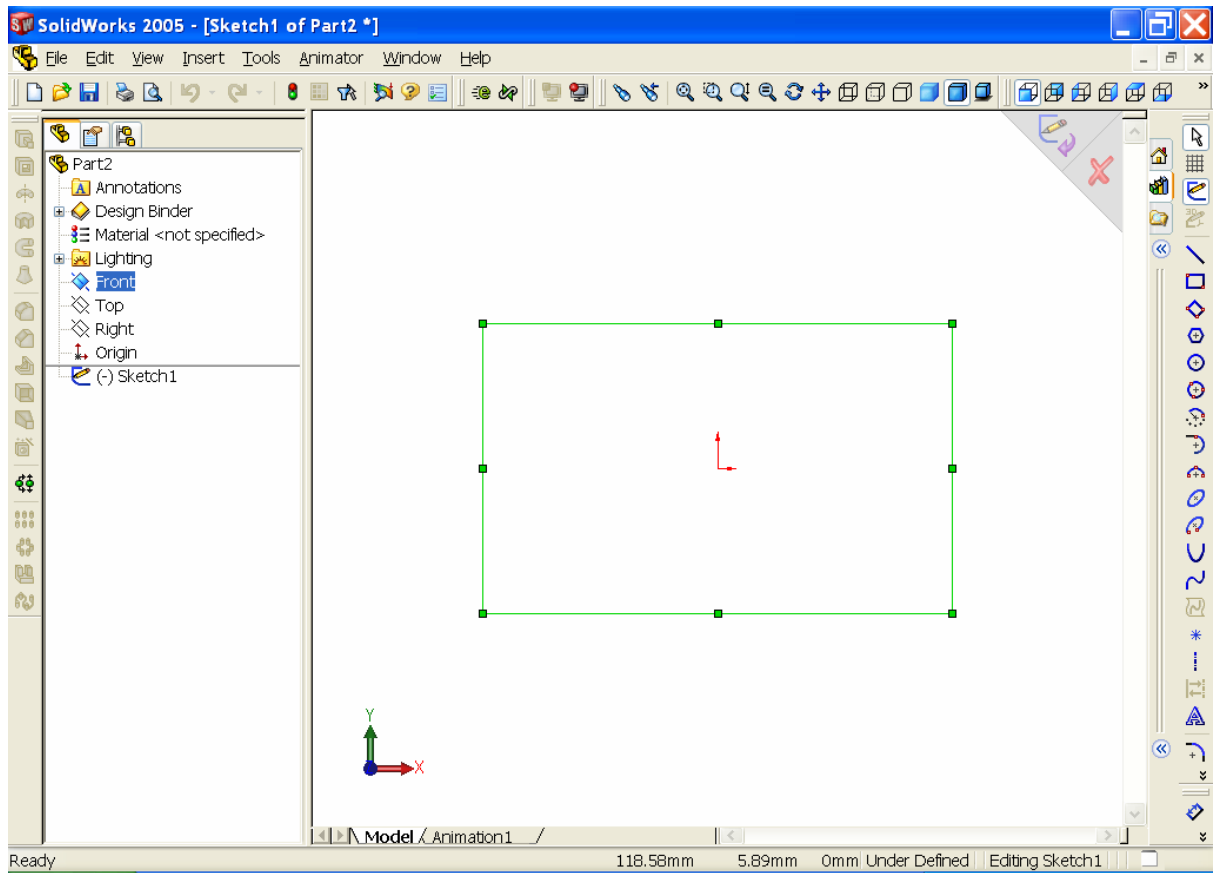
Figure 4 Starting a new sketch.

Setting the Units and Grid

1. Choose **Tools> Options** from the menu bar to invoke the **System Options -General** dialog box.

2. Choose the **Document Properties** tab. The name of the dialog box is changed to the **Document Properties -Detailing** dialog box.

3. Select the Units option from the area on the left to display the options related to linear and angular units.

4. Select **millimeters** from the drop-down list available in the **Length units** area. Also, select the **Degrees** option from the drop-down list provided in the **Angular units** area.

5. Select **Grid/Snap** from the area on the left. Set the value of the **Major grid spacing** spinner to 100 and the value of the **Minor-lines per major** spinner to 10.

6. Select the **Go to System Snaps,** check **Grid** box in the **Sketch Snap** area, if it is cleared. Choose OK to exit the dialog box.

Drawing the Outer Loop of the Sketch

It is a good practice to draw the sketch on one side of the origin, preferably in the first quadrant. This is because while generating the part program for manufacturing the part, you will have a reference for work origin in advance.

The sketch of the model consists of an outer loop, two circles inside the outer loop, and a cavity. Therefore, it will be drawn using the Line and the Circle tools. You will first draw the outer loop and then the inner entities. Note that in the sketcher environment, the lower right corner of the SolidWorks window displays three areas. The first area displays the X, Y, and Z

coordinates of the current location of the cursor. These coordinates will be modified as you move the cursor around the drawing area. You will use the coordinate display to draw the sketch of the model.

You will start drawing the sketch from the lower left corner of the sketch and the outer; loop will be drawn using the continuous lines.

1. Click **Line** button on the Sketch toolbar to invoke the **Line** tool. The arrow cursor will be replaced by the line cursor.

2. Move the cursor in the first quadrant close to the origin. The coordinates of the point are displayed close to the lower right corner of the screen.

3. Press the left mouse button at the point whose coordinates are **10 mm 10 mm 0 mm** and then move the cursor horizontally toward the right.

   You will notice that the symbol of **Horizontal** relation is displayed below the line cursor and the length of the line is displayed above the line cursor. As the length of the first horizontal line at the lower left comer is 10 mm, you will move the mouse until the length of the line is shown as 10 above the line cursor.

4. Press the left mouse button when the length of the line that is displayed above the line cursor shows a value of **10**.

   The first horizontal line is drawn. Because you are drawing continuous lines, the endpoint of the last line is automatically selected as the start point of the next line.

5. Move the line cursor vertically upward. The symbol of **Vertical** relation is displayed below the line cursor and the length of the line is displayed above the line cursor.

6. Press the left mouse button when the length of the line displayed above the line cursor shows a value of **10**.

   A vertical line of length **10 mm** will be drawn and will be displayed in green color. Also, as this is the line that is selected now, the previous line will no more be highlighted and therefore will be displayed in blue color.

7. Move the line cursor horizontally toward the right. Press the left mouse button when the length of the line above the line cursor shows a value of **10**. This draws the next horizontal line of **10 mm** length.

8. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor shows a value of **10**.

9. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of **30**.

10. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor shows a value of **10**.

11. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of **10**.

12. Move the line cursor vertically downward and press the left mouse button when the length of the line on the cursor shows a value of **10**.

13. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of **10**.

14. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **40**.

The next line that you will draw is an inclined line that makes an angle of 135°. To draw this line, you will move the cursor in a direction that makes an angle of 135°.

The aligned length of the line is not given for this line. Instead, the delta X and delta Y values are given. Therefore, you will draw a line at this point such that the delta X and delta Y value of this line is **10** mm each. These values can be viewed in the **Line Property Manager**. The last spinners in the **Line PropertyManager** are for the delta values.

15. Move the line cursor such that the line is drawn at an angle of 135-degree. The current angle will be displayed in the spinner above the delta spinners in the **Line PropertyManager**.

16. Press the left mouse button when the delta X value in the **Delta X** spinner shows a value of **10** and the value of delta Y in the **Delta Y** spinner displays a value of **10** in the **Line Property Manager**. At this stage, the length of the line will be displayed as 14.14 above the line cursor and also in the **Length** spinner in the **Line PropertyManager**.

17. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **50**.

You will notice that some blue and brown inferencing lines are displayed when you move the cursor.

18. Move the line cursor in the direction of 225-degree angle.

You will notice that two blue inferencing lines are displayed. The first one originates from the start point of the first inclined line and the other one from the start point of the first line of the sketch.

19. Press the left mouse button at the point where the two inferencing lines intersect.

You will notice that at this point, the length of the line on the line cursor shows a value of 14.14. Also, the delta X and delta Y values in the **Line Property Manager** are displayed as **10** each and the angle is shown as 225-degree.

20. Move the cursor vertically downward to the start point of the first line.

You will notice that when you move the cursor close to the start point of the first line, a red circle is displayed. Also, the line cursor turns yellow in color and an orange-colored box is displayed below the line cursor. The length of the line shows a value of **40**.

21. Press the left mouse button when the red circle is displayed. Right-click to display the shortcut menu and choose the **Select** option to exit the **Line** tool.

This completes the sketch of the outer loop. Note that the display of the sketch is small. Therefore, you need to modify the drawing display area such that the sketch fits the screen. This is done using the **Zoom to Fit** tool.

22. Choose the **Zoom to Fit** button from the **View** toolbar to fit the current sketch on the screen.

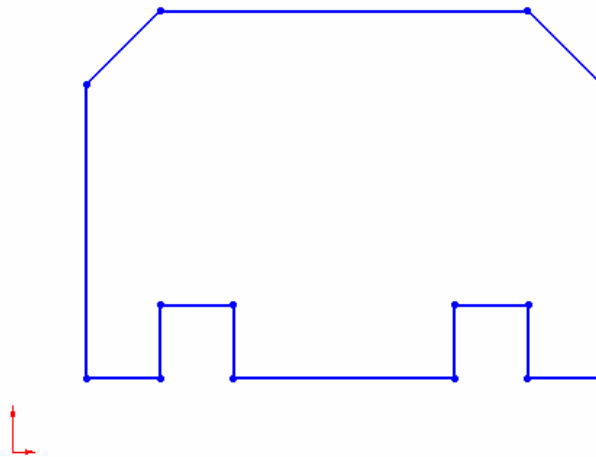The outer loop of the sketch is completed and is shown in Figure 5.

Figure 5 Outer loop of the sketch.

Drawing Circles

The circles will be drawn using the **Circle** tool. You will use the inferencing line originating from the start points and endpoints of the inclined lines to specify the center point of the circles. As mentioned earlier, the cursor in the sketching environment jumps through a distance of 10 mm by default. Therefore, when you move the cursor to define the radius *of* the circle, the minimum value that can be set is 10 mm. But the radius of the circle is 5 mm and to specify this value, the cursor needs to jump only through a distance of 5 mm. This is the reason you need to modify the document settings so that the cursor jumps through a distance of 5 mm.

1. Choose **Tools> Options** from the menu bar to invoke the **System Options -General** dialog box. Choose the **Document Properties** tab.

2. Select the **Grid/Snap** option from the area on the left to display the options related to linear and angular units. Set the value of the **Major grid spacing** spinner to **50** and choose **OK**.

   Setting this value to 50 will force the cursor to jump through a distance of 5 mm instead of 10 mm. Therefore, when you move the cursor now, the values shown on the cursor and the coordinates of the points shown close to the lower right corner of the SolidWorks window will be in the increment of 5 mm.

3. Choose the **Circle** ⊕ button from the **sketch toolbar** to invoke the Circle tool.

   As the **Select** ⬚ tool was active earlier, the cursor earlier was the arrow cursor. But when you invoke the **Circle** ⊕ tool, the arrow cursor will be replaced by the circle cursor.

4. Move the circle cursor close to the lower endpoint of the right inclined line and then move it toward the left. Remember that you will not press the left mouse button at this moment.

   An inferencing line is displayed generating from the lower endpoint of the right inclined line. When you move the cursor toward the left, you will notice that at the point where the cursor is vertically in line with the upper endpoint of the right inclined

line, another inferencing line is generated from the upper endpoint of the right inclined line. This inferencing line will intersect the inferencing line generated from the lower endpoint of the inclined line.

5. Press the left mouse button at the point where the inferencing lines from both the endpoints of the inclined lines intersect. Now, move the circle cursor toward the left to define a circle.

6. Press the left mouse button when the radius of the circle displayed above the circle cursor shows a value of **5**.

7. Similarly, draw the circle on the left using the inferencing lines generating from the endpoints of the left inclined line. The sketch after drawing the two circles inside the outer loop is shown in Figure 6.

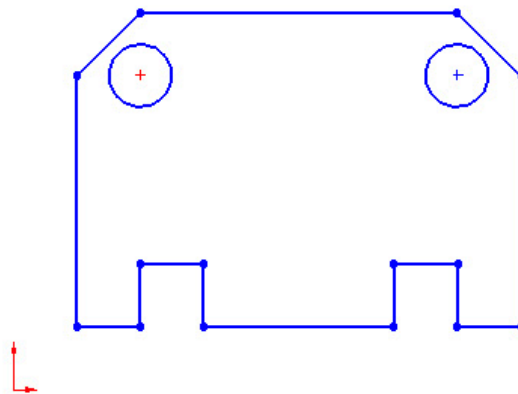8. Right-click in the drawing area and choose Select to exit the Circle tool.

Figure 6 Outer loop of the sketch including circles.

Drawing the Sketch of the Inner Cavity

Next, you will draw the sketch of the inner cavity. To draw the sketch of the inner cavity, you will start drawing with the lower horizontal line.

1. Choose the **Line** button from the Sketch toolbar. The arrow cursor is replaced by the line cursor.

2. Move the line cursor to a location whose coordinates are **30 mm 25 mm 0 mm**.

3. Press the left mouse button at this point and move the cursor horizontally toward the right. Press the left mouse button when the length of the line above the line cursor shows a value of **30**.

4. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **10**.

5. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **10**.

6. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of **5**.

7. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **10**.

8. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **5**.

9. Move the line horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **10**.

10. Move the line cursor vertically downward to the start point of the first line. Press the left mouse button when the red circle is displayed. The length of the line at this point will show a Value of **10**.

11. Right-click and choose Select from the shortcut menu. This completes the sketch for this Tutorial. The final completed sketch is shown in Figure 7.
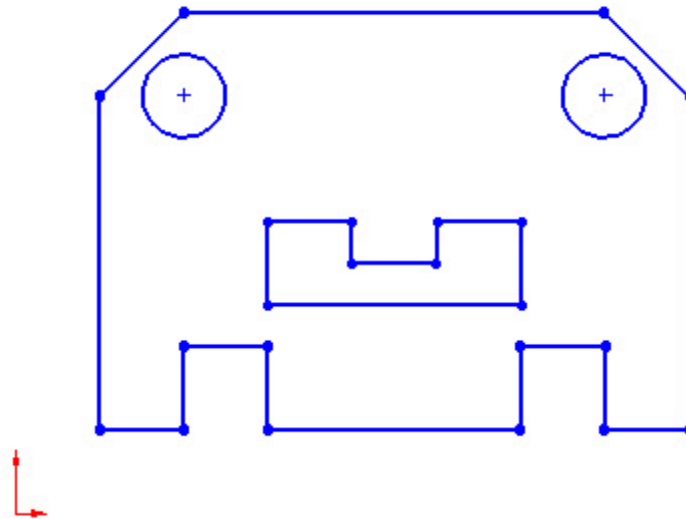
Figure 7 Completed sketch for the model.

Saving the Sketch and model

1. Choose the **Save** ![save icon] button from the Standard toolbar to invoke the **Save As** dialog box.

2. Enter the name of the document as *tutorial1.sldprt* in the File name edit box and choose the Save button. The document will be saved in the *specified* folder. It is recommended that you create a separate folder for SolidWorks files.
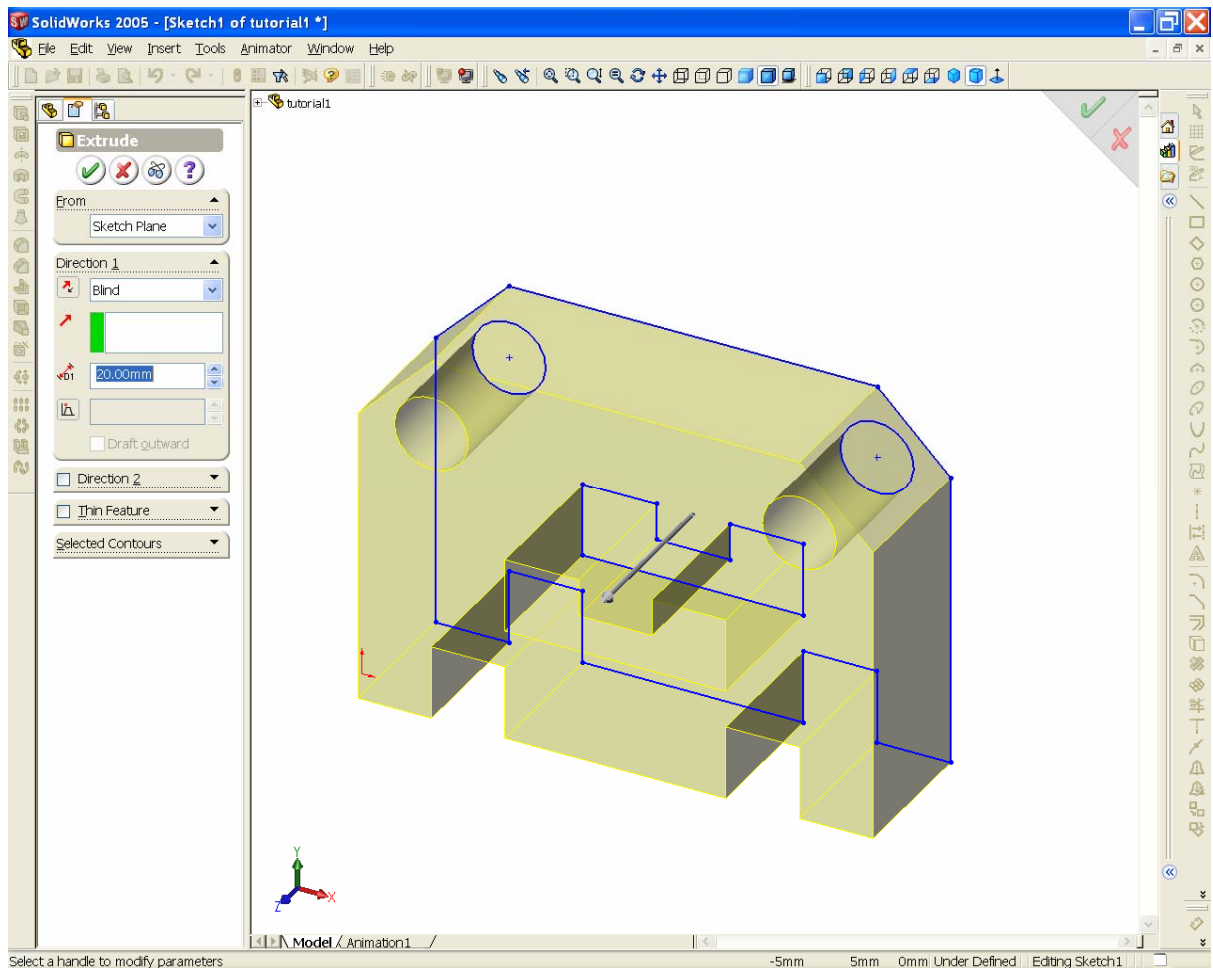
Figure 8

3. Select **Extrude Boss/Base** on the Features toolbar. The profile changes its orientation to pictorial. The Extrude Property Manager appears as shown in Figure 8.

4. In the **Depth** text box, type **20 mm** as shown in Figure 8. Click **OK** to finish the extruded base feature.

5. Choose the **Save** button from the Standard toolbar to save the model.

6. Close the document by choosing **File> Close** from the menu bar.