DEPARTMENT OF MECHANICAL AND INDUSTRIAL ENGINEERING
NORTHEASTERN UNIVERSITY

CAPSULE PROGRAM
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Tutorial 01
Getting Started

SolidWorks 2010
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Getting Started

In this tutorial you start learning 3D modeling by creating a 3D Model of a Chamfered Block.

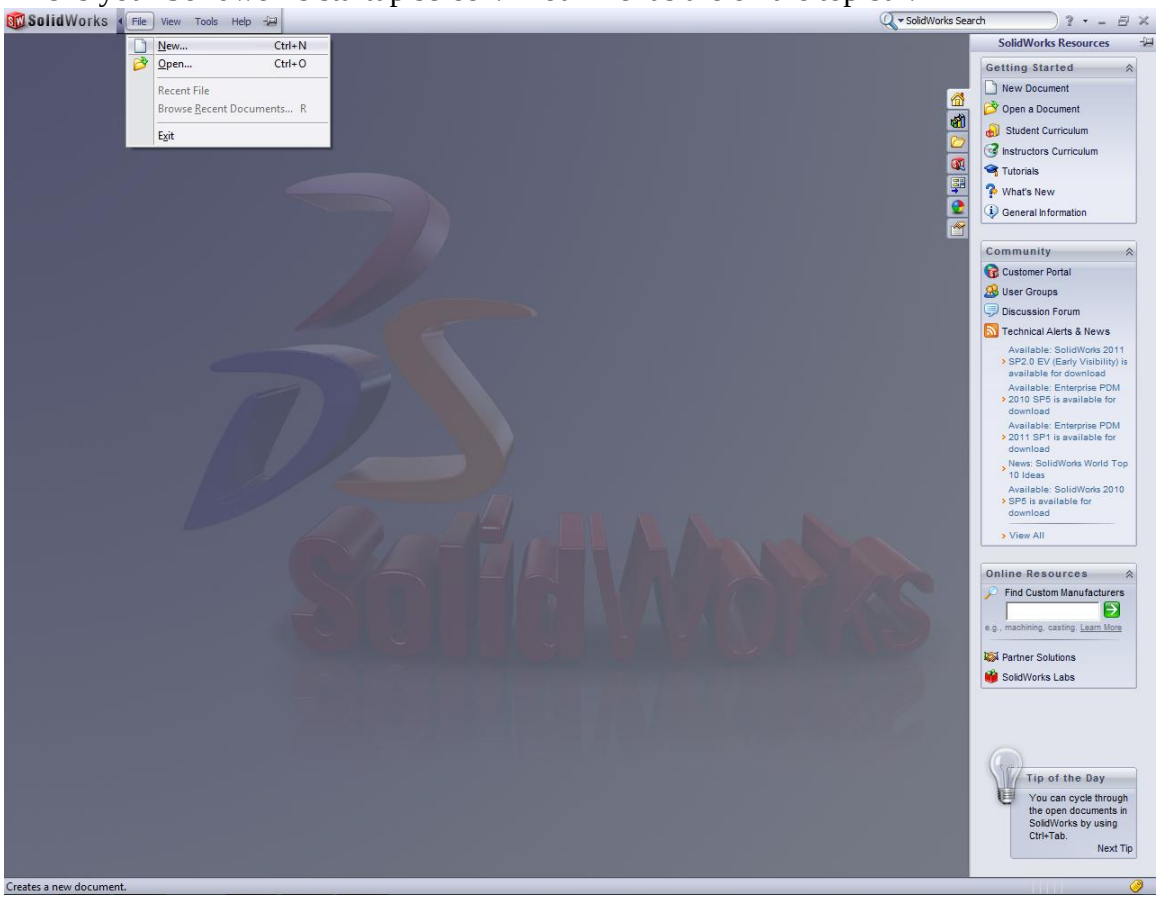
But first, what is Solidworks? Solidworks is a 3D modeling program where you can create 3 dimensional objects on your computer. As if you had the object in your hand, you can look at it from all angles, zoom in and look at the details. The primary use for Solidworks and programs similar is to reduce manufacturing costs. Editing a part on a computer isn't as expensive as making 3,4, or 10 versions of the same part to try and get it right. With Solidworks you can modify the dimensions, you can modify the shape and look of your part as well as change the material based on what you need.

Solidworks Interface

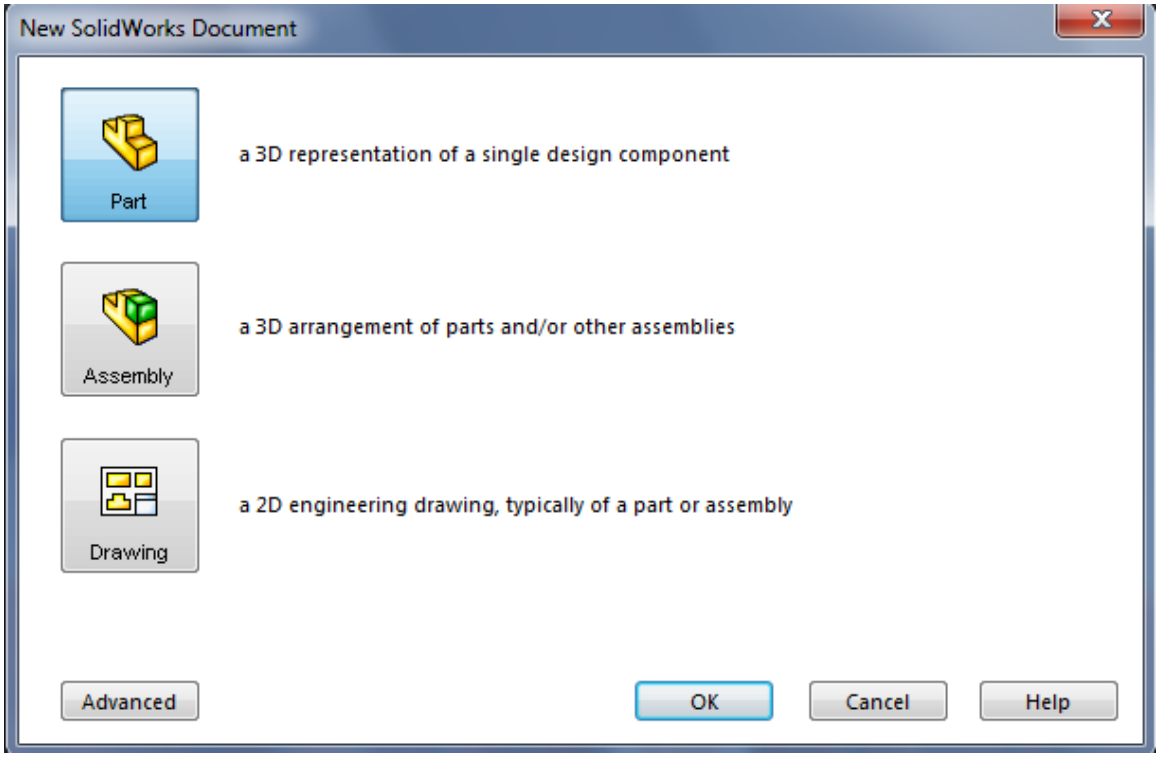
Tutorial 1: Drawing a circle and extruding it to a cylinder

Step 1: Creating a new part file

This is your Solidworks startup screen. Your menus are on the top bar.



To open a new part, go to **File → New**. This screen will appear on your monitor.



In Solidworks, you have three options of file types. The first file type is a **part**. This type will allow you to design the individual components to your final part. For example, with a Dasani water bottle, you have two parts – the bottle and the cap. In Solidworks, you would design the bottle as a part and the bottle cap as a part. So, in the end, you have **two** part files.

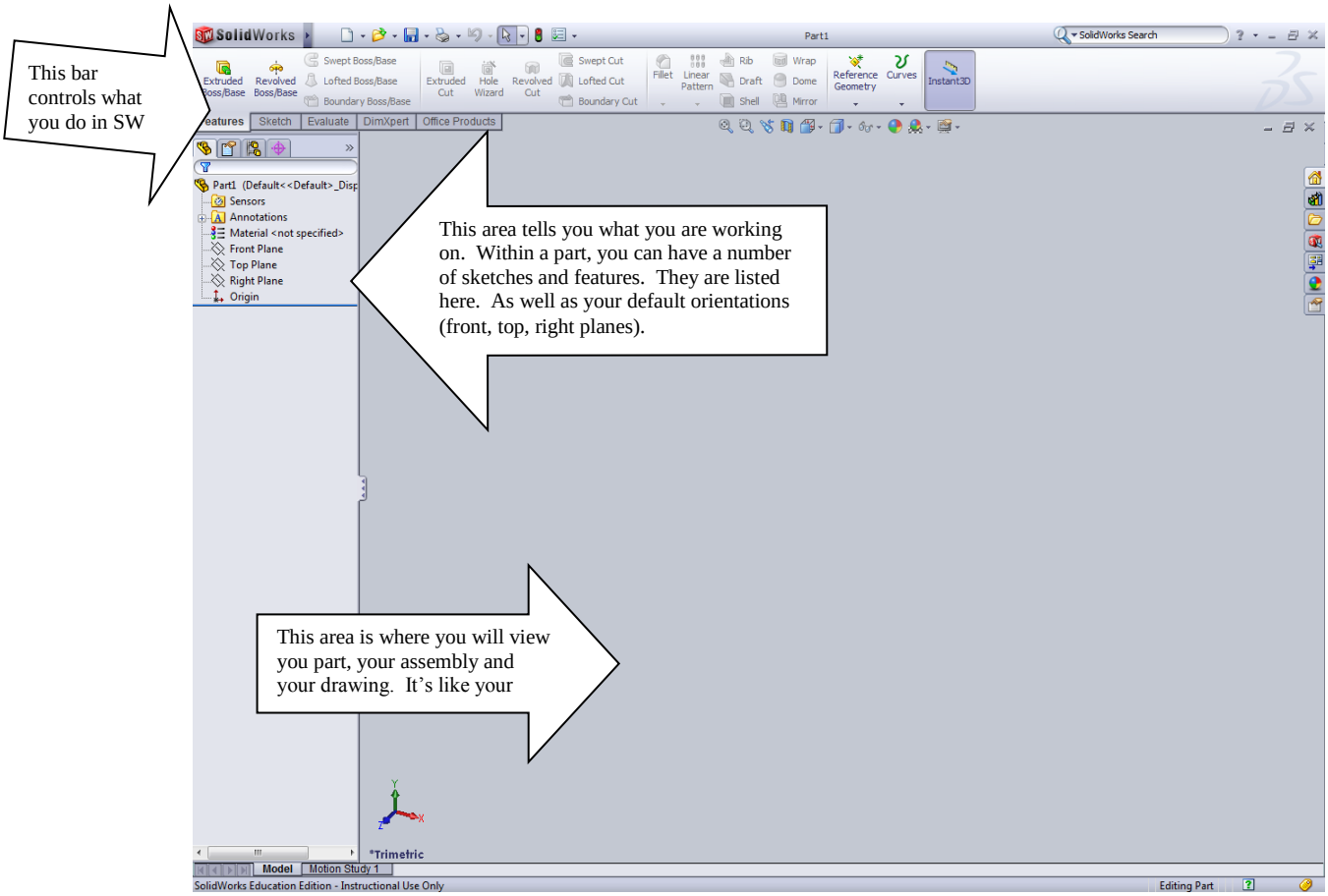
An **assembly** file type allows you to create your final product. So, returning back to the water bottle example… an assembly file type will allow you to put the bottle part and the cap part together in one file, to create product. At the end of an **assembly** you will have one file with two parts.

A **drawing** file is a blueprint of the assembly **or** part that you created. It's primarily used to clearly see dimensions from all sides of your product (i.e. front, right, top and isometric views). A drawing file allows a manufacturer to clearly see what you created on a 2D plane.

Now, select **part** and **select OK.**

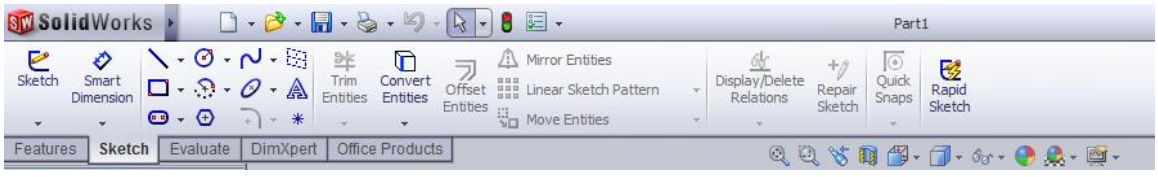
You will get the following view of Solidworks.

Step 2: Solidworks 2010 front interface

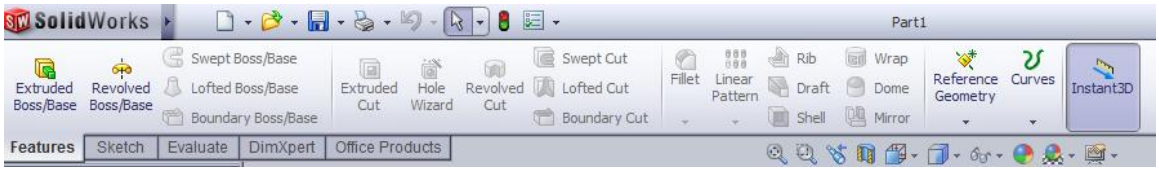


There are two primary tabs that you will be working with in Solidworks. The first tab being the "**Sketch**" tab. The **Sketch** tab allows you to draw in Solidworks. For example, if someone asked you to draw a circle on a piece of paper, that's really easy. Same with Solidworks – except you are drawing the circle on the computer (BUT still in a 2D plane… like a piece of paper). That is all the sketch tab is. It allows you to draw in SW. You will notice in the tab below you have circles, lines, rectangles, etc. You can draw any shape, any size.

You always need to sketch on a plane. Those can be found on your left hand side under the **Feature Manager Tree**. Do you see above where it states: Front Plane, Top Plane, Right Plane. Any one of these are a sketchable plane.

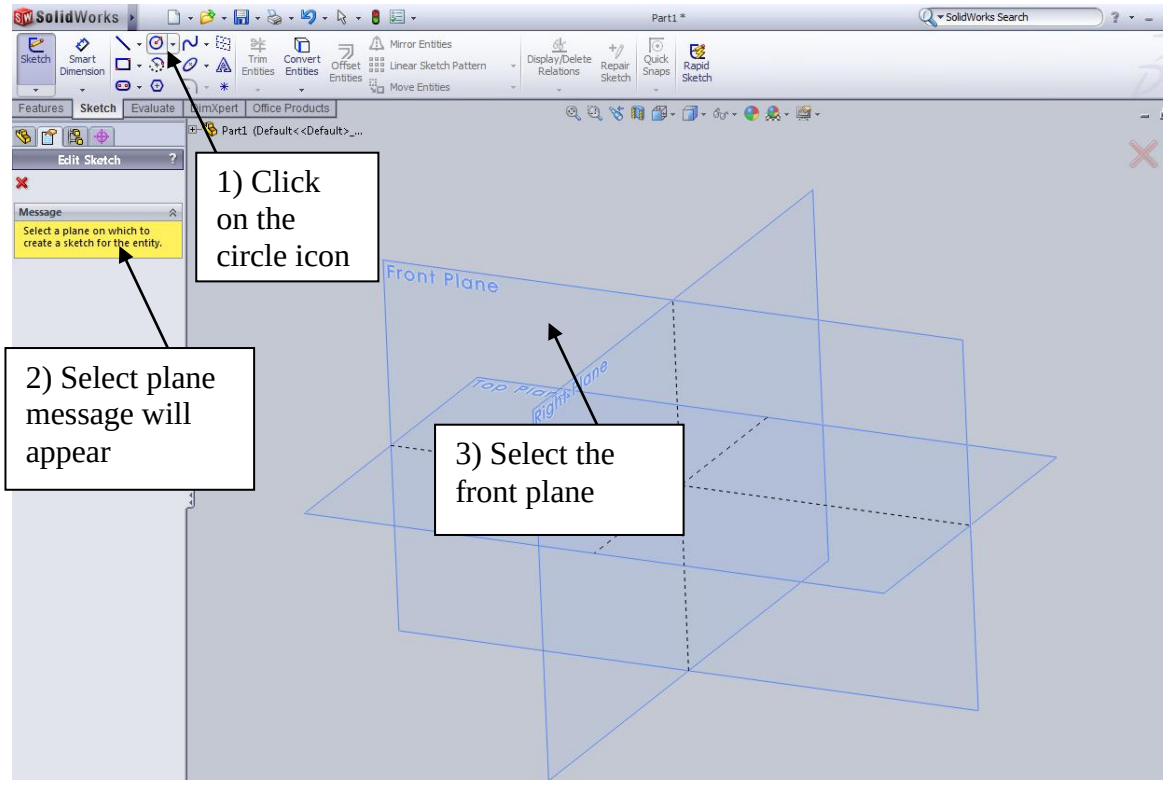


Now, take that circle you just drew on a piece of paper and make it a cylinder… You can't, can you? There is no way to make the circle you drew on your paper a 3D object. In SolidWorks, you CAN. You can make a circle that you draw a 3D object. That is what the **Features** tab is used for. The **Features** tab is used to make a 2D object 3D. Using this tab, you can also make tubes, revolved objects (like a bottle) as well as objects with holes.

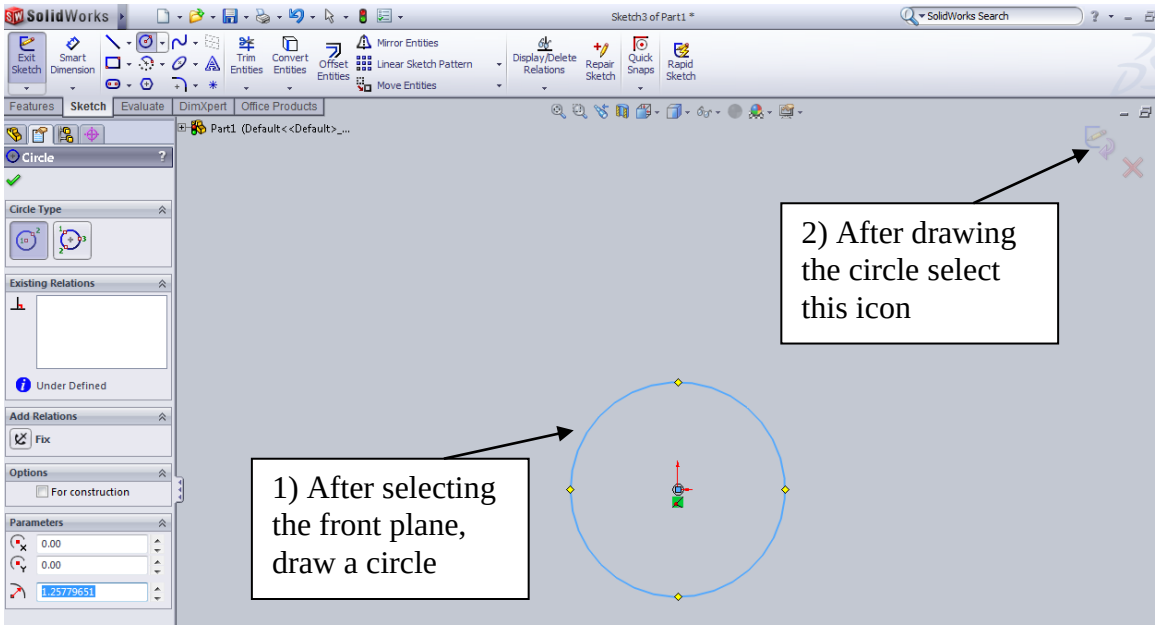


Step 3: Drawing a circle

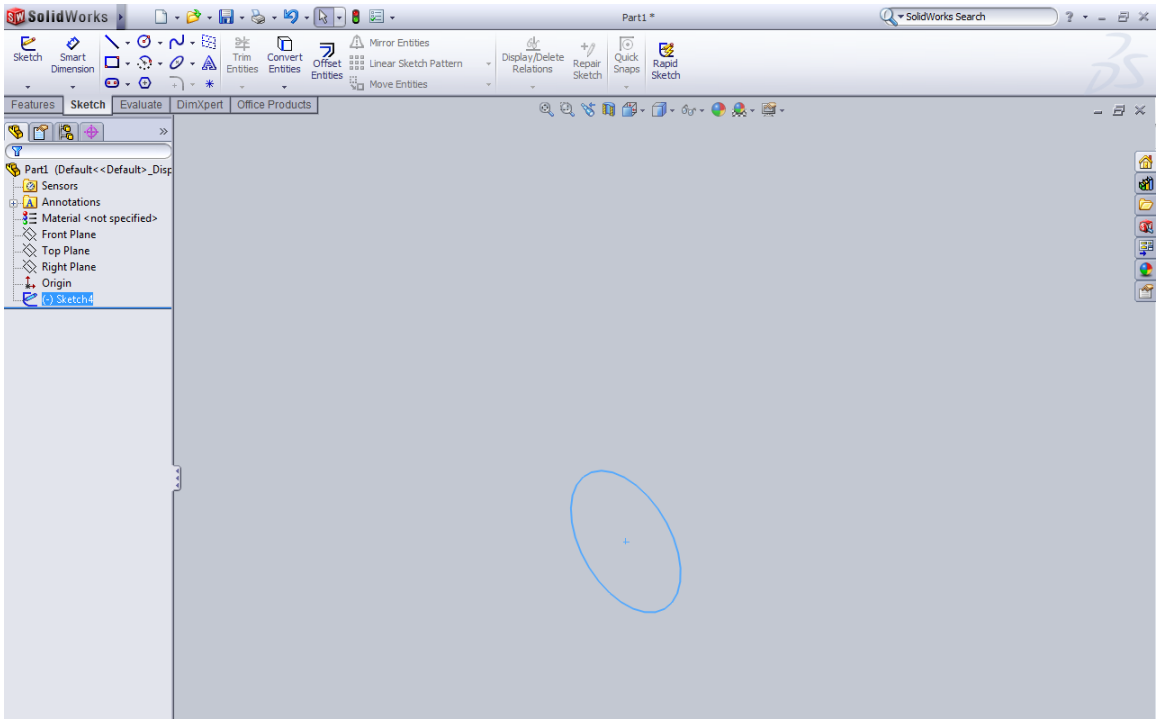
Here is an example of what was just described. First, I sketch a circle – a 2D object with no thickness.



When you click on the **circle** icon the Solidworks interface will prompt you to **select a plane**. For now select the **front plane**.



Once the **front plane** is selected, draw the circle and select the icon as shown above.

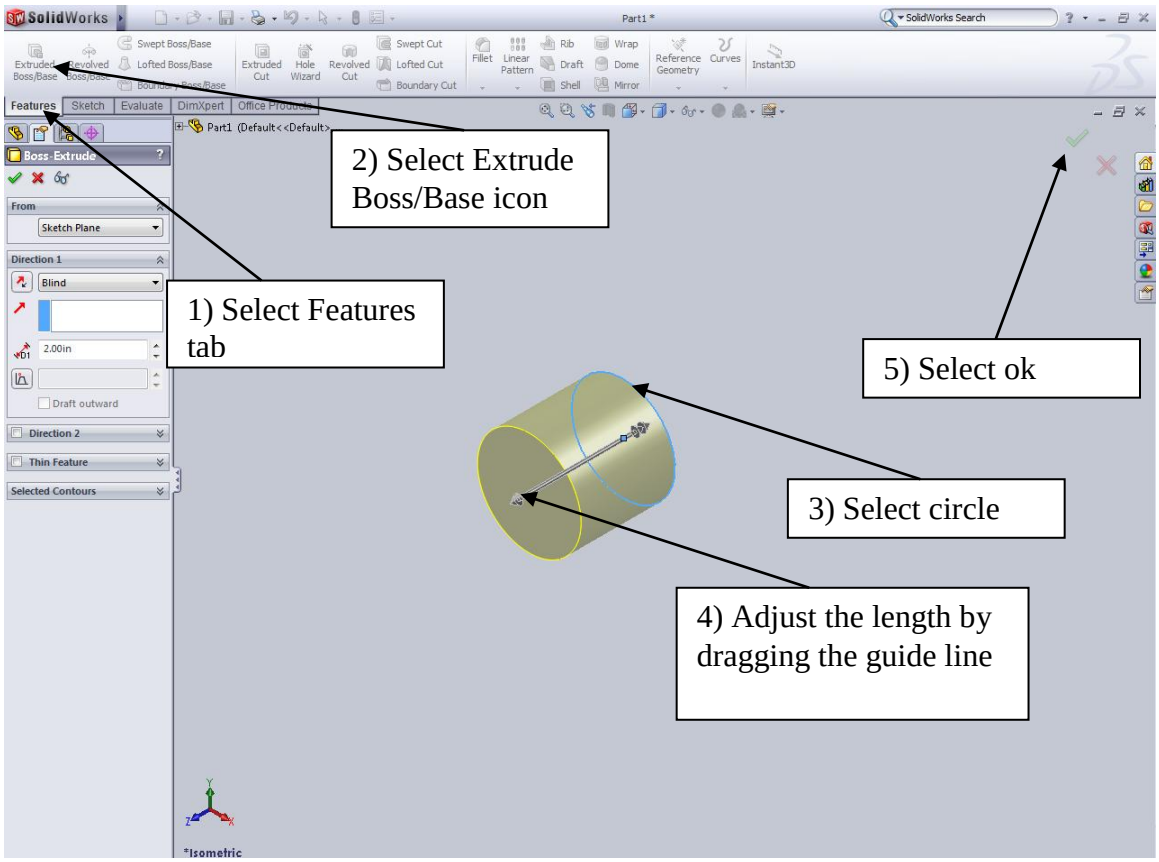


The drawing below is the same circle… the left view is looking at the circle from the front (it looks like what it would look like on a piece of paper right)? The right picture is the circle from an **isometric** view – simply an angled view of the object. You can see,

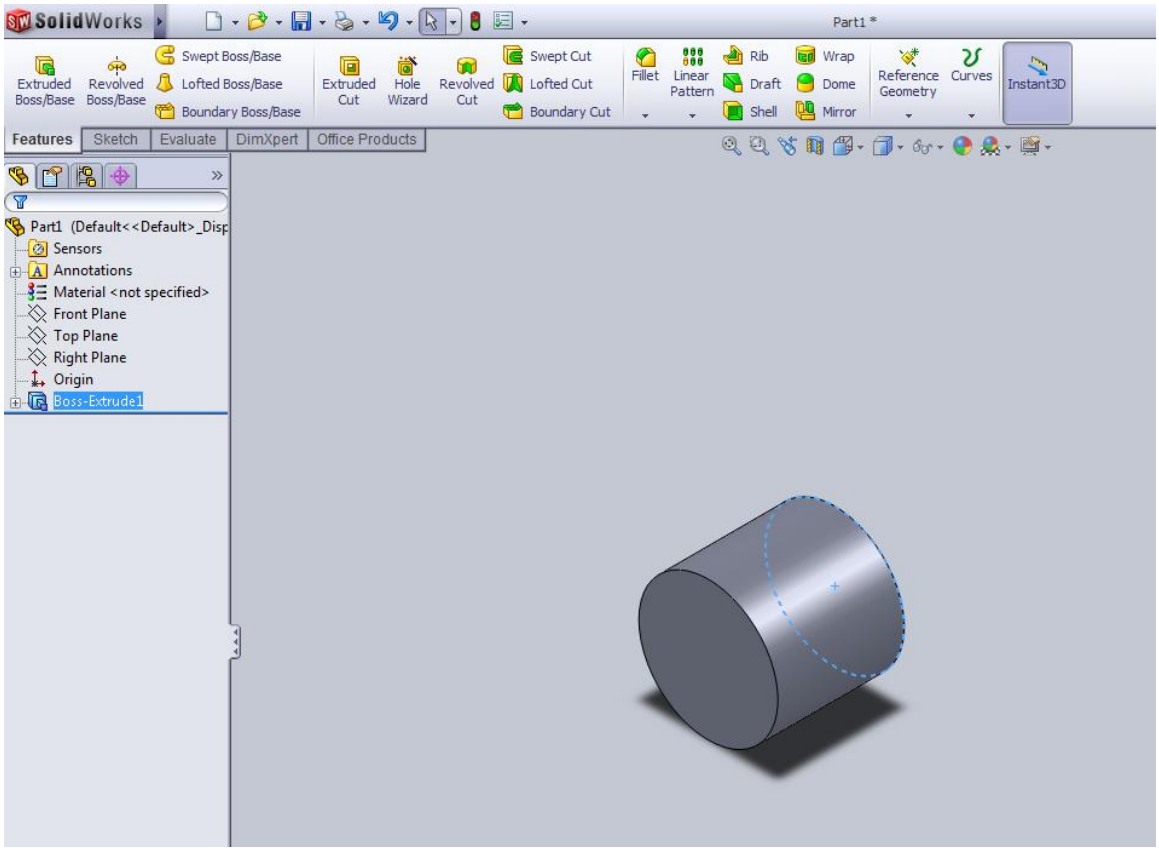
there is no thickness to the circle. Much like the circle on a piece of paper. Think about sketching as drawing the cross section of a part.

Step 4: Extruding a circle

Now, we are going to **extrude the circle.** This will give it 3 dimensions – also known as a height, a width and a depth. The picture on the left is the circle in the process of being **extruded**. You will be able to choose how far you (or how deep) you want your cylinder to be… but that will be discussed later. For me, my cylinder is 3" deep. The right screenshot is your final cylinder. So now, you have a 3D object, in SolidWorks.



Do you see how the sketch is the **cross section** of your cylinder? In the left picture above, your sketch is highlighted in **blue.** That is because you are going from **sketching** to adding a **feature**.



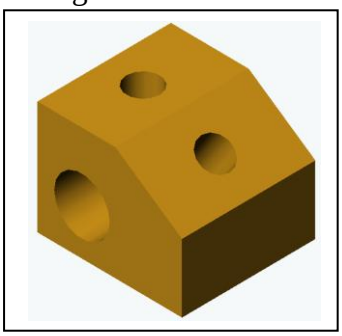
This will be the end result after the extrude operation.

General planning Strategy

In this tutorial you will learn about general planning and strategy that accompanies each step thought the solid modeling process.

Tutorial 2

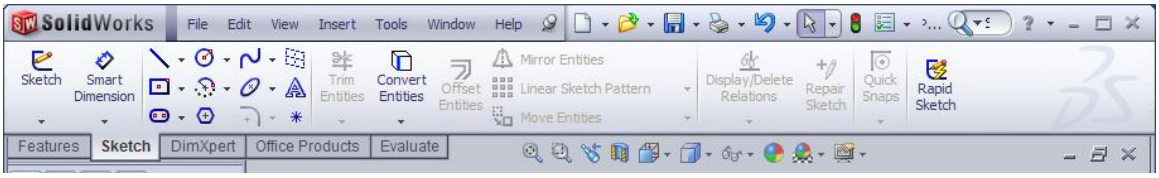
The figure shown below is the image of the model that we are going to create in this tutorial.

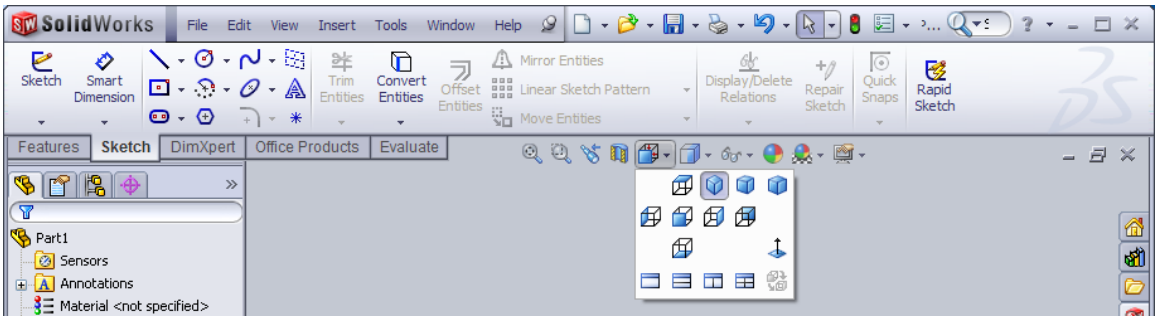


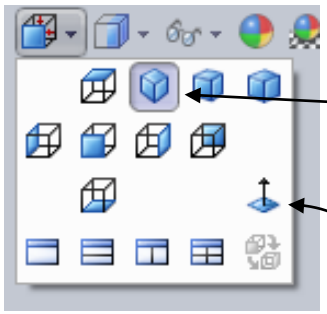
We shall follow the following general steps in order to create it:

1. Sketch and extrude the profile of the block.
2. Sketch and cut extrude the circle on the side face to generate the horizontal hole.
3. Sketch the circle on the inclined plane by defining an inclined WCS i.e. inclined sketch plane and cut extrude it to generate the inclined hole.
4. Sketch and cut extrude the circle on top face to generate the vertical hole.

The toolbars used in this exercise (note the icons):



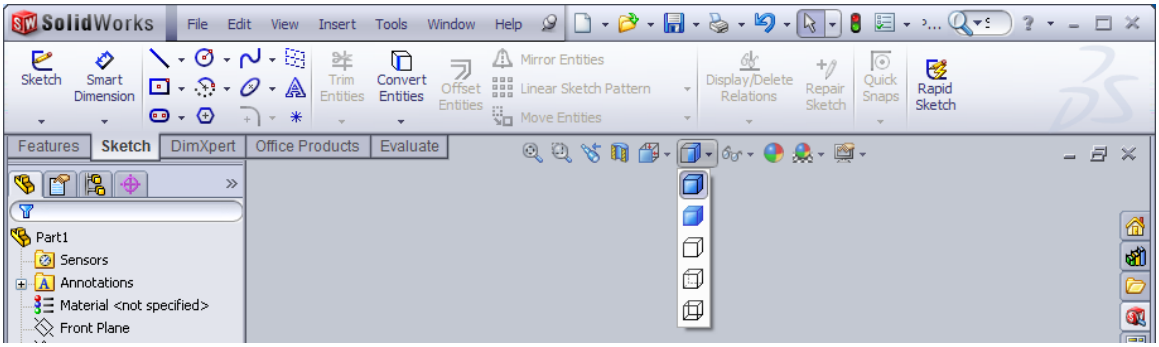




Your views from left to right, top to bottom

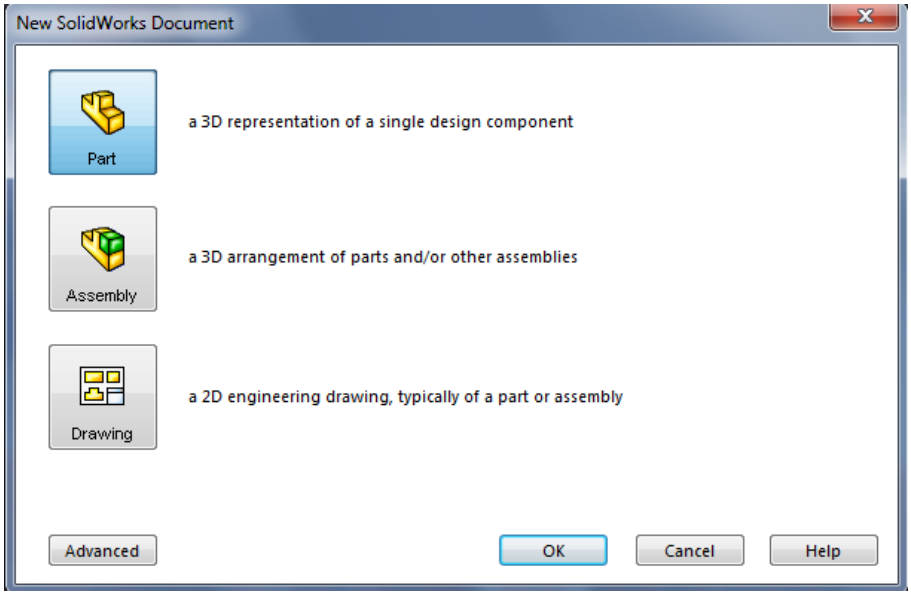
1. Top
2. Isometric
3. Trimetric
4. Dimetric
5. Left
6. Front
7. Right
8. Back
9. Bottom
10. Normal View (The icon with the upward arrow)

These are your different **display styles** on how your part will look on screen. The first one is **shaded with edges** which is primarily what you'll use. Play around with the display styles and see how the look of your part will change. It won't change the actual part, just its appearance.



Step 1: Creating a new part file

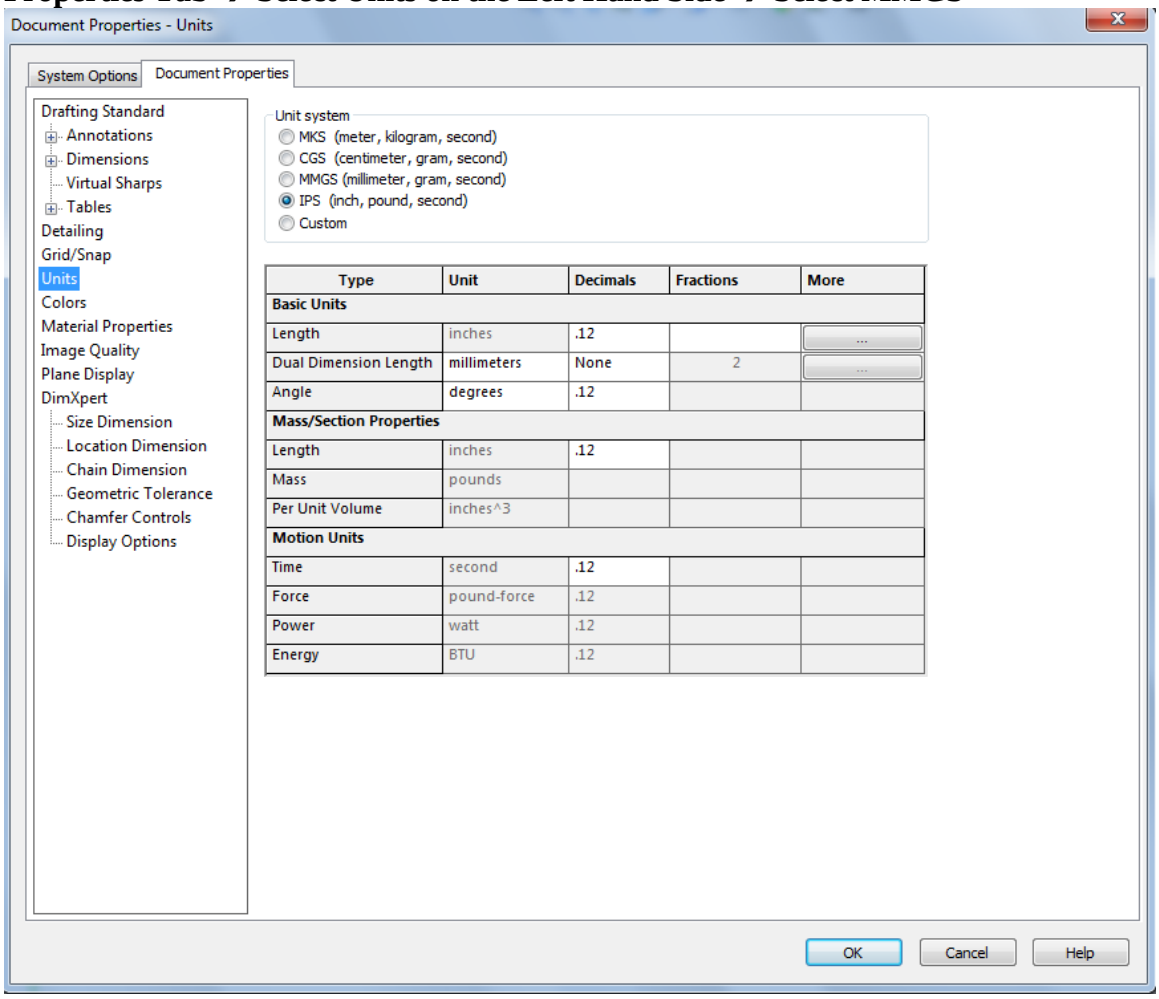
Start Solid Works. In the main window click on **File** then click on **New**, select **Part** and click **OK**.



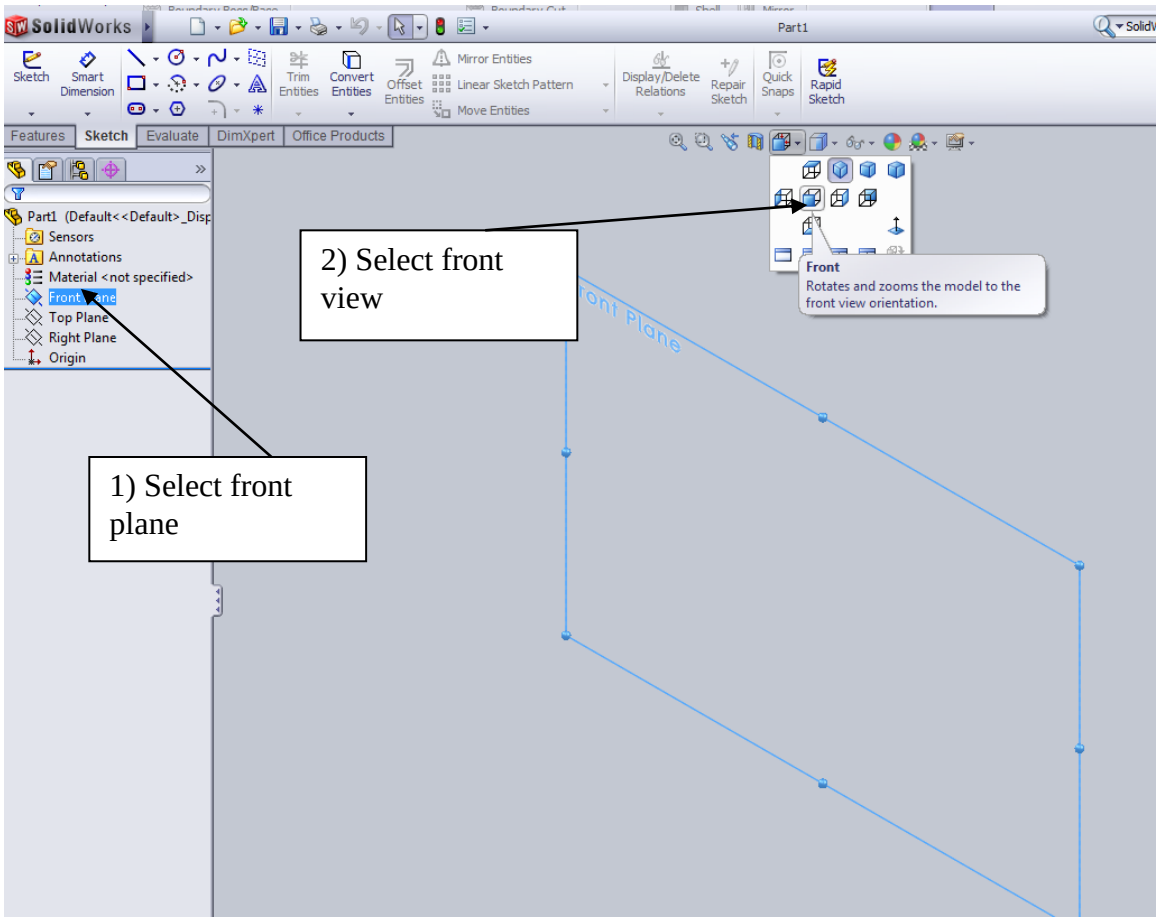
Step 2: Extruding the block

Step 2.1: Setting up the Sketching plane

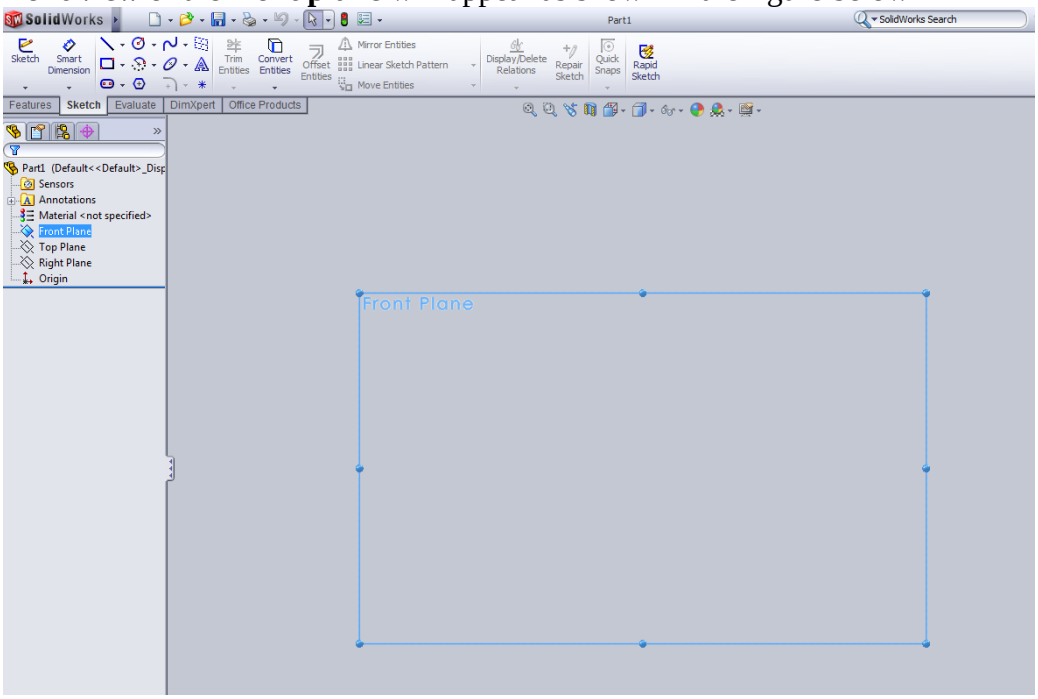
To change your units from inches to millimeters, to go: **Tools → Options → Document Properties Tab → Select Units on the Left Hand Side → Select MMGS**



Select **Front** from the **Feature Manager Tree.** Click on the **Sketch** icon from the **Sketch** toolbar.

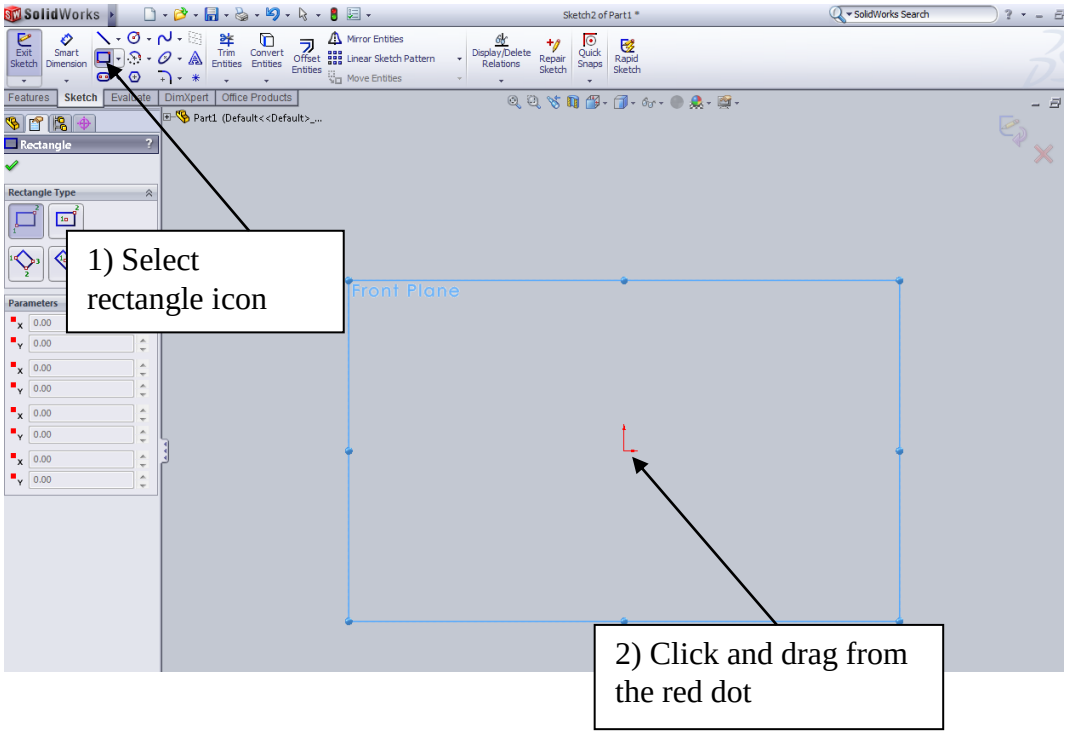


The **front view** of the **front plane** will appear as shown in the figure below

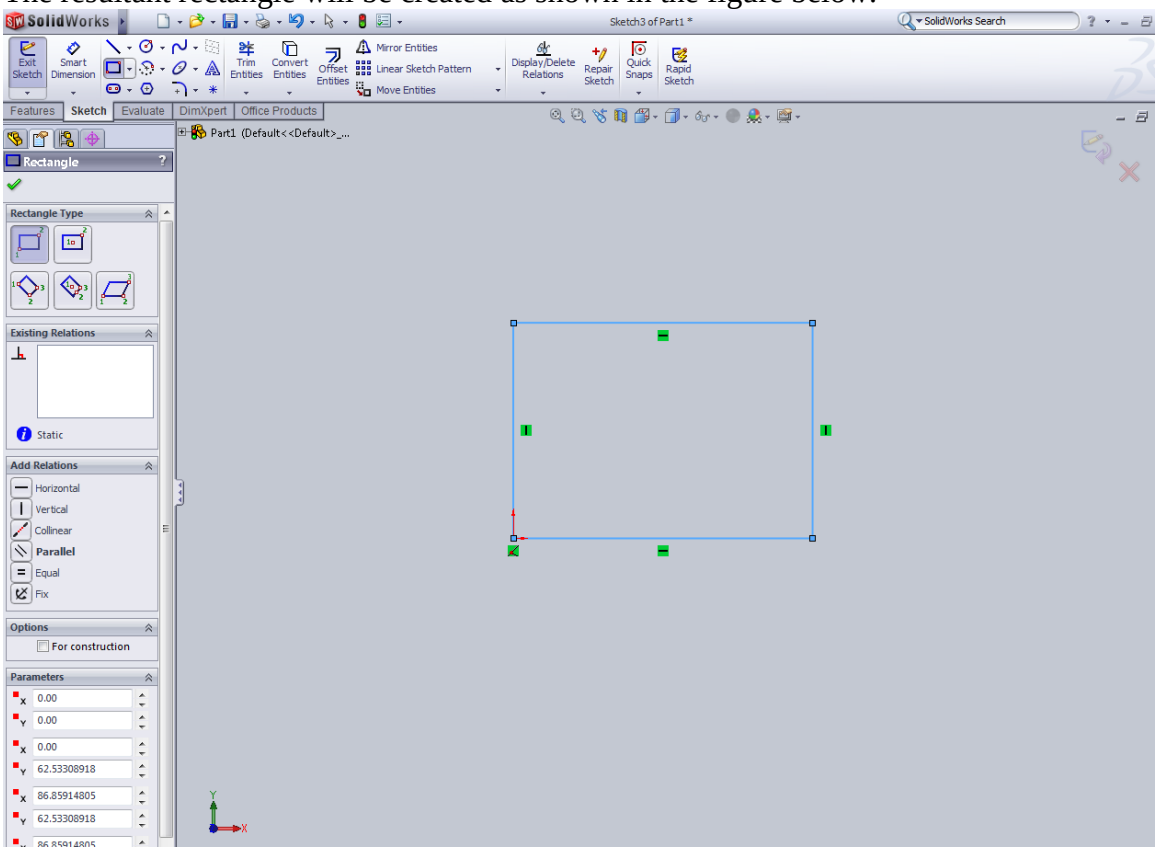


Step 2.2: Sketching the profile

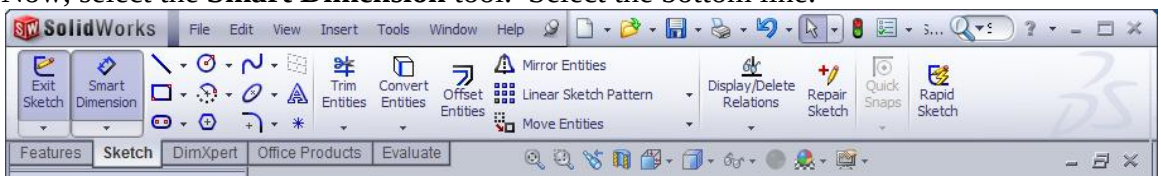
Draw a rectangle beginning from the origin (the red dot). Click and drag to the right and up and create an arbitrarily sized rectangle.



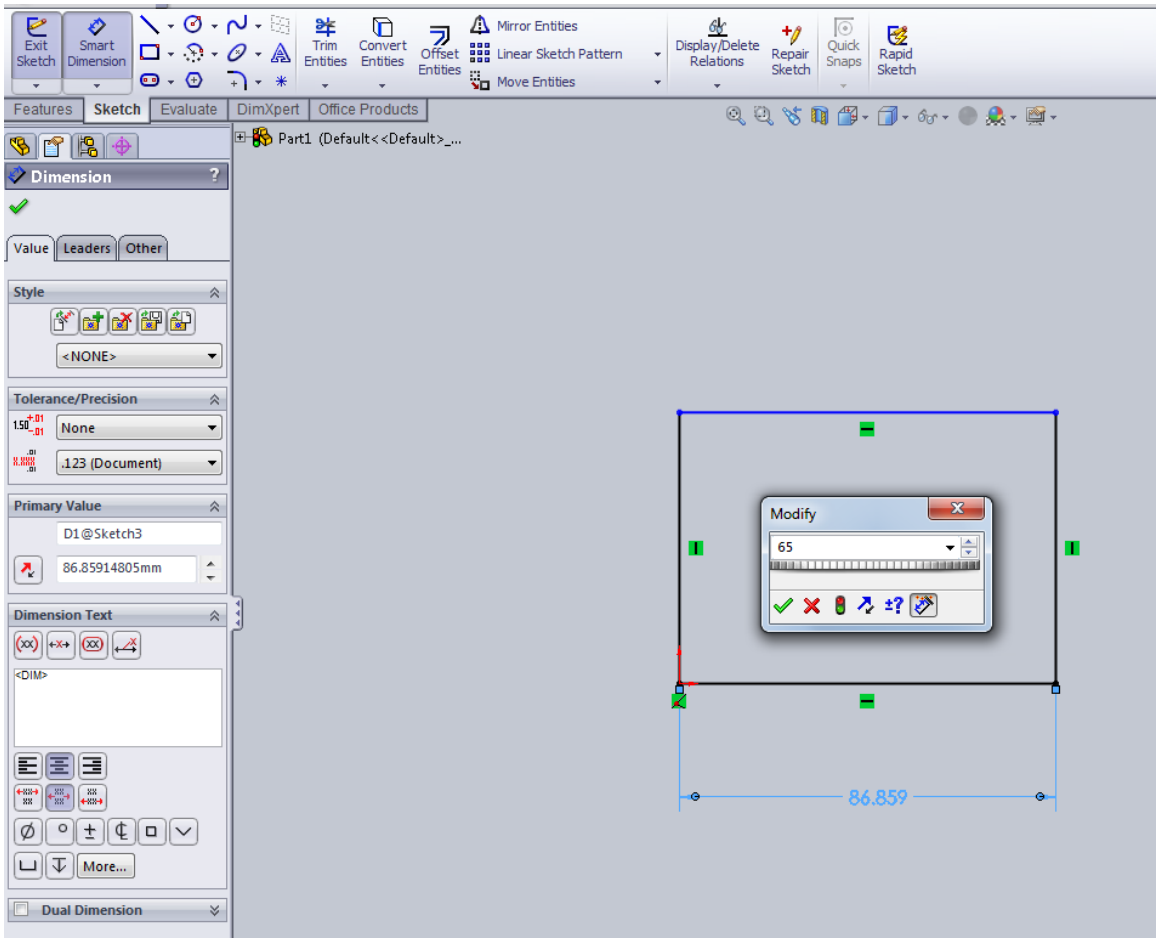
The resultant rectangle will be created as shown in the figure below.



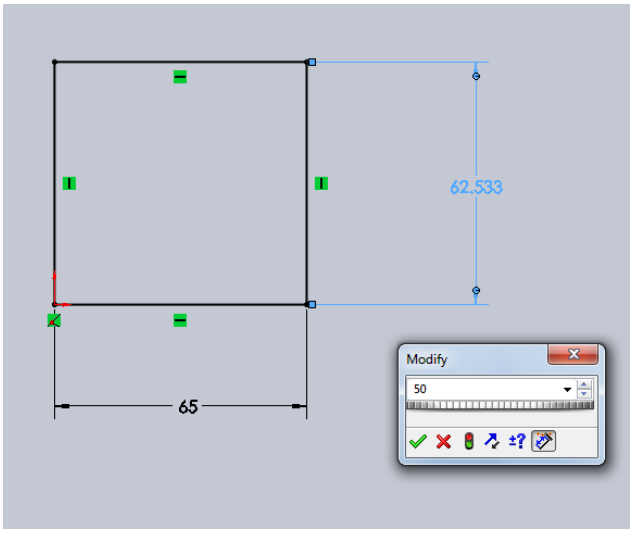
Now, select the **Smart Dimension** tool. Select the bottom line.



A box will pop up and ask you to type in a value of 65mm. Click the green check box.

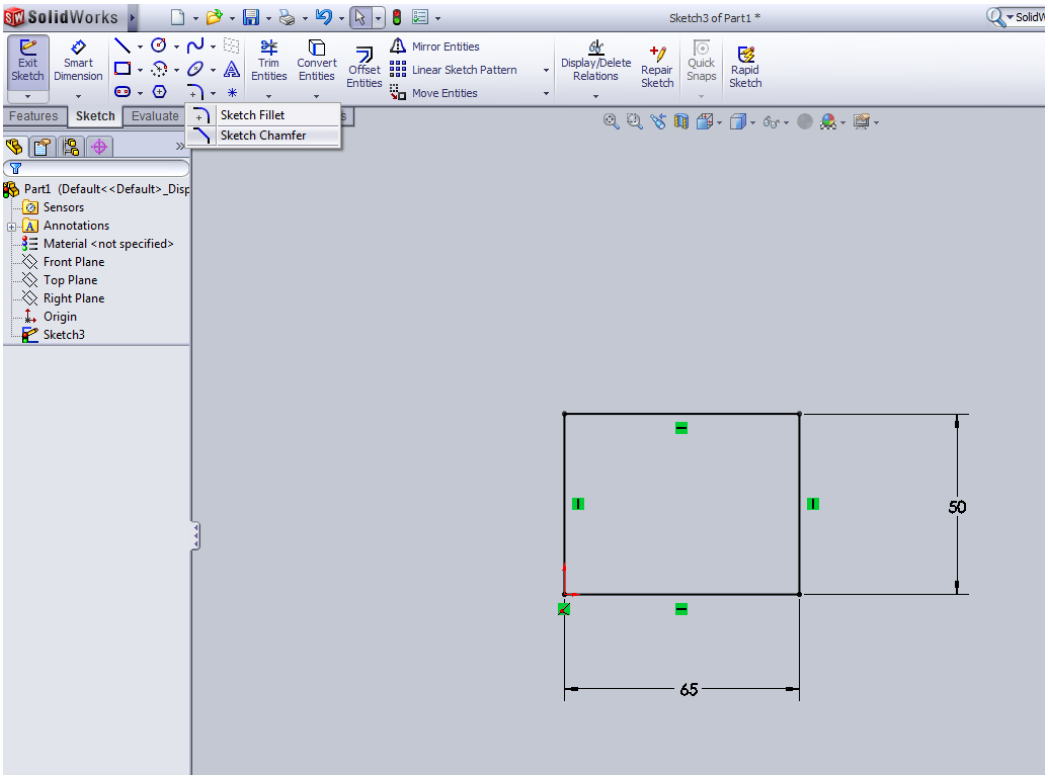


Do the same thing for the vertical side except type in 50 mm.

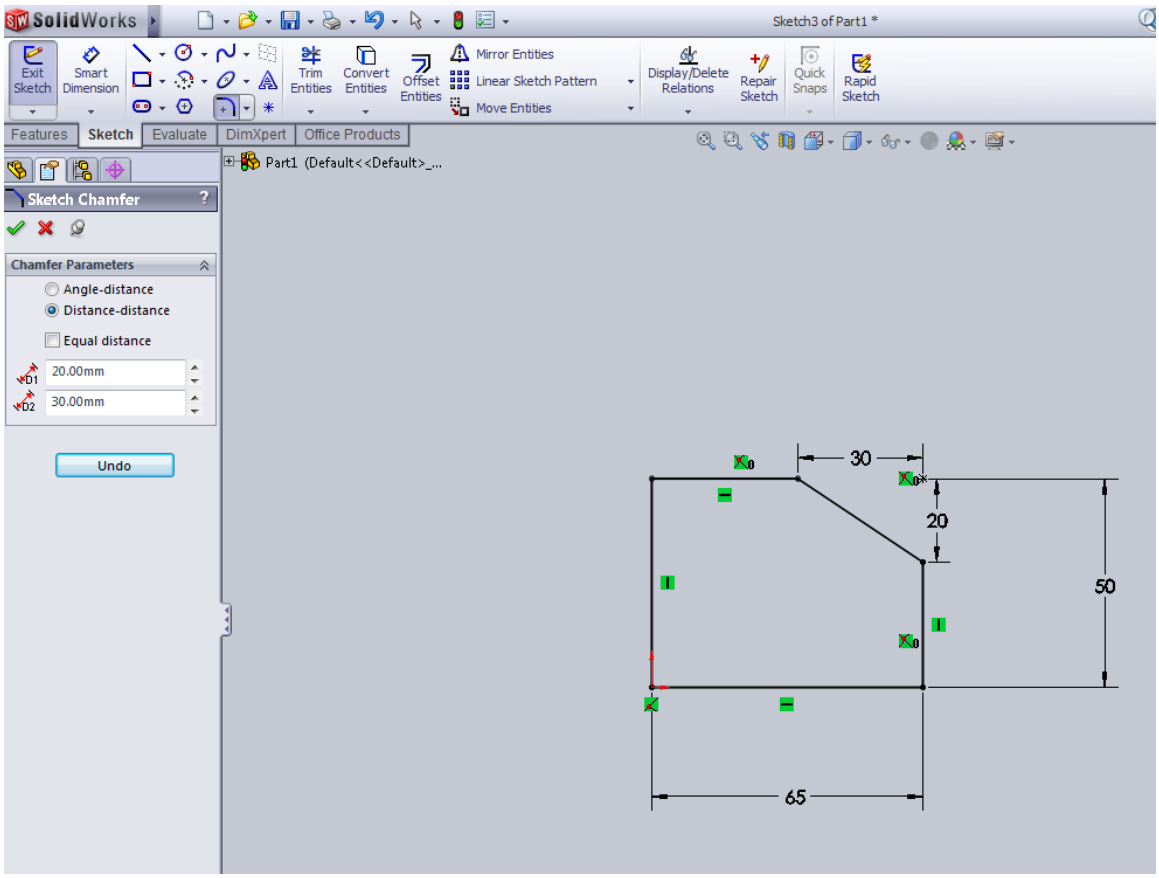


Step 2.3: Chamfering the top right corner

Select the **Sketch Chamfer** icon from the **Sketch Tools** toolbar. The **Sketch Chamfer** menu appears in place of the **Feature Manager Tree.**



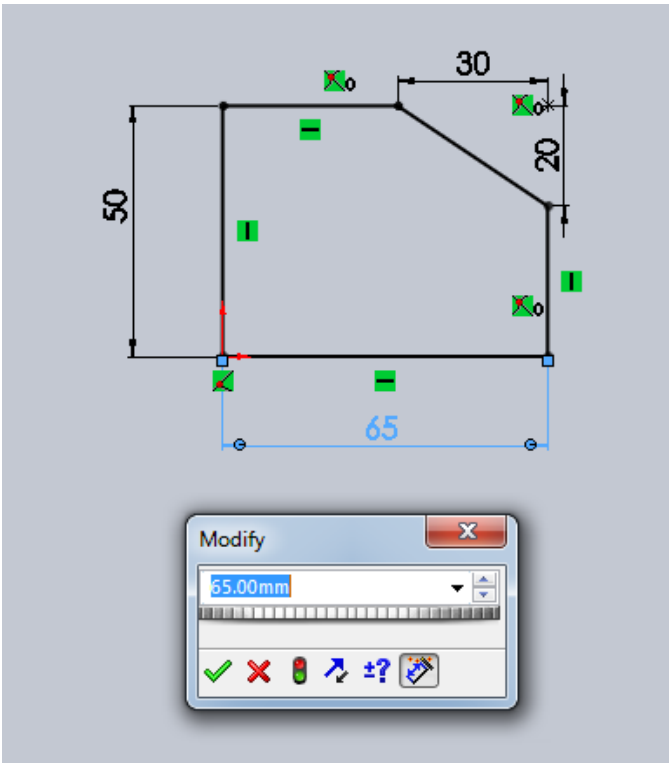
You will see a box appear on the left hand side. Select the upper right most corner. Toggle **Distance-Distance** option on and toggle the **Equal distance** option off. Enter the first distance as 20mm and second distance as 30mm and now select the vertical line and the horizontal line respectively and press **Enter**.



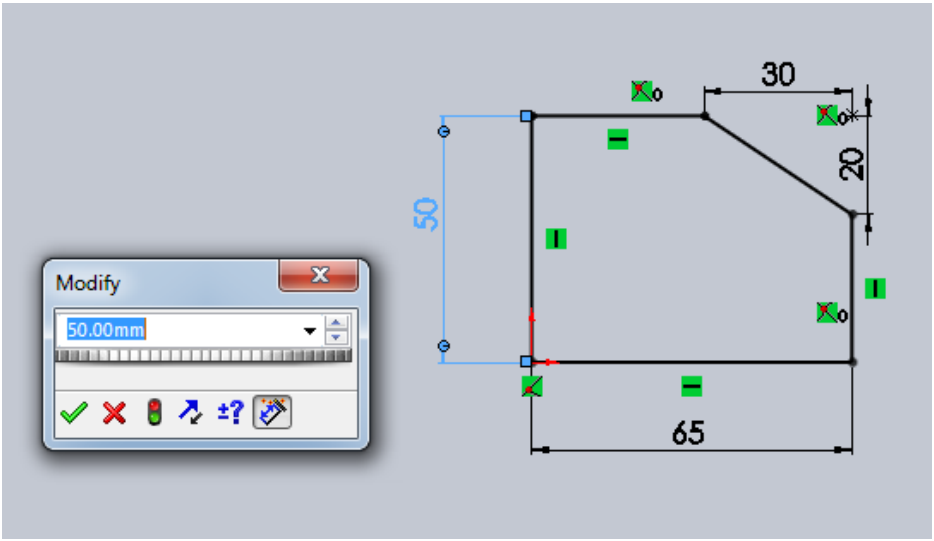
Note: The dimensions 20mm and 30mm affect the order of line selection. The first dimension is given to the line we select first.

Step 2.4: Modifying the dimensions of the block

Click on the **Dimension** icon from the **Sketch** toolbar. Select the horizontal line and place the dimension at an appropriate location, enter 65 in the **Modify** window and hit **Enter**.



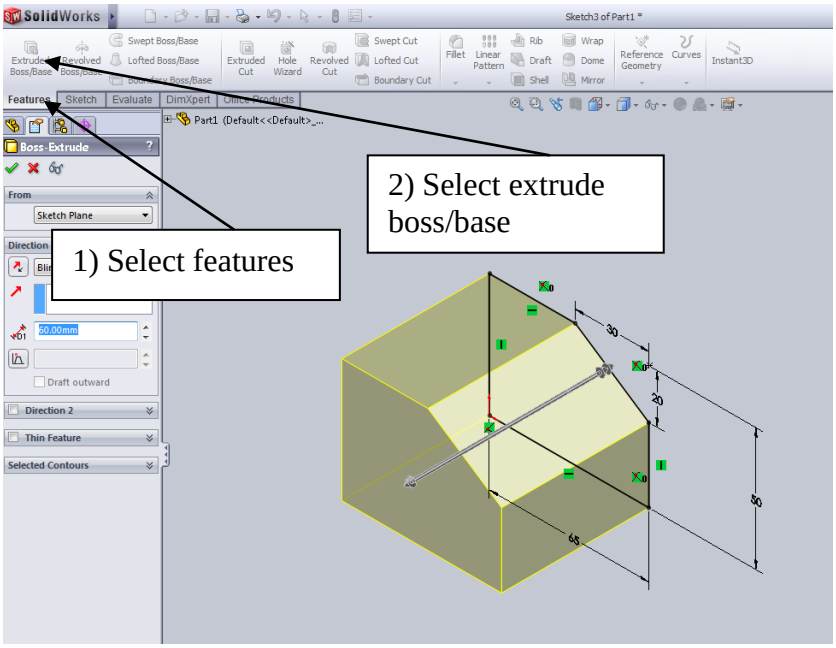
Similarly, modify the vertical dimension to an exact value of 50mm.



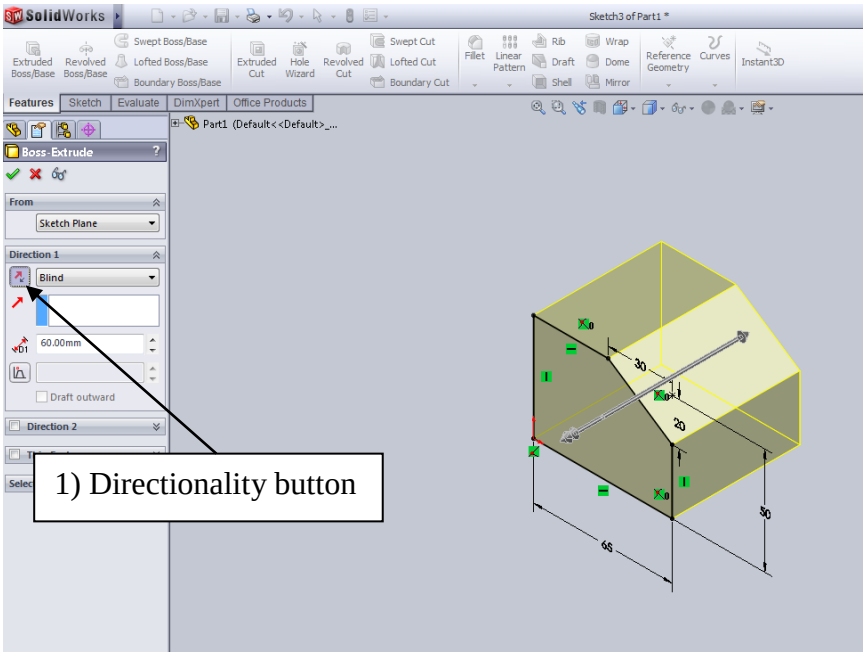
This completes the sketch of the profile of the block.

Step 2.5: Extruding the sketch to obtain the 3-D block

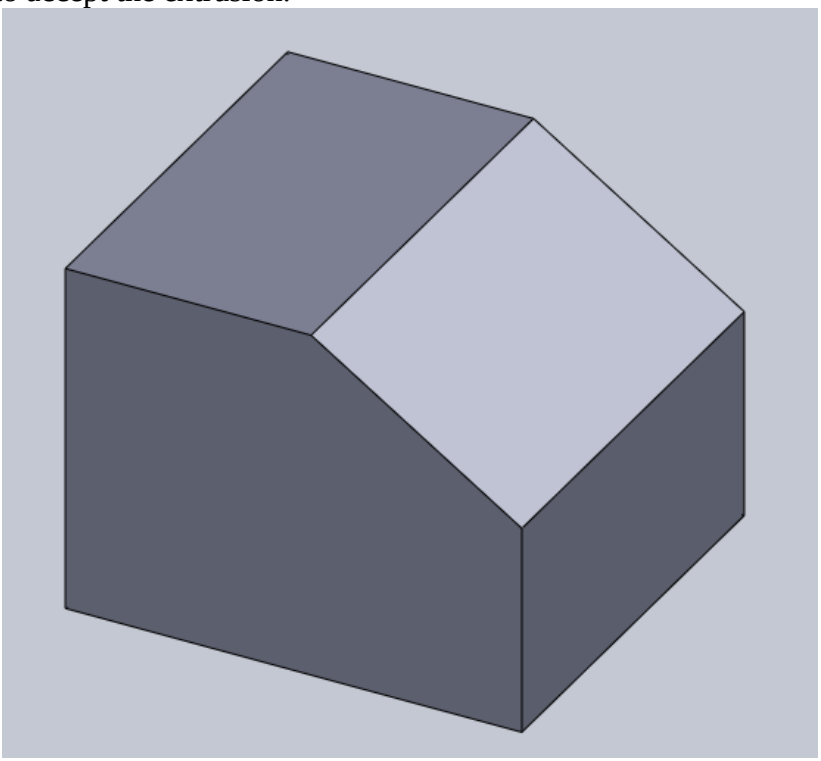
Click on the **Extruded Boss/Base** icon on the **Features** toolbar. The orientation of the sketch changes to isometric from orthographic.



Now select the directionality button next to “Blind” to change the direction of the arrow,



Press **Enter** to accept the extrusion.

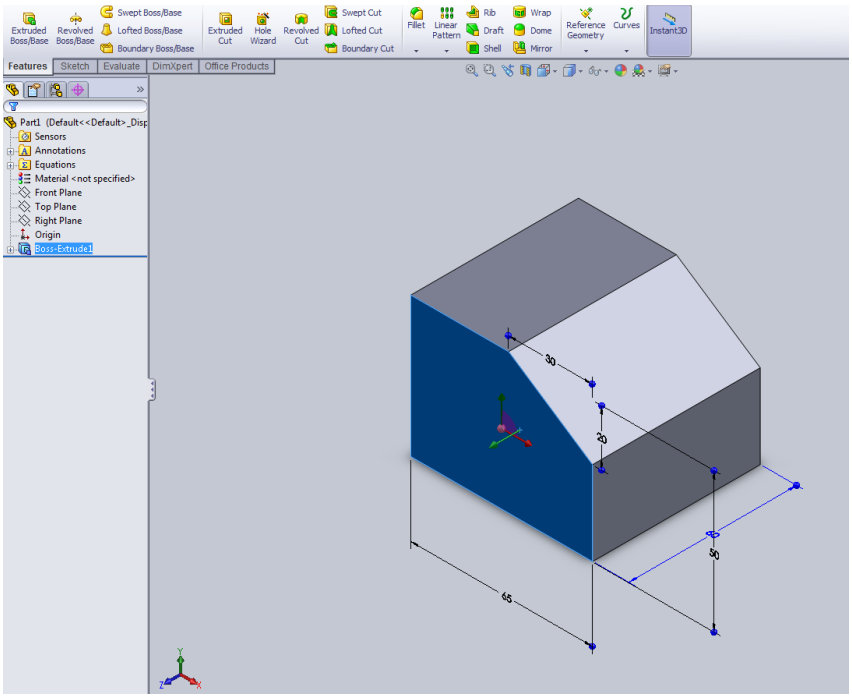


This completes the extrusion of the block.

Step 3: Creating the holes

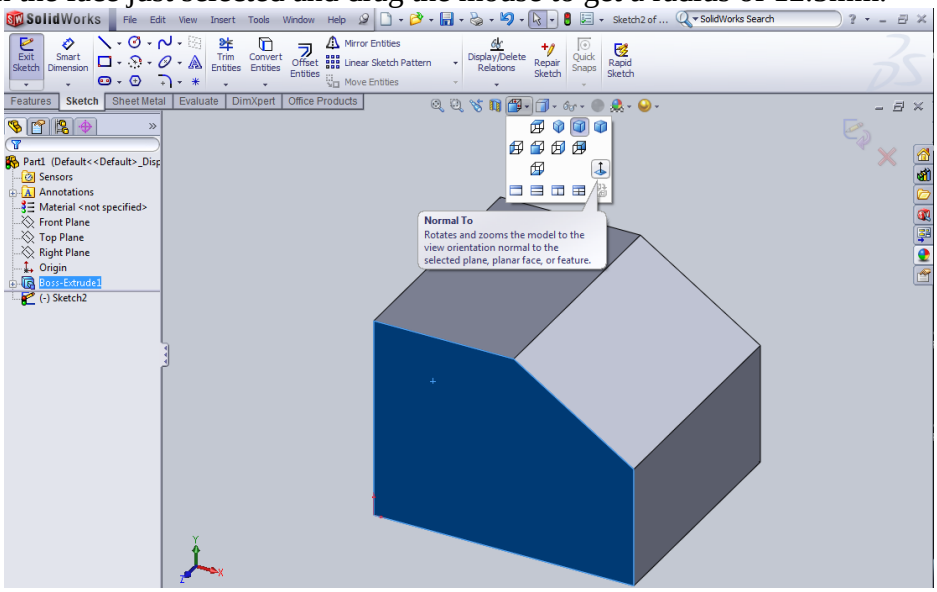
Step 3.1: Setting up the sketching plane

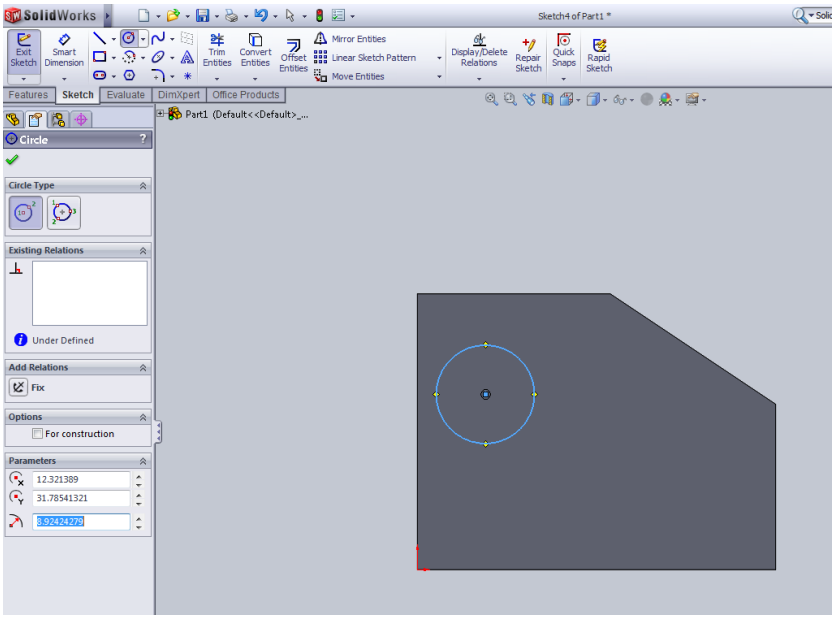
Select the face of the object as shown in the figure below.



Step 3.1: Sketching the circle

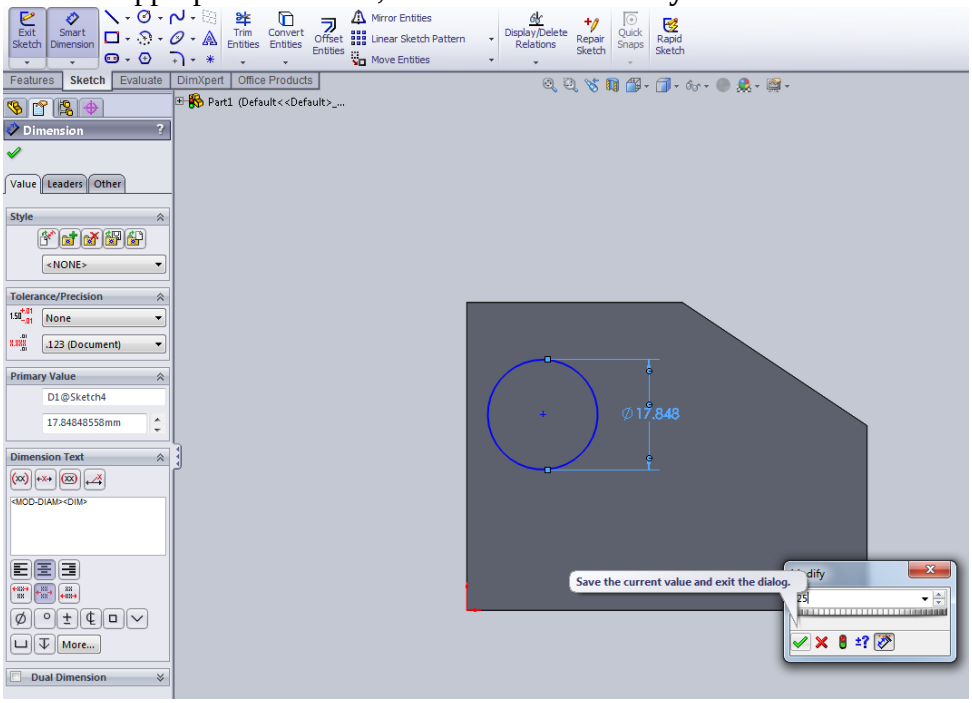
Click on the **Circle** icon from the **Sketch Tools** toolbar. Draw a circle by clicking for the center on the face just selected and drag the mouse to get a radius of 12.5mm.



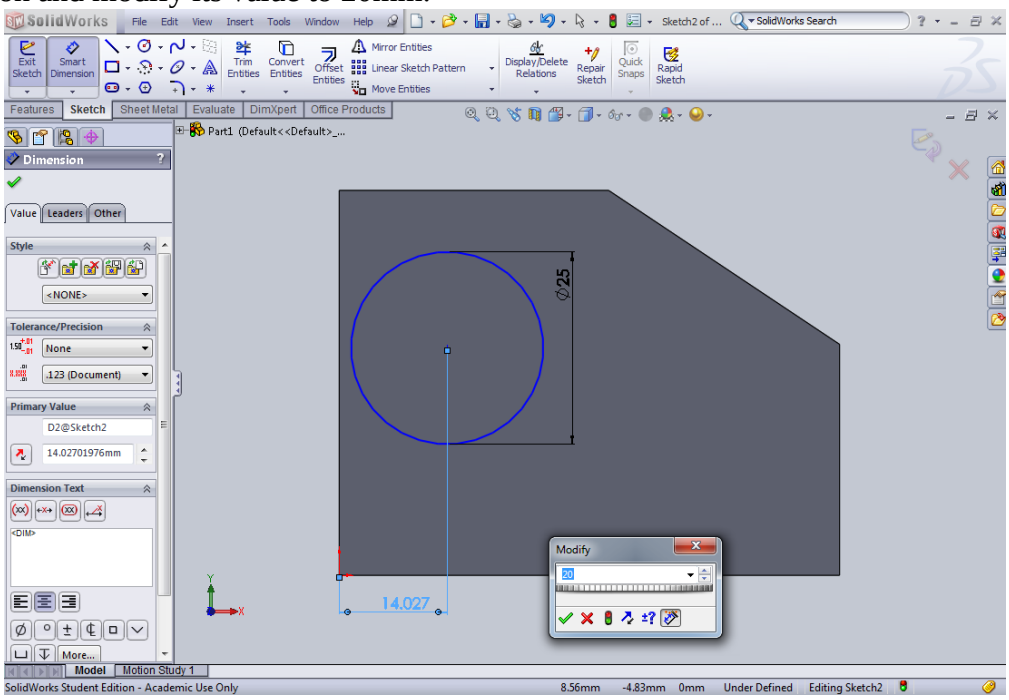


Step 3.3Modifying the dimensions of the circle

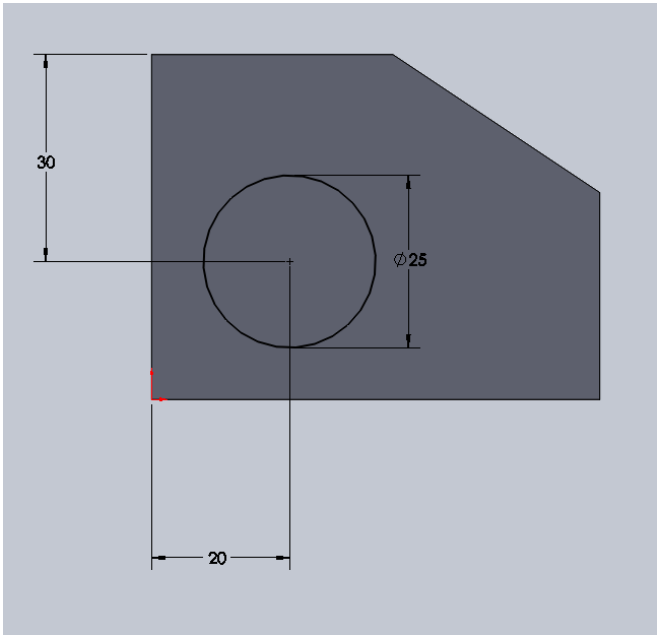
Click on the **Dimension** icon from the **Sketch** toolbar. Select the circle and place the dimension at an appropriate location, enter 25 in the **Modify** window and hit **Enter**.



Now click on the center of the circle and the origin and move the mouse so that the horizontal dimension between these two points is displayed. Place it at the appropriate location and modify its value to 20mm.

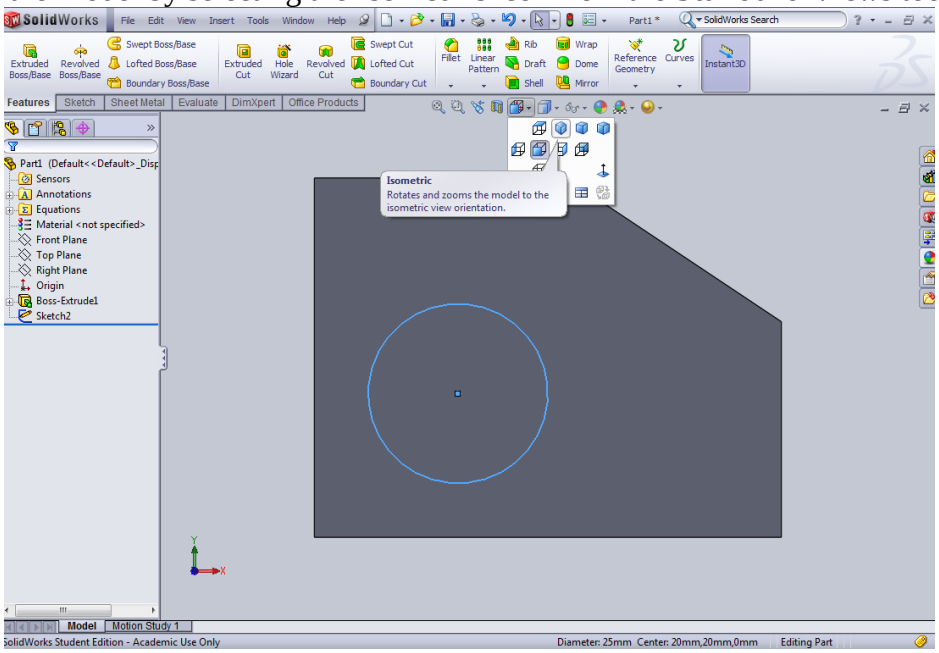


Similarly modify the vertical dimension to 30mm.

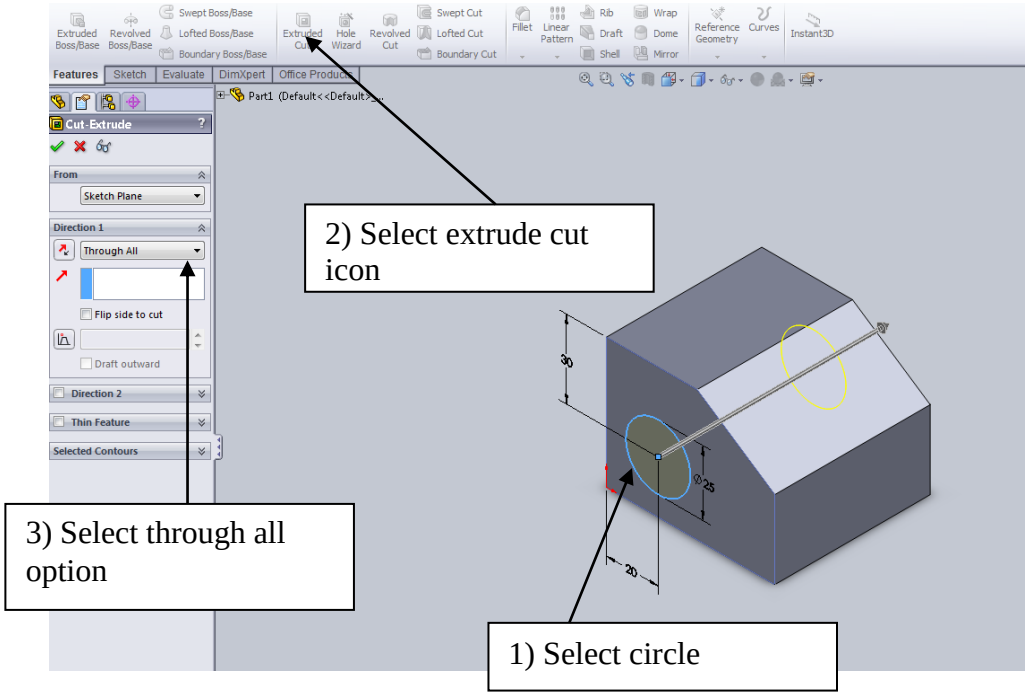


Click on the **Green Tick Mark** in the **Dimension Property Manager** to complete dimensioning.

Reorient the Model by selecting the **Isometric** icon from the **Standard Views** toolbar.



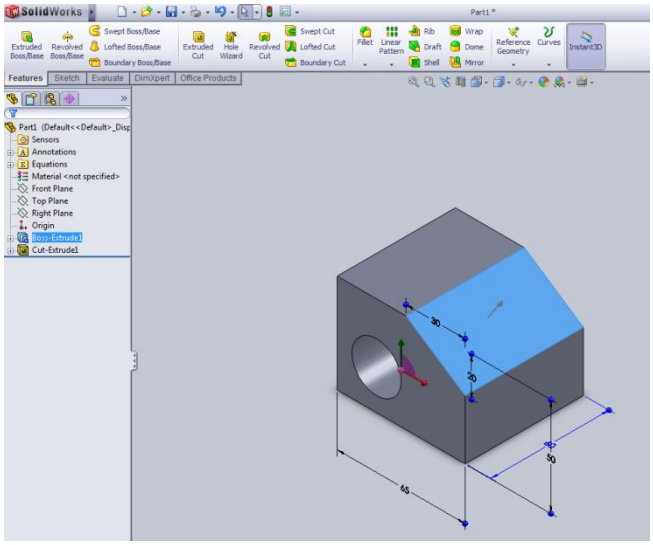
Step 3.4: Extruding the circle to create the hole:
Click on the **Extruded Cut** icon in the **Feature** toolbar. Set the **Direction** Field to **Through All**. By moving the mouse select the direction of extrusion so that the hole is created.



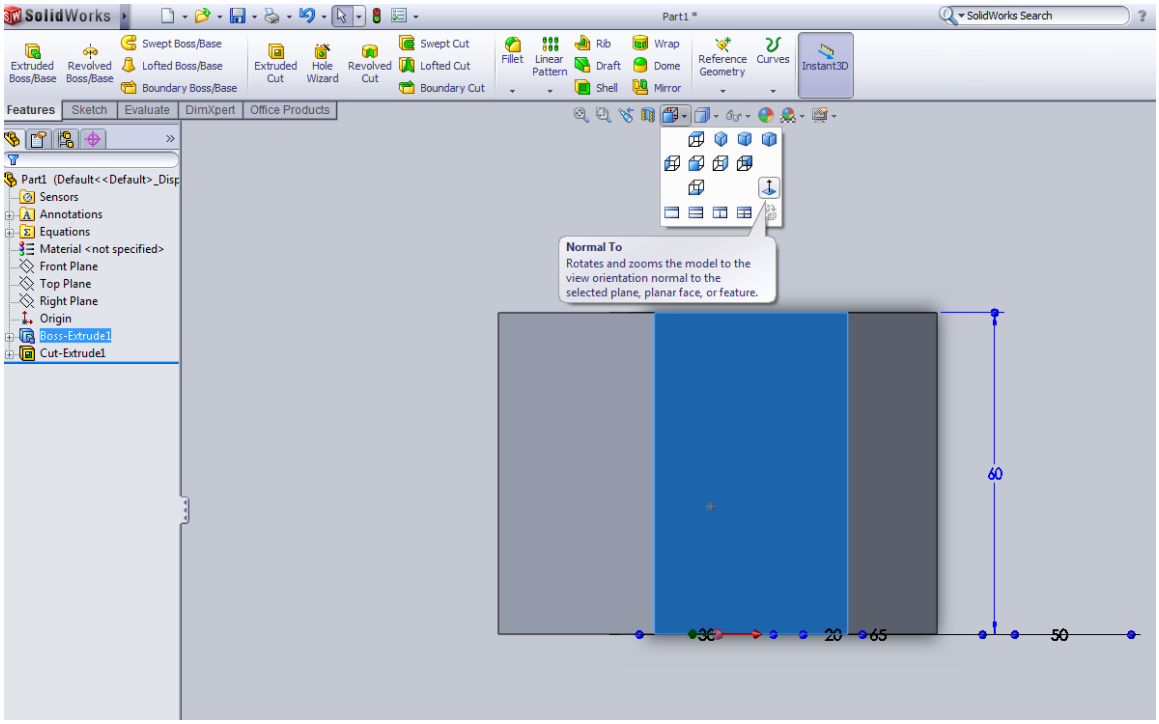
Hit **Enter** to finish extrusion cut.

Step 4: Creating the inclined hole

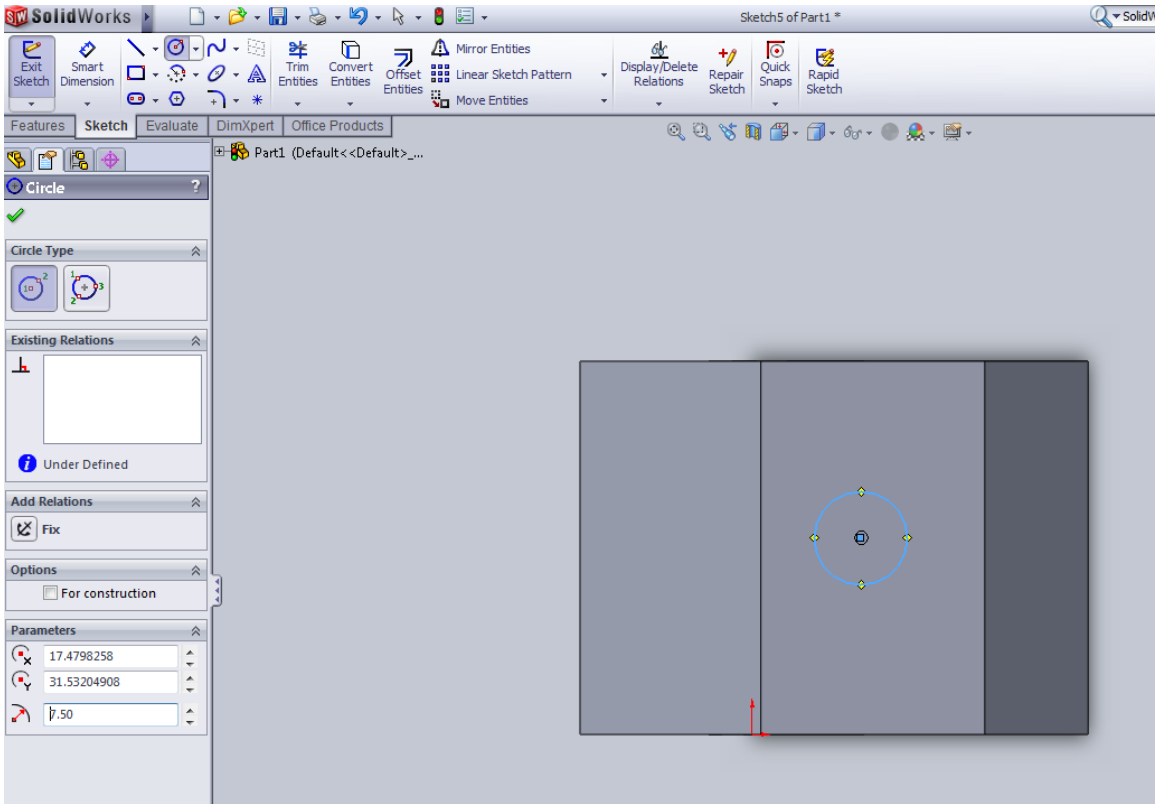
Step 4.1: Setting up the Sketching plane and sketching the circle:
Select the inclined face of the block and click the **Sketch** icon from the **Sketch** toolbar



Click the **Normal To** icon from the **Standard Views** toolbar.

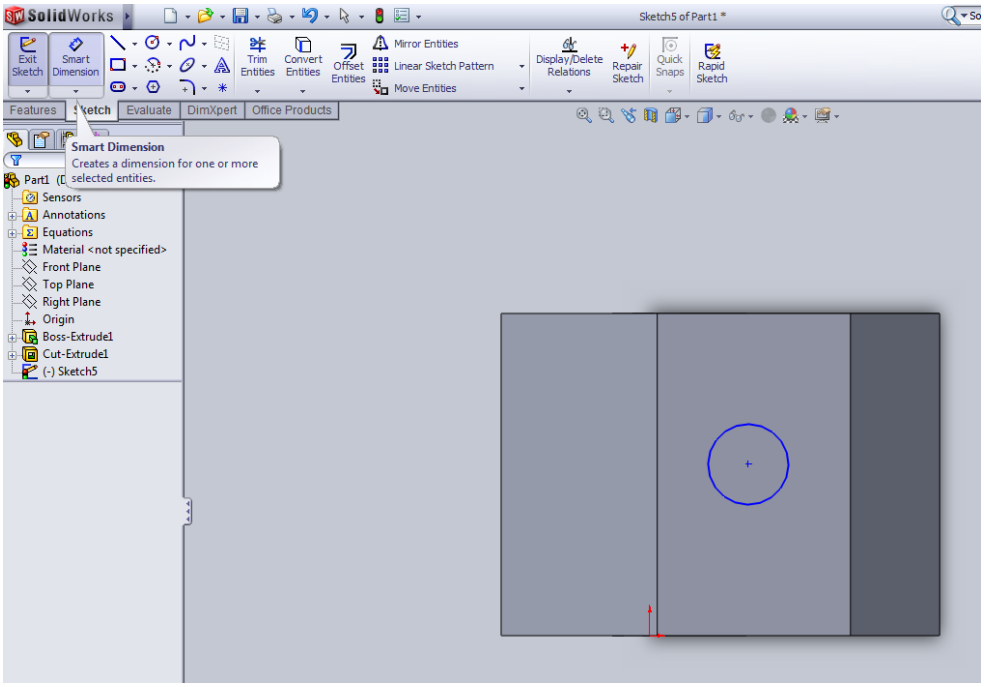


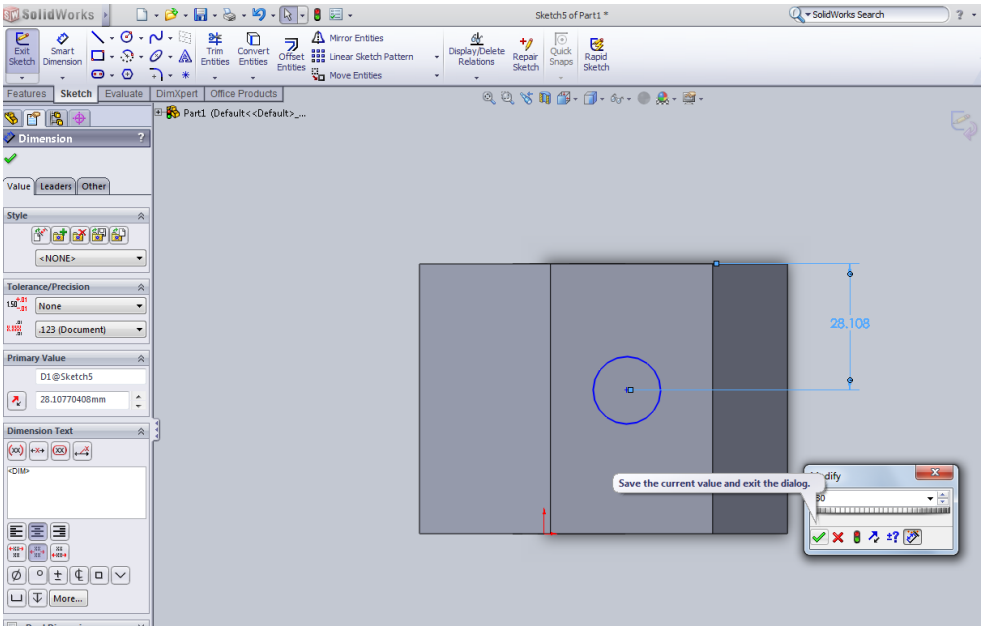
Click the **Circle** icon from the **Sketch Tools** toolbar. Sketch the circle with approximate radius of 7.5mm

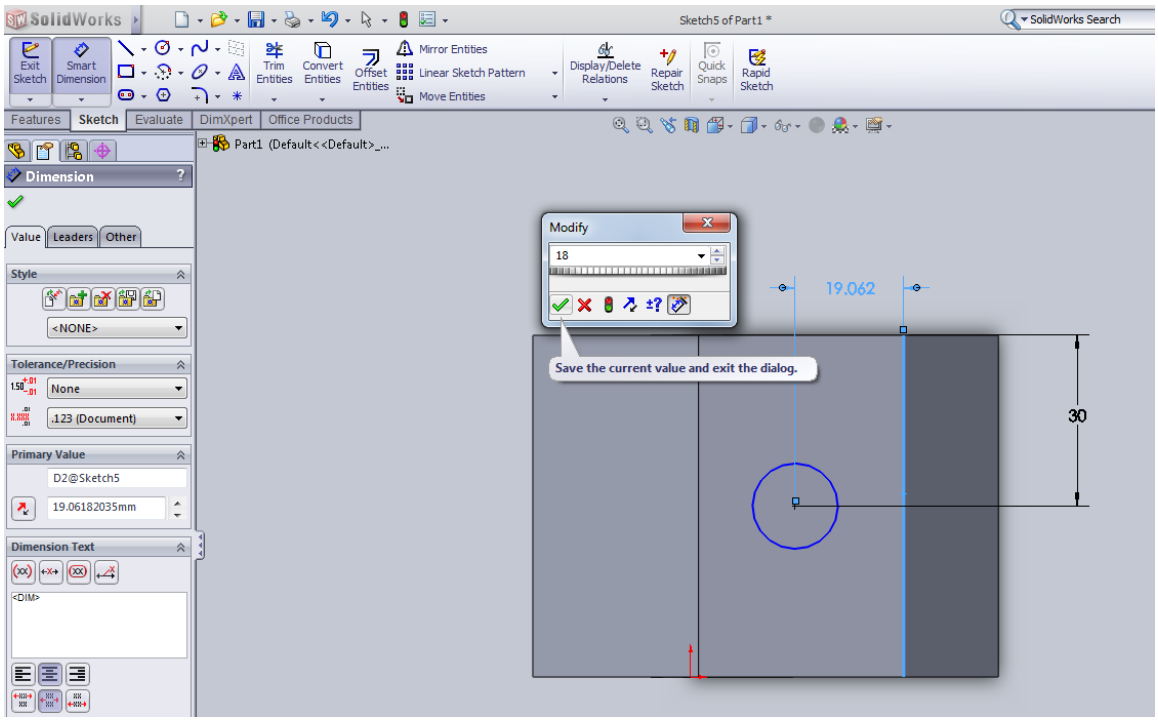


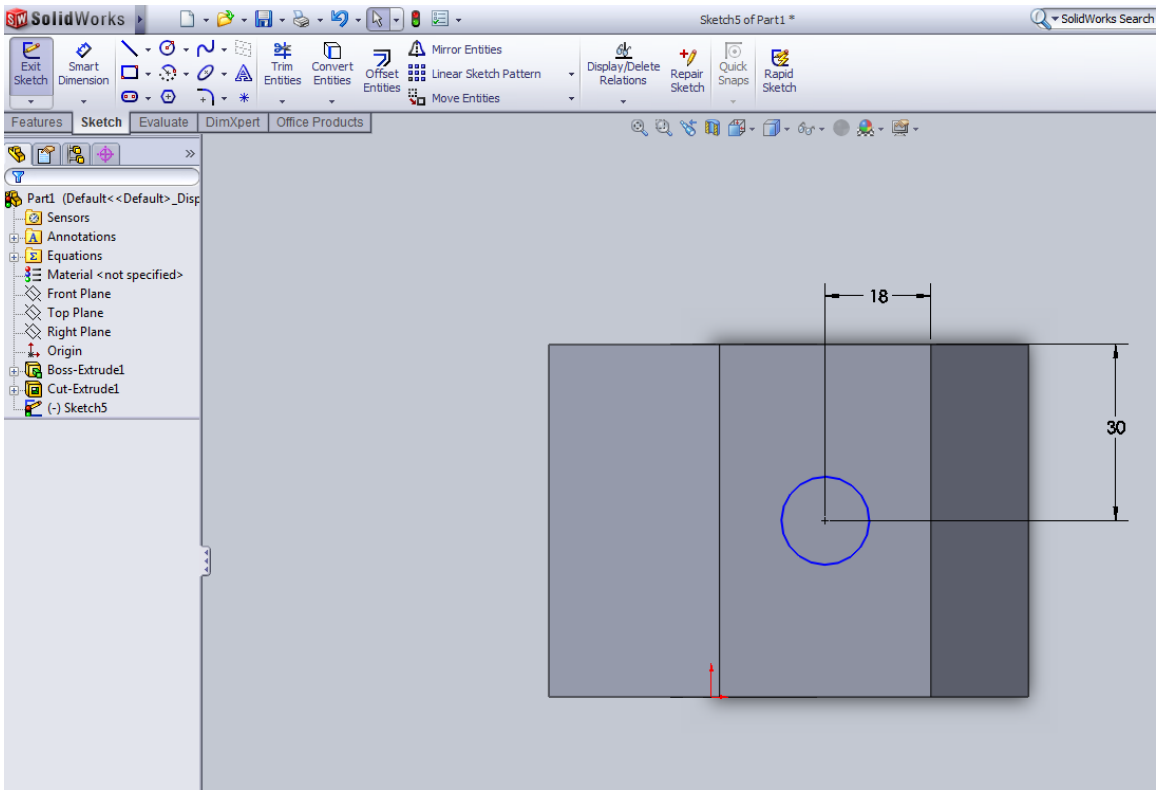
Step 4.2: Modifying the dimensions

Modify the dimensions using the procedure discussed earlier to match to those shown in the figure below.





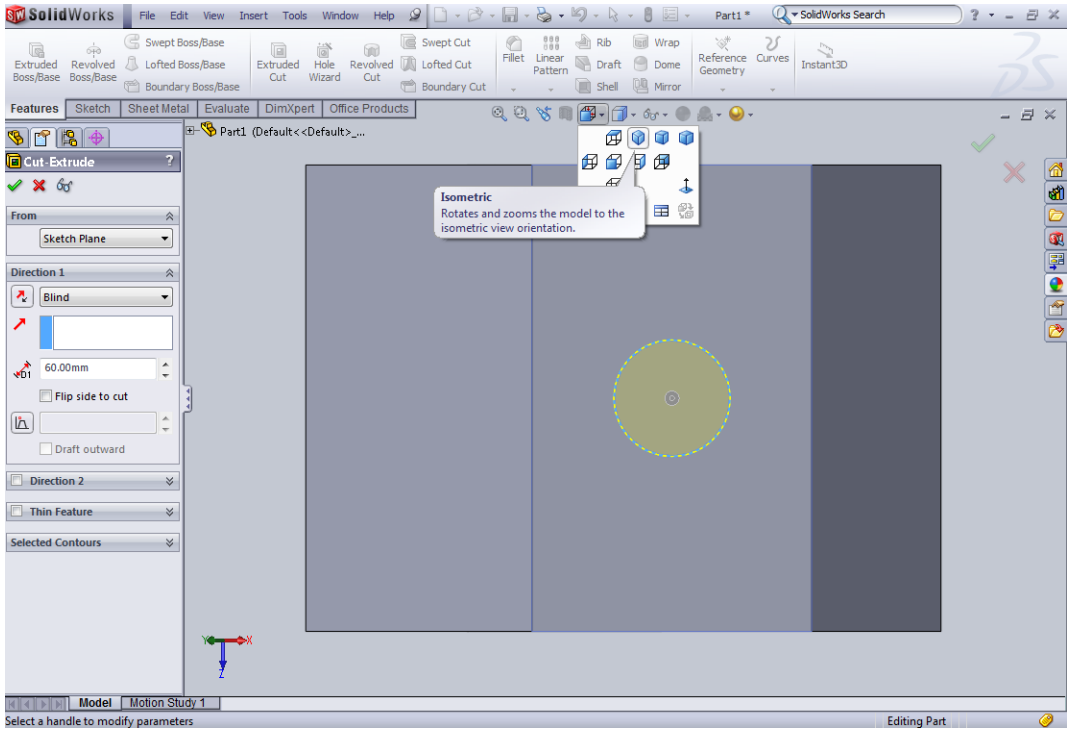


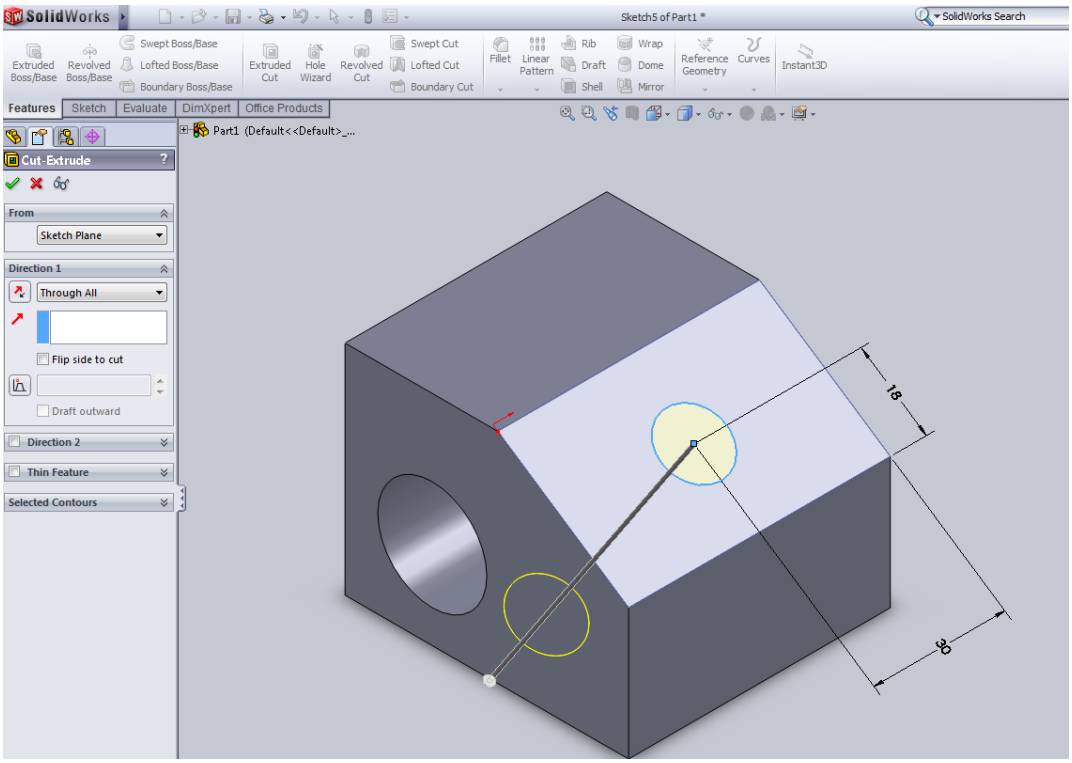


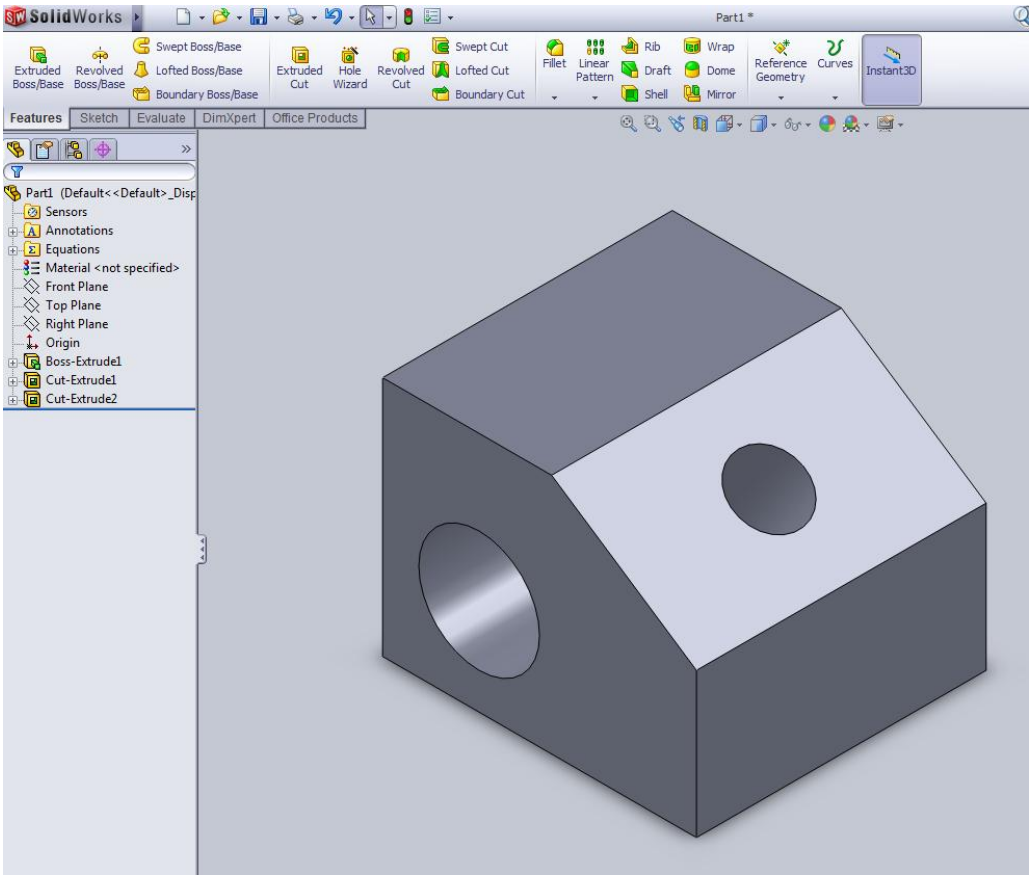
Thus the hole is located centrally on the inclined plane. Now orient the model to an isometric view by clicking the **Isometric** icon in the **Standard Views** toolbar.

Step 4.3: Extruding the circle

Using the procedure described in step 3.4 create the hole. (the depth is **Through All**). Hit Enter to finish the extrusion of the hole.

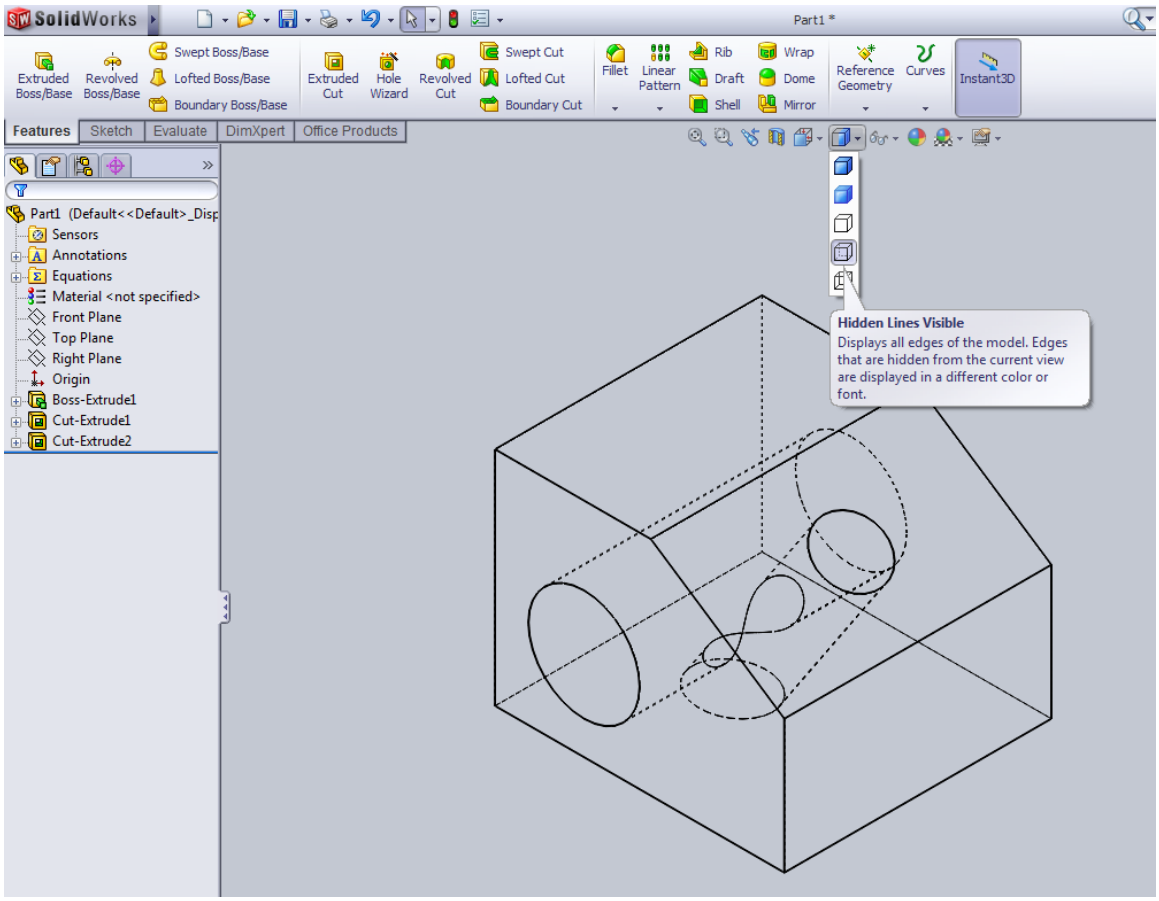





SolidWorks
Part1 *
Extruded Boss/Base
Revolved Boss/Base
Swept Boss/Base
Lofted Boss/Base
Boundary Boss/Base
Extruded Cut
Hole Wizard
Revolved Cut
Swept Cut
Lofted Cut
Boundary Cut
Fillet
Linear Pattern
Rib
Draft
Shell
Wrap
Dome
Mirror
Reference Geometry
Curves
Instant3D
Features
Sketch
Evaluate
DimXpert
Office Products
Part1 (Default<<Default>_Disp
Sensors
Annotations
Equations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Boss-Extrude1
Cut-Extrude1
Cut-Extrude2

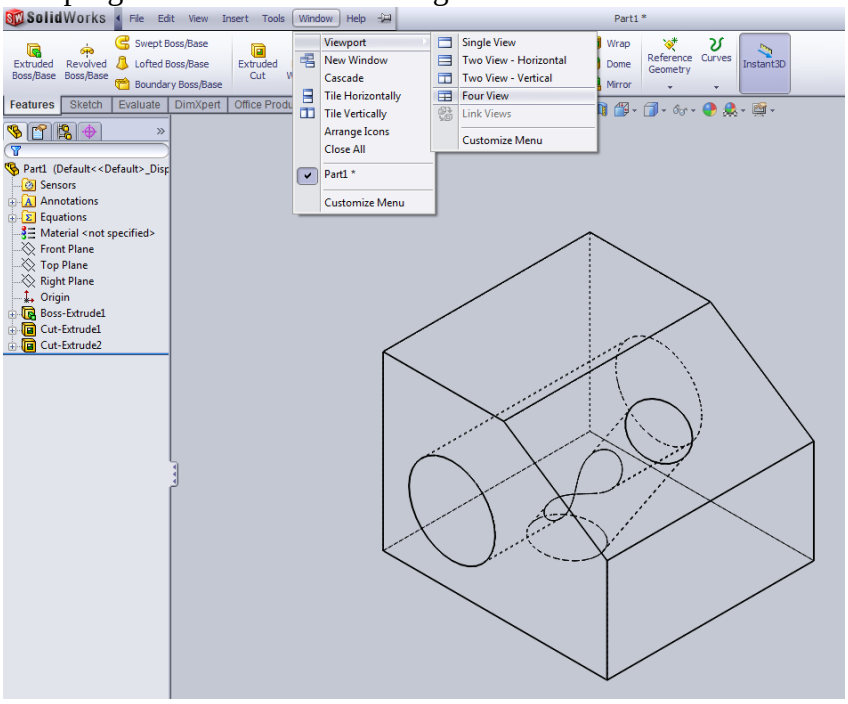
Step 5: Modifying the isometric views

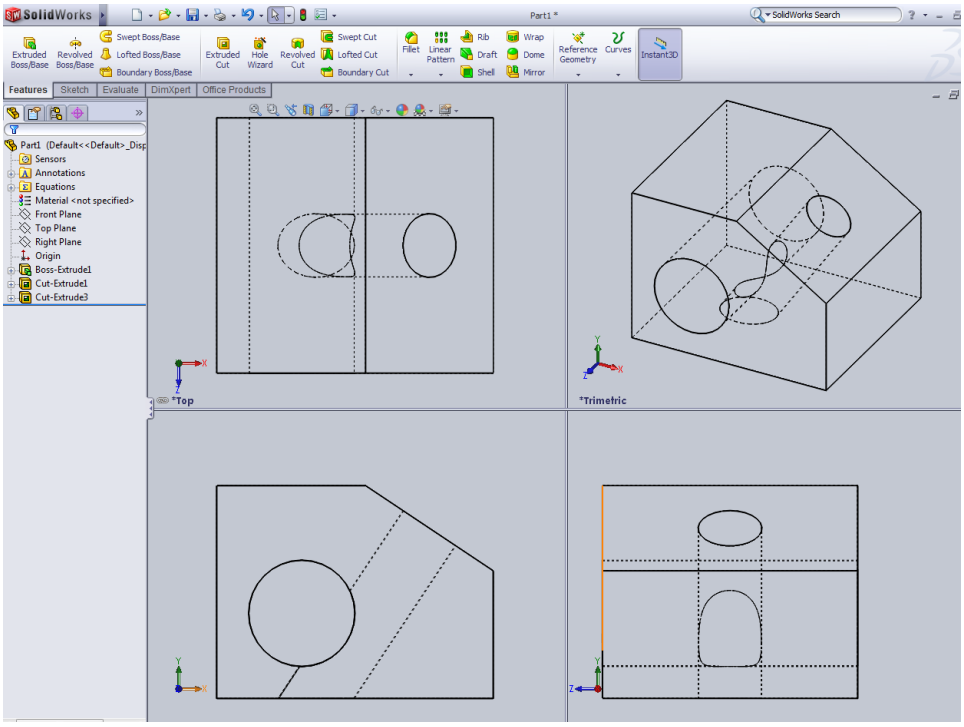
The view shown in the above figure is the shaded image of the isometric view of the model. To obtain a wireframe isometric image of the model click on the **Wireframe** icon from the **View** toolbar or click on the **Hidden line visible** from **View** toolbar.



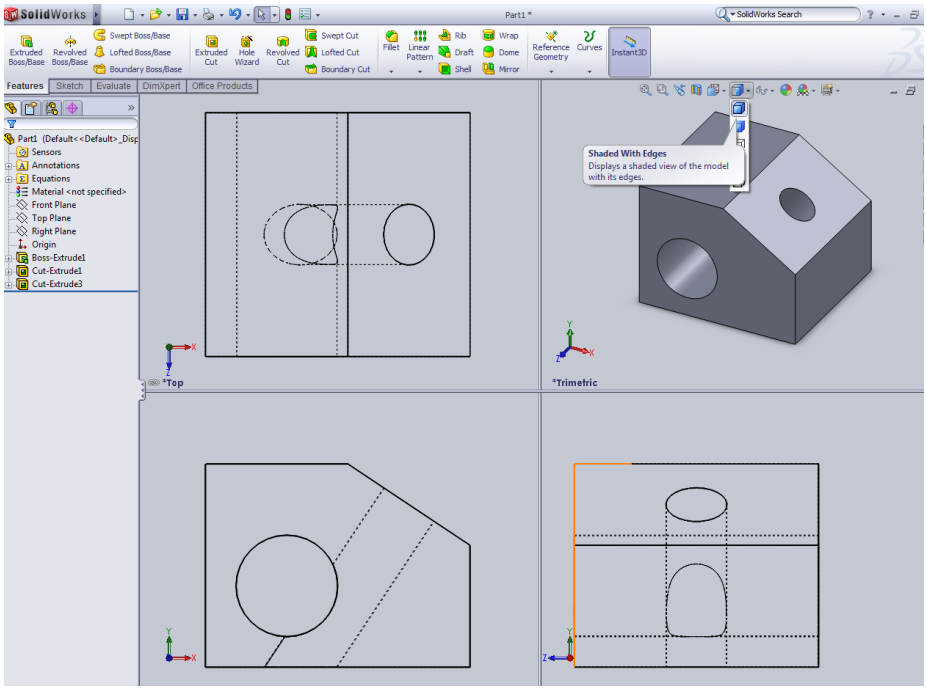
Step 6: Converting to a 4 – view layout

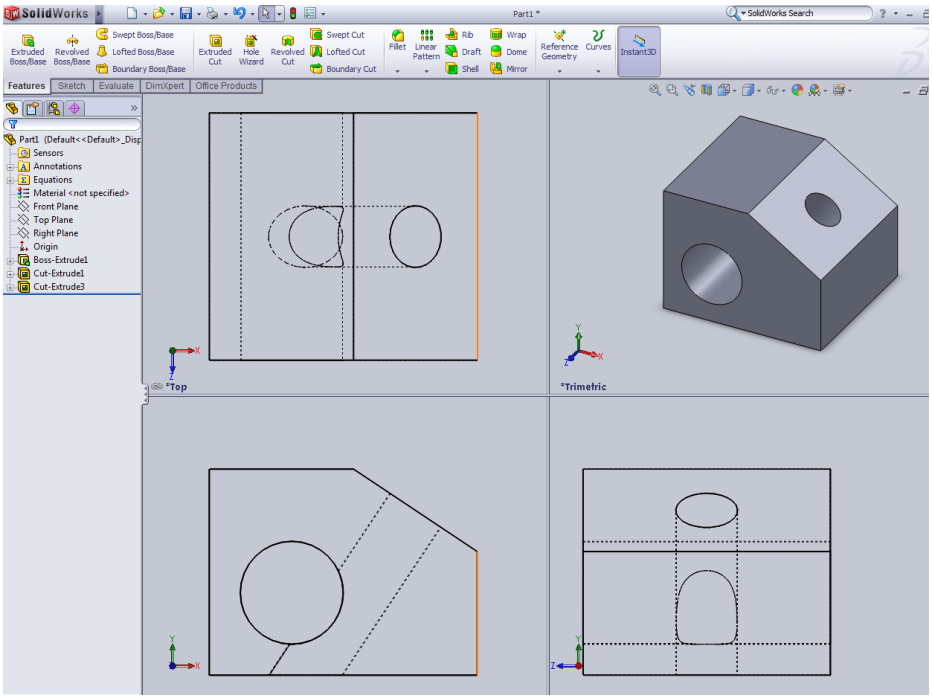
We can obtain a 4-view layout by dragging the horizontal and vertical split controls which are located near the horizontal and the vertical scroll bars near the bottom left corner and the top right corner of the modeling window.





Now we can set the appropriate views in each window by clicking in the window and selecting suitable options from the **View** and **Standard Views** toolbar. The following table shows various combinations.



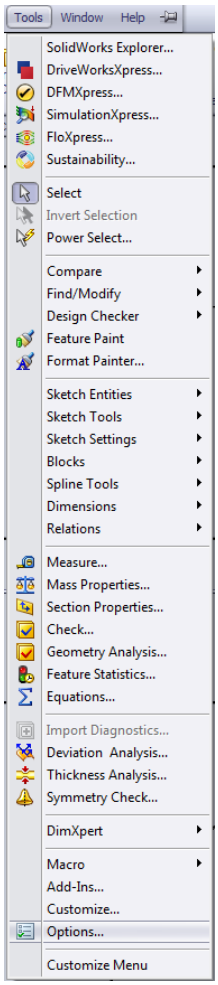


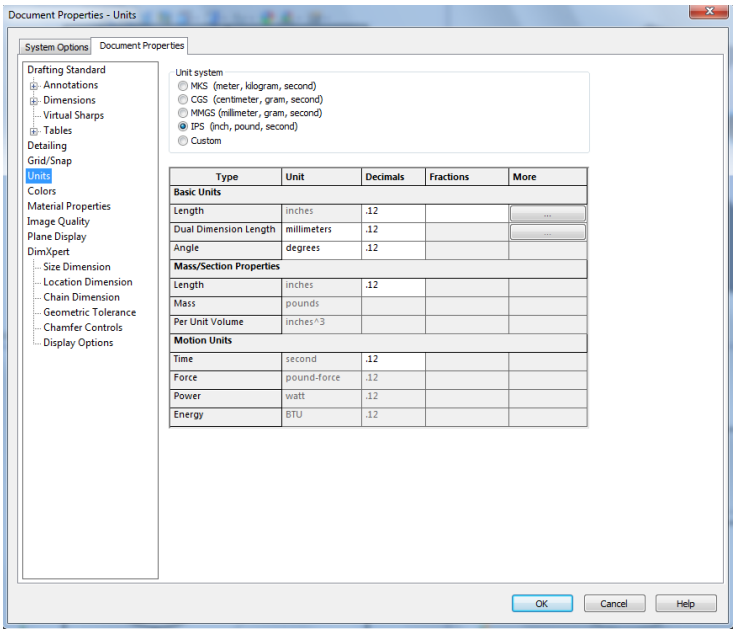
The combinations to create the views in respective windows:

Window	Icon in Standard View toolbar	Icon in Views toolbar
Top-left	Top	Hidden in gray
Bottom-left	Front	Hidden in gray
Bottom-right	Right	Hidden in gray
Top-right	Isometric	Shaded

Step 7: Converting the model database units

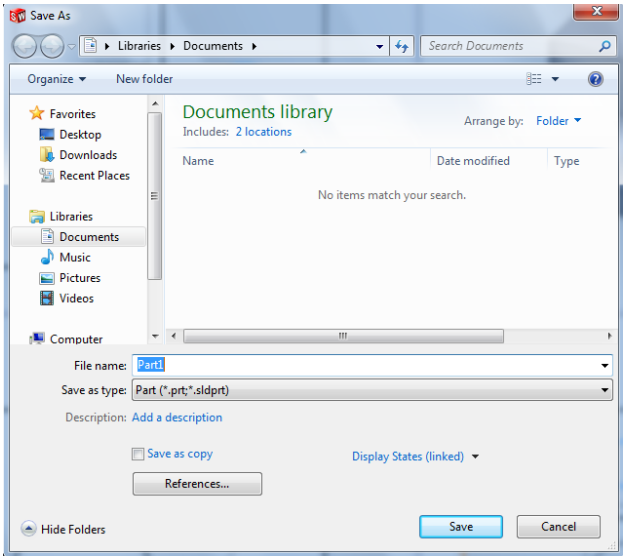
The database units while we created the model were in mm. Now we shall see how to convert the database units from mm. to inches. Go to **Tools**, click **Options**. Click the tab **Document Properties**, select **Units,** set units to "IPS". The drawing units are now in inches.





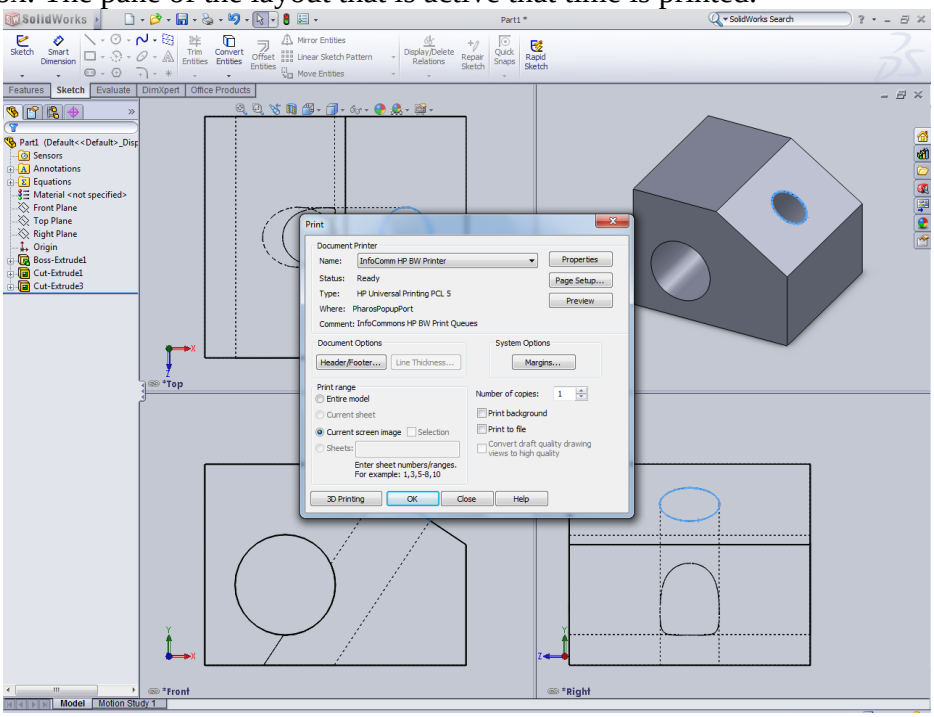
Step 8: Saving a file

In the main window, click on **File,** go to **Save As** and enter the suitable file name and click on **Save**.



Step 9: Printing a file

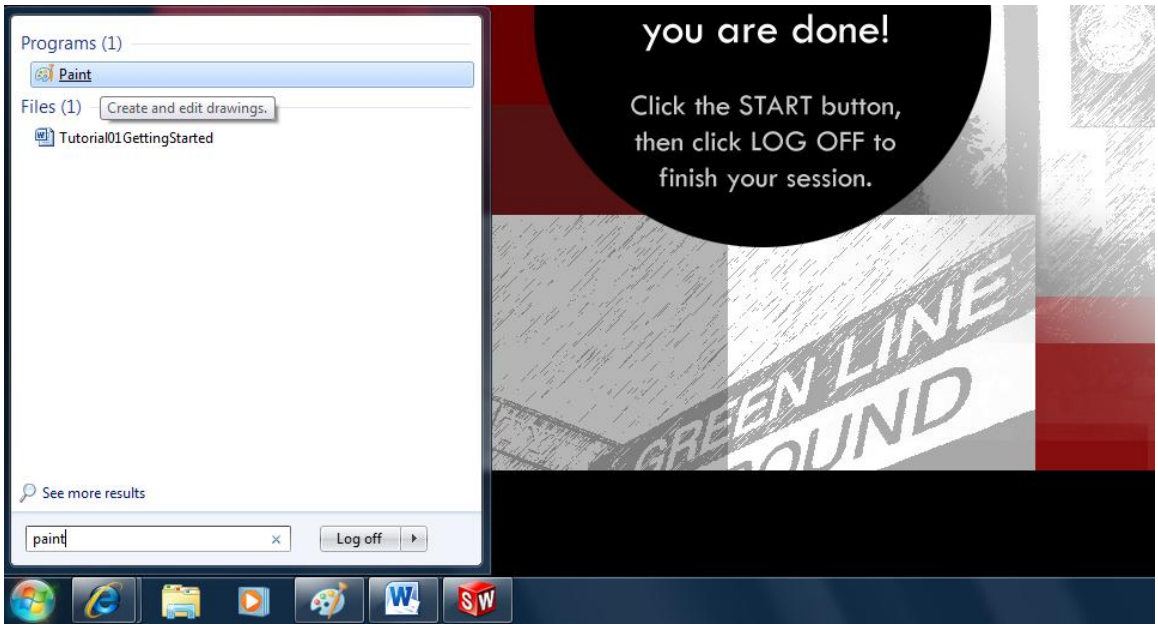
In the main window, click on **File**, go to **Print** and click. Then click **OK** in the **Print** dialog box. The pane of the layout that is active that time is printed.



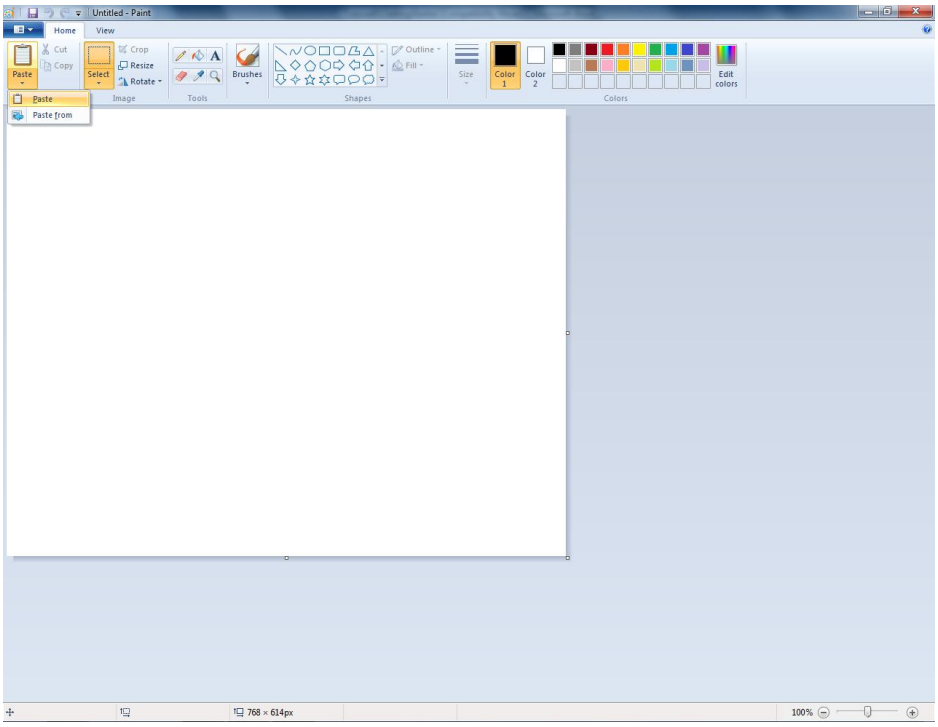
Thus by selecting each pane one by one we can print all the views.

Step 10: Screen captures

Hit the **Print Screen** key on the keyboard with the desired window maximized. Open **Paint** application by clicking on **Start** and **search for paint**. Then click **paint option from the top**.



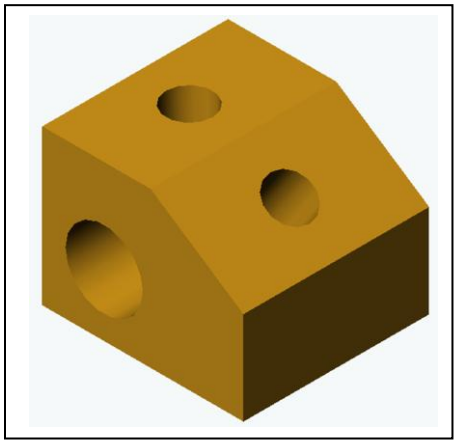
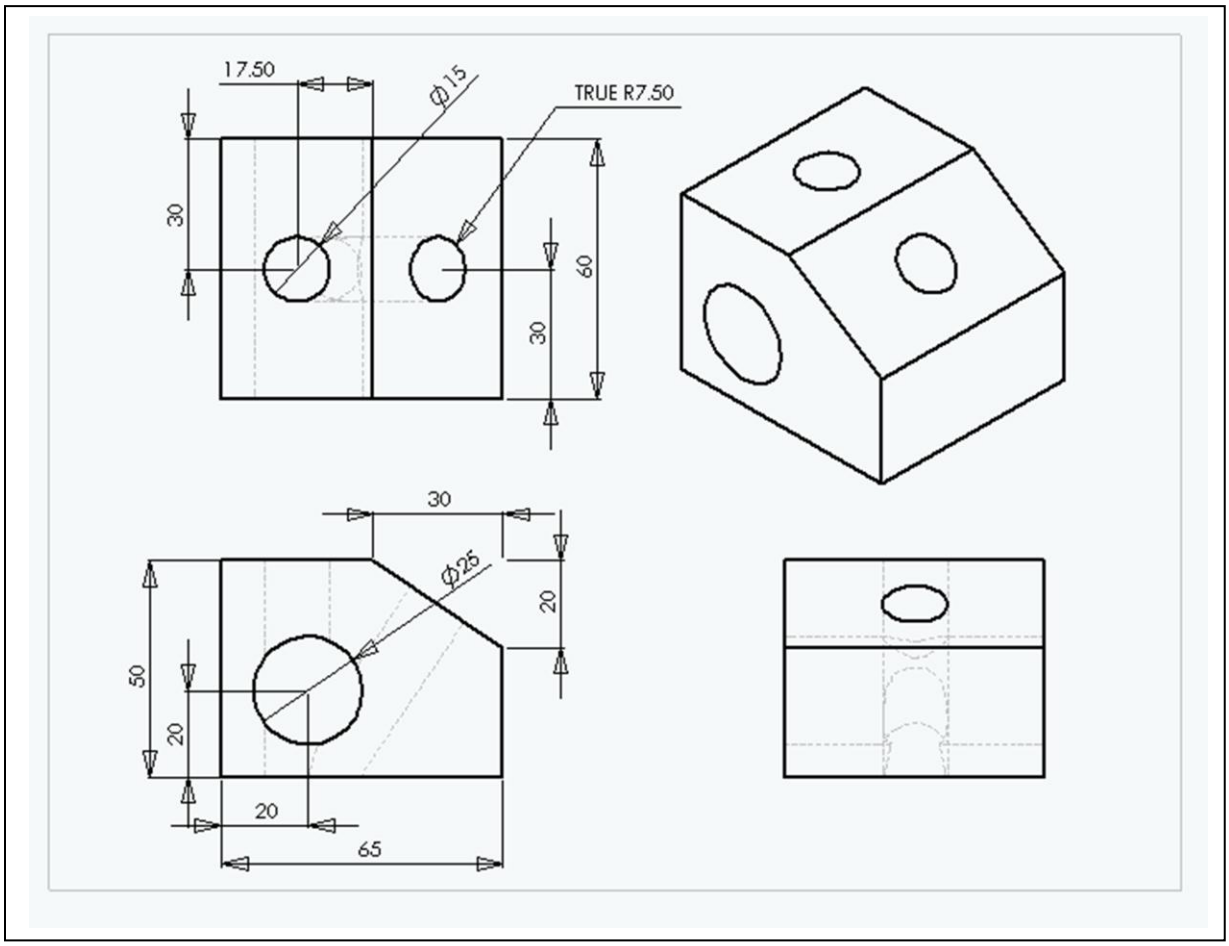
Go to **paste** and click **paste.**



Open **Word document**, go to **Edit** and click **Paste** to obtain an image of printable size. In the main window of **Word**, go to **File**, click **Print** and **OK** (Just as we do in the main window of SolidWorks).

Class Hands-on

Create a centrally placed, vertical hole of diameter 15 mm. in the top face of the block that we created in this tutorial as in the image below.



This completes the modeling of the chamfered block. Shown below is the dimensioned drawing of the model.