

DEPARTMENT OF MECHANICAL AND INDUSTRIAL ENGINEERING
NORTHEASTERN UNIVERSITY

CAPSULE PROGRAM
Funded by NSF grant #0833636



Tutorial 02
3D Part Modeling

SolidWorks 2010
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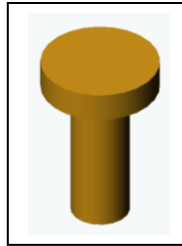
3D Part Modeling

In this tutorial you learn 3D part modeling and sketch control. We create three parts: pin, rectangular plate, and base plate

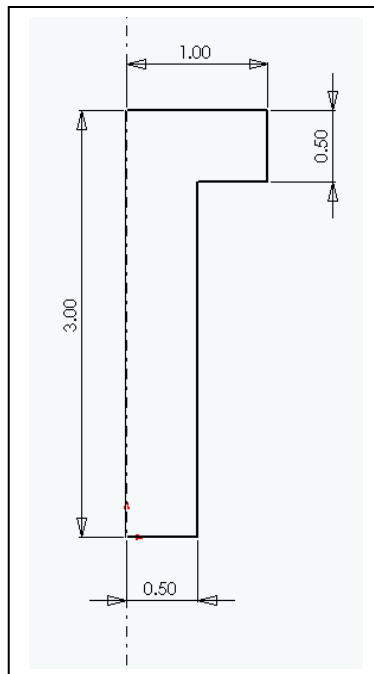
Pin

Step 0: General Planning Strategy:

We shall create the part shown in the image below. Listed below the figure are the steps in brief that we must perform to create the part.



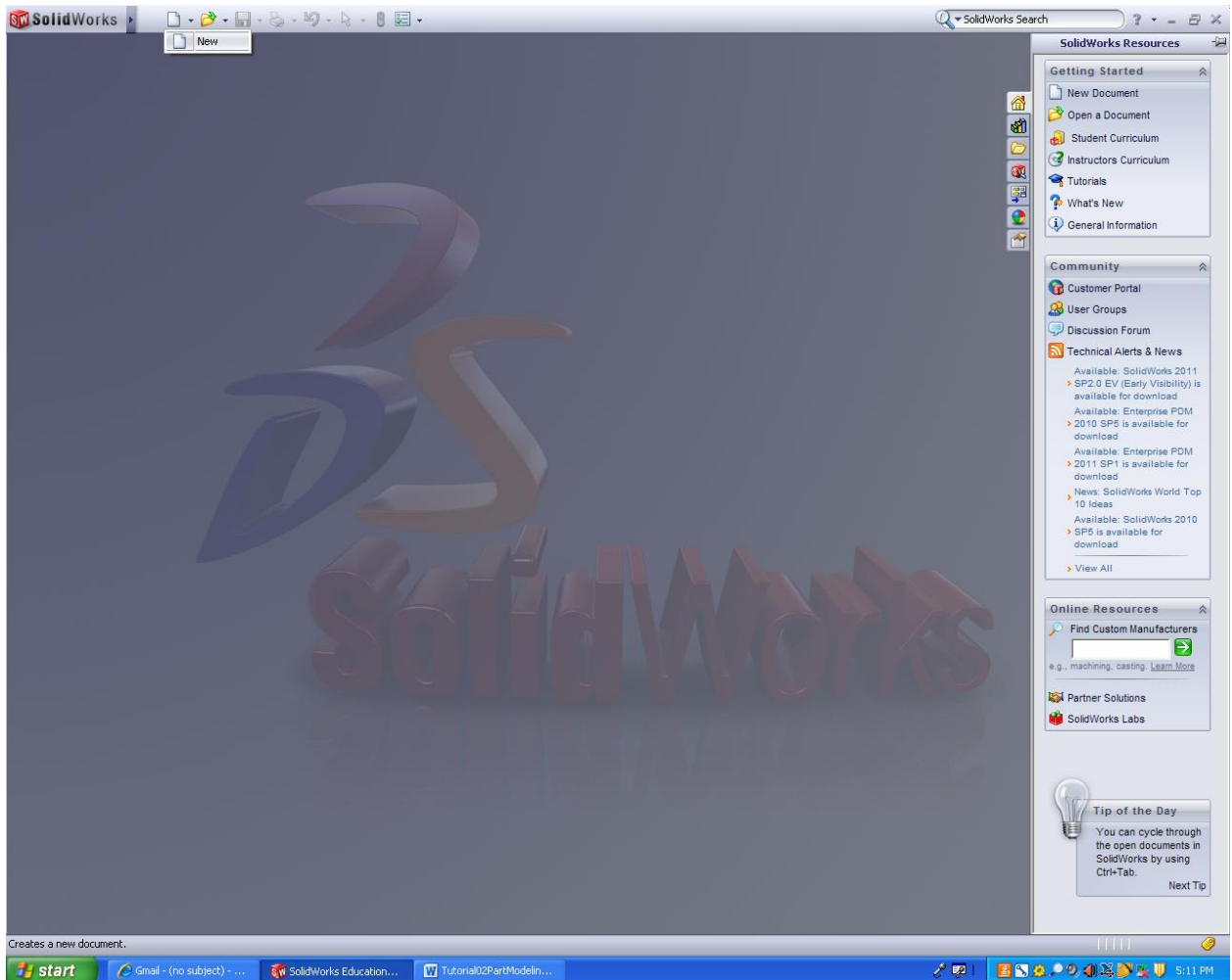
1. Create and revolve the profile.



Creating the Pin

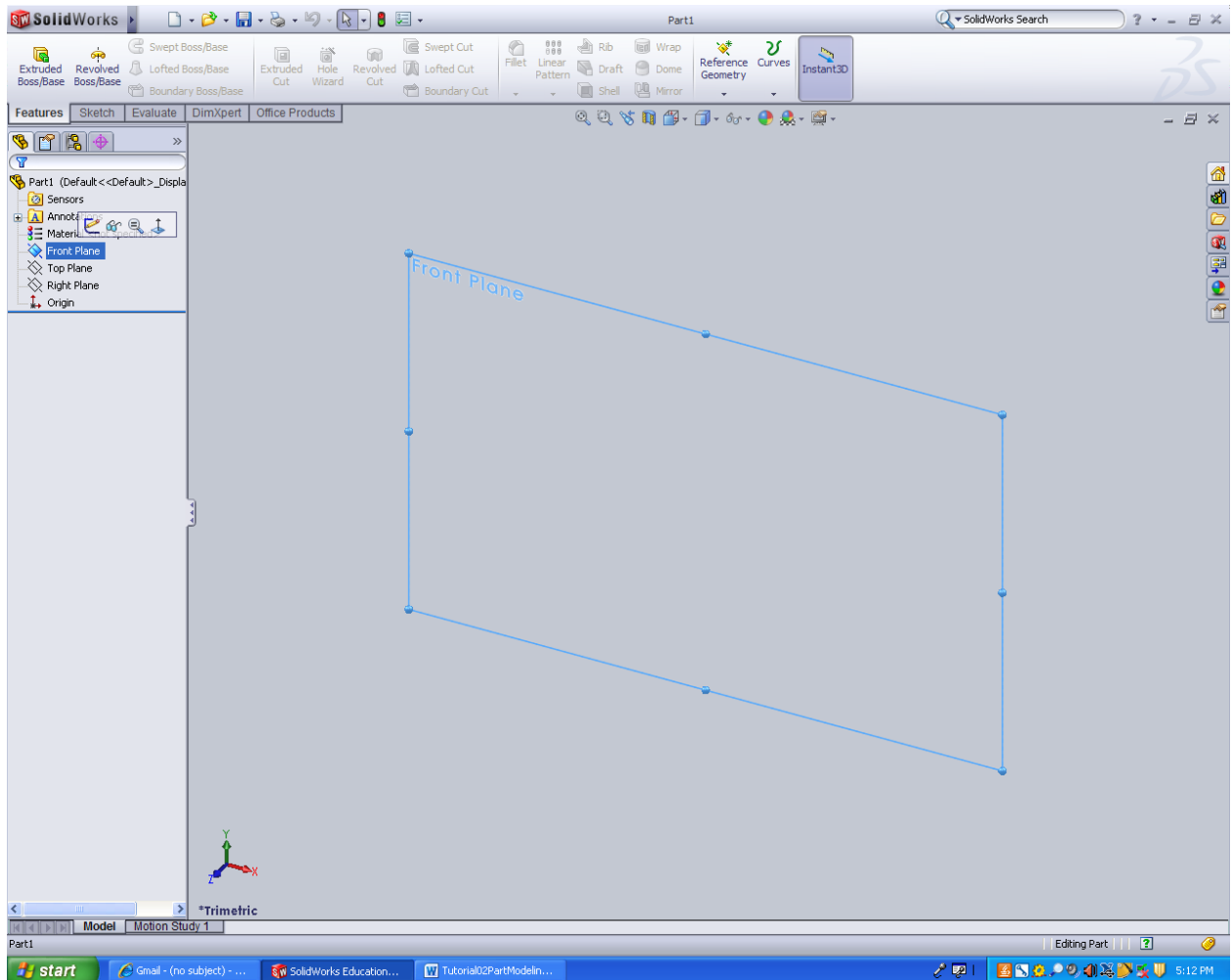
Step 1

File → New → Part



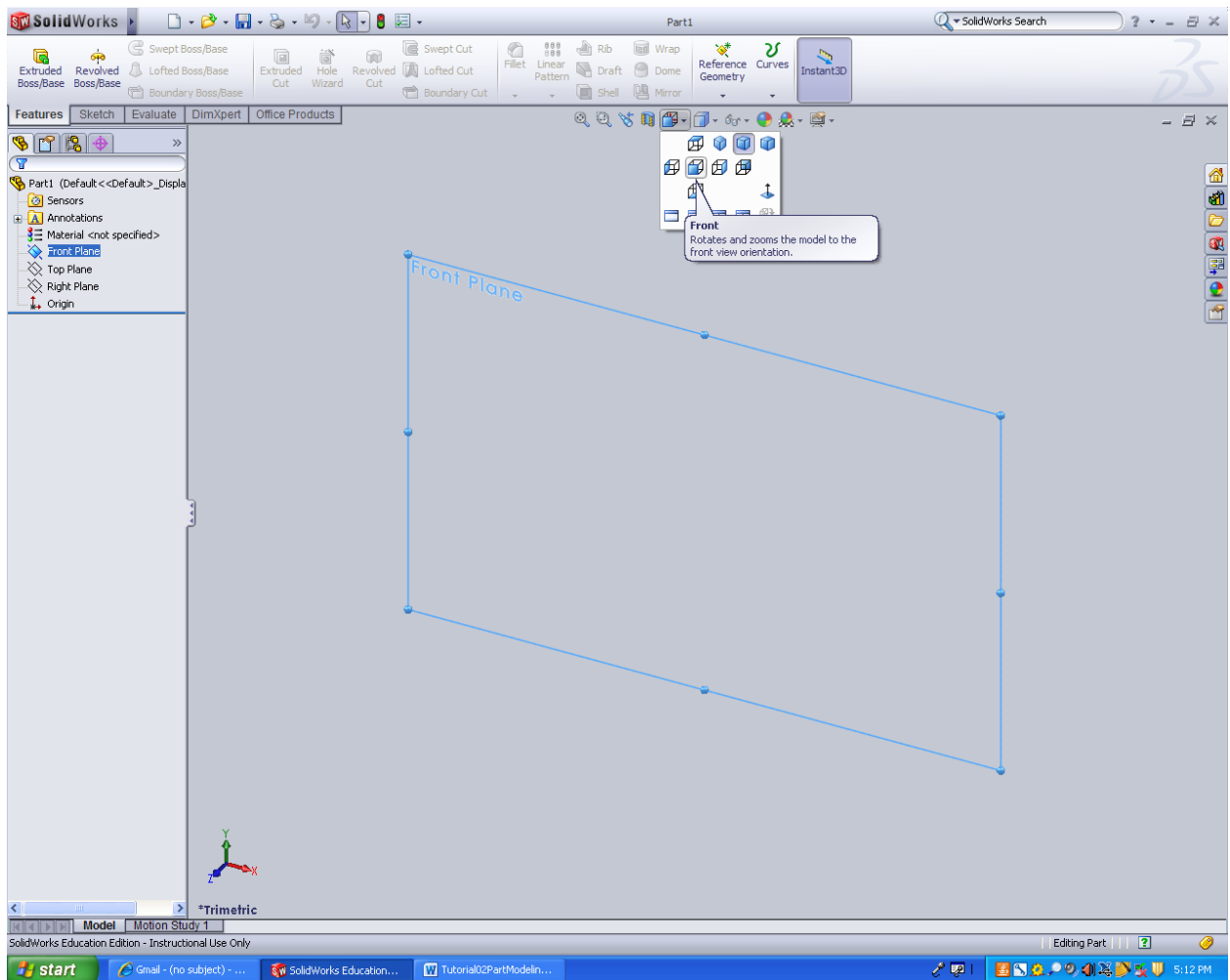
Step 2

Select **Front Plane** from the feature manager



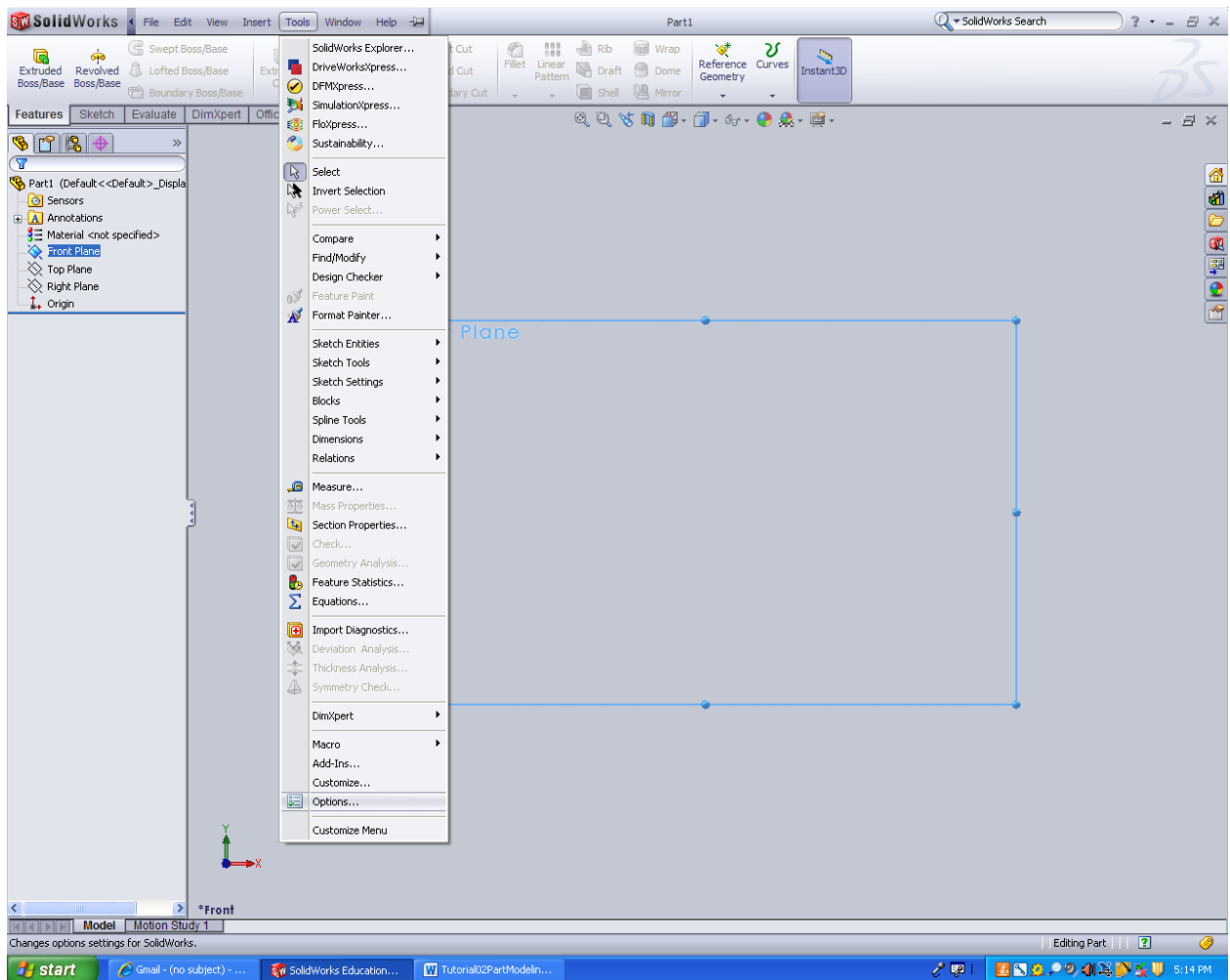
Step 3

Select **Front View** from view orientation icon



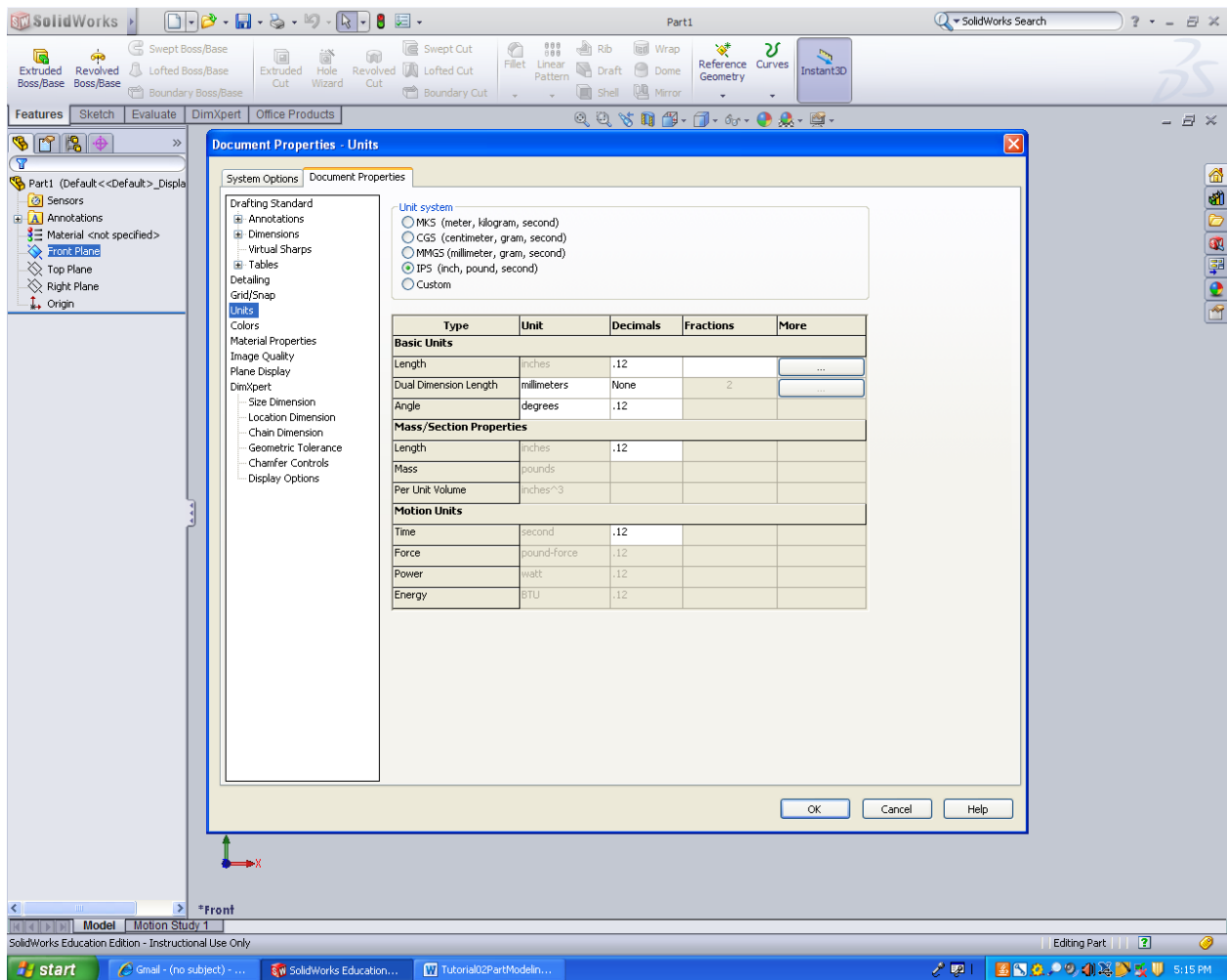
Step 4

Go to Tools → Options

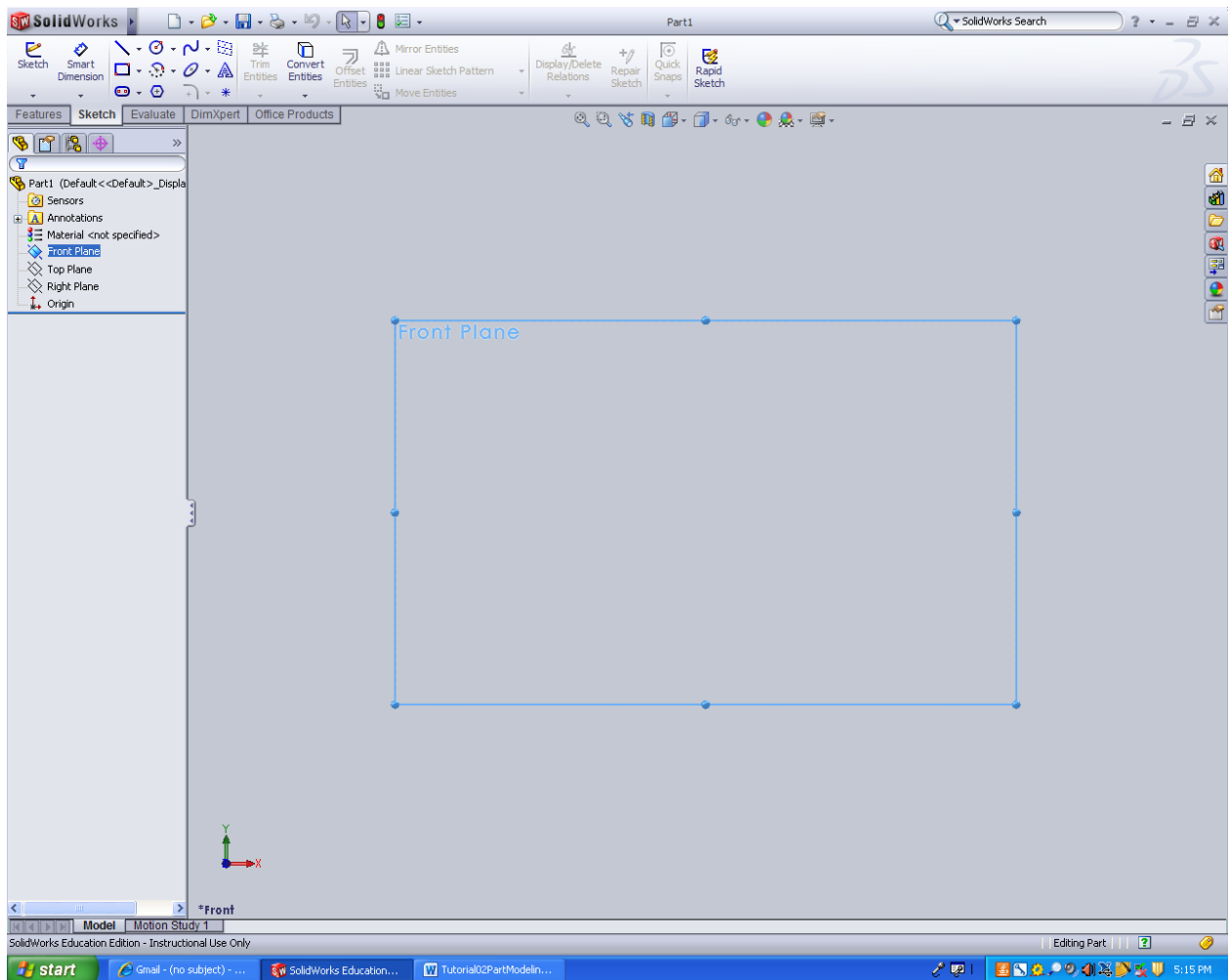


Step 5

Go to Document properties → Units → IPS



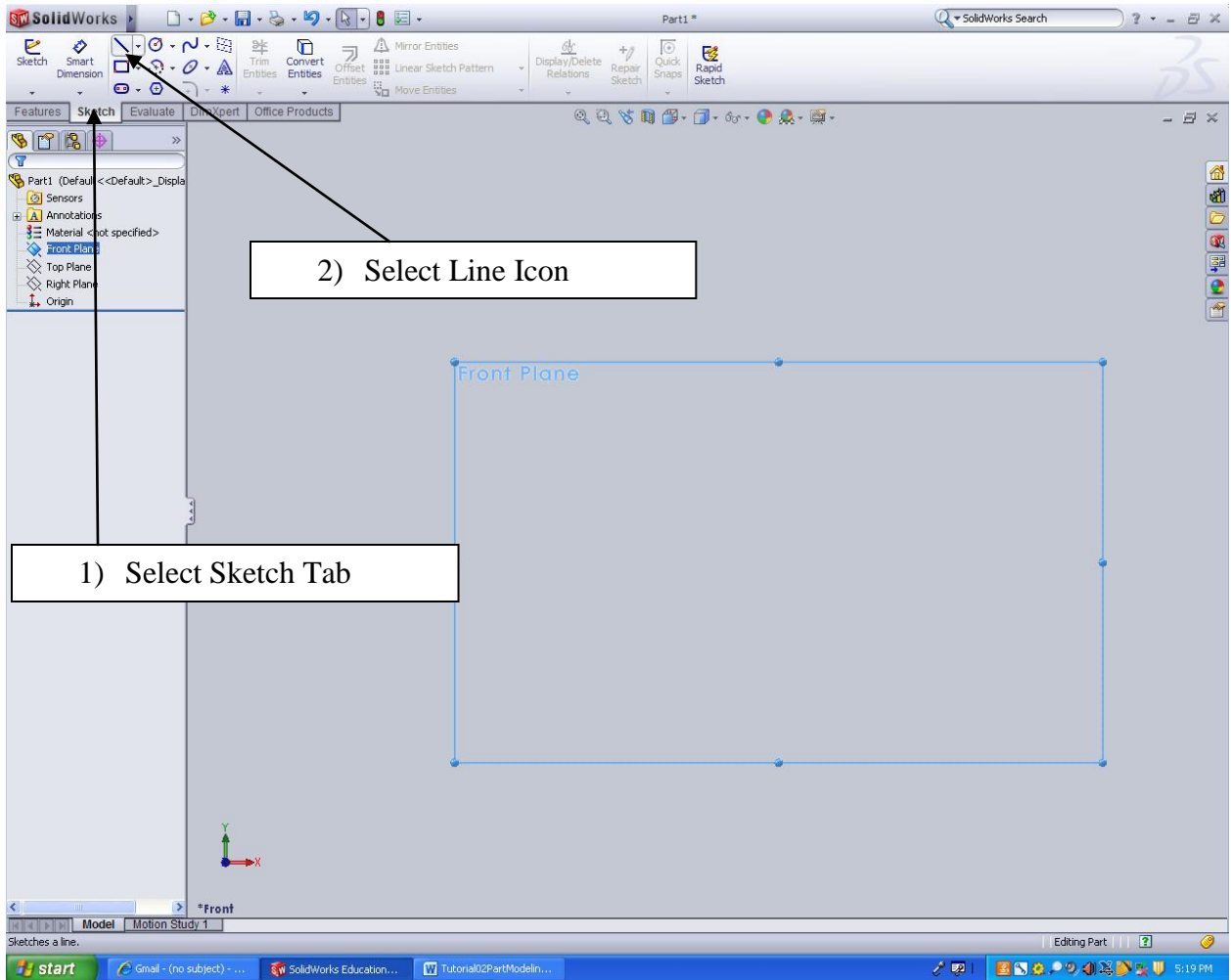
The IPS unit is selected for the drawing. The drawing will now have inches as its unit.



Step 6

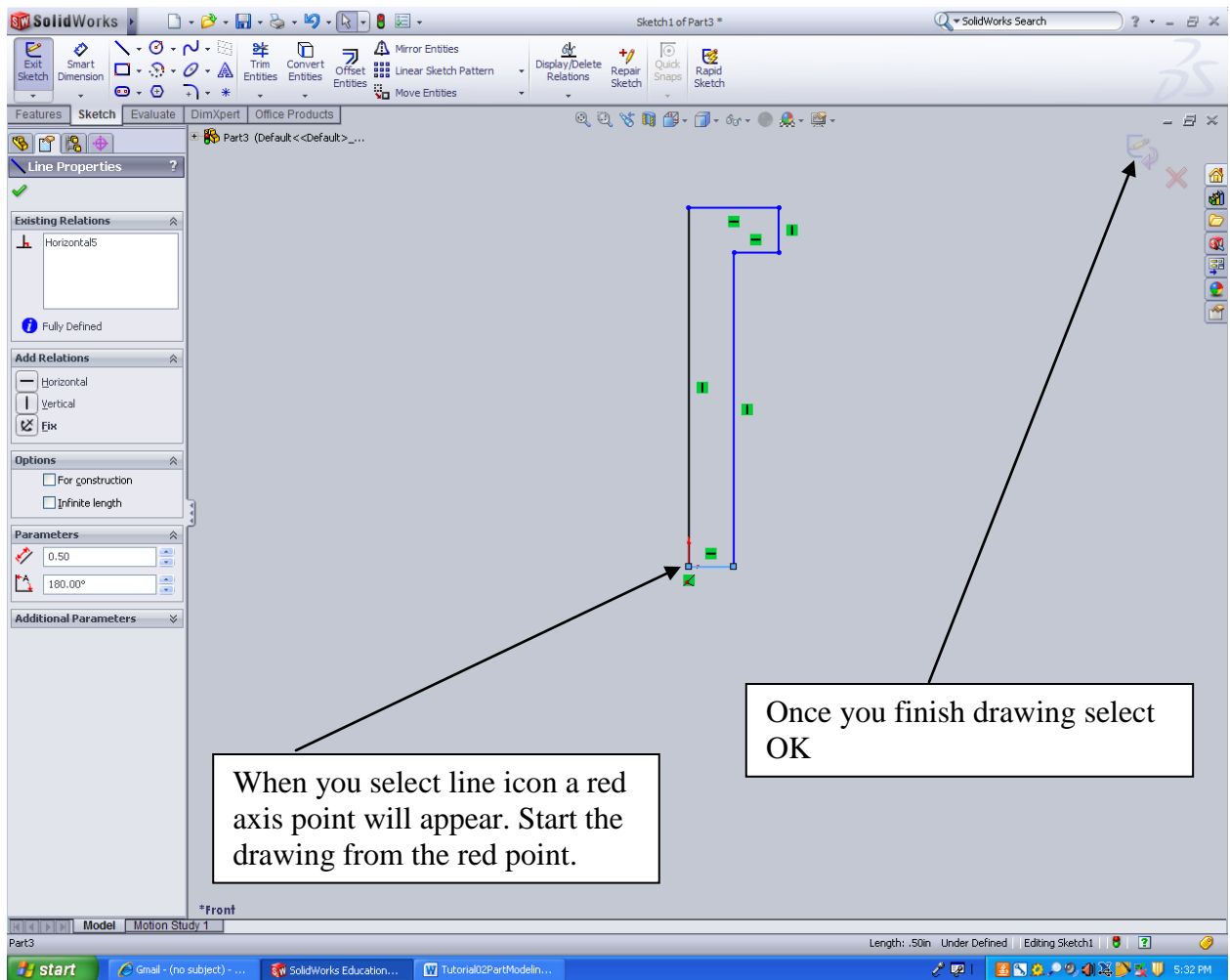
Select **Sketch Tab**

Then select **Line Icon**



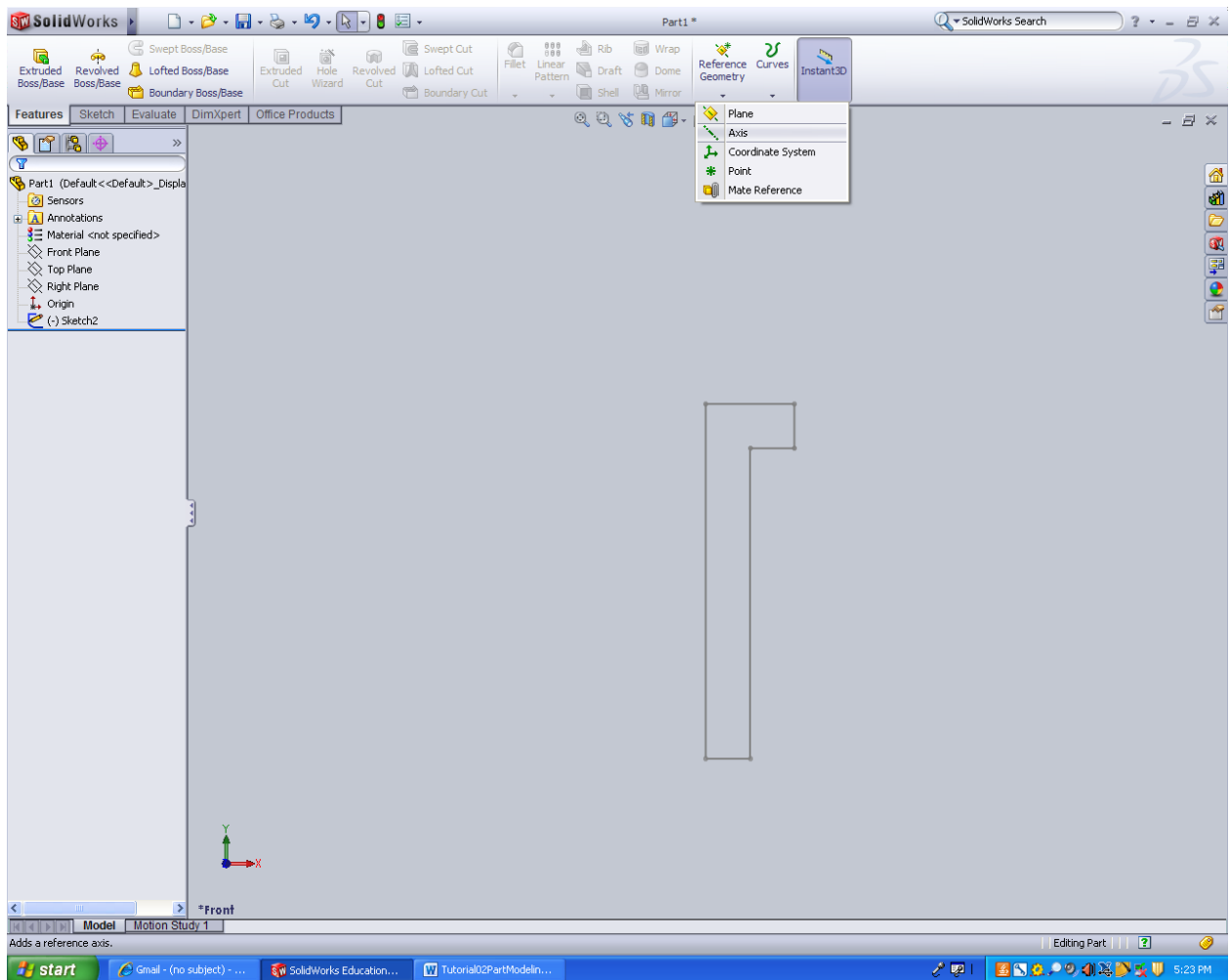
Step 7

Draw the drawing as per required dimensions

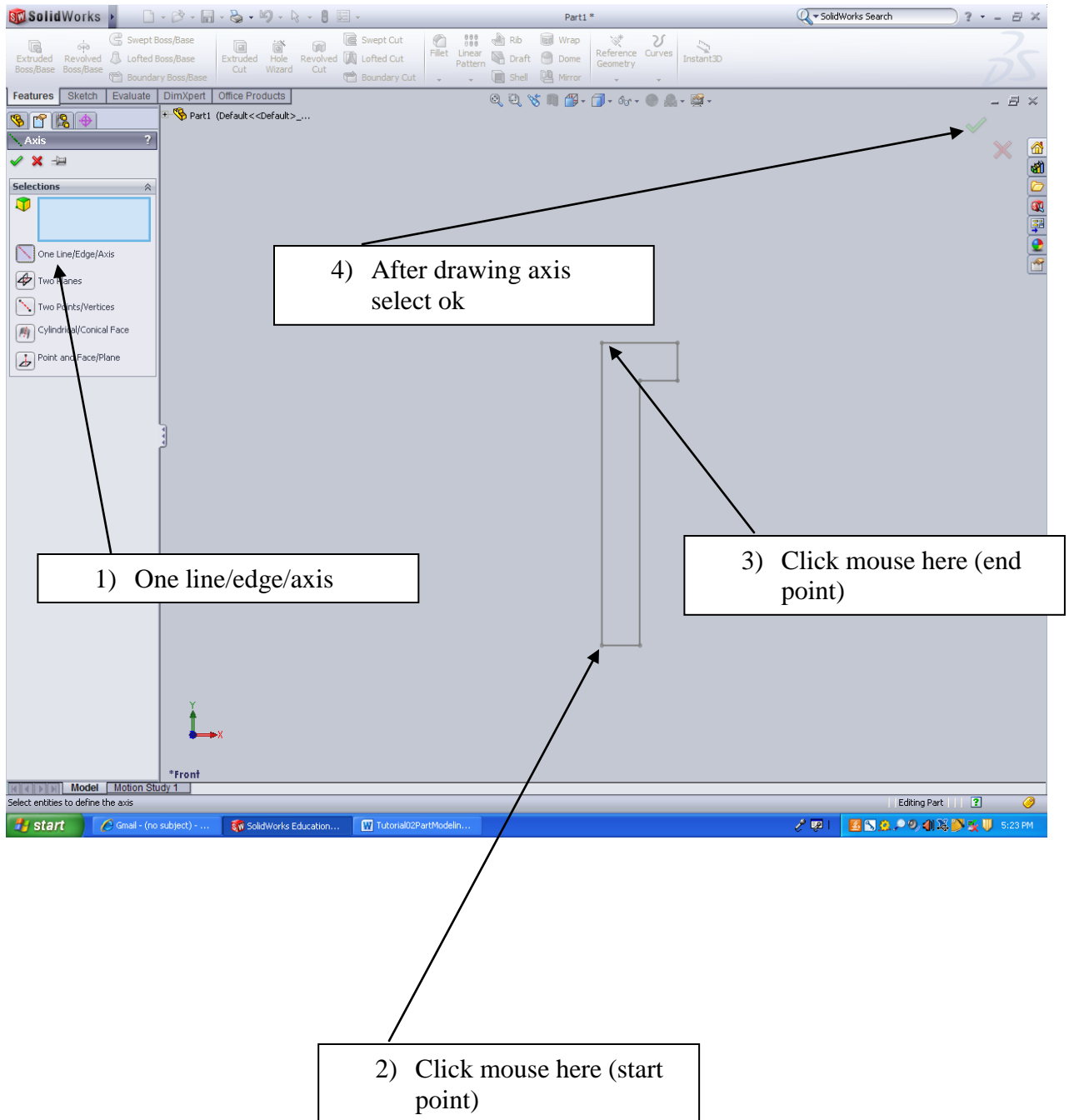


Step 8

Go to **Features Tab** → **Reference Geometries** → **Axis**

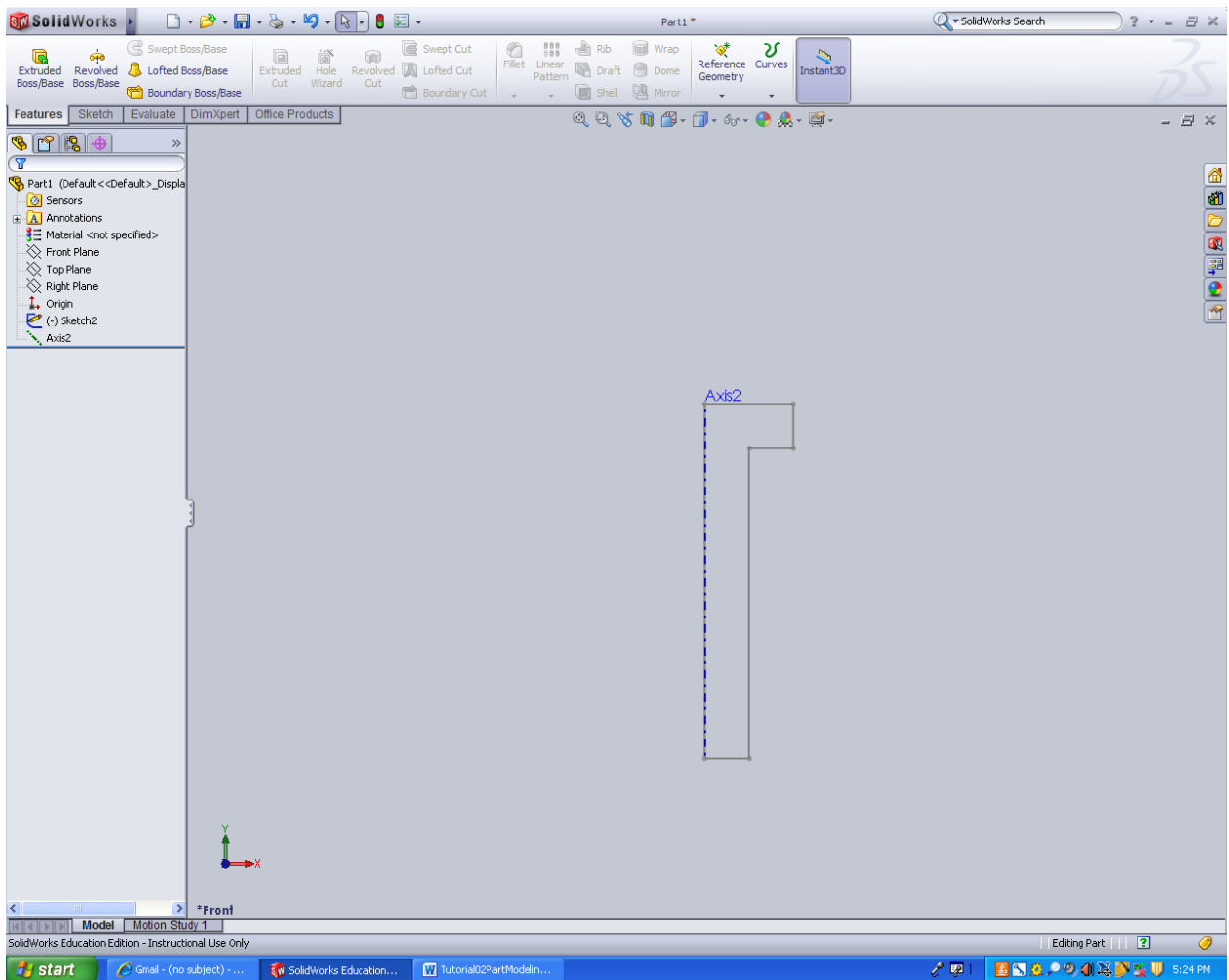


Step 9



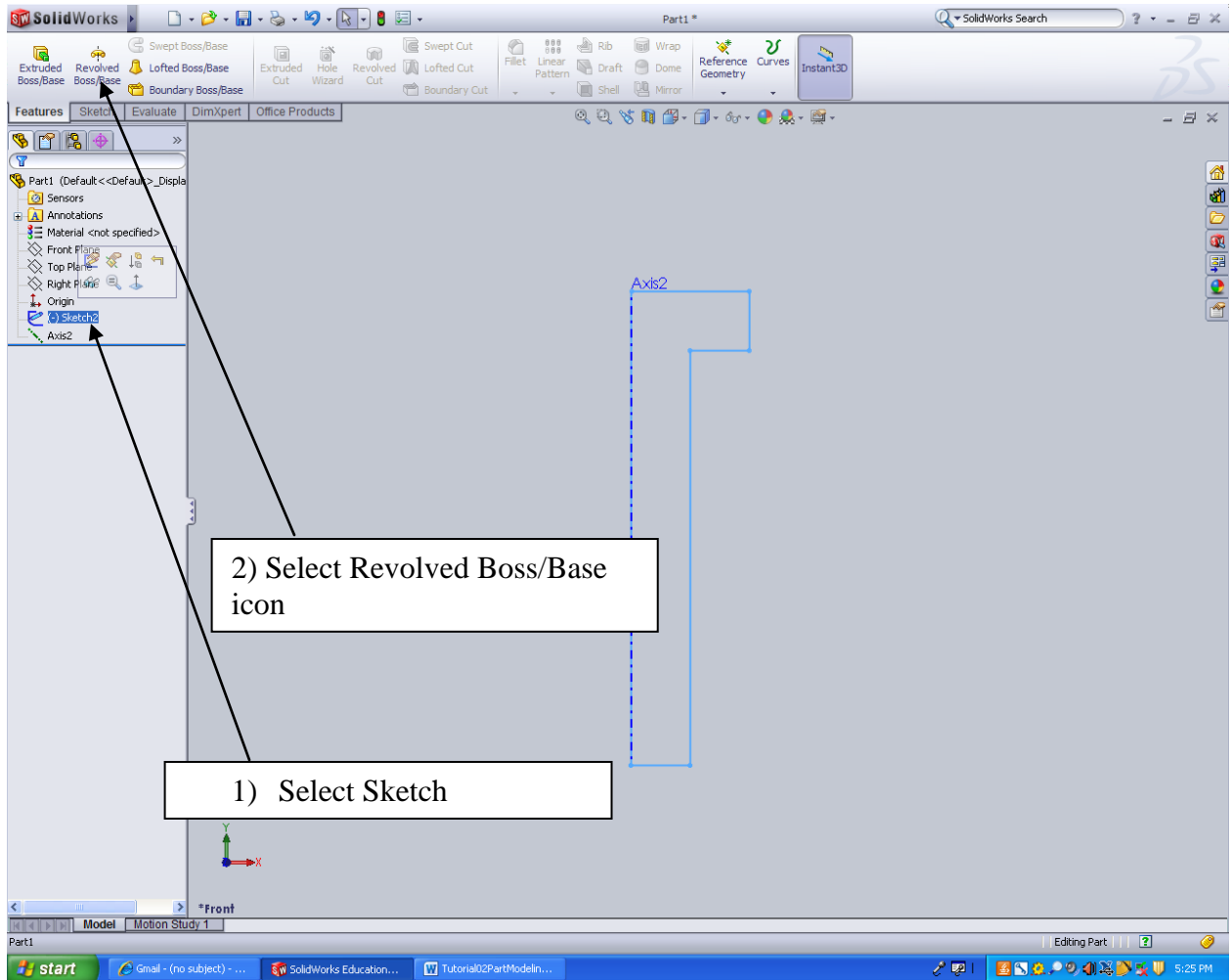
Step 10

Now we are ready to perform the revolve operation.

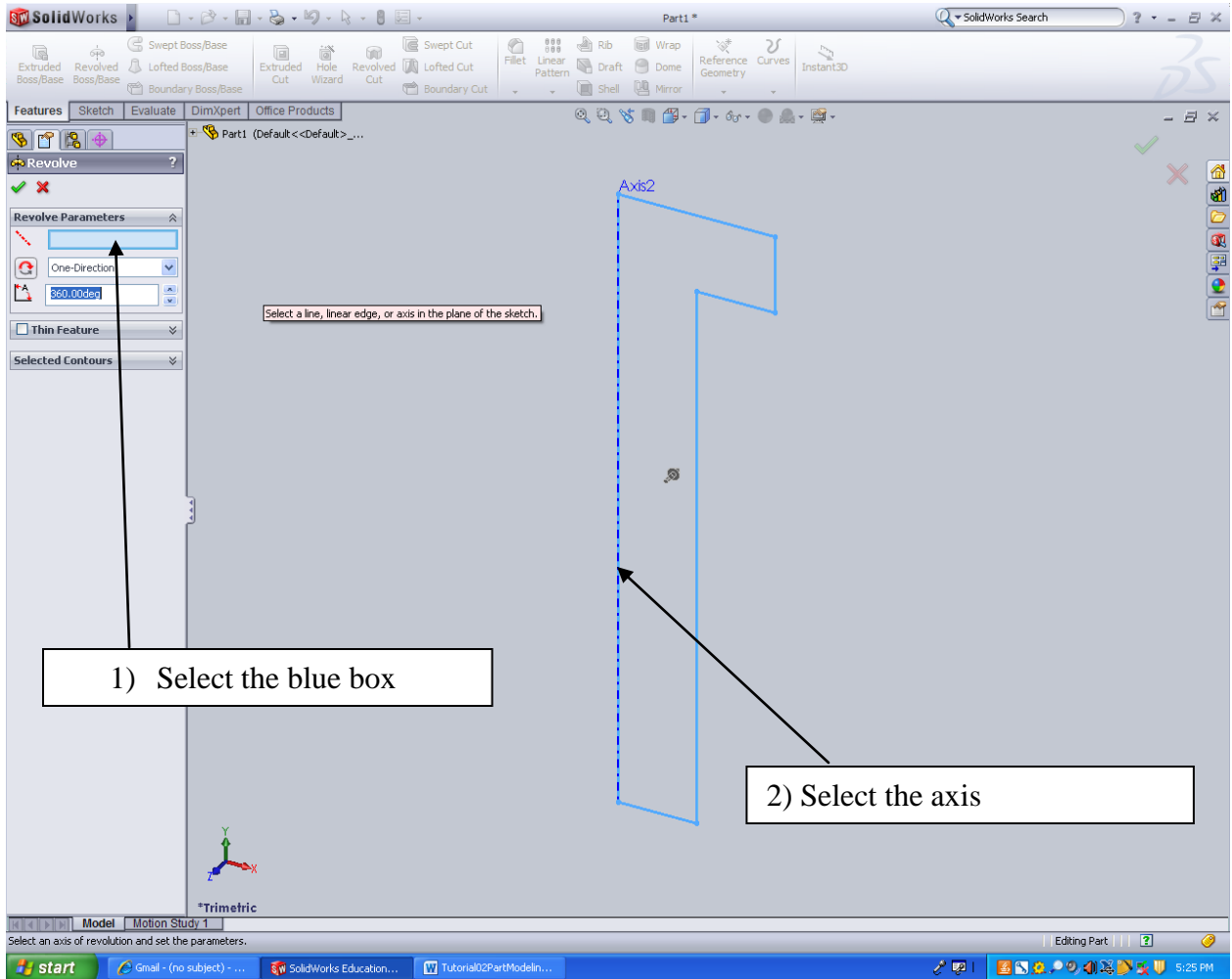


Step 11

Go to **Feature Manager** and select **Sketch**
Then select **Revolved Boss/Base** icon

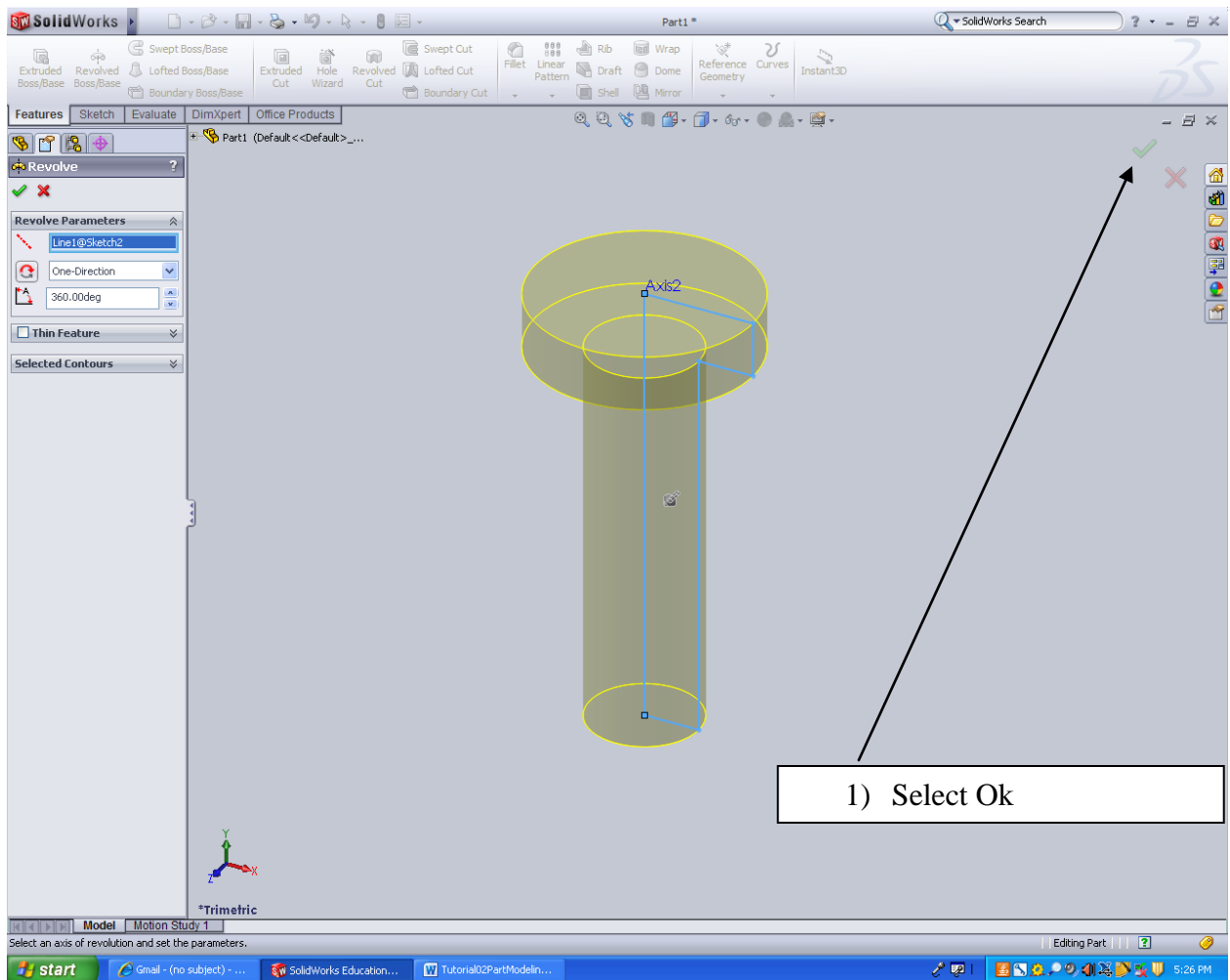


Step 12

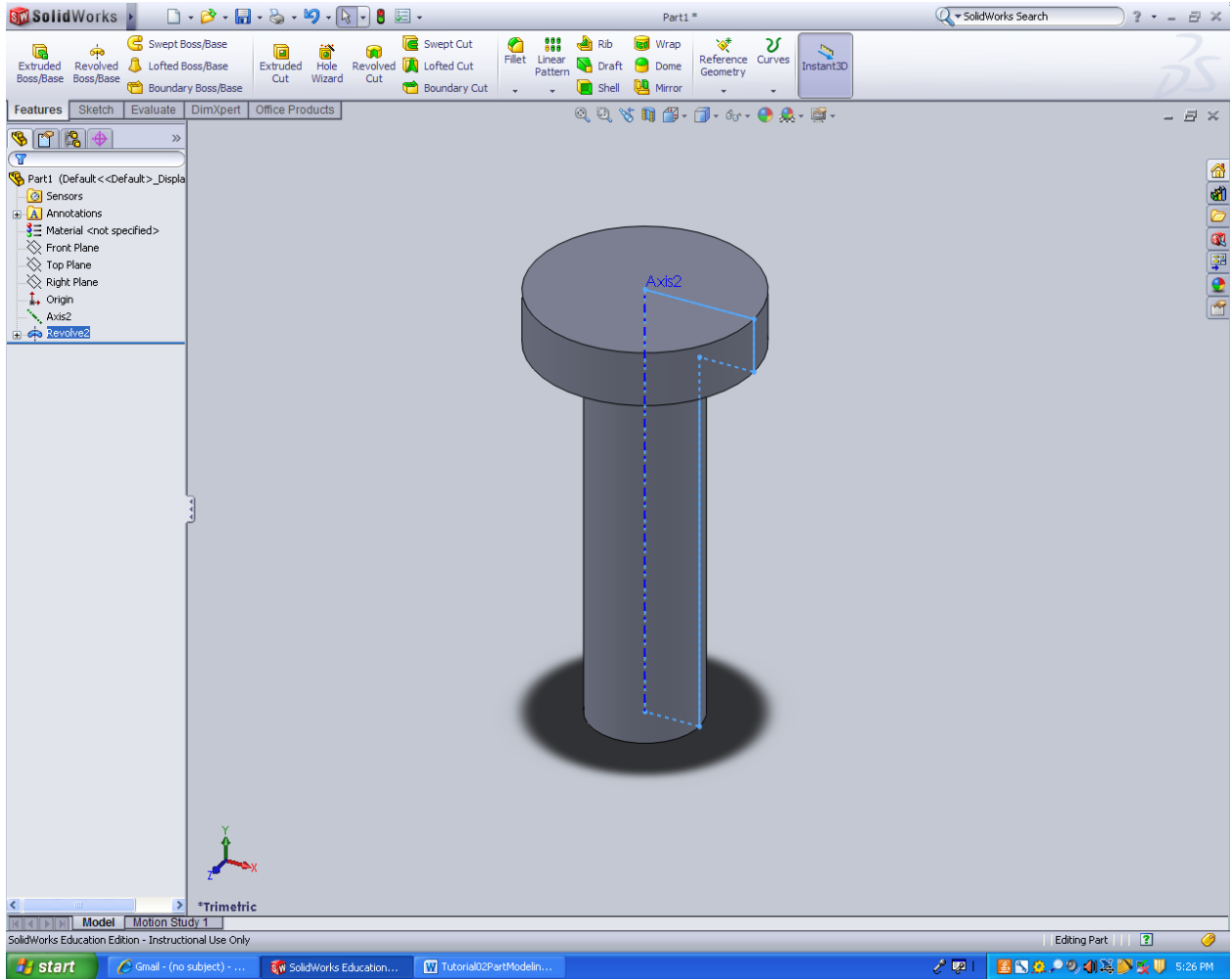


Step 13

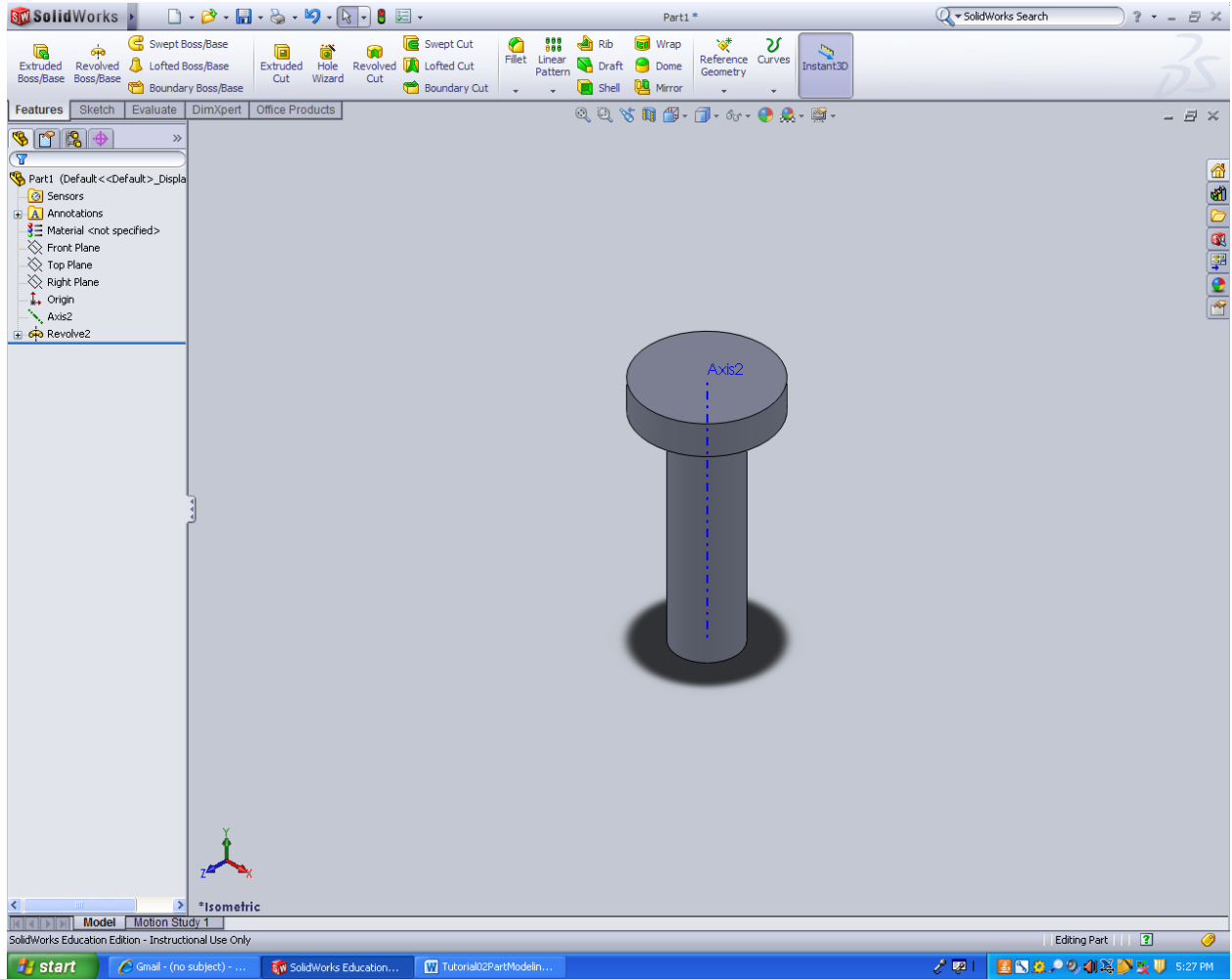
The rotated entity preview will be shown as in the image below



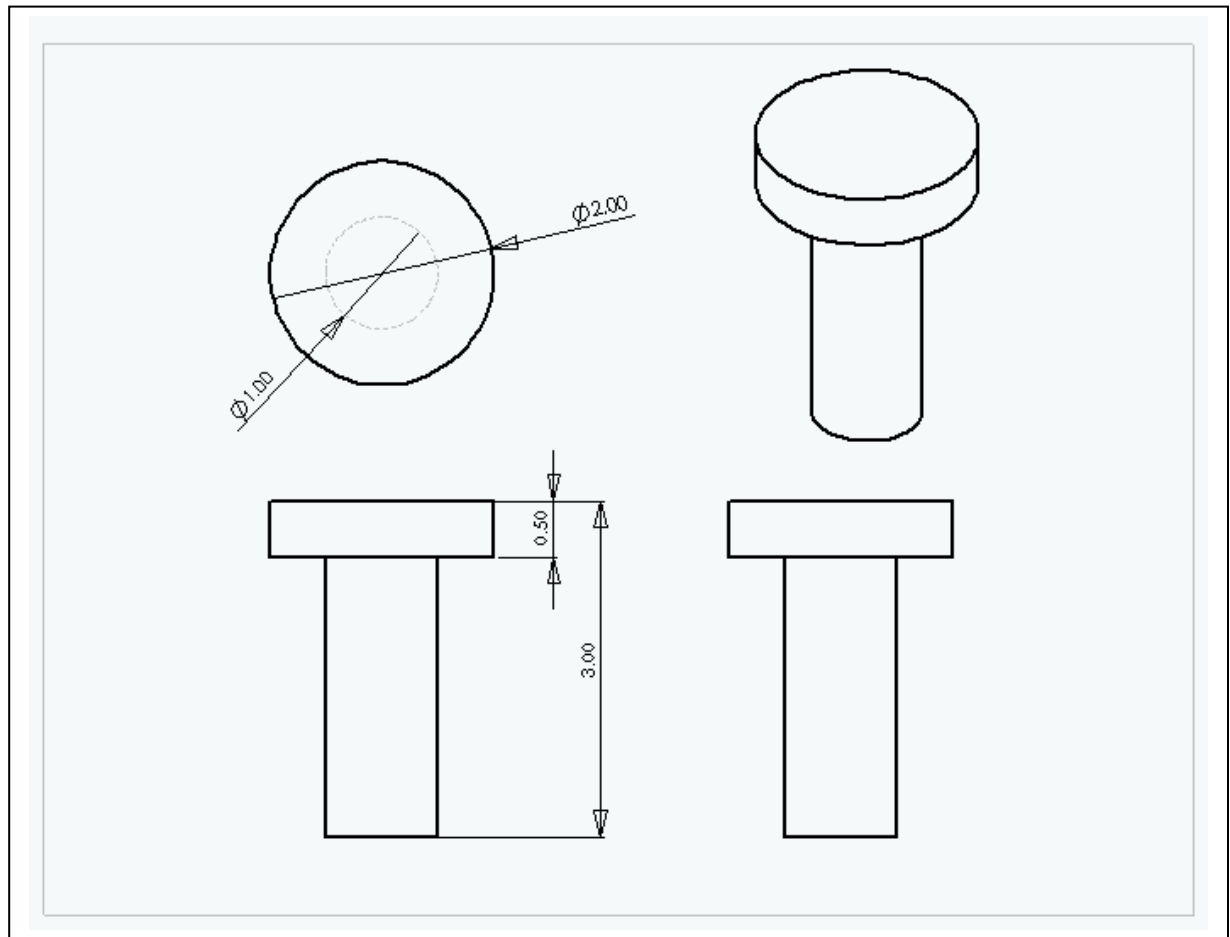
The rotated entity preview will be shown as in the image below



The rotated entity preview will be shown as in the image below



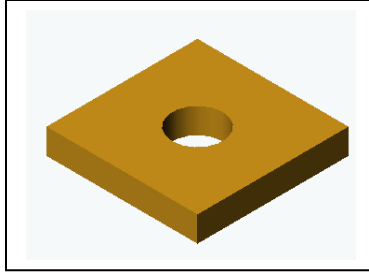
The dimensions of the pin are shown in the image below.



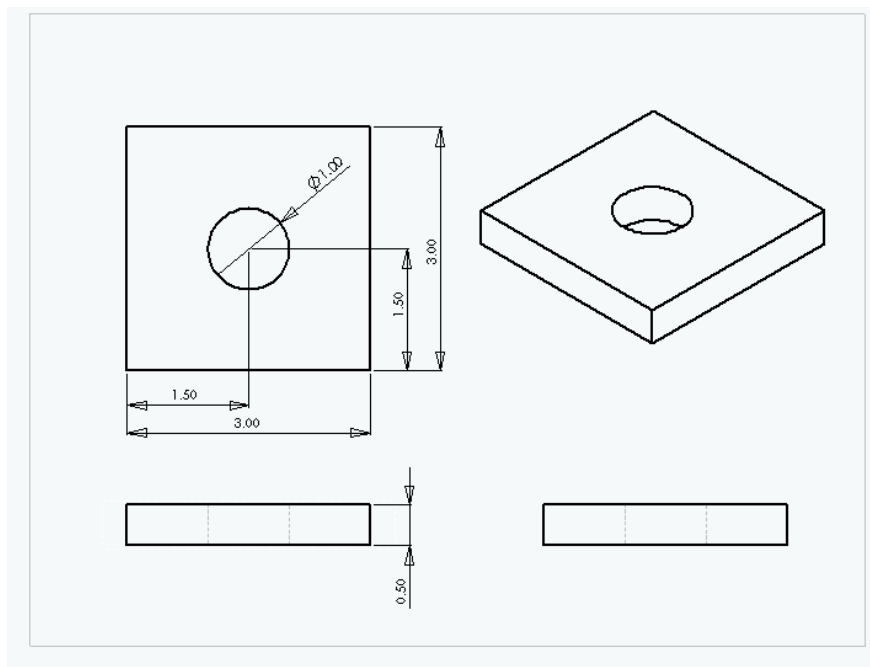
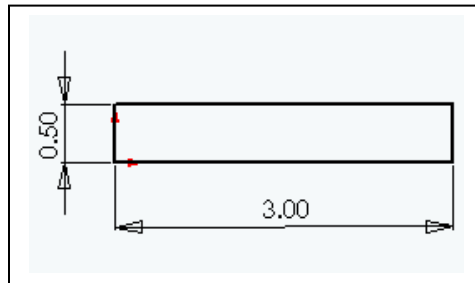
Rectangular Plate

General Planning Strategy

We shall create the part shown in the image below. Listed below the figure are the steps in brief that we must perform to create the part.

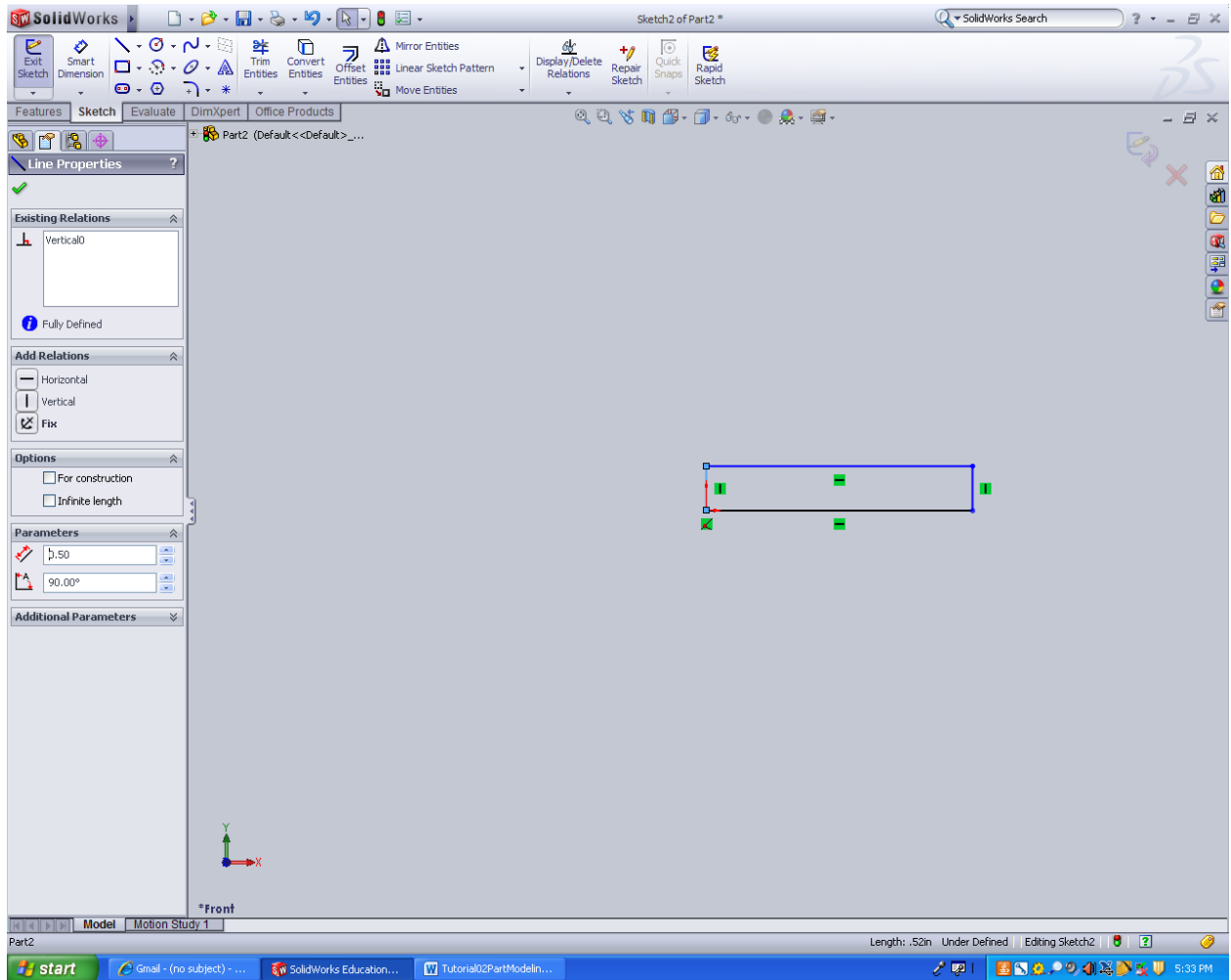


2. Create the profile and extrude it to create the plate.
3. Create the hole.



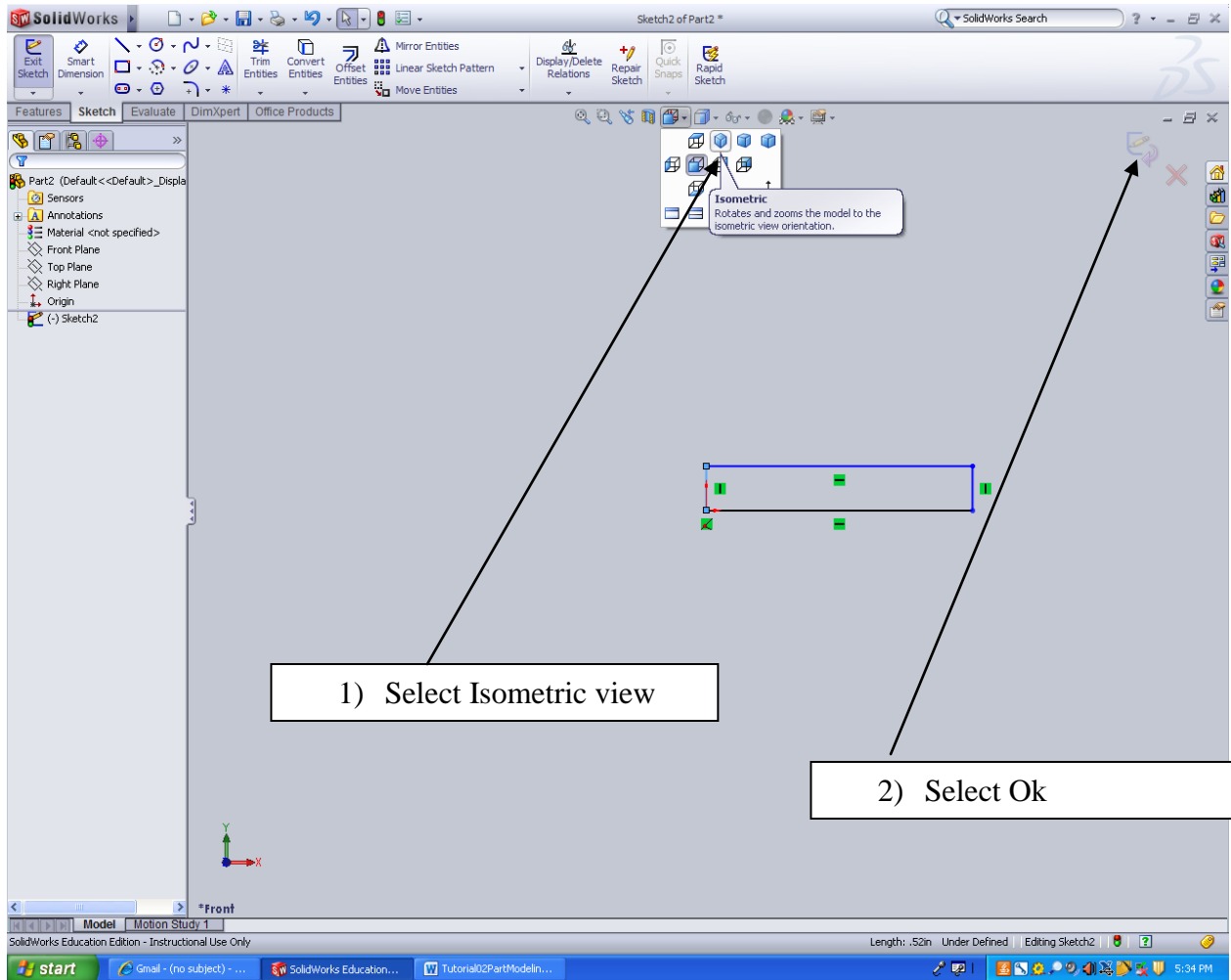
Step 1

Create the part file and draw the rectangle according to required dimensions. For creating part file and drawing refer to the pin tutorial.



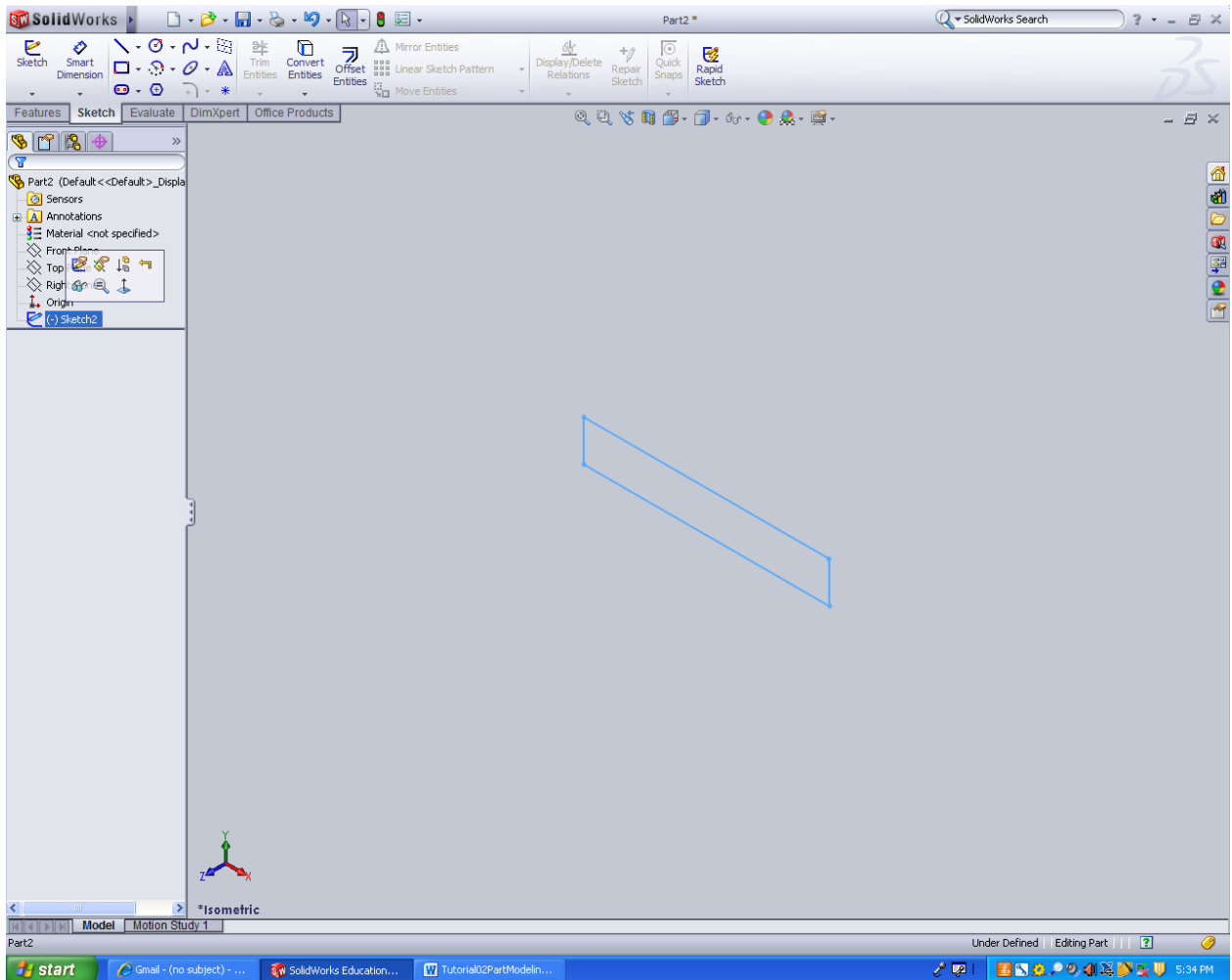
Step2

After completing the drawing change the view to isometric and select OK.



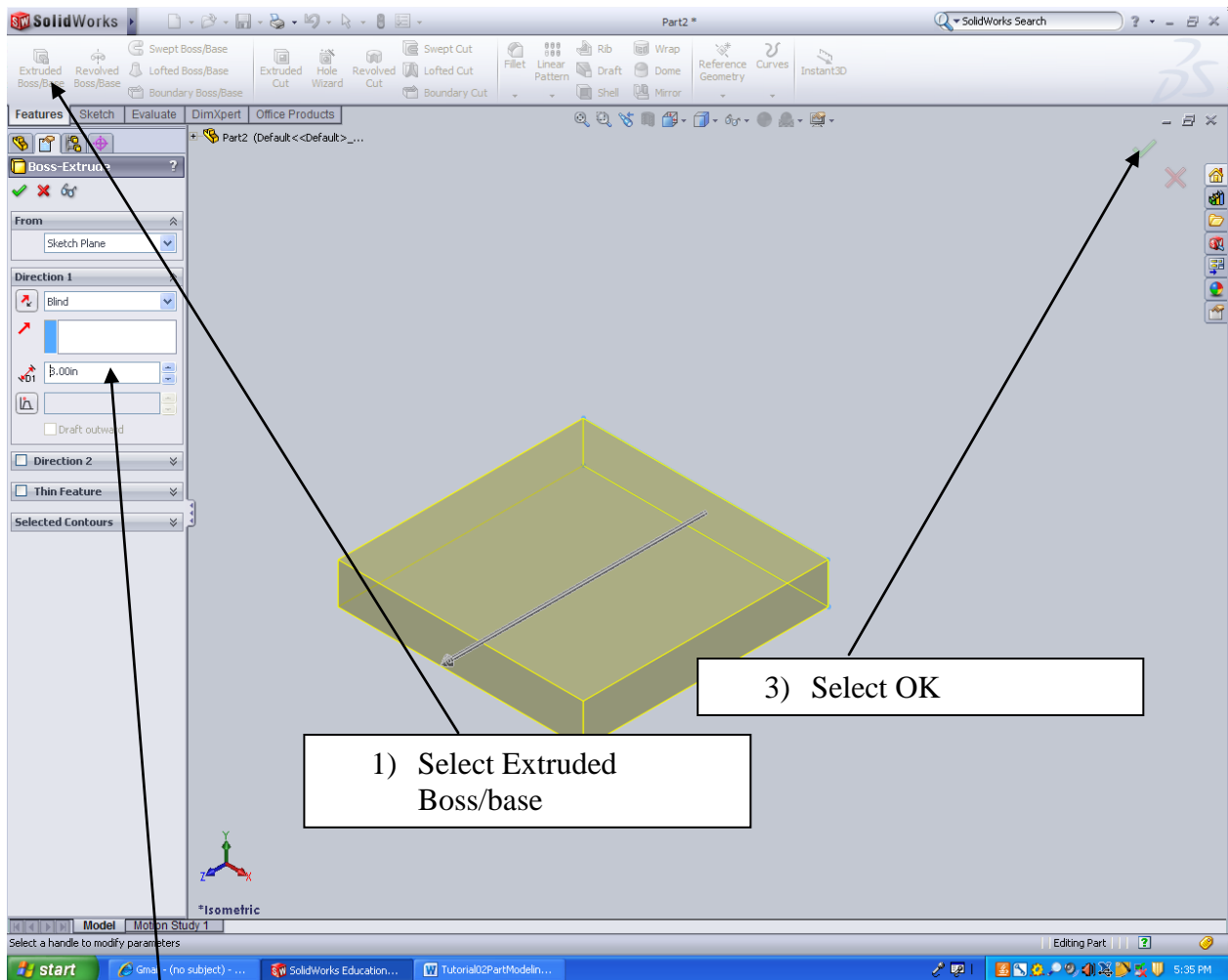
Step 3

Select sketch from the **Feature Manager**.



Step 4

Select **Extruded Boss/Base** from Features Tab. Enter the dimension as 3 inches.

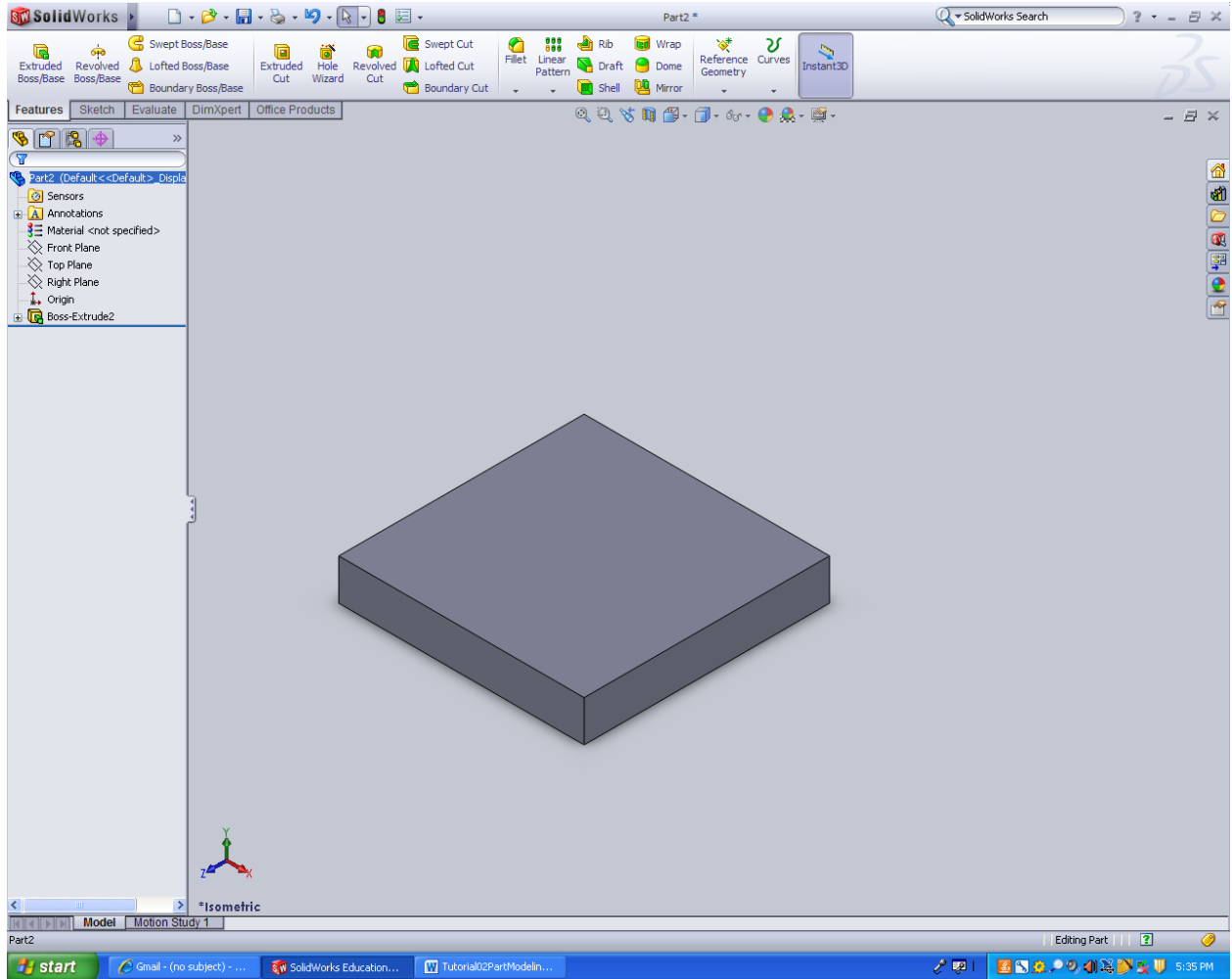


1) Select Extruded Boss/base

2) Enter dimension as 3 inches

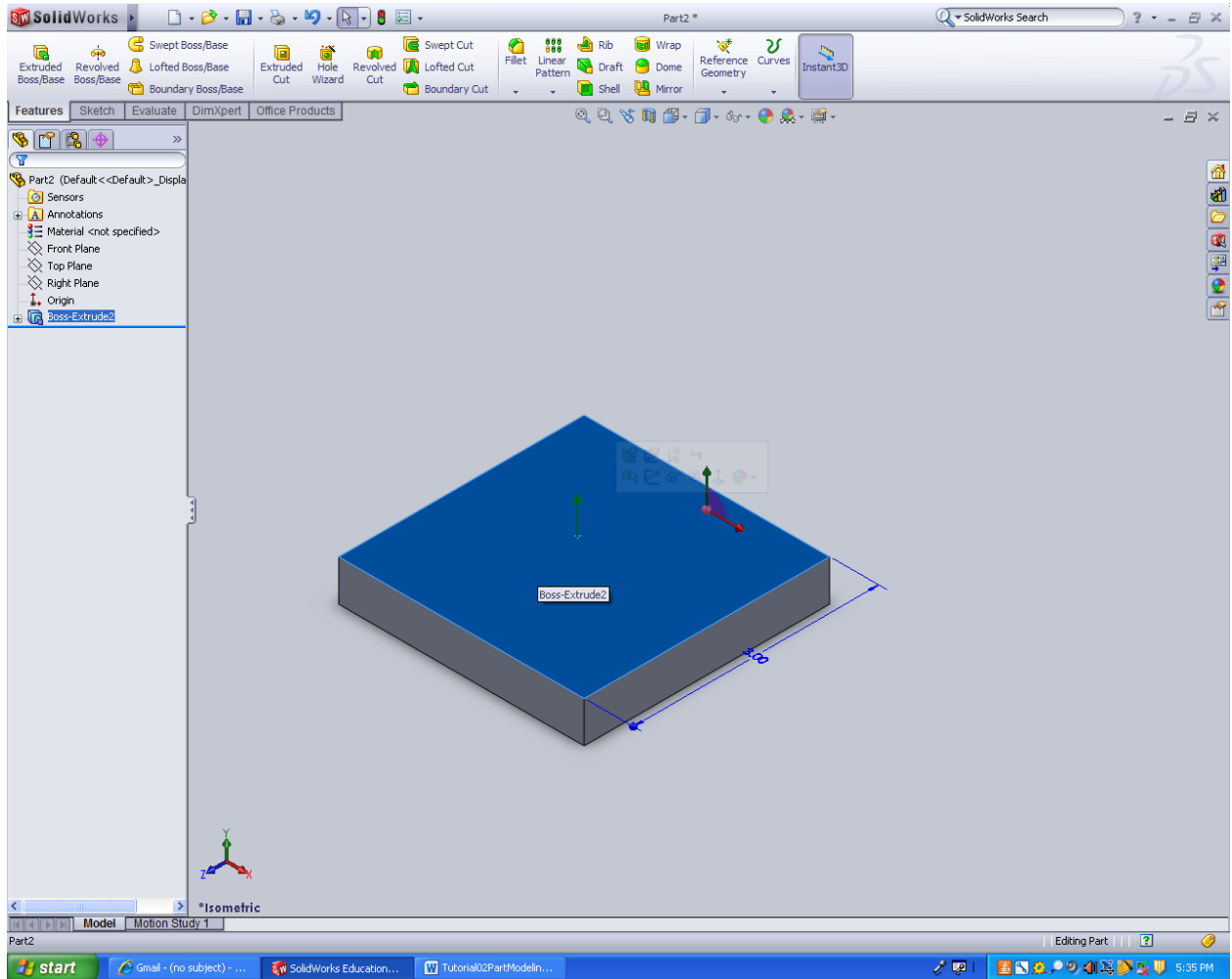
3) Select OK

Preview of the rectangular plate.



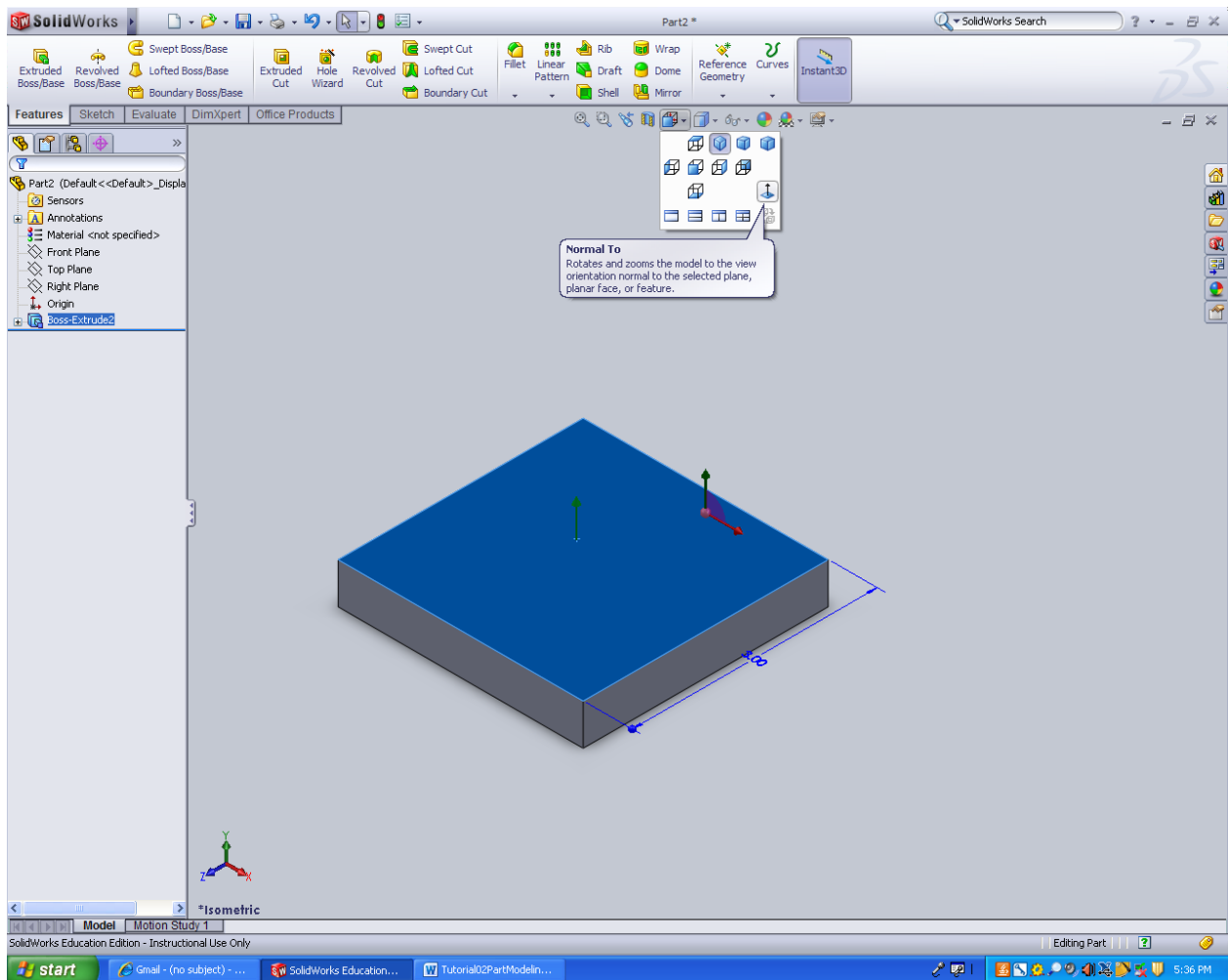
Step 5

Select the top plane of the rectangular plate.



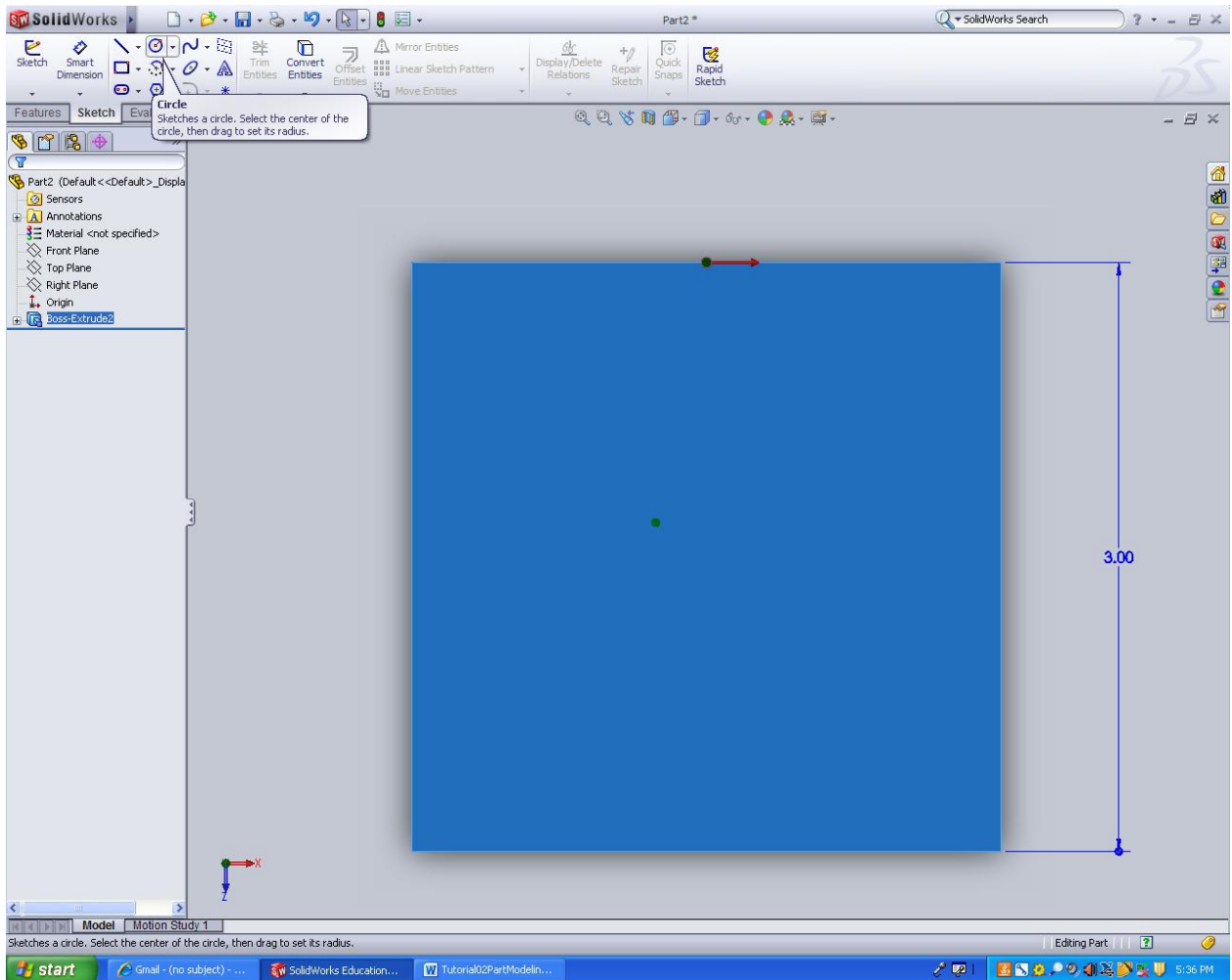
Step 6

Go to **View Icon** and select **Normal To** option



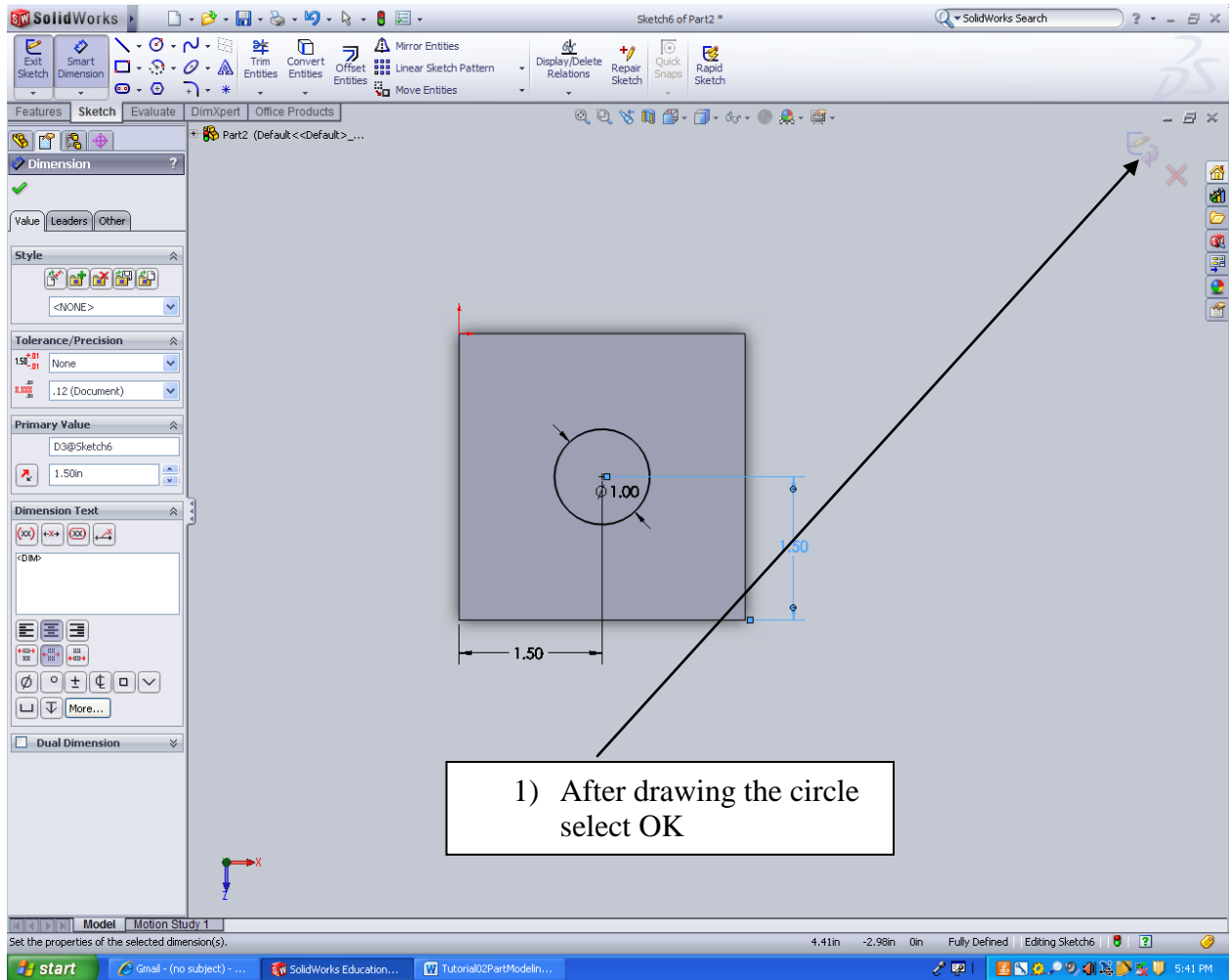
Step 7

Select **Circle Icon**.

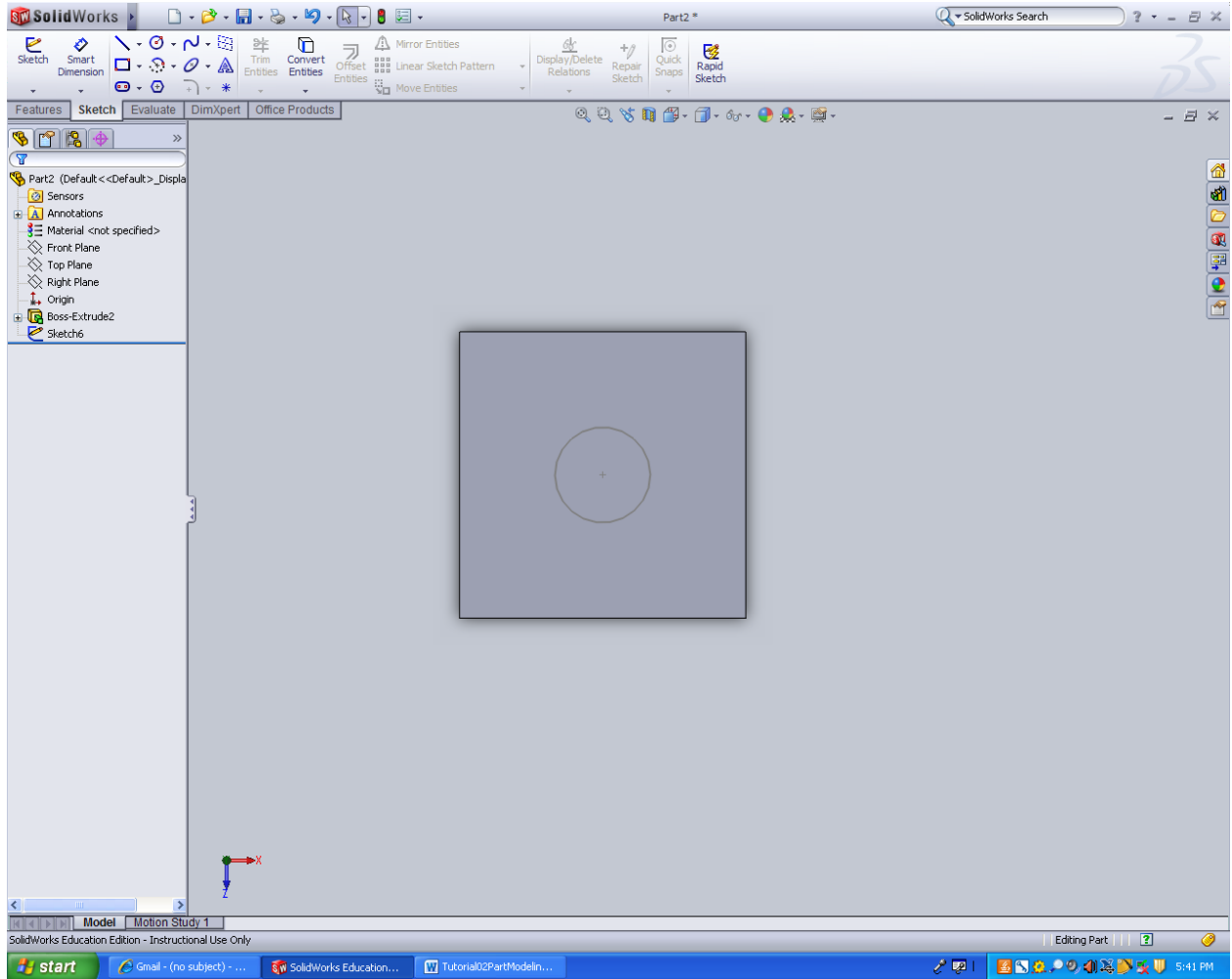


Step 8

Draw a circle of diameter 1 inch. Adjust the placement of the circle using smart dimension option. Tutorial 1 has the steps on how to use the smart dimension option.

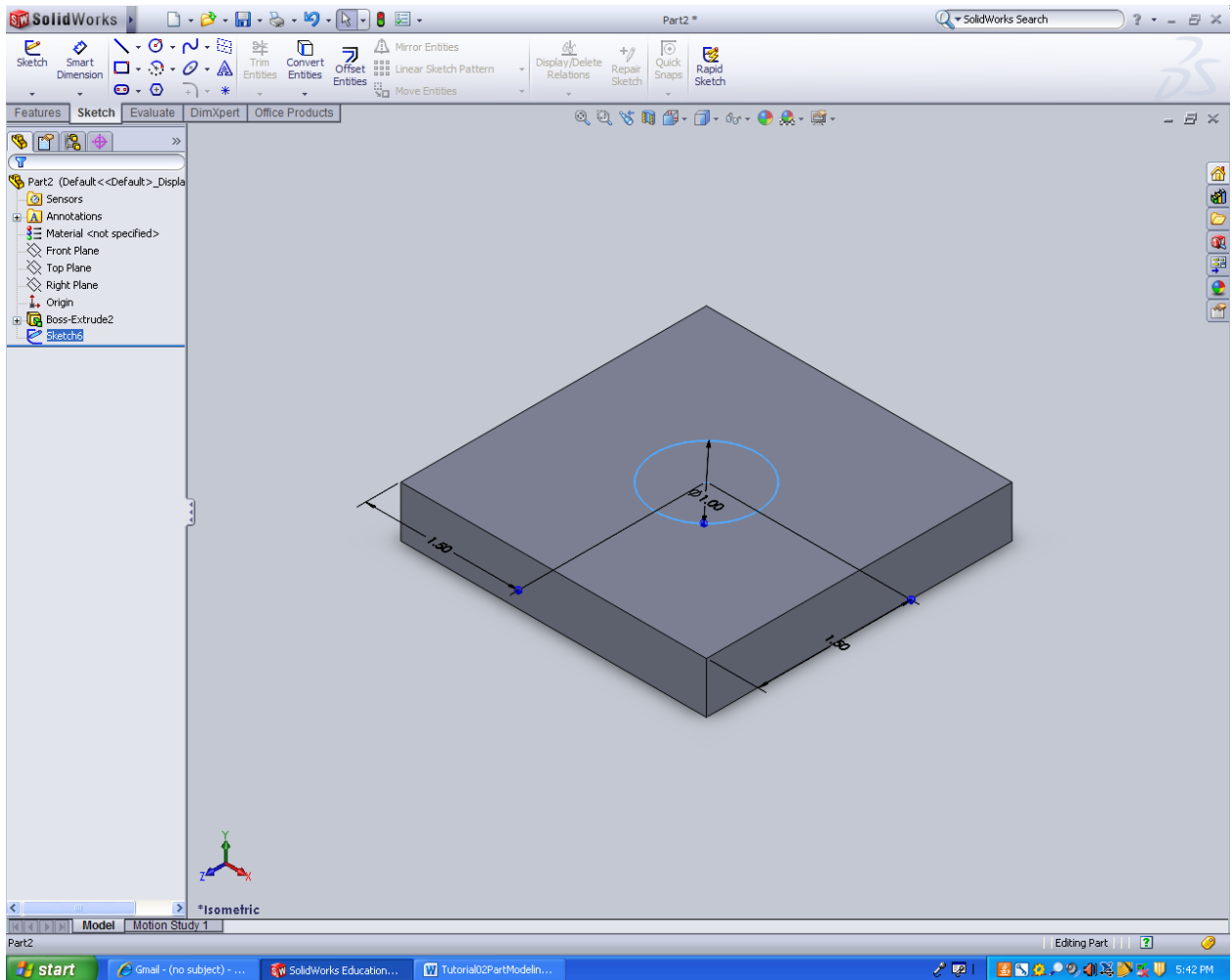


Preview of the Rectangular plate.



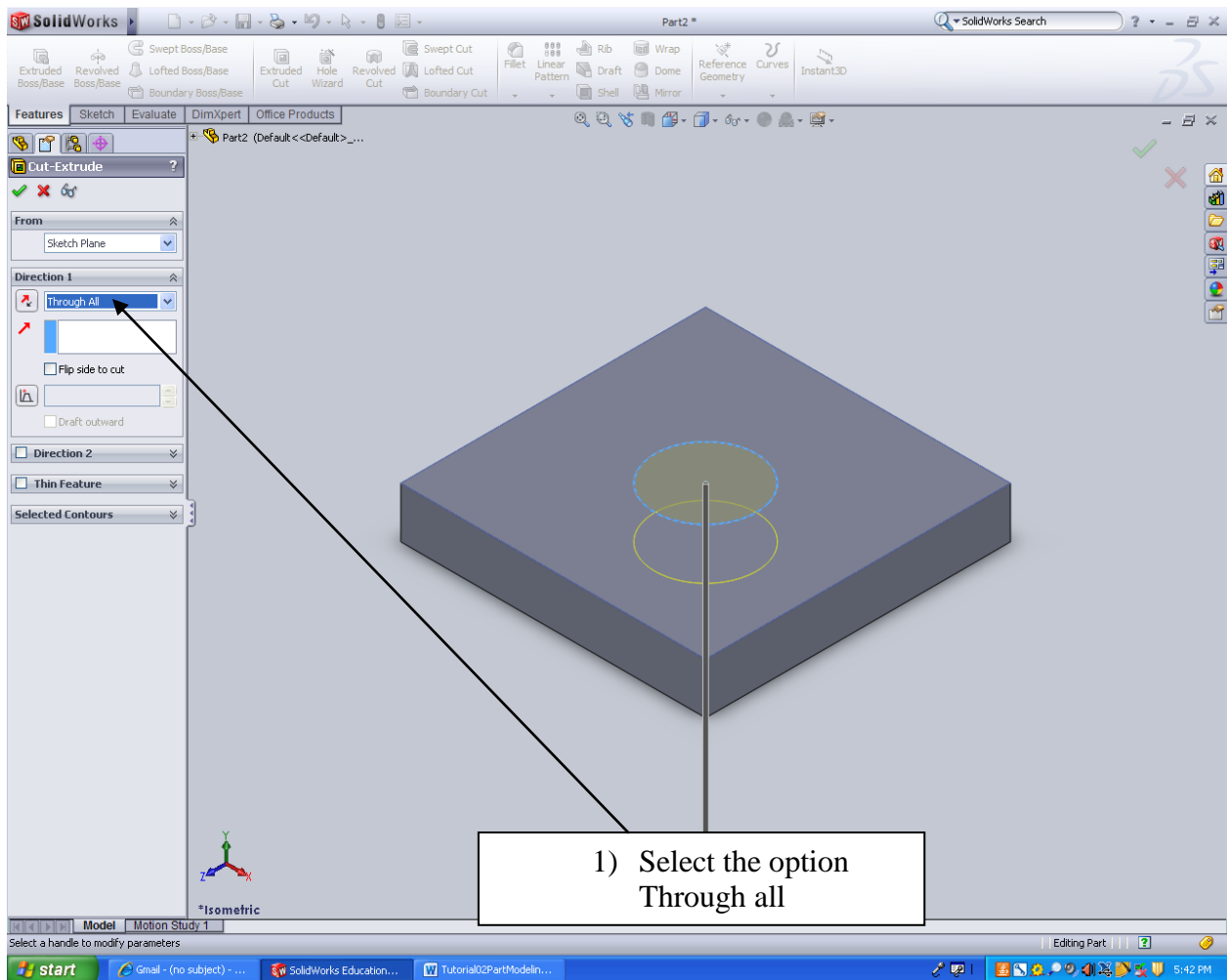
Step 9

Select sketch from the **Feature Manager**.

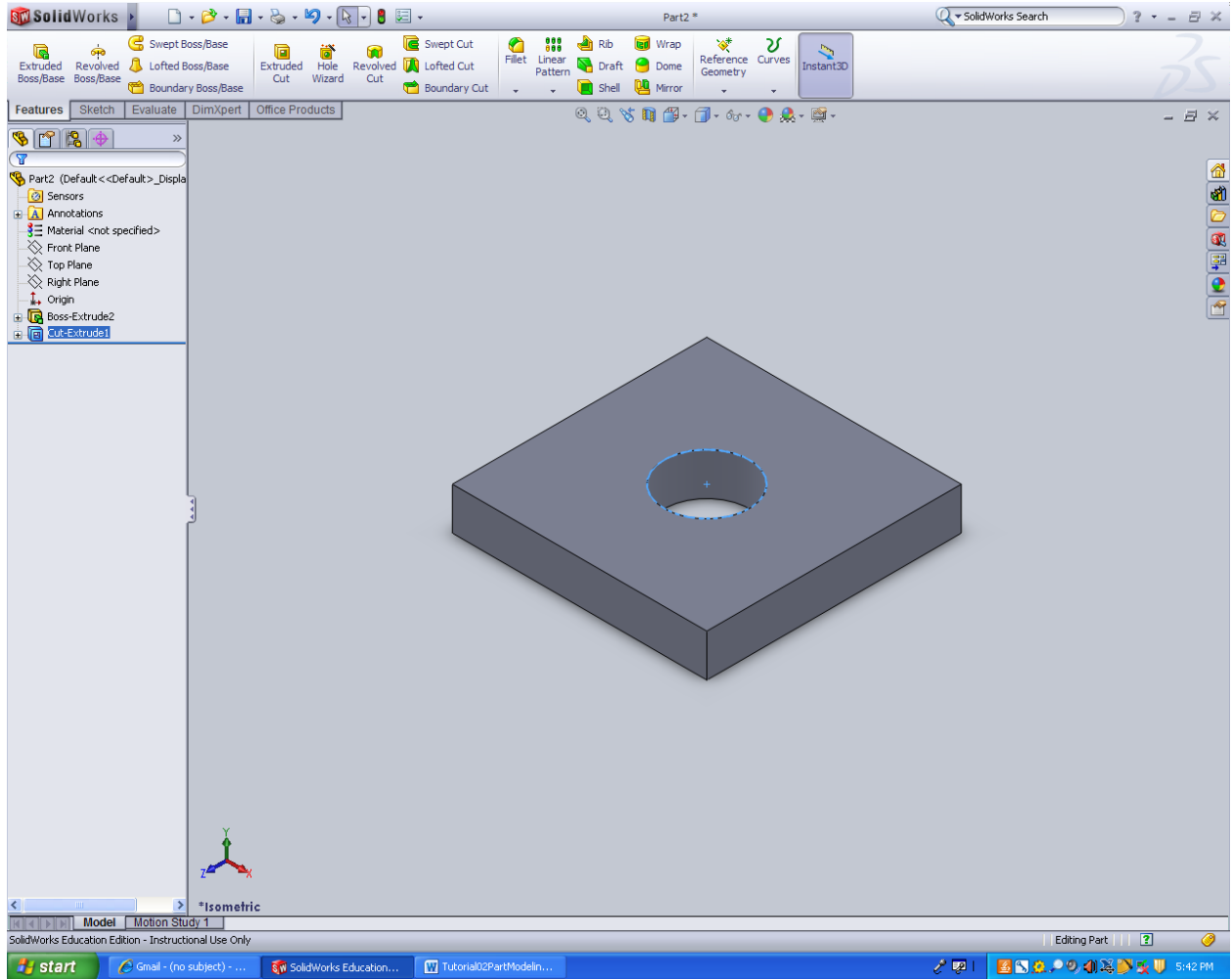


Step 10

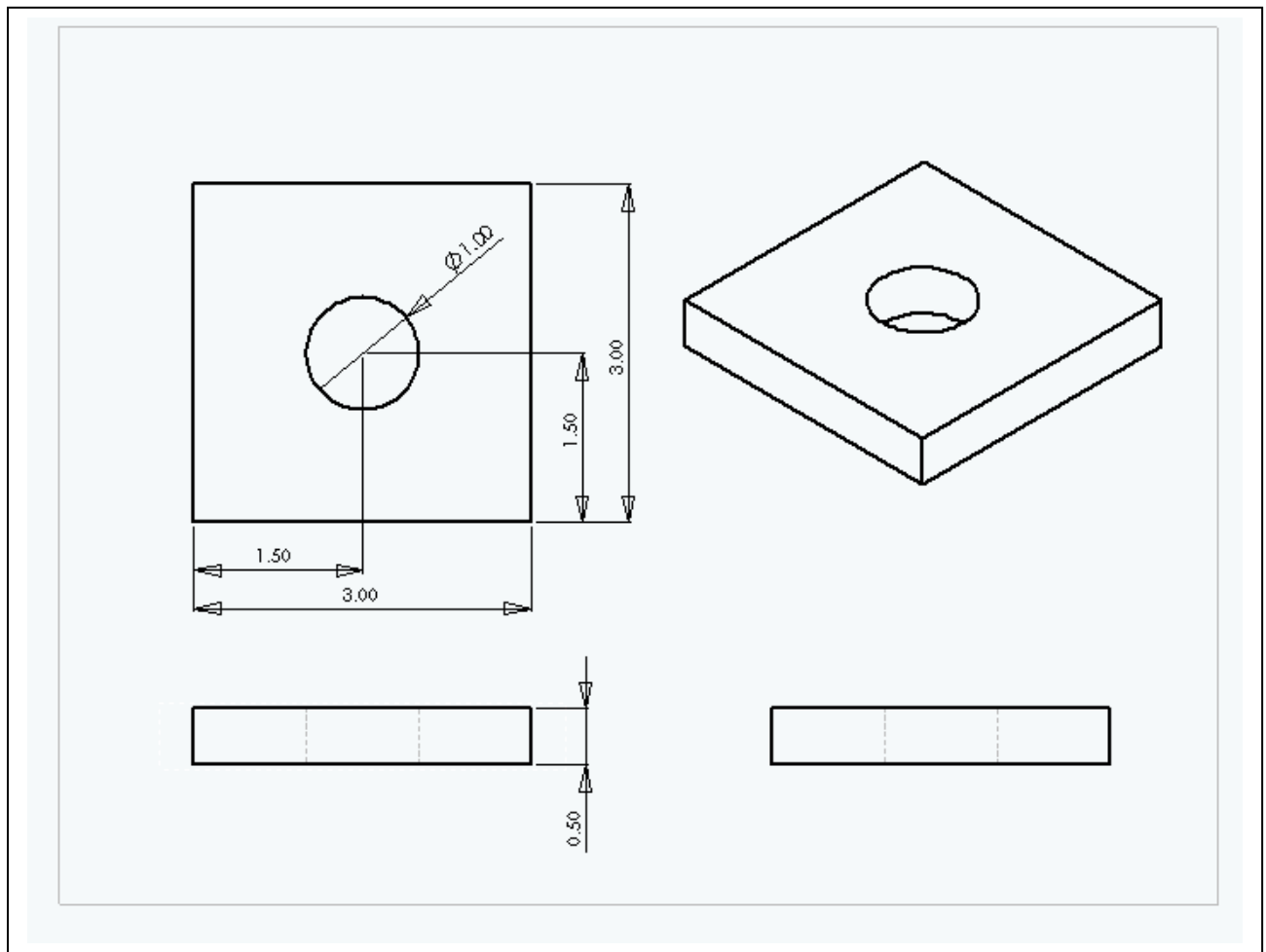
Go to Features **Tab**→**Extruded Boss/base**. Select the option **Through All**.



Preview of the completed Rectangular Plate.



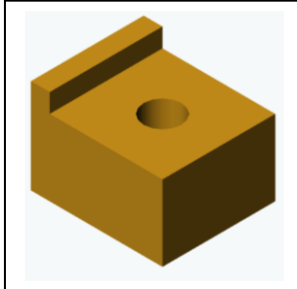
This completes the model. Shown below is the dimensioned drawing view of the model that we just created.



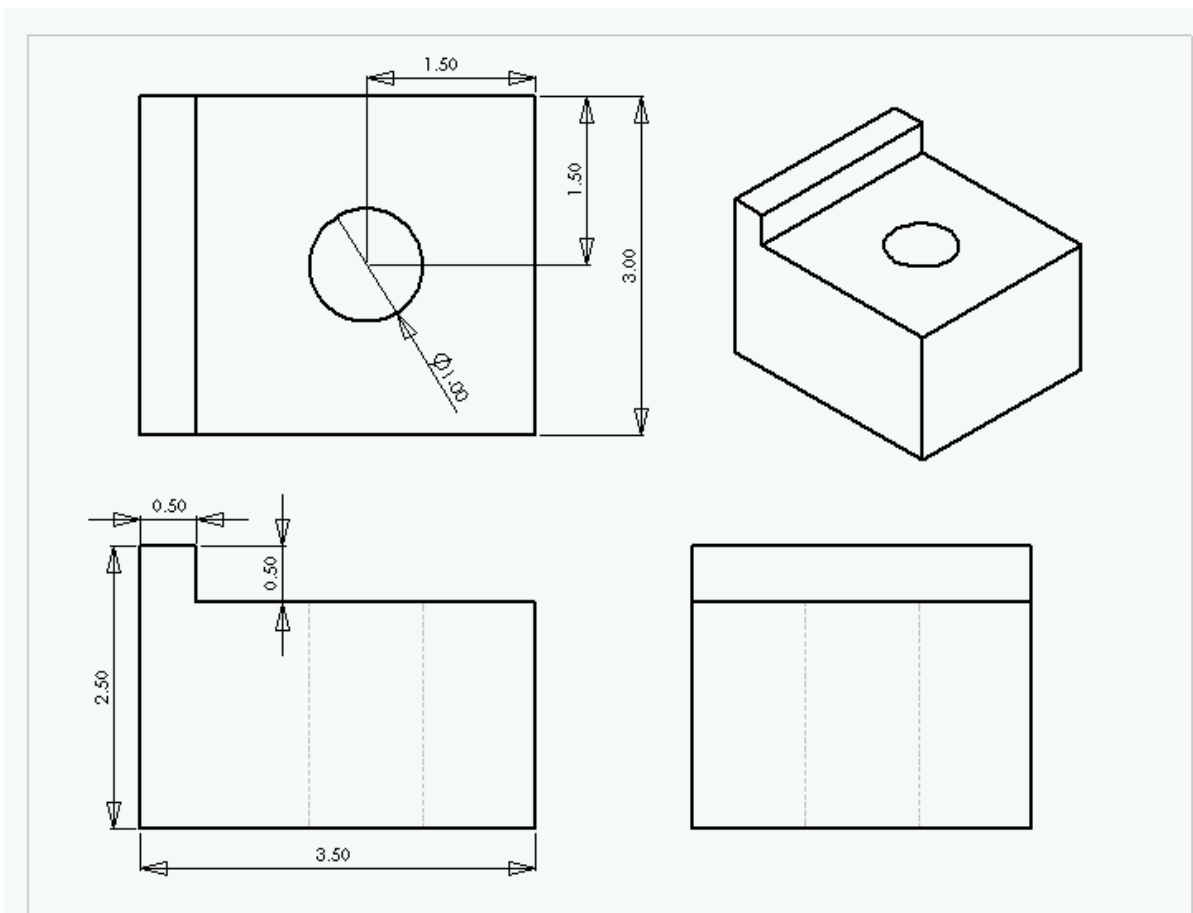
Base Plate

General Planning Strategy

We shall create the part shown in the image below. Listed below the figure are the steps in brief that we must perform to create the part.

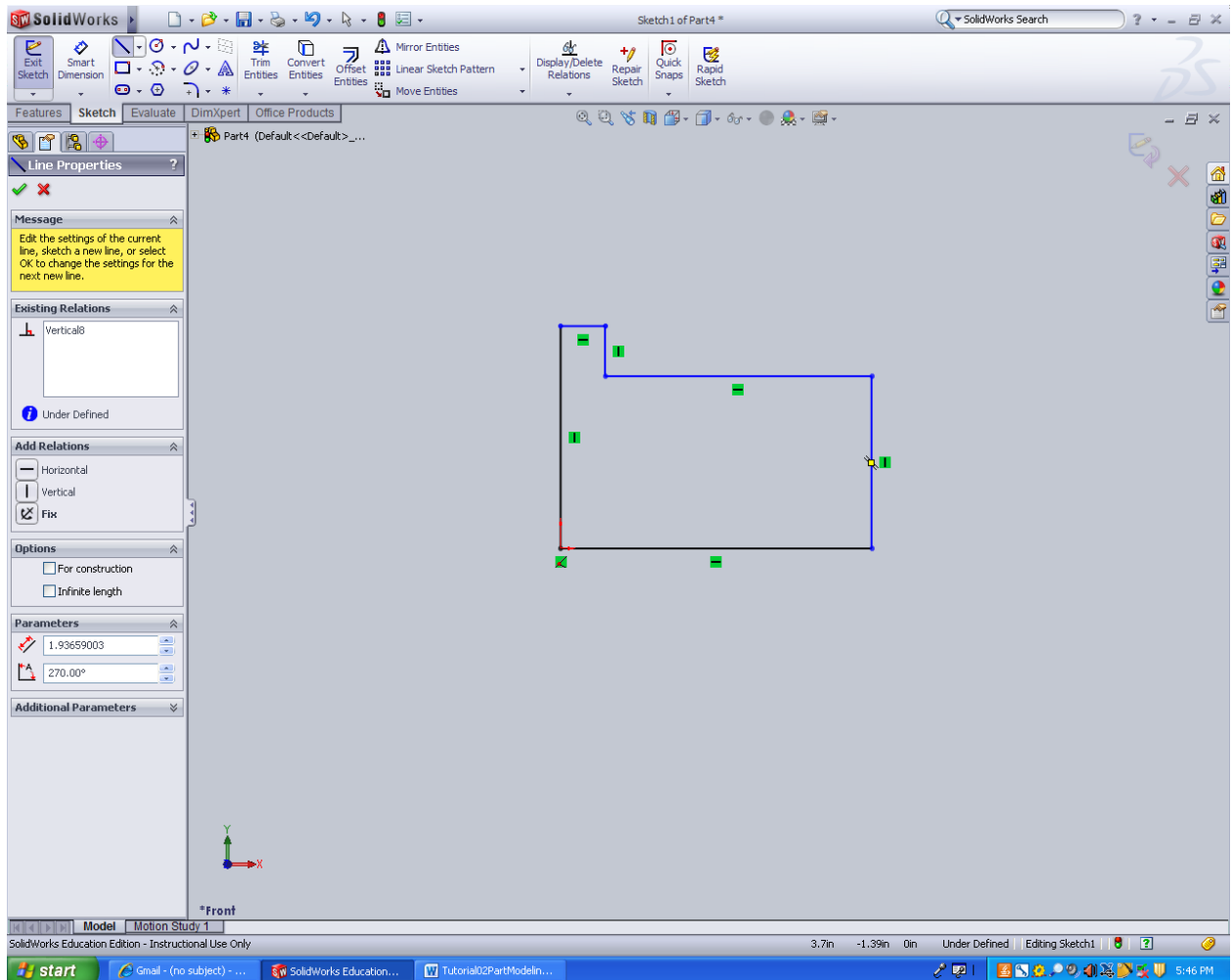


4. Create the profile and extrude it to create the plate.
5. Create the hole.



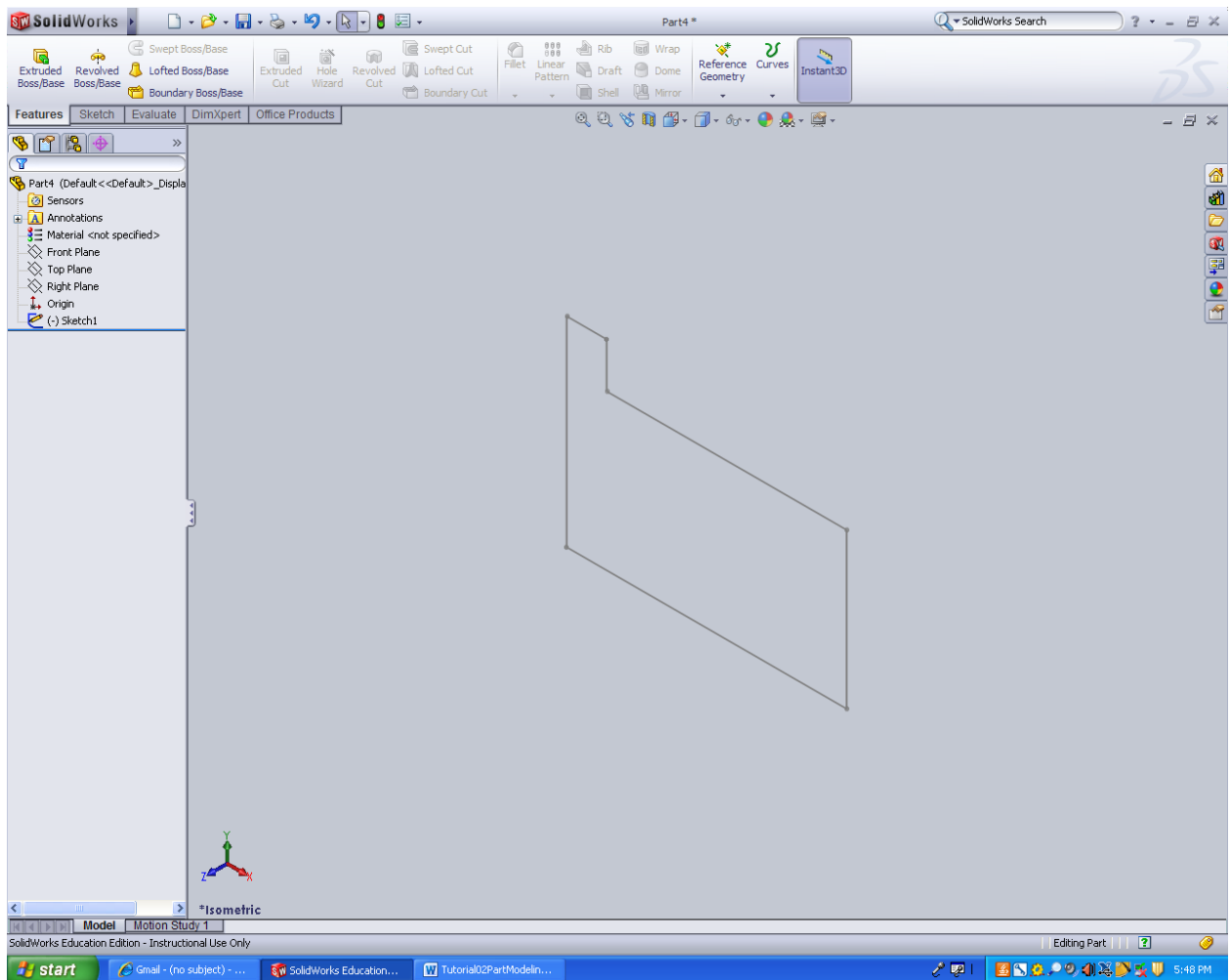
Step 1

Create the part file and draw the figure according to required dimensions. For creating part file and drawing refer to the pin tutorial.



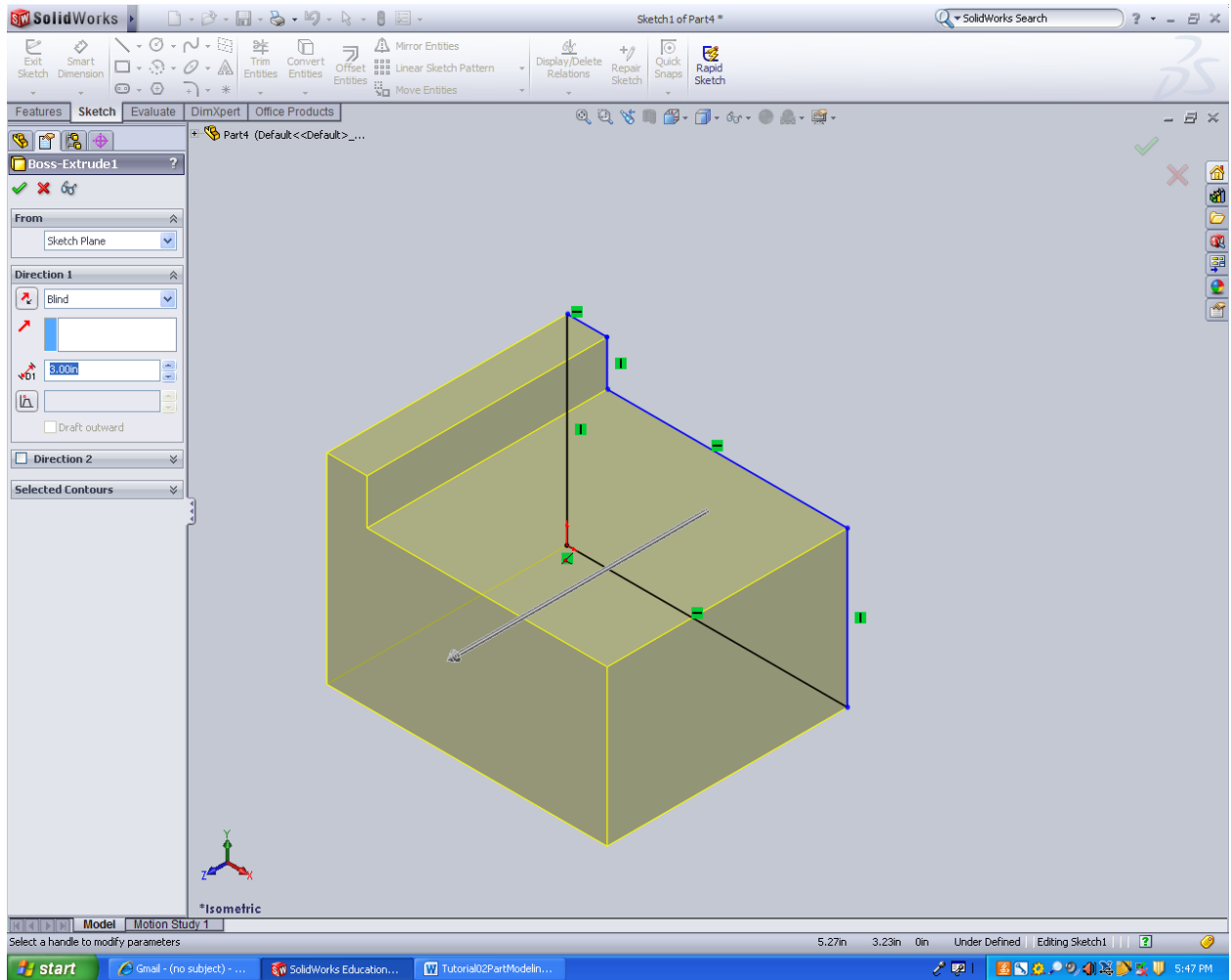
Step 2

View the drawing in **Isometric View**. Select sketch from the **Feature Manager**.

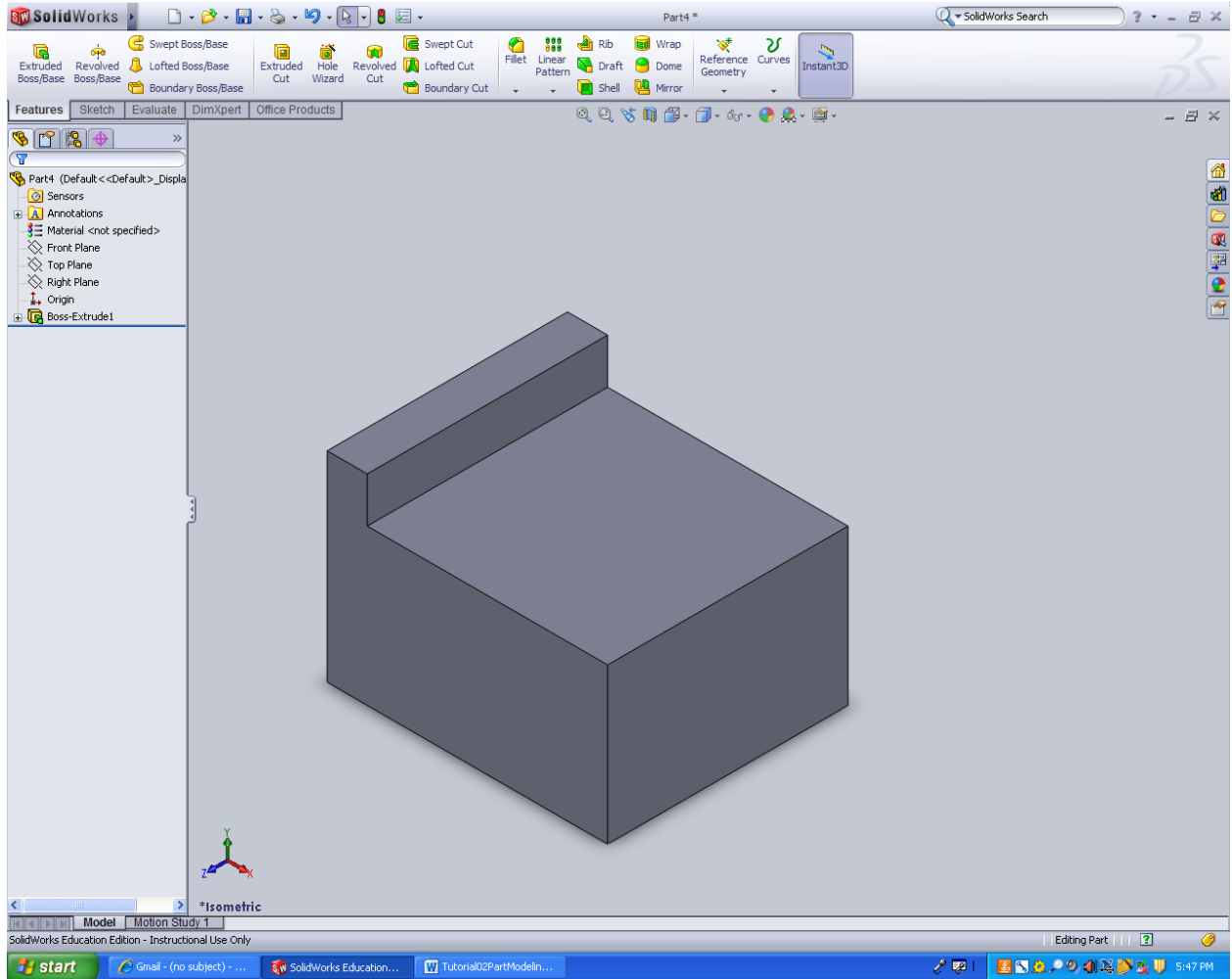


Step 3

Go to Features **Tab** → **Extruded Boss/base** and extrude the figure to a length of 3 inches. Select Ok (green tick mark on top right corner) to complete extrusion.

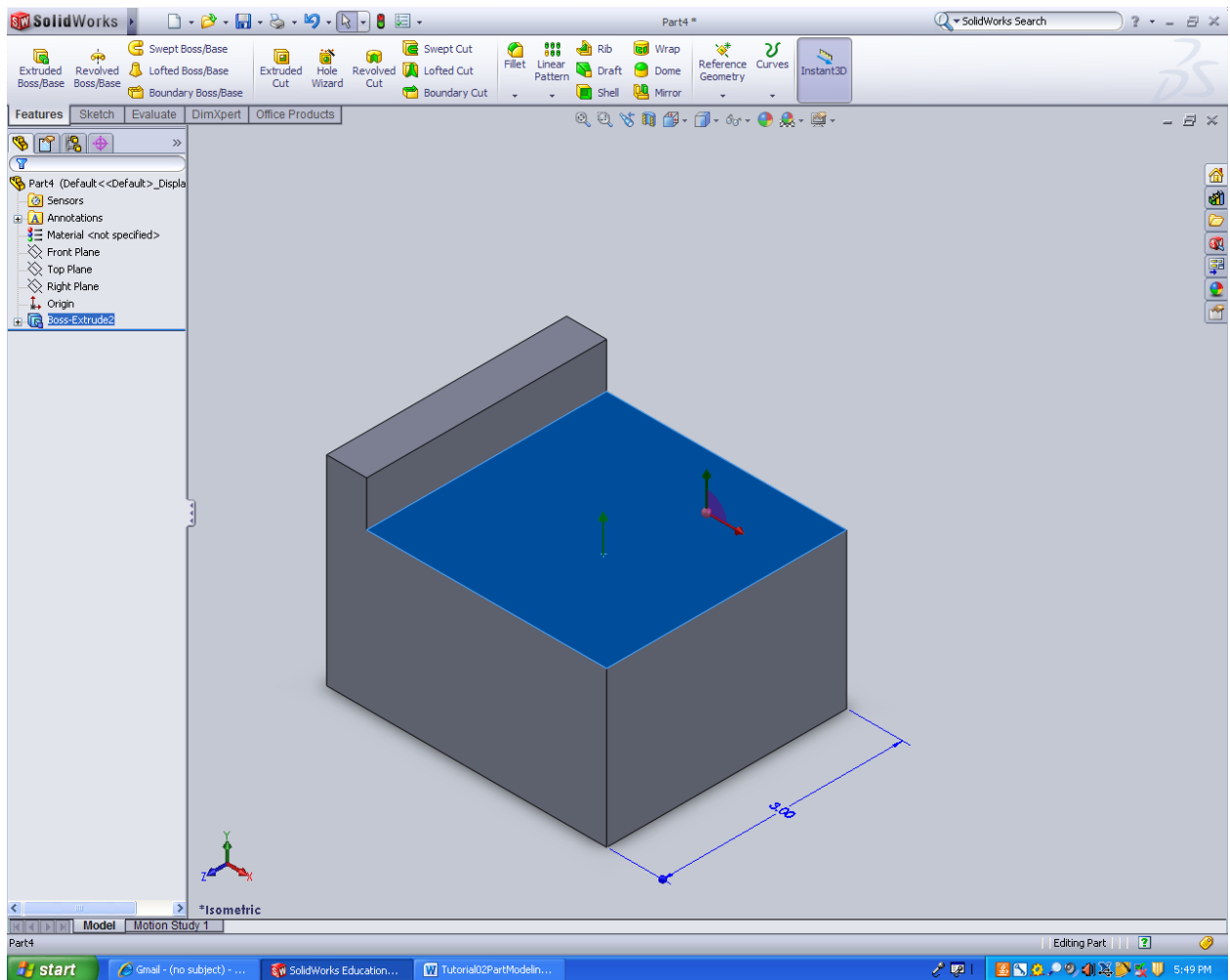


Preview of the extruded part.



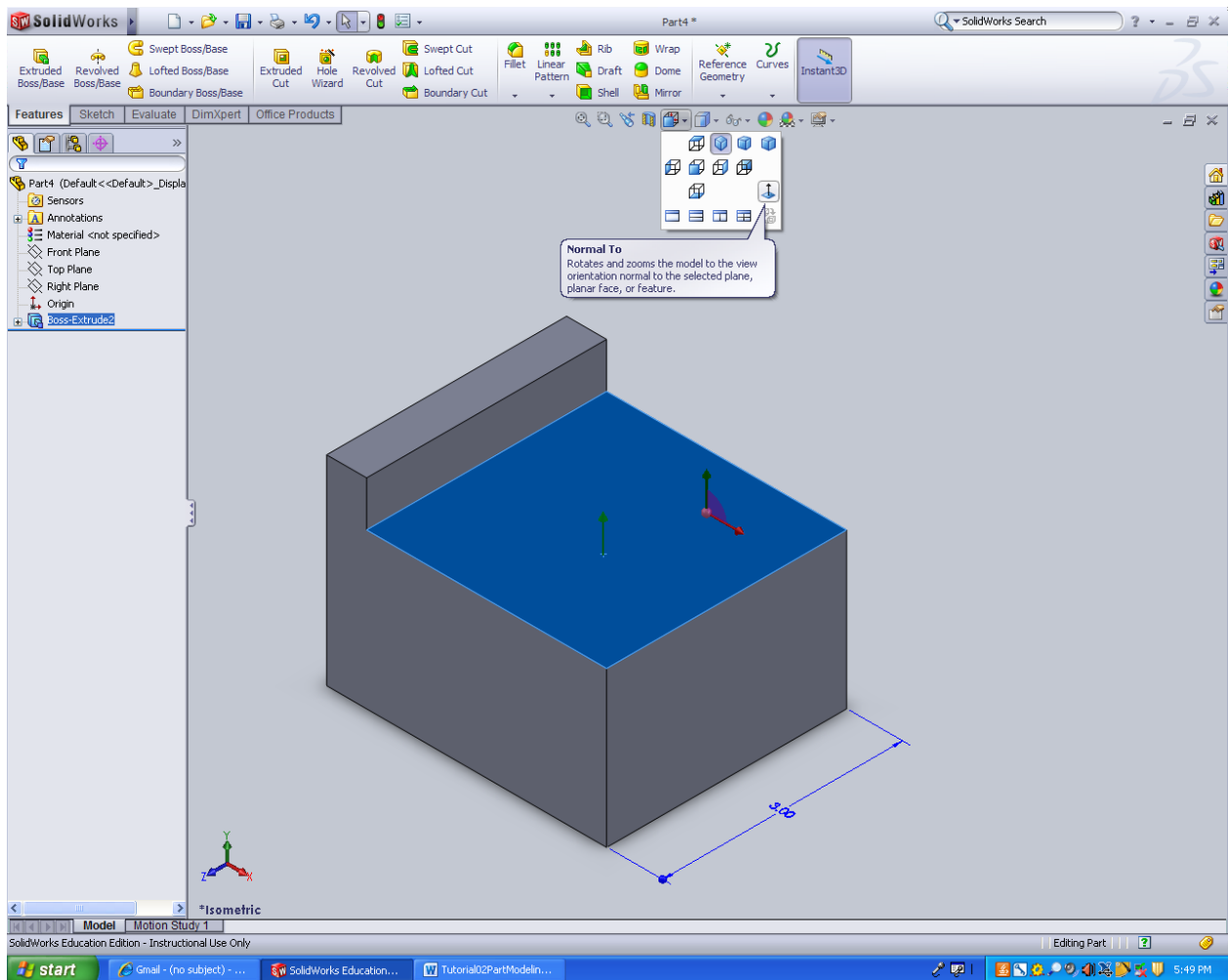
Step 4

Select the plane as shown in the figure below.



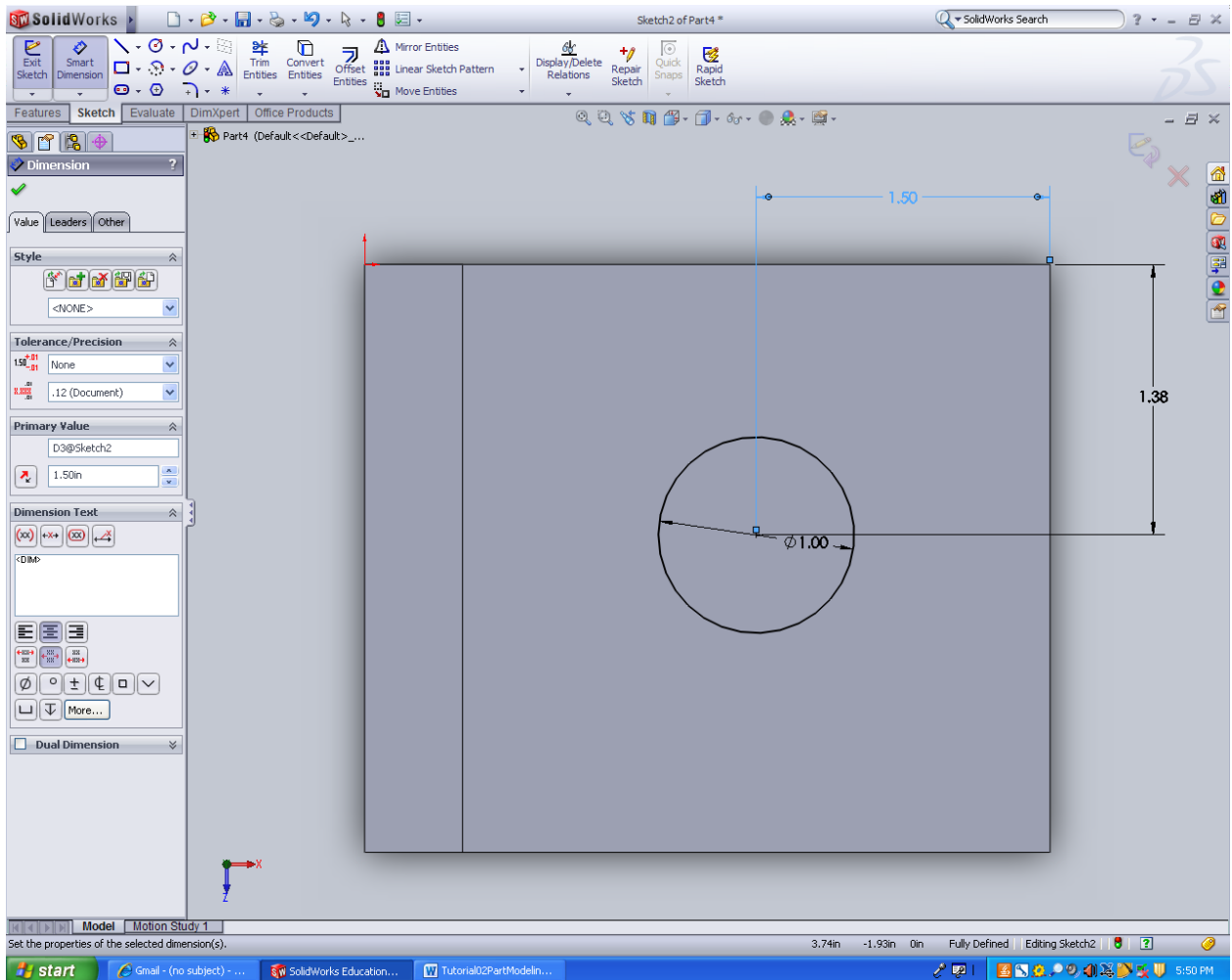
Step 5

Select **Normal To** option from the **View Icon**.



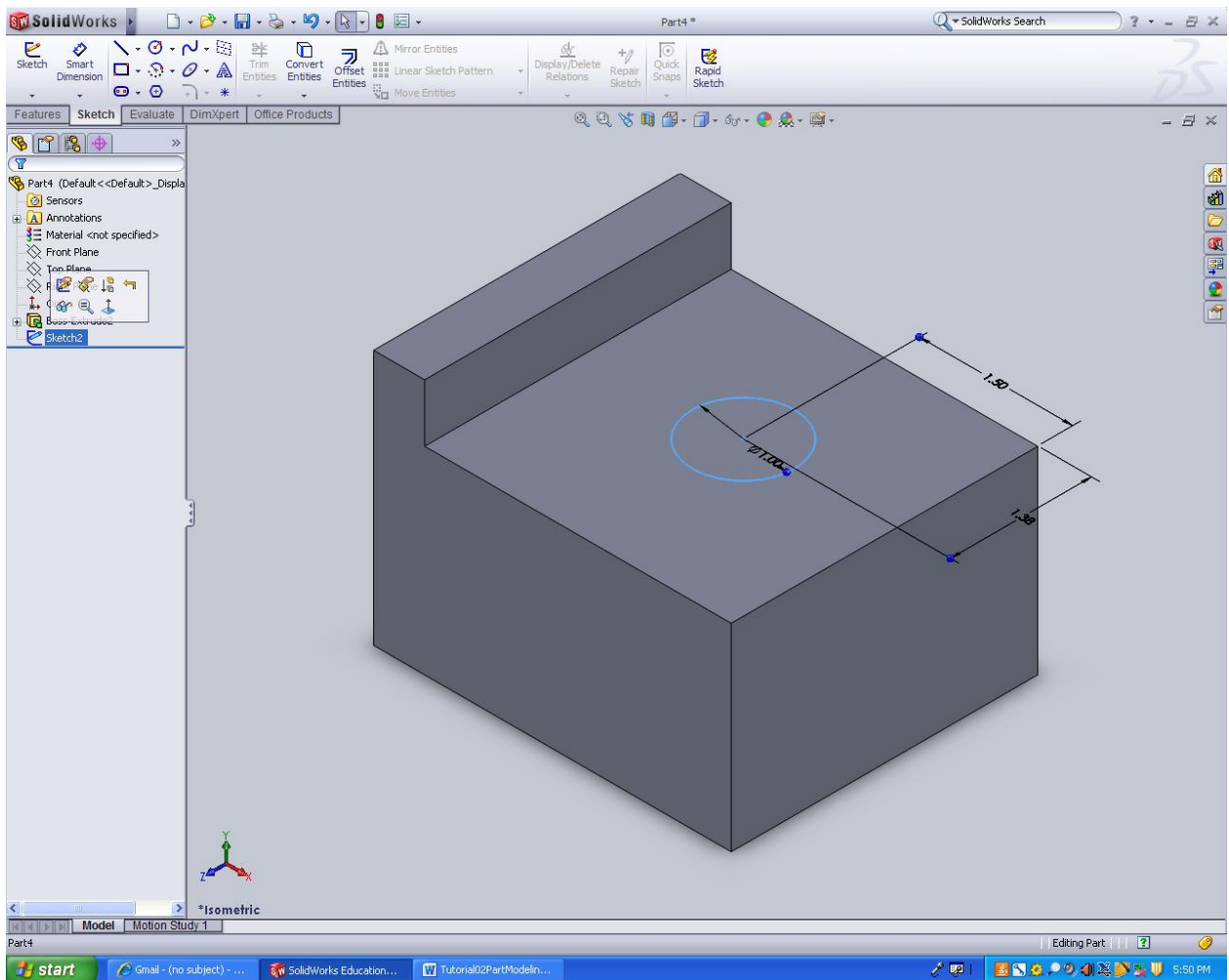
Step 6

Draw a circle of diameter 1 inch. Position the circle using smart dimension option. Follow the steps given in the tutorial for rectangular plate on how to use the smart dimension option. Select OK to complete the process.



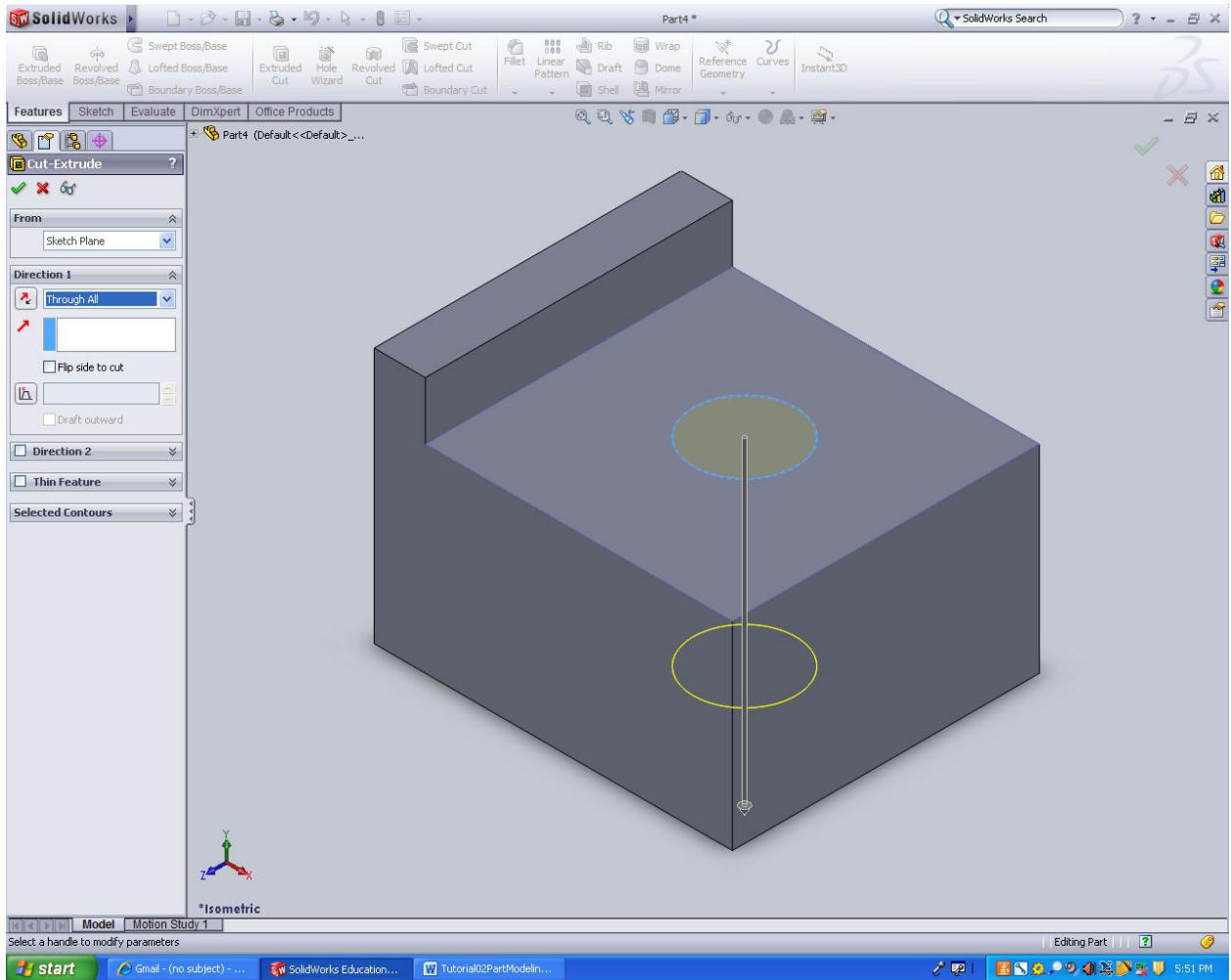
Step 7

Select the circle sketch from the **Feature Manager Window**.

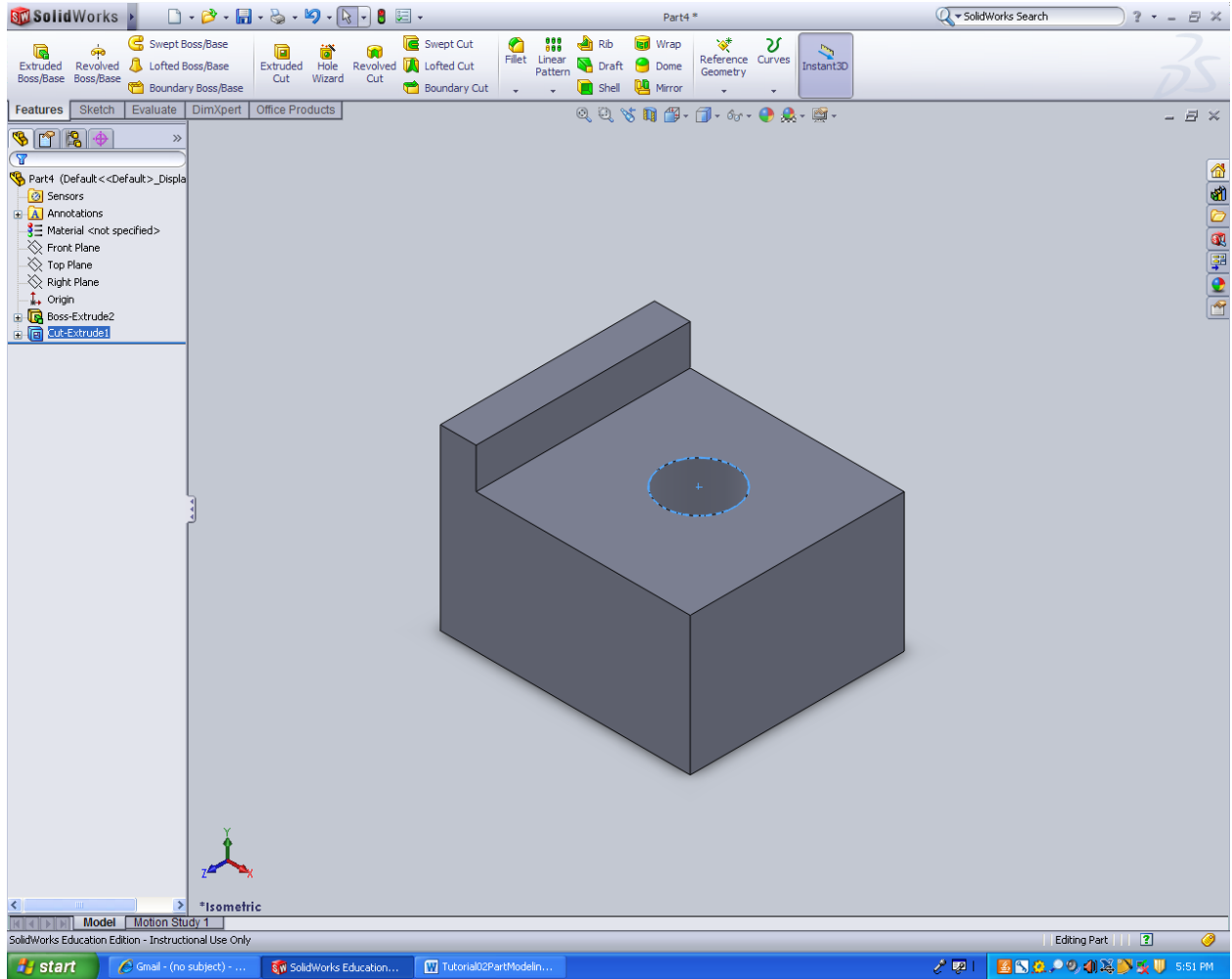


Step 8

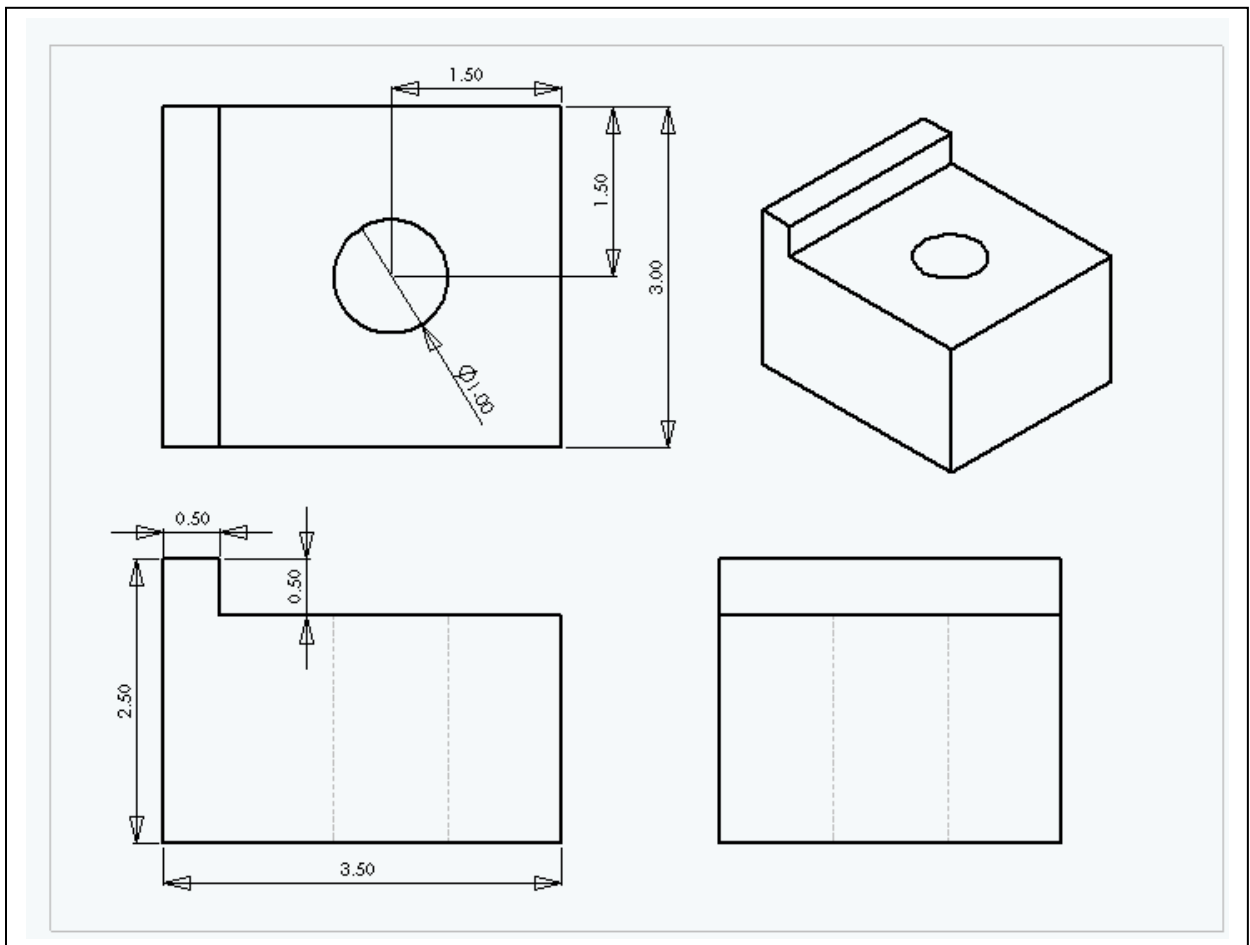
Go to Features **Tab**→ **Extruded Boss/base** and use the option **Through All**. Select OK to complete the extrusion.



Preview of the completed model.



This completes the model. Shown below is the dimensioned drawing view of the model that we just created.



Save the models with the name pin, rectangular plate and base plate.