DEPARTMENT OF MECHANICAL AND INDUSTRIAL ENGINEERING
NORTHEASTERN UNIVERSITY

CAPSULE PROGRAM
Funded by NSF grant #0833636

Tutorial 03
Engineering Drawings

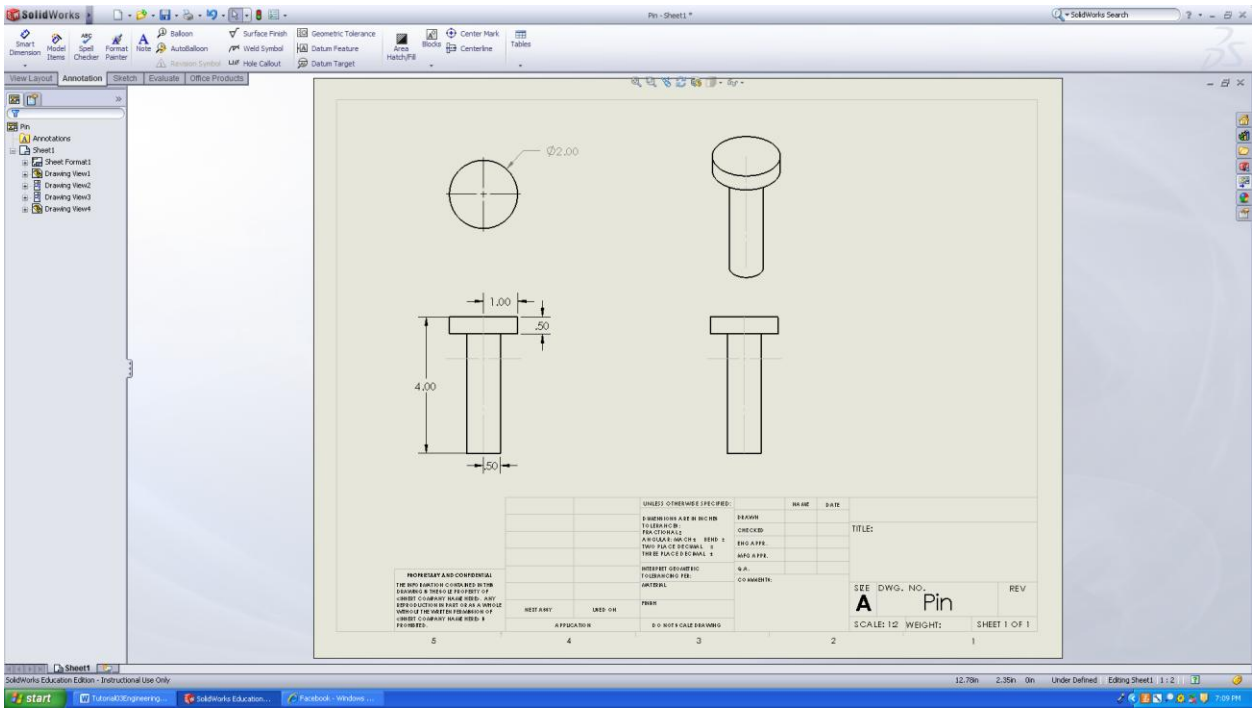
SolidWorks 2010

Drawing and Dimensioning

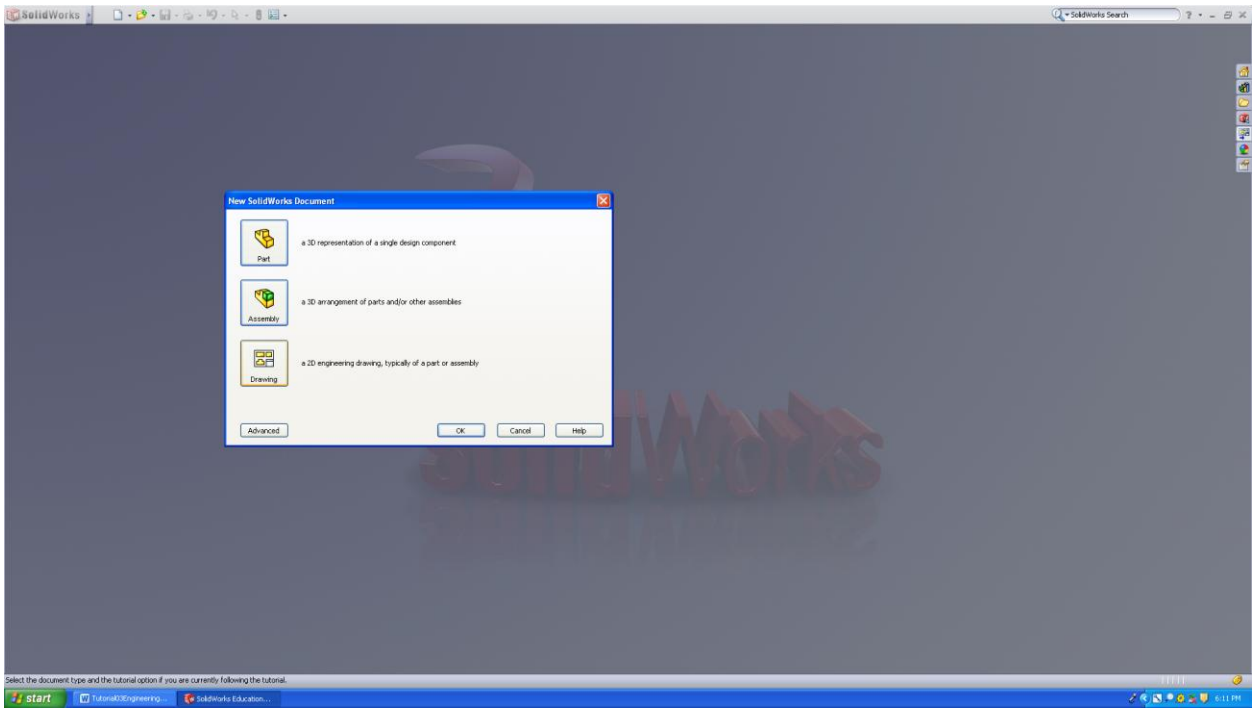
In this tutorial you create an engineering drawing of a part

General planning strategy

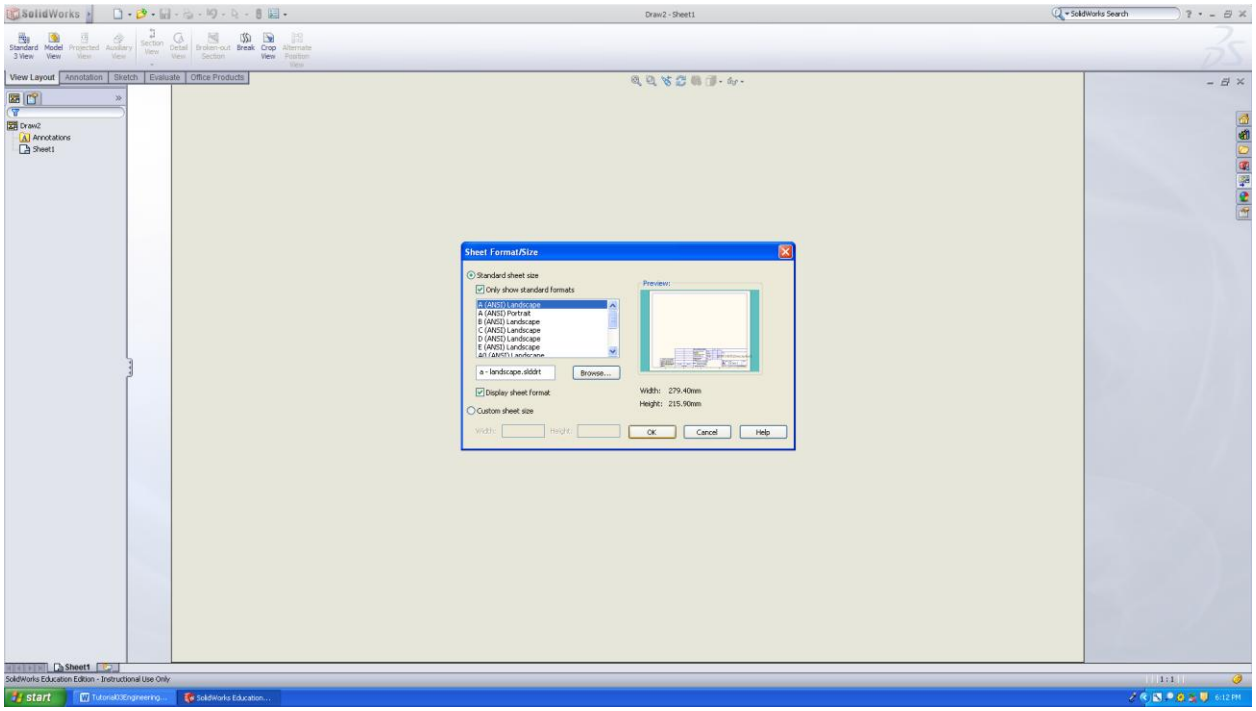
At the end of this tutorial we shall have the drawing/drafting layout shown below:



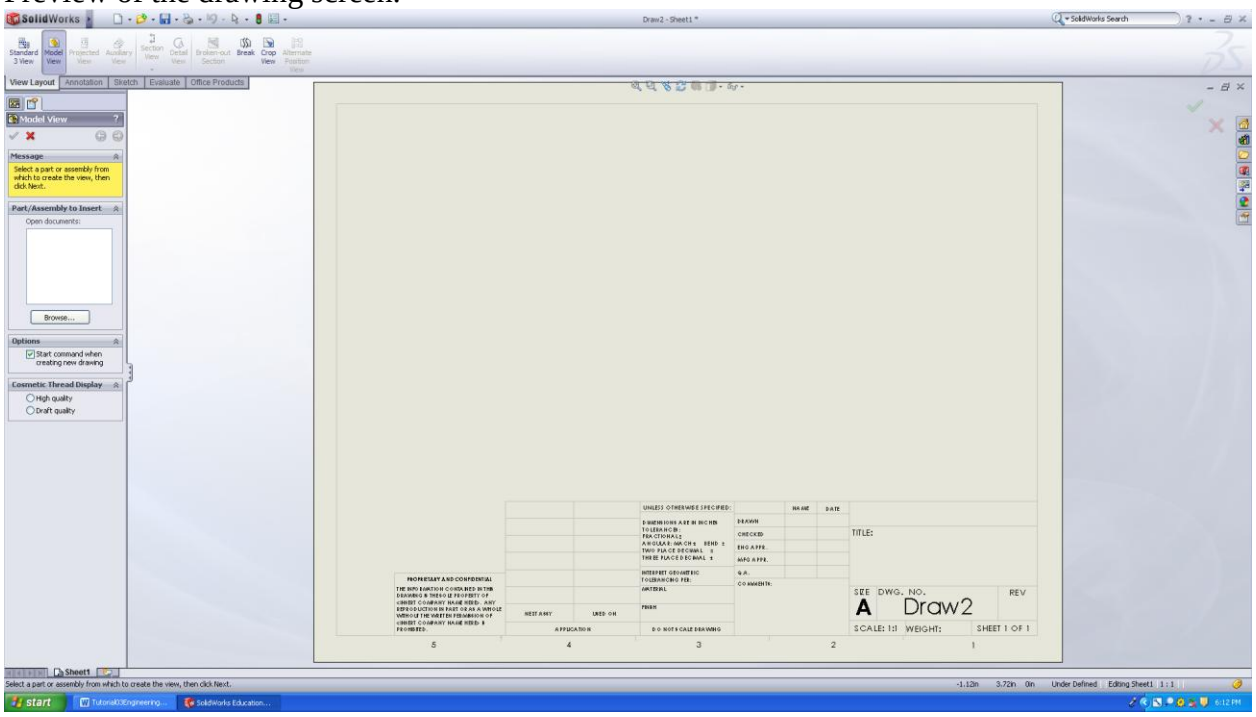
Step1
Go to **New→Drawing**.



Select **A(ANSI)Landscape** and **Select OK**

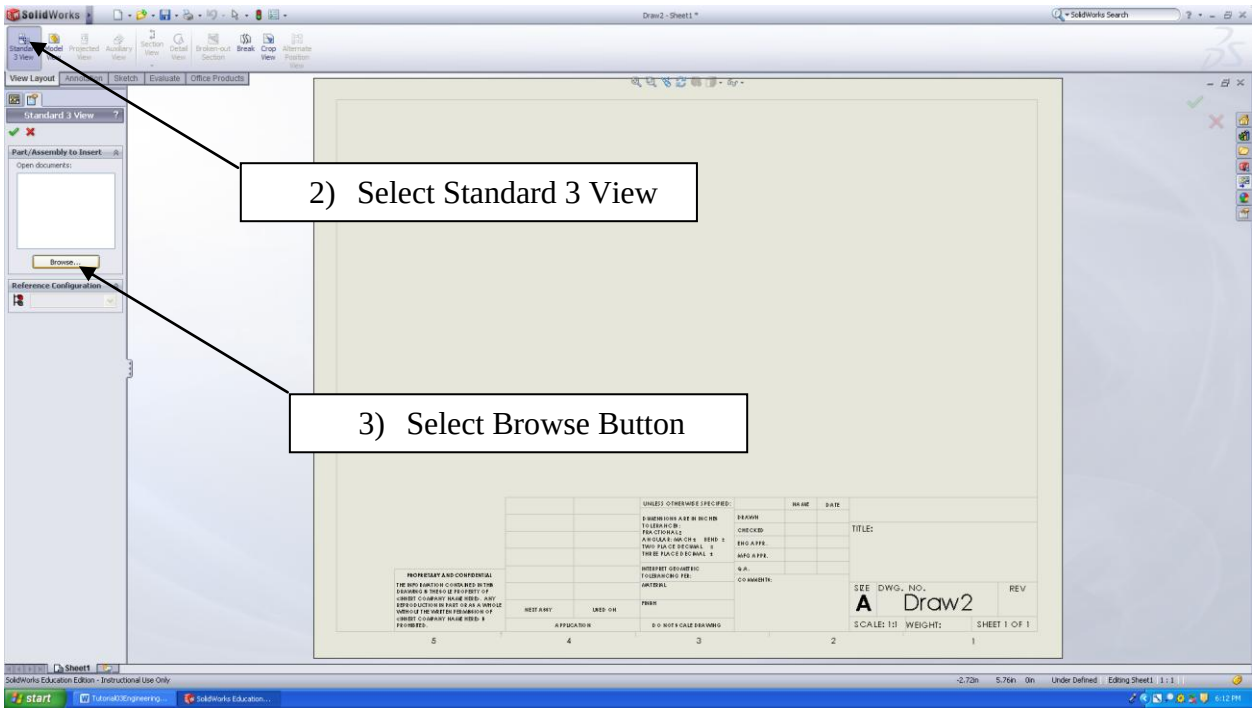


Preview of the drawing screen.

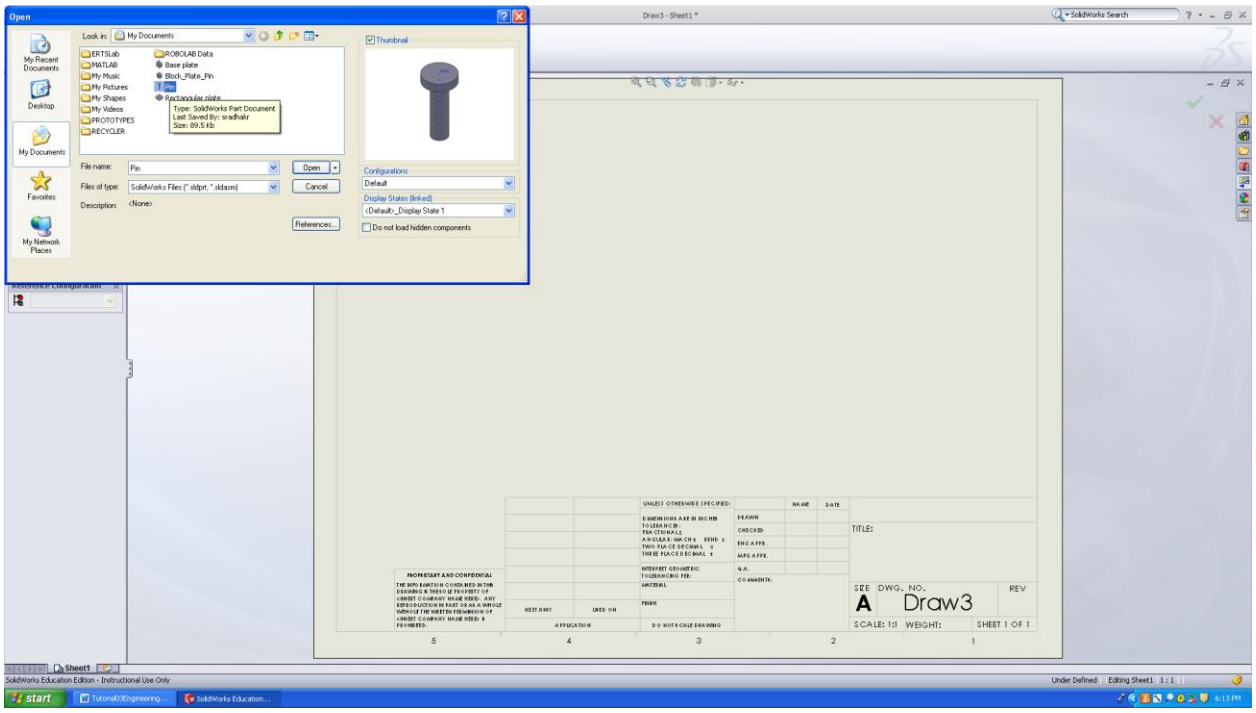


Step 2

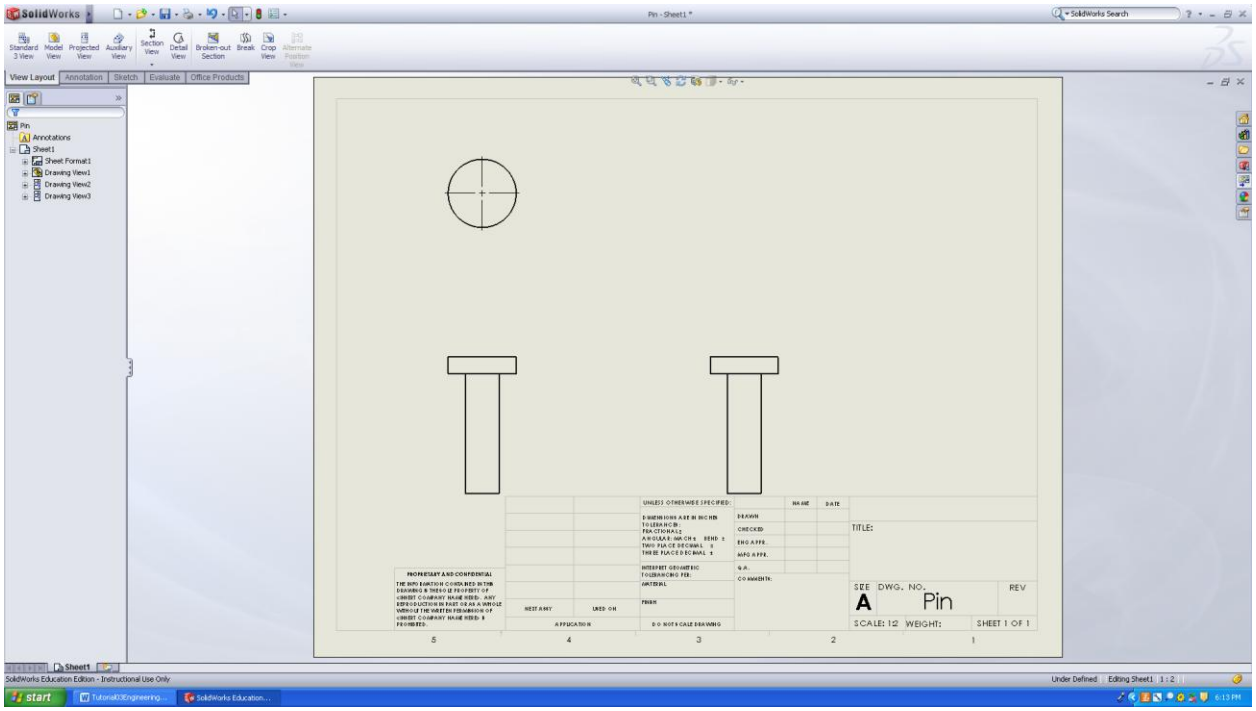
Go to **Standard 3 View** and **Select Browse**.



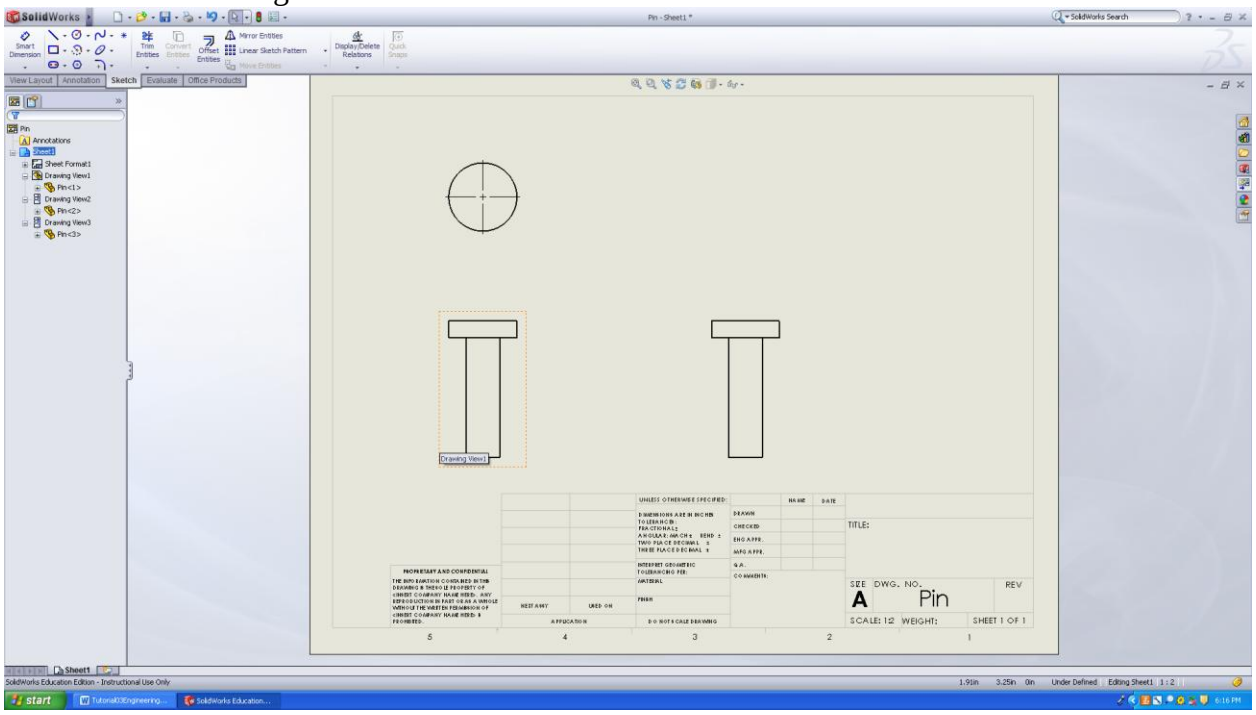
Step3
Select the part file named “pin” and select **open button.**



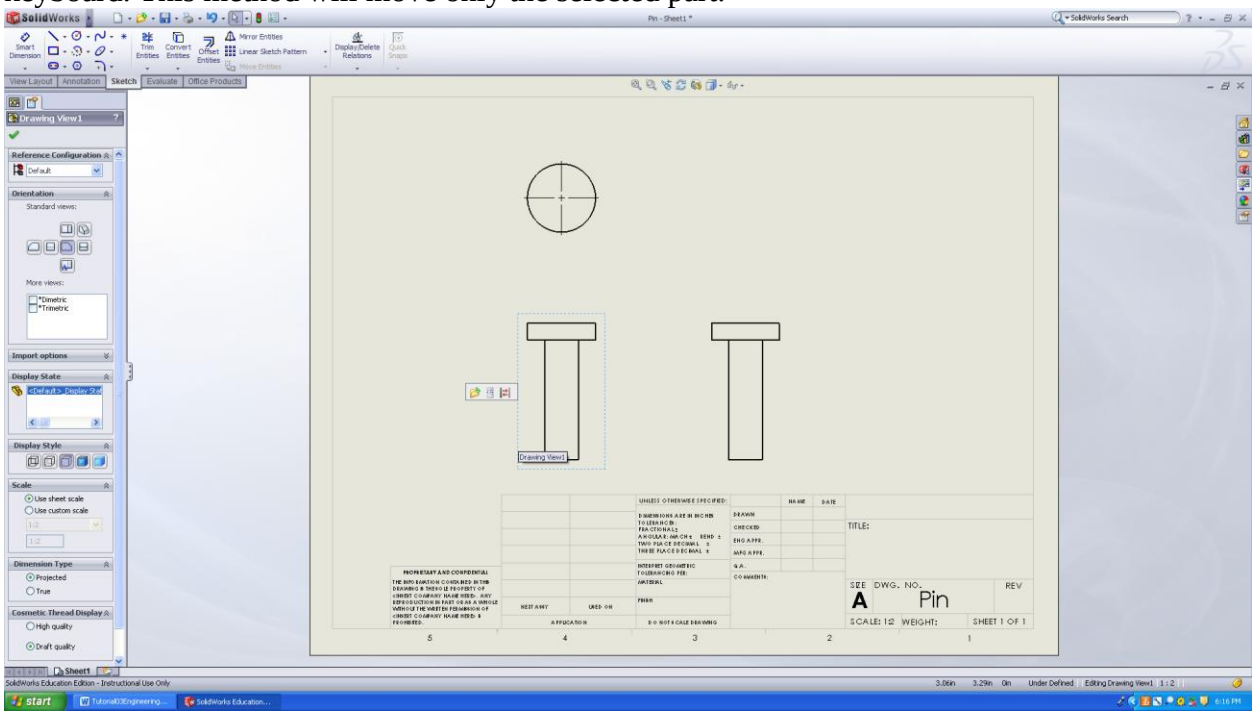
Preview of the pin model drawing.



When you hover the mouse on a drawing, it gets highlighted by an orange box. You can the select the orange box and move using the mouse. Remember when you move one section of the drawing the other sections also moves.

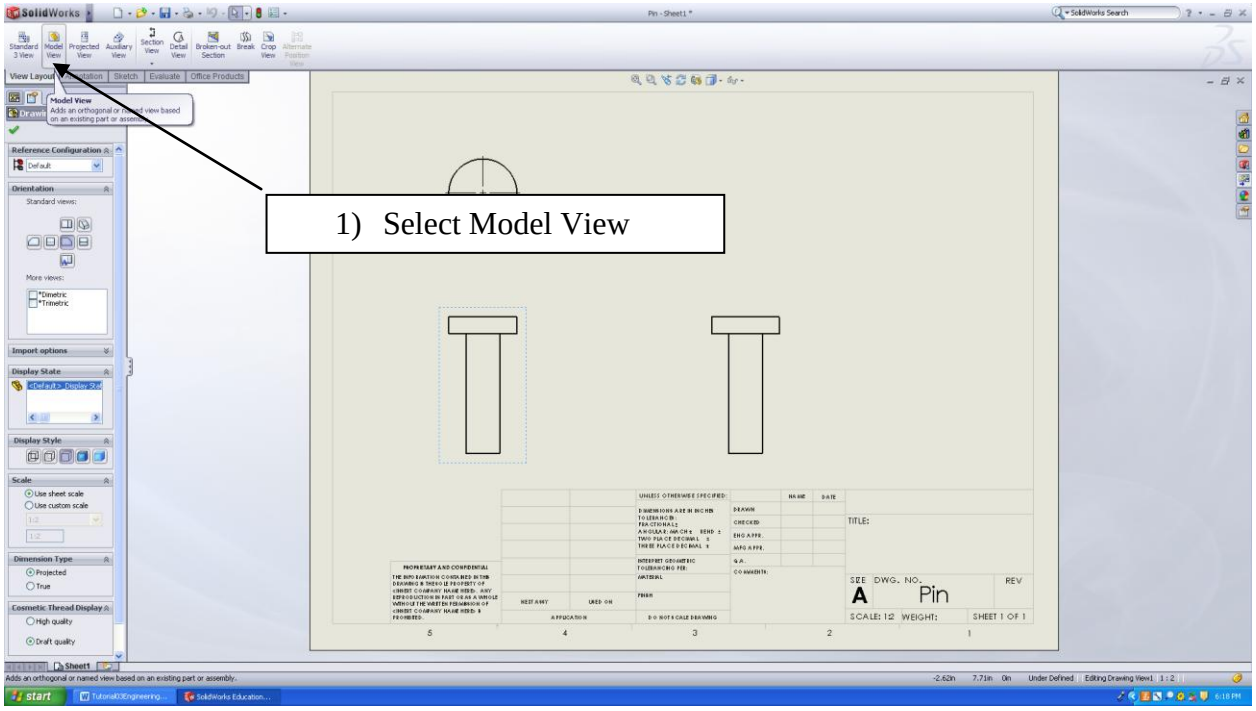


You can select individual sections of the drawing and move using arrow keys on the keyboard. This method will move only the selected part.

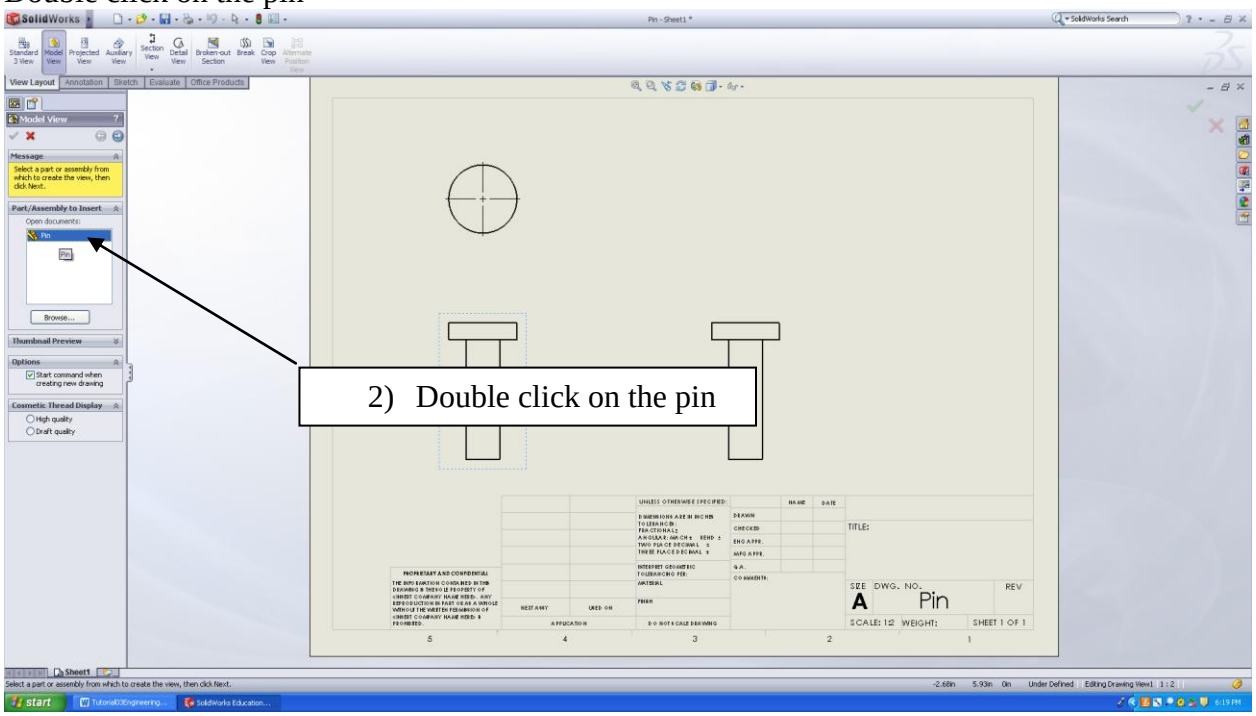


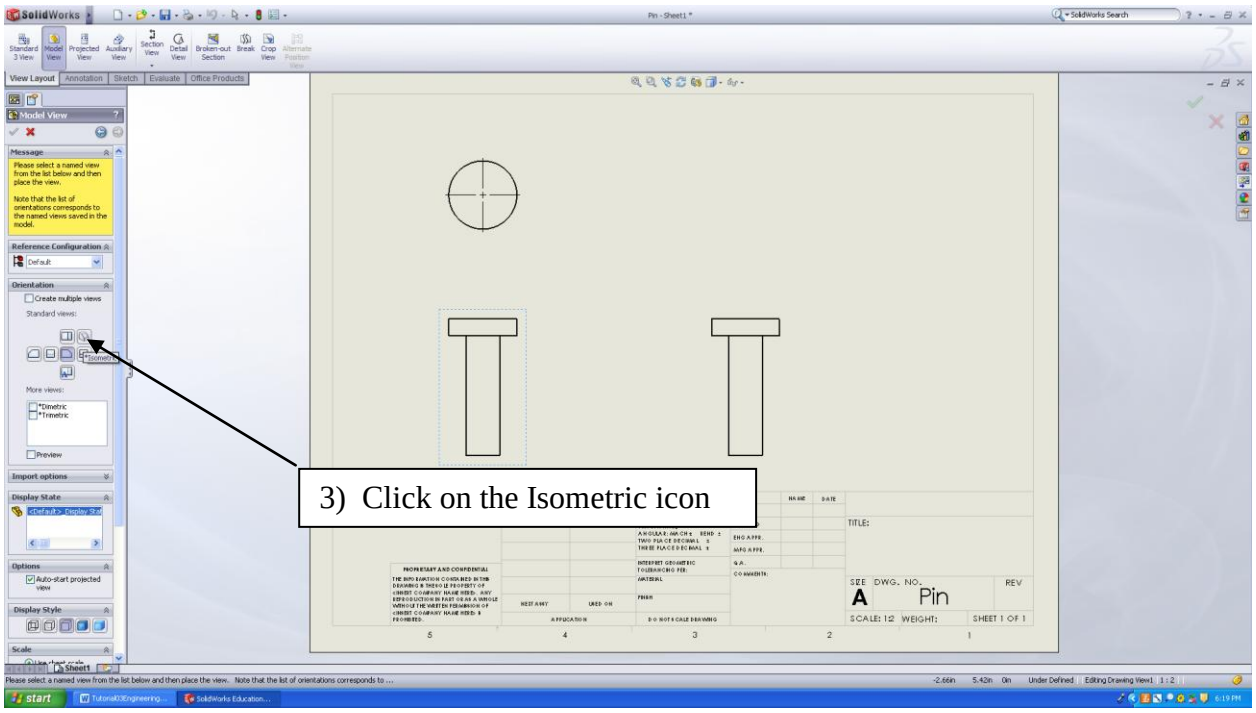
Step 4

Select **Model View**

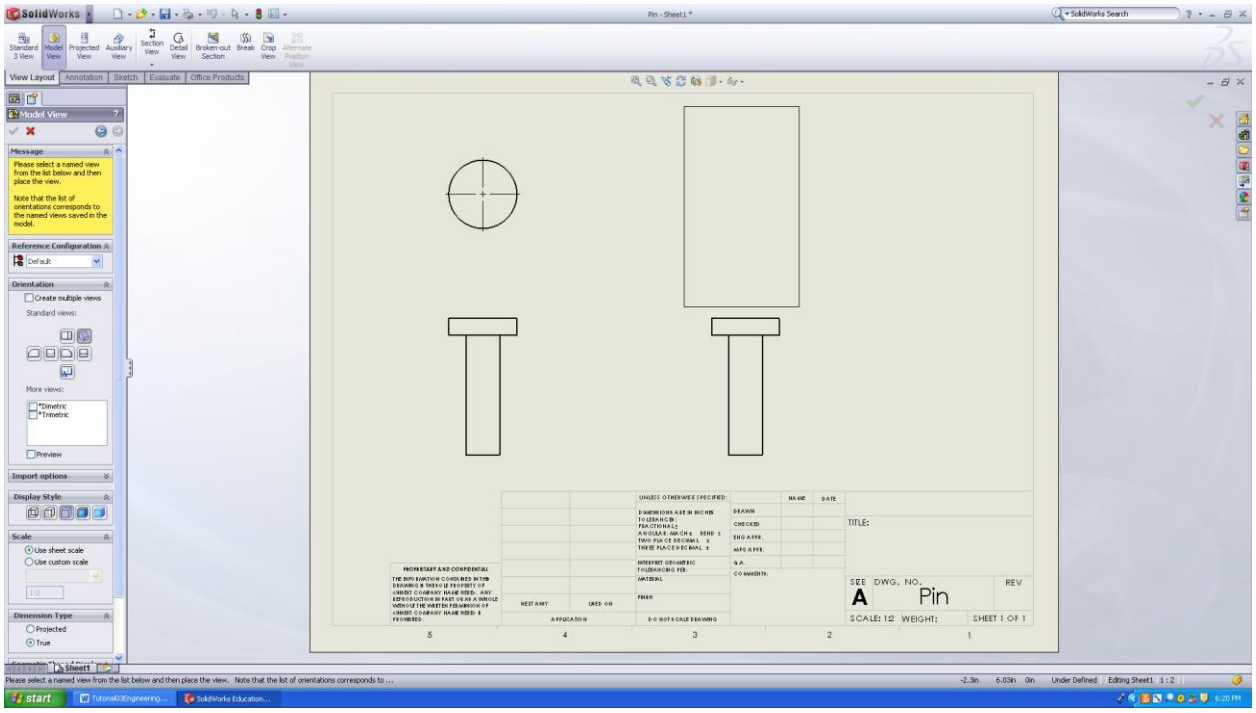


Double click on the pin

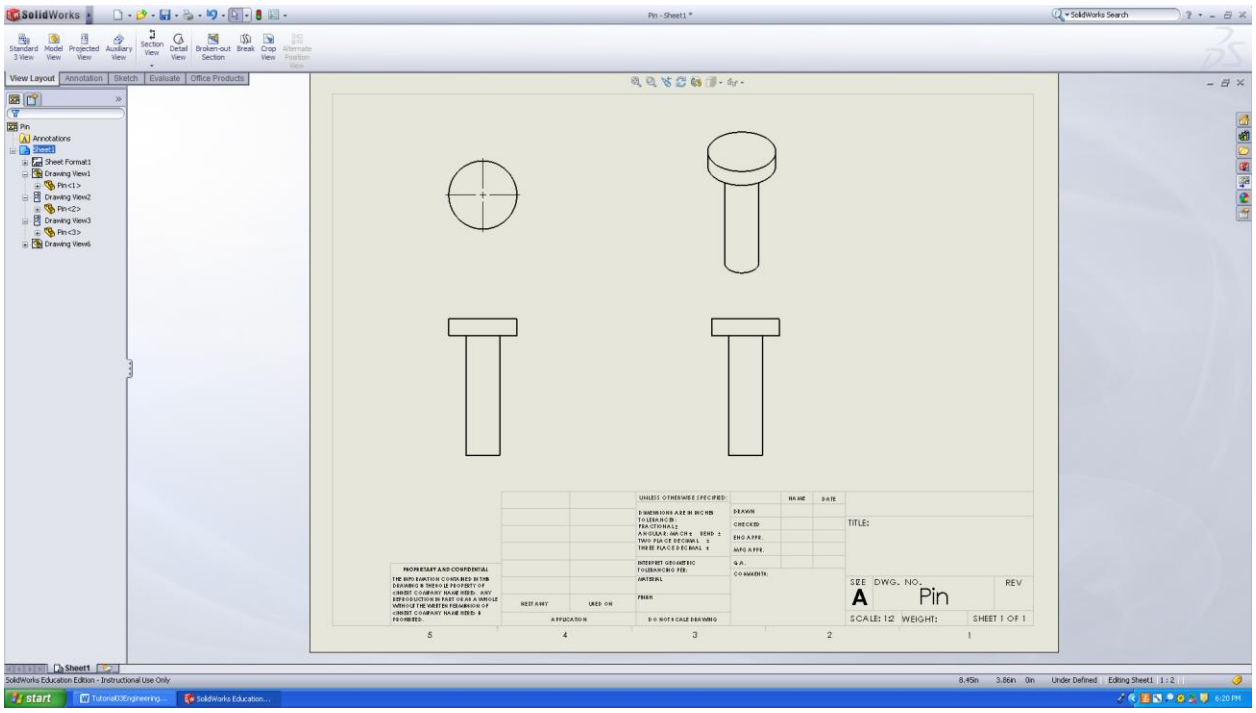




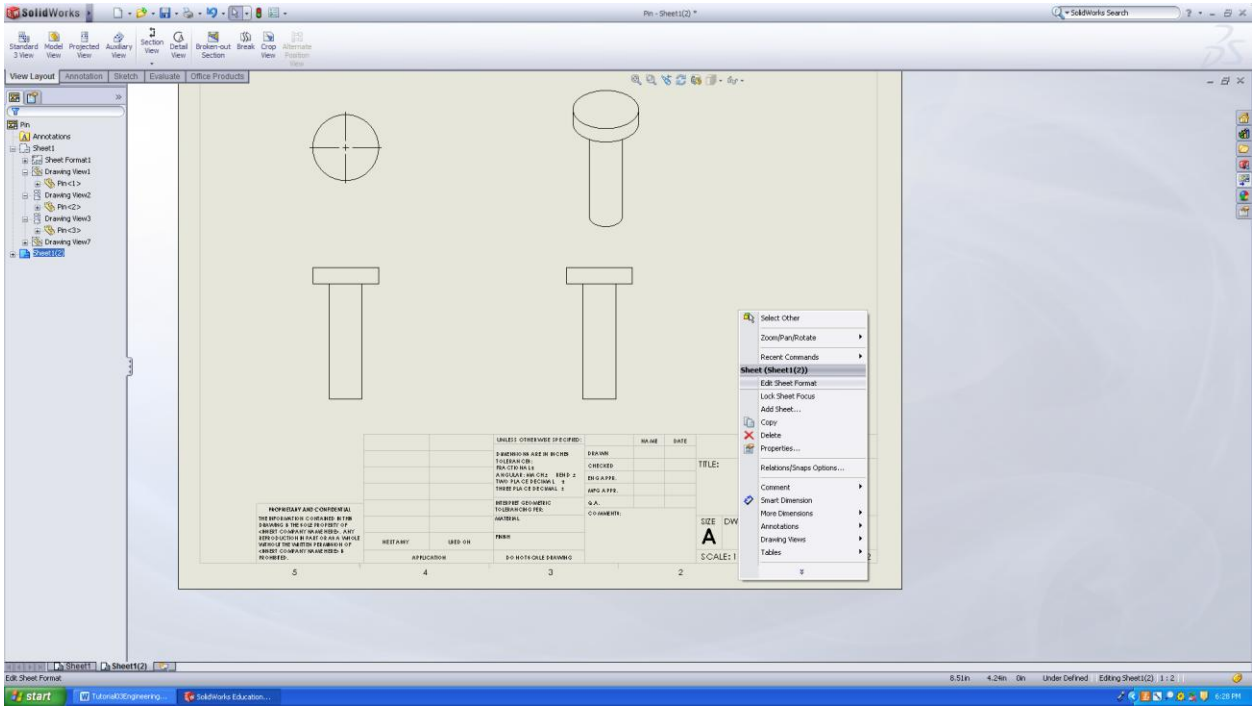
Once you selected the Isometric icon, a box will follow your mouse pointer. Click on the drawing area to fix the box.



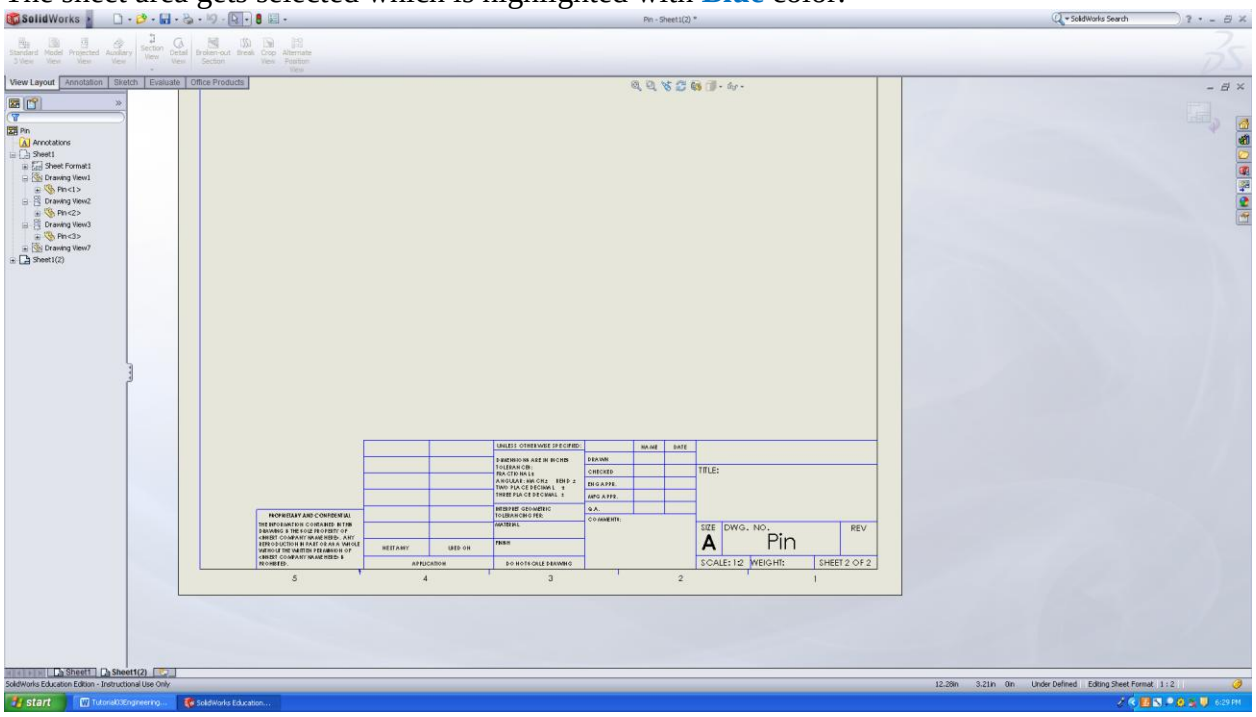
Preview of the Isometric Model.



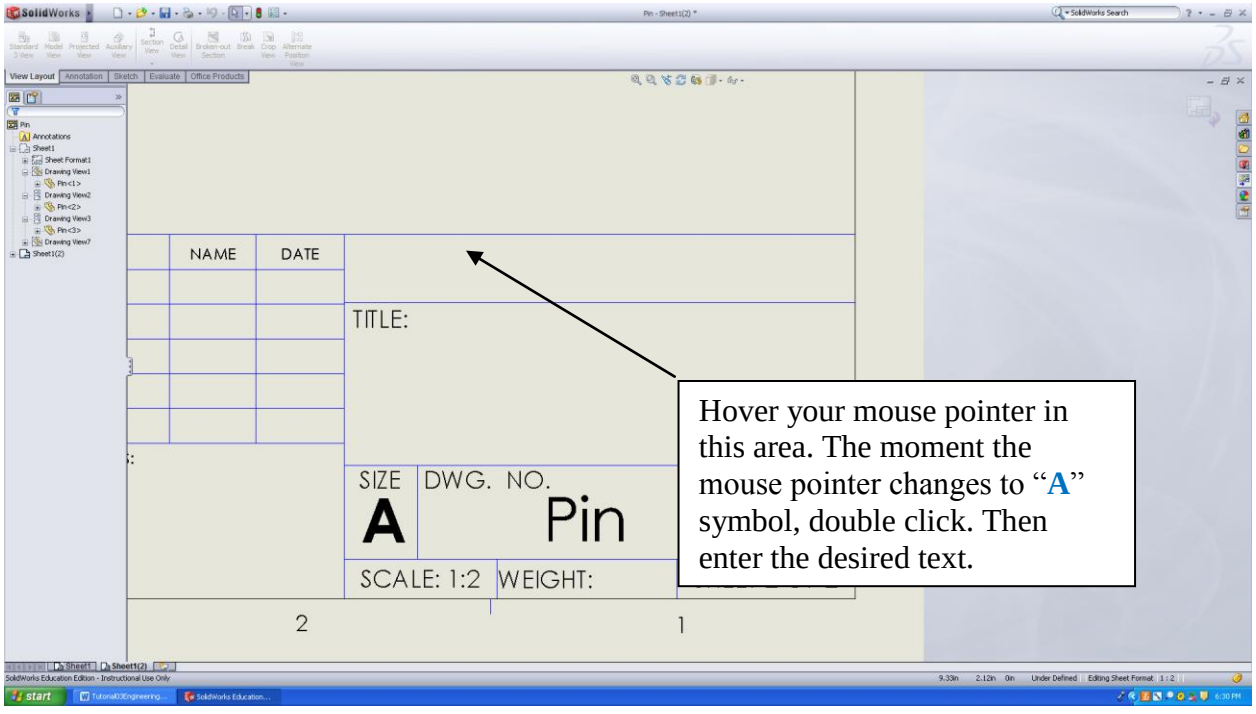
Step 5
Right click on the drawing area and select **Edit Sheet Format**.



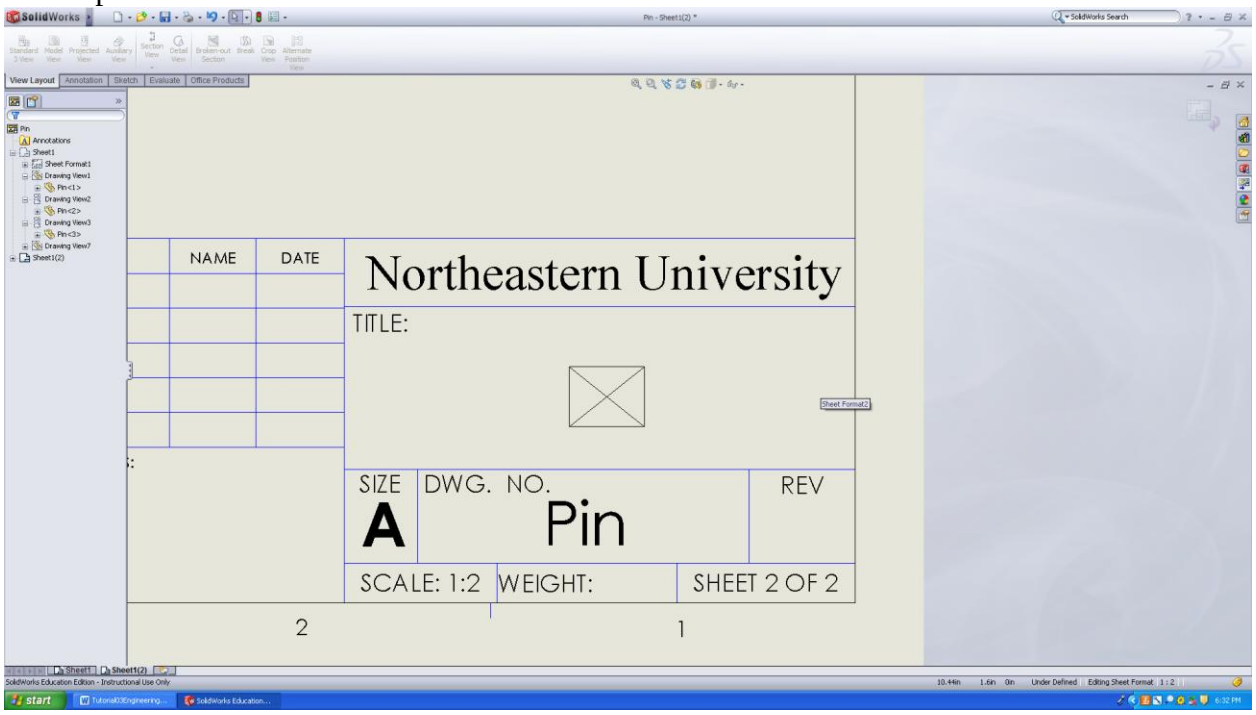
The sheet area gets selected which is highlighted with **Blue** color.



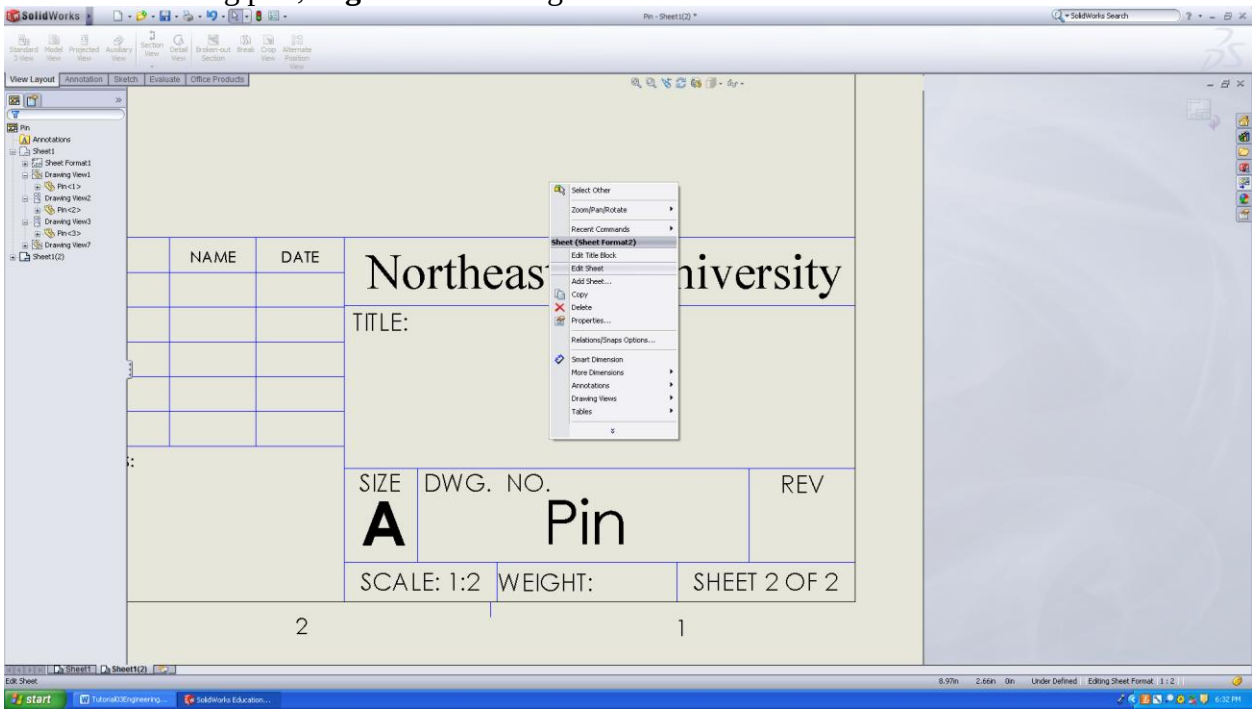
Zoom to the Title Block area.



Follow the same procedure to enter text value in different text boxes. Sometimes the text gets outside the boxes. They can be made to fit the boxes by clicking and dragging the mouse pointers on small blue boxes that surround the text box.

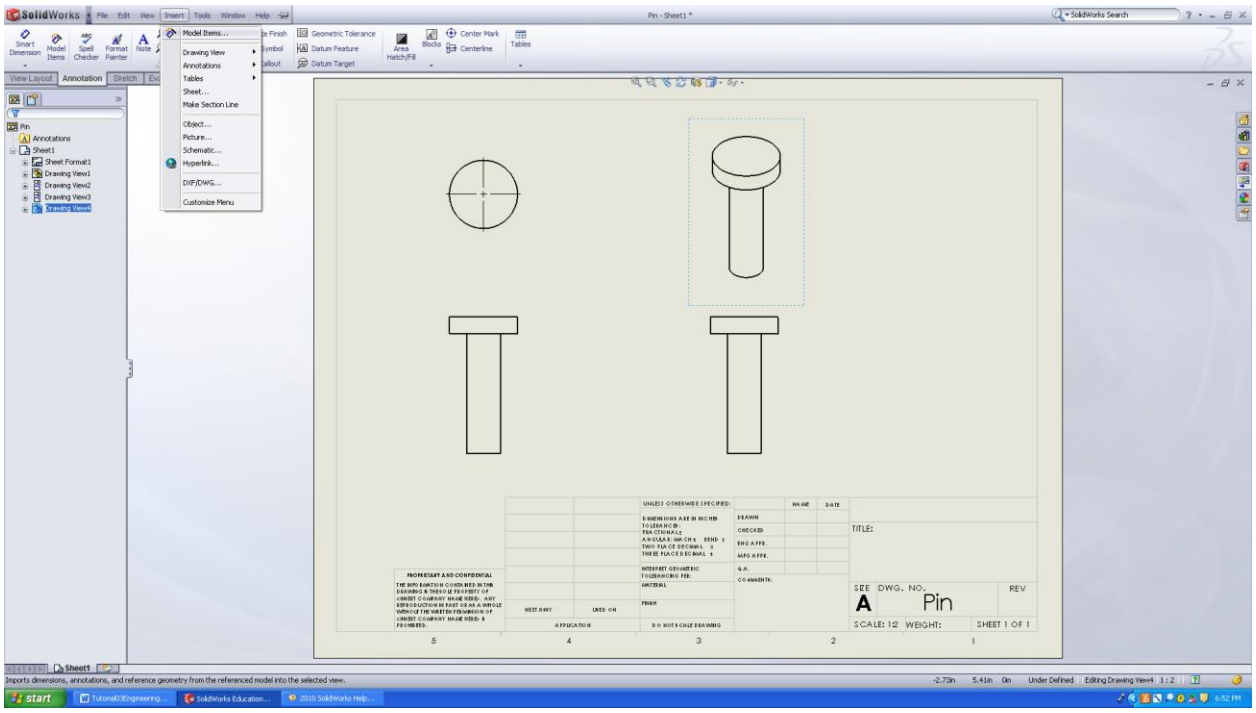


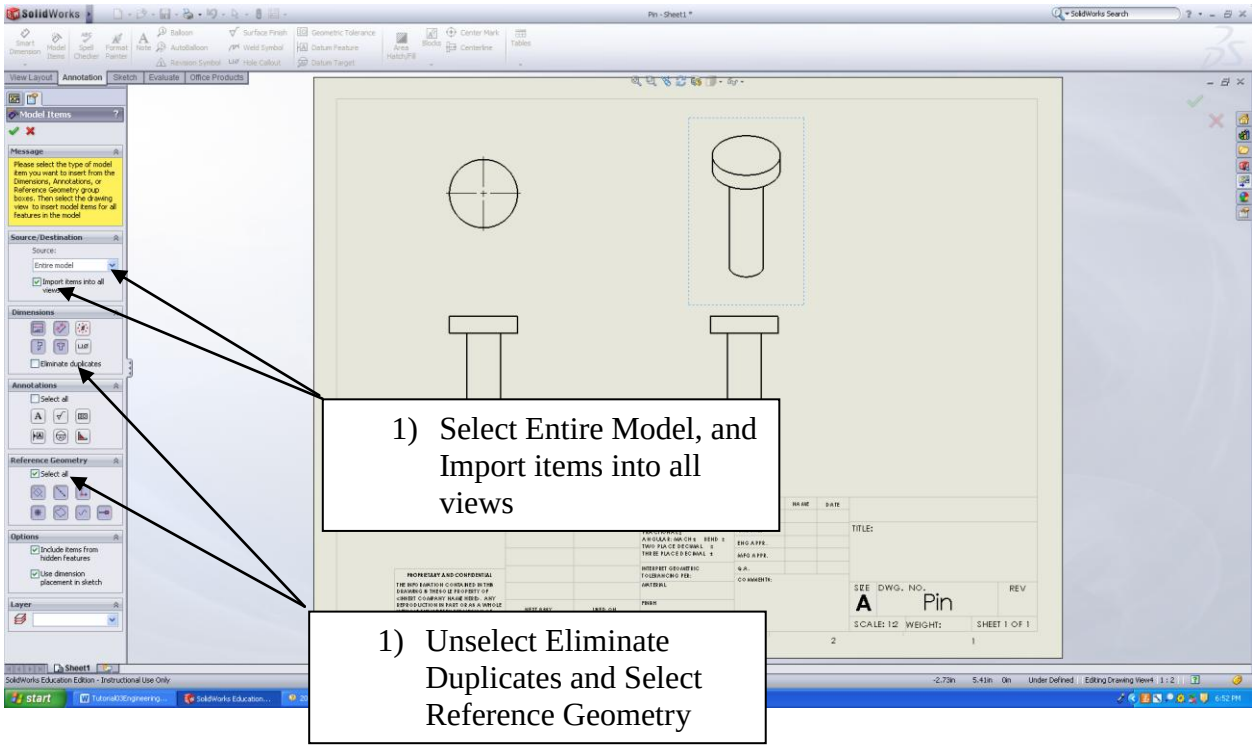
To exit the editing part, **Right Click** and again select **Edit Sheet.**



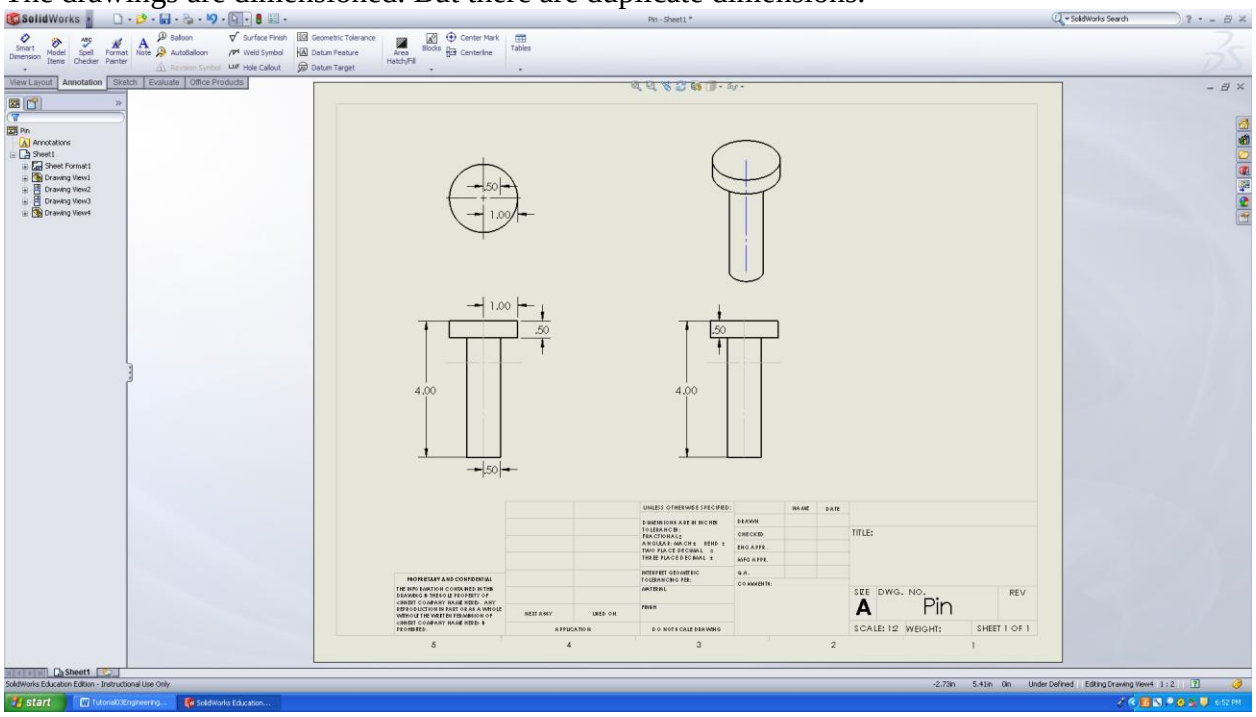
Step 6

In order to dimension go to **Insert→Model Items**.

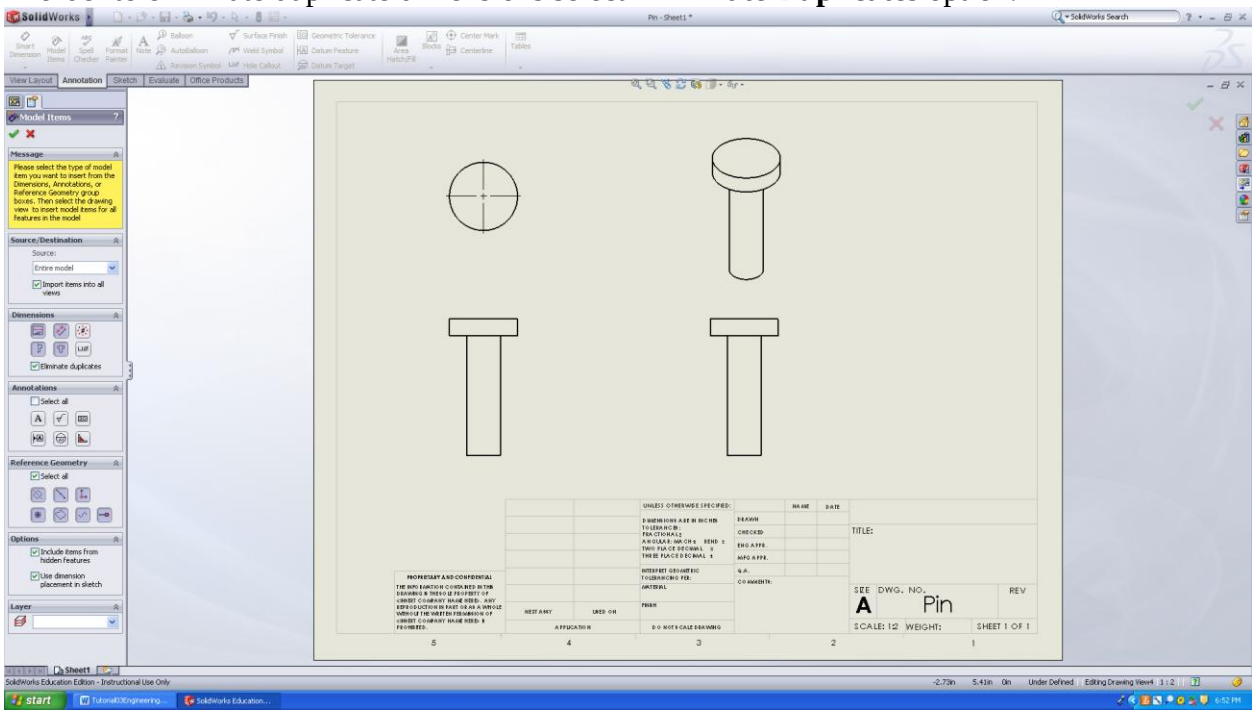




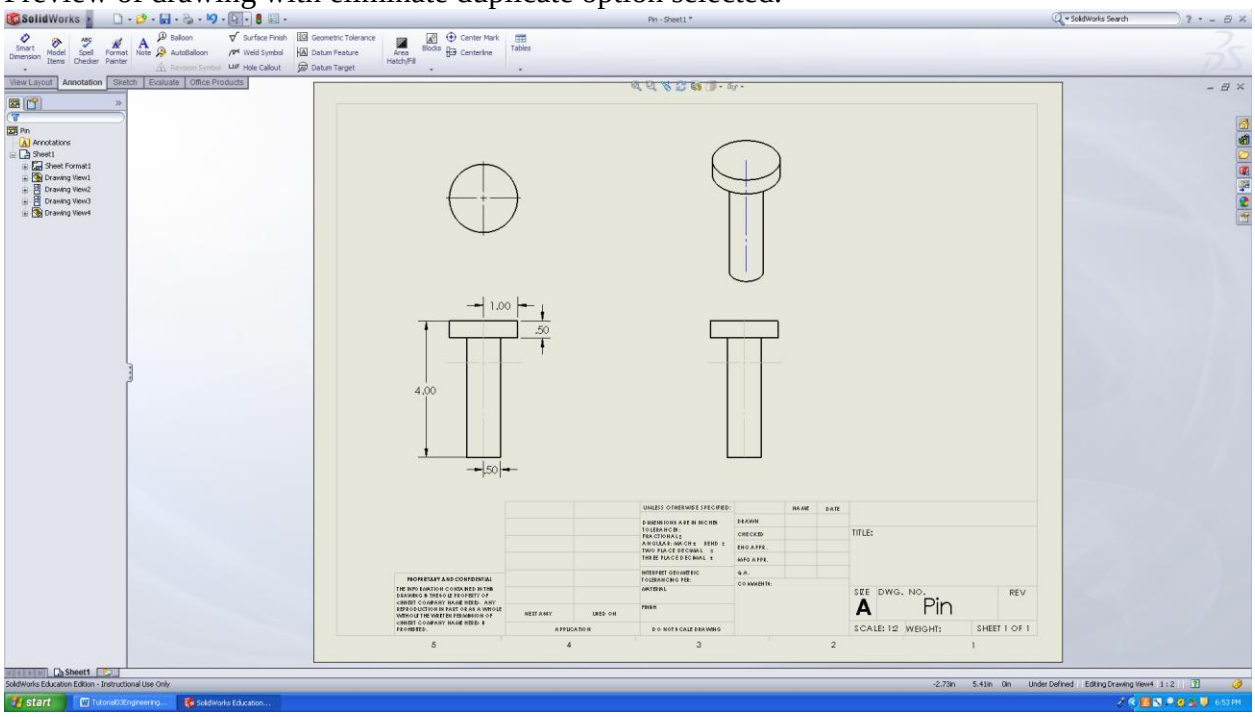
The drawings are dimensioned. But there are duplicate dimensions.



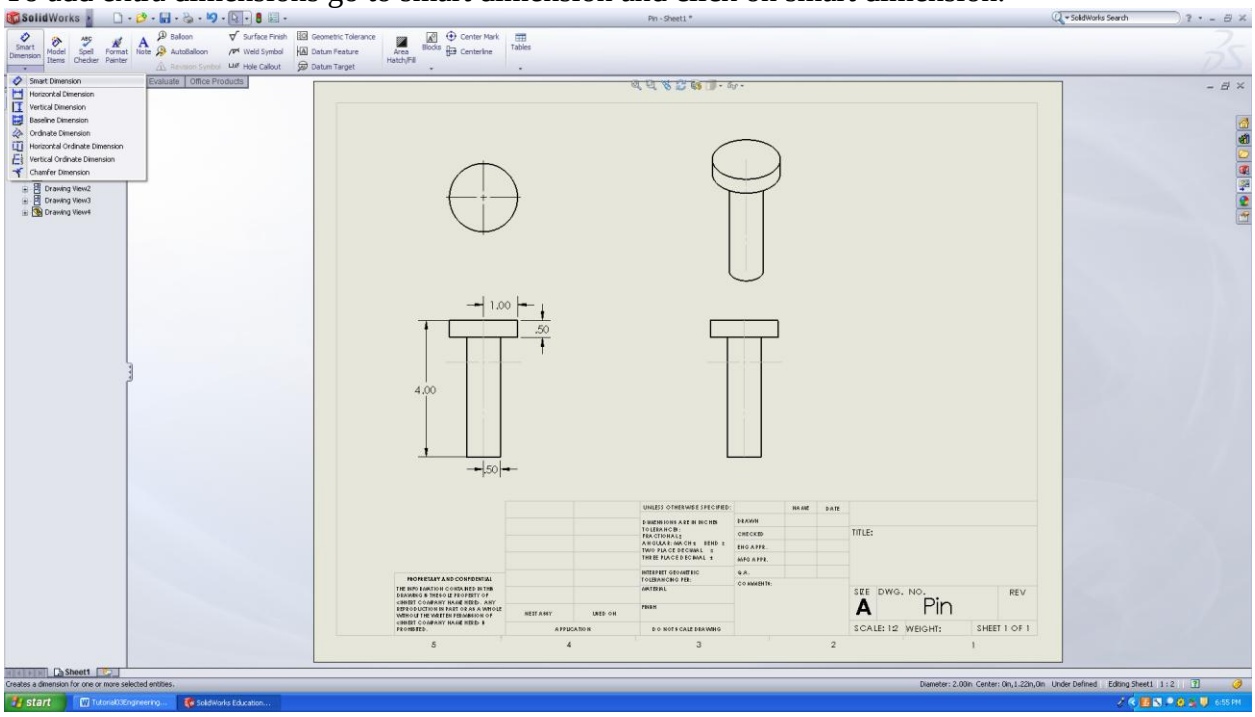
In order to eliminate duplicate dimensions select **Eliminate Duplicates** option.



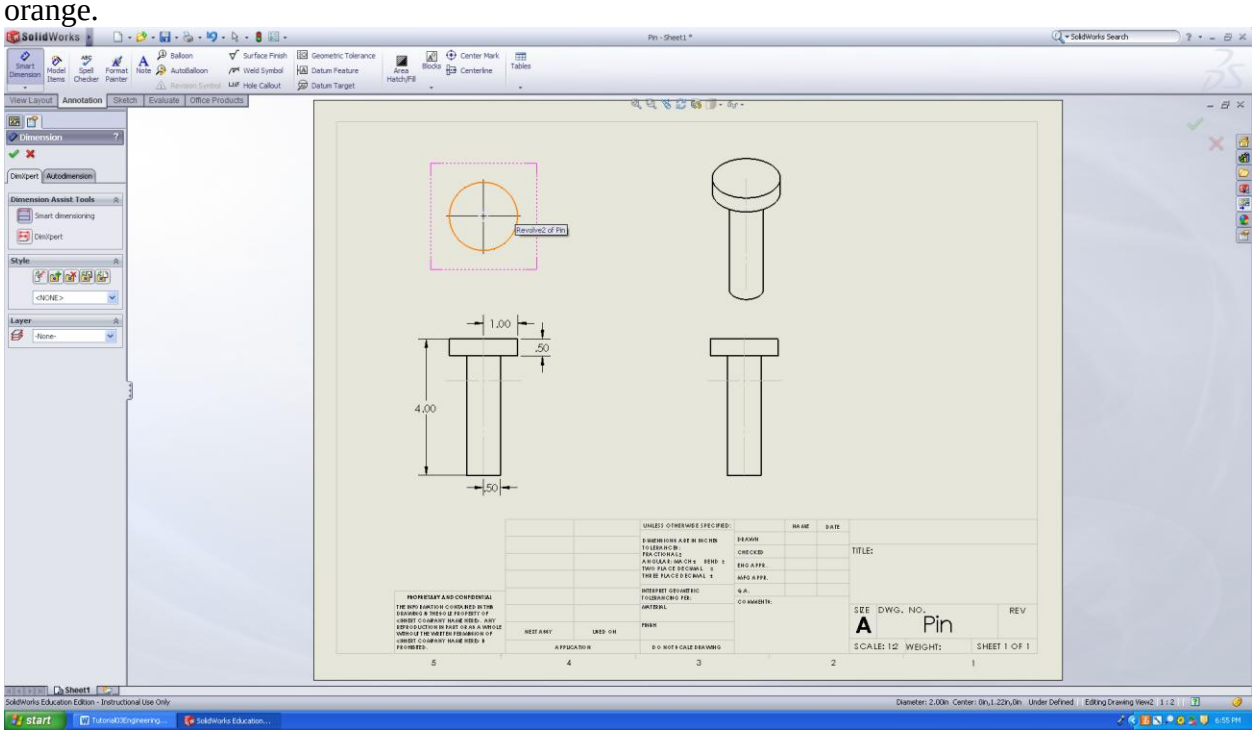
Preview of drawing with eliminate duplicate option selected.



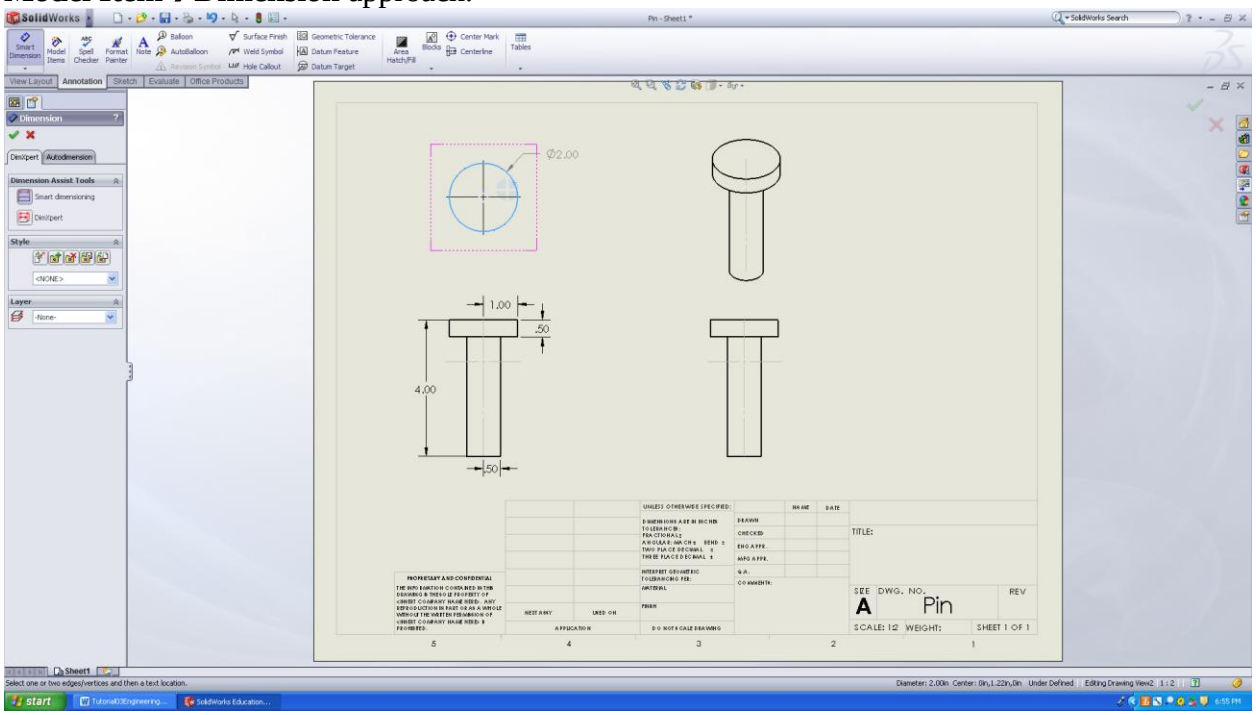
To add extra dimensions go to smart dimension and click on smart dimension.



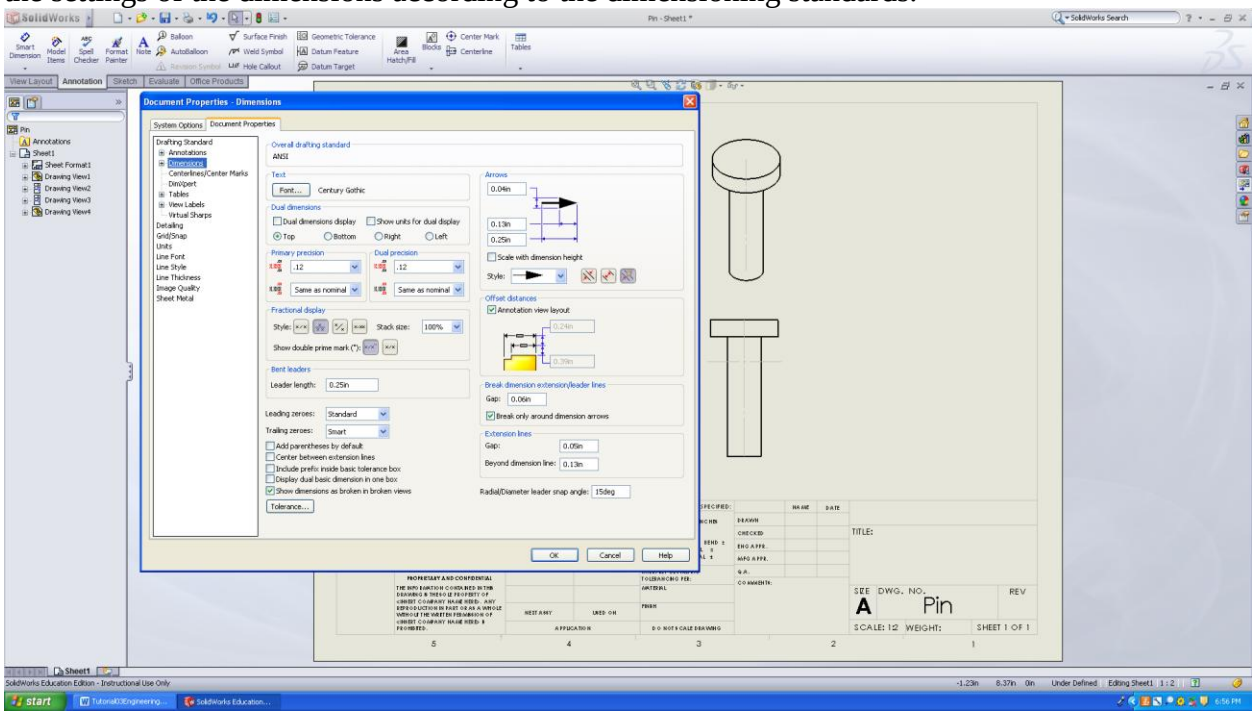
Hover mouse pointer over the desired geometry. The color of that geometry will turn orange.



Click on the geometry and drag outside the figure to get the desired dimension. You may dimension entire model using smart dimension. But it is more time consuming that the **Model Item→Dimension** approach.



You can go to **Tools→options→Document Properties Tab→Dimensions** and change the settings of the dimensions according to the dimensioning standards.



Step 7
It is important that you save the file as you shall be using it in other tutorials. In the **Main** window go to **File** and click on **Save As,** select an appropriate file name and click **OK**.

Class Hands-on
Create the 3 standard views and an isometric view of the Support Bracket as shown in the figure below.