

Lab 1: Pro-ENGINEER Part Modeling

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Part 1: Setup

In the HAC lab, Pro-E is located under the start menu.

Start Menu – Programs – Engineering Applications – Pro Engineer

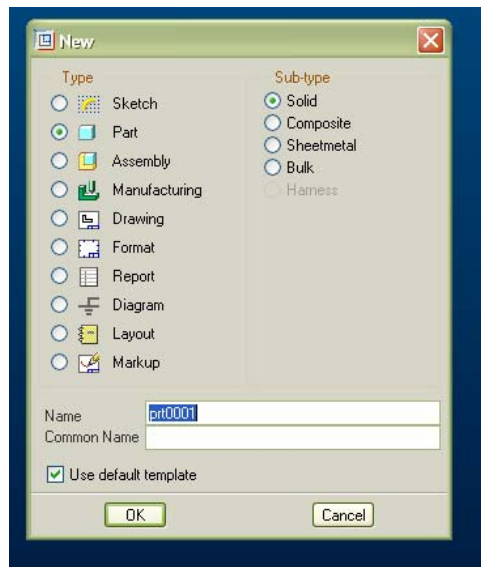
The first thing to do after the program finishes loading is to assign a working directory. It's recommended that you bring a flash drive, and create a folder on it for your part and other files. In the top left corner of Pro-E, look for:

File – Set Working Directory

Navigate to find the folder you created, double-click on it, and then press **OK**.

Now, it's time to create a part. In the top left of the main Pro/E screen, press: **File – New**

The following menu should appear. Keep everything as is, and enter a name of your choosing for the part. Then press **OK**

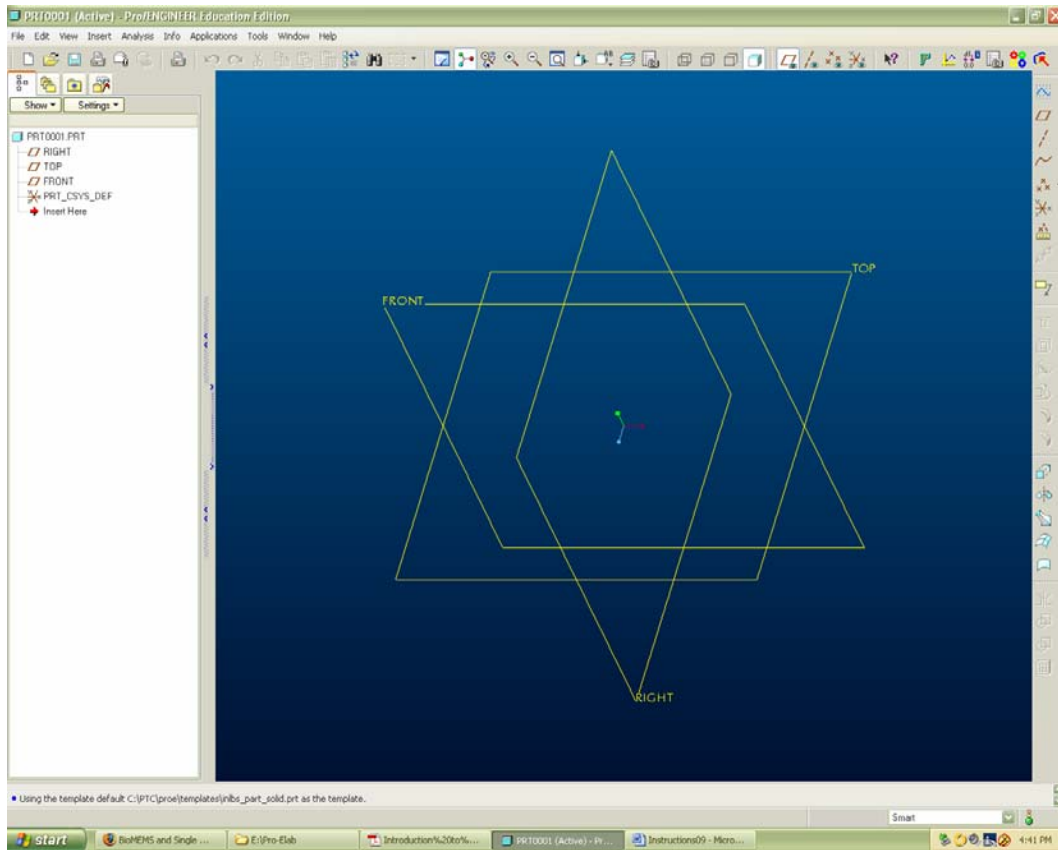


(*Note – the template controls the units. Default uses inches for length, and pounds for force. Un-clicking the template box will open up a menu to change units. For this lab, stick with default)

A blue screen should appear. If it's not already selected, click on the '**Datum planes on/off**' button to toggle on a 3D set of planes (the button should be on the top of your screen):



You should now see this:

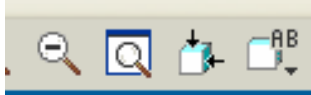


You can rotate the three dimensional planes with the following controls:

- Translate: Hold down *Shift* and the *Middle Mouse Button*, and then move the mouse
- Rotate Planes: Hold down the *Middle Mouse Button*, and move the mouse
- Zoom: Scroll with the *Middle Mouse Button*

To return things to their original view:

- Click on the '**Saved view list button**' along the top of the screen:



(it's the one with the little 'AB' on it)

And select '**Default Orientation**' from the menu that appears.

We're going to create the part by choosing one of the three planes, sketching a 2D picture on it, and then pulling it out into a third dimension.

- First, click the '**Extrude Tool**', located along the right side of the screen:



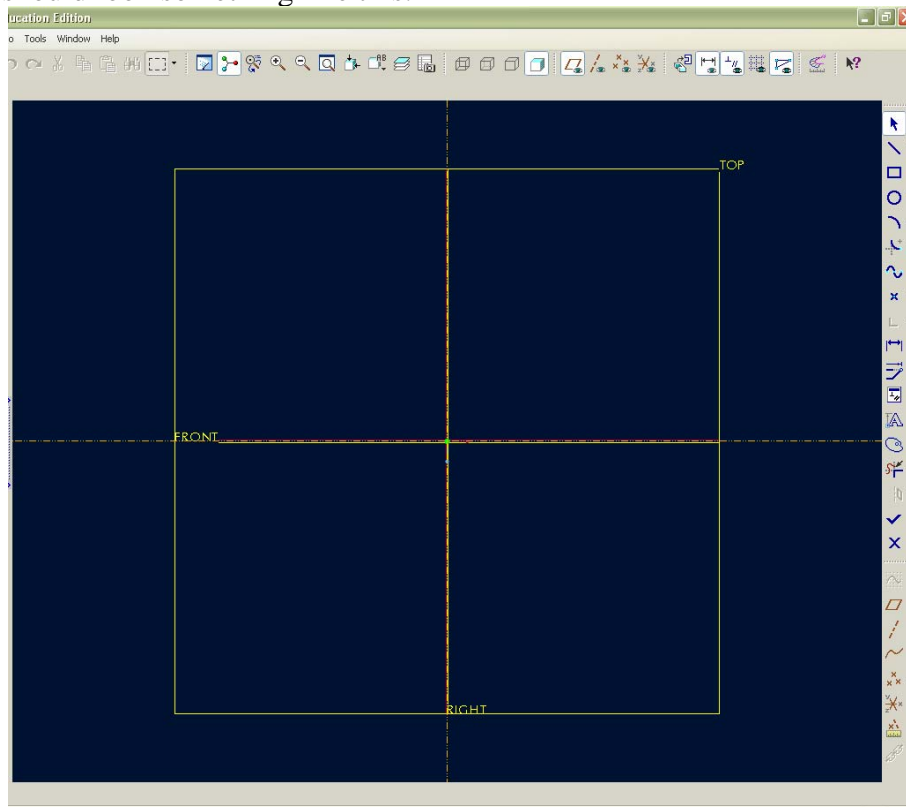
A new menu will now appear along the bottom of the screen. The word 'Placement' will appear in red, probably on the left side of the new menu.

- Click on '**Placement**'
- Click on the '**Define**' button that appears.

Another menu pops up on the screen entitled 'Sketch' at this point. Now, we're going to select one of the three datum planes to draw the sketch on.

- Click on one the three planes. The choice is arbitrary.
- Click on the '**Sketch**' button in the 'Sketch' menu.

Your screen should look something like this:



Part 2: Sketching

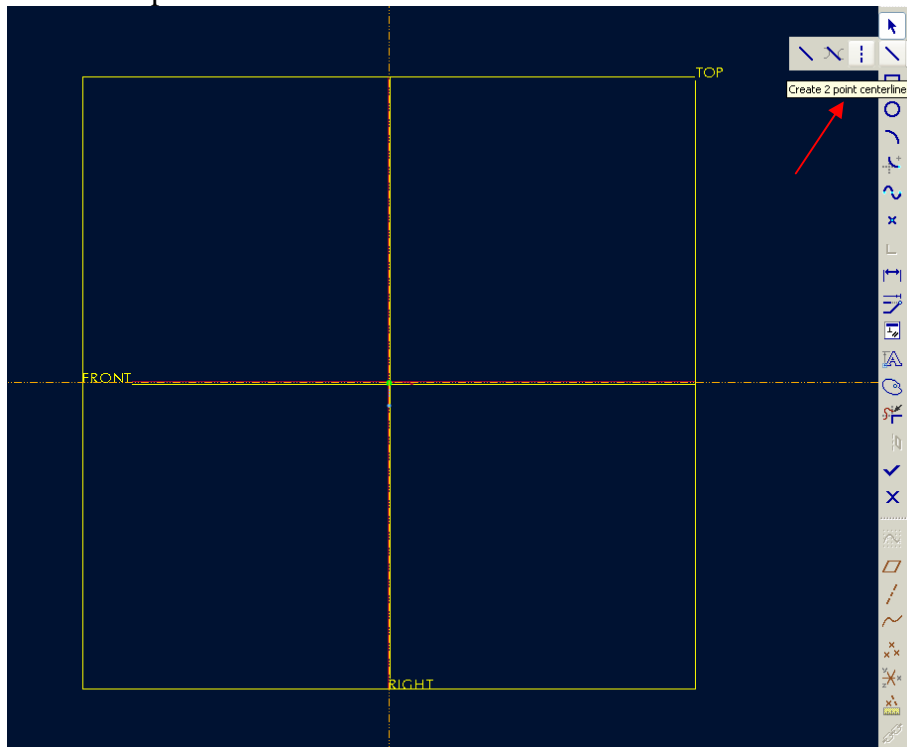
From the lab assignment hand-out, pick one of the three shapes to sketch. You're now going to make a 2D sketch of one of the shapes. It is important that you make the dimensions of the shape fit the stress-concentration curves! For example, if you pick shape two, you'll want the H/h ratio to be between 1.5 and 1.01, and the r/h ratio between 0.05 and 0.30. Otherwise, you won't be able to compare your results from lab 2 with the stress concentration plots.

Below are instructions on sketching in Pro/E, specifically for shape one. There are many, many correct ways to sketch in Pro/E, but these instructions should provide a basic and straightforward tutorial. Feel free to use as much or as little of the next section as you'd like.

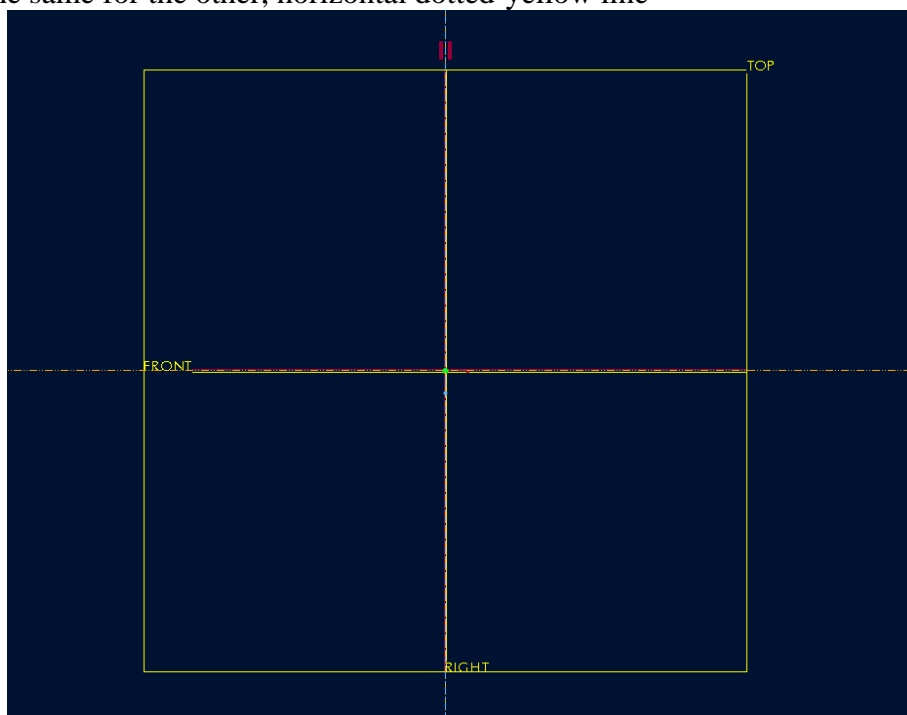
Creating the Centerlines:

Centerlines will be used as a symmetry line during the process of sketching.

- Click on the drop down menu next to the blue line
- Select “Create 2 point centerlines.”

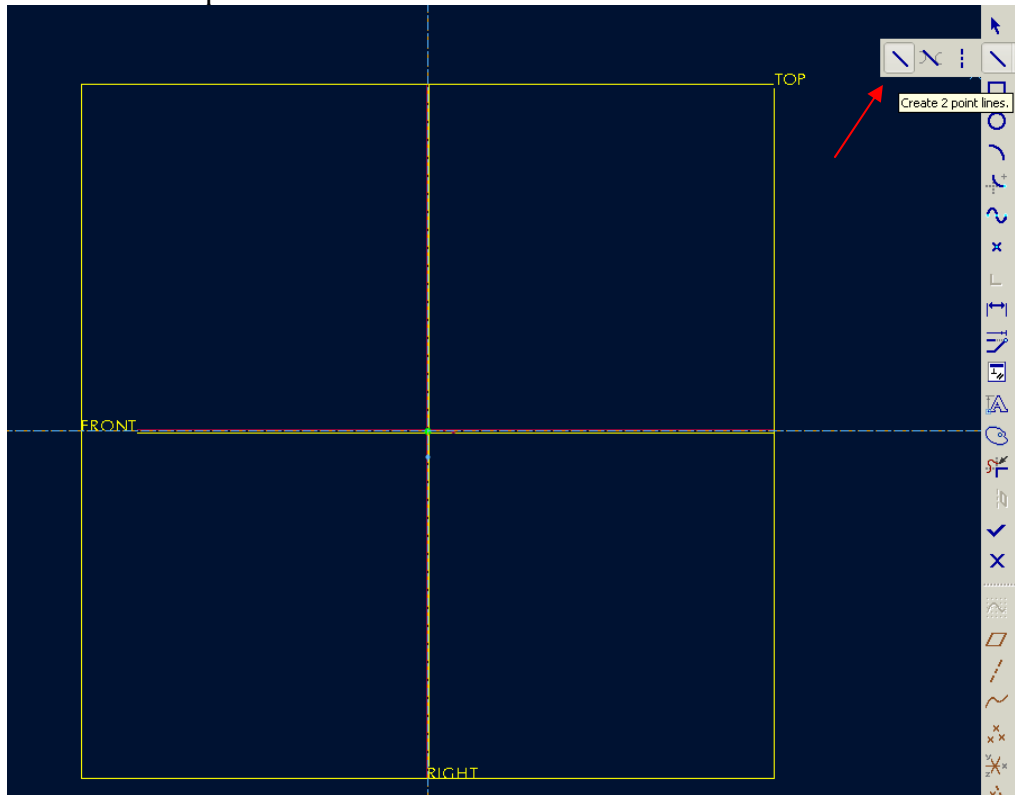


- Click on the vertical dotted-yellow line
- Align the blue line with the dotted-yellow line and click again to complete the line
- Do the same for the other, horizontal dotted-yellow line

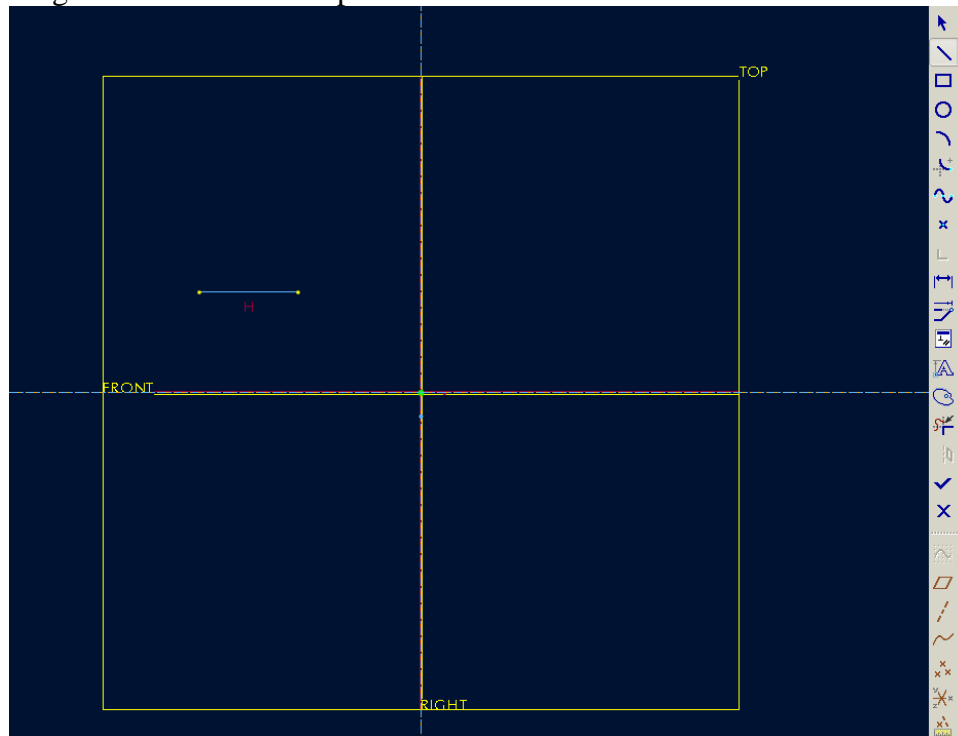


Sketching Straight Lines:

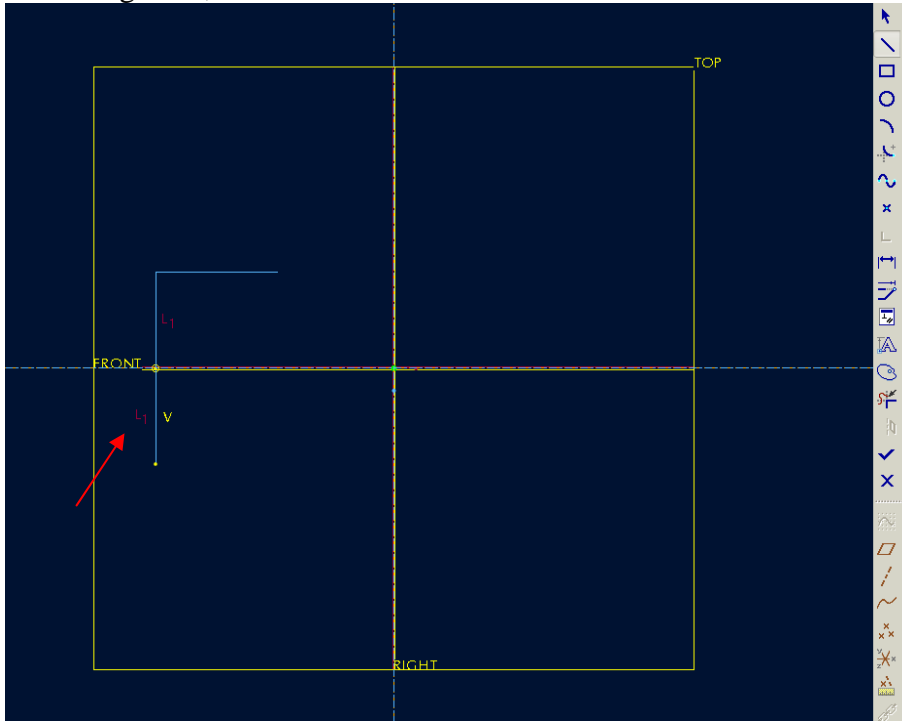
- Click on the drop down menu next to the blue line
- Select “Create 2 point lines.”



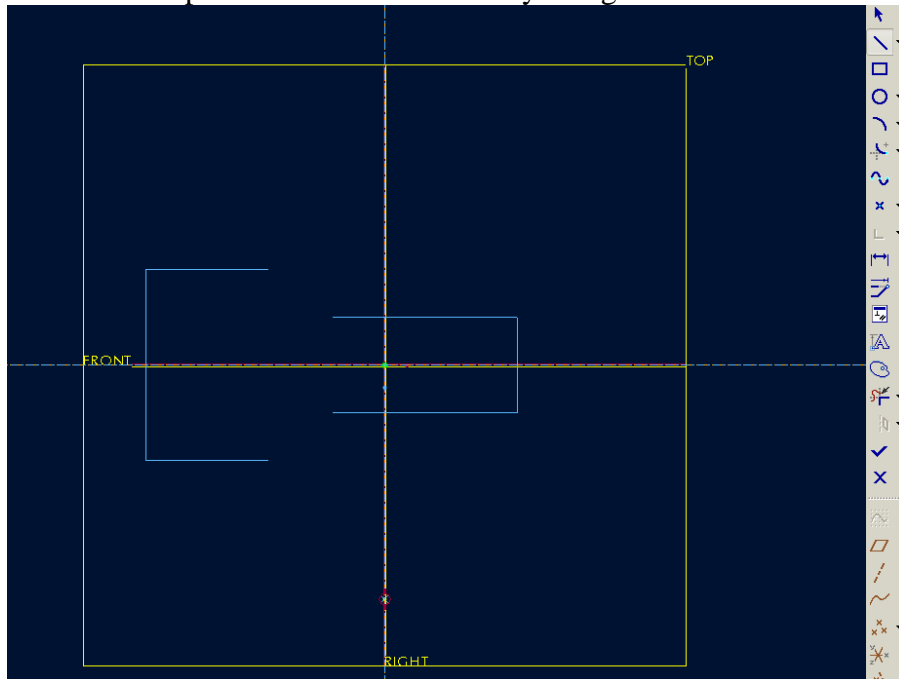
- Click on the screen to start creating a line
- Click again to create another point



- The letter L_1 shown in red indicates that two lines have equal length
- To end drawing lines, click on the center button

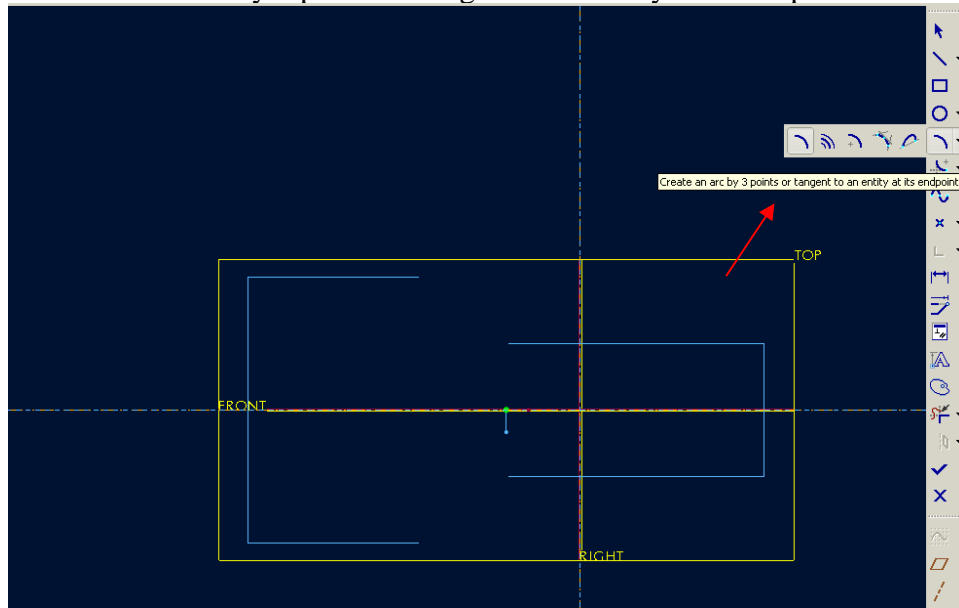


- Follow the same steps to draw all the necessary straight lines

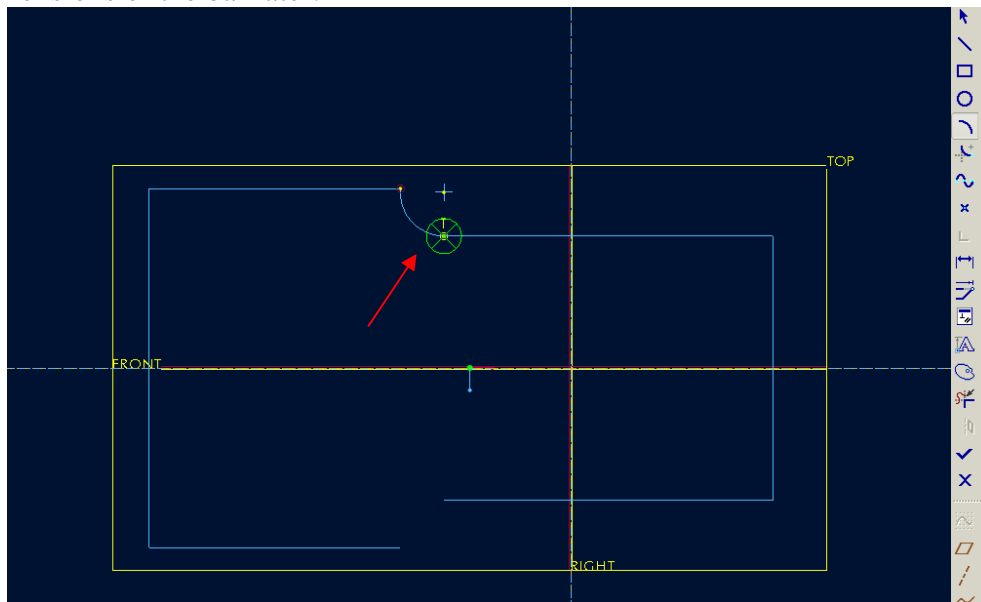


Sketching Curves:

- Click on the drop down menu next to the quarter-circle
- Select “Create an arc by 3 points or tangent to an entity at its endpoint.”

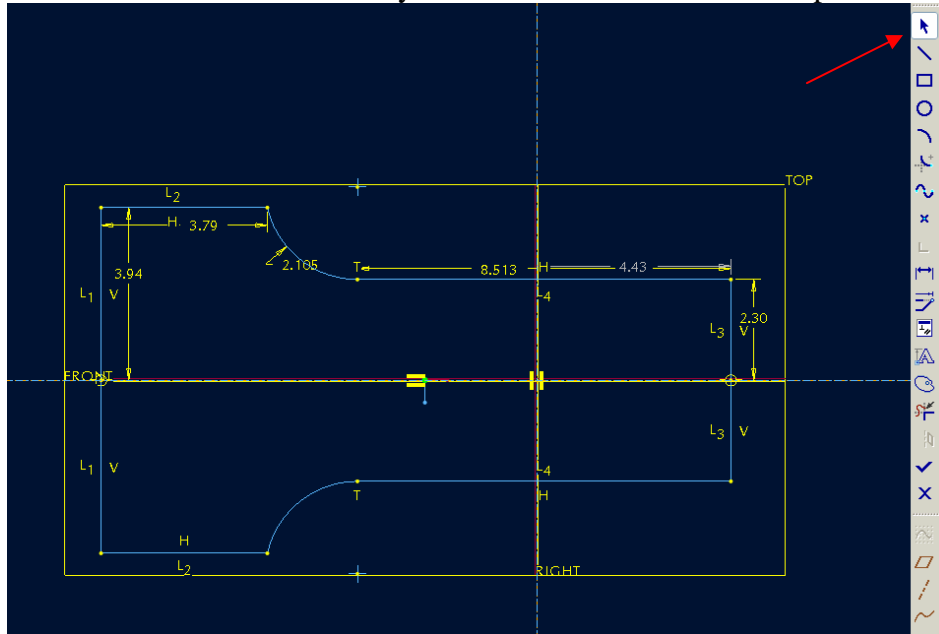


- Click on the end-point of one of the lines
- Make sure that the curve is tangent to the line (indicated by the yellow letter “T”)
- Click on the end-point of the other line
- Note: Sometimes you can change the radius of the curve after clicking on two end-points by moving around the mouse. If this happens, you can finish creating the curve by clicking again.
- ***Don't worry if you cannot adjust the radius of the curve. You can change the dimensions of the bar later.

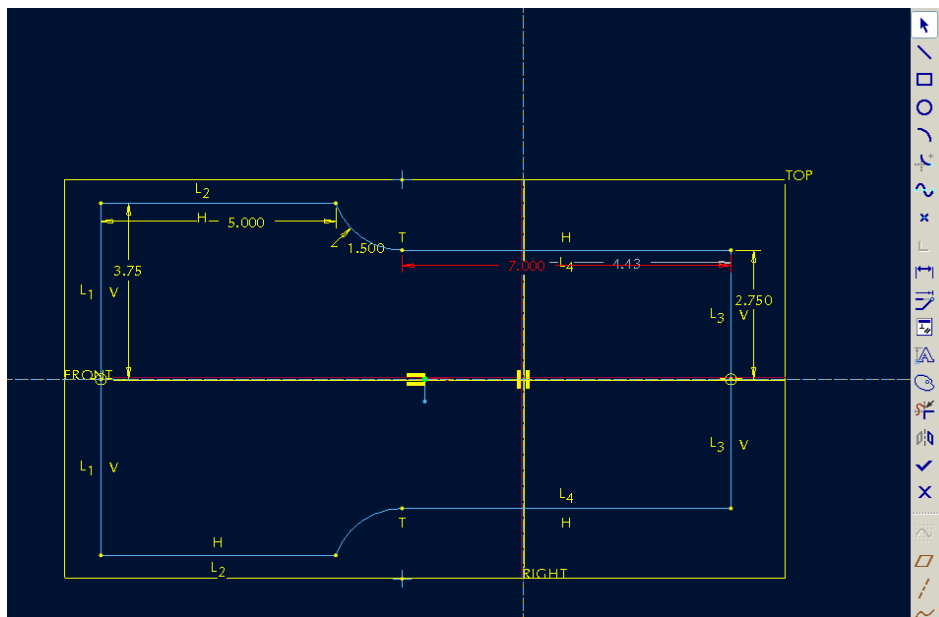


Changing the Dimensions:

- Click on “Select items” to show the dimensions
- You can change the dimension by double clicking on the number
- You can also change the dimensions by clicking and dragging on yellow points
- ***Make sure that the dimensions you choose are within the ratios presented in the table



(Before changing the dimensions)

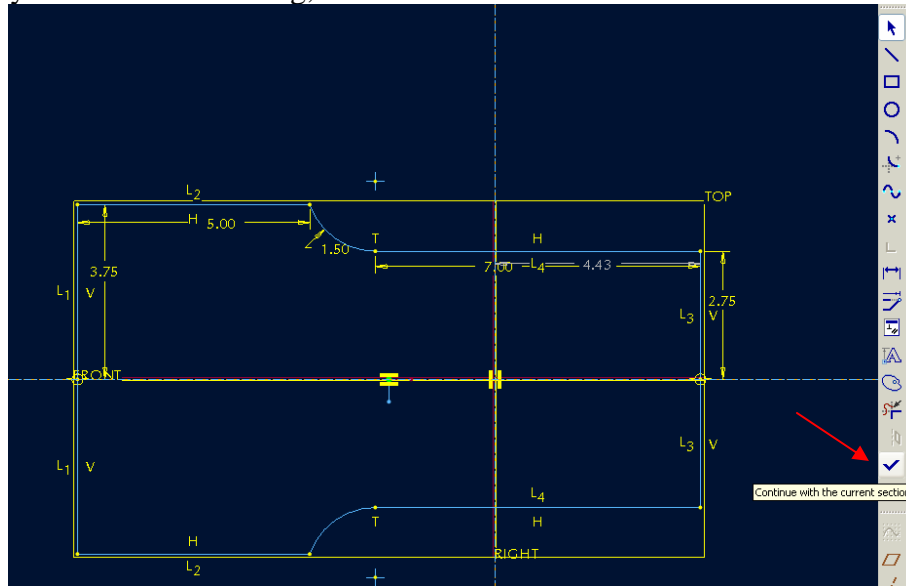


(After changing the dimensions)

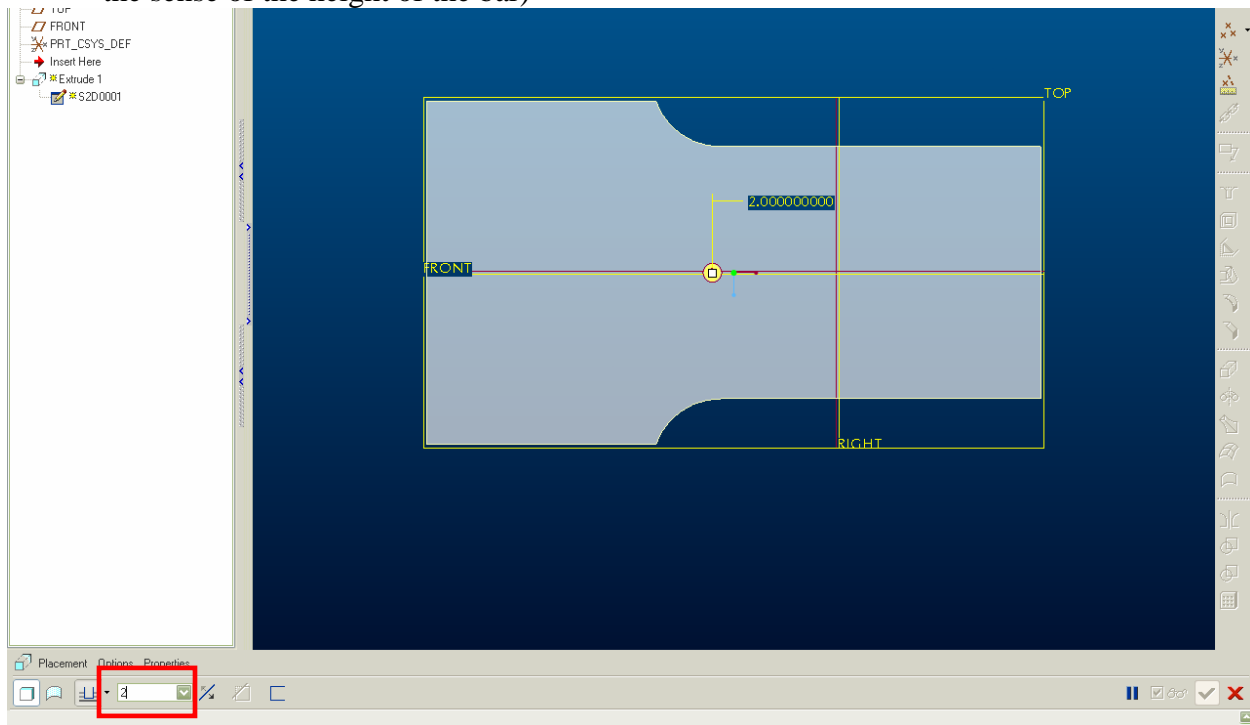
Also, the smaller the dimensions, the less time the computer will need to run a simulation in lab 2. Anywhere from an overall length of .5 to 12 inches will work fine.

Choosing the Height:

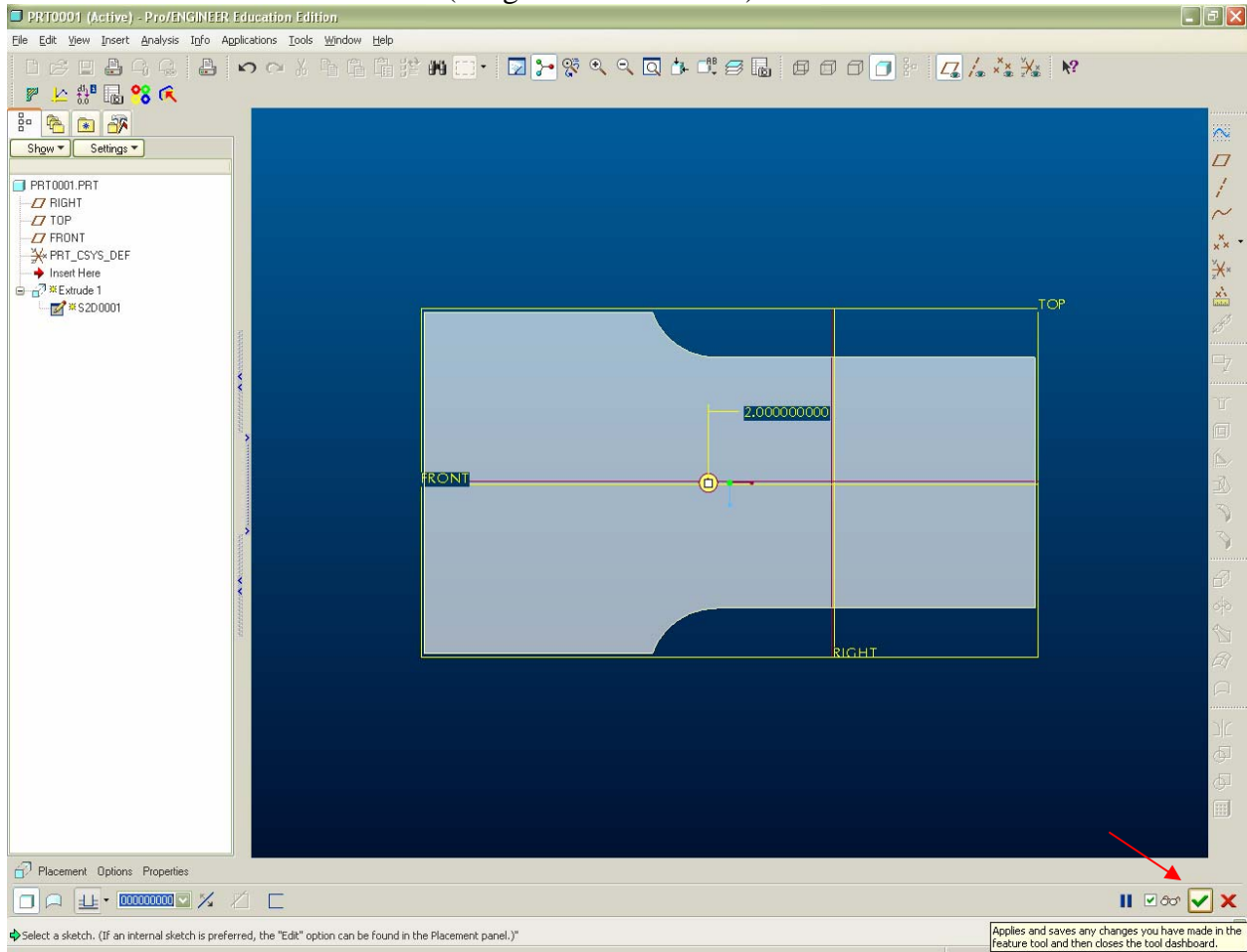
- Once you are done sketching, click on “Continue with the current section.”



- Choose the height/depth of the bar by entering the dimension in the box
- (You can move the mouse while clicking on to the middle button to rotate the bar to get the sense of the height of the bar)



- Click on “Applies and saves any changes you have made in the feature tool and then closes the tool dashboard” (the green check button)



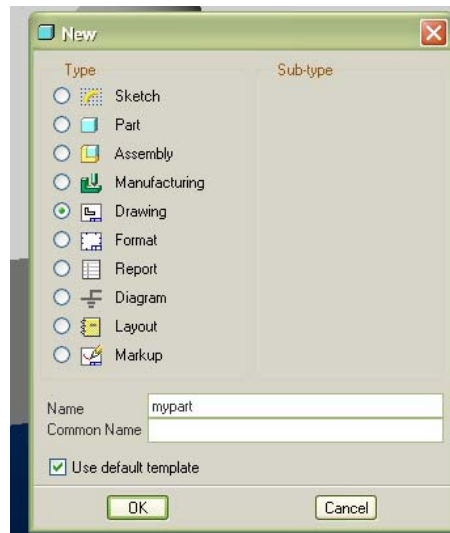
Now that the part has been created, you'll need to print out a 3D, color image of it. Rotate the part to a nice angle to see the depth, and then print it.

(A quick and fast way to print is to use the 'Print Screen' button on the top right of the keyboard. For anybody who hasn't used this technique before, whenever you press this button it makes an image of your current computer screen. Then, you can open up MS Word, and press paste: the image will be placed on the word document. Very handy!).

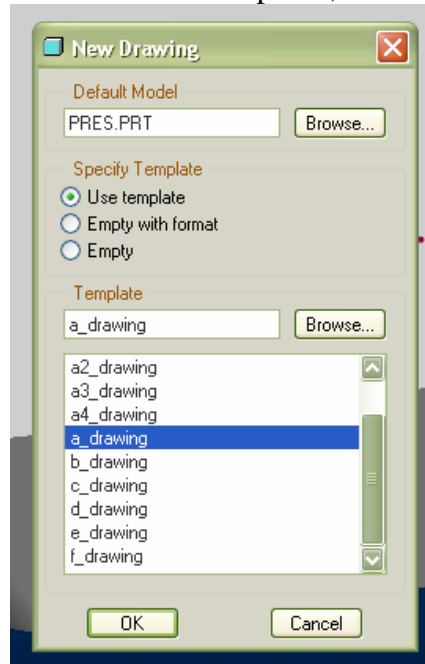
Part 3: CAD drawing

The final thing to do for this lab is to create an engineering CAD drawing of the part. Once the part is completed, and you are viewing it in the main window,

- Select **File – New** from the top menu
- In the pop-up box that appears, pick **Drawing**
- Type in whatever name you like, keep ‘Use default template’ highlighted, and press **OK**

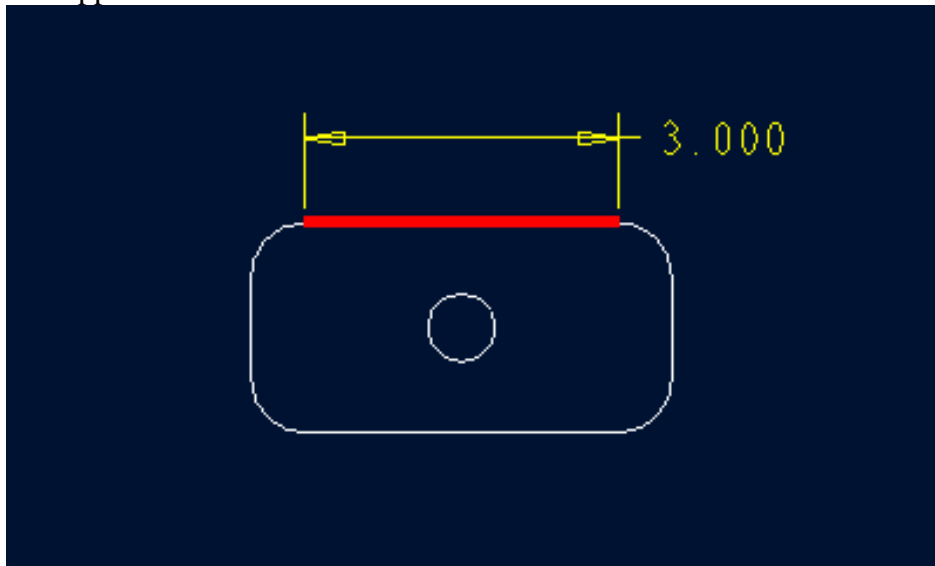


- Another menu will appear. Select ‘Use template’, ‘a-drawing’, and press **OK**

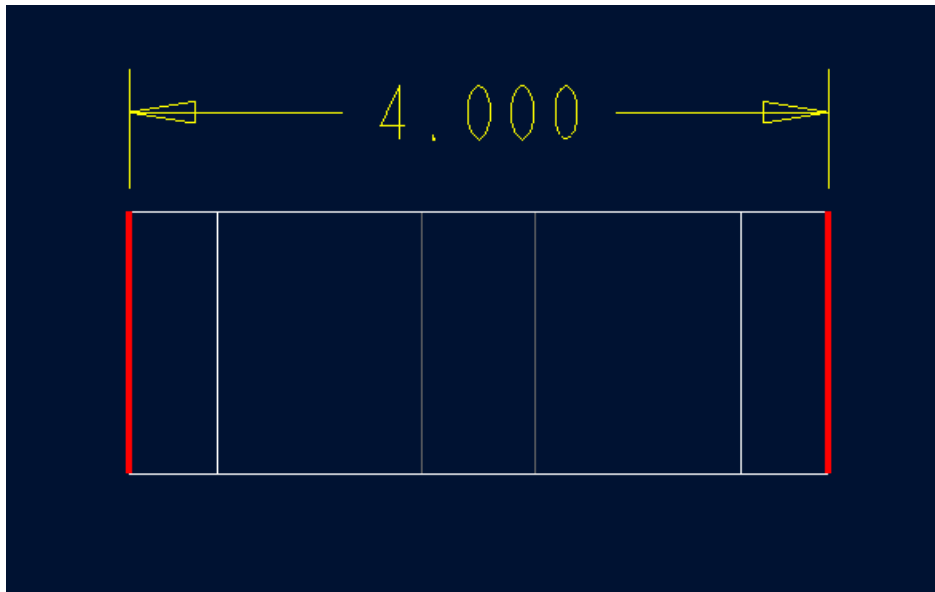


A CAD drawing will appear! Now you just have to add the dimensions. The easiest way to add dimensions is to go to the top menu and press ‘**Insert – Dimension – New Reference**’. The mouse will turn into a pencil icon.

The first way to create dimensions is to select a single line with the *left mouse button* (the line will turn red). Then, click the *middle mouse button*, and where-ever you click, the dimension will appear.

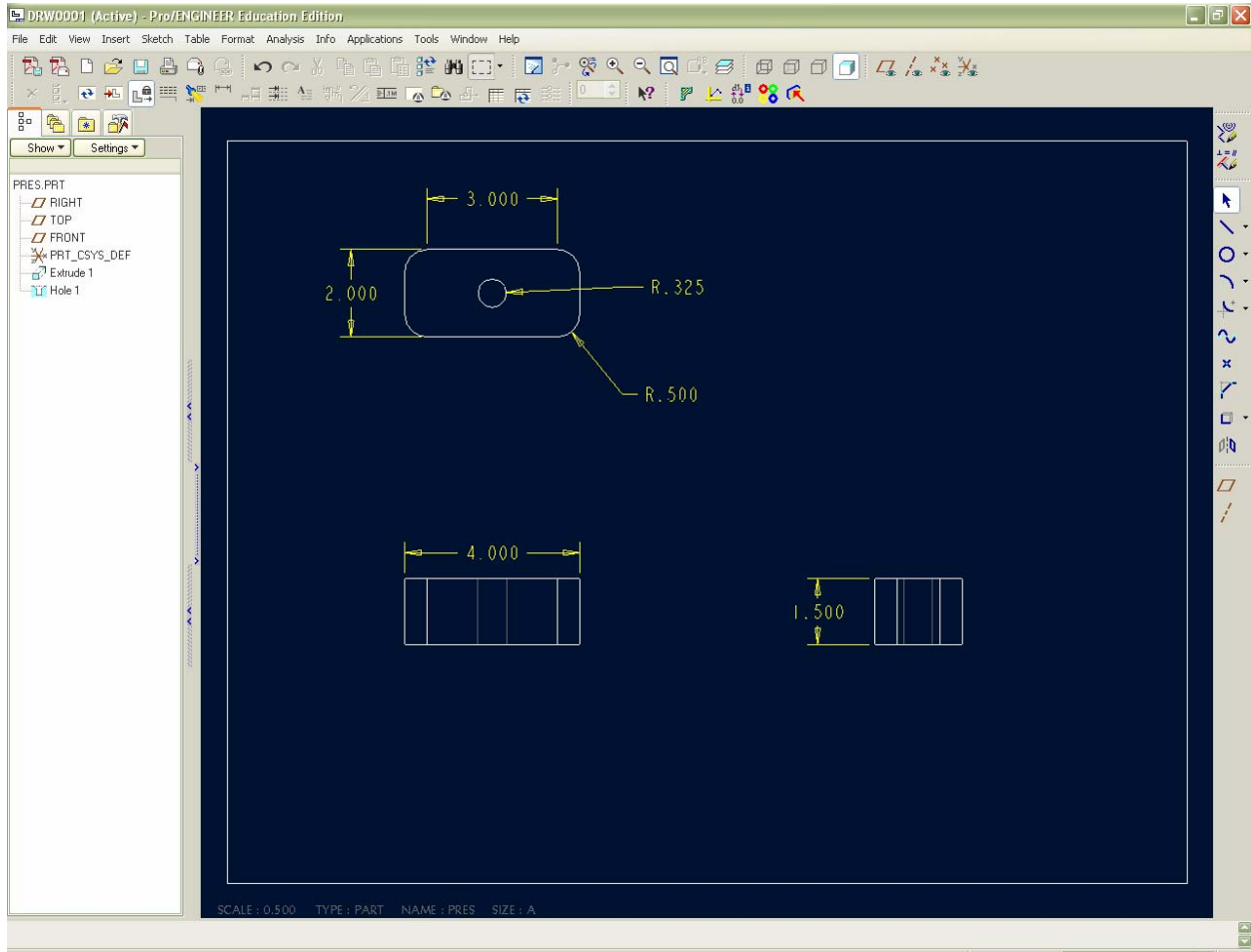


Or, you can select two parallel dimensions, by clicking on one then another with the left mouse button. Once both are highlighted, click with the middle mouse button off to the side, and the distance between the two lines will appear.



You need to make sure that a few specific dimensions are included:

- The dimensions from the chart ratios: **r**, **h**, **H**, etc, and the **depth**.
- Once the dimensions are all in place, print the drawing sheet out (this can be in black and white)



Summary

For the first lab, you will need to turn in three things:

1. The CAD drawing with appropriate dimensions
2. A 3D image of your part, in color
3. A few comments or thoughts on using Pro/E and this lab