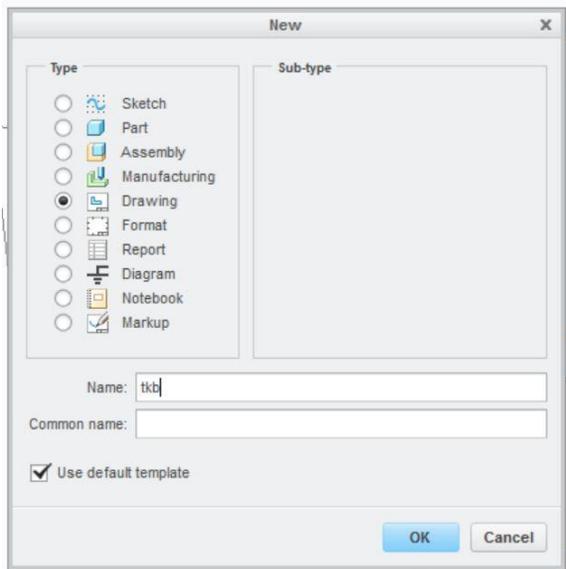


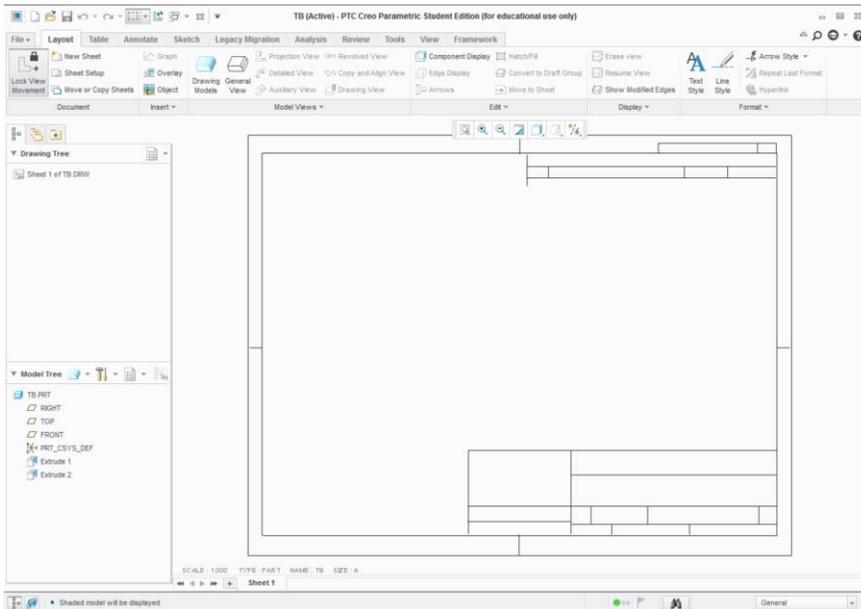
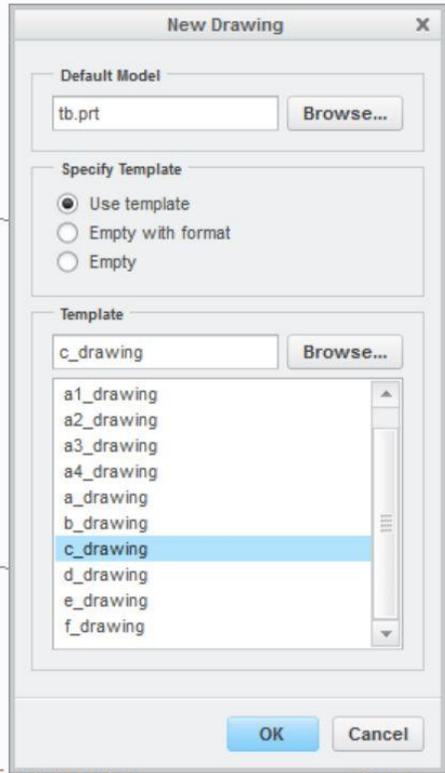
## HOW TO CREATE A DRAWING IN CREO

Step 1: have your 3D model open

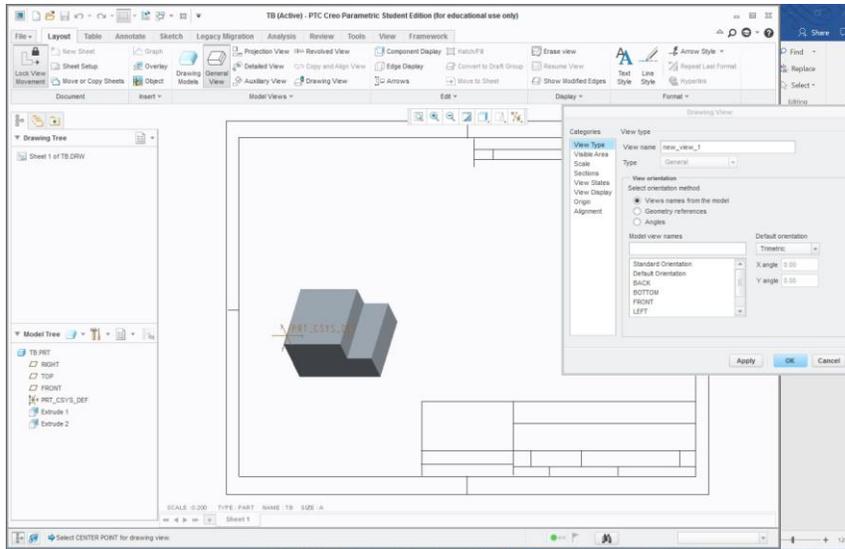
Step 2: Click file, new, drawing. The following menu should appear.



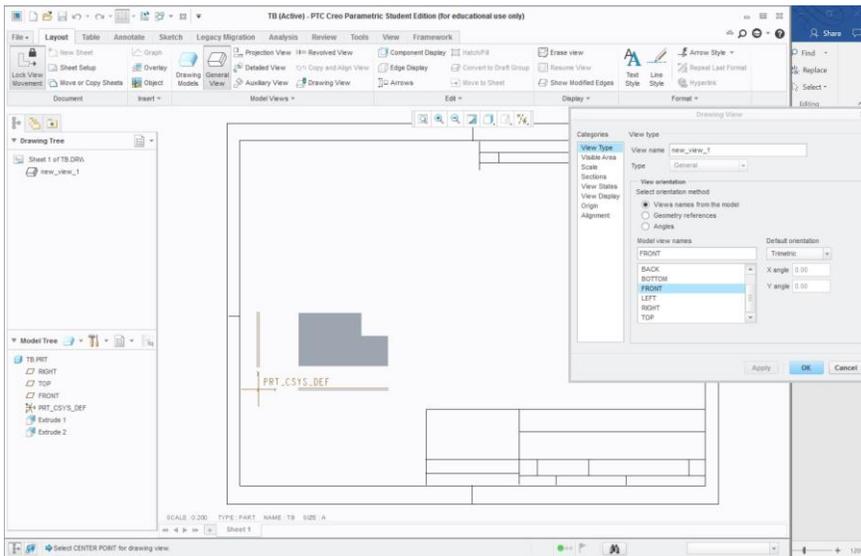
Step 3: Name your part drawing and click okay. The following menu will appear. Click empty with format and choose A.frm. The second image in this section is what your screen should currently look like.



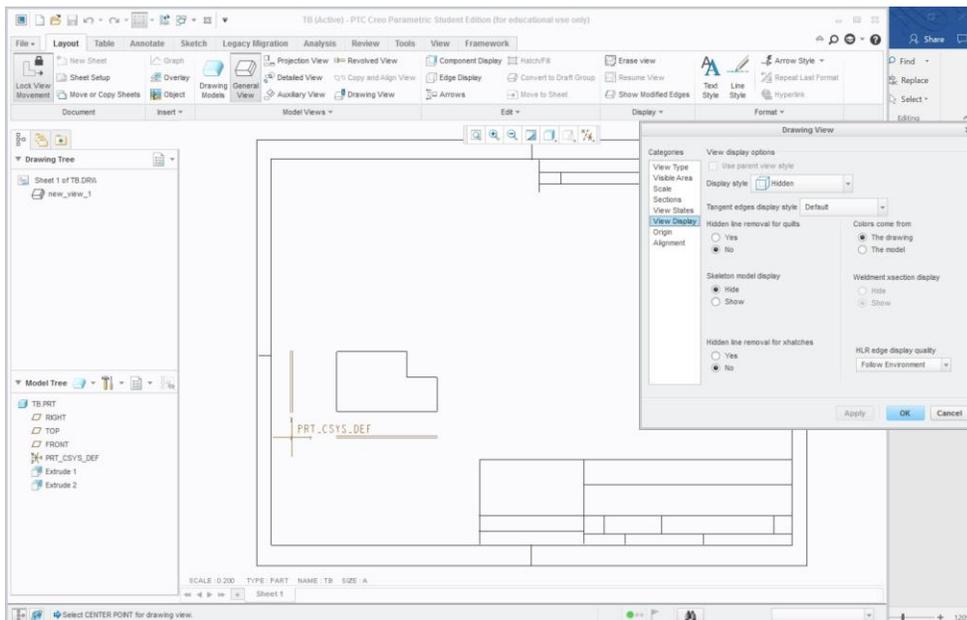
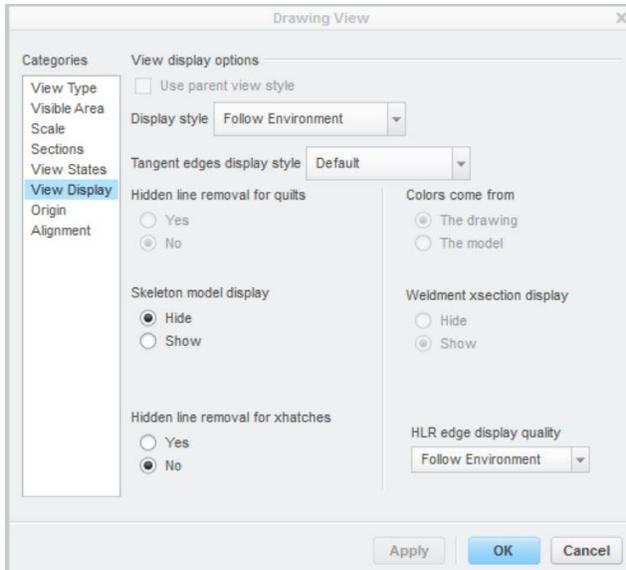
Step 4: On the top ribbon, click general view and okay and select the placement for your front view. Your screen should look like the image below, except with your part.



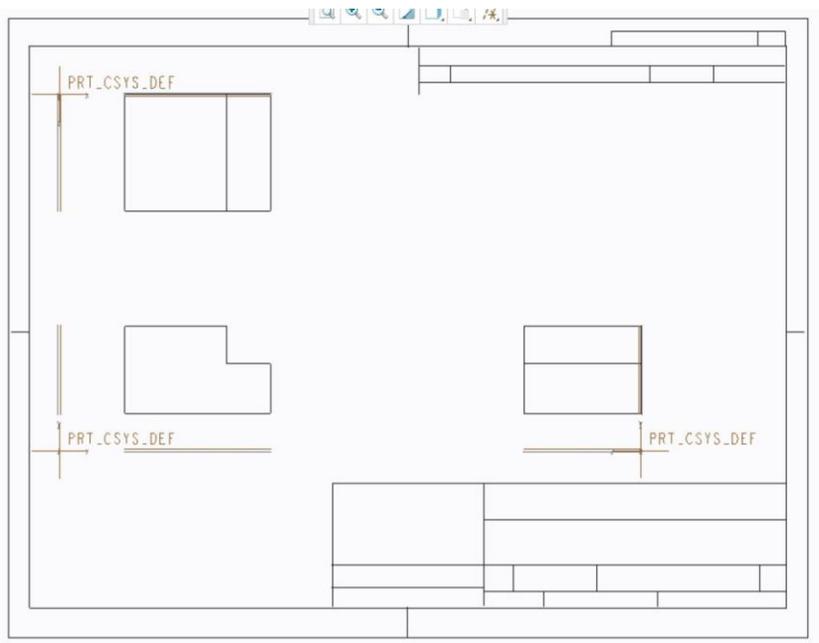
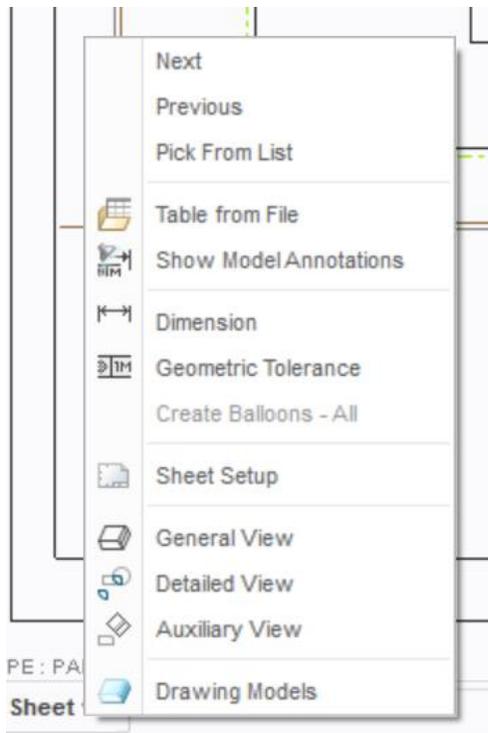
Step 4.1: if your part is at an angle like mine, follow the next direction. If not skip to step 5. In the box that says “Standard orientation, default orientation,” etc. Select front and click apply. If your “front” is not in fact the front of the part, select the view that is the front and click apply. It should now look like the image below.



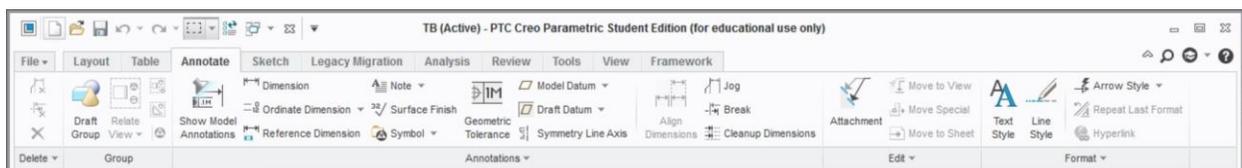
Step 5: Now click “view display” in the menu sidebar. The menu should now look like the image shown below. Change display style from follow environment to hidden. Your screen should now look like the second image shown in this section. Click okay to close the menu.



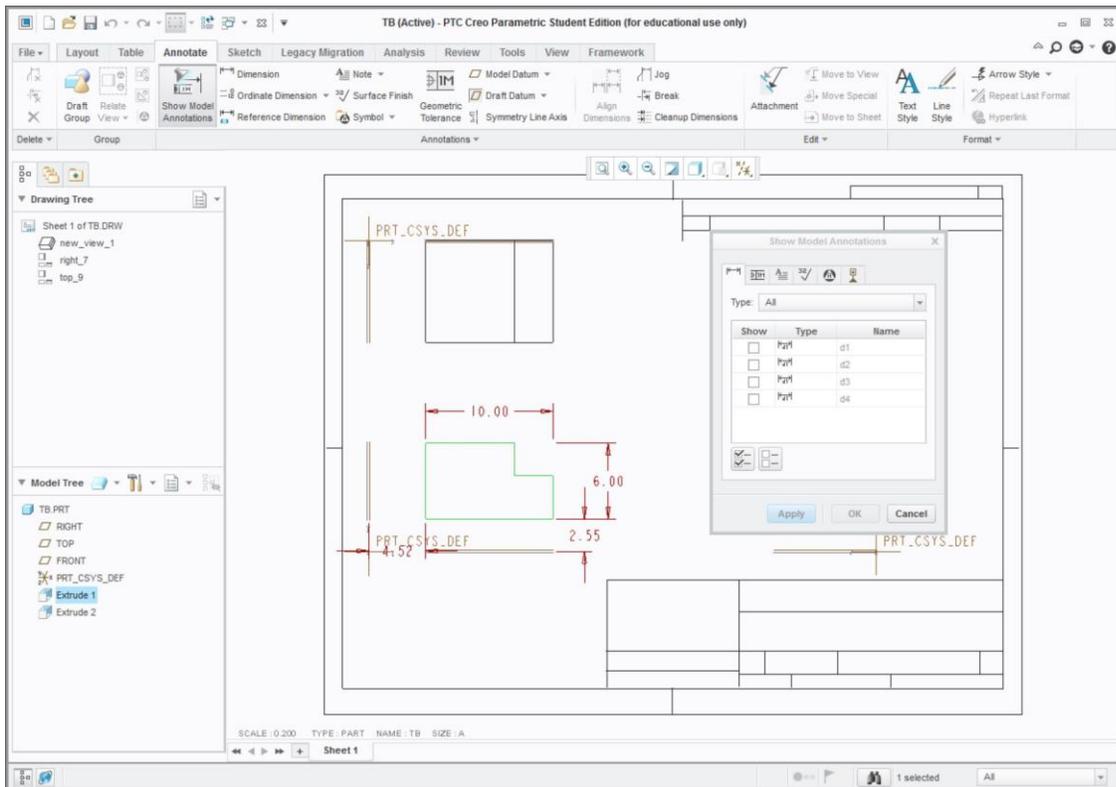
Step 6: From here we will make projection views from the front view. Click the front view and hold the right click button until a new menu appears. The menu is shown below. Here you can select “projection view” and drag your mouse to the side or above the front view, depend on which view you would like to add. Click the screen where you want to place your new view. Shown in the second image of this section I have added both the top and the right views of my part. To remove the shading of your views, repeat step 5 for each view. That menu will appear if you right click, hold and select “properties” for each view.

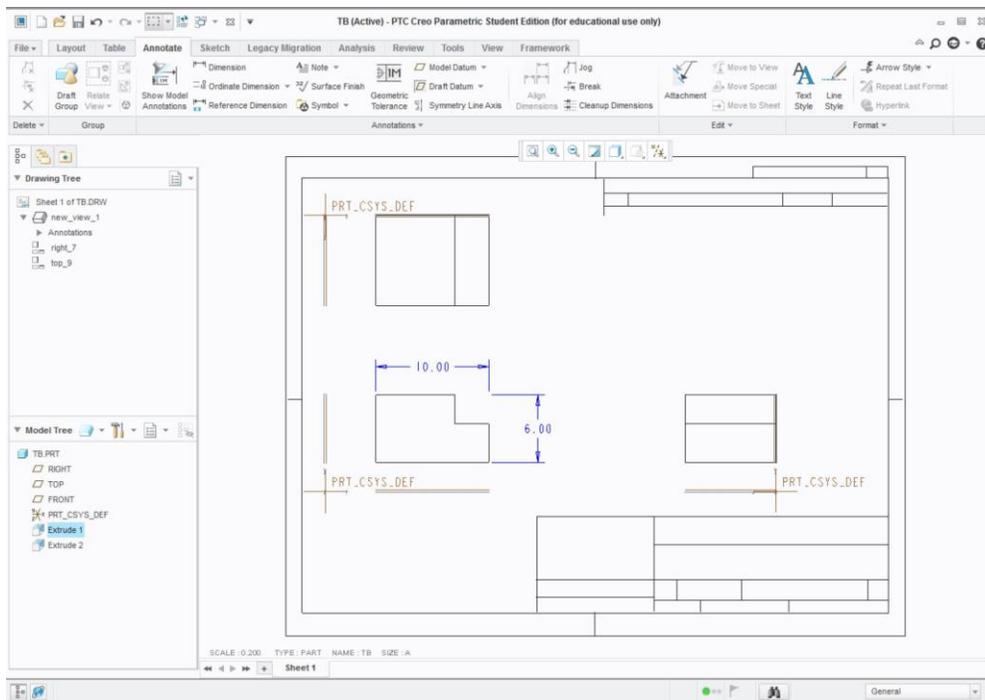


Step 7: You can add dimension in two ways. To do either way you can select “annotate” in the top ribbon. The annotate ribbon is shown below.

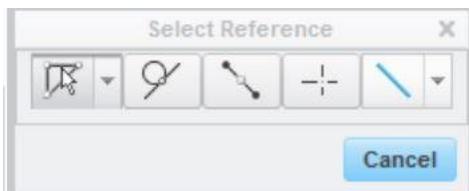


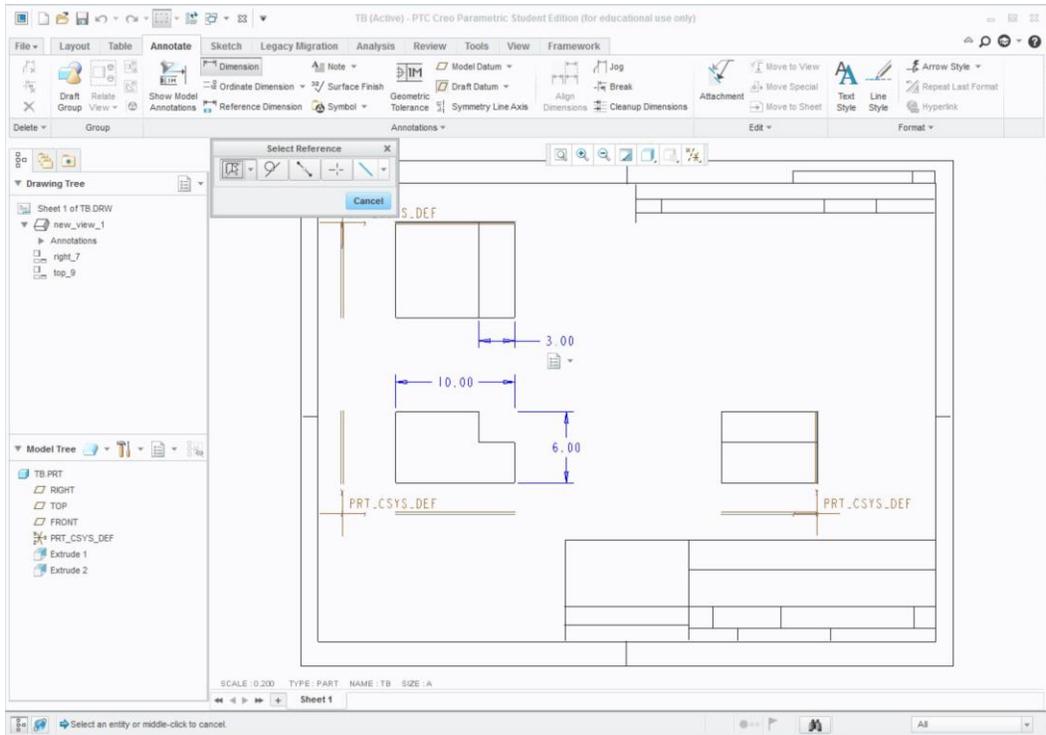
Step 7.1: The first way to dimension a view is to select “Show model annotations.” Once selecting this from the ribbon, select the view you would like to make dimension on. In the image below I chose my front view. The menu shown allows you to select the boxes for which red dimensions you would like to keep. You can also select the dimension you want to keep directly from the view. Once you have selected the dimensions you want, you can click apply to save the selection and cancel to close the menu. Then the second image shows the selections I made after clicking apply and cancel.





Step 7.2: What if the dimensions you want do not appear. This brings us to the second way to dimension your part. Click the “dimension” button in the top ribbon. Here the shown menu will appear. If you would like to dimension the length of a line, diameter of a circle, or radius of a curve you can simply select the object and middle click where you want the dimension to be placed. However, to dimension a distance between two lines or objects, you must click the first object, hold the CTRL button, click the second object, and then middle click to drop the dimension in place. The dimensions will be grey until you middle click a location. They will then turn blue when saved in place. When you are done dimensioning, click cancel in the menu and save your drawing.





Step 8: Check to make sure all of the needed dimensions are shown and are placed neatly. Save your drawing in the same folder you saved your part. If you are sending the drawing to someone else, not as a pdf, you must save the drawing and part in a zip folder together and then send the folder. They will not be able to open the drawing without the part.