Creo 2.0, Basic Modeling Tutorial

Trust me, I'm an Engineer
1. Open Creo Parametric 2.0

![Creo Parametric 2.0 Icon](image)

2. Hit Select Working Directory on the top bar and select whatever folder you want your new part to go into.

![Select Working Directory](image)

3. Next hit the New Button, make sure the type is set to part. Change the name to whatever you want to name your part and hit OK.

![New Part](image)
4. Your screen should now look like this. Hit the View tab along the top ribbon and hit the plane tag display button so you can see the names of the planes.

5. Next, select the Top plane, return to the model tab, and hit the extrude button.
6. Then you should be in sketch mode. Hit the sketch view button on the small ribbon in the work area. So that you are viewing the top plane in 2D. Then select Rectangle in the sketch ribbon and click and drag a rectangle in your workspace.

7. Once you click again and then click the select button you will see that inside the rectangle turns orange, and dimensions appear. Change the dimensions so that it has a length and width of 6.00 and the dimensions to the centerlines are 3.00. This is where you start to have freedom. If you want to make something smaller you can, but this is the maximum size you can have for your part. Once you are satisfied hit the check mark to finish the sketch.
8. Next, enter .25 as the thickness for the extrude. This is the thickness of the HDPE you will be cutting. Then hit the check mark.

9. Now, if you want to import an image to sketch over so you can cut it into your HDPE you go to the View tab, click the drop down menu of Model Display, and select Images.
10. Then you hit the Add button and select the top plane of your part. This will bring up browser that you can look through and find your picture in your computer.

11. Then place your picture where you want it and hit the check mark for OK.
12. Now return to the model tab and select the top plane of your part again and hit the Extrude button again. Put another Rectangle over your image. I would leave about .25 inches from the edge of your board.

13. Now you need to use a combination of lines and arcs to sketch out the image you want to have on your board. Then hit the check mark to show that you are finished.

14. Now to make this cut into the board instead of come out of it you need to set the depth to .125 and then hit the arrows next to the depth box to reverse the direction and the remove material button next to the reverse direction button to make it a cut.
15. Then go to the Image menu again and hit the remove button to take away the image you were sketching over.

16. Now hit the save button and you can move on to the Introductory Creo CAM Tutorial to make the .NC files to manufacture your model.