Two Ways to Skin a Cat: Dan's Plate

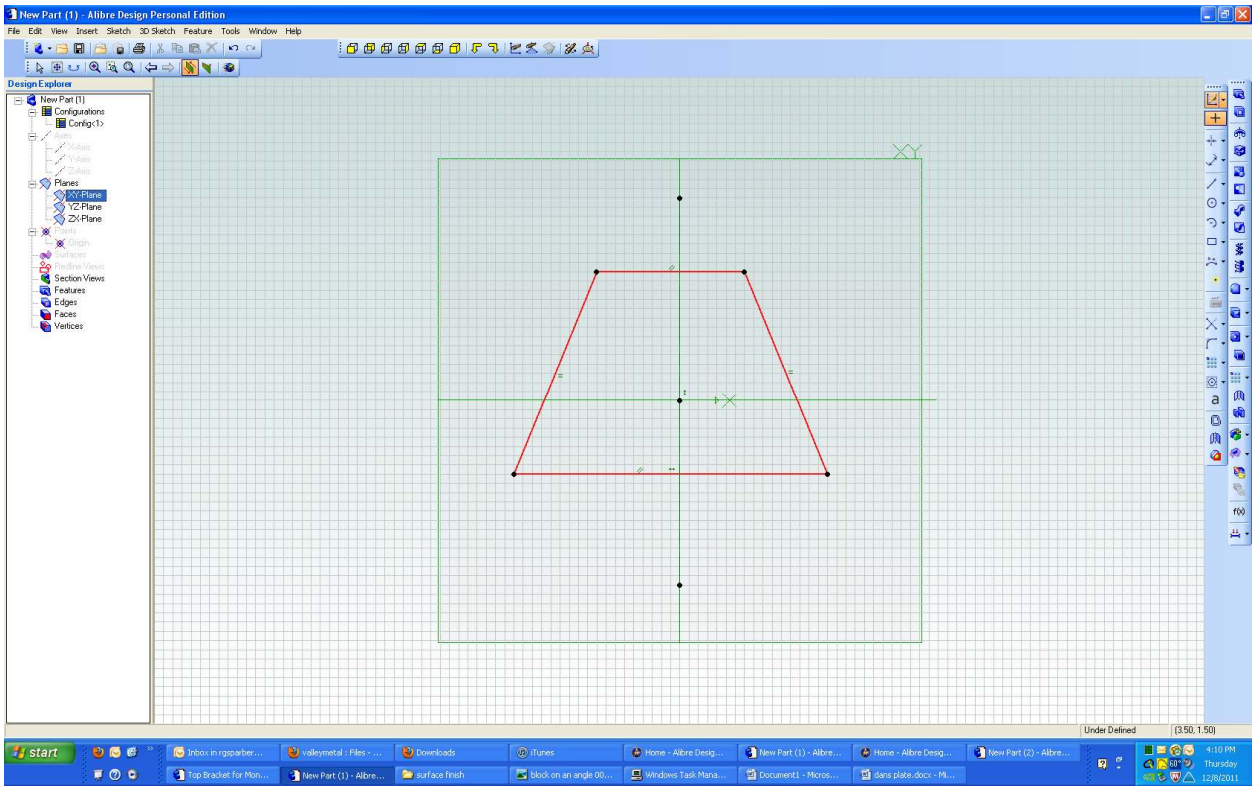
By R. G. Sparber

I will present two ways I have drawn a plate for my friend Dan using Alibre. If you have Alibre, I encourage you to follow along with this article in one window and the Alibre work area flanking it.

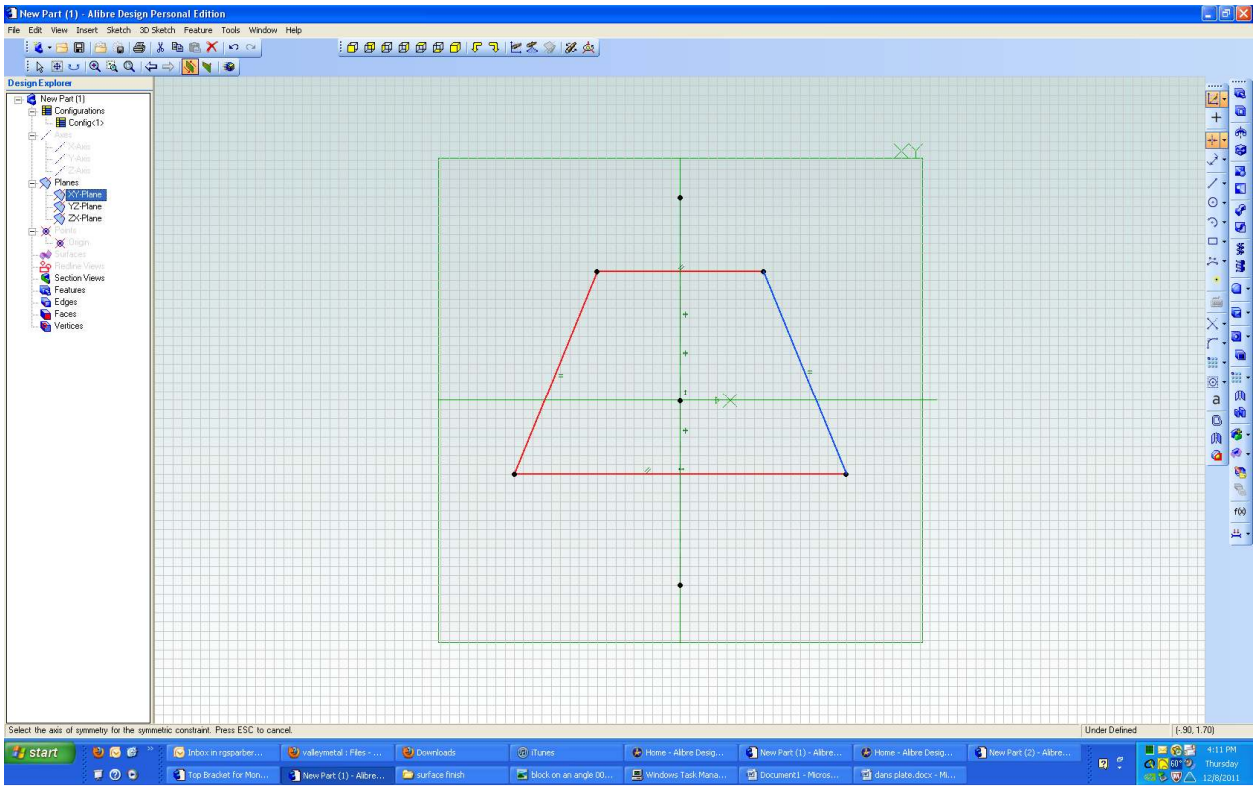
[1] You are free to copy and distribute this document but not change it.

First Method: Work Mostly in 2D

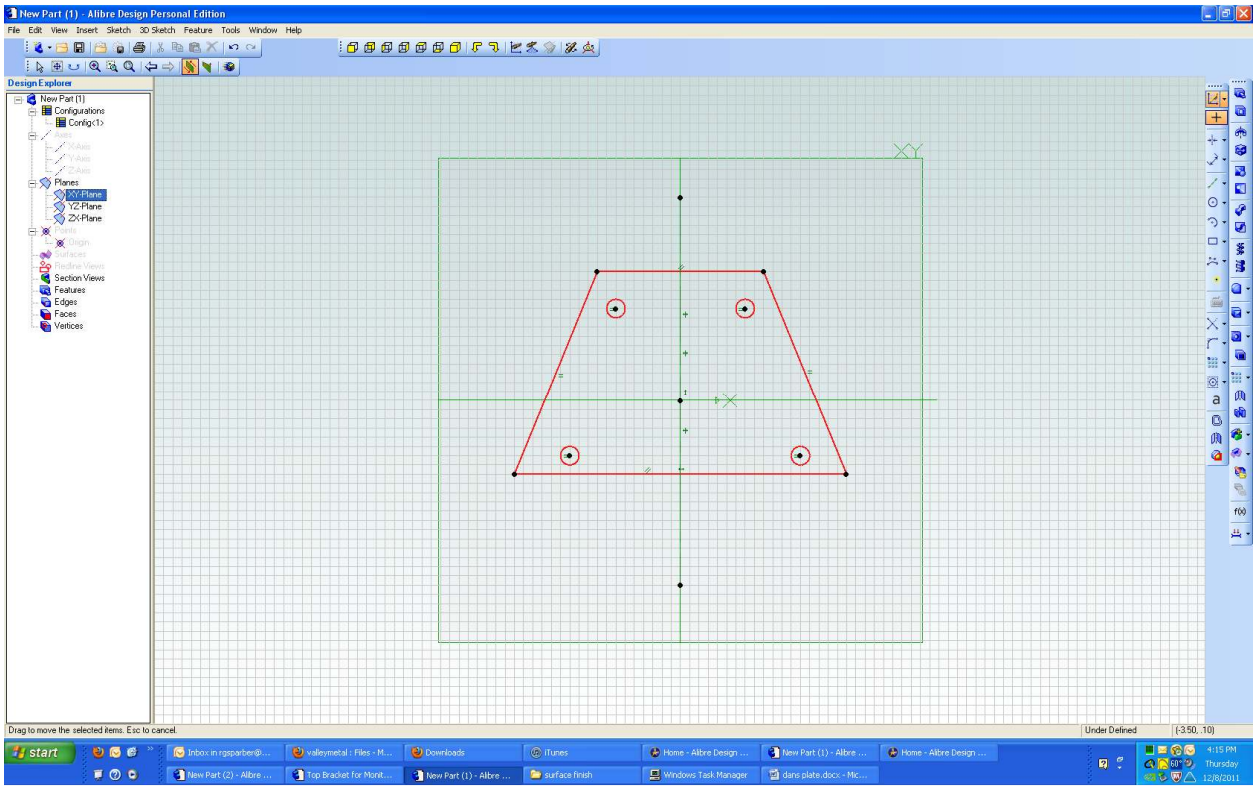
Advantage: one stop shopping
Disadvantage: you can only easily make straight through holes



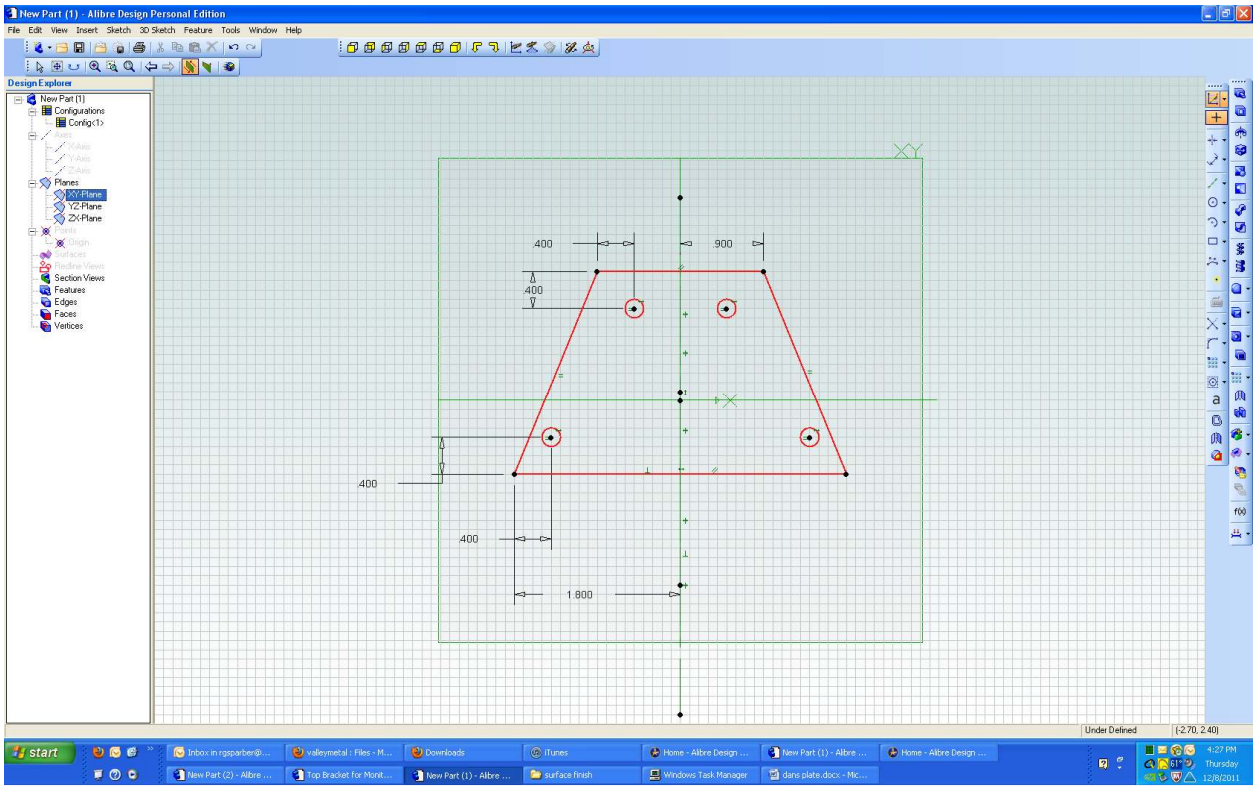
I start with my XY plane selected and draw a reference line vertically through the origin. The plate is symmetric about this line and I will call this my Line Of Symmetry (LOS). Then I drew my outline. It is not constrained to be symmetric yet.



Next I selected the symmetric constraint tool and clicked first on my LOS. Then I clicked on the two sloped lines which now snap into being mirror image.

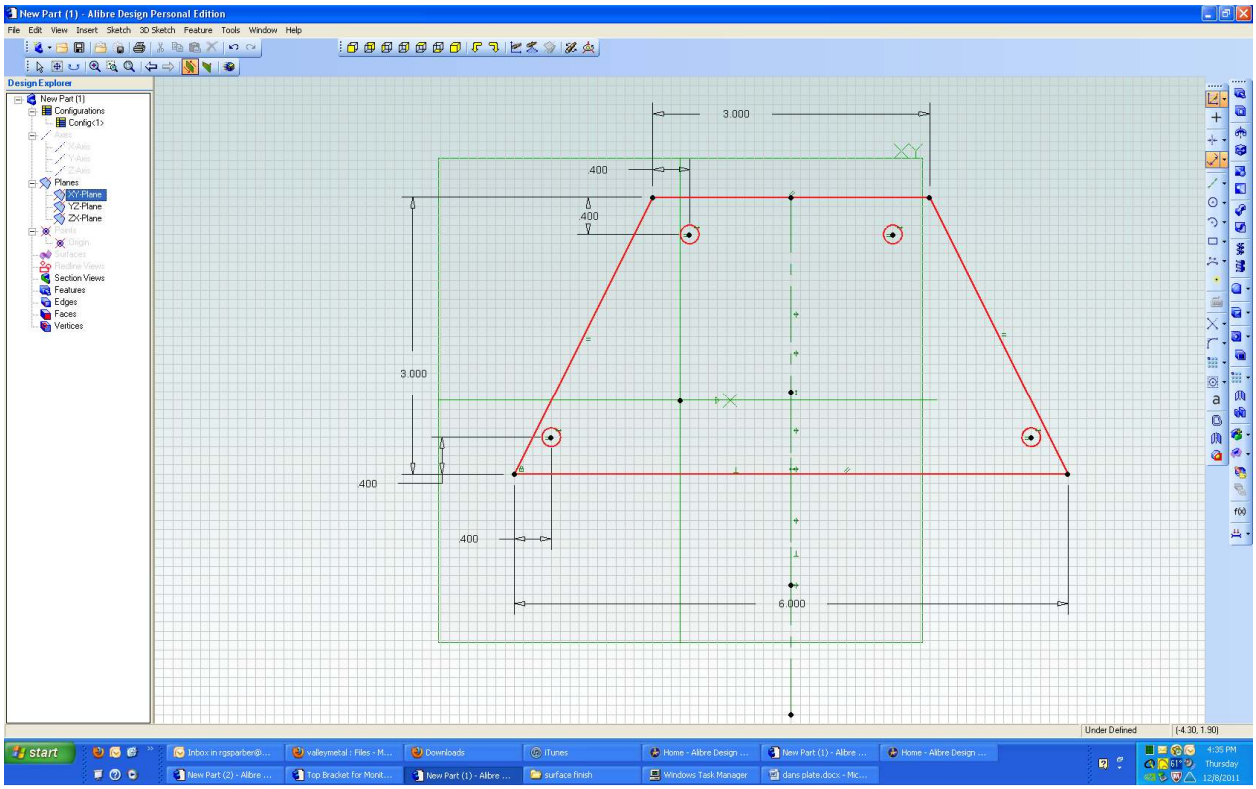


I have roughly placed the four holes. Since all holes are the same size, I drew my first hole, and then when I drew the rest, I adjusted the size until I saw an "=" sign. Their diameters are then locked to be the same.

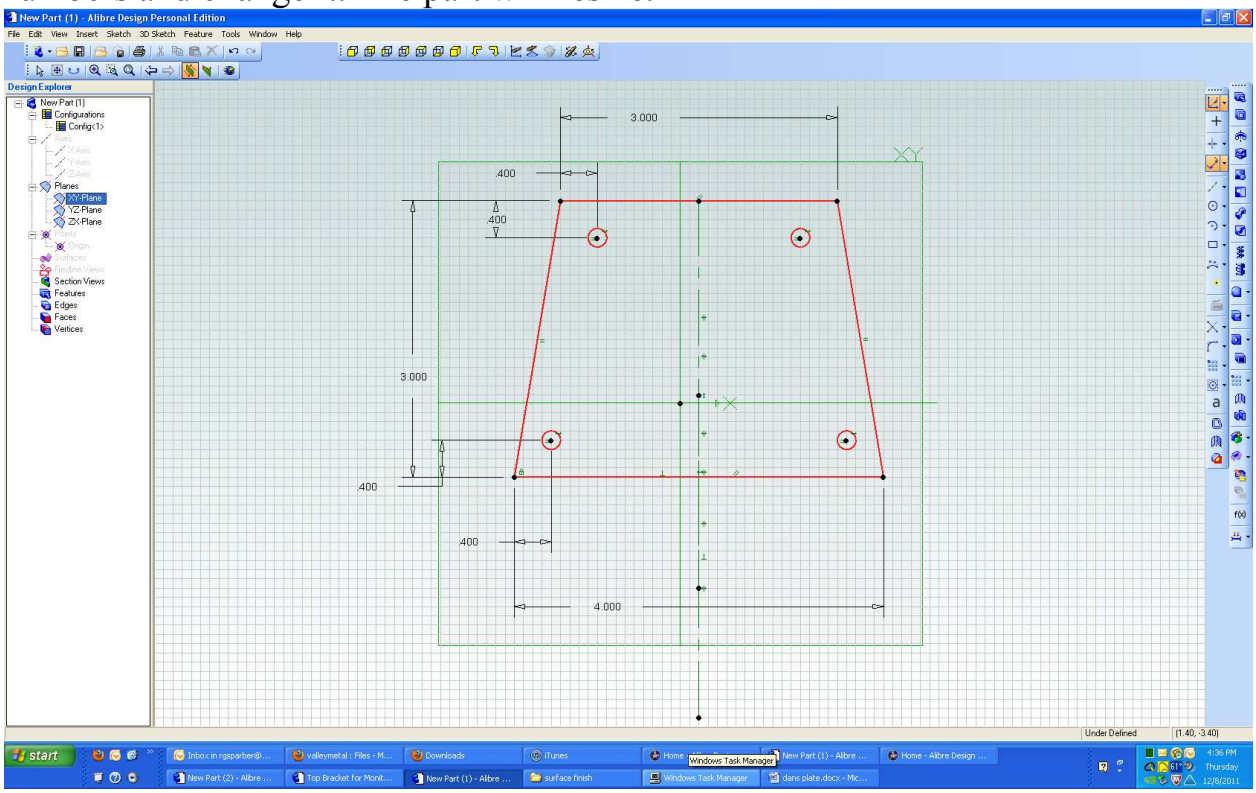


I placed my left two circles in their exact position relative to the corners of the part. Then I used the horizontal constraint to force the bottom right circle to be at the same height as the bottom left circle. The same was done with the upper two circles.

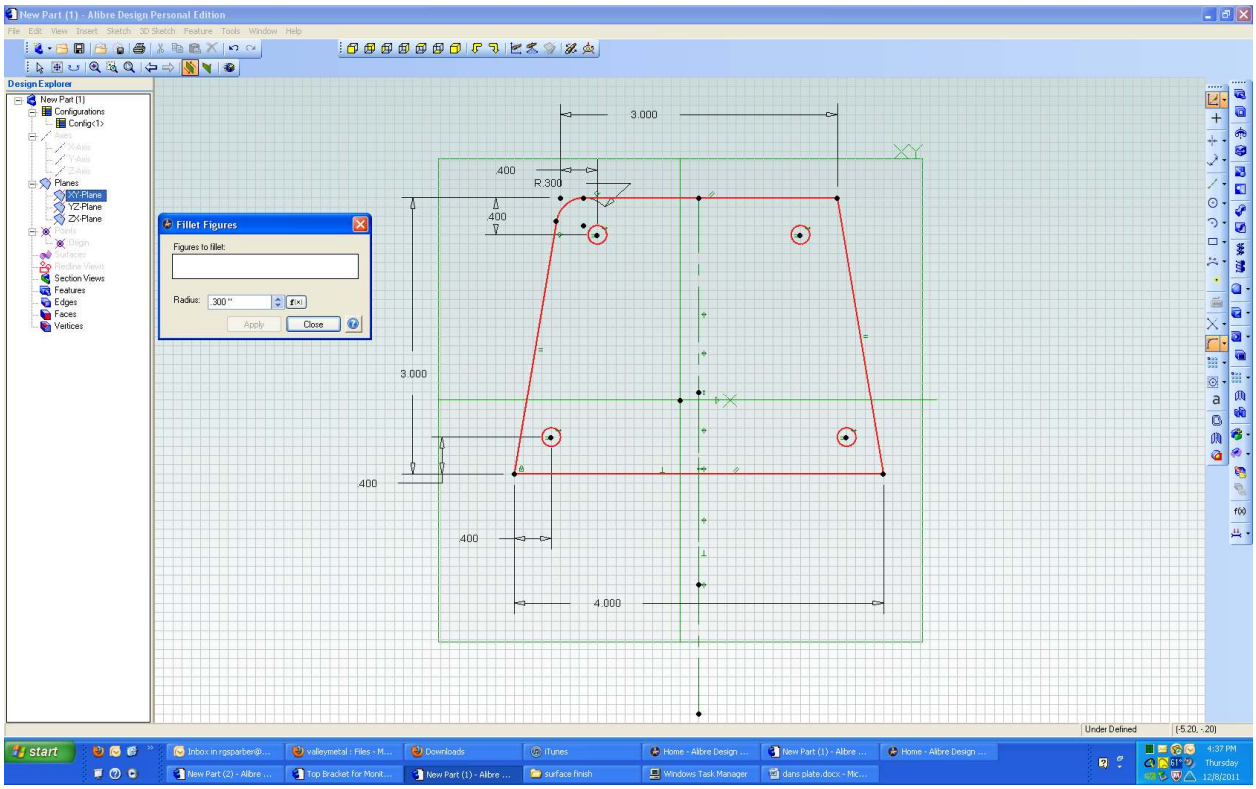
I then used the symmetric constraint to force the right upper circle to be the same distance from my LOS as the left upper circle. And finally, I did the same symmetric constraint for the lower right circle. Any change in position in the upper or lower left circles will be followed by the upper or lower right circles.



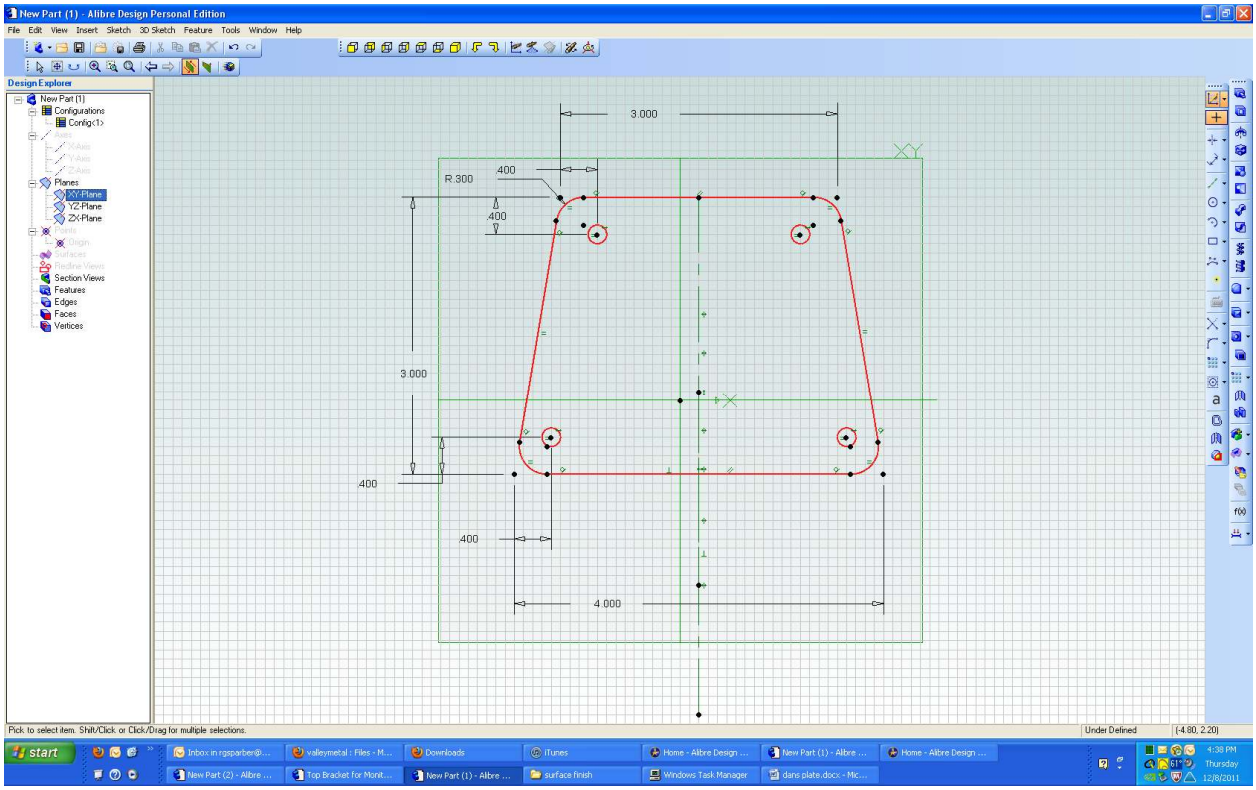
I have dimensioned the outline of the part. At any time I can click on one of these numbers and change it. The part will resize.



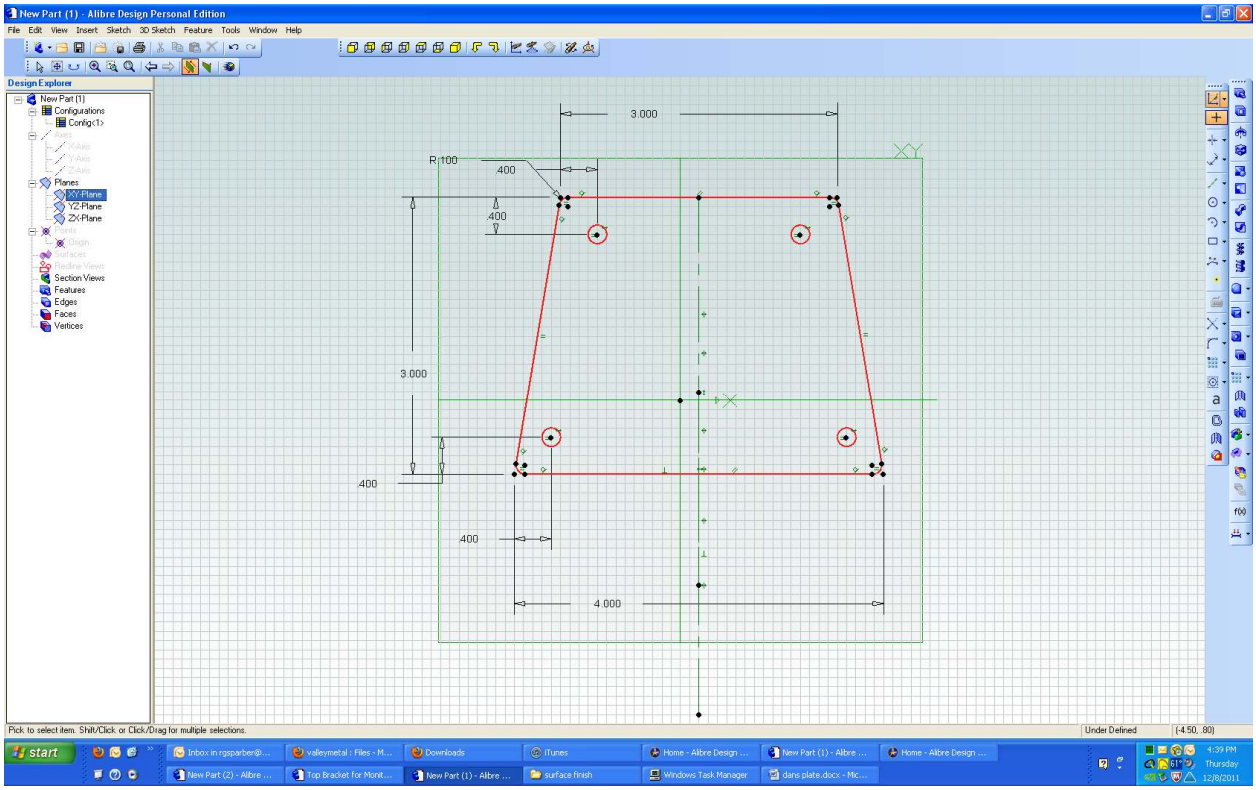
I just changed the base from 6” down to 4”.



Here I am using the fillet tool to round my first corner. I set the radius at 0.3”.



I did the other 3 corners. Note that there is an "=" sign on each radius. I used the same 0.3" and the program assumed I wanted all radii to be equal.

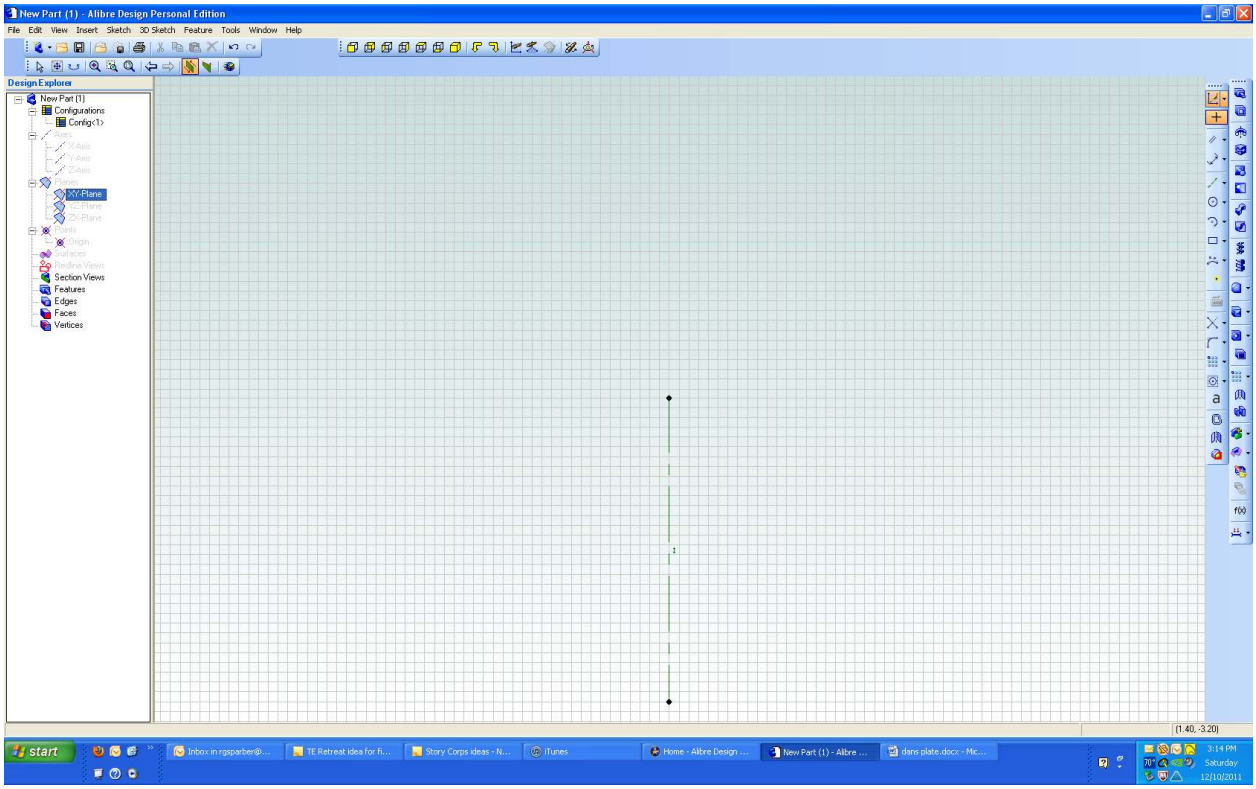


Here I have modified the radius of the upper left corner and all four corners are now at 0.1".

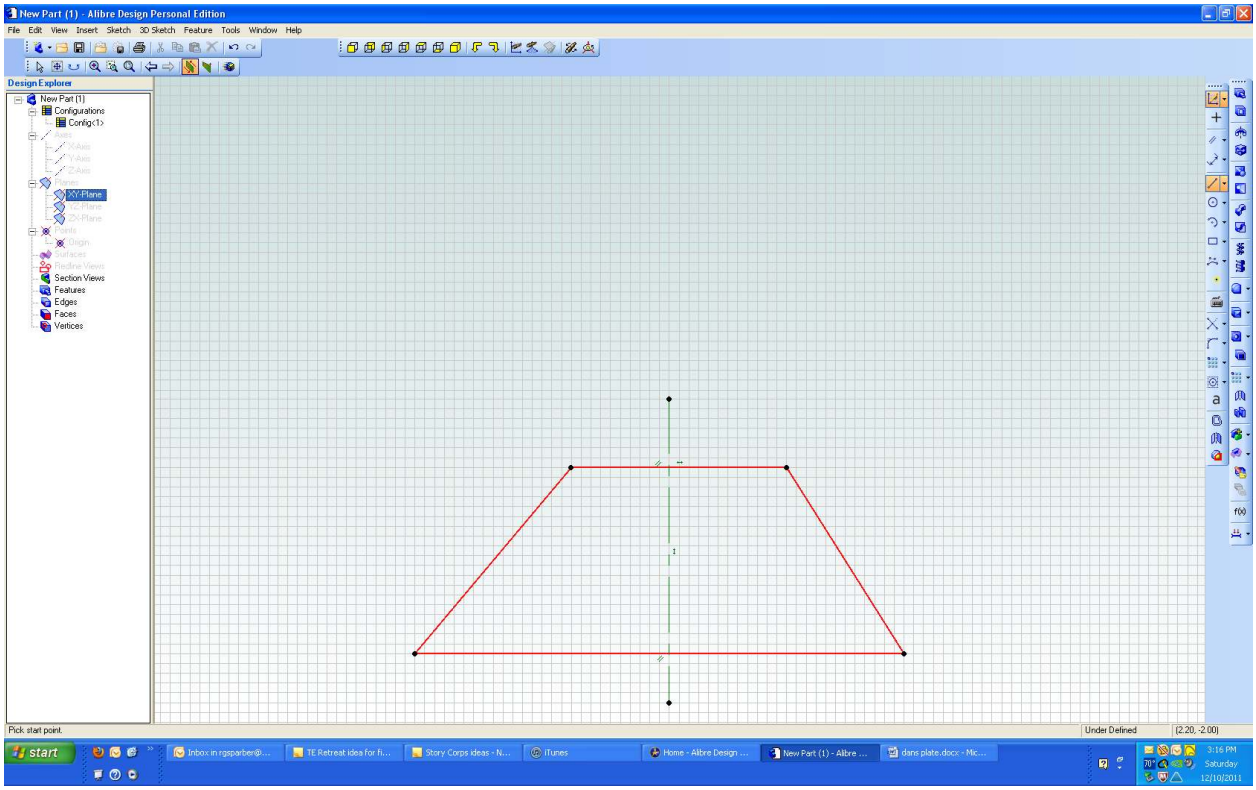
So, in general, the shape is correct but I know all dimensions are not what Dan needs. He just has to edit each dimension to bring the part into spec. Then just extrude this 2D drawing into a plate.

Second Method: Work Mostly in 3D

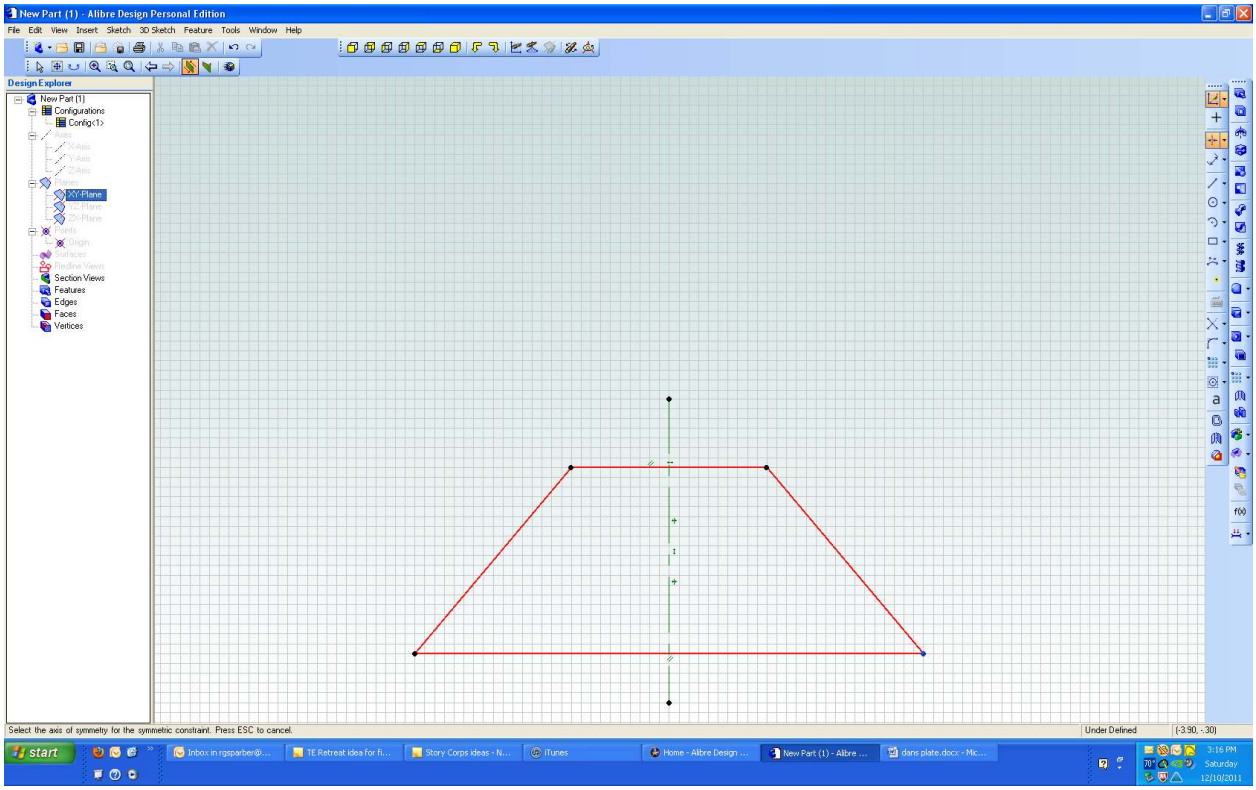
Advantage: holes can easily be counterbored, countersunk, or straight.
Disadvantage: it takes a few more steps



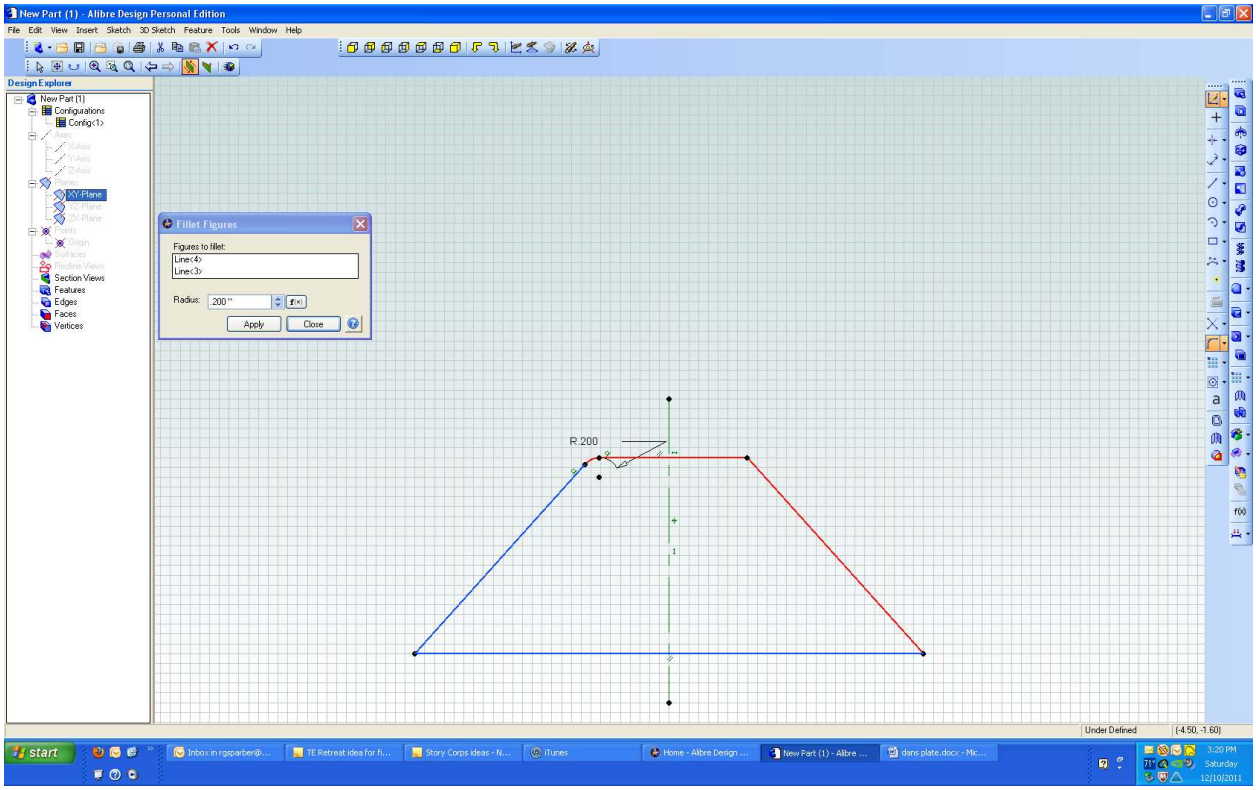
After opening a new window and selected the XY plane, I drew a reference line from the origin down to around 3".



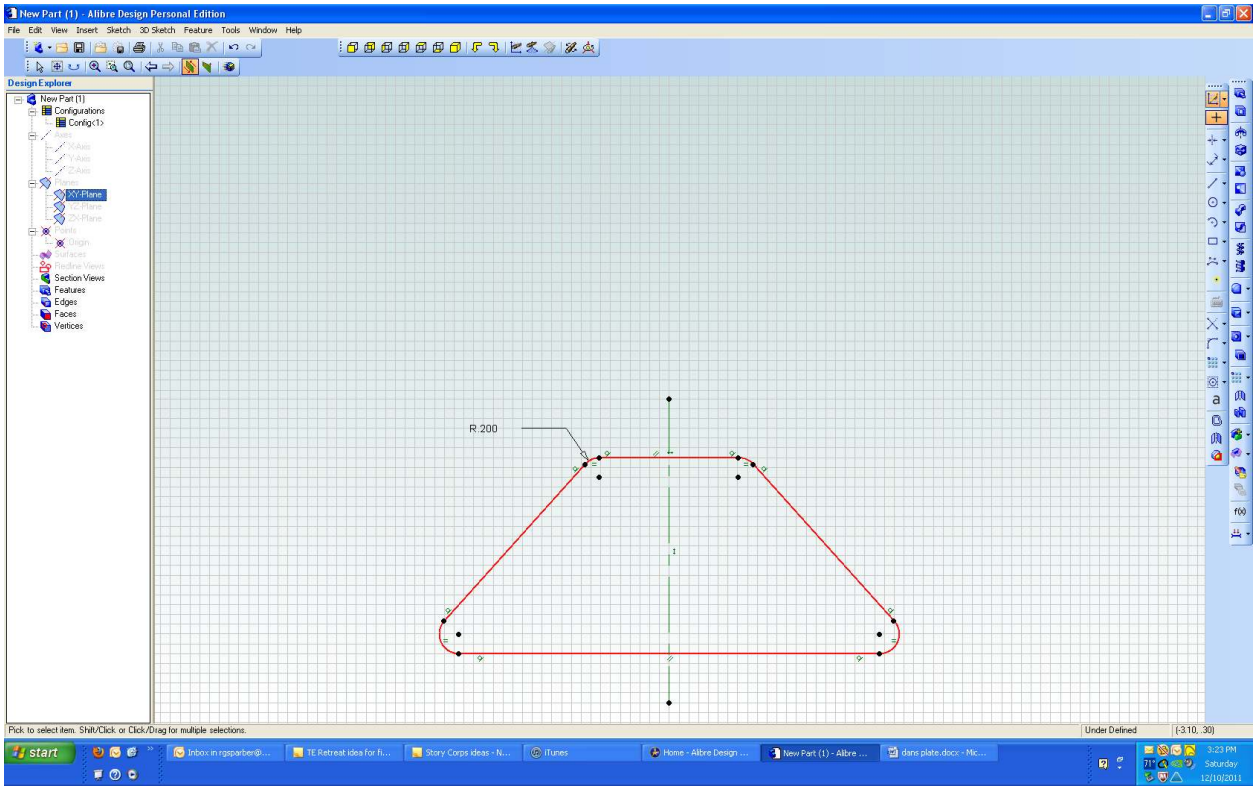
Changing to a solid line, I roughly drew the outline of the part.



Using the symmetry constraint tool, I defined the top two corners to be symmetric by first clicking my Line of Symmetry (LOS). Then I clicked the upper left corner and the upper right corner. Moving either of these corners will move the other one. I did the same thing with the lower left and right corners: click the LOS followed by clicking each of these two lower corners.

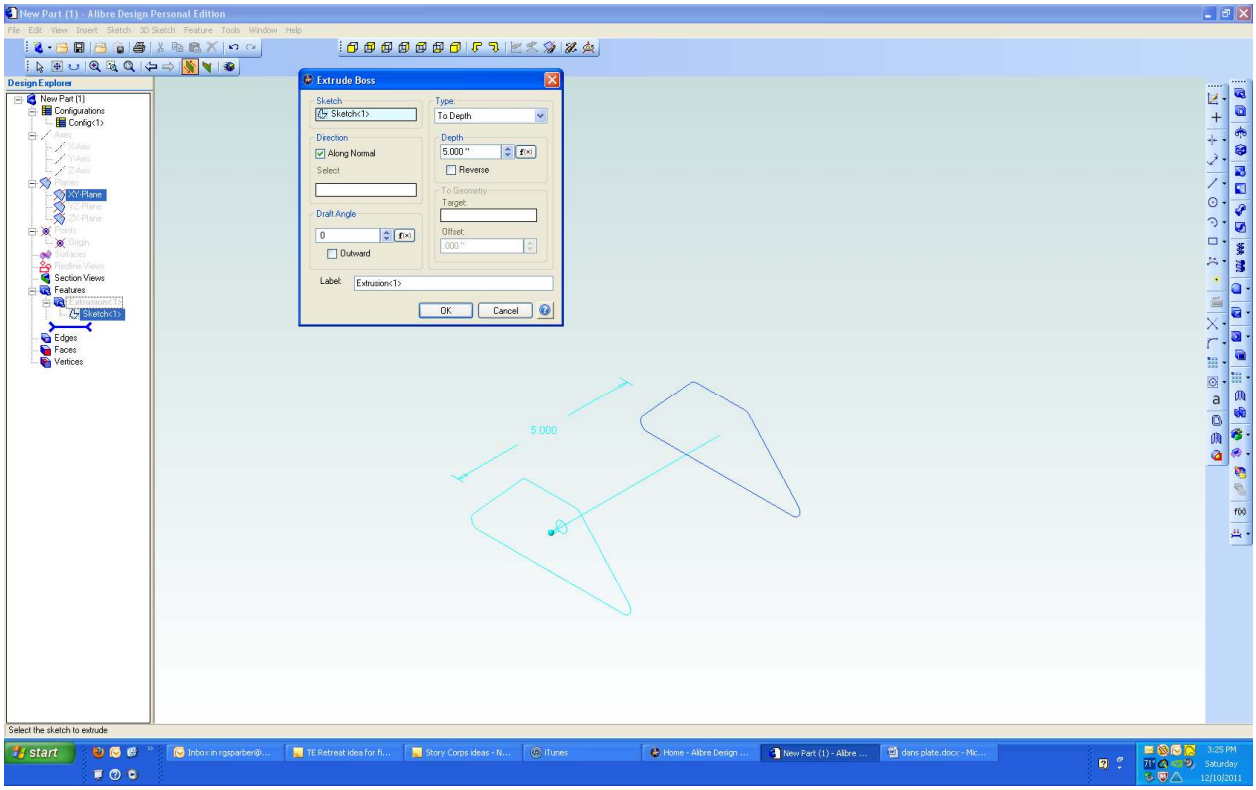


Next I put a radius on the upper left corner. For now I set it at 0.200”.



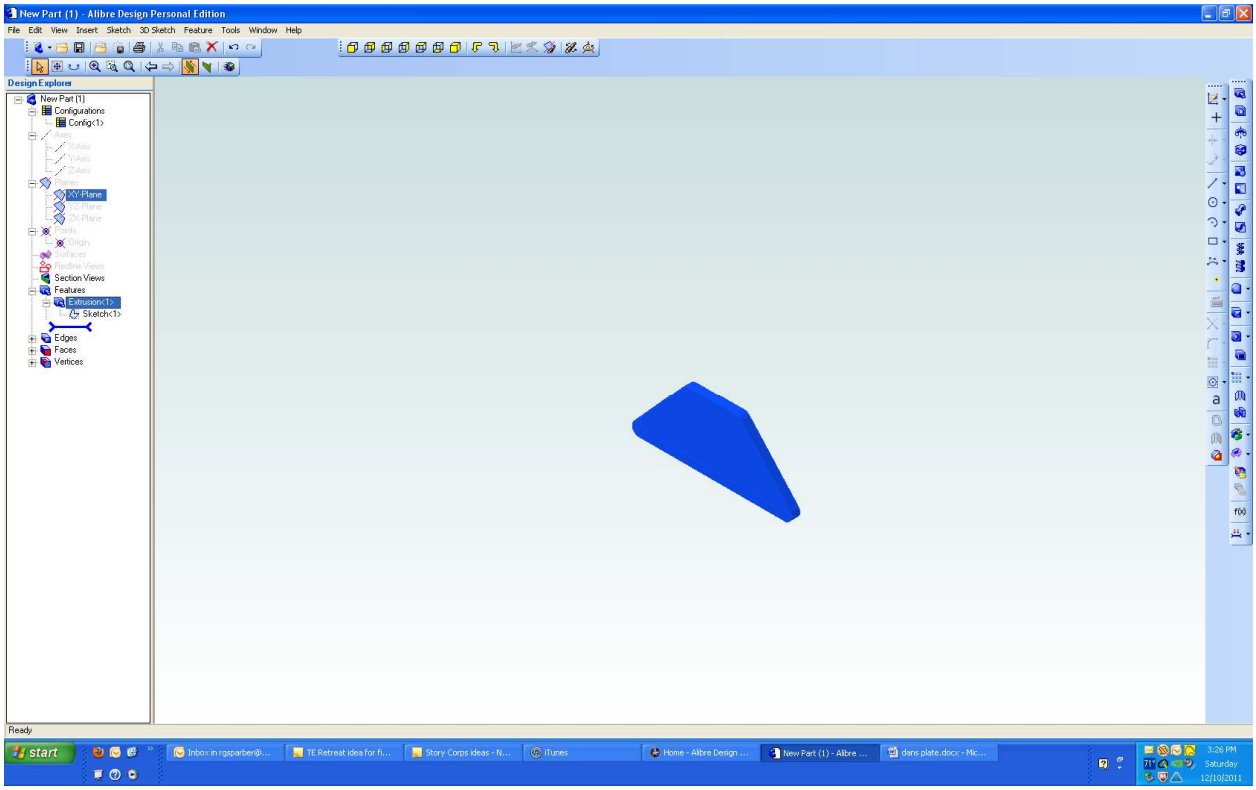
Without putting the radius tool away, I applied the same radius to the other three corners. Note that each one has an "=" sign next to it. This means that changing the one that is dimensioned, I will change all of the corners by the same amount.

I am defining a 2D figure that will be stored in a file called Sketch(1). Look down the Design Explorer column to see it.

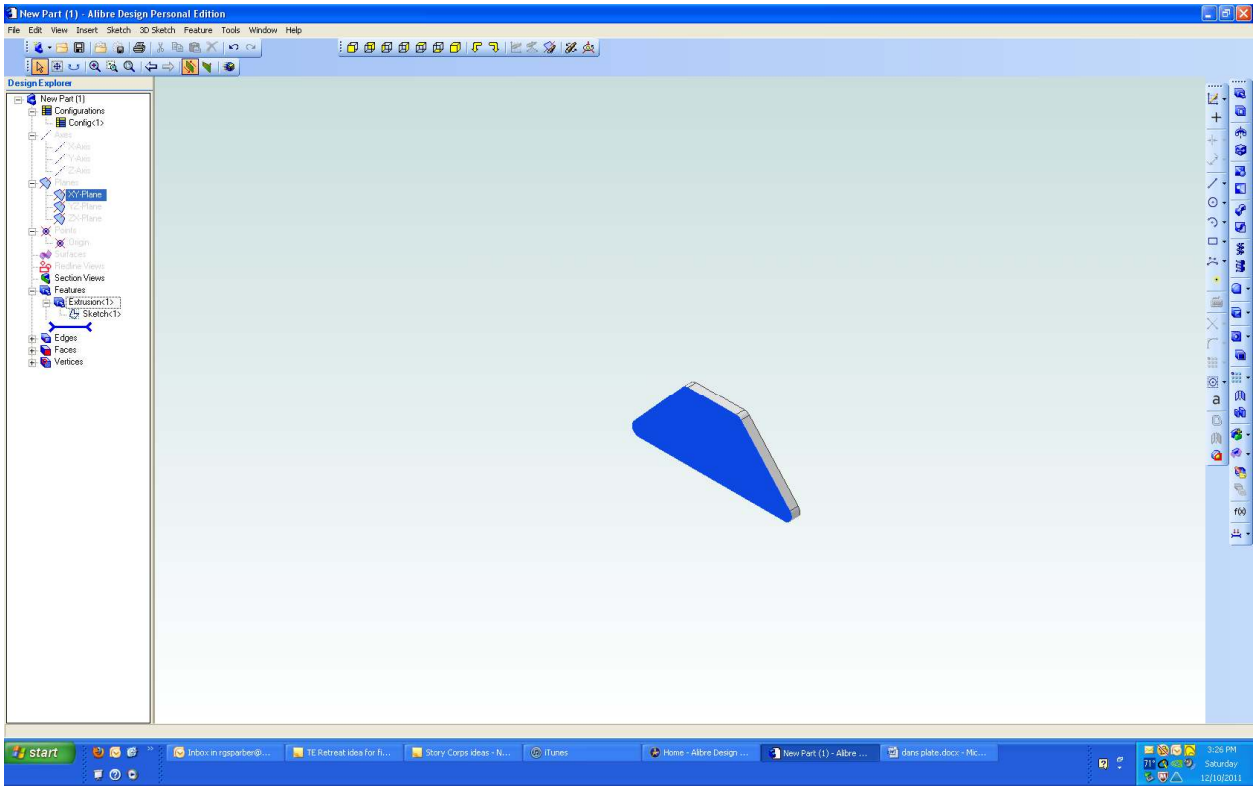


I next selected the extrusion tool which defaults to 5.000".

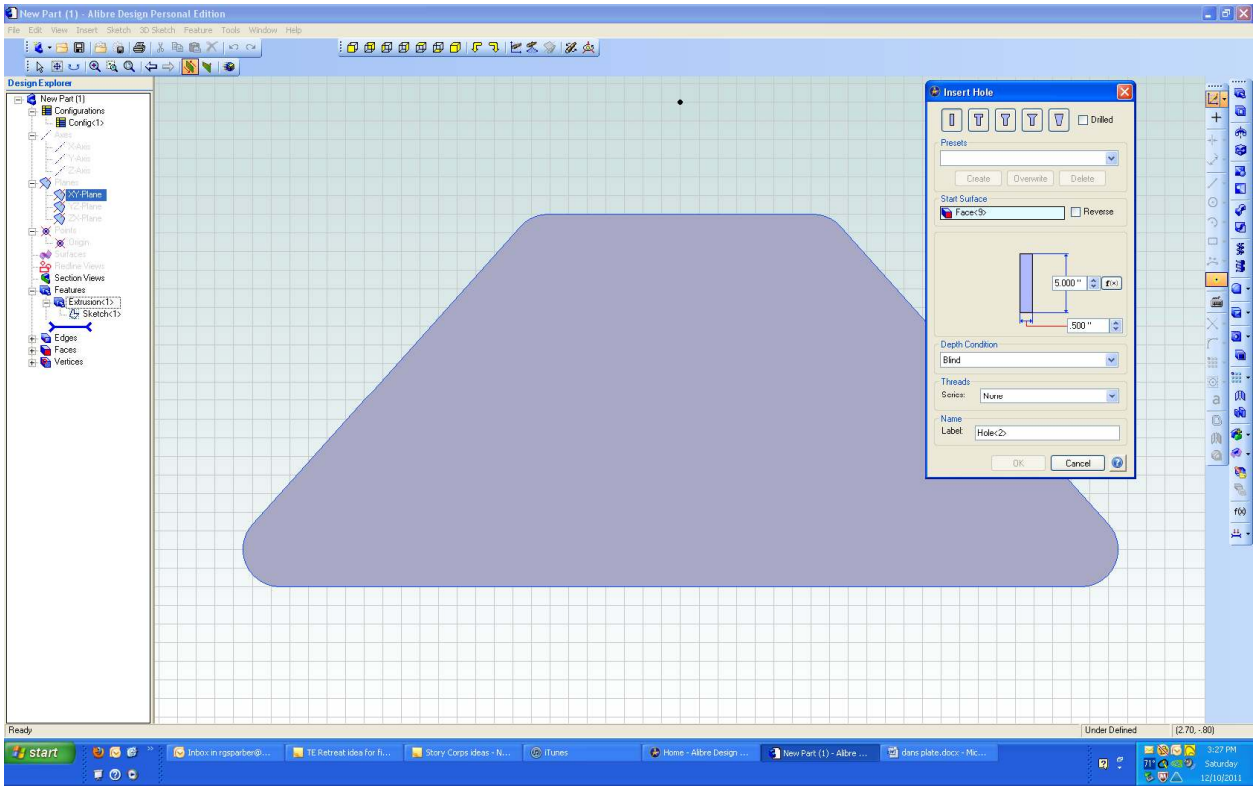
This action creates a 3D figure that is dependent on my 2D drawing. The 2D drawing is stored in the Sketch(1) entry in the Design Explorer. Above Sketch(1) is the dependent 3D entry Extrusion(1).



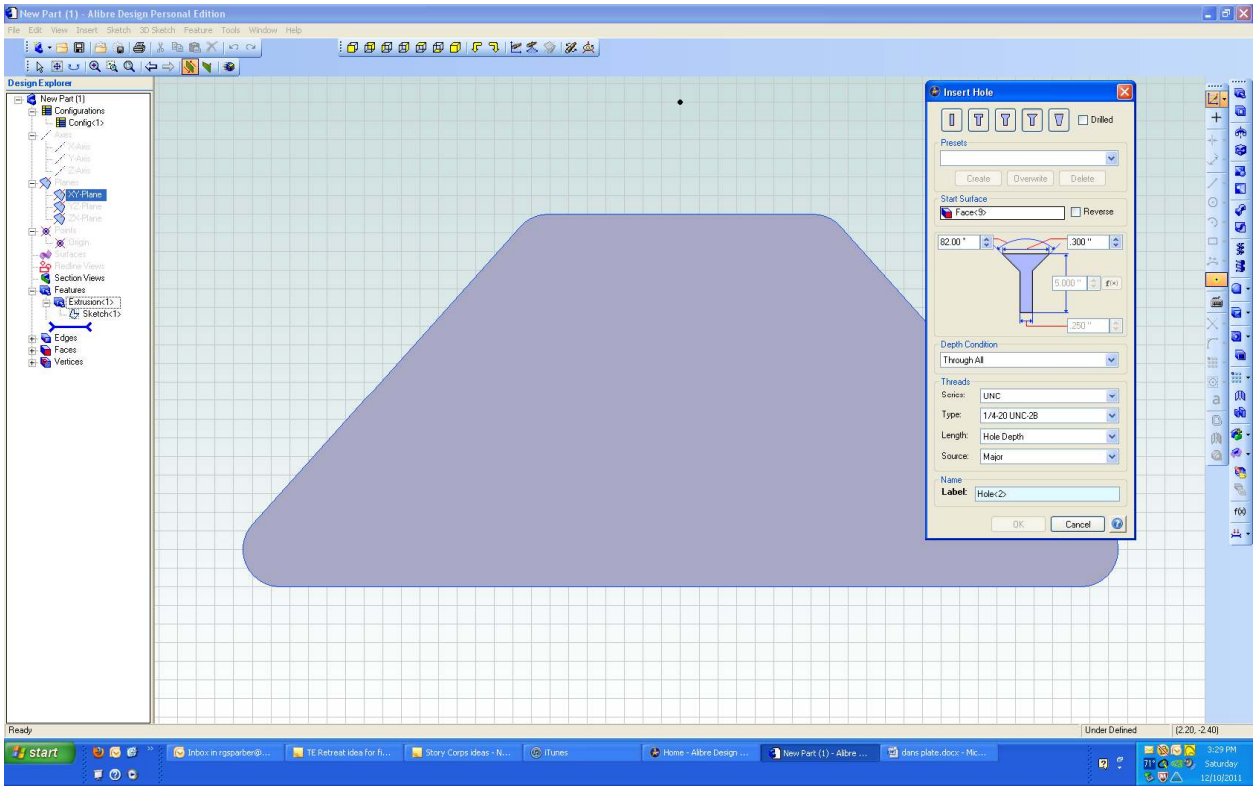
After entering 0.25”, I closed the dialog box and now have my ¼” thick plate.



My next step was to select the face of the part that will receive my holes. Note that this face is blue while the rest of the part's faces are gray.

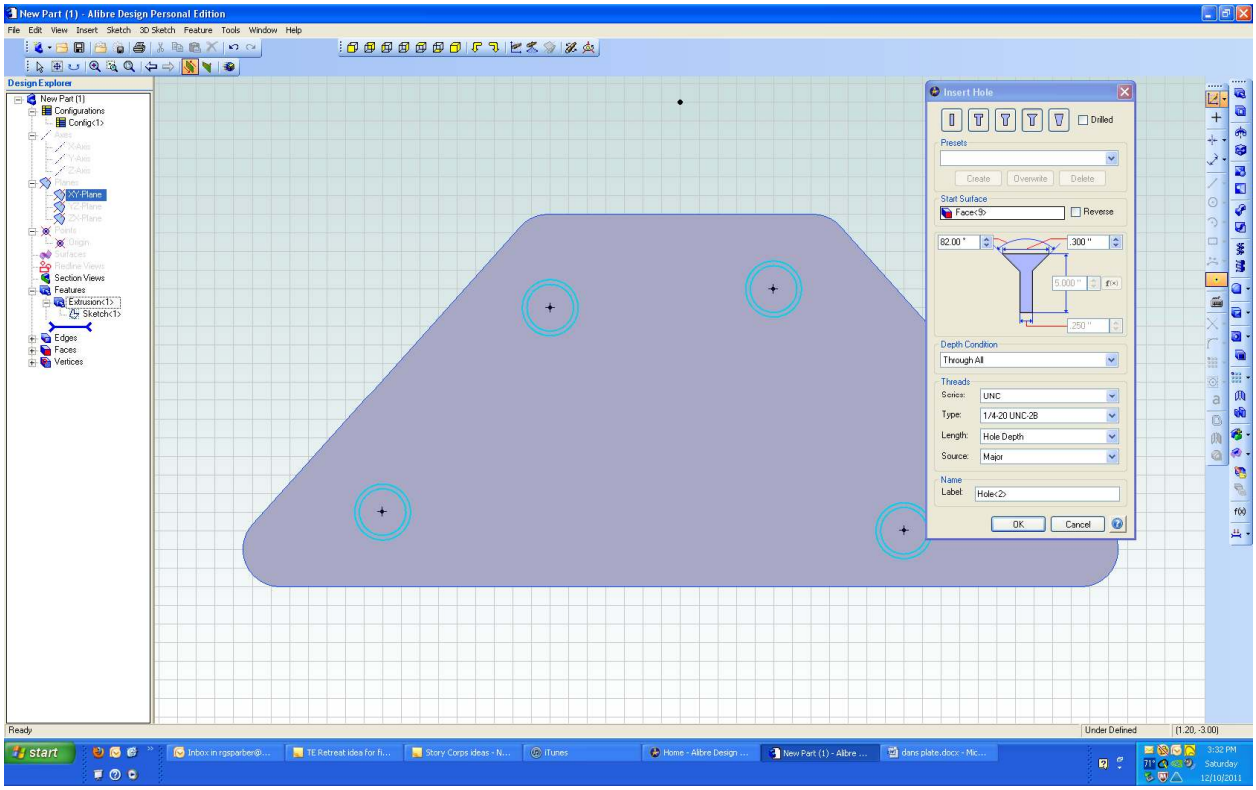


When I selected the 3D hole tool, the part face I will be modifying now faces squarely towards me and I can select what kind of hole to drill.

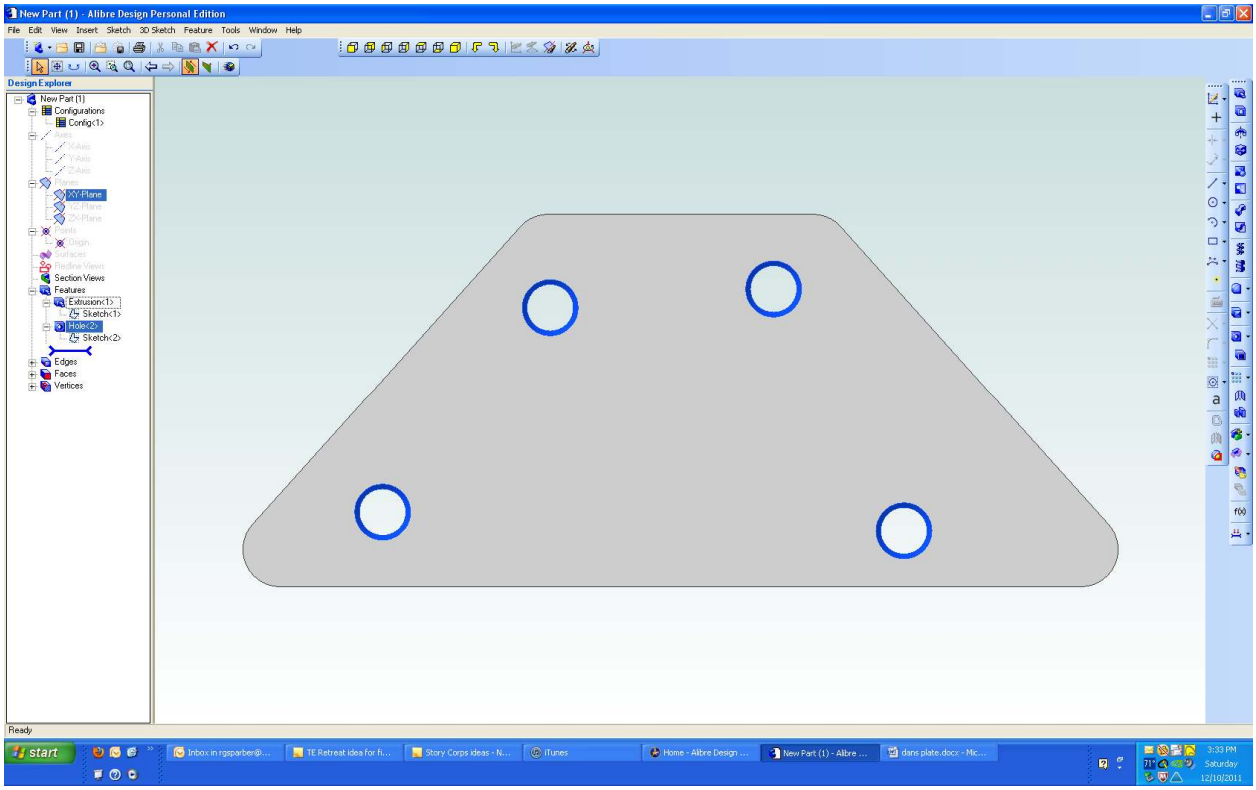


I changed the top of the countersunk hole to be 0.300” across. My depth condition is “through all”, my thread is UNC ¼-20, and my hole is major diameter. I really wanted a clearance hole here but don’t see that as an option.

Note that the OK button is gray. That is because I have not placed any holes yet.



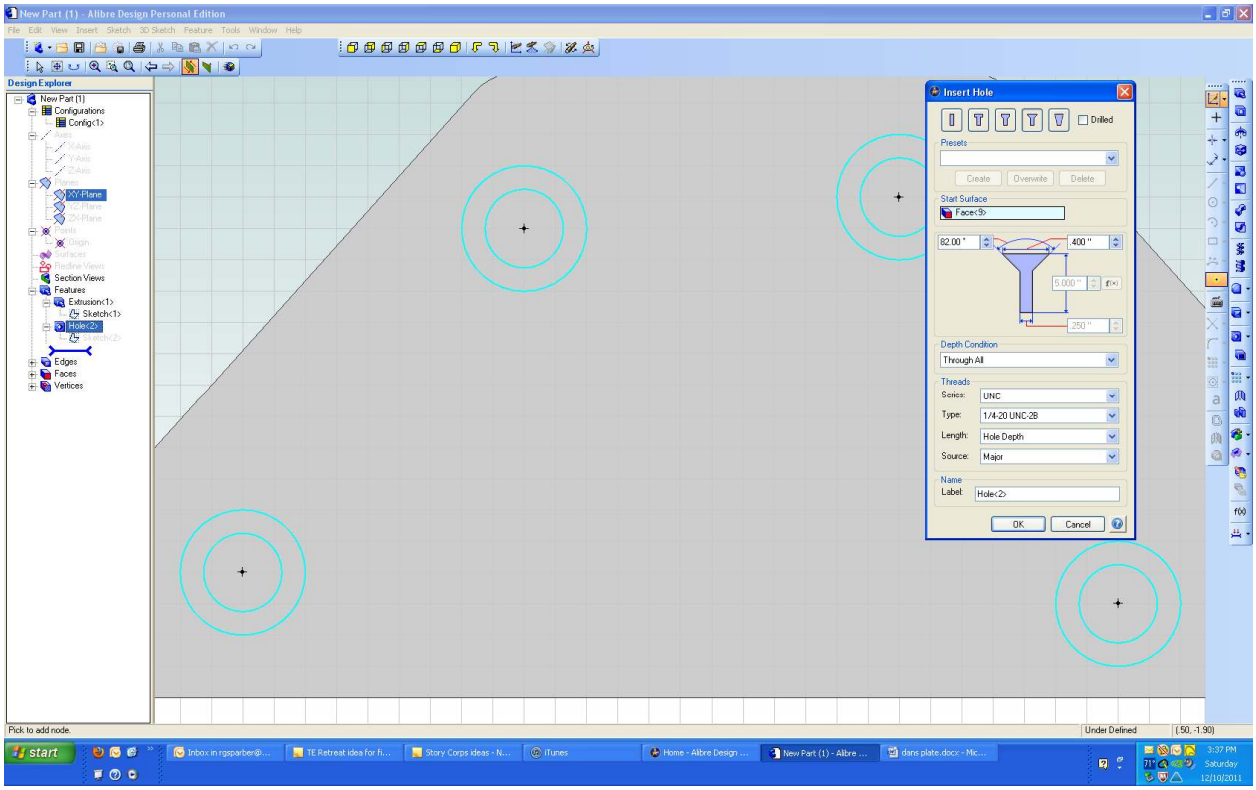
After placing my four holes, the OK button is now available to be clicked.



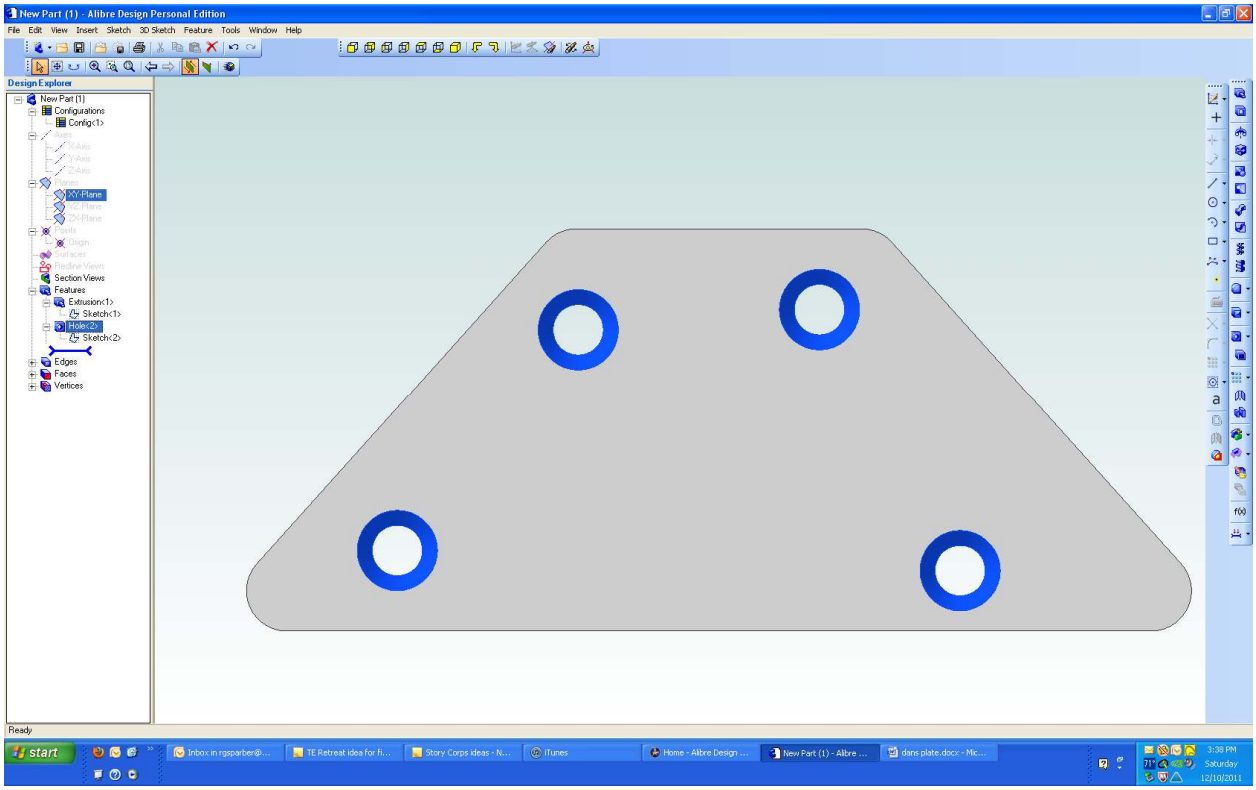
I now see my holes with countersinks in place.

In the Design Explorer I now have an entry marked Hole(2) and under it Sketch(2). I can change the shape of these four holes by moving my cursor over the Hole(2) entry, right click, and select edit.

These holes don't look right so I decided to edit Hole(2).

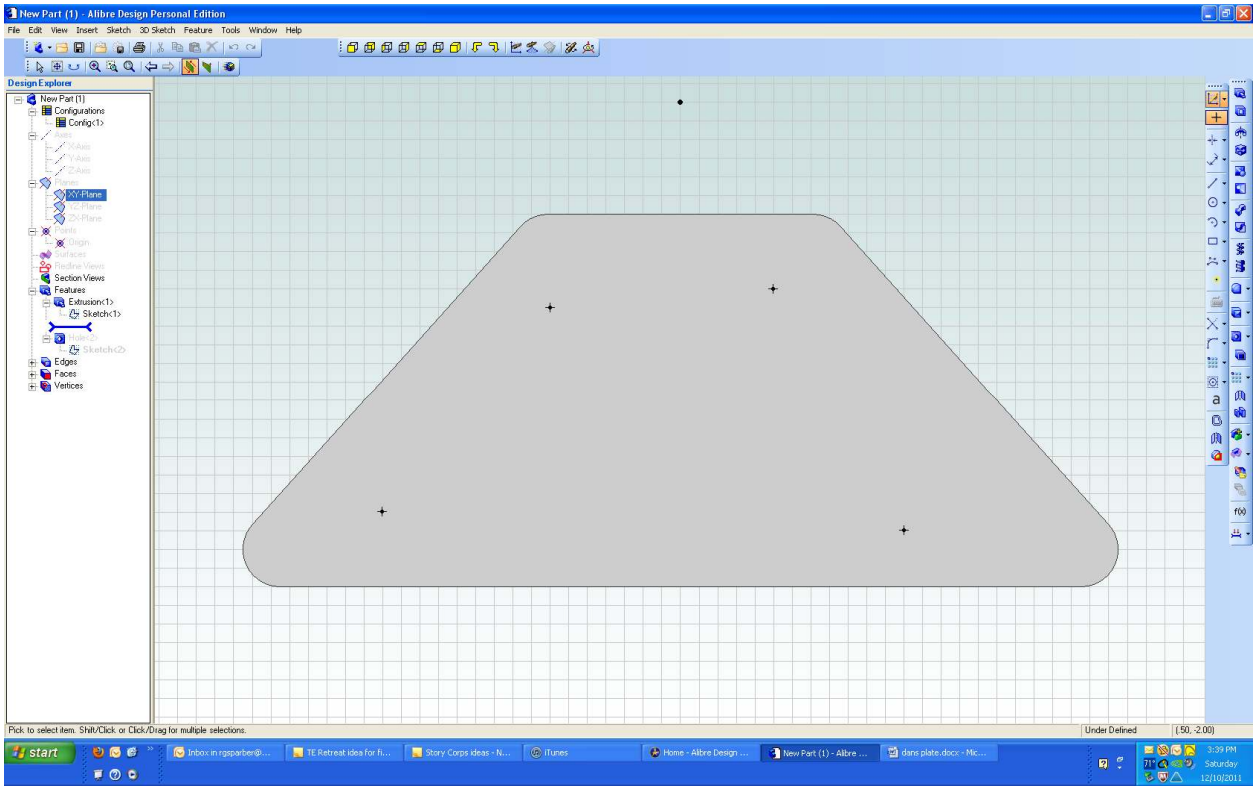


I changed the top of the countersink to 0.400”. Then I clicked OK to take me back to 3D.

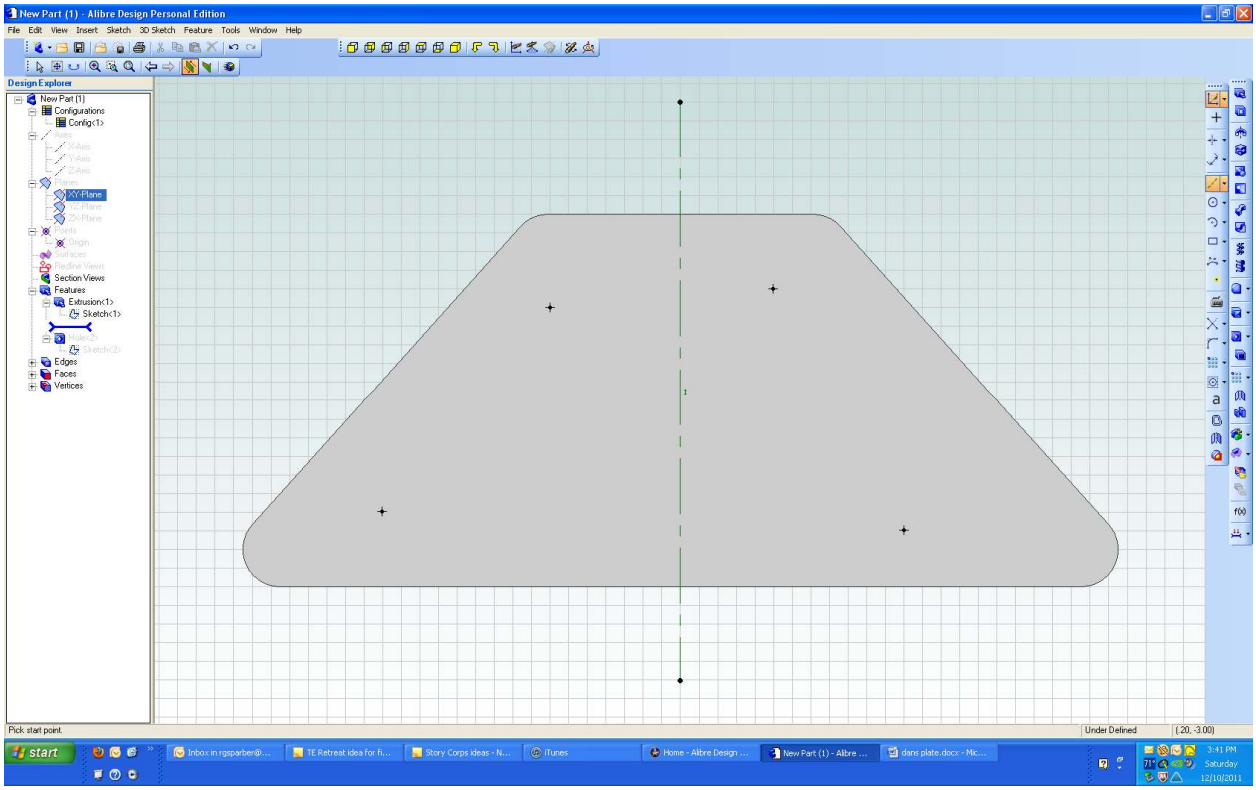


That looks better. Nothing is permanent so I can change these holes at any time, even when I get to the shop drawing.

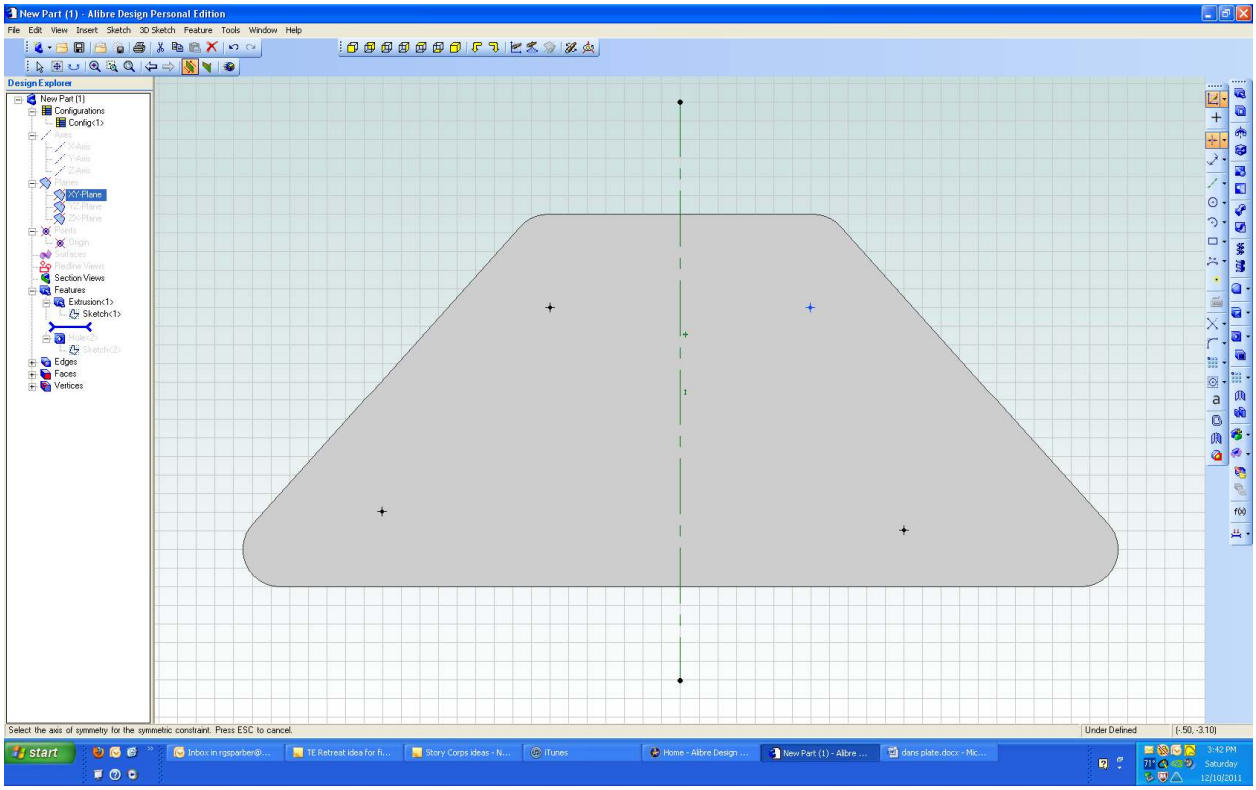
OK, the holes are about the right size but certainly not in the right place. To fix that, I move the cursor over the Sketch(2) file name, right click, and select edit.



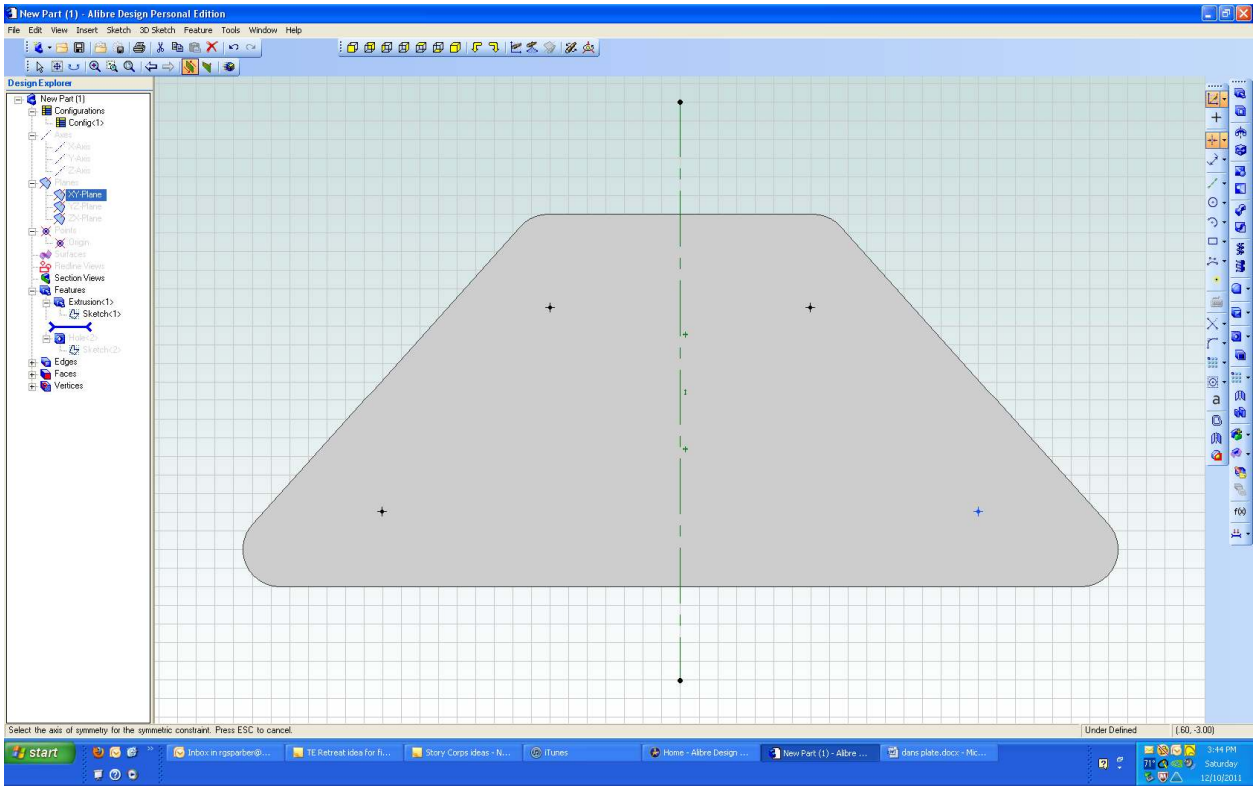
I now get the outline of the part plus 5 nodes. The top node which is off the part is my origin. The four nodes on the part are the centers of all holes.



The first thing I want to do is draw a reference line from the origin and down across the part. This will enable me to place my holes in a symmetric pattern.



As with the outline of the part, I again used my symmetric constraint tool to force the top two hole centers to be mirror images. I click on the LOS and then the two top centers. Since I clicked the left center first, the right center moved to be symmetric with it after I clicked it.

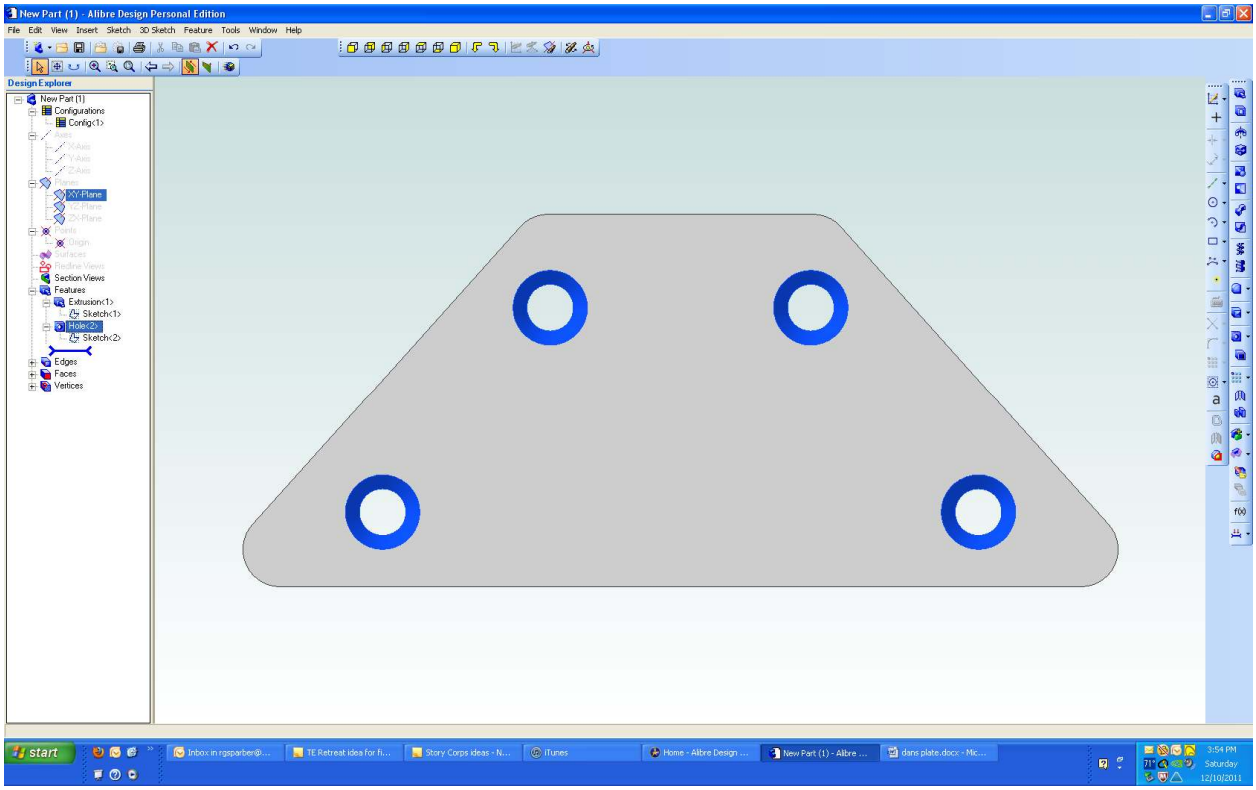


Repeating the process, my bottom two centers are now mirror images.

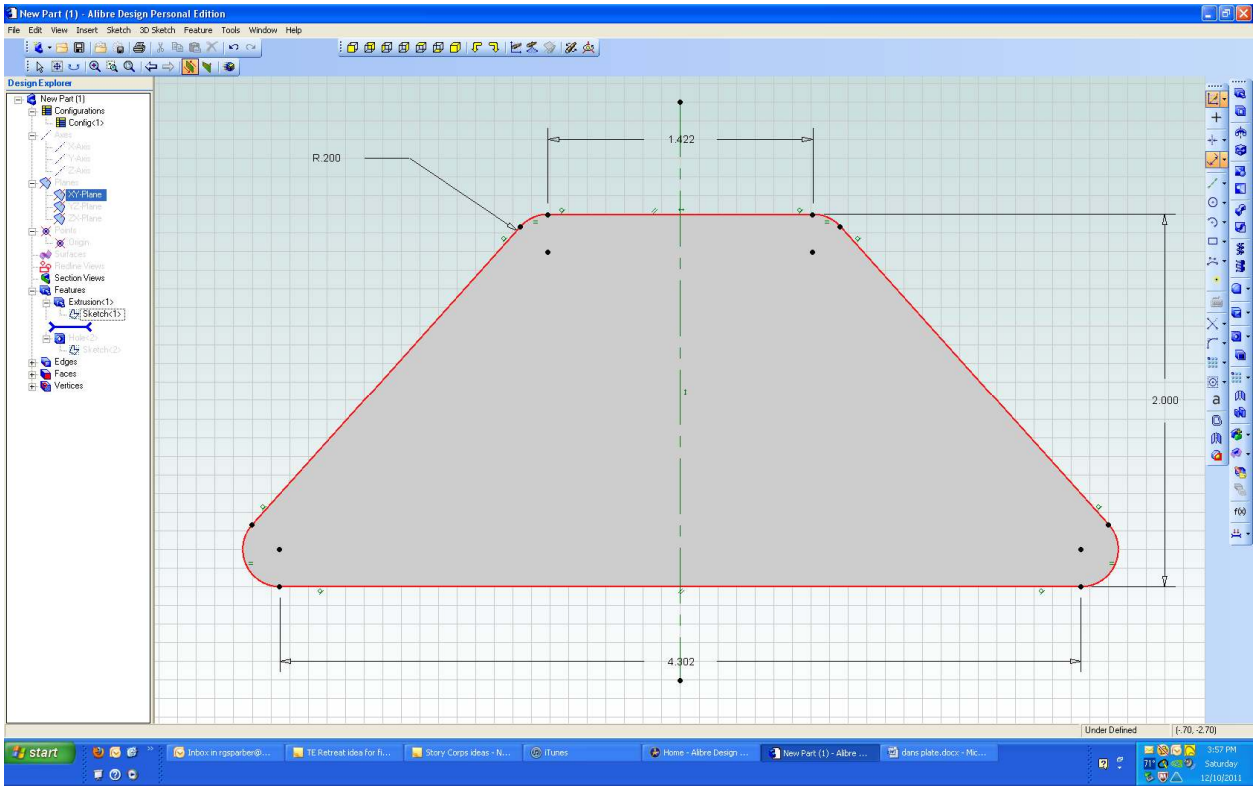
Note that I am working on the 2D file called Sketch(2) which is tied to my 3D file Hole(2). I could modify the outline of the part from Sketch(2) but this would turn into a mess. Sketch(1) defines the outline of the part and that is where I should go to change it.

Recall that I first drew a 2D outline which became Sketch(1) and then extruded it to become Extrusion(1). In the design explorer Sketch(1) appeared first and later Extrusion(1) appeared above it.

When creating the holes, I first drew the holes in 3D and created the entry Hole(2). It, in turn, generated the 2D entry Sketch(2). So the order of creation is opposite for these two cases. No matter, you still edit the entry you wish to change.

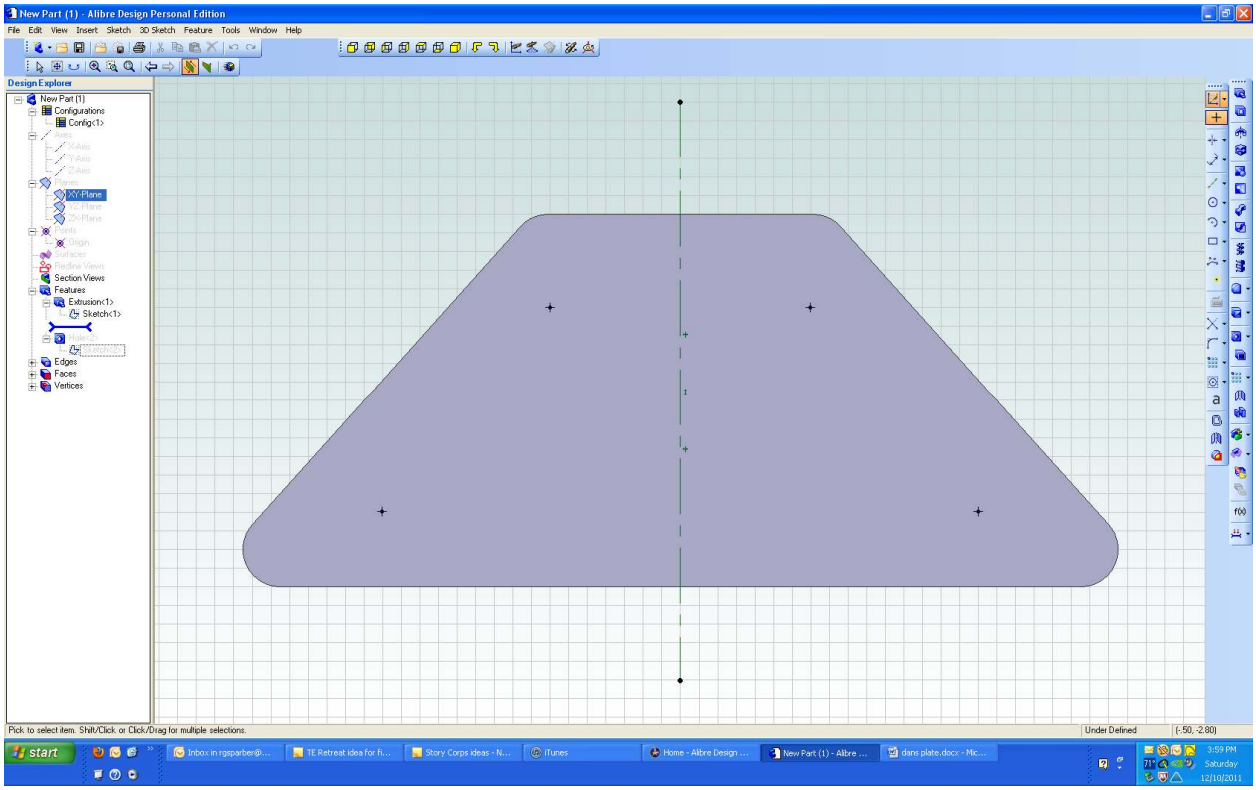


Exiting the 2D edit mode, I now have my 3D plate with four countersunk holes in it. I liked the look of it but exact dimensions have not been applied so this is not yet usable.

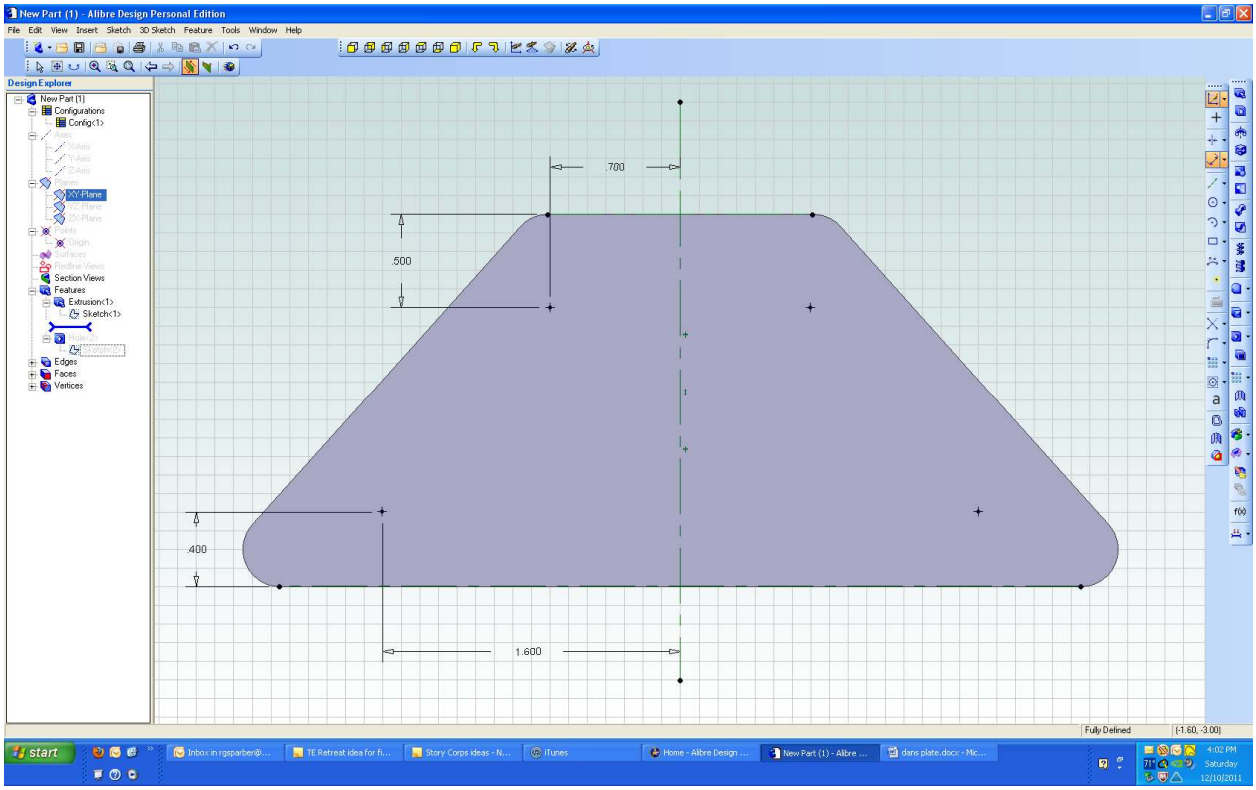


I went back to my Design Explorer and right clicked on Sketch(1). Then I selected edit which lets me 2D edit the outline of the part. I then used my dimension tool to define three distances. No effort was made to change the values at this time.

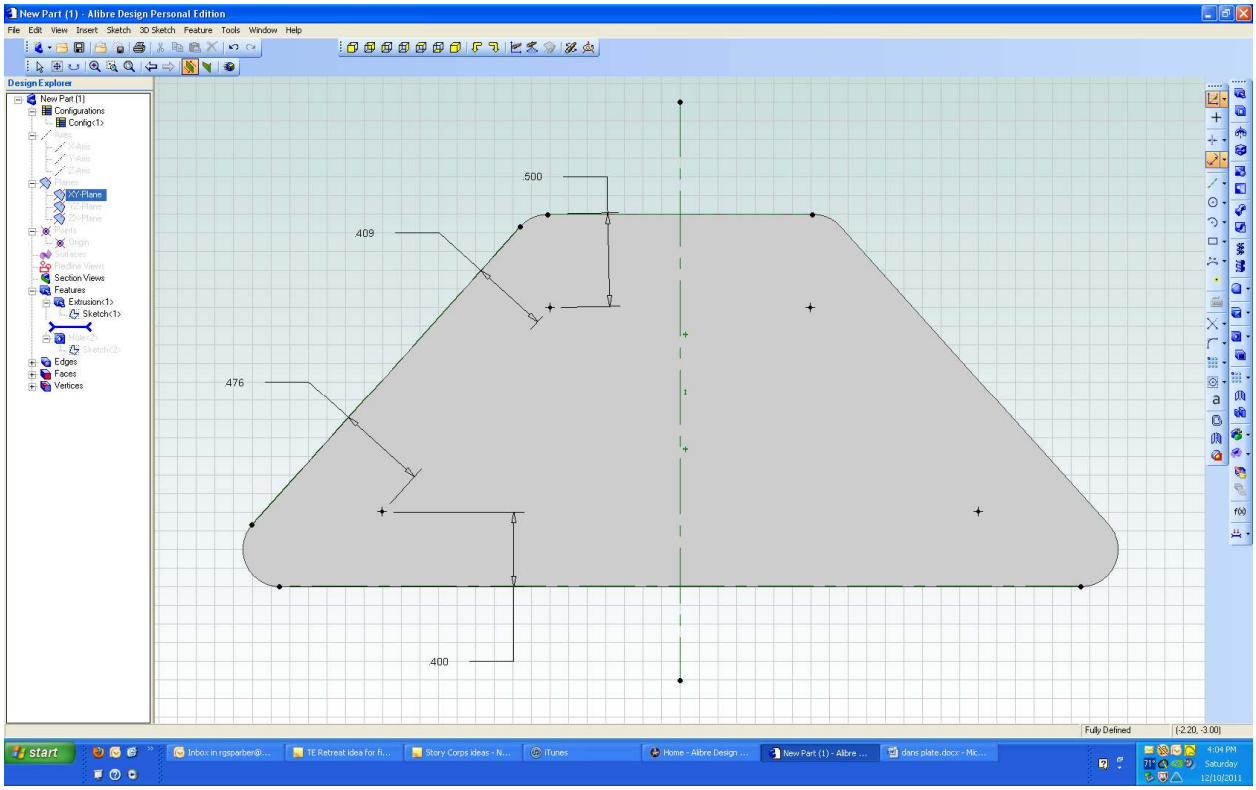
I then left the 2D edit mode by clicking on the edit icon near the upper right corner of the edit window.



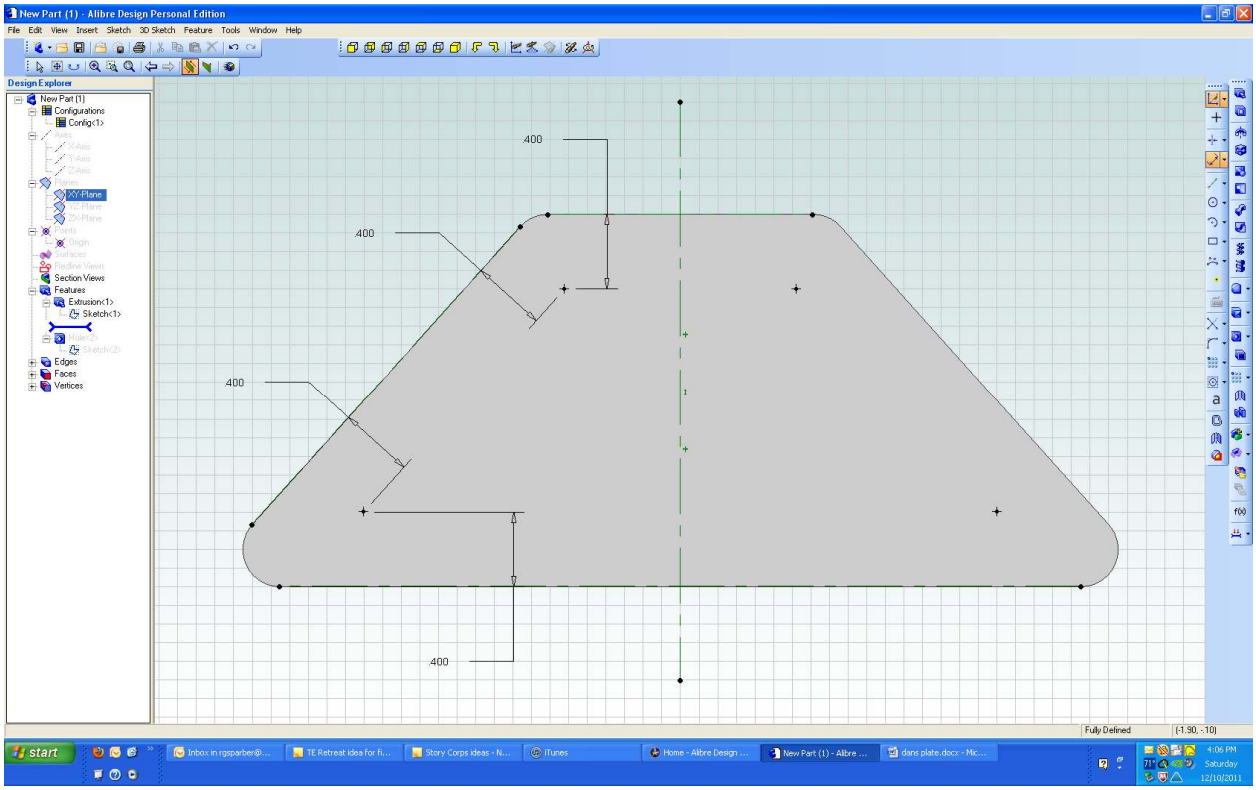
Next I moved my cursor to Sketch(2), right clicked, and selected edit. I am now in the 2D edit mode looking at my hole centers.



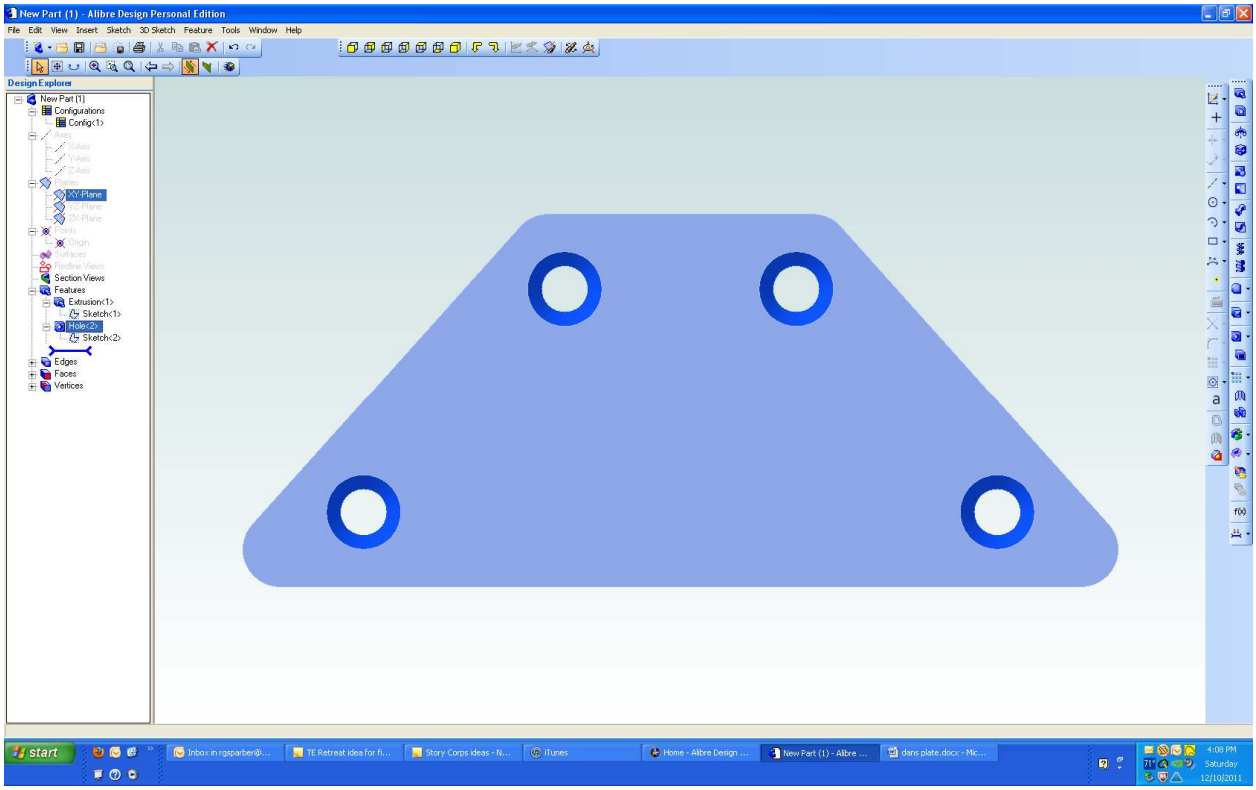
Using my dimension tool, I have completely defined my hole locations with respect to my LOS.



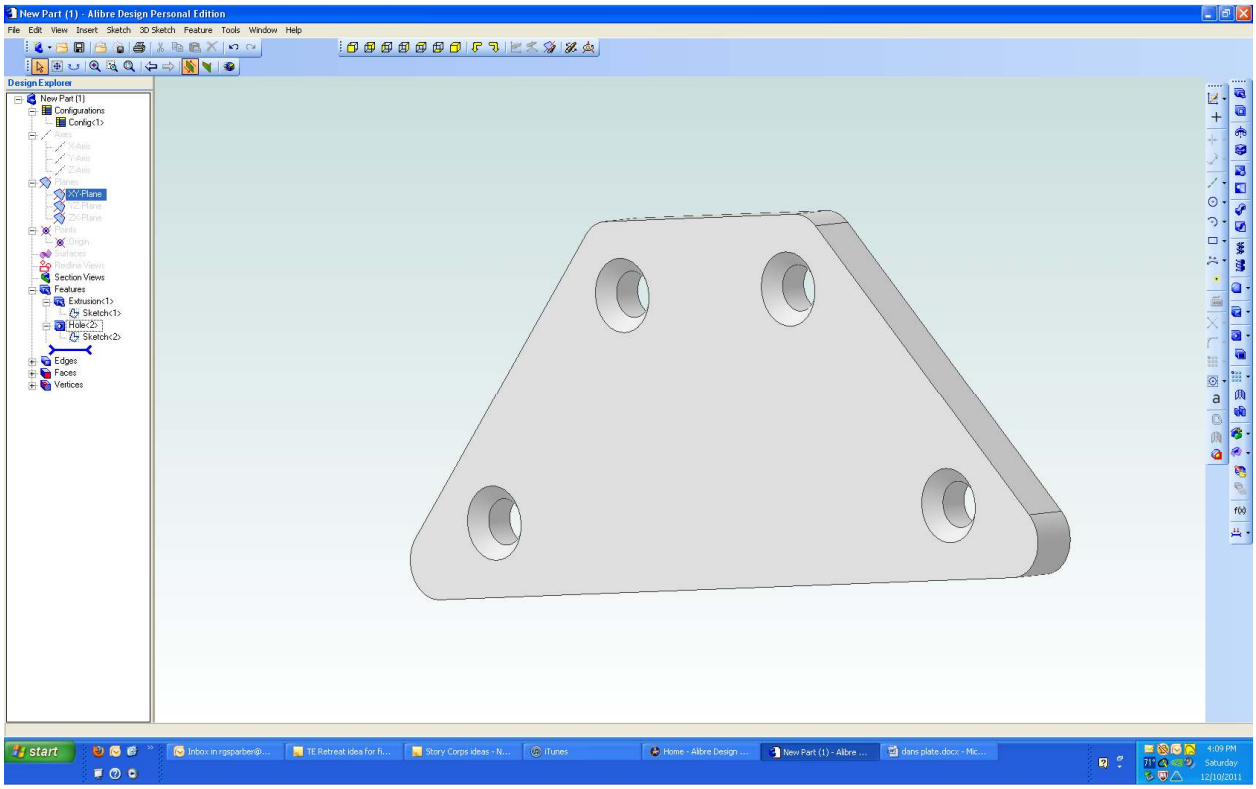
I changed my mind. I want the centers to be with respect to the edges of the part. No problem. Just delete the old dimensions and add new ones.

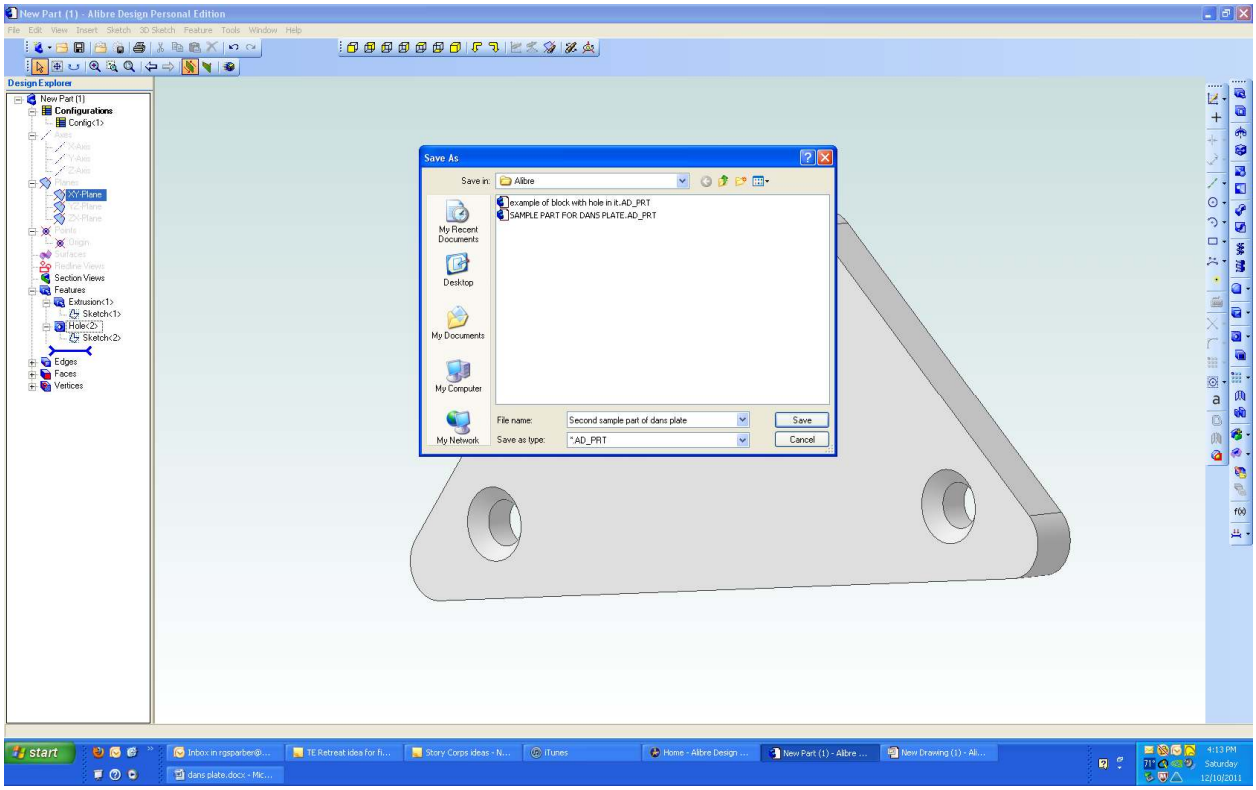


I changed all of the distances to 0.400". I could have changed one of them and used the function option in the dimension tool to make the rest equal to this value. But I will leave that discussion for another day.

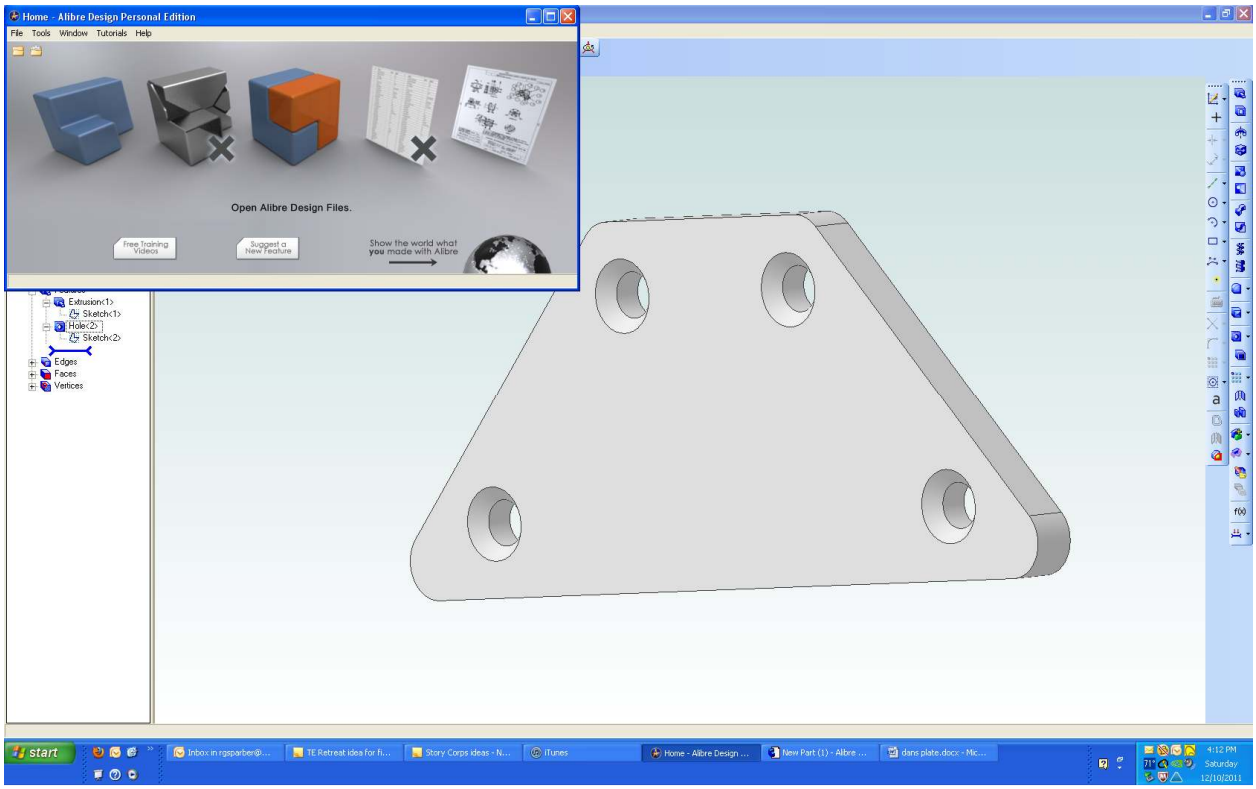


Leaving the 2D editor, I then had a decent looking plate with four countersunk holes.

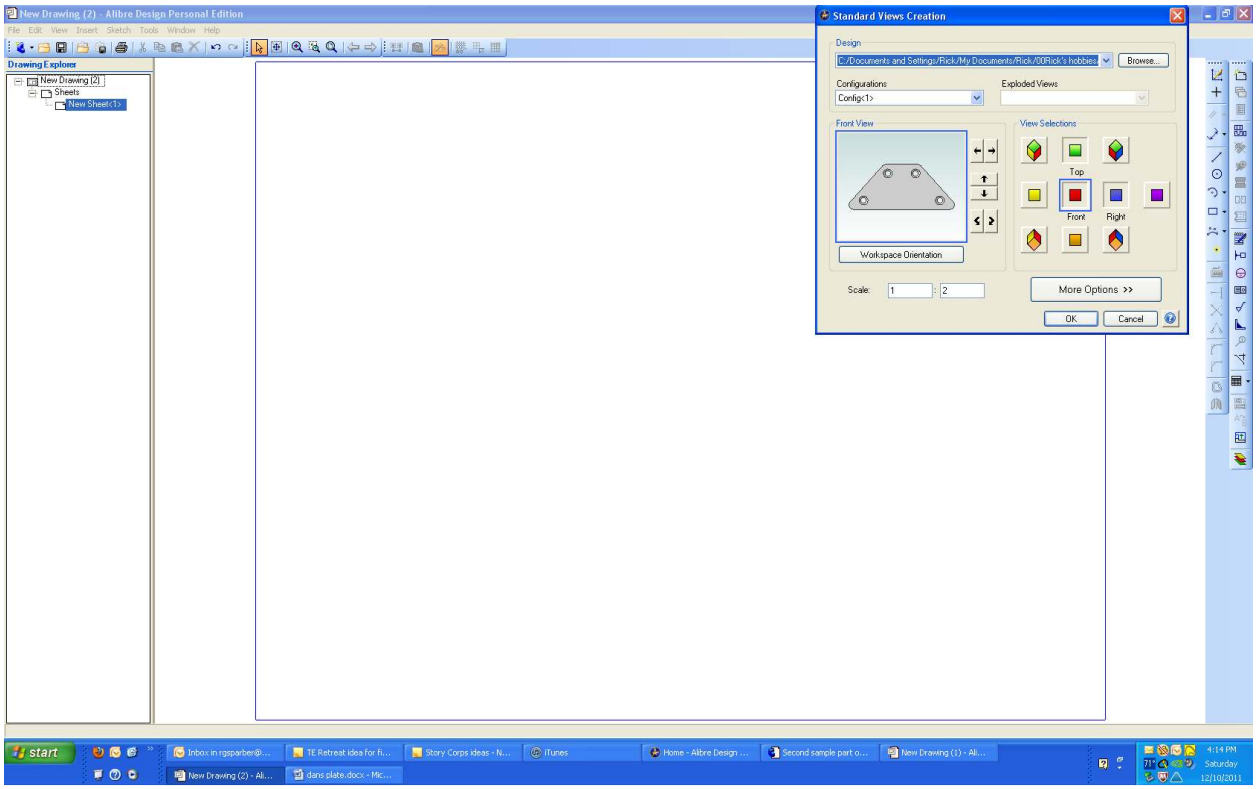




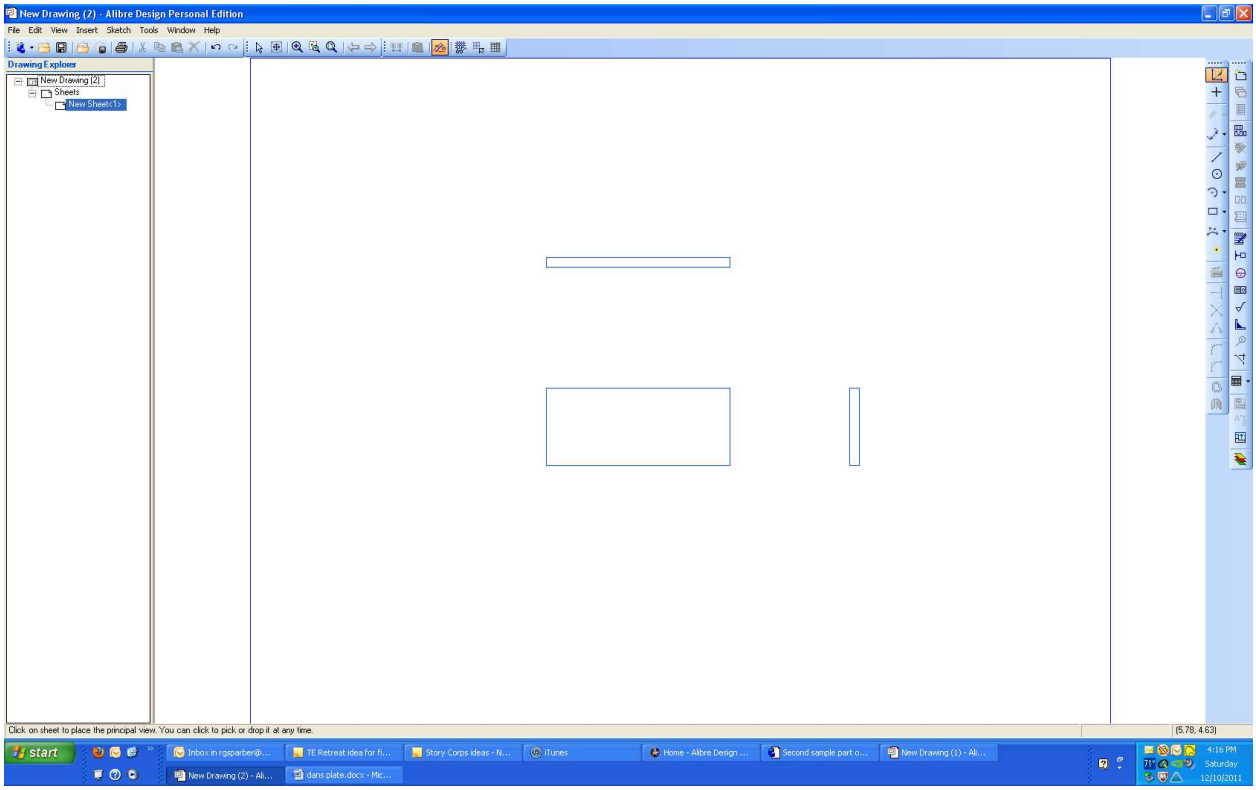
Before I can quit this part of the design effort, I must save my work.



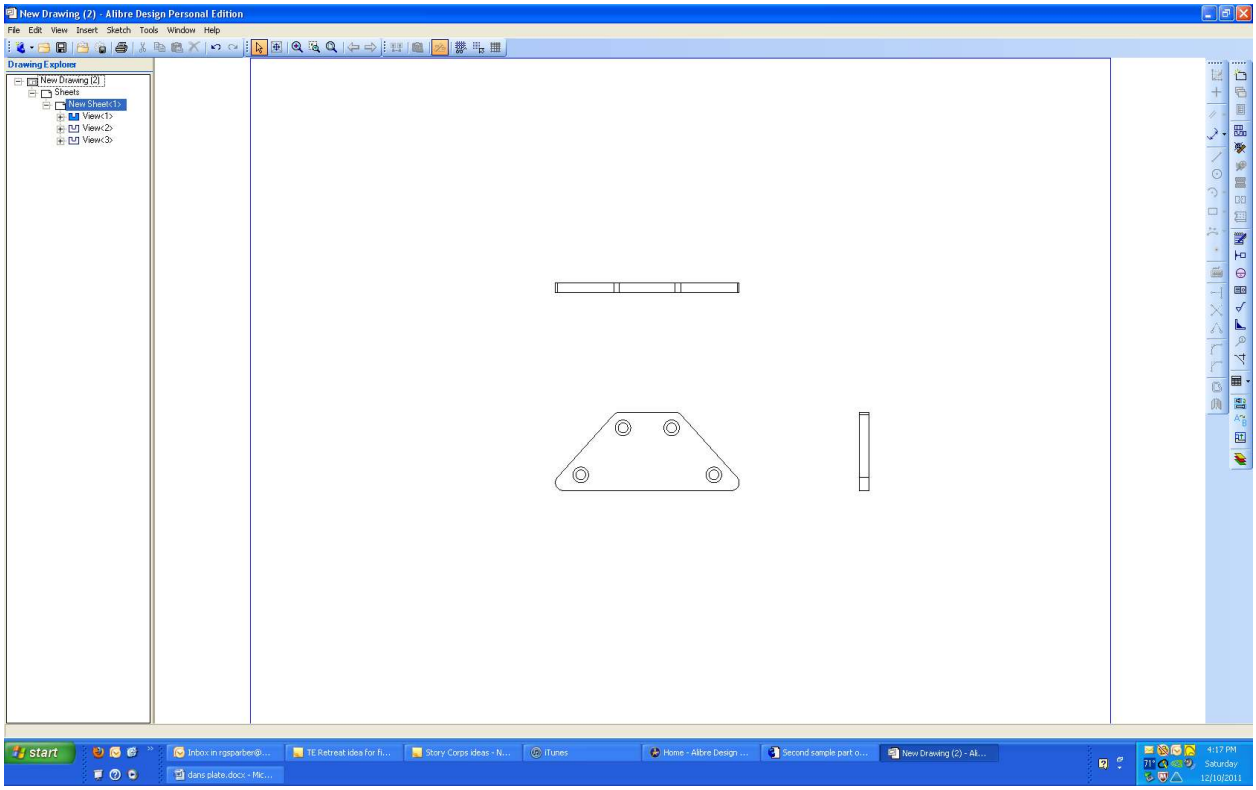
The final step is to generate a shop drawing. I went back to my Home screen and clicked on the far left icon.



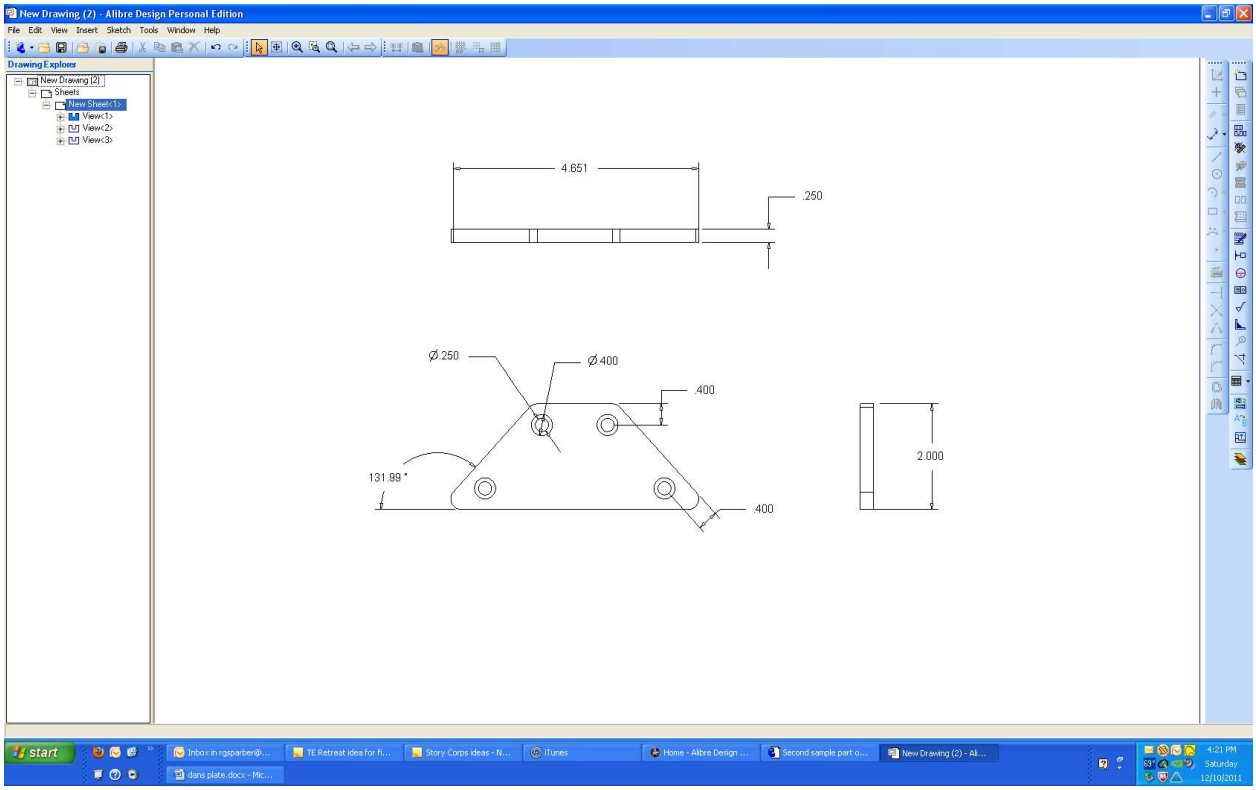
The program assumes that I want to generate a shop drawing for the last part I touched. I will skip the huge number of options here and just accept all defaults.



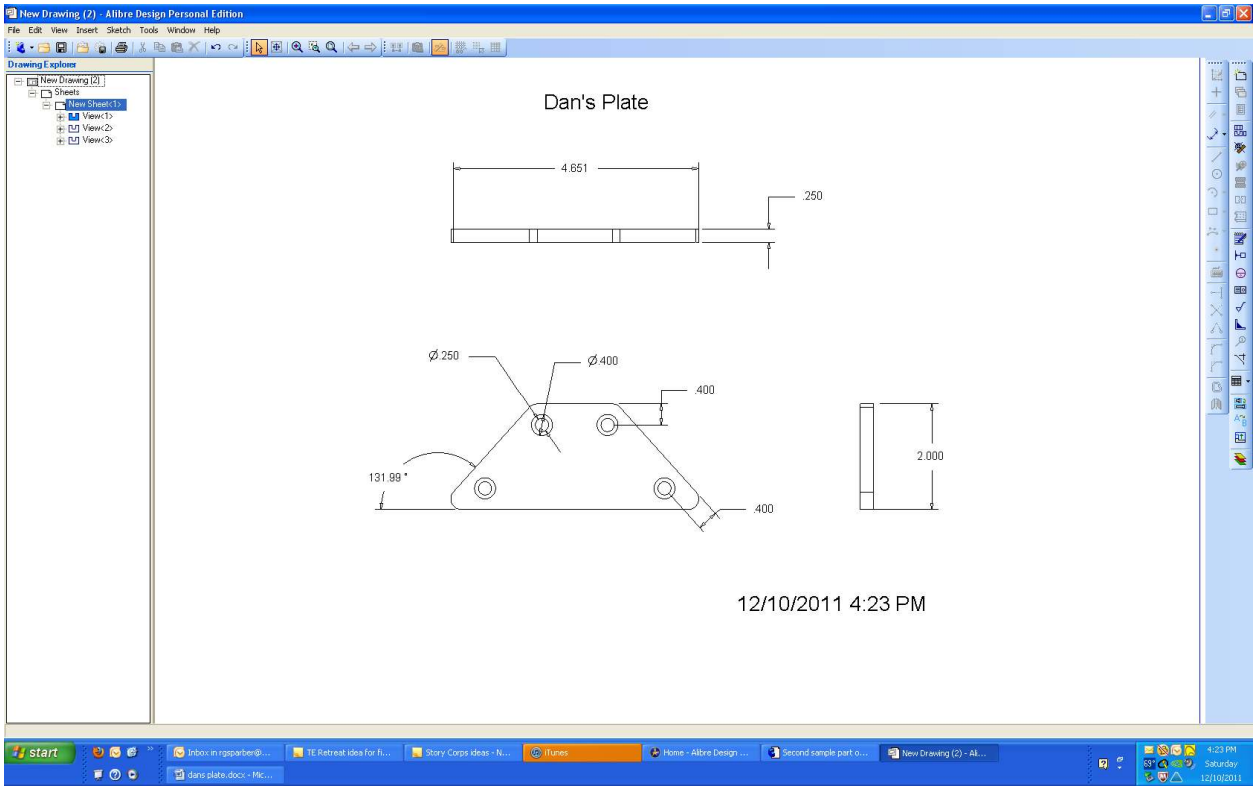
When I click OK, I am shown these 3 rectangles which move around as my cursor moves. Left click locks them down.



There are the default views of the part.



Using the dimension tool, I marked the distances and angle that I thought would be helpful in the shop. If I change any of these values, it will modify the part that I drew. That too is an advanced topic better left for another day.



My final touch was to go to the menu, select Insert and then annotation. Then select Note. The Note tool was then used to label the part and add the date and time.

The final step is to save the drawing. I can't believe it! Alibre crashed and I lost my shop drawing. That has never happened before… Crap. I don't see any auto-recovery file either.

Oh well, I hope you get the idea.

I welcome your comments and questions.

Rick Sparber
Rgsparber@aol.com
Rick.Sparber.org