Knurling with Alibre

By R. G. Sparber

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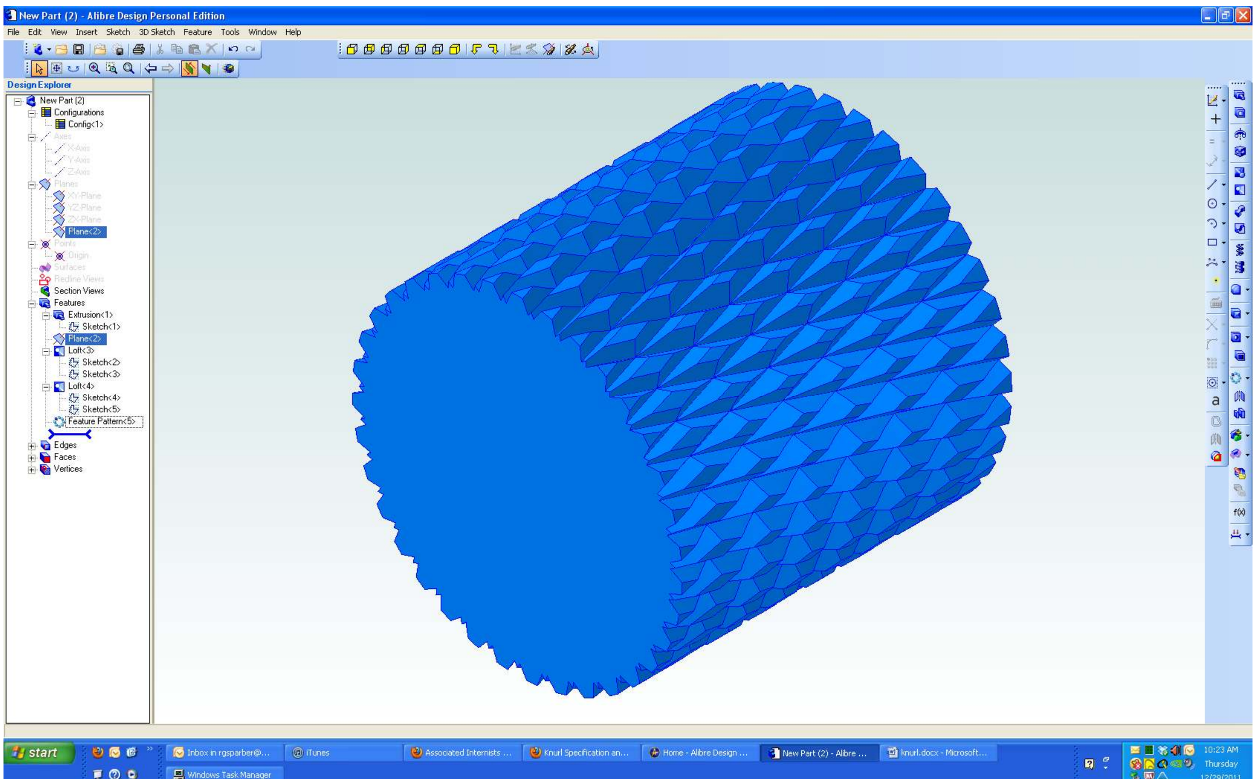

I read on the Alibre® BBS how someone drew the knurling pattern using a helical cut. I wondered if there was another way that would use straight lines. Here is the result.

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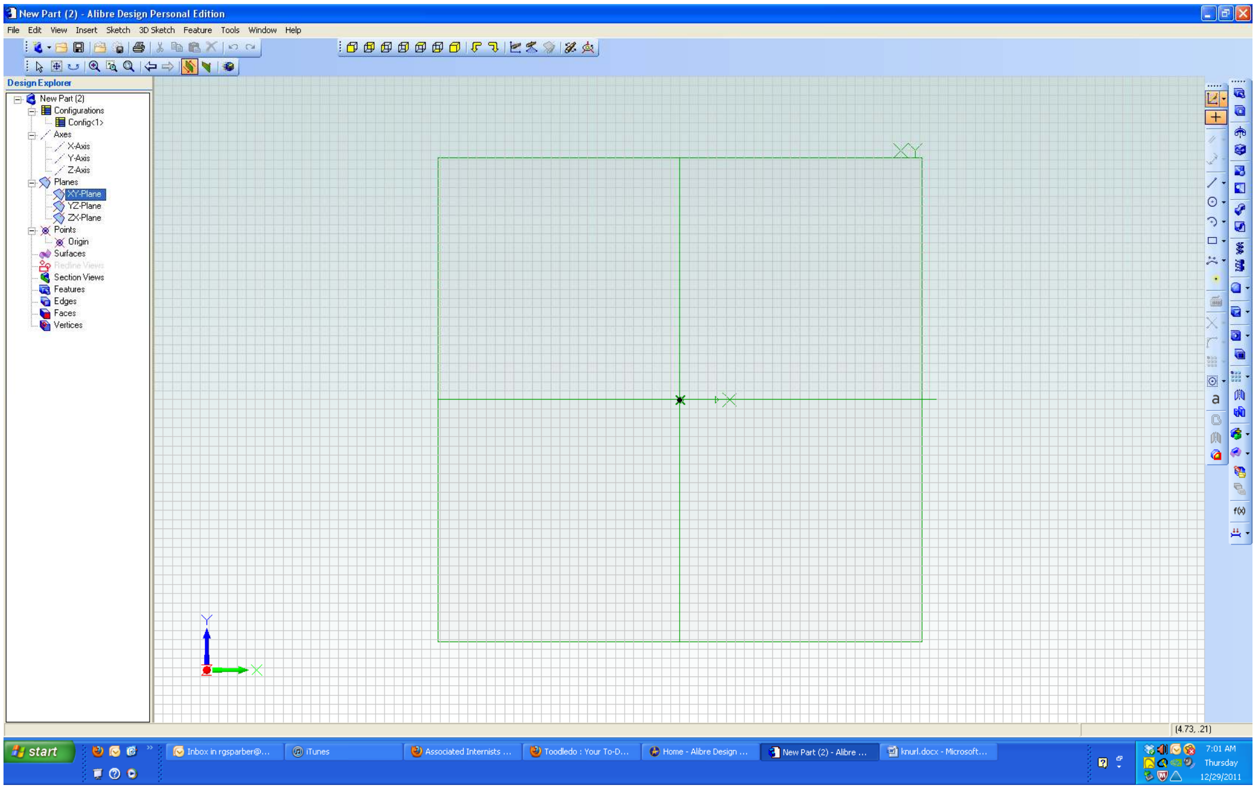
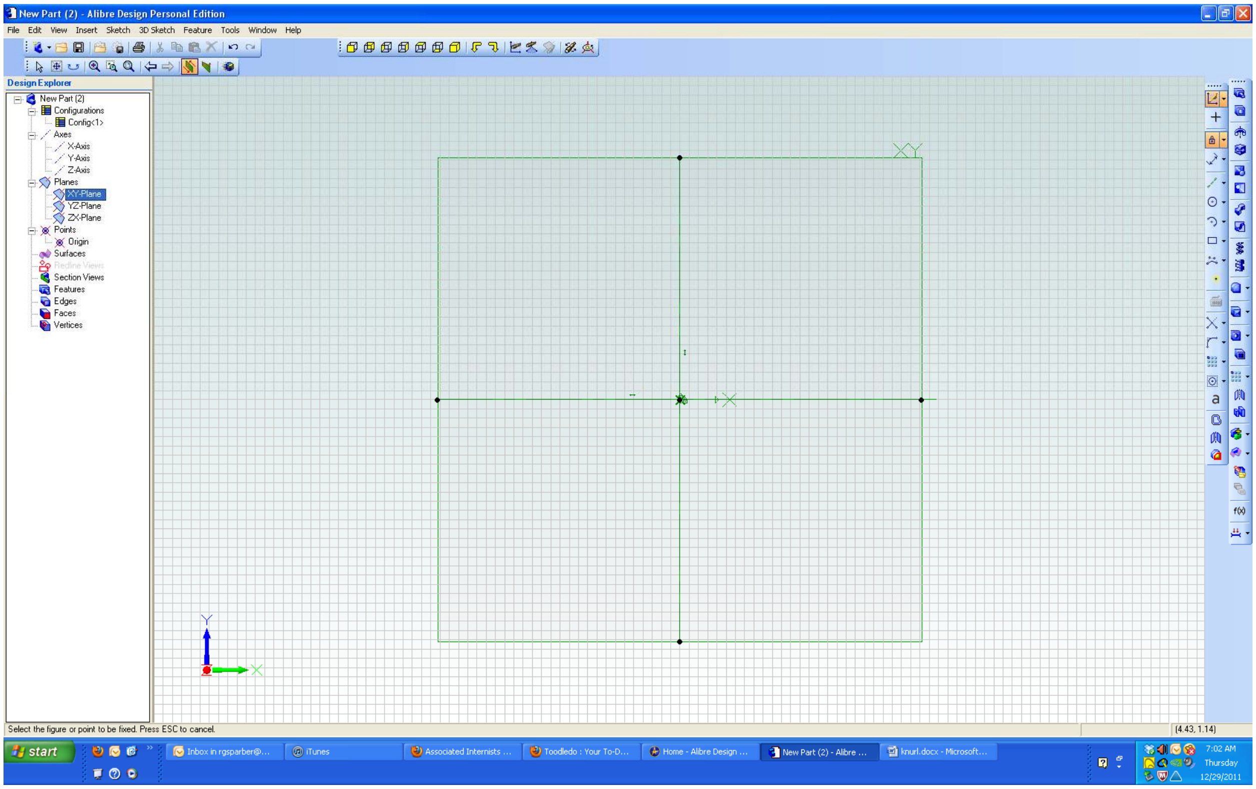
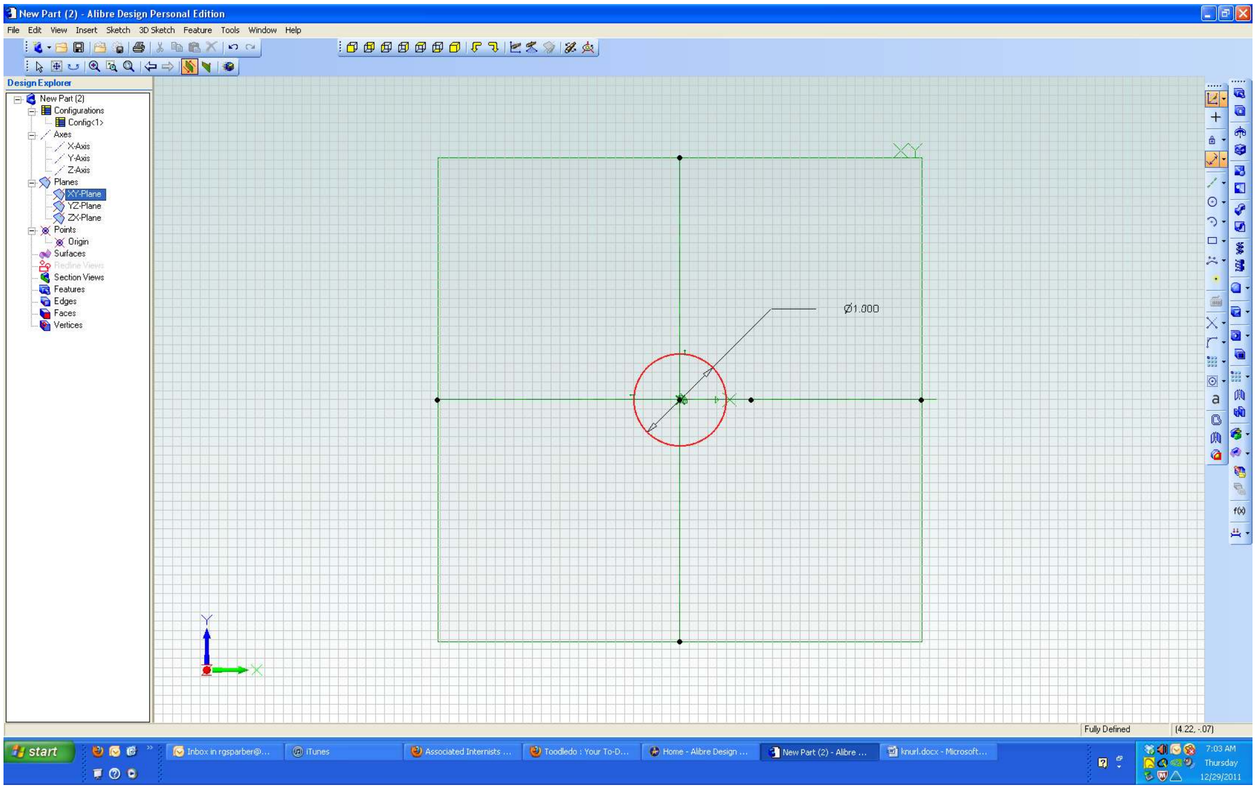


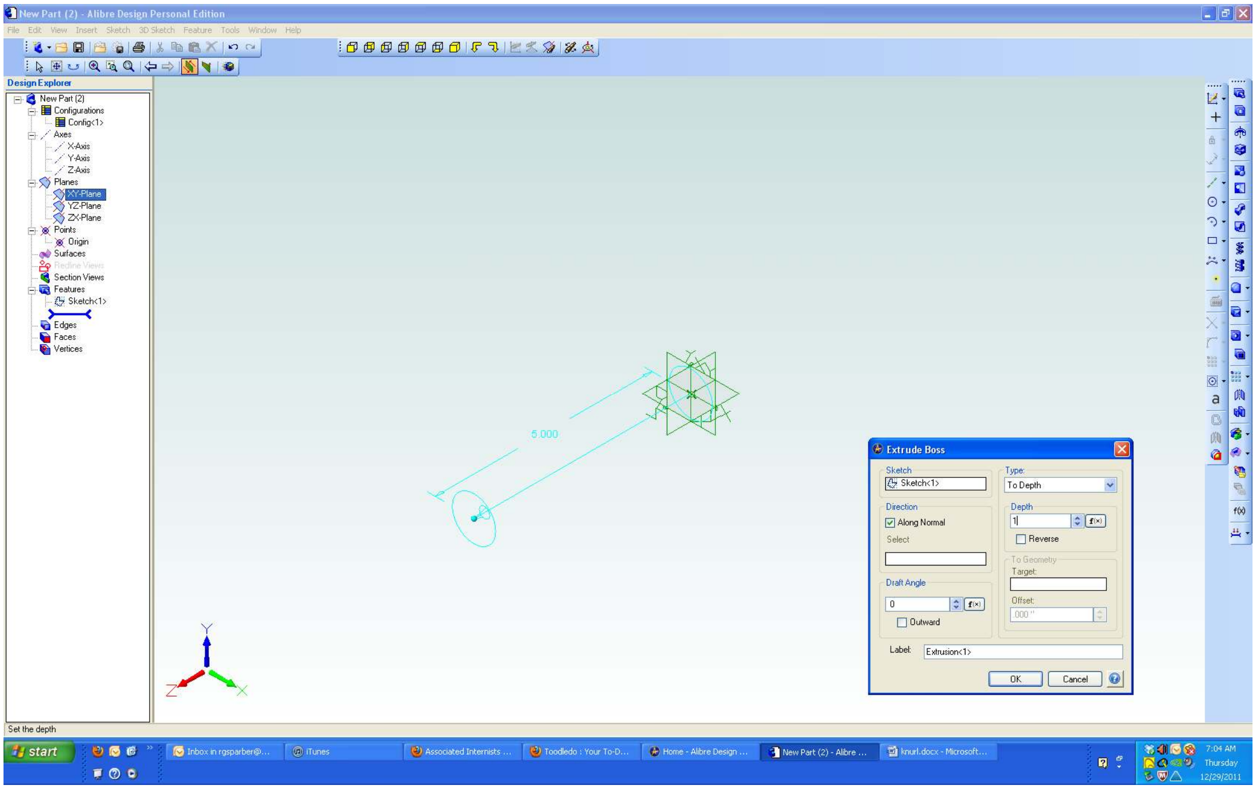
I started by selecting my 2D Sketch tool and the XY plane.



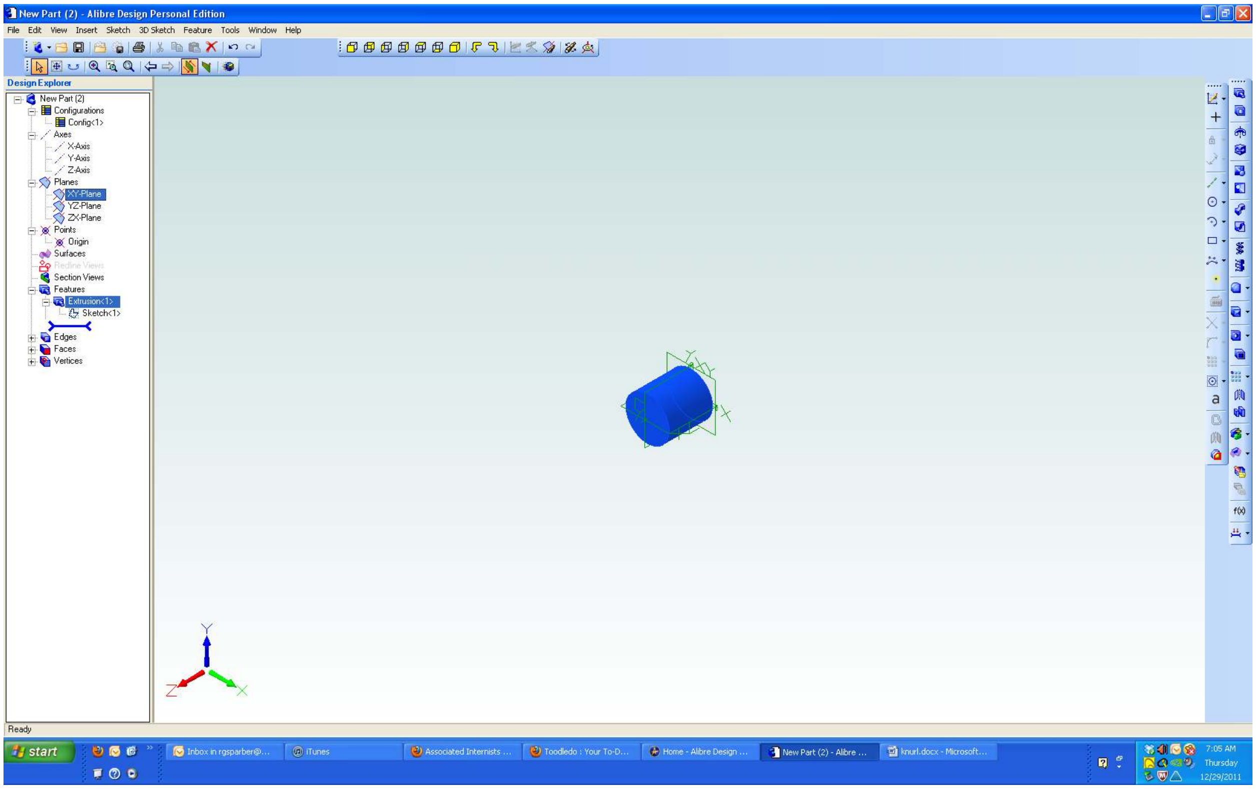
Next, I drew reference lines on the X and Y axis and locked them in place with the lock constraint tool. Lee told me that this is just good practice.



I drew a circle and dimensioned it to 1".

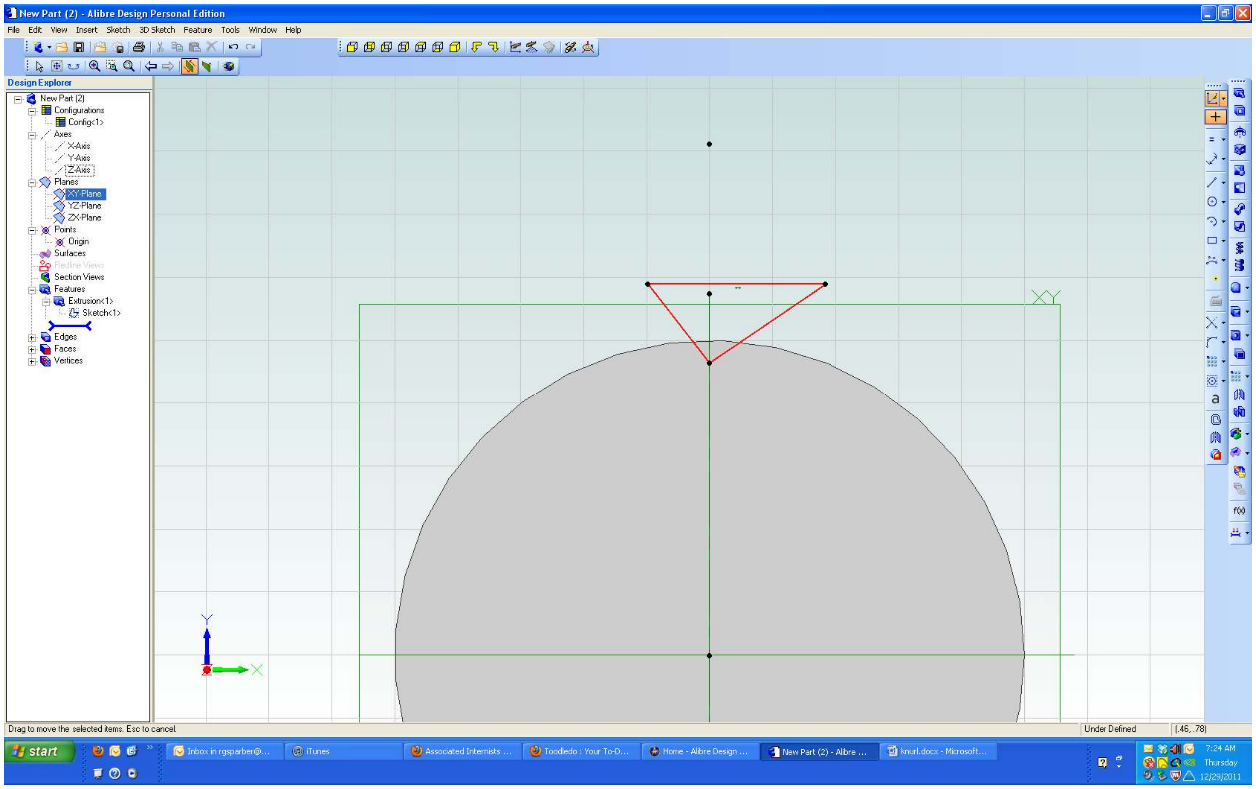


I used the extrude boss tool to form my cylinder. The default length is 5” but I have input 1”.

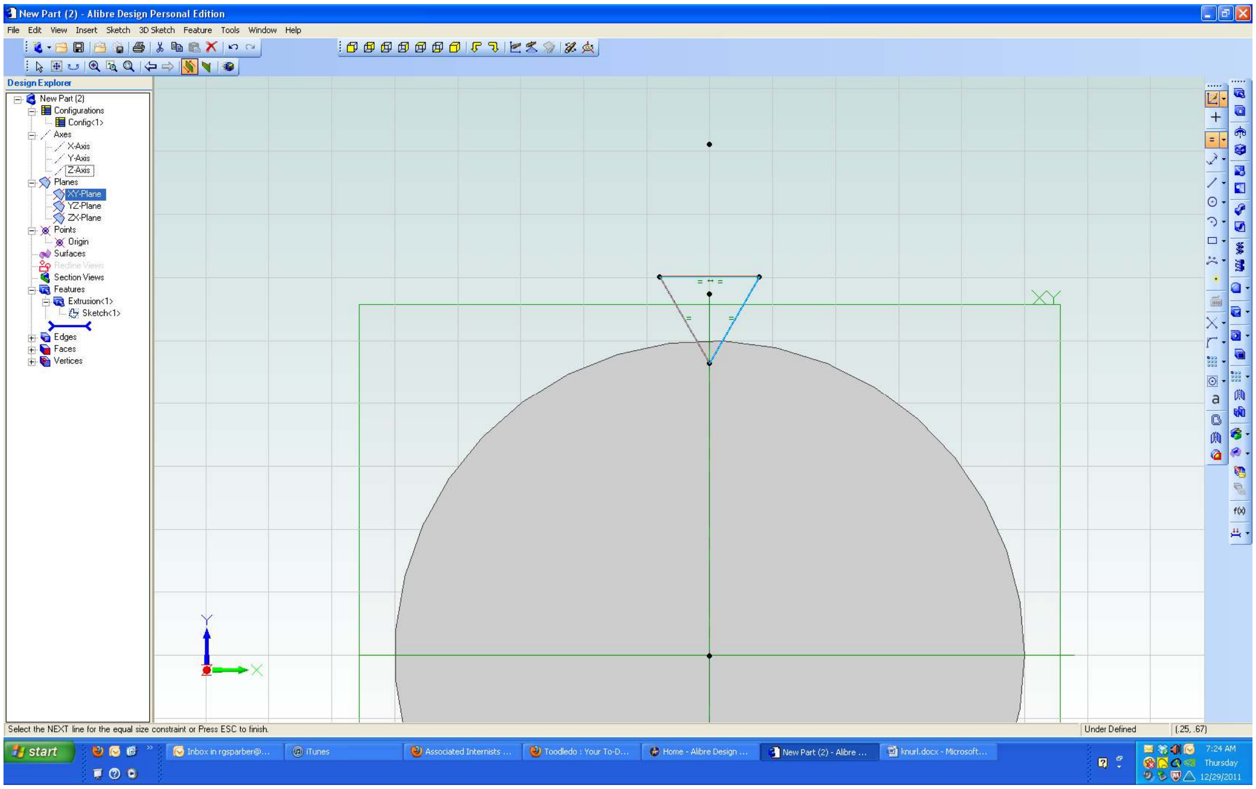


When I hit OK, my 1” diameter and 1” long cylinder appears.

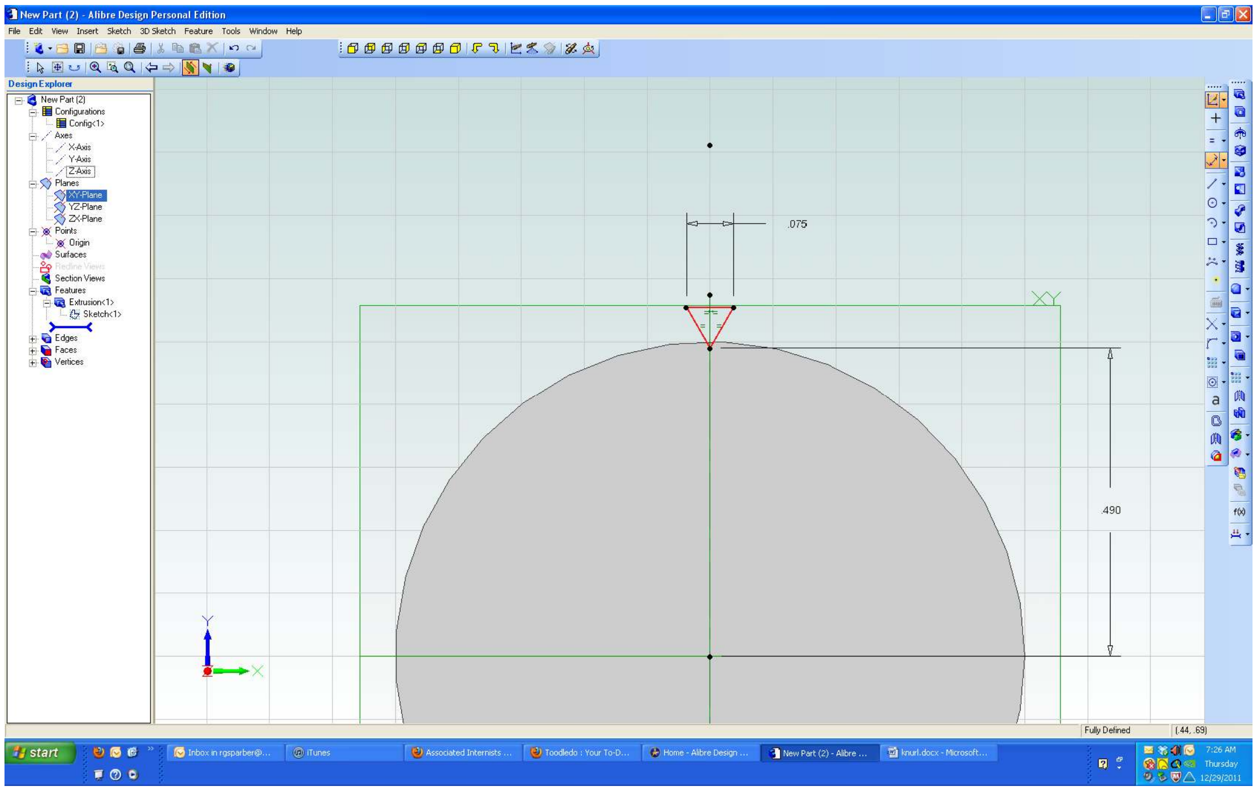
Now I’m ready to define my knurling feature.



I selected my 2D sketch tool and the XY plane. Then I drew this triangle. It will be one end of a cut over the surface of the cylinder.

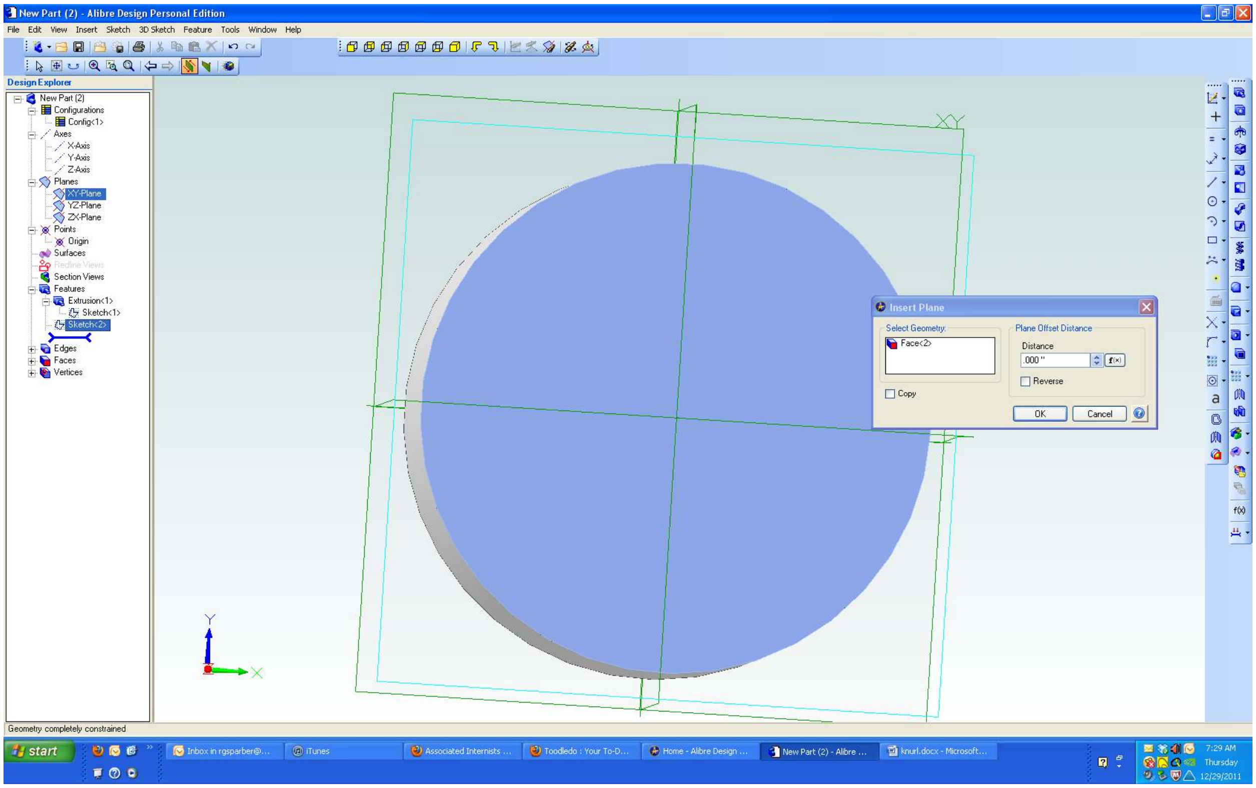


I used the equal constraint to set all sides of my triangle to be of the same length.

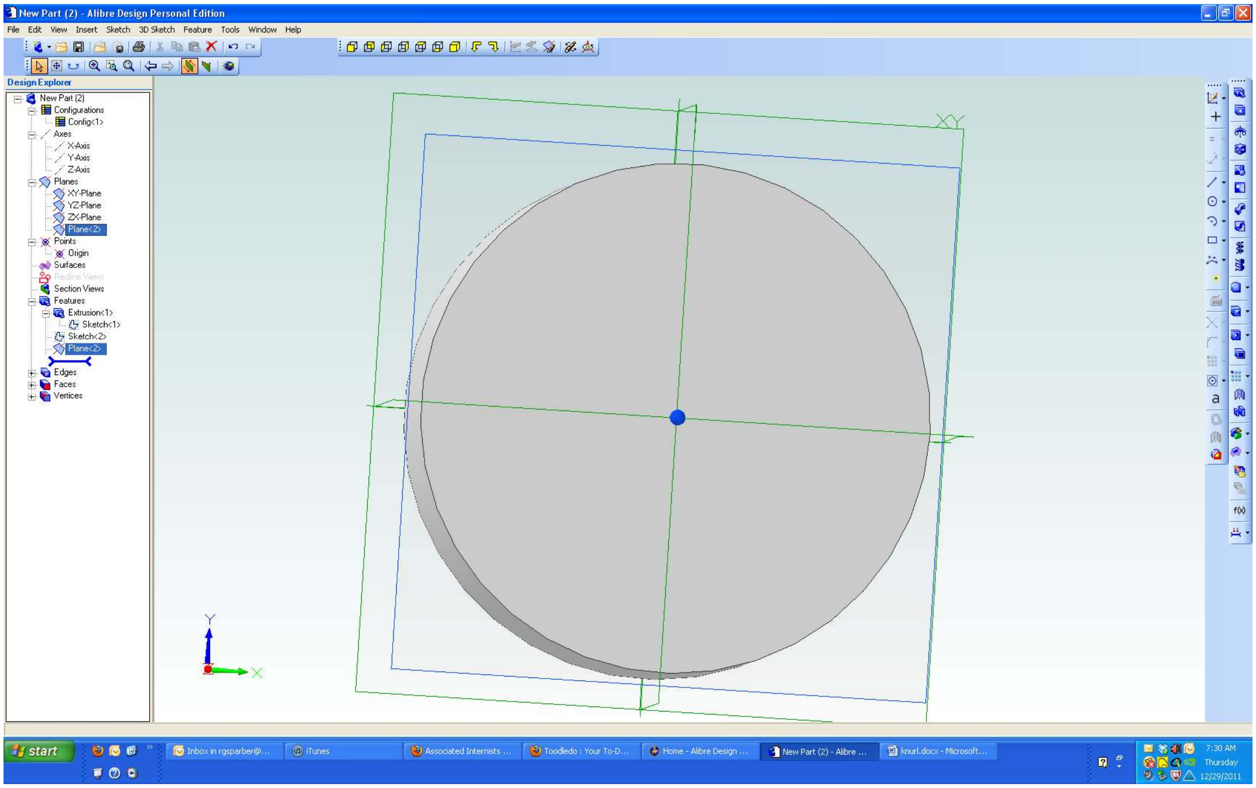


I set one side of the triangle to 0.075” which forces the other two sides to be this long too. Then I set the bottom of the triangle to be 0.49” from the center of my cylinder. This means that my knurl has a depth of 0.01”.

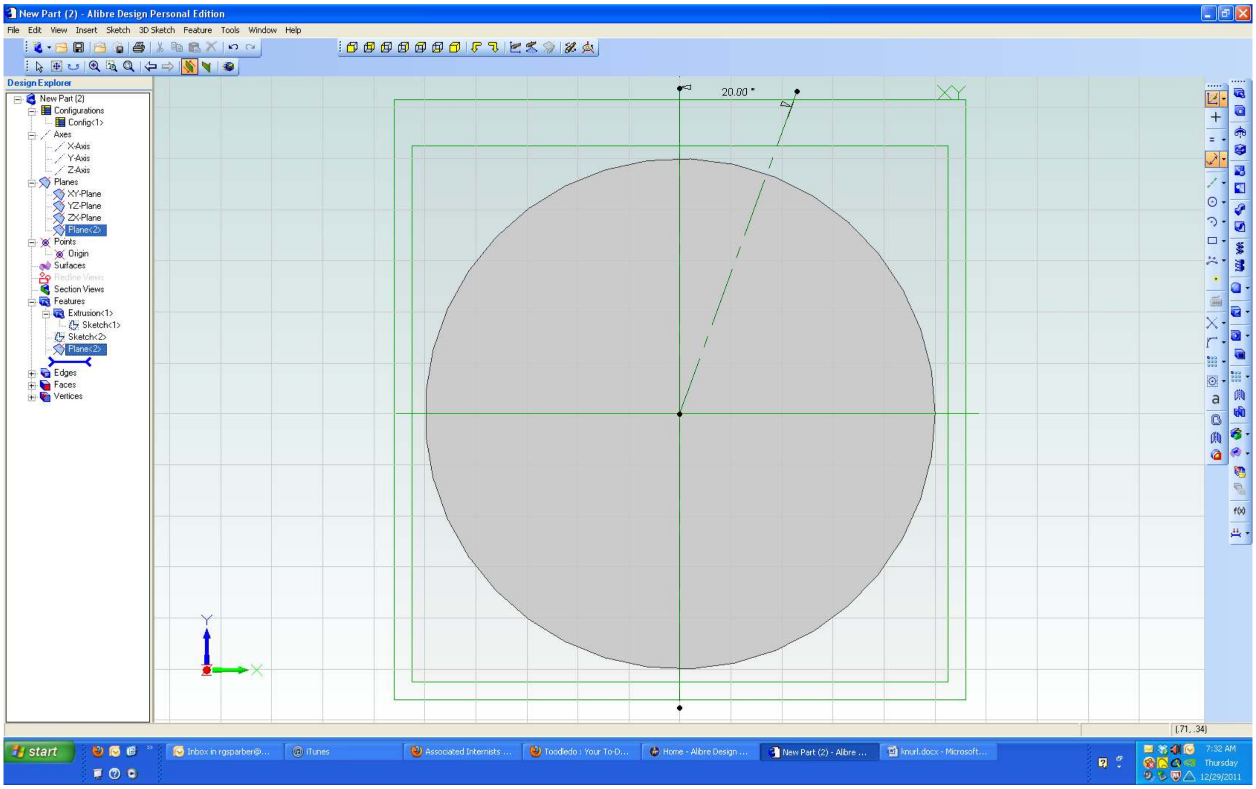
When I exited my 2D sketch, my triangle was defined as Sketch<2>.



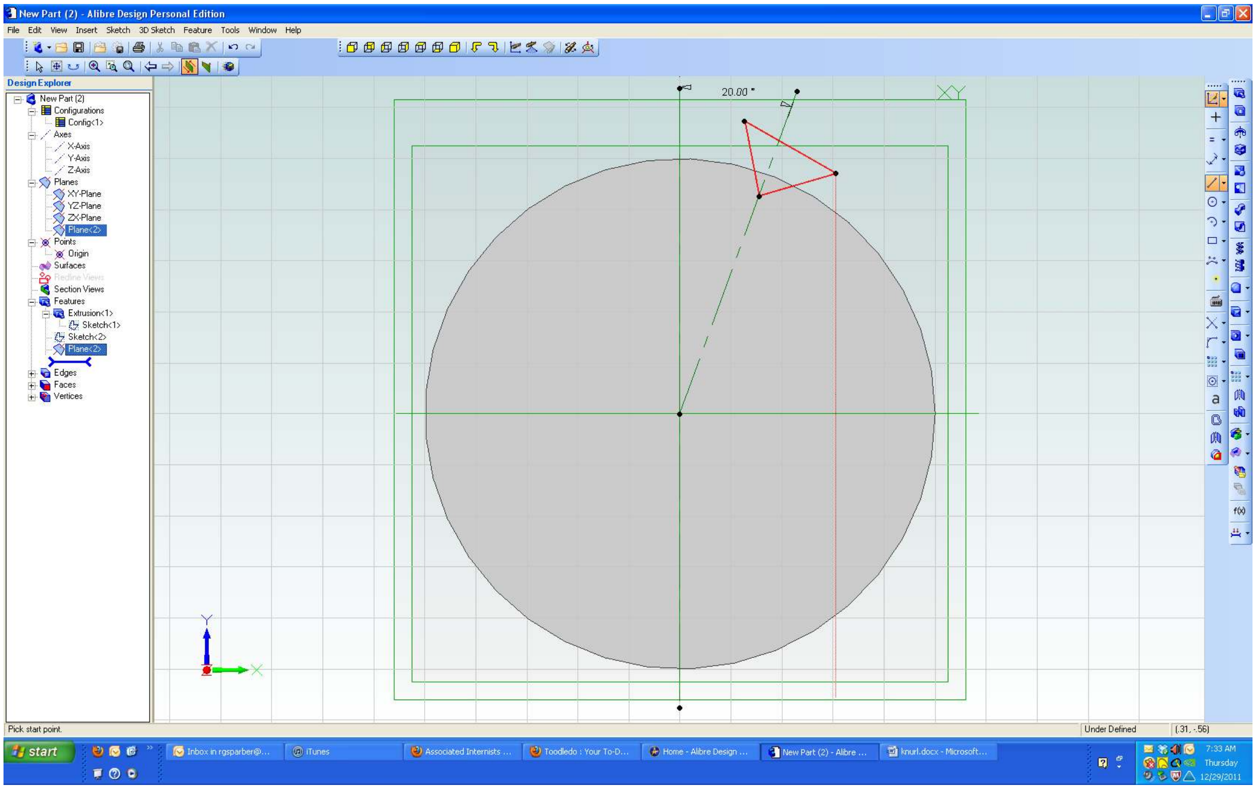
In order to draw the other end of the cut on the opposite face, I had to first define a new plane that sits on the other end of the cylinder.



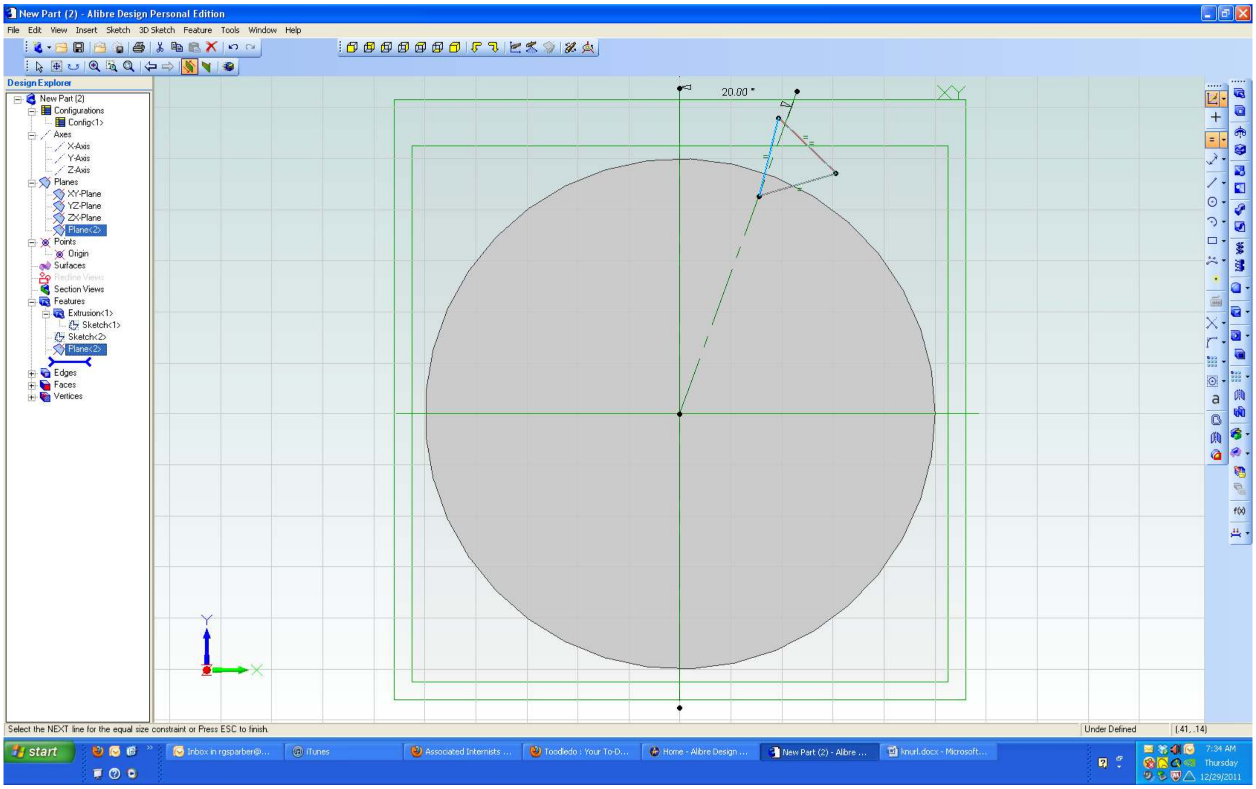
Alibre defined Plane<2> as I left the 2D sketch.



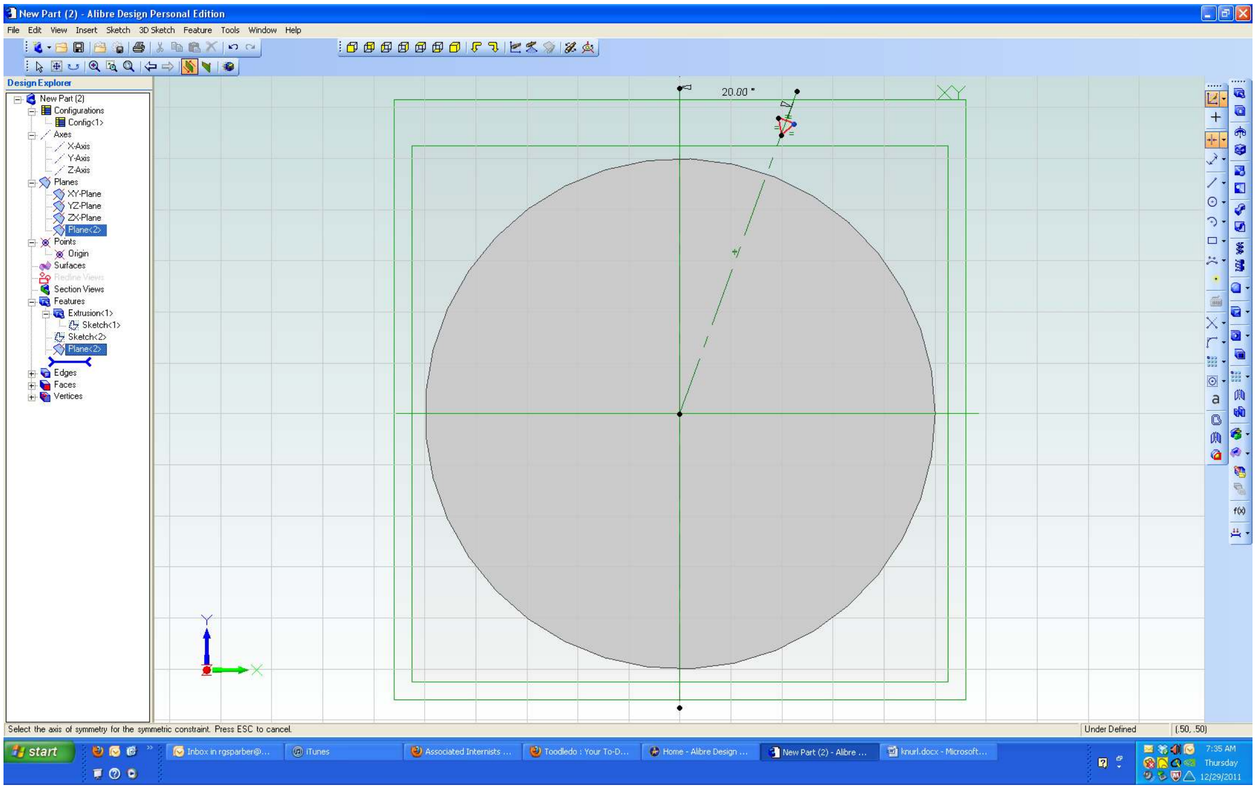
This time I used my 2D sketch tool on my new Plane<2>. I drew a reference line from the center to beyond the perimeter of the cylinder face. Then I set the angle to 20°.



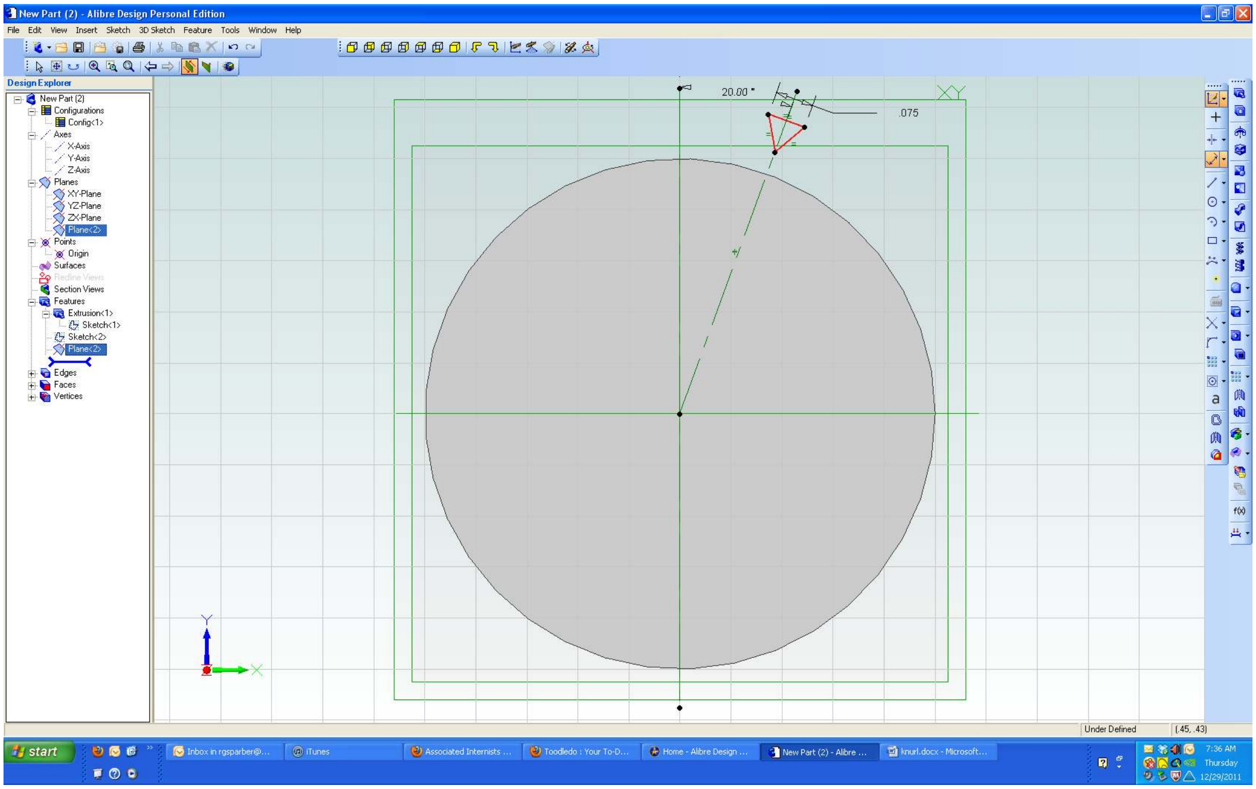
Here is my second triangle.



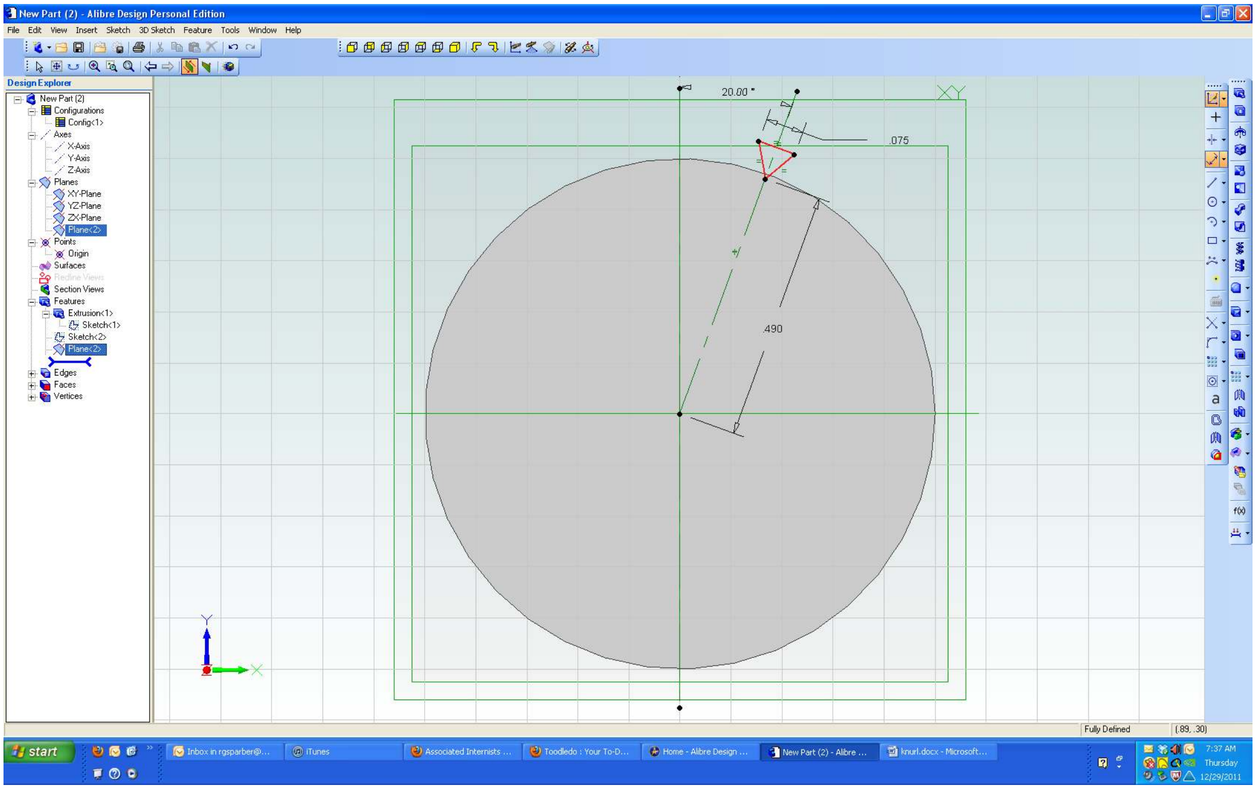
I applied the equal constraint to force all sides to be of the same length.



Then I used my symmetric constraint to force the top of the triangle to be centered on my reference line.

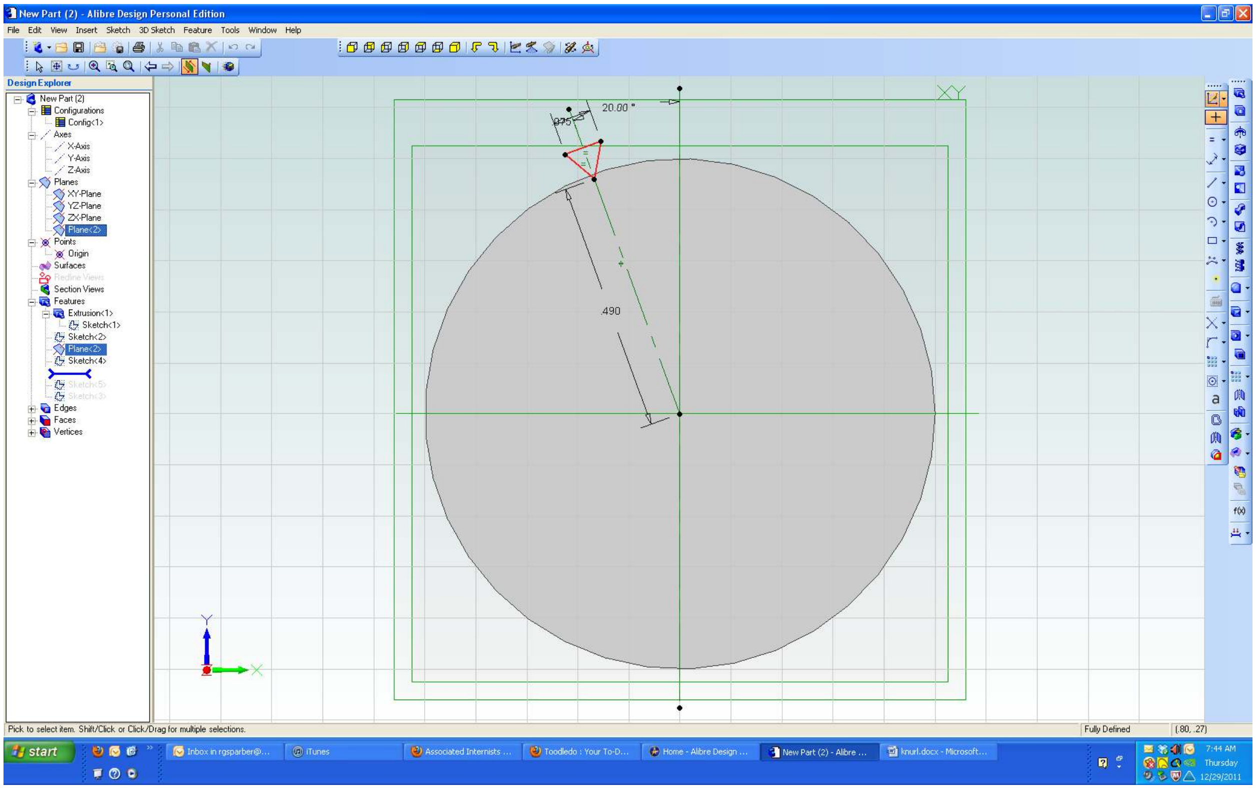


When I dimensioned one side to be 0.075” long, the other two sides of my triangle changed to be the same length.



And finally, I set the distance from center to the bottom end of my triangle.

When I exited 2D sketch, my triangle was given the tag Sketch<3>.

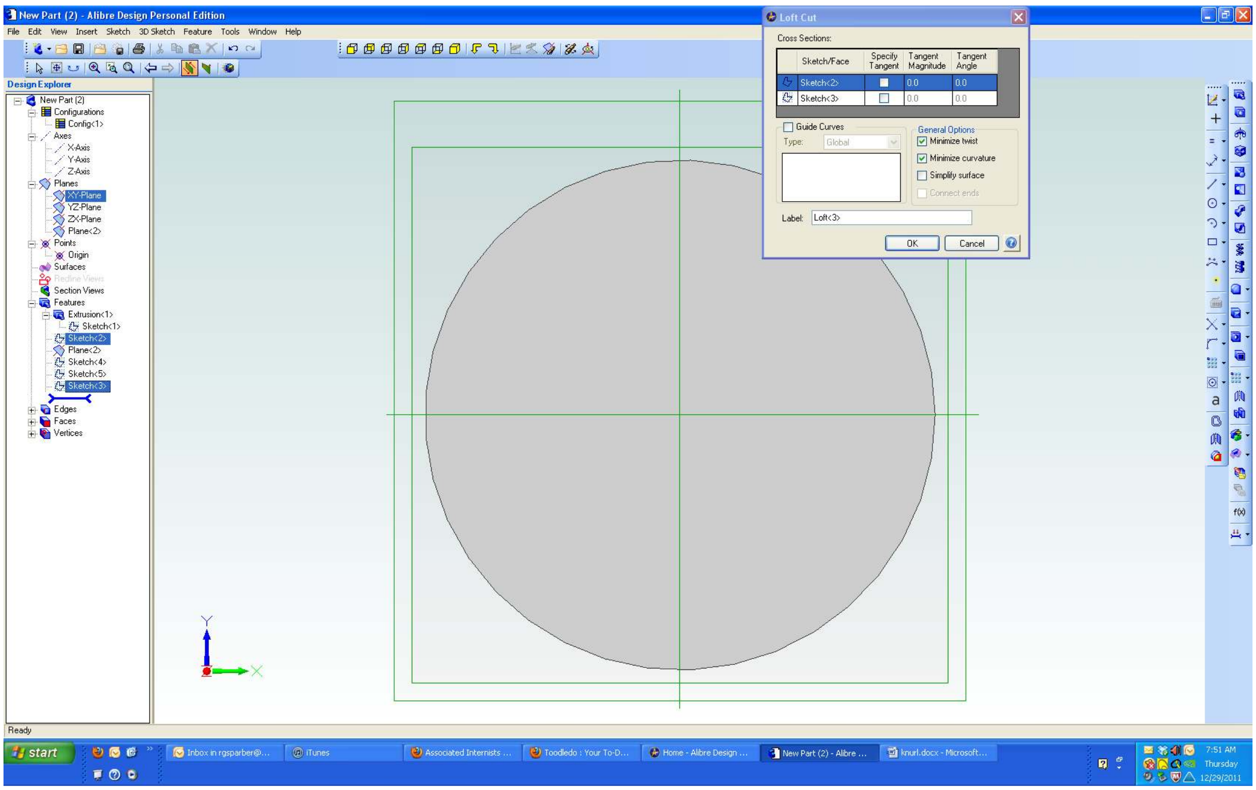


Back on my Plane<2>, I drew a triangle similar to Sketch<3> but 20° to the left of the centerline. When I exited, this was tagged Sketch<5>.

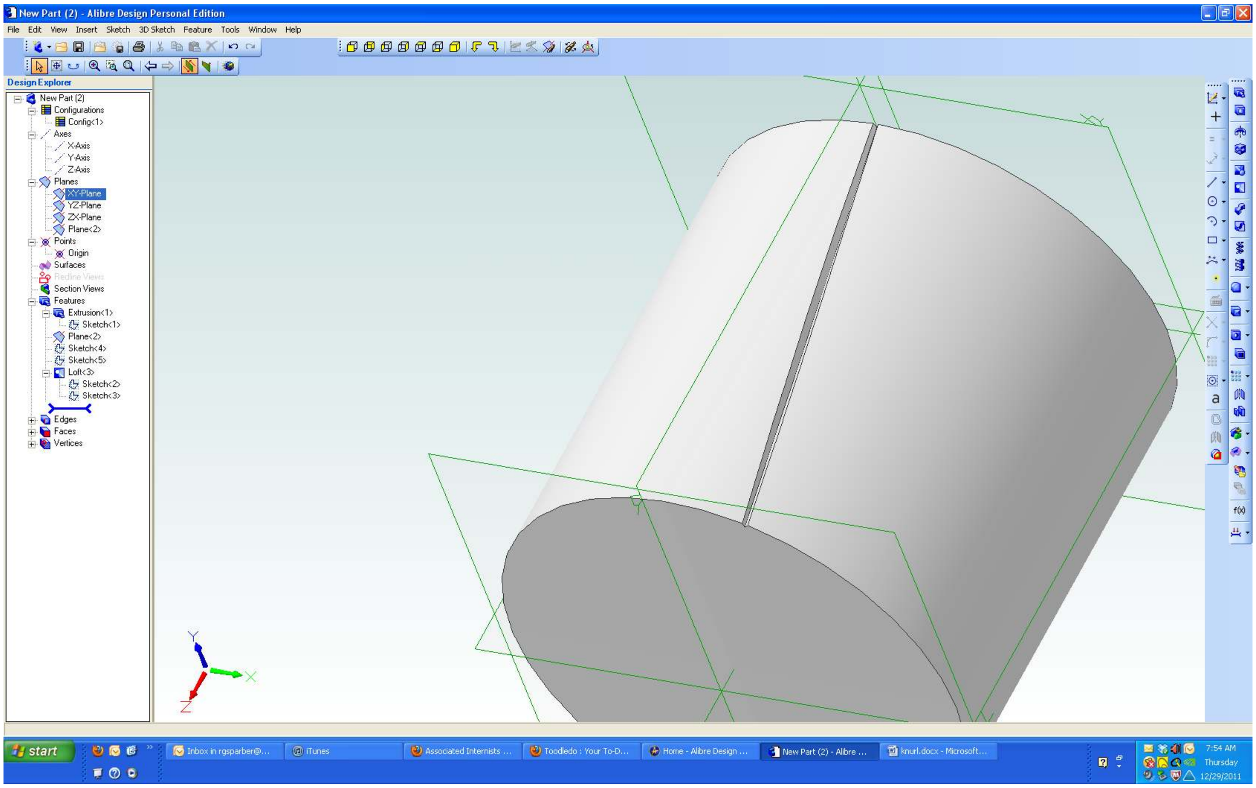
So now I have Sketch<3> with my triangle angled 20° to the right of my centerline and Sketch<5> angled to the left of my centerline.

Using my 2D sketch tool with the XY plane, I drew another triangle identical to the one found in Sketch<2>.

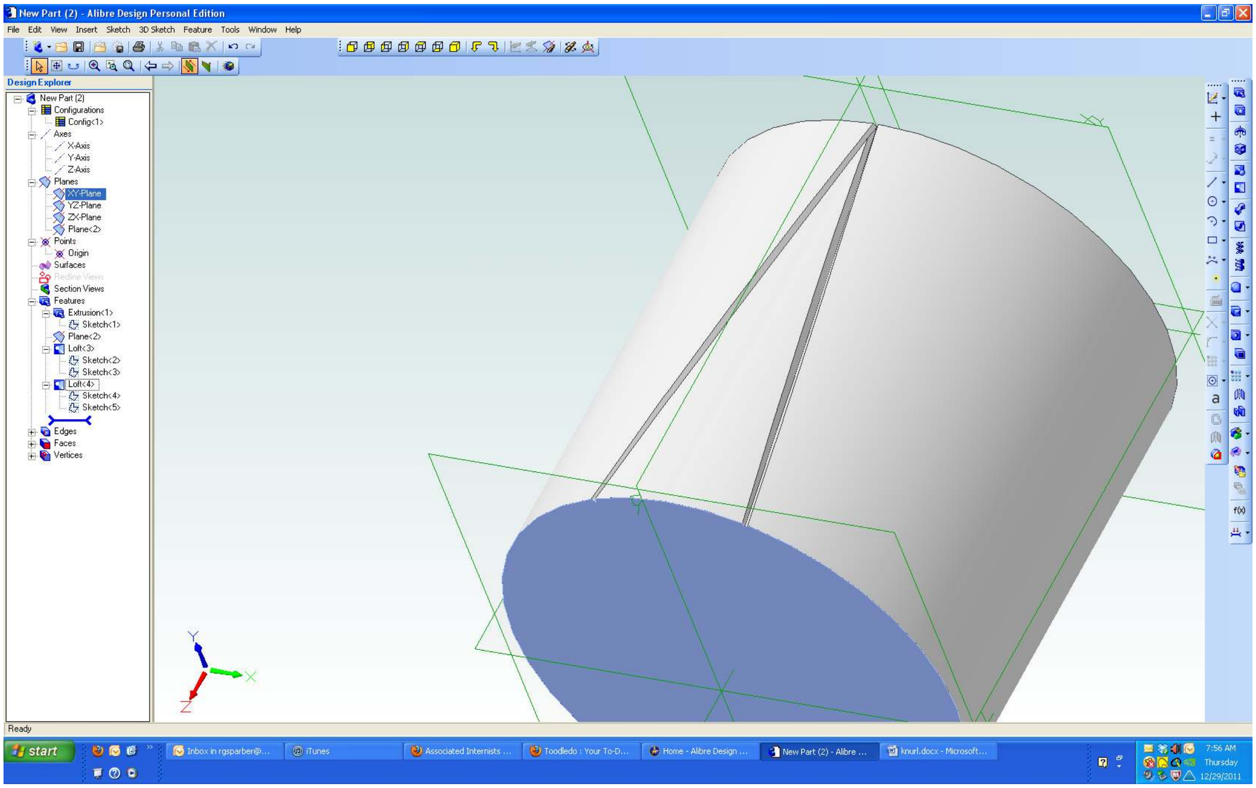
So now I have Sketch<2> with a triangle on my vertical line and Sketch<4> which looks the same. Why do this? Because the tool that will cut the groove between sketches must use unique end figures. If I use Sketch<2> and Sketch<3> to define a groove, I cannot use Sketch<2> again with Sketch<5> to define another groove.



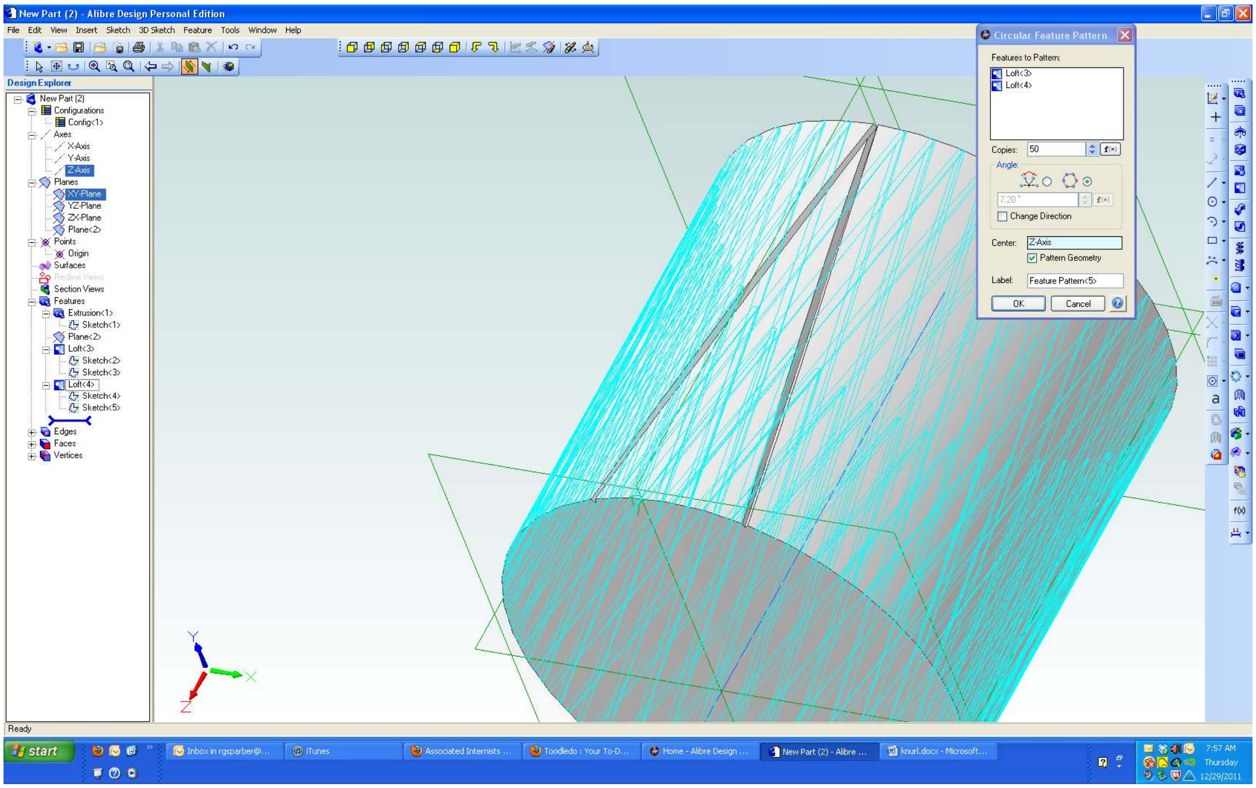
I had brought up my Loft cut tool and selected Sketch<2> and Sketch<3> as the cross sections. At first the OK button did not turn black. Then I realized that Sketch<3> was below the line that looks like this: >-------<. I used the tool in the lower right corner of the window with the same shape to enable all hidden features. Then I was able to select both Sketches and get the OK button to turn black.



Here is my first cut.

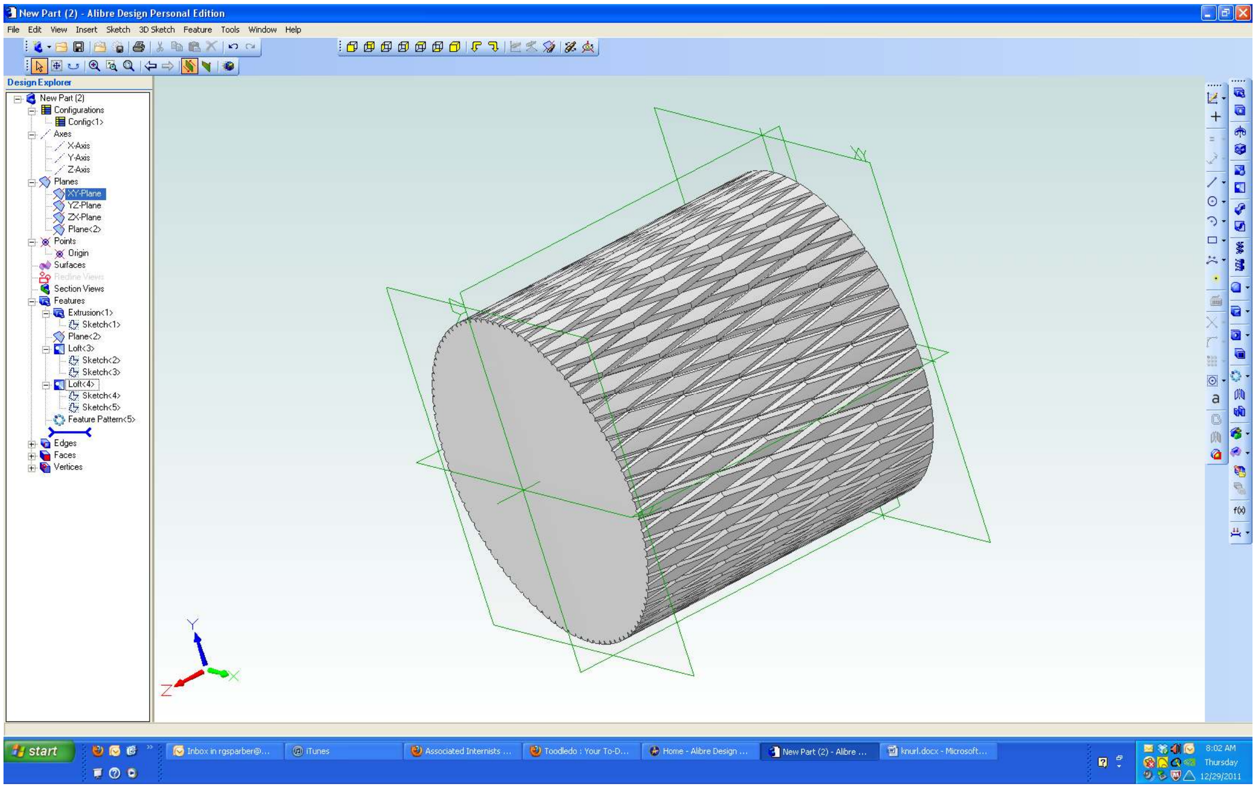


I repeated the process using Sketch<4> and Sketch<5>.

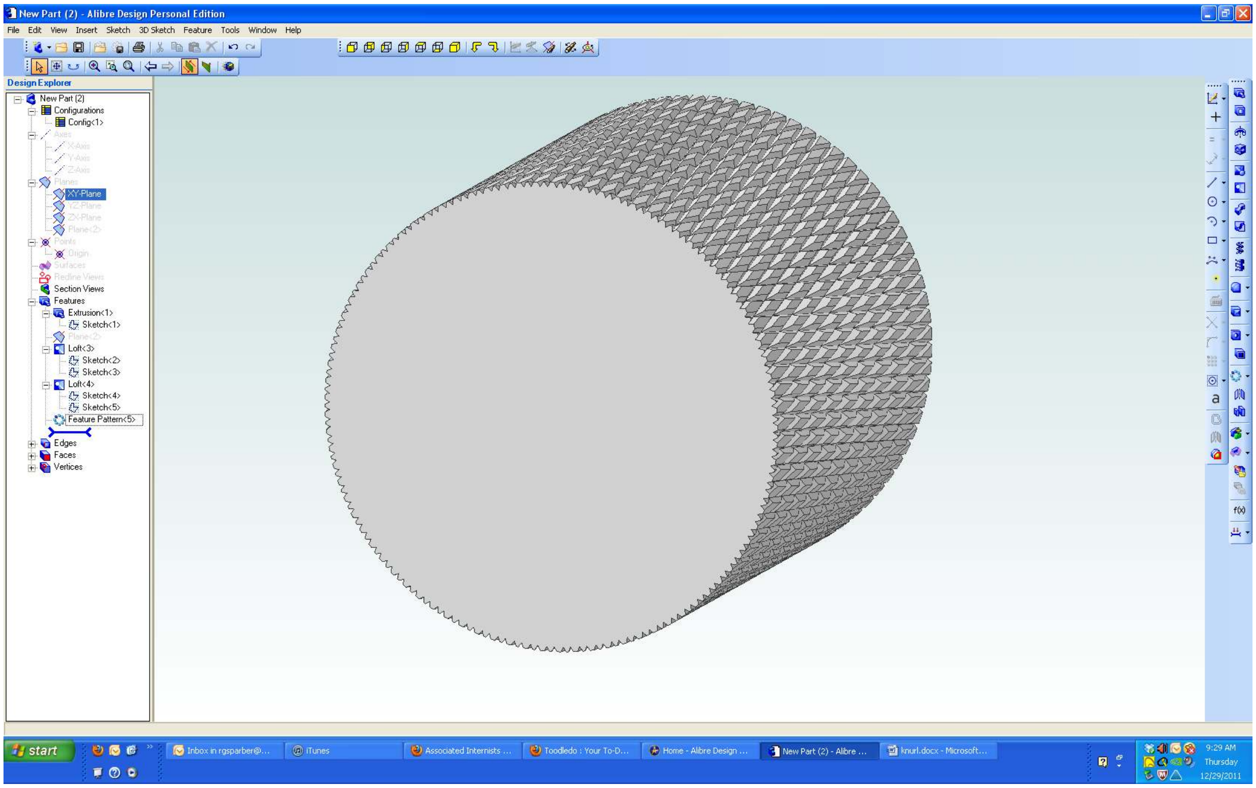


I brought up my Circular feature pattern. Then I selected the two cuts, Loft<3> and Loft<4> to put in my "Features to be patterned" box. I put in 30 for the number of copies. If this doesn't look right, I can change it later. Next I clicked on Center dialog box and clicked my Z axis. This will cause my copied features to be cut around this axis. At this point, my OK button turns black and I click it.

A lot of computation must occur so it takes a while for the new surface to appear.

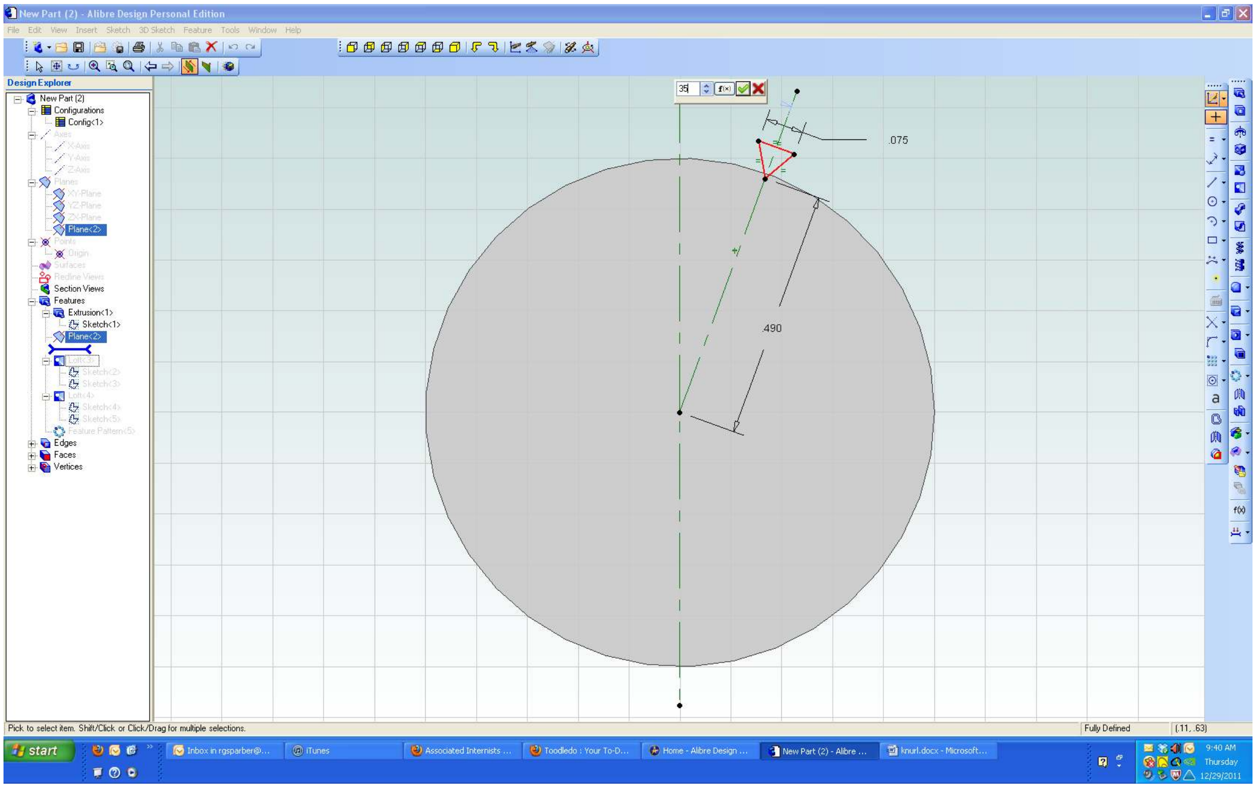


After waiting 25 seconds, this appears. The general shape is OK but there is too much flat between cuts.



This time I used 85 copies. It looks about right but I decided to go into my shop and measure my knurling tool. I will next modify my drawing to match what it forms.

My knurling tool consists of two wheels. Each one has a series of ridges 35° from their axial centerline.



I went back to Sketch<3> and changed the angle from 20° to 35°. I did the same thing to Sketch<5>.

I also measured the diameter of the knurling wheel. It was 0.758”. Circumference equals π x Diameter so my wheel comes out to 3.14159 x 0.758” = 2.381”.

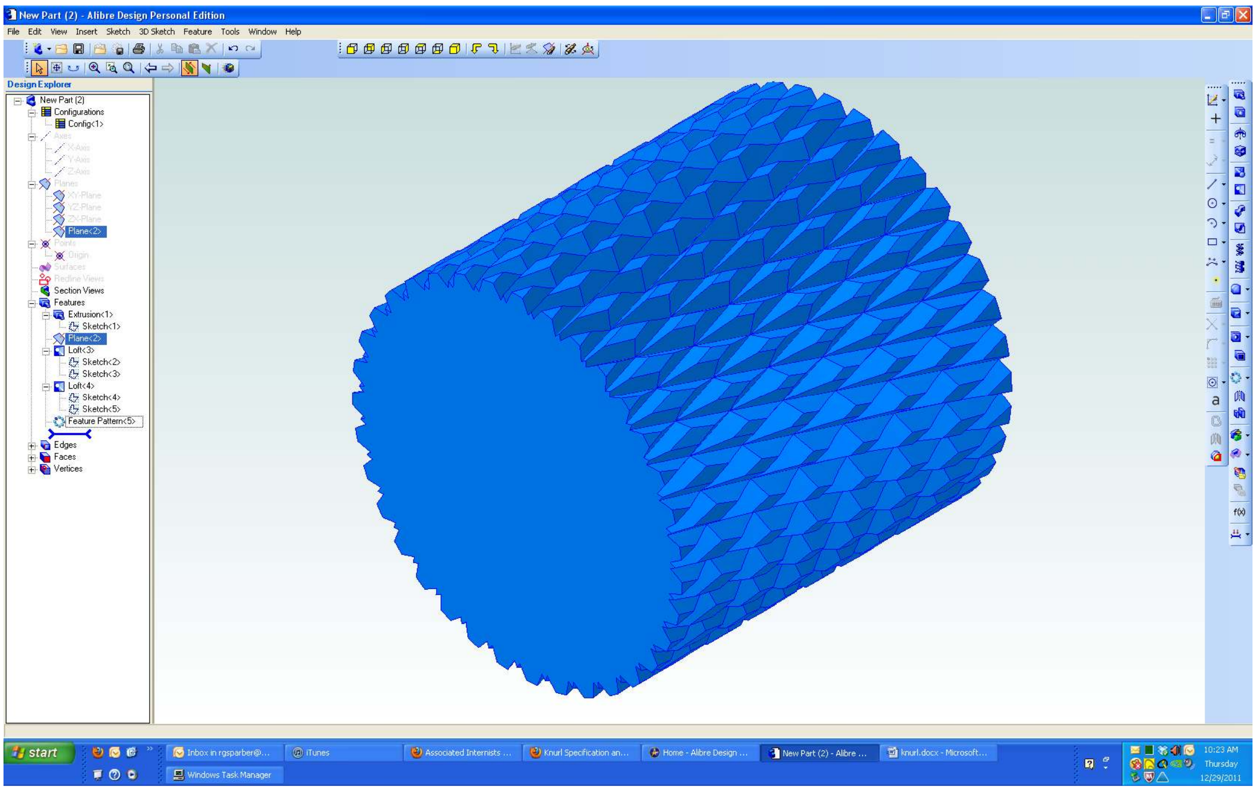
The distance between ridges measured out at 0.0625”. I can figure the angle by realizing that when I go around the entire circumference, it is 360°. So to go the distance between ridges, I go

$$angle\ between\ ridges = \frac{0.0625"}{2.381"} x\ 360°$$

$$angle\ between\ ridges = 9.45°$$

These ridges are equally space. If I divide 9.45° into 360° I get 38.1 ridges. I can’t have a fraction of a ridge so the correct number of ridges must be 38. My guess of 85 was way off. I was going around the part more than once.

I then measured the depth of the knurl and found it to be around 0.035”. So I went back to Sketches 2 through 5 and updated the distance from center to bottom of triangle to 0.465”. This caused the top of my triangle to be under the surface of the cylinder in the middle so I then had to change the size of the triangle from 0.075” on a side to 0.1”. Since the sizes of the triangle were constrained to be equal, I only had to make this change on the size I dimensioned.



I am focused on drawing the knurl pattern, not on actually knurling. The numbers given here are not exact. If you wish to learn more about knurling, one reference is

http://www.engineersedge.com/manufacturing_spec/knurl_chart.htm

I welcome your comments and questions.

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