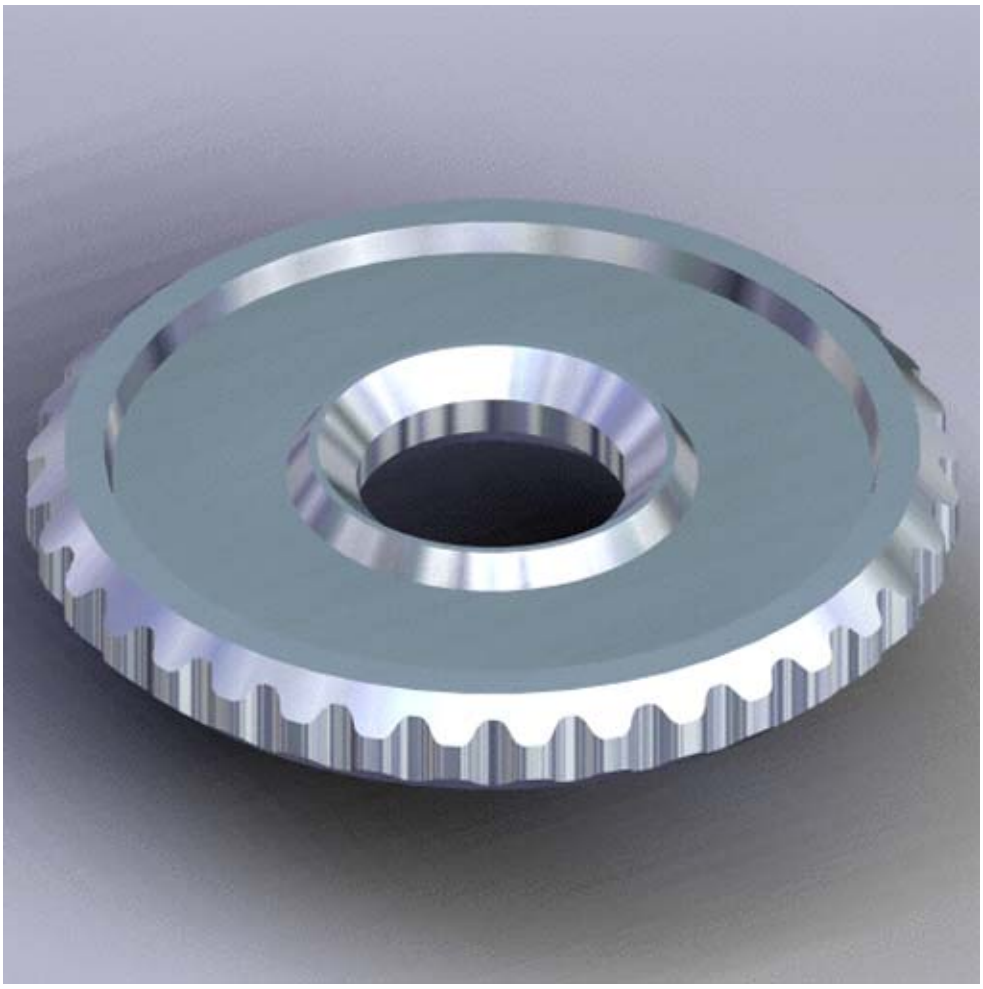
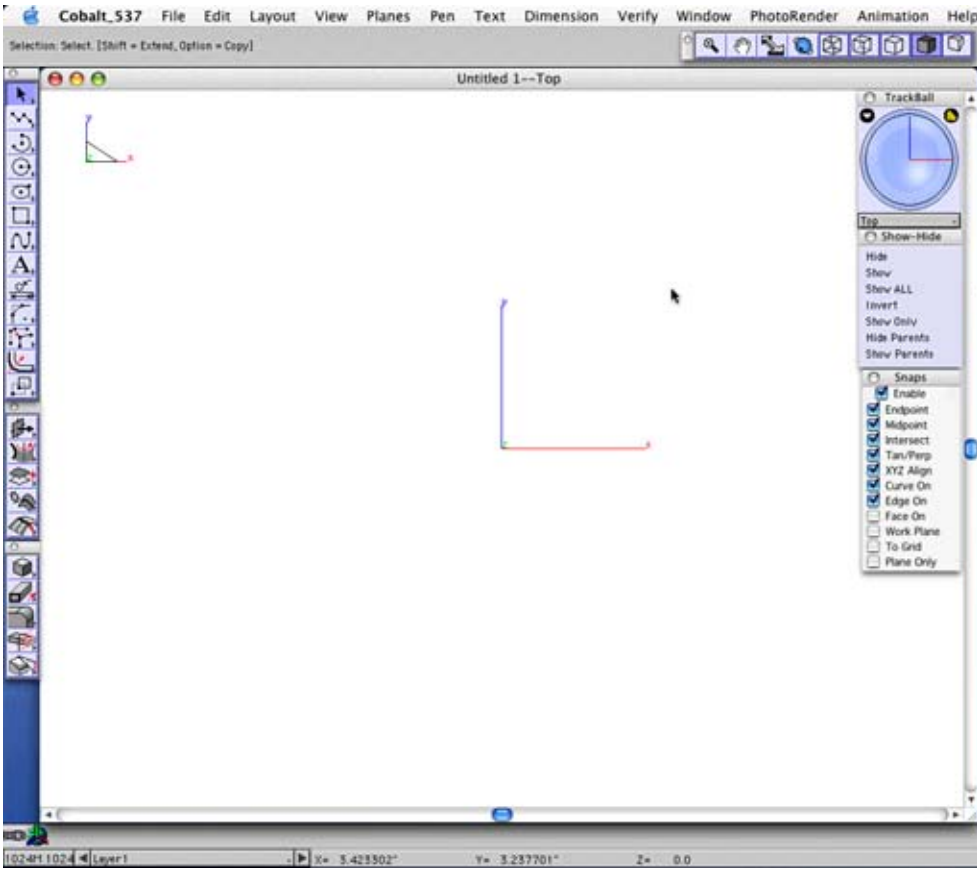
Modeling a Gear



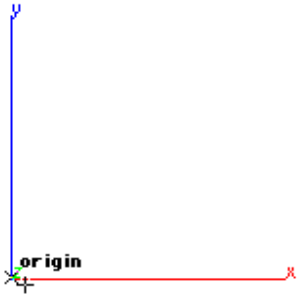
This tutorial describes how to create a toothed gear. It combines using wireframe, solid, and surface modeling together to create a part. The model was created in standard units. To begin, the screen is arranged as shown below. The **Standard Tools, Surface Tools** and **Solid Tool** palettes are on the left side of the screen. The **View, Trackball, Show-Hide** and **Snaps** palettes are along the right side of the screen. All of these tools are accessible from the ***Window*** menu.



The first step in creating our gear is to create the tooth profile. The bad news is that our gear is going to have 36 teeth. The good news is that we only have to create one of them.

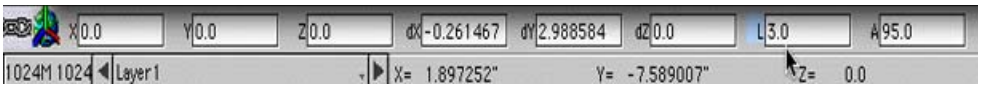
Select the **Line Tool** from the Tools Palette.

The lines we are about to create are guide lines for us to build our tooth. Place the first point of the line at the origin as shown below.



Place the next line straight up along the Y-axis. How far isn't critical because we will type the length and angle after we create it.

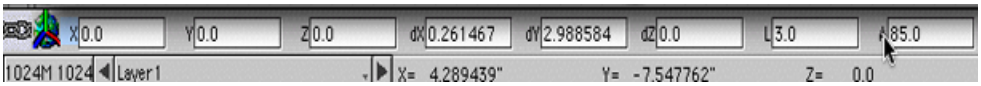
After placing the last point, type **3** into the **L (Length)** field and **95** into the **A (Angle)** field. This will give us exactly the size line we need.



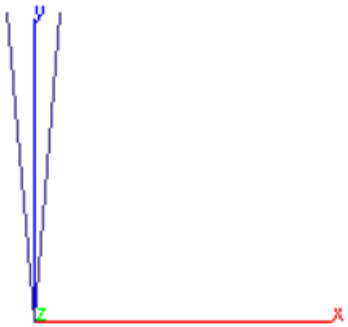
We will create a second line just like we did the first except this time after creating it we will put **85** into the Angle field instead of **95**.

Tip:

Just type 85 into the Angle field and hit enter. A second line will be created that is three inches long and at 85 degrees.



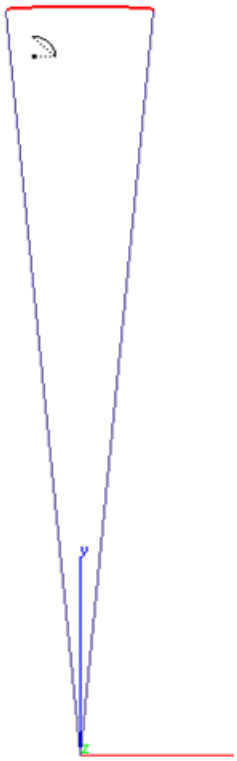
You can see below what the lines should look like after you complete these steps.



Next we will create an arc along these two points.

Select the **Center-Point Arc** tool from the Tools Palette.

The first piece of information that is asked for is the center point. Pick the **Origin** for our center point.

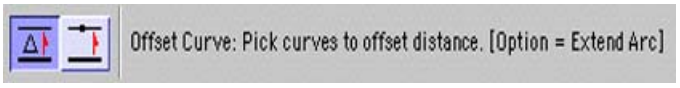


The other two points that it needs are the points along the arc. Select the two endpoints of the lines for those points. The arc created should look like the image below. You can hit **CTRL+F** on Windows or **Command+F** on a Mac to fit the model to the screen (this is the shortcut for Zoom All). You could also go to ***View>Zoom All***.

We have just created the outer section of our tooth. Now we will create the inner section. We don't have to go through creating all of those lines again, we can just offset the arc we already made a little bit to create it.

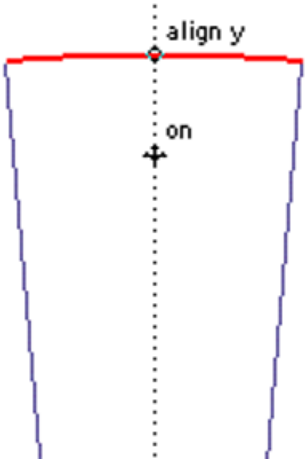
Select the **Offset by Specified Distance** tool from the Tools Palette.

Make sure that the option for (offsetting to value) is selected in the Message Line.

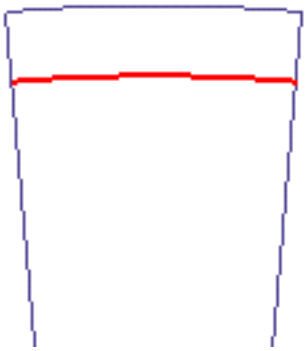


Set the distance to offset to **.125**".

Select the outer arc that we created for the piece to offset. Select a point below the arc as shown below.



The second point that we selected tells the program which way to offset.

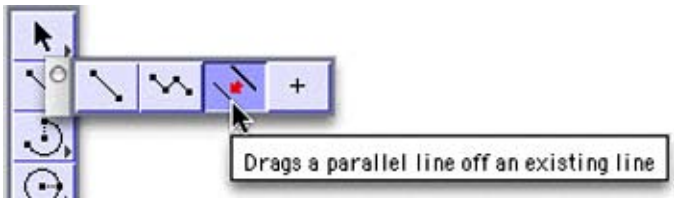


Now we need some construction lines to help us build more of the tooth. We may not actually HAVE to have them, but they sure do help.

While holding **Shift+CTRL** on Windows or **Command** on a Mac, click the origin of the model and drag the mouse straight up. A red dotted line should appear on the screen. This a very short and quick way to create construction or helper lines to create your geometry.

We need a couple of them next to each other to create the rest of the tooth.

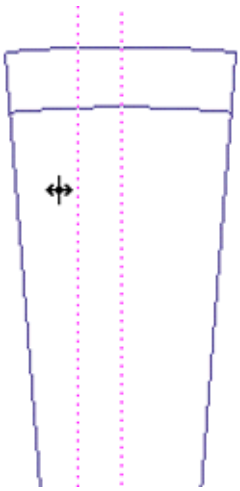
Select the **Parallel Line Tool**.



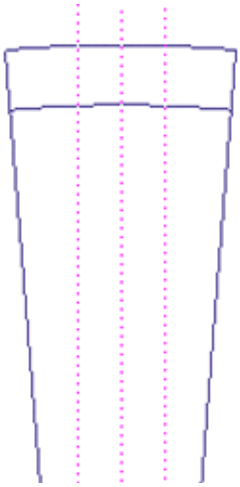
Select the construction line that we created and drag it to the left. After dragging, type **.1** into the Offset field. This sets the offset to exactly .1 inches.

This tool automatically creates a copy of the original line and has the name suggests, it only works with lines.

The new construction line should look similar to the image below.



Next, we will do the exact same thing for the right side. You can drag a new line off of the center construction line or you can just enter a negative value in the status line (**-.10**), and hit the enter key and a new construction line is created.

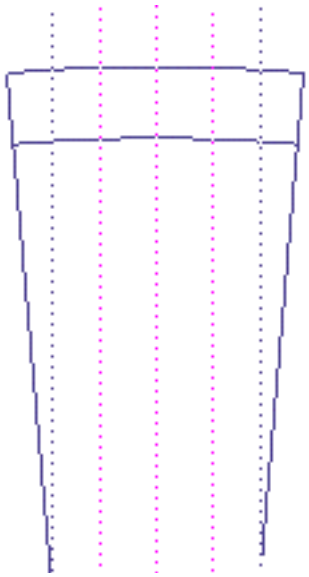


Tip:

You only have to create one parallel line off of the original, the rest of the lines can be created by entering different + & - offset values into the Status Line data box.

We need to do the exact same thing to the original one more time to both sides, but this time type in **.180** and **-.180** for the offset amount.

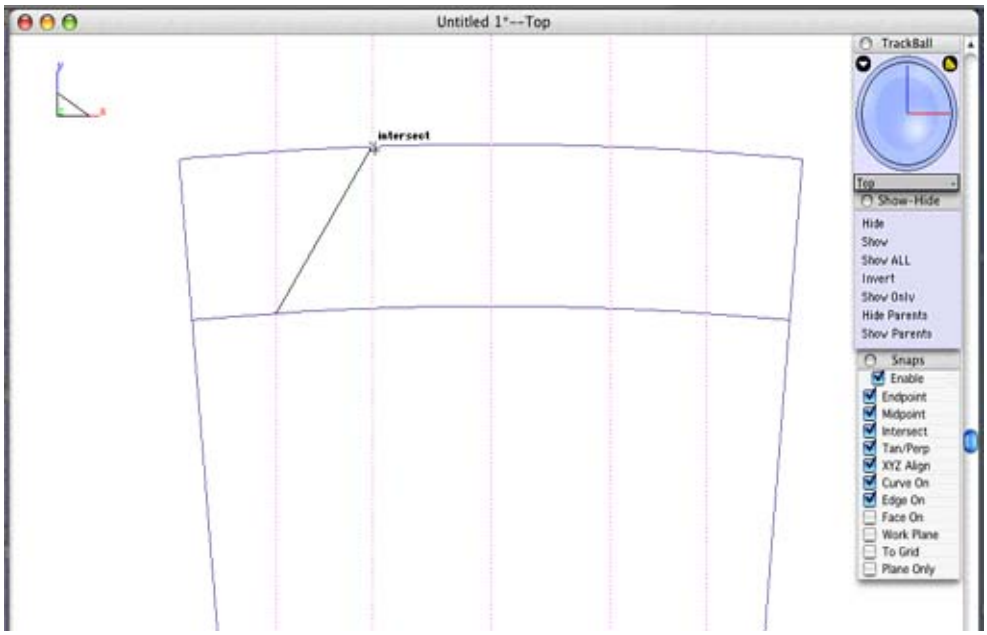
Now that we have our construction lines, lets finish this tooth.



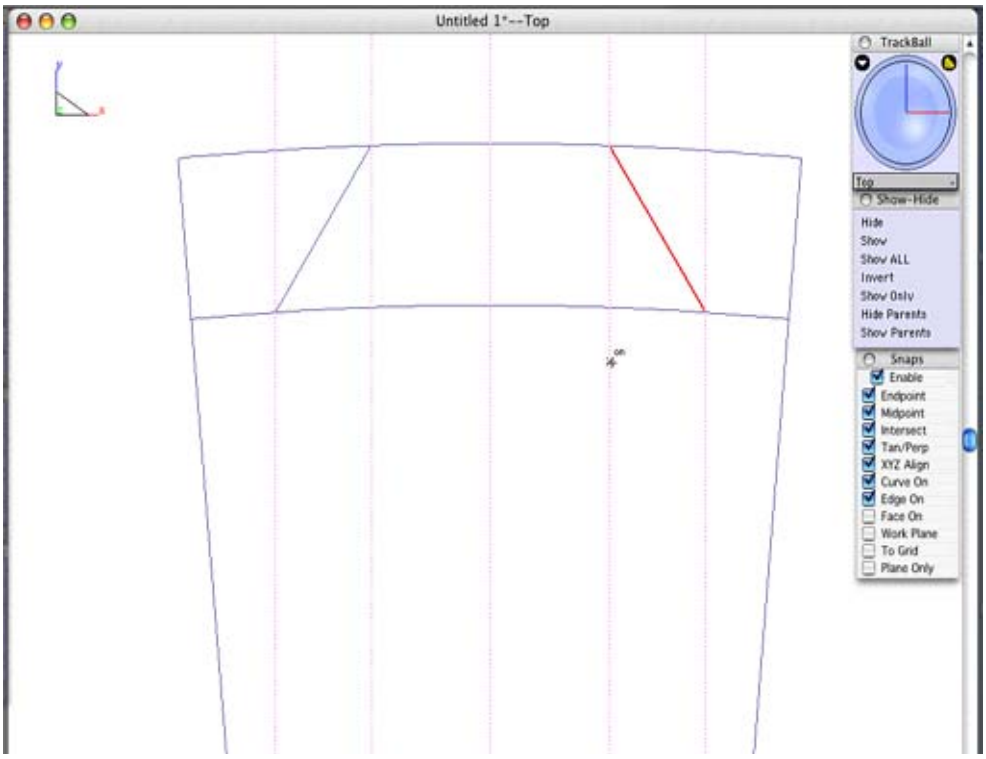
Zoom in on the upper area of the model. It is the only area we are working on right now and we need to get a closer view to make it a little easier to create new geometry.

Select the **Single Line** Tool.

Create a line as shown below. Use the snaps to let you create the lines at the intersections of the arcs and the construction lines.



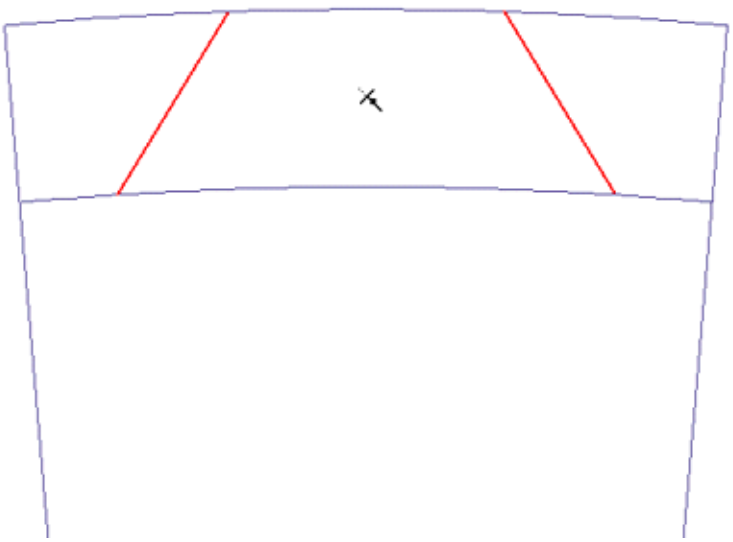
Create another line on the opposite side.



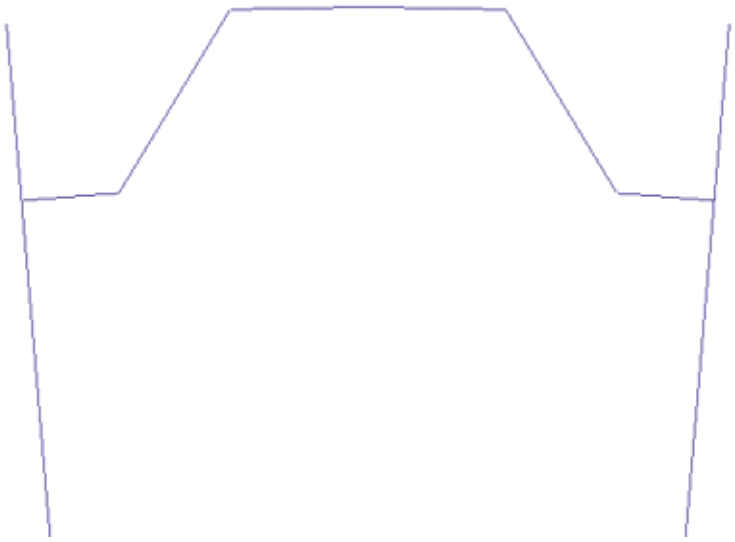
You will no longer need the construction lines, so you can hide them or deleted them.

Select the **Trim Tool** from the Tools Palette.

While holding the **Shift** key select both of the new lines that we just created.



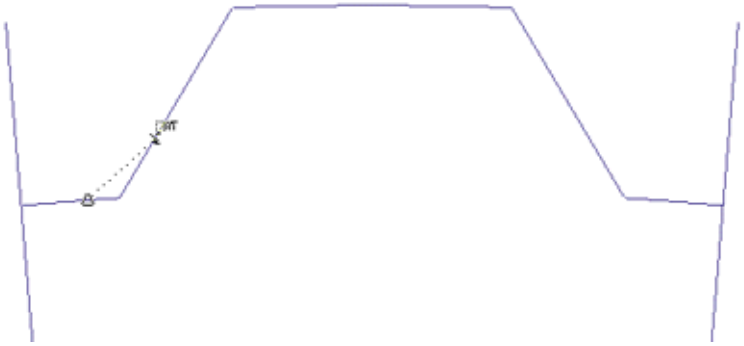
Now we have to select the parts of the arcs to be trimmed away. On the upper arc select the two parts **outside** of the two line. On the inner arc, select the part **inside** the two lines. Your tooth should similar to the one below.



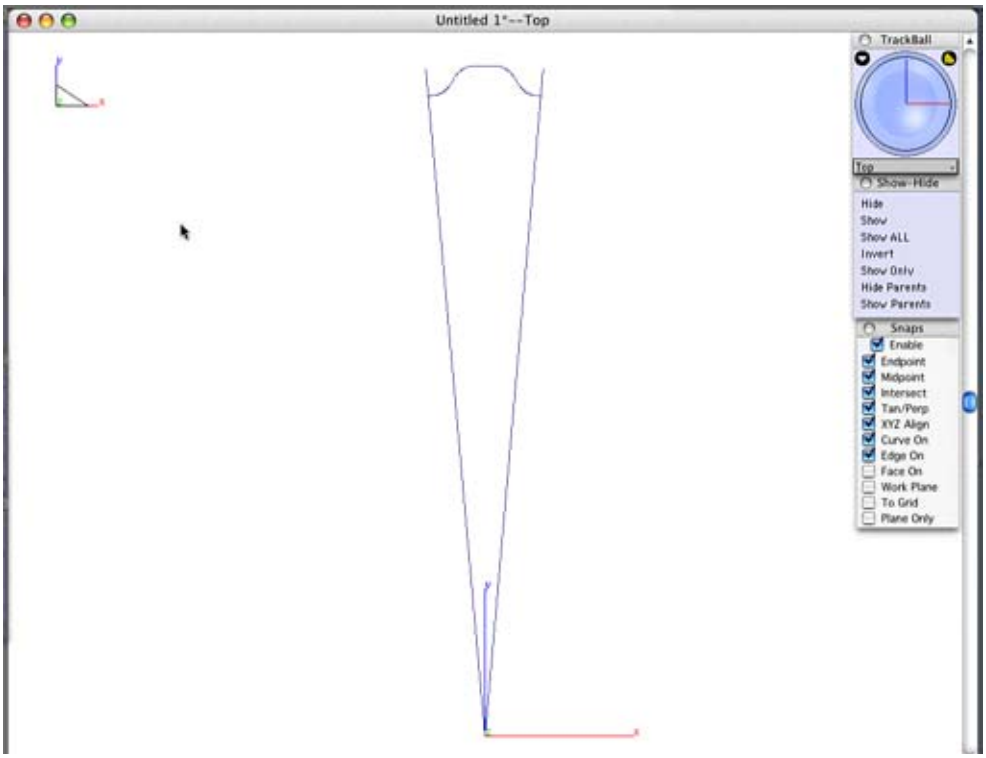
Select the **2-Entity Fillet** Tool from the Tools Palette.

Type in **.1** for the radius.

We will put fillet on all four of the sharp corners of the tooth. Select two lines at a time to fillet them.



Once you've completed all of the fillets your tooth should be rounded out and look like the image below.



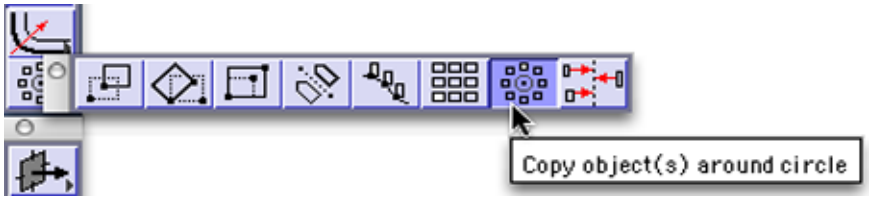
Now that we have finished our one tooth, we can create our solid gear.

Save your file onto your computer

Creating the Gear

With our one tooth created we can copy it around in a circle to create a 2D gear shape. Then we will extrude that shape into a solid gear.

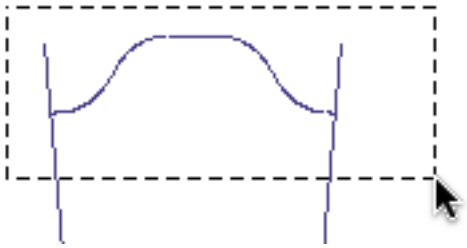
Select the Polar Duplicate Tool.



Select the Tooth that we created. A simple way to do this is drag a selection fence around .

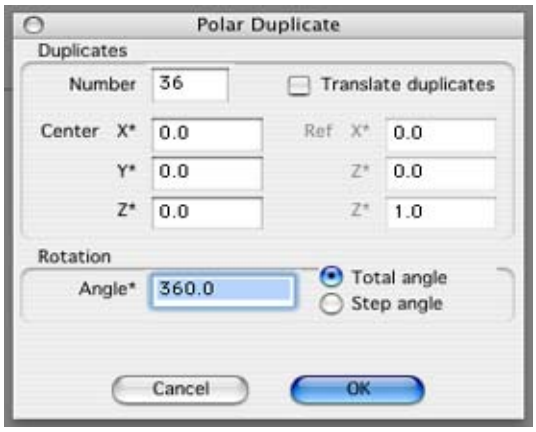
NOTE: The two large lines that we created at the very beginning of this tutorial are not part of the tooth.

Once you have done this and let go of the **Shift** key a dialog will appear. These are the setting for our polar duplicate.

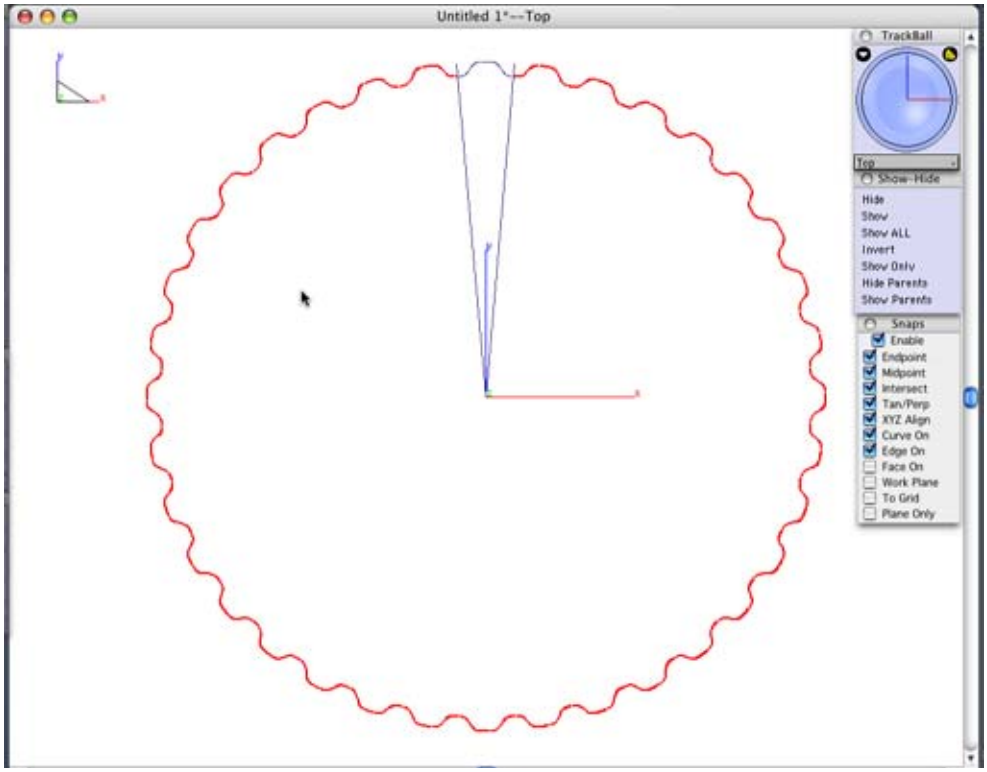


Since our tooth is 10 degrees and we need to create a full circle we will need **36** teeth. The only other option we need to worry about right now is the center of revolution which should default to the location of the last polar duplicate that you performed. If not, you can either type **0** into all of the fields, or you can select the origin in your model and the program will fill in the data from where you clicked.

Select the **OK** button.



There you have it. Now we will use this profile to create a solid.



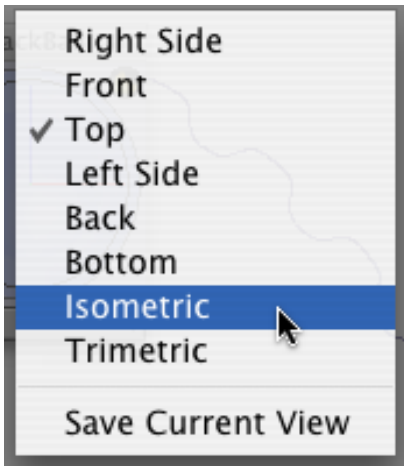
Use the **Select Tool** and pick a point in the drawing area where there is no geometry. This will deselect everything.

The two lines created at the start of this tutorial are no longer necessary and can be hidden.

Select Hide from the Show-Hide Palette.

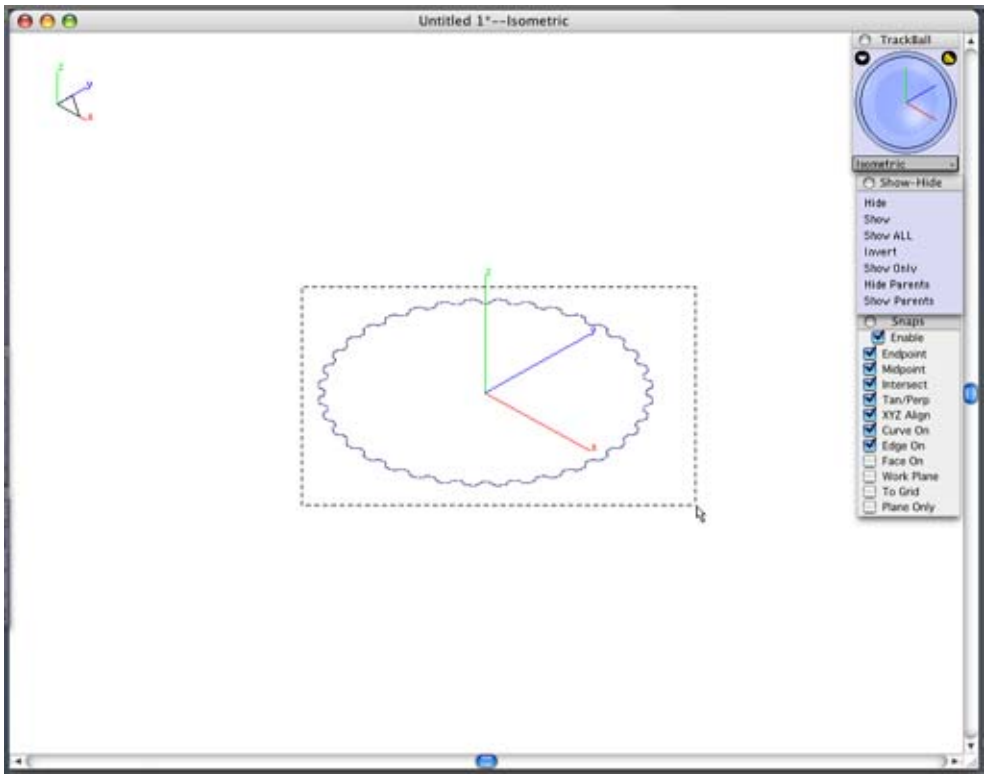
While holding shift select the two long lines and they will hide.

Go to the **Isometric view**. You can select this view from the drop down list at the bottom of the Trackball. There is also a shortcut for this view and it is the letter **f**.

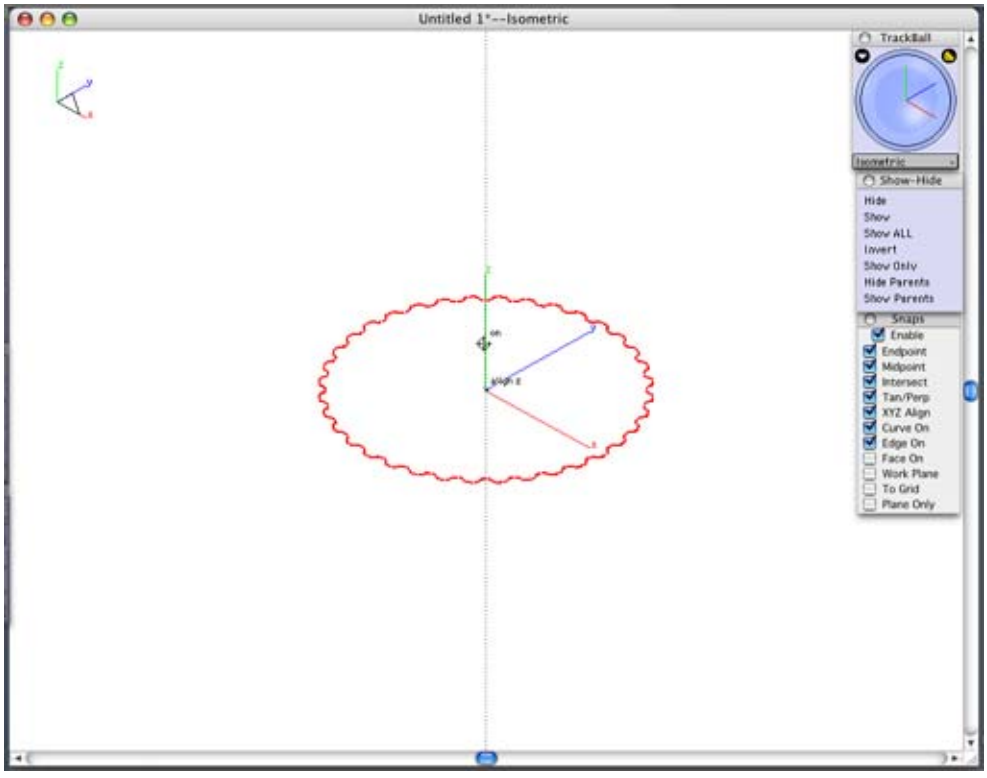


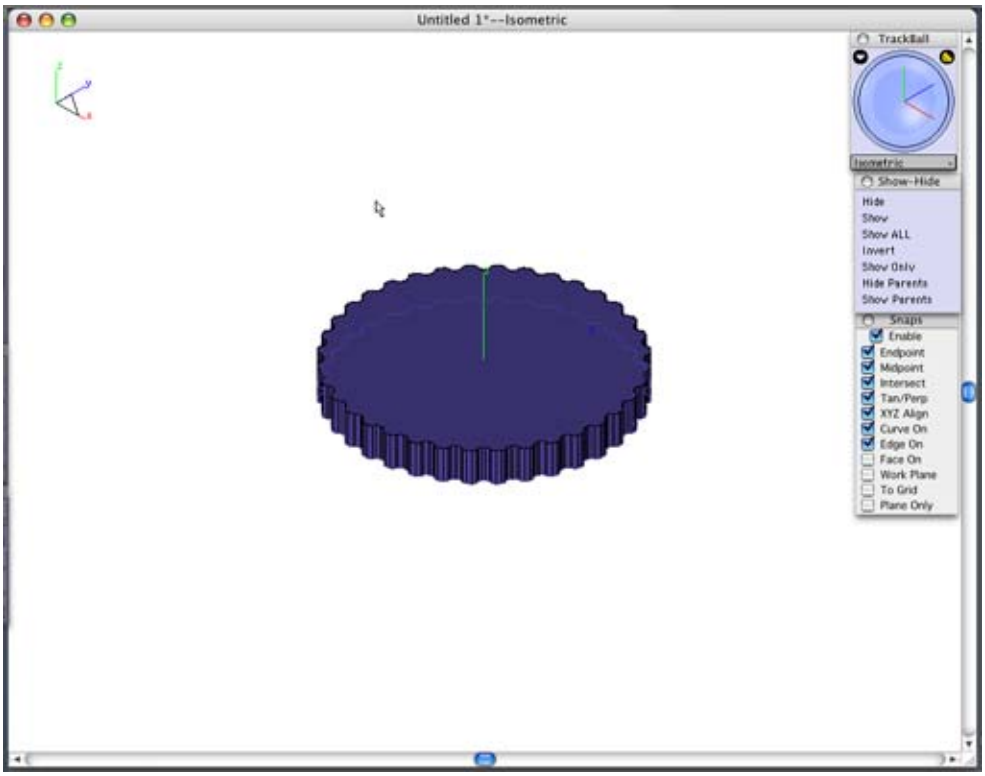
Select the Extrude tool from the Solids Menu.

Select the whole gear profile to extrude. Draw a selection fence around the whole drawing as shown below.



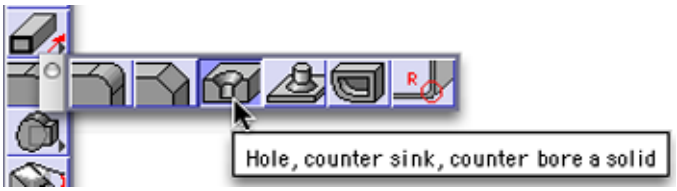
Now it needs a direction to extrude the profile. Select the **origin** for the first point and straight up along the **Z-axis** for the second.





Type in **.75** for the Distance in the Status Line and press **Enter**.

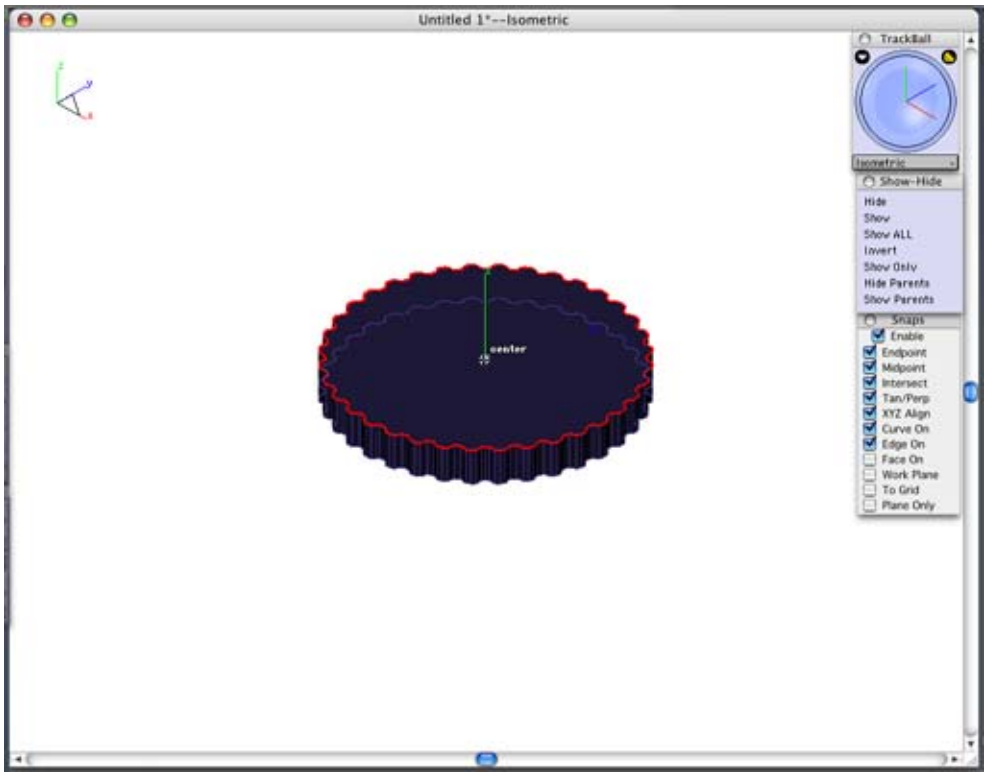
Next, we need a hole in the center of the part. Select the **Hole Tool**.



Set the options in the message line as shown below. We are making a **Simple hole** all the way **through** the gear and **normal** to the face of the solid.

Pick the top of the gear for the face to put the hole in. It will give you a choice of two faces to pick from (i.e. the top or the bottom face). It can't tell if you want to put the hole on the top face of the gear or the bottom face of the gear. Since our hole is going all the way through the gear, it doesn't matter which face we pick. If it did matter, you may want to try picking the face from a different view.

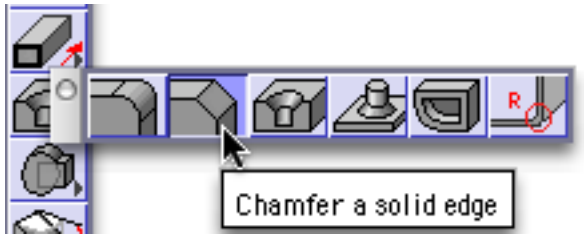
Then it will ask where to put the hole. We want it in the center, so select the center.



Type in **1.75** inches for the Diameter as shown below and press **Enter**.

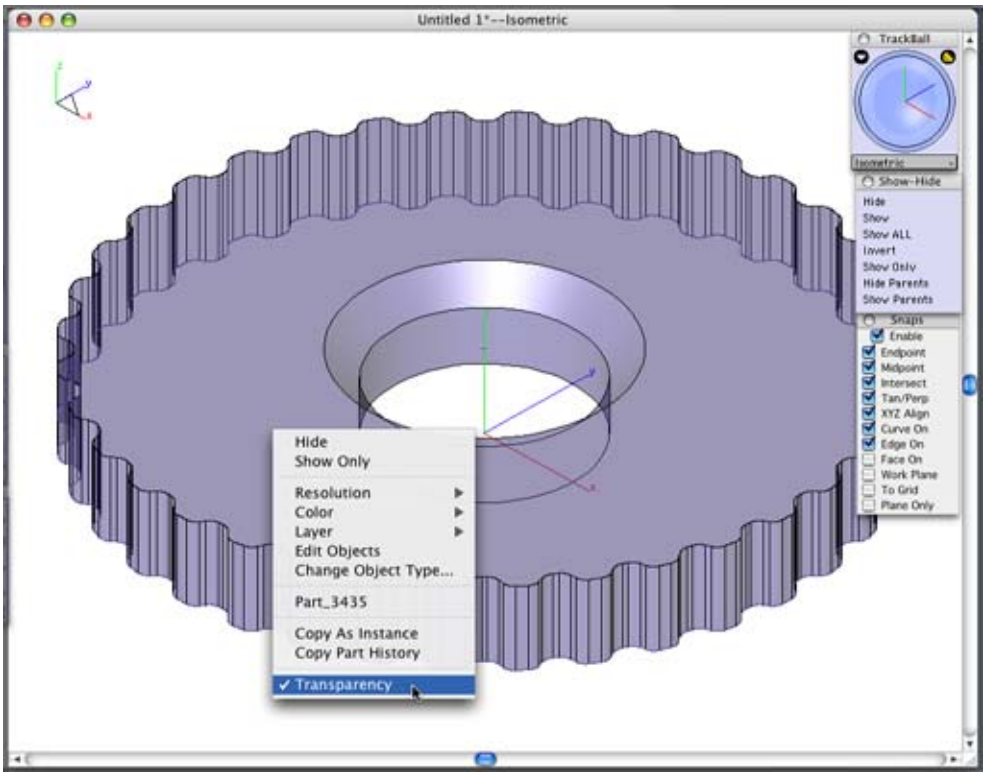
We are now going to put a chamfer on the edges of the holes.

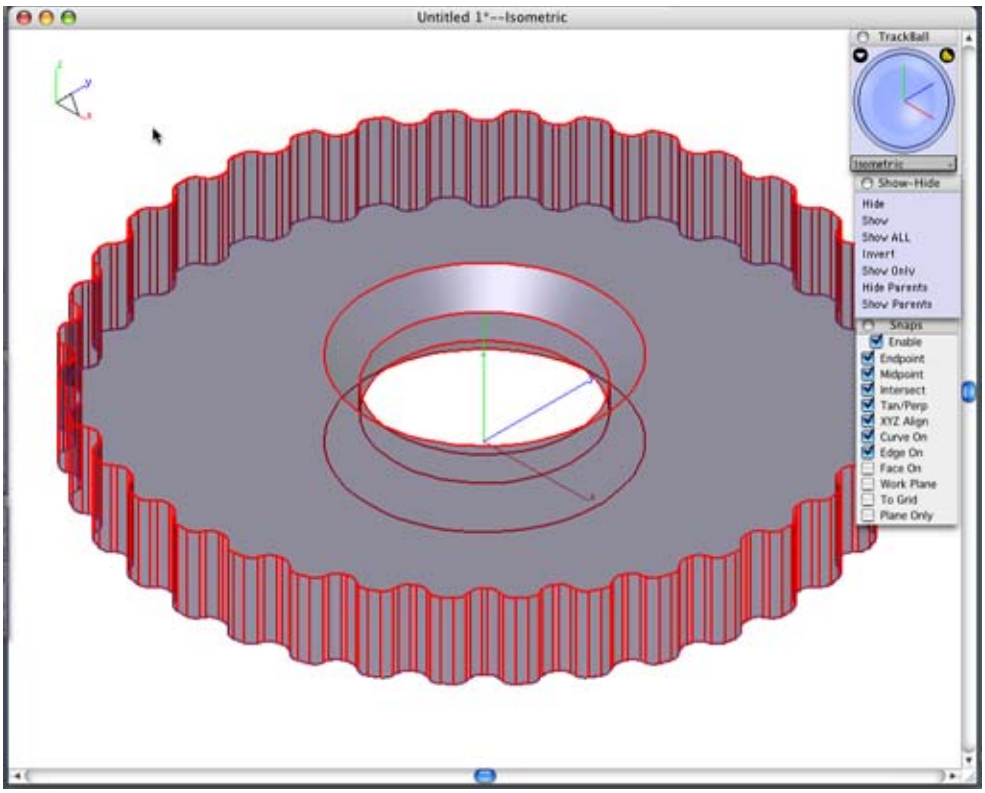
Select the **Chamfer** a solid edge tool.



Set the Length to **.25** inches and press **Enter**.

Select the top and bottom edges of the hole. After you create the top chamfer you may need to turn on transparency to see the bottom edge of the hole. Right Click on the part and select Transparency.





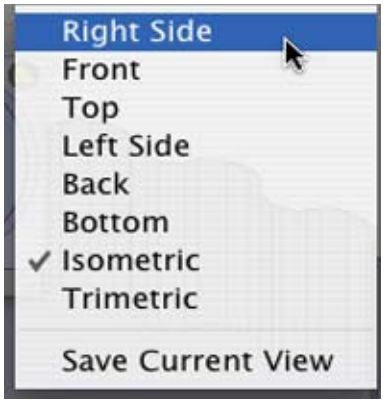
The next step of our process is to put some finishing details on the model.

Save your file.

Trimming the Details

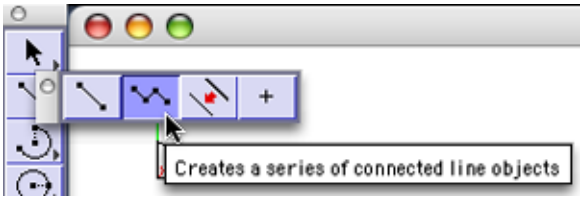
This last section puts a little detail into the gear to make it more realistic. If this part were really being made, we could say these details reduced weight or mass, or reduced the required material to make the part.

To add some detail to the model, we will create some surfaces to trim out the gear.



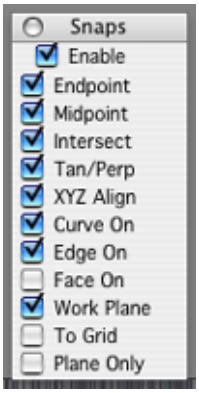
Change your view to the right side of the model by selecting it from the drop down list at the bottom of the Trackball, or by tapping the **a** key on your keyboard.

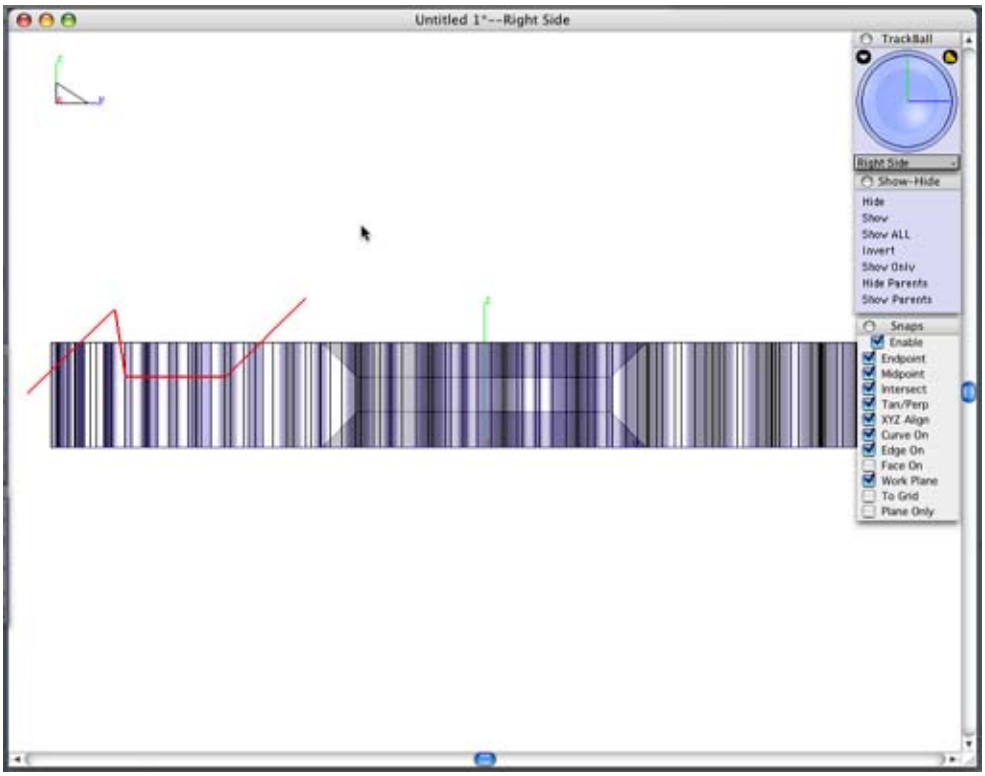
Select the **Connected Line** tool.



Turn ***ON*** the **Work Plane** Snap.

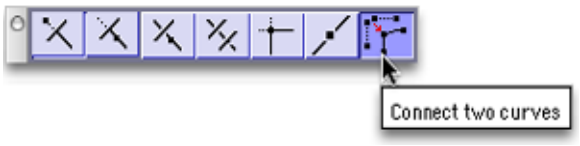
Create four connected lines as shown below in red. Make sure that the bottom red line does not go below half of the gear. **Double-click** on the last point to end the connected line.





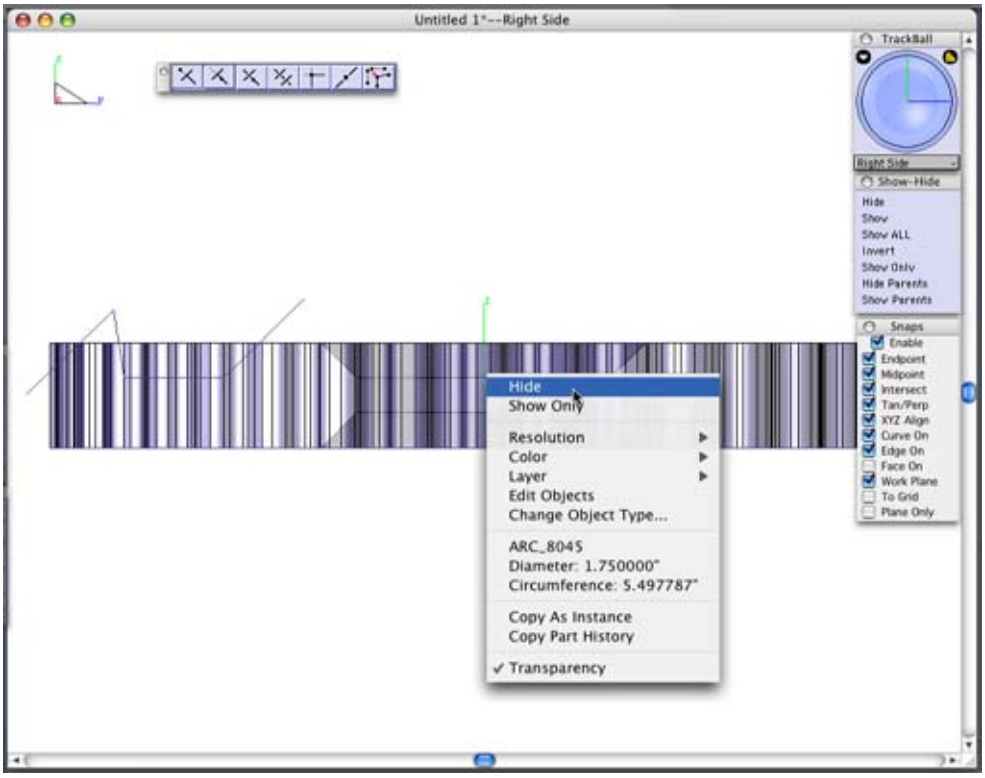
The lines we created are connected at their endpoints, but they are not one piece of geometry.When we create our surface from this piece of geometry, if the pieces are not joined, four separate surfaces will be created (one for each line). The Connect two curves tool has an option to join lines so that we can avoid this situation.

Select the **Connect two curves** tool.

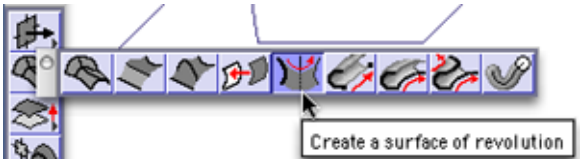


While holding **CTRL** on Windows or **Option** on a Mac select two of the lines that are touching. This joins those two into one piece of geometry. Repeat this to join all four lines.

Hide the solid part. Right-Click on the part, choose **Hide** from the pop-up menu.

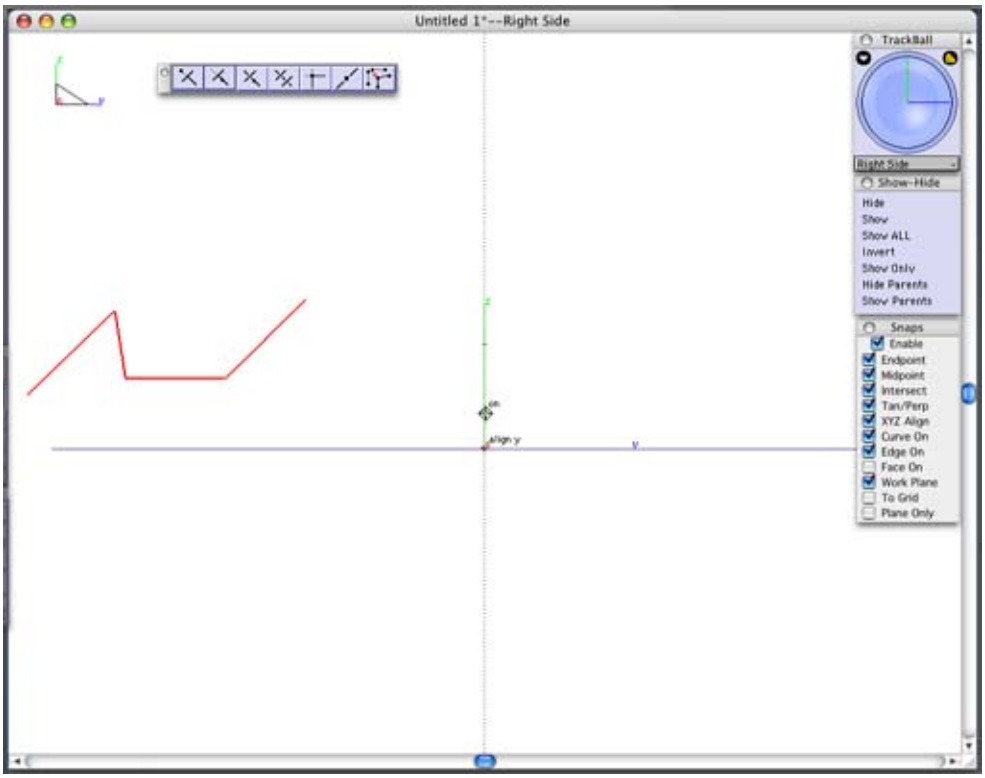


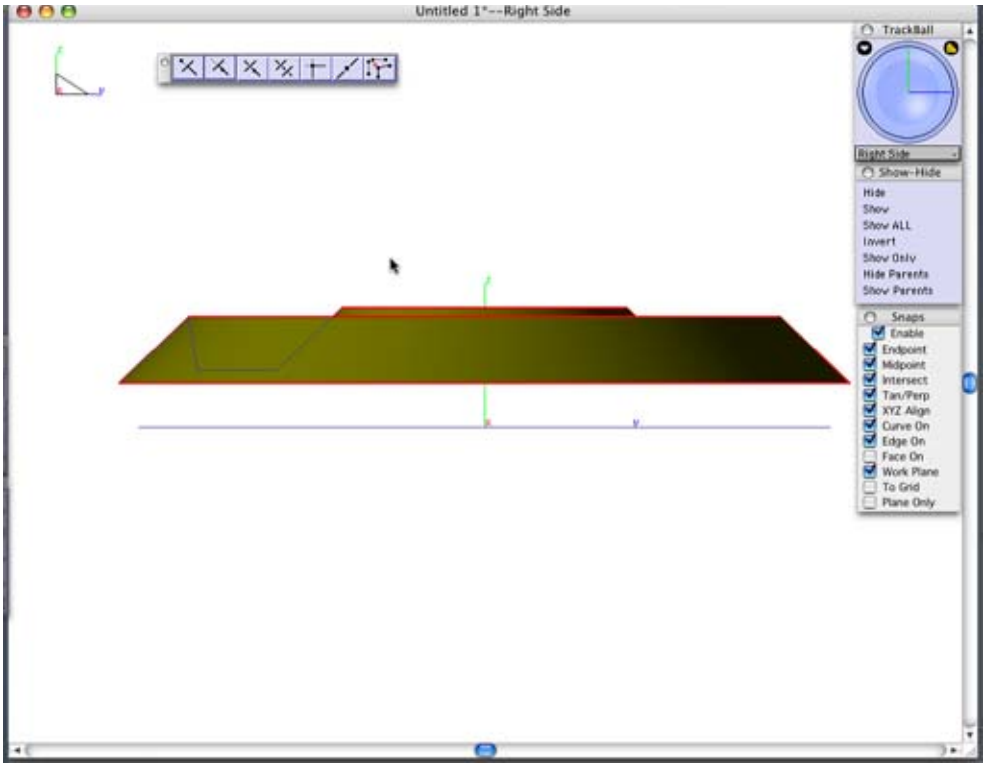
Select the **Revolved Surface Tool** from the surfaces tool palette.



Choose the first option in the Message Line, “Revolve around a specified axis.”

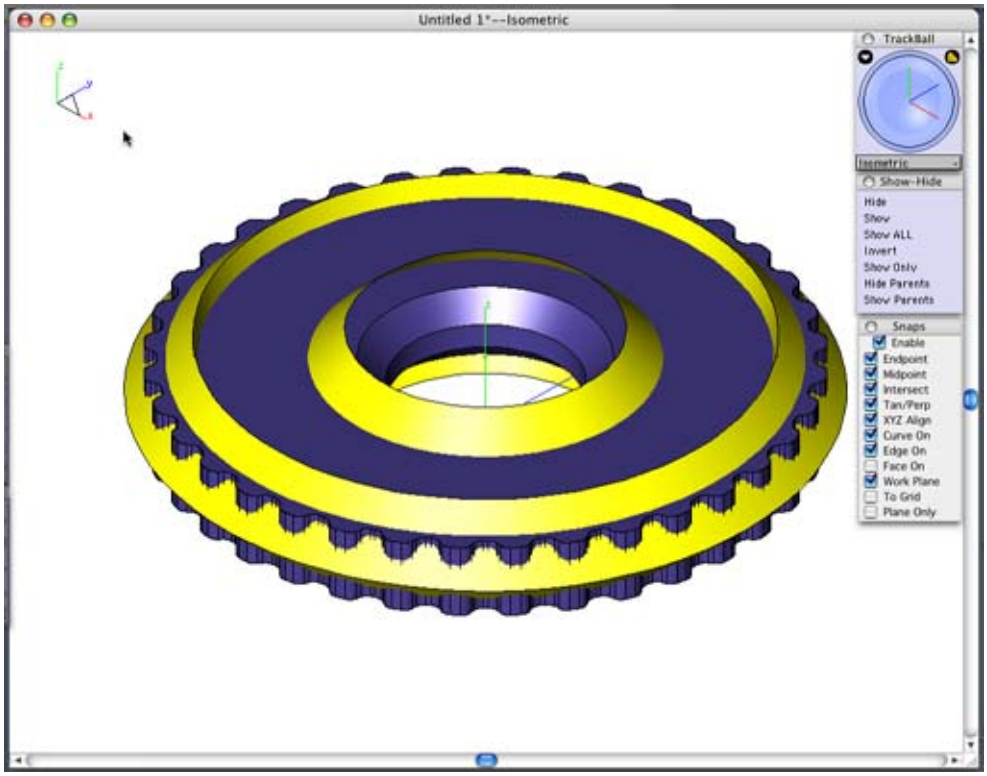
For the axis of revolution select the first point at the **origin** and the other point straight along the **Z-axis**.





Select the **Mirror Tool** from the Tools Palette.

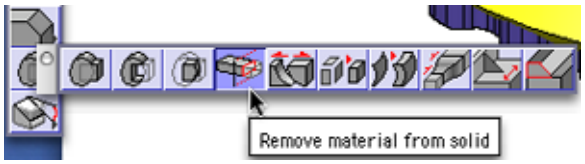
Select the surface you just created, hold down the **CTRL** key on a PC or the **Option** key on a Mac mirror a copy of the surface about the **midpoint** of the gear.



Now we need to trim the part outside of the surface off the gear.

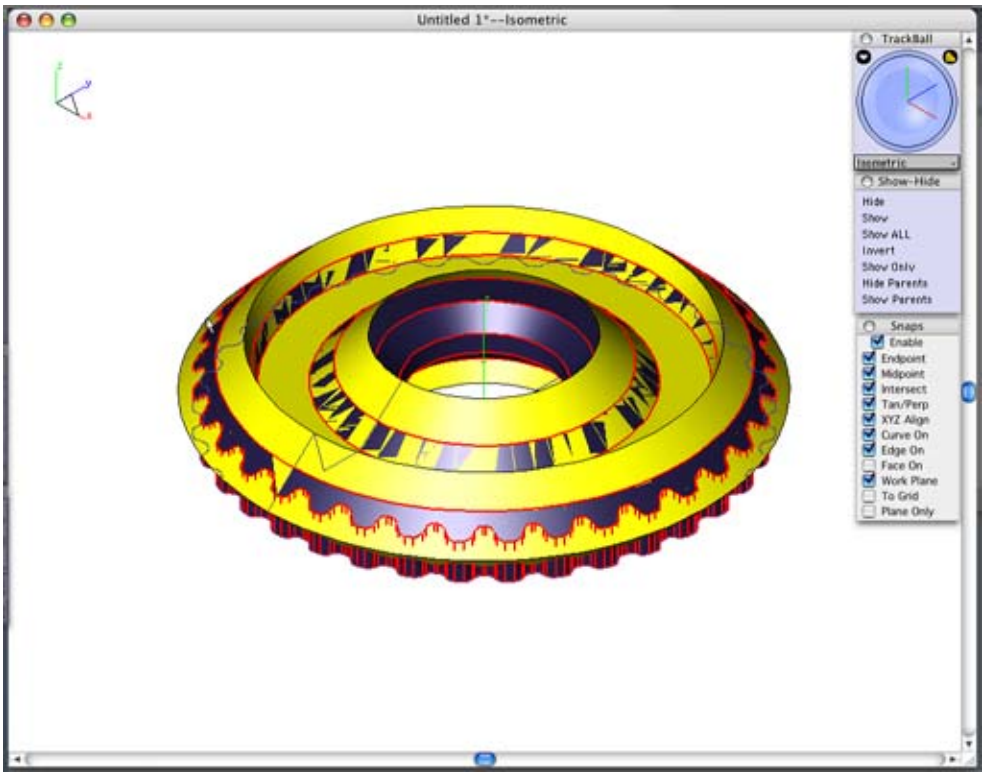
Switch to the Isometric view, tap the **f** key on your keyboard.

Select the **Trim Solid tool** from the solids tool palette.



Select the top surface that we created. Then select the solid.

We want to trim the little part off the top so the big part should be remaining, if it trimmed the opposite part you can hit the **CTRL** key on Windows or **Option** on a Mac to flip the direction of the trim.

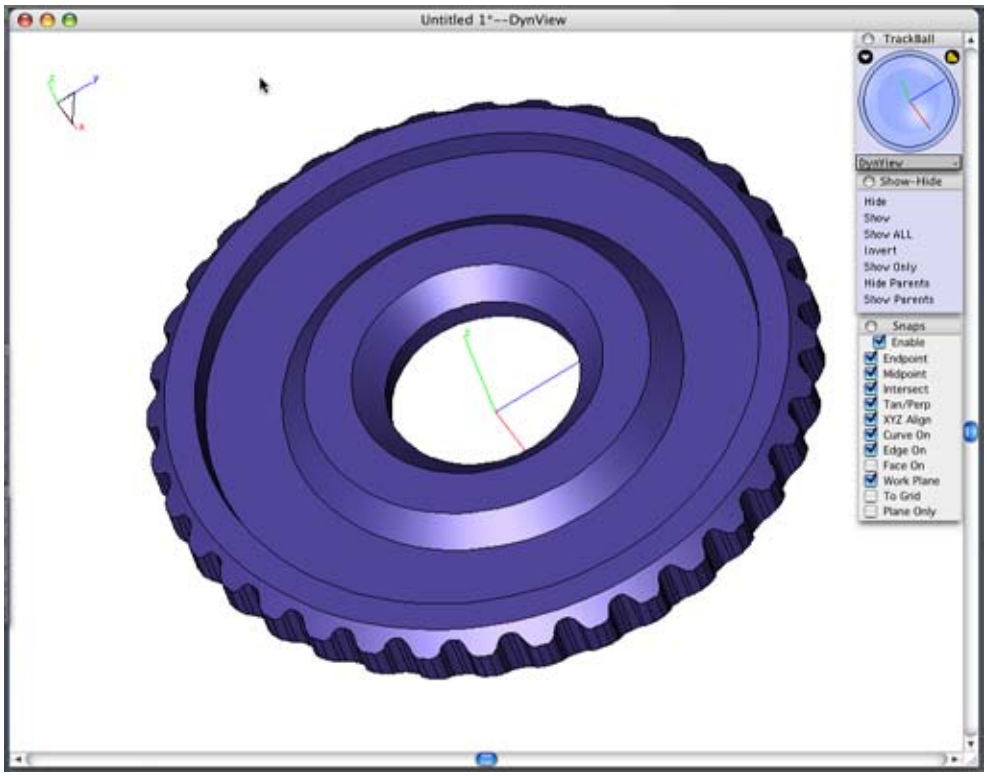


Repeat this for the bottom surface.

Select the **Select Tool** from the Tools palette.

Select the gear and select **Show Only** from the Show-Hide palette.

That completes our gear. Use the Trackball to rotate the view so that you can see what it all looks like when you are finished. I have changed the color of mine by selecting it with the Select Tool and choosing ***Pen>Color*** to change the color of it.



Save your file.

This completes this tutorial.