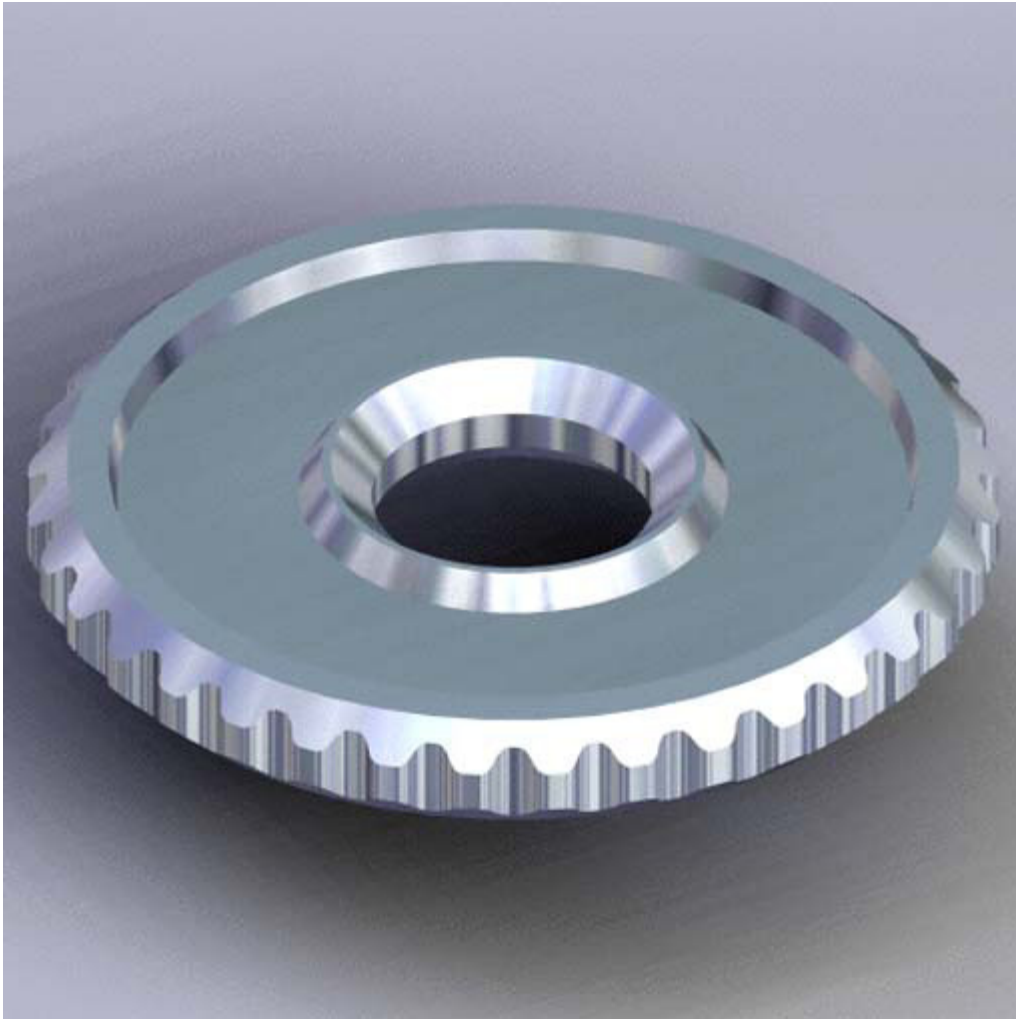# **Modeling a Gear Tutorial**

Using Cobalt™, Xenon™, Argon™

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 on the next page. The **Standard, Surface** 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.
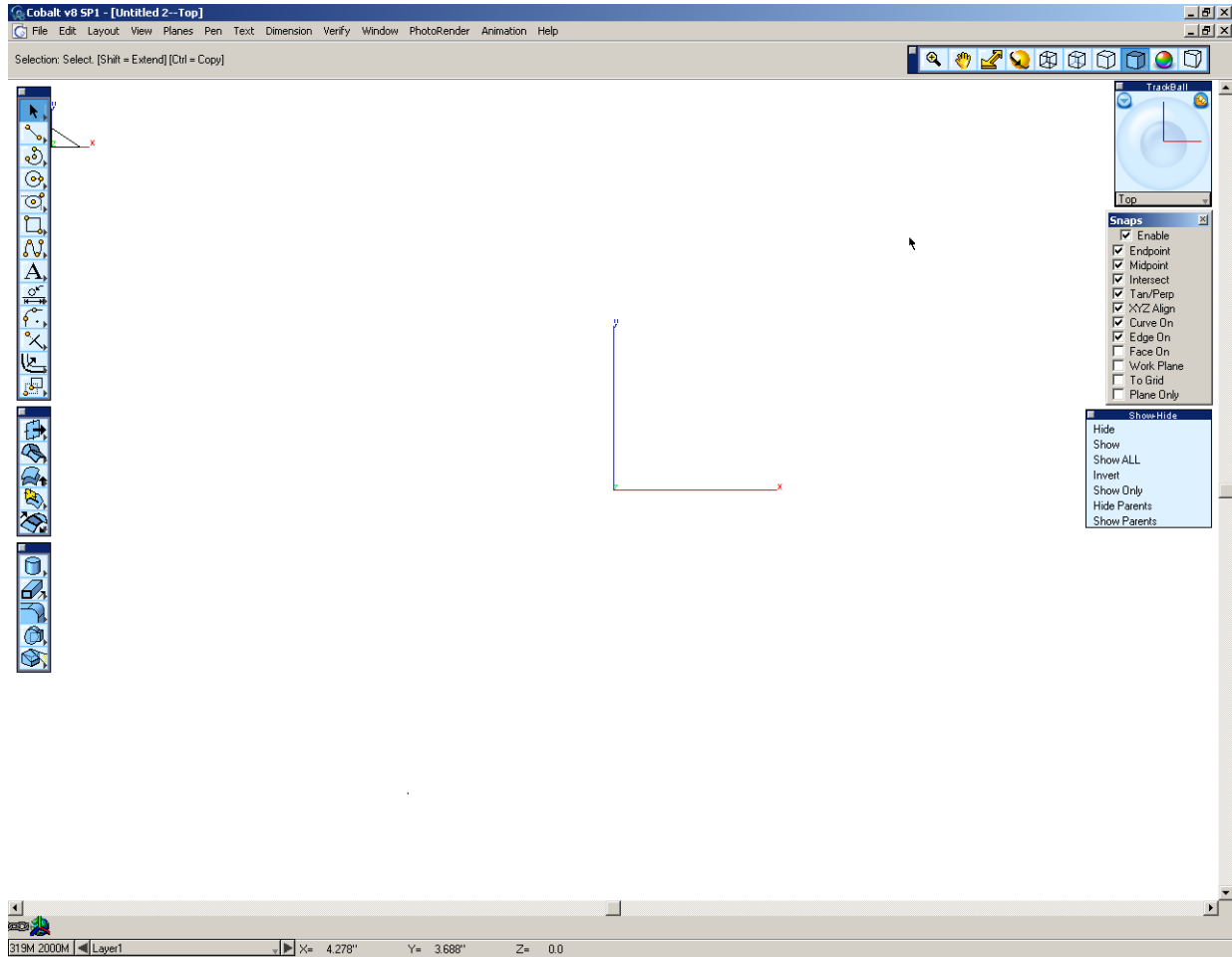
1. The first step in creating the gear is to create the tooth profile. The gear is going to have 36 teeth, though only one of them will be created.

   • Select the **Single Line** tool from the **Tools** palette.

   • Create the guide lines to build the tooth. Place the first point of the line at the origin as shown below.
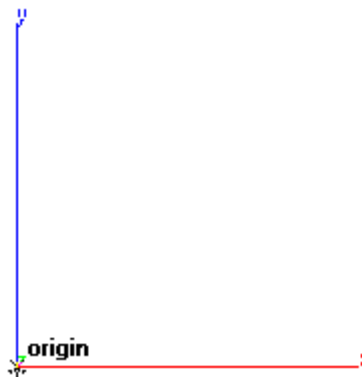
- Place the next line straight up along the Y-axis. How far isn't critical because the length and angle can be typed after creating.

- After placing the last point, type **3** into the **L** (Length) field and **95** into the **A** (Angle) field. This gives necessary line size.
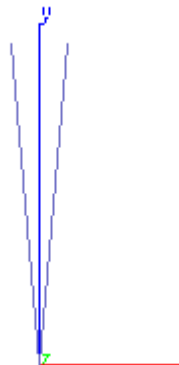

- Create a second line just like the first except this time after creating it put 85 into the Angle field instead of 95.

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.

*See below what the lines should look like after completing these steps.*

2. Next create an arc along these two points.

- Select the **Center-Point Arc** tool from the **Tools** palette. The Message Line prompts to pick the center point. Pick the Origin for the center point.

- The other two points are the points along the arc. Select the two endpoints of the lines for those points. The arc created should look like the image on the left. Press CTRL+F (Windows) or COMMAND+F (Mac) to fit the model to the screen (this is the shortcut for Zoom All). Or go to **View>Zoom All**.

    The outer section of the tooth is created. To create the inner section, it is not necessary to go through creating all of those lines again, just offset the arc we already made a little bit to create it.
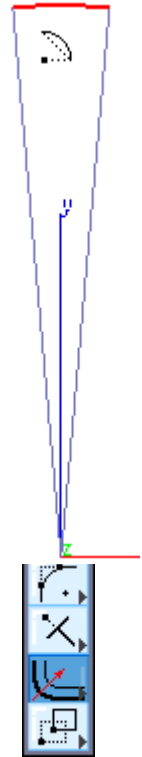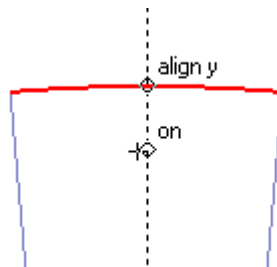
3. Select the **Offset Curve** tool from the Tools Palette. Make sure that the option **Offset to Value** is selected in the Message Line.

    Offset Curve: Pick curves to offset distance. [Ctrl = Extend Corner] [Shift = Extend]

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

     Offset .125

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

     align y

     on

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

4. Now some construction lines will help to build more of the tooth. They are not structurally necessary but are often helpful.

   - While holding SHIFT+CTRL (Windows) or COMMAND (Mac), click the origin of the model and drag the mouse straight up. A red dotted line should appear on the screen. This is a very short and quick way to create construction or helper lines to create the geometry.

- It is necessary to have a couple of them next to each other to create the rest of the tooth.

5. Select the **Parallel Line** tool..

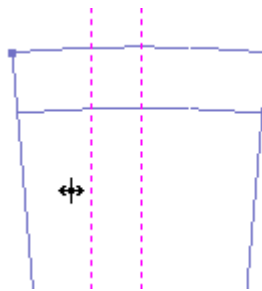Drags a parallel line off an existing line
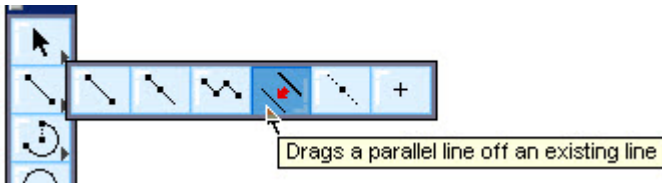
- Select the construction line created and drag it to the left. After dragging, type **.1** into the Offset field. This sets the offset to exactly .1 inch.


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

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

- Next, do the same thing for the right side. Drag a new line off of the center construction line or just enter a negative value in the status line (-.10), and press the ENTER key and a new construction line is created.
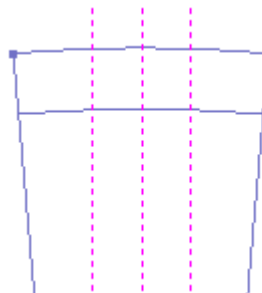

Create one parallel line off of the original. The rest of the lines can be created by entering different + & - offset values into the Status Line data box.

- Do the same thing to the original one more time to both sides, but this time type in **.180** and **-.180** for the offset amount.
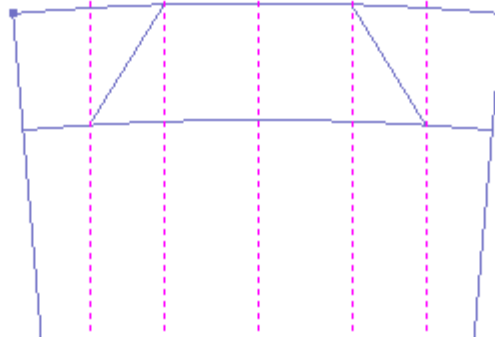
6. Now with the construction lines, finish this tooth.

    • Zoom in on the upper area of the model to make it a little easier to create new geometry.
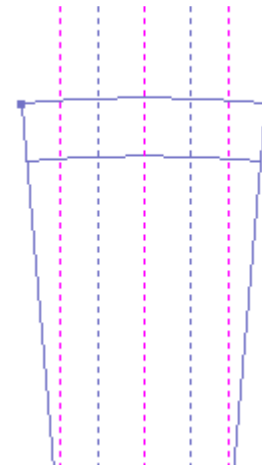
    • Select the **Single Line** tool.

    • Create a line as shown below. Use the snaps to create the lines at the intersections of the arcs and the construction lines.
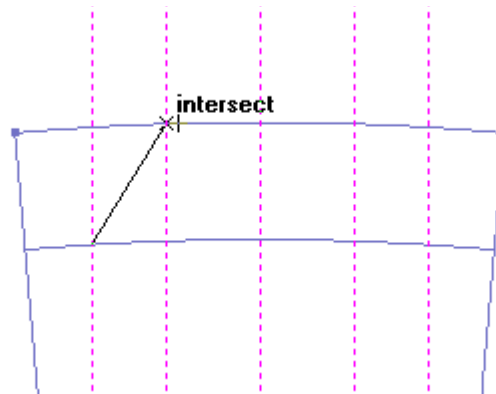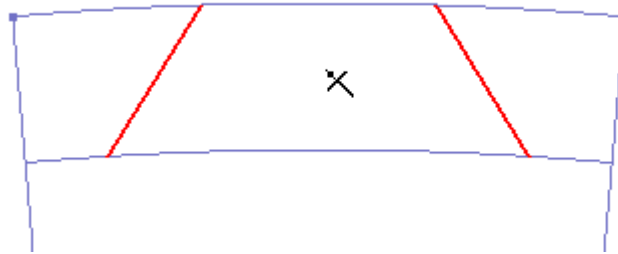
    • Create another line on the opposite side.

   • The construction lines are no longer needed, so hide or delete them.

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

- While holding the SHIFT key select both of the new lines just created.

- 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. The tooth should be similar to the one below.
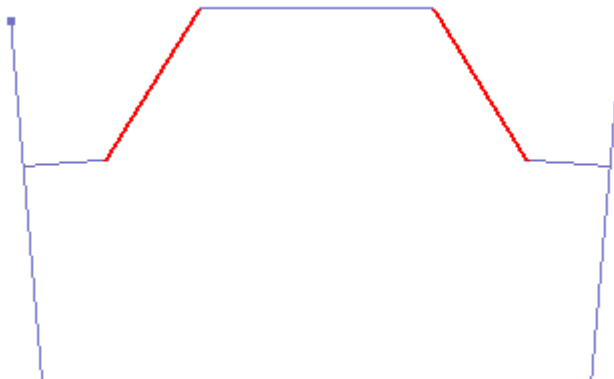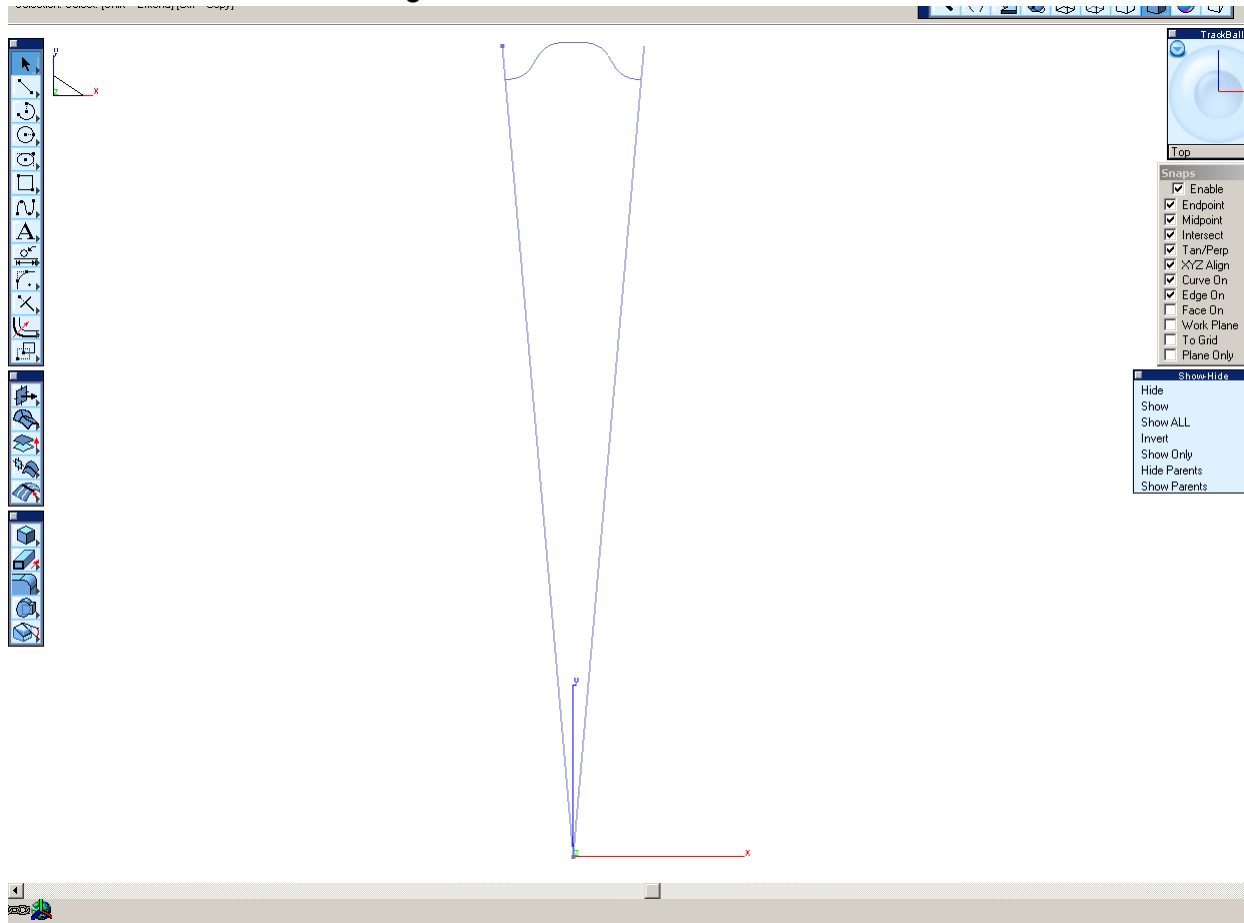

8. Select the **2-Entity Fillet** tool from the **Tools** palette.

- Type **.1** for the radius.
- Fillet all four of the sharp corners of the tooth. Select two lines at a time to fillet them.

Once all of the fillets are completed, the tooth should be rounded out and look like the image below.

After finishing the tooth, create the solid gear.

Save the file on the computer.

Creating the Gear

9. Copy the tooth created around in a circle to create a 2D gear shape. Then extrude that shape into a solid gear.

   • Select the **Polar Duplicate** tool.

   Copy object(s) around circle

   • Select the created tooth. A simple way to do this is to drag a selection fence around.

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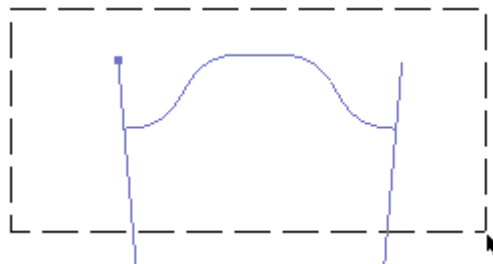

   • Do this, let off the SHIFT key and the dialog box will appear. These are the settings for the **Polar Duplicate**.

     Since the tooth is 10 degrees and it is necessary to create a full circle, there will be 36 teeth. The only other option is the center of revolution which should be default to the location of the last polar duplicate that you performed. If not, either type 0 into all of the fields, or select the origin in the model and the program will fill in the data from where it clicked.
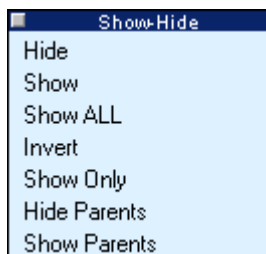
   • Click the OK button.
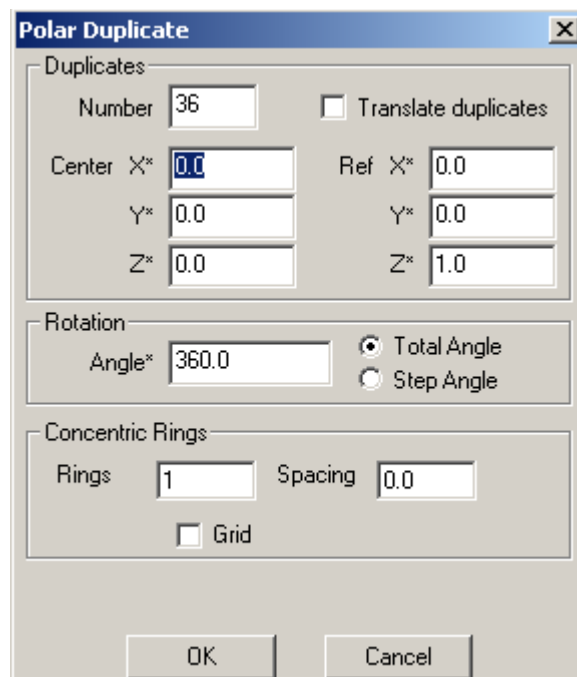
10. There it is. Now use this profile to create a solid.
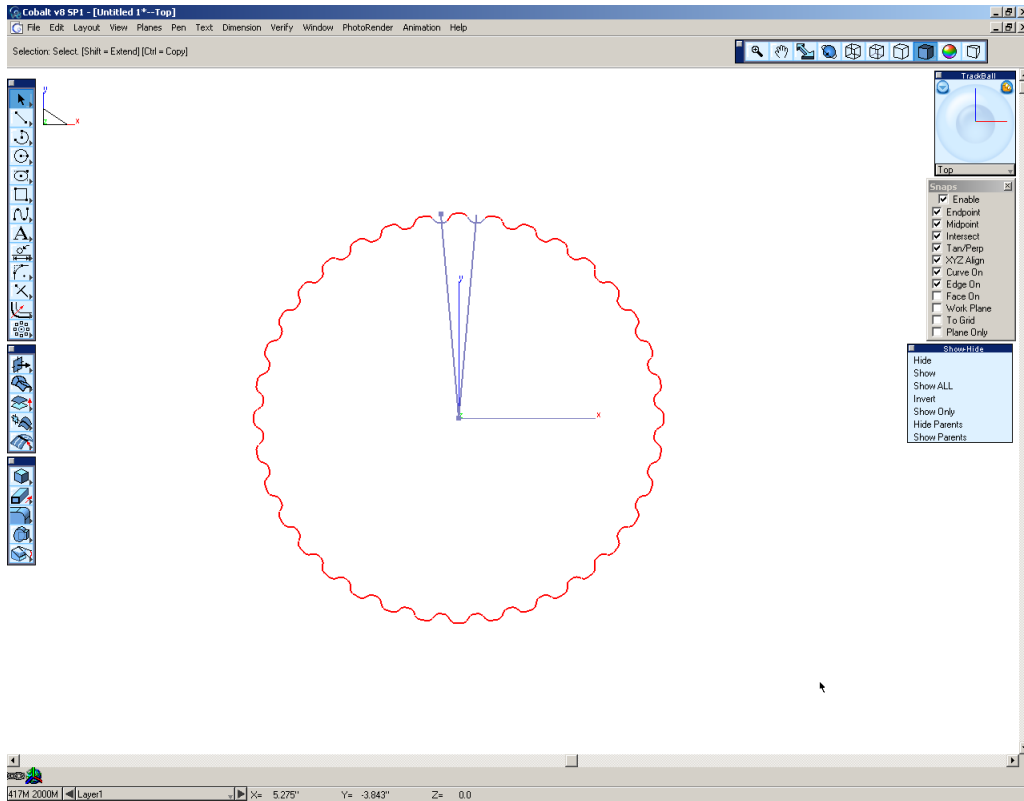
- Use the **Select Tool** and pick a point in the drawing area where there is no geometry. This will deselect everything.

- The two lines created at the start of this tutorial are no longer necessary and can be hidden or deleted.
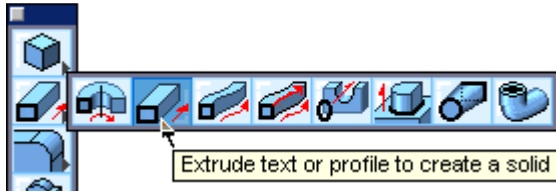
- Select **Hide** from the **Show-Hide** palette.

While holding SHIFT, select the two long lines to 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.
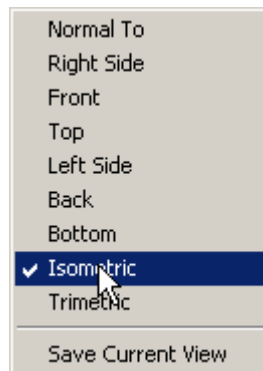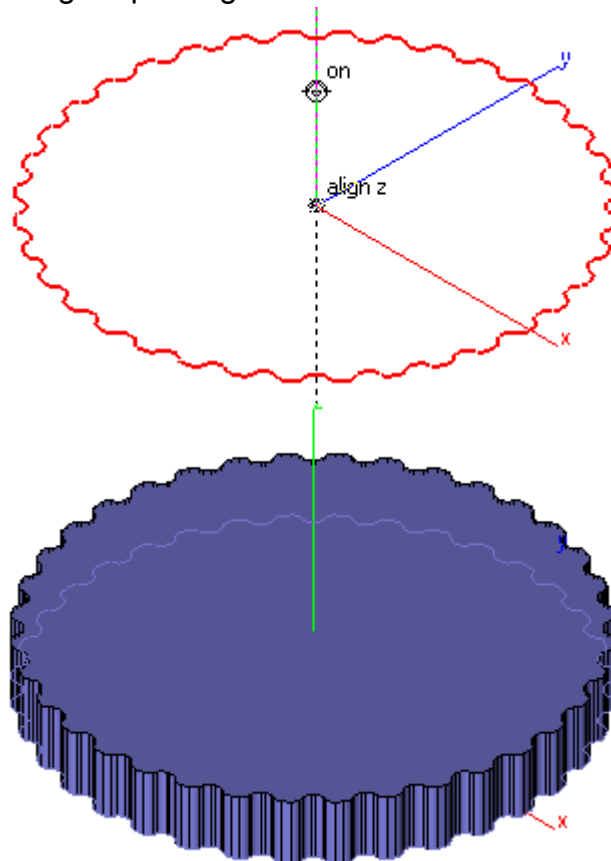
11. Select the **Extrude** tool from the **Solids** palette.

Extrude text or profile to create a solid

- Select the whole gear profile to extrude. Draw a selection fence around the whole drawing as shown below.
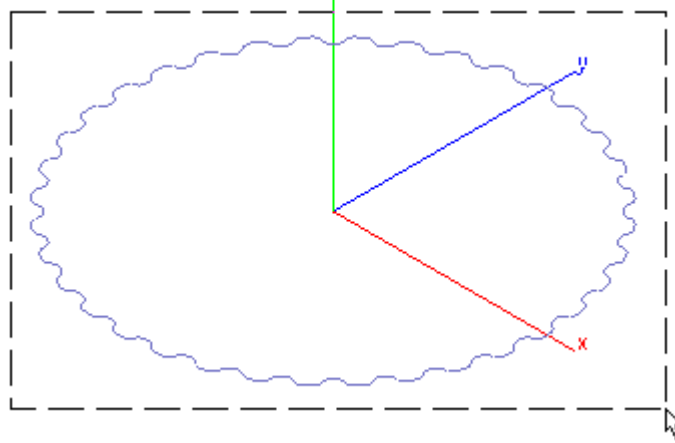
- Now it needs a direction to extrude the profile. Select the origin for the first point and straight up along the *z-axis* for the second.

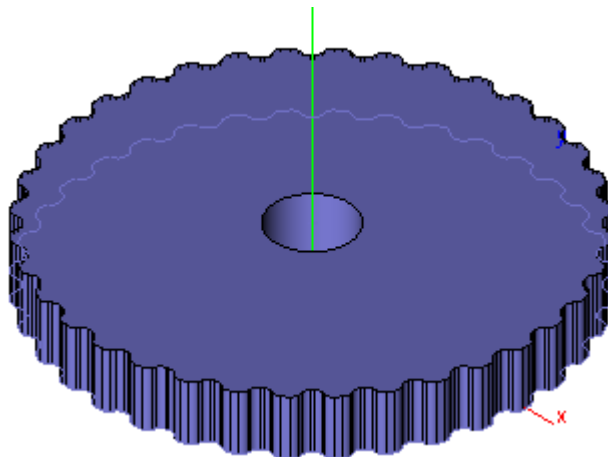- Type **.75** for the Distance in the Status Line and press ENTER.

⊡⊙ 🔧 dX 0.0	dY 0.0	dZ 0.750	Distance 0.750	Draft Angle 0°
418M 2000M ◀ Layer1		▶ X= 1.457"	Y= -12.503"	Z=

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

Hole, counter sink, counter bore a solid

- 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.

⊔ ⊔ ⊔	To Depth ▾	Through ▾	Normal ▾	Project on Face ▾
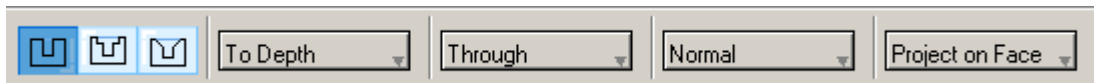
- Pick the top of the gear for the face in which to put the hole in. It can't tell if you put the hole on the top face of the gear or the bottom face of the gear. Since the hole is going all the way through the gear, it doesn't matter. If it did matter, try picking the face from a different view.

- The Message Line asks where to put the center of the hole. Select the center.

- Type **1.75** inches for the Diameter as shown below and press ENTER.

⊡⊙ 🔧 X 0.0	Y -0.0	Z 0.750	Depth 2.250	Diameter 1.75	Draft Angle 0°
430M 2000M ◀ Layer1		▶ X= 6.677"	Y= -11.320"	Z= 0.0	

13. Put a chamfer on the edges of the holes.
- Select the **Chamfer** tool.
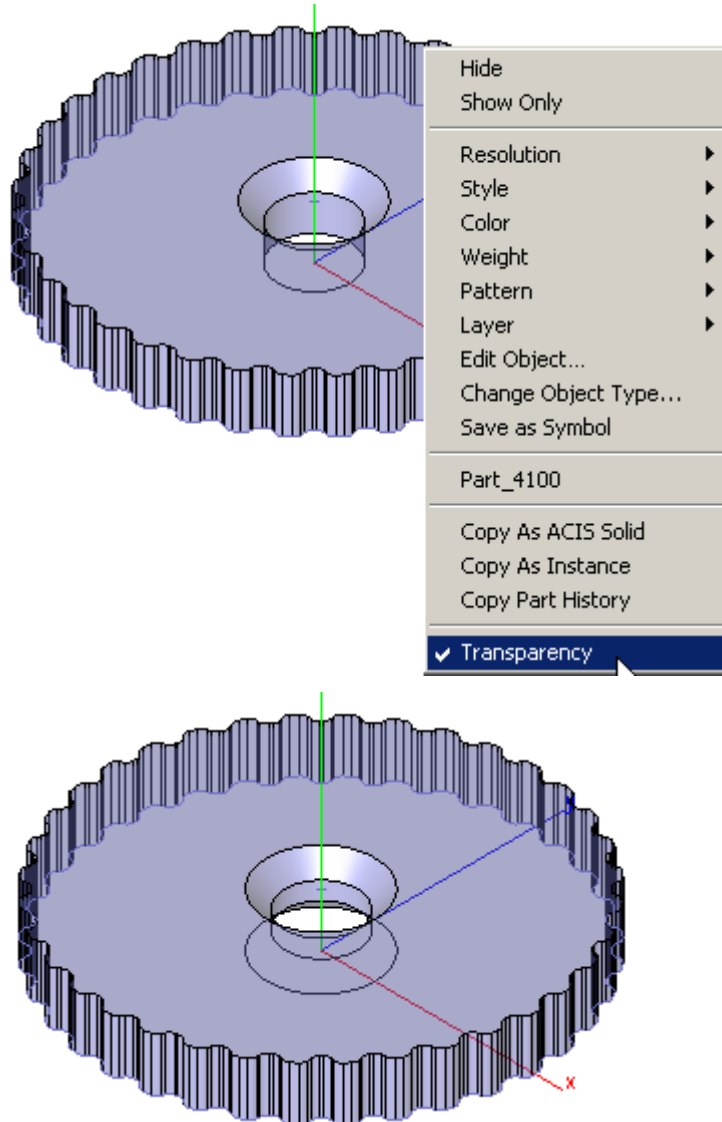
Chamfer a solid edge

- Set the Length to **.25** inches and press ENTER.

- Select the top and bottom edges of the hole. After creating the top chamfer, turn on transparency to see the bottom edge of the hole. Right click on the part and select Transparency.


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

Save the 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.

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

   • Change the 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 the keyboard.

   • Select the **Connected Line** tool.
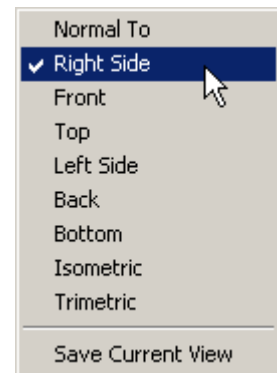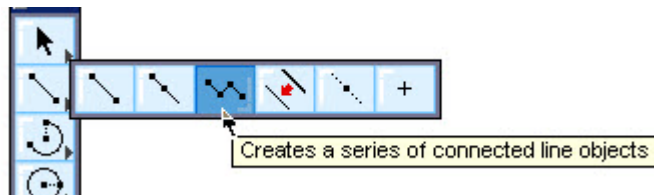
   • Turn ON the **Work Plane** snap.

   • 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.
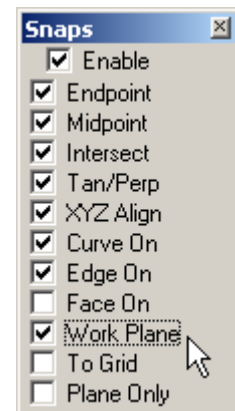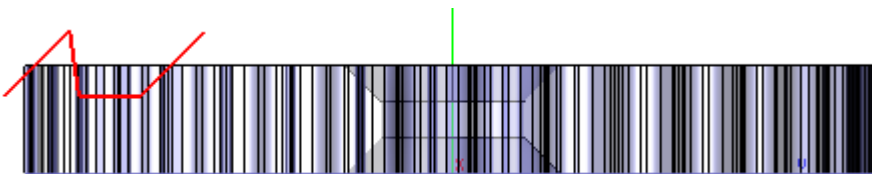
   *The lines created are connected at their endpoints, but they are not one piece of geometry. When creating the surface from this piece of geometry, if the pieces are not joined, four separate surfaces will be created (one for each line). The **Connect Curve** tool has an option to join lines to avoid this situation.*
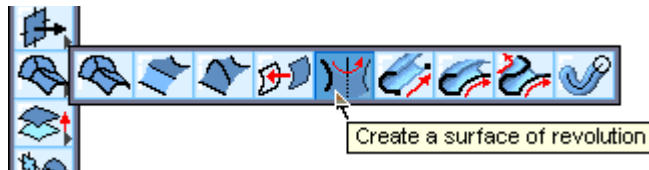
15. Select the **Connect Curve** tool.
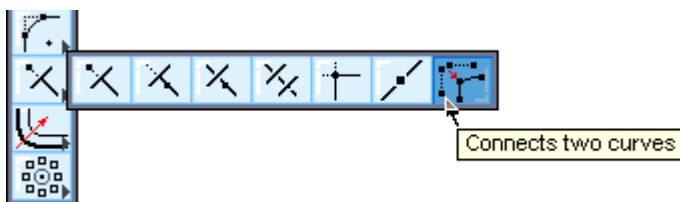
   • While holding the CTRL (Windows) or OPTION (Mac) key select two of the lines that are touching. This joins those two into one piece of geometry. Repeat this to join all four lines.
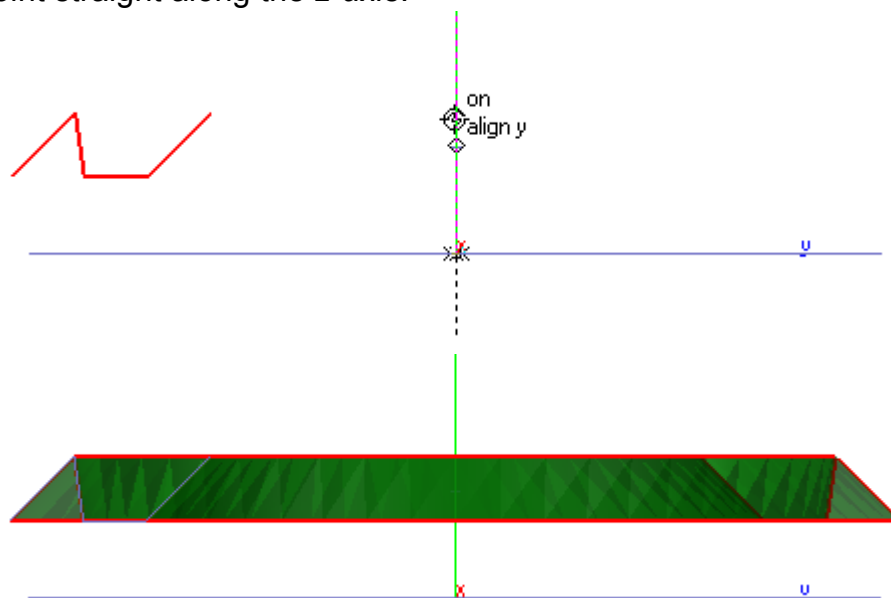
16. Hide the solid part. Right-click on the part, choose Hide from the pop-up menu.
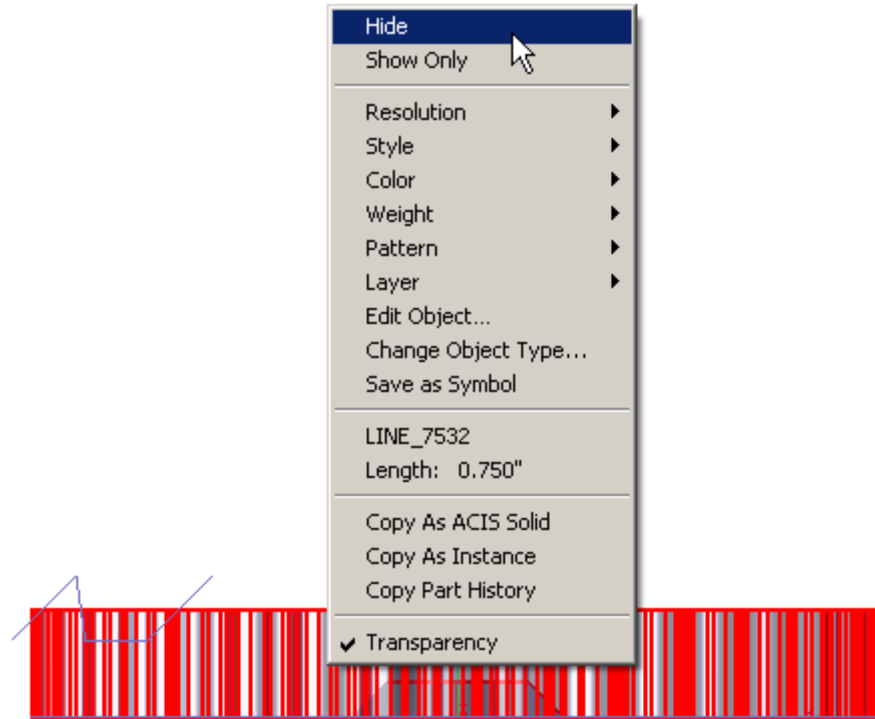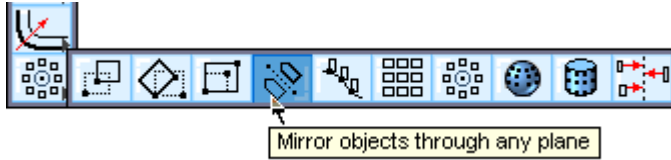

17. Select the **Revolved Surface** tool from the **Surfaces** tool palette.

- Choose the first option **About Two Points** in the Message Line.

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

18. Select the **Mirror** tool from the **Tools** palette.

Mirror objects through any plane

- Select the surface just created, hold down the CTRL key (Windows) or the OPTION key (Mac), mirror a copy of the surface about the midpoint of the gear.
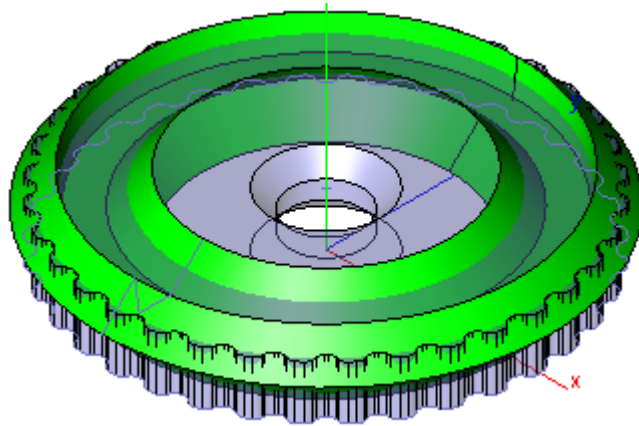

19. Now, trim the part outside of the surface off the gear.

- To switch to the Isometric view, tap the F key on the keyboard.

- Select the **Trim Solid** tool from the **Solids** tool palette.


Remove material from solid

- Select the top surface created. Then select the solid.

- Trim the little part off the top so the big part is remaining. If it trimmed the opposite part, press the CTRL key (Windows) or OPTION (Mac) to flip the direction of the trim.
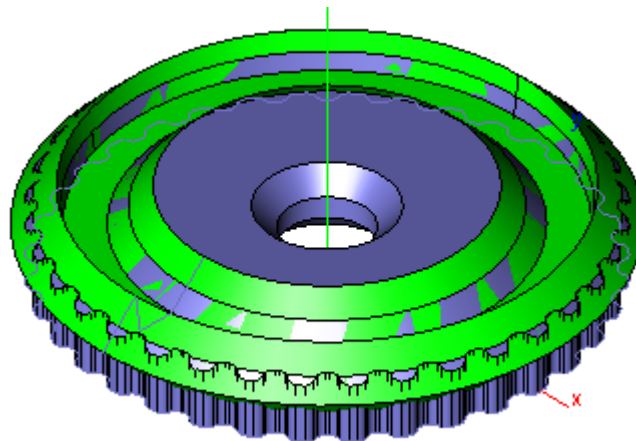

- Repeat this for the bottom surface.

20. Choose the **Select** tool from the **Tools** palette.

    • Select the gear and choose **Show Only** from the **Show-Hide** palette.
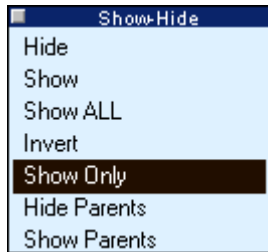
That completes the gear. Use the Trackball to rotate the view to see what it all looks like when it is finished. Change the color by selecting it with the **Select** tool and choosing *Pen>Color* or right-click, choose All Selected and choose Color from the Context menu.
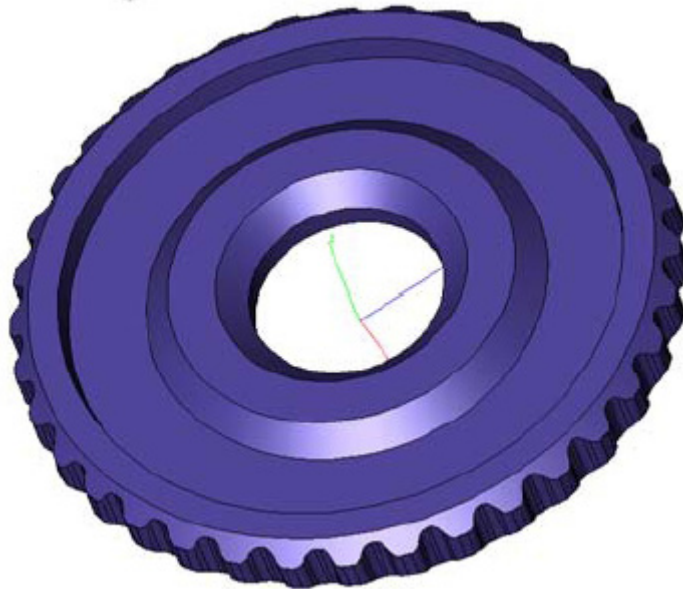
Save the file.

This completes this tutorial.