

Creating an Involute Gear On Creo Parametric 3.0

1. Use the following Formulas to calculate the dimensions of the four circles needed:
 - N is a given Value, PA(pitch angle) is a given value, and PD (Pitch Diameter are given values)
 - $P = N/PD$; $a = 1/P$
 - $Db = PD\cos(PA)$
 - $OD = PD + 2a$
 - $DR = PD - 2b$, where $b = ht - a$, where $ht = 2.157/P$
2. Once you have these values, use the extrude tool and choose the xy plane. Create a circle of diameter OD. Then choose an appropriate depth and click OK.
3. After this, use the sketch tool to create the dimensions of the remaining 3 circles, PD, Db, and DR.
4. Then click the drop down datum menu and choose “curve from equation.” Then choose the reference as the origin and then click the equation button on the top. Input the following:
 $r = Db/2$ (NOTE: this is not what you type in, you type in “r =” and then whatever the value of Db/2 is)
 $ang = t*90$
 $s = (PI*r*t)/2$
 $xc = r*\cos(ang)$
 $yc = r*\sin(ang)$
 $x = xc + (s*\sin(ang))$
 $y = yc - (s*\cos(ang))$
 $z = 0$
5. After you click ok, it will give you your involute curve. You should then click “point.” Place one point on the origin. Then click the point option again, hold ctrl, and then click both the involute curve and the 3rd largest circle next to the curve. Then let go of ctrl and it will give you your point.
6. Then click the datum dropdown menu and click “curve through points.” Click both points to create the line. (NOTE: once you choose the option, one point may already be highlighted, so only choose the one that isn’t, or it might not work)
7. At this point, you should choose “Hidden Line” option to make it easier to see what is being constructed.

8. Then choose the "Analysis" tab and choose the "Measure" dropdown to select "Angle." Hold control and click the straight line you just created and the horizontal plane directly below it. This will give you an angle measure, and you should then click the save option to the right and save it as "Measure."
9. Then select the involute curve, and choose "copy" then "Paste Special" and then check the second option. Then make sure you click the rotate button to the left of the green box. Then choose the dashed z-axis as a reference and then input the value given from $-2(360/N)$. Then click OK.
10. Then double click the copied curve, and then double click its dimension. Type in angle:FID_MEASURE + $(360/(4N))$. Then click Yes.
11. Then single click the new curve just generated and click the mirror option. Use the reference plane as the horizontal plane directly above it. Click the check mark.
12. Click the curve again that we generated in step 10 and use the copy-paste special option (make sure again that you check the second box). Click the rotate button again and make sure the reference is the z axis. Then rotate it by a value of $360/N$.
13. Then click extrude and click the face of the gear. Then choose the project tool. Choose the outermost circle, the inner most circle, the uppermost involute curve, and the only involute curve facing in a different direction.
14. You will quickly notice that it doesn't fully connect. Choose the line tool. You want to connect end of the upper most curve to the smallest circle at a tangent. Do not connect it until you see a "T" appear to signify that it is tangent. Use the line tool again to connect the other selected involute curve to the innermost circle at a tangent.
15. Then select the delete segment tool to delete all orange segments that are NOT part of the cut we are trying to make. The cut we are trying to make is the intersection of the 4 curves we selected, so cut out all other orange segments.
16. Then click the check mark and select the remove material option. Make sure it is directed in the correct direction so that actually does remove all of the material.
17. Make sure the extrude is still selected and choose the pattern option. Change the dropdown from "dimension" to "axis" and then choose the dashed z-axis as the reference. The first value should be N and the second should be $360/N$. Then click the checkmark.

18. You can then use the round tool. Hold control after selecting round and choose the bottom 2 edges perpendicular to the inner circle, and then follow all the way up to the two corresponding involute curves and choose the straight line at the top of each curve parallel to the two lines you already selected. (This is kind of confusing unfortunately). Choose a value for the round and click okay.
19. You can then immediately pattern the round and click the check mark. You can also use the extrude hole to create a circular hole of whatever diameter you choose in the middle of the gear. Your gear is now complete.

Creating the Frame for the Gear

1. Create two circles with the same diameter as your hole created in step 19. Separate them by a distance of $PD + .015PD$. Use the line tool to create two horizontal lines connecting the two circles tangentially. Use the delete segment tool to delete the inner portions of both circles so just the outer semicircles are left. What should be left should be similar to the shape of a rectangle with two semi circles at the end. Give this whatever depth you see fit.
2. Then use the extrude tool to set up two circles again with the same diameter and same separation distance. They should go on top of the part created in step 1 and should be given depth. The gears fit on to these round parts.