Parametric Design of a Spiral Gear Process

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Abstract

The objective of this project was to develop an automated process for modeling spiral bevel gears to reduce gear design time. As the popularity of five-axis CNC machine tools and multi-axis CAM software has increased, such tools are now being used to manufacture these types of gears in small size lots. However, an accurate 3D representation of the gears' defining geometry is not always readily available. The goal of this project was to create a system that will accurately define this geometry in CAD software. The outcome was a well-defined set of steps that can be used to accurately create gear models. The final step was to streamline this process by taking advantage of the features of the CAD software.

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1) Introduction

The spiral bevel gear has some very distinct advantages over other bevel gears, the most notable of which is its ability to handle greater loads and torque. The spiral-formed teeth allow multiple teeth to be in contact at one time. Distributing the load over multiple teeth reduces the stresses placed on each tooth. The most prevalent disadvantage to these gears is wear since the teeth continually change contact and slide along its mating teeth increasing the wear. The goal of this project is to assist CNC Software, inc. in the parametric design of this complex gear which currently does not exist.

Parametric gear designs are not readily available. We initiated the process with a simpler spur gear, then advanced to the straight bevel gear and finally defined the governing parametric equations for a spiral bevel gear. SolidWorks was our vehicle for modeling the 3D geometry. However, the process is applicable to multiple advanced CAD systems, such as Pro Engineer.

2) Background

The gear is one of the most important devices used in many types of machinery. Gears allow the user to translate power, motion and torque. Gears have a power transmission efficiency of up to 98% and are some of the most durable torque transmitting machine elements (Hamrock et al. 2005, pg. 607). The applications of gears are limitless and useful in many different settings. This chapter discusses the history, purpose and manufacture of gears.

3.1) History

The first primitive gears can be traced back to over 3000 years ago. They were made of wood and had teeth of engaged pins. Early Greeks used metal gears with wedge shaped teeth; Romans used gears in their mills; stone gears were used in Sweden in the Middle Ages (Gears Manufacturers). All of these cultures found reasons to use basic gearing to convert energy or motion in one form to a form they could use in devices for the technological advancement of their societies.

Gears were used by early engineers for lifting heavy loads by taking advantage of their force-multiplying properties. One example of this type of application was in ship anchor hoists and catapults. Gear technology made its biggest leaps during the industrial revolution in Great Britain during the eighteenth century (eFunda). As machines became more sophisticated throughout the years, gear technology and manufacturing also developed at a rapid pace. Gears became essential elements in countless devices, from clocks to complex machinery. Today, gears are used in many of the machines people depend on every day such as automobiles.

3.2) Purpose

The primary purpose of gearing is the manipulation of motion into a more potent or usable form. The various types of gears allow for endless possibilities in this manipulation. The three major classes of gears are parallel-axis gears, nonparallel-coplanar gears and nonparallel-noncoplanar gears (Hamrock et al. 2005, pg. 607-610).

3.3) Parallel-Axis Gears

The most basic types of gears are used to alter the amount of shaft rotation by meshing gears of different sizes. Parallel-axis gears can are highly efficient and can transfer large amounts of power (Hamrock et al. 2005, pg. 608). The simplest gears in this category are called spur gears because of their shape. Spur gears are advantageous because of their low cost and simple design. By altering the amount of rotation, more energy can be manifested from a process. An example of this is a windmill. One disadvantage of spur gears is that they can produce significant noise levels.

The spur gear is useful in many applications, but may not be ideal in situations that require very large torques because the tooth contact ratio is one-to-one. For higher torque applications, the helical gear can be used. This gear uses angled teeth to increase the contact ratio between the teeth of two meshed gears. Other advantages of helical gears are that they run more quietly than spur gears and that a smaller helical gear can transmit the same load as a larger spur gear (Hamrock et al. 2005, pg. 608). One disadvantage of helical gears is that they produce an additional end thrust along the axis of the shaft which much be compensated for. They also tend to be slightly less efficient than spur gears because efficiency is based on normal tooth load, which is higher in spur gears (Hamrock et al. 2005, pg. 608).

3.4) Nonparallel-Coplanar Gears

The other main group of gears is used to translate power and rotation in a different direction. The bevel gear is the most common type of the nonparallel-coplanar gears. The face of a bevel gear is angled so that the shafts of two meshed bevel gears can translate rotation in a different direction. These gears can also be used in different gear ratios, so that the direction of rotation and amount of power can be altered in one step. This type of gear is very important in many automotive applications. Bevel gears are generally mounted perpendicularly, but can be mounted at almost any shaft angle. Straight bevel gears are the least costly type of bevel gears, but they are also limited in application for similar reasons as the spur gear (Hamrock et al. 2005, pg. 678-680).

The next type of gear, and the most advanced gear discussed in this report, is the spiral bevel gear. This gear has the angled face of a bevel gear and also has angled teeth similar to those of a helical gear. The angle of the teeth varies along the face of the gear, which creates a curved tooth shape. These gears allow several teeth to be in contact at once, meaning they translate much more power than a standard bevel gear, but still share the property of changing the direction of the motion. Spiral gears are best suited for higher speed applications than the straight bevel gear. However, the thrust force generated by a spiral gear is much greater than the straight bevel gear and must be accommodated for. The cost of spiral bevel gear sets is also very high as compared to most other types of gears (Hamrock et al. 2005, pg. 678-680).

Other gears in this category, which are not discussed in this report, are the hypoid bevel gear and the Zerol bevel gear. The Nonparallel-noncoplanar category of gears, which primarily includes worm gears, is also omitted from this report.

3.5) Manufacture

Gear manufacturing requires advanced and highly specialized procedures for most types of gears. As the apparent complexity of the gear geometry increases, so does the complexity of manufacture. Most gear manufacture is done on specialized machine tools designed specifically for creating gears. However, as more advanced, multiple axis CNC machine tools have become available, more gearing can be done using these tools, as long as accurate models can be imported to the machine.

Gears can be made of many materials including a variety of metals and non-metals. The most common gear metals are cast iron, steel alloys, and bronze. Metal gears are preferred for applications with high loads and rotational speeds. The primary characteristics of the metals used in gearing are shear strength, resistance to bending, and resistance to wear and pitting. Cast iron is one of the most widely used metals because of its resistance to wear, good strength and ease of manufacturing. Casting is a process that can produce a large variety of shapes that can be made very close to tolerance. Non-metal materials, such as nylon, are generally used in low-load gearing applications to reduce cost and also to reduce noise during operation (Agro Engineers).

Once a material is selected the gear manufacturing process begins by creating a gear blank, which is completely stress relieved to minimize distortion that may have taken place during the initial manufacturing step. The gear blank is basically a gear without any teeth. Gear blanks can be produced using a number of processes because of their simple shape. The gear teeth are then cut out, with an allowance given for the subsequent grinding which will take them down to the exact desired shape and size. Gears also typically undergo broaching, hobbing, heat treatment, shaving and deburring to create a gear to the necessary tolerances (Gears Hub). The process is long and complex, but necessary to create an accurate and well performing gear. Any imperfection in even the simplest types of gears can cause critical failures for not only the gears, but also the machines they are used within.

As mentioned above, the machine tools used for gear cutting are generally highly specialized pieces of equipment, but it is becoming more desirable for companies to use multi-axis CNC machine tools for gearing as these machines have become more popular. These machine tools require an accurate CAD model that can be imported into the machine tool's computer in order to operate at their highest level of performance and robustness. Most types of gears can be modeled quickly and easily using a variety of CAD software, but more complex types of gears have more complex geometries which are much more difficult to generate. One such gear is the spiral bevel gear. The variable spiral angle makes the gear extremely difficult to model using typical CAD modeling techniques. A commercially available CAD model of the spiral bevel gear would revolutionize the manufacture of these types of gears, but does not currently exist. This project seeks to lay out the steps necessary to create an accurate model of the spiral bevel gear in SolidWorks in order to pave the way for a modeling system that can create models of spiral bevel gears of any size and orientation.

3) Methodology

Many steps were taken to complete the goal of this project. These steps are separated into sections based on the type of gear we designed and modeled. We started with the simpler spur gear, then advanced to the straight bevel gear and finally to the spiral bevel gear. This chapter explains the process we used

4.1) Spur Gear

The biggest challenge in modeling the spur gear was to parametrically define the involute tooth geometry and the undercut of the teeth. The first attempt we made at correctly defining the involute curve required careful dimensioning of a series of curves to attain an approximation of the full involute. Our most important discovery in the process of modeling the spur gear was the parametric equation of the involute of a circle. Defining the curve by a parametric equation reduced the amount of time for creating the curve by 99% over our original process. The full process we used for modeling these gears is shown in the following chapter.

4.2) Bevel Gear

Once the spur gear process was fully defined, we began working with the bevel gear. Our initial modeling process involved extruding the teeth of the gear onto a conical shaped body. However, the defining geometries of the bevel gear teeth restricted us from using this method because we could not fully define the tooth geometries. Our next approach was to create a gear blank and cut the teeth out of this blank. We found a method of defining the tooth geometry on the back face of the gear, called Tregold's Approximation, which we used to sketch an equivalent spur gear on the back face of the gear blank. We used the gap between adjacent teeth of the equivalent spur gear and lofted a cut of that profile which would terminate at the apex of the gear. This allowed us to make a very accurate approximation of the bevel gear geometries.

4.3) Spiral Bevel Gear

The spiral bevel gear was the most complex gear we dealt with and was the next step in our project. The hardest part of the modeling of this gear was defining the three dimensional guide curve for the spiral tooth to follow during the lofted cut. As with the spur gear, our initial idea was to create this curve piecewise. We realized that the curve could not be fully defined, nor could it be accurate using this method. Thus we tried several different methods for defining this path. The first was a parametric equation of a spiral, which we manipulated into the three-dimensional space of the bevel gear. The idea behind this method was to constrain the spiral curve to the bottom land of the gear tooth for the entire length of the cut. The other major method we attempted was to use an equation that described the variable spiral angle. All of the processes we used are described fully in the next chapter.

4) Results

5.1) Spur Gear

5.1.1) Geometry

In making the spur gear we accomplished the correct involute profile of the gear teeth, along with the process in which to accurately build the profile with the proper fillets and undercuts as needed. The proper dimensions were also found by following sets of equations to make sure the geometries are correct. The equations were defined in the AGMA handbook. The use of these equations helped define dimensions used when modeling the gear in the SolidWorks CAD software. The most important defining equation for the spur gear was the parametric equation of the involute, which is shown in the following section. However, some dimensions were not found in the handbook, such as the thickness of the gear and the dimensions of the bore and keyway. The bore and keyway are cut from the center of the gear so it can be fitted to a shaft. At first these dimensions were assumed to be arbitrary, but were later found to be related to stress and strength values of the finished spur gear.

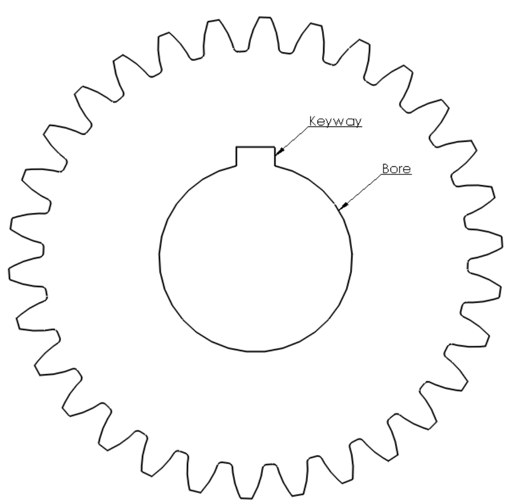


Figure 1: Finished Spur Gear

5.1.2) Process

To model a spur gear several steps must be followed. These steps ensure that the proper geometries are made following the governing equations. Once all the necessary geometric parameters have been calculated using the defining equations, the gear can be made. All of the equations used to find the parameters necessary to model the gear are listed in this chapter. Equations for additional geometric parameters of the spur gear are listed in Appendix 1. Most of the equations in this chapter can be used for any standard set of units, except where the units are specified.

To model a spur gear the first step is to draw a circle with a diameter equal to the form diameter, defined by equation 1, where d_f is the form diameter, r_l is the theoretical limit radius, and P is the diametral pitch of the gear. The theoretical limit radius is a necessary parameter for finding the form diameter, but is merely an engineering parameter and is not used when modeling the gear. Finding the form diameter requires first defining a set of other parameters, which can be done using a series of equations listed in appendix 1 of this report. Determination of which datum plane this circle should be drawn in will depend on how the gear will be manufactured. The designer needs to consider which plane the machine tool will most easily be able to cut the gear in.

$$\text{Form Diameter: } d_f = 2(r_l - (.025/P)) \qquad \text{(Eq. 1)}$$

Extruding this circle to the determined thickness of the gear results in a blank gear to which the teeth will be added. To make a proper involute tooth, create a new sketch on one of the flat faces of the gear. In this sketch, draw and define the base circle, pitch circle, and outside diameter circle constraining each of their centers at the origin of the blank. The typical design approach for the spur gear is to define the pitch and outside diameter of the gear as user inputs.

Equation 2 is used for finding the base diameter. The involute curve originates at the base circle of the gear. The involute curve is defined by the set of parametric equations (equations 3 and 4) below. In the following equations, D_b represents the base diameter, D represents the pitch diameter, φ represents the pressure angle and r_b represents the base radius. The values of t in the parametric equations are used to define an interval over which to draw the curve.

Base Diameter: $D_b = D*\cos(\varphi)$ (Eq. 2)

Parametric Involute Equations: $x(t)=r_b(\cos(t)+t*\sin(t))$ (Eq. 3)

$y(t)= r_b(\sin(t)-t*\cos(t))$ (Eq. 4)

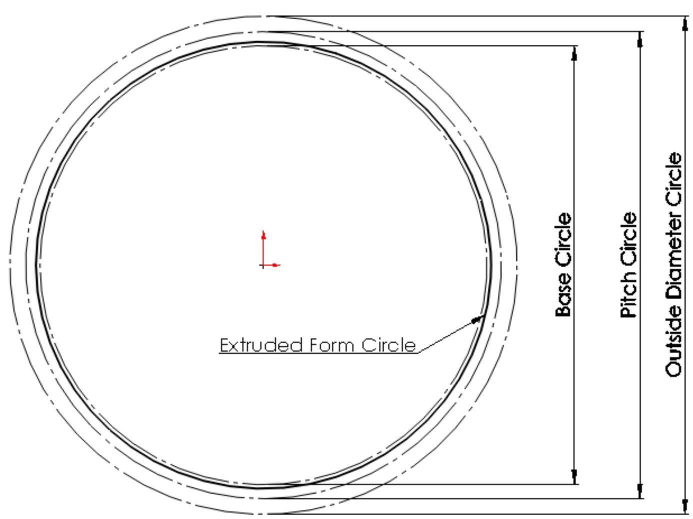


Figure 2: Sketch of the Necessary Construction Circles for the Spur Gear

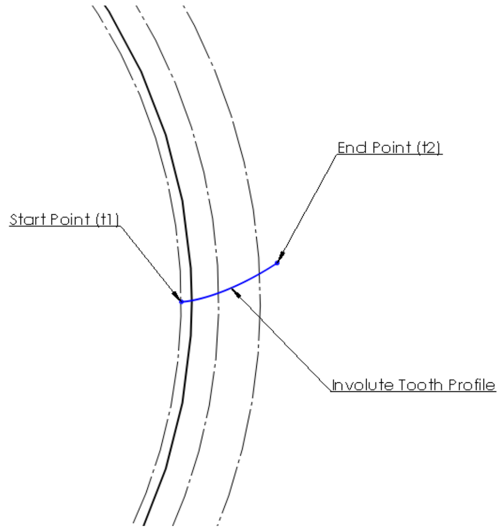


Figure 3: Involute Tooth Profile for the Spur Gear

The start and end points of the parametric curve are referred to as t_1 and t_2. t_1 should be constrained to 0 and t_2 should be left unconstrained initially. Constrain the start of the involute circle to the base circle and the end to the outside diameter circle. To finish the tooth profile a center line must be created. Using half the tooth thickness parameter, which is defined in equation 5, define the distance between the center line and involute along the pitch diameter.

Circular Tooth Thickness: $t = \pi /(2*P)$ (Eq. 5)

To complete the tooth profile, mirror the involute about the centerline. Close the profile by drawing three point arcs along the outside diameter between the ends of the involute curves and along the base circle between the start of the involute curves. Now extrude the tooth profile along the thickness of the gear to finish one tooth. To construct the desired number of teeth use a circular pattern about the center axis of the spur gear.

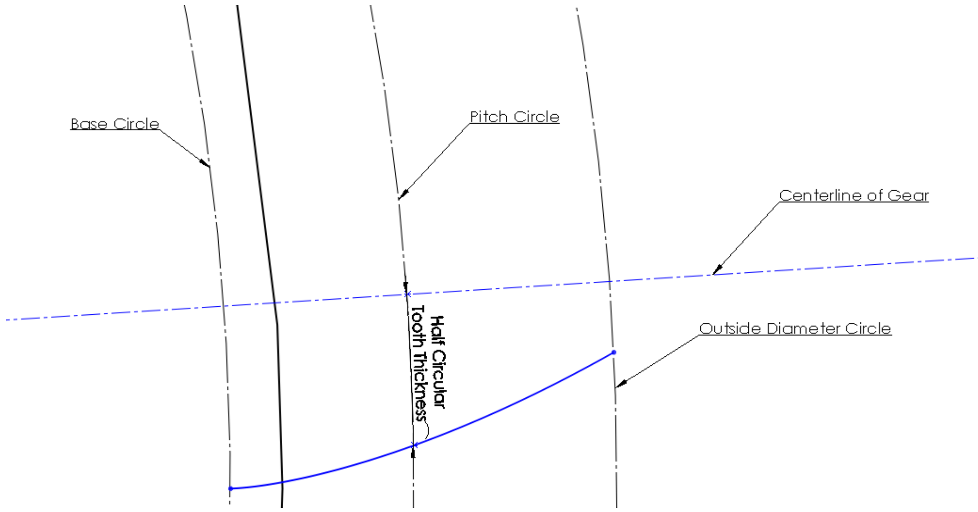


Figure 4: Fully Constrained Involute Curve

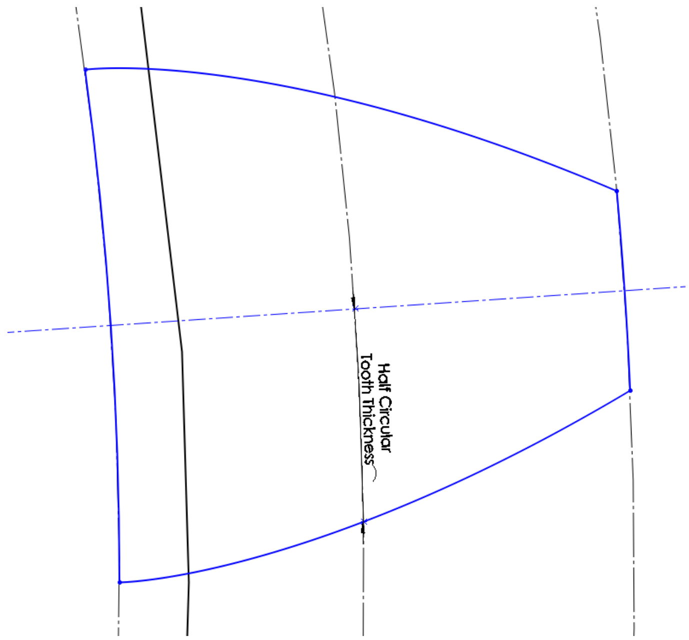


Figure 5: Finished Involute Tooth Profile for the Spur Gear

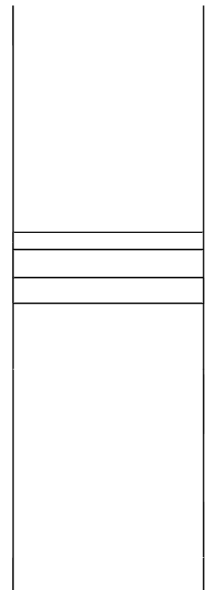


Figure 6: Top View of the Single Extruded Spur Gear Tooth

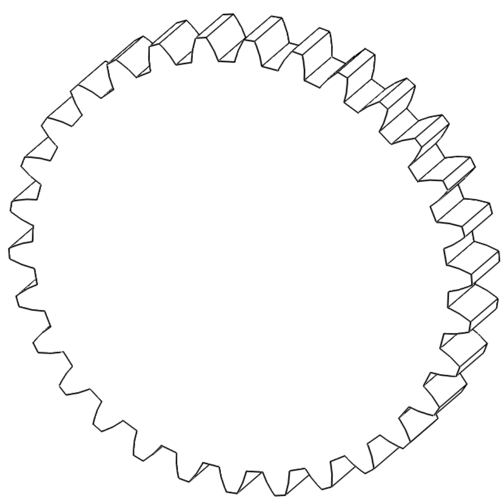


Figure 7: Spur Gear with Involute Teeth

The teeth now need to be cut down to the correct depth. The current tooth profile represents the surfaces of the gear teeth which come into contact with one another during gear meshing, but the tips of the gear teeth also need space to rotate beneath the current form circle extrusion. To make the whole depth of the teeth (defined by equation 6) correct, a section must be removed from between each tooth. To do this, create a new sketch on one of the flat faces of the gear. Draw and dimension the root circle of the gear and confine its center to the origin of the gear. The root diameter is defined by equation 7. Draw two lines that are tangent with the start of the involute curve and end at the root circle. Draw two arcs to close the shape of the cut. The first is along the form circle extrusion between the points where the two lines meet the involute curve. The other arc is along the root circle and between the other two ends of the lines. Finish the profile of the cut by adding fillets between each of the tangent lines and the root circle. The fillet radius is defined by equation 8.

Whole Depth: $h_t = 2.2/P + .002$ in. (Eq. 6)

Root Diameter: $D_R = D_o - 2h_t$ (Eq. 7)

Fillet Radius: $r_f = .3/P$ (Eq. 8)

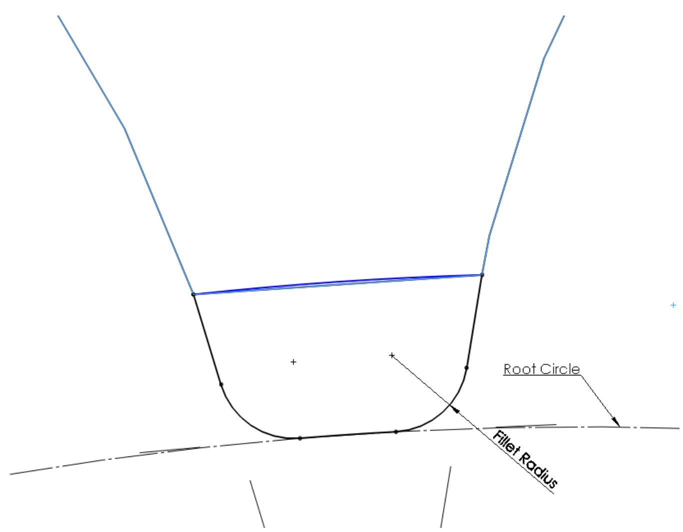


Figure 8: Sketch of Fillet Area of Spur Gear

In the above equations, P represents the diametral pitch and D_o represents the outside diameter. Now make a through all extruded cut of this profile. Selecting a through all cut will ensure that the cut is made to the correct depth. Use a circular pattern of this feature about the center axis of the gear to finish the teeth of the gear. The number of instances of the pattern needs to be the same as the number of teeth used in the earlier pattern.

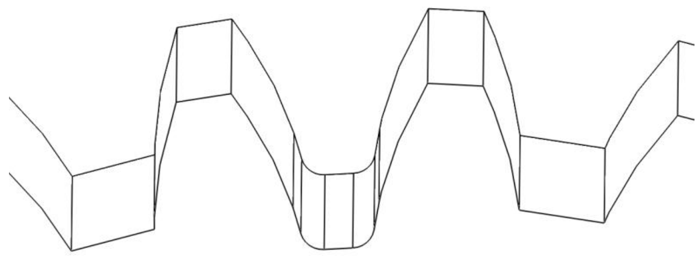


Figure 9: The First Cut between Two Spur Gear Teeth

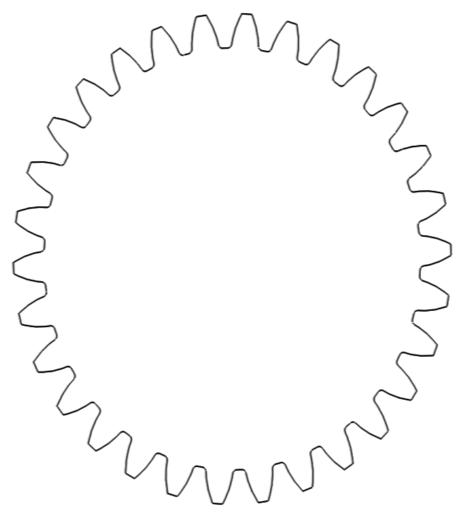


Figure 10: Spur Gear Model with Cuts between All of the Teeth

The last step in modeling the spur gear is cutting a bore and keyway in the center of the gear. Make a new sketch on one of the flat faces of the gear. Draw a circle of the diameter appropriate for the bore and cut the bore through the entire thickness of the gear. To correctly draw the keyway, first draw a vertical centerline from the center of the gear on your sketch. Now draw a vertical line to either side of the centerline which starts on the bore circle and has a length equal to the height of the keyway. Now dimension the distance between this vertical line and the centerline to be equal to one half the keyway width. Draw a horizontal line from the free end of this vertical line and define its length as the full keyway width. Add a final vertical line between the free end of the horizontal line and the bore circle to finish the keyway sketch. Close the sketch with an arc along the bore circle and between the two ends of the lines then cut the keyway through the entire thickness of the gear. The steps for creating the proper keyway sketch are labeled in figure 11. Consult a gear catalogue for standard bore and keyway dimensions.

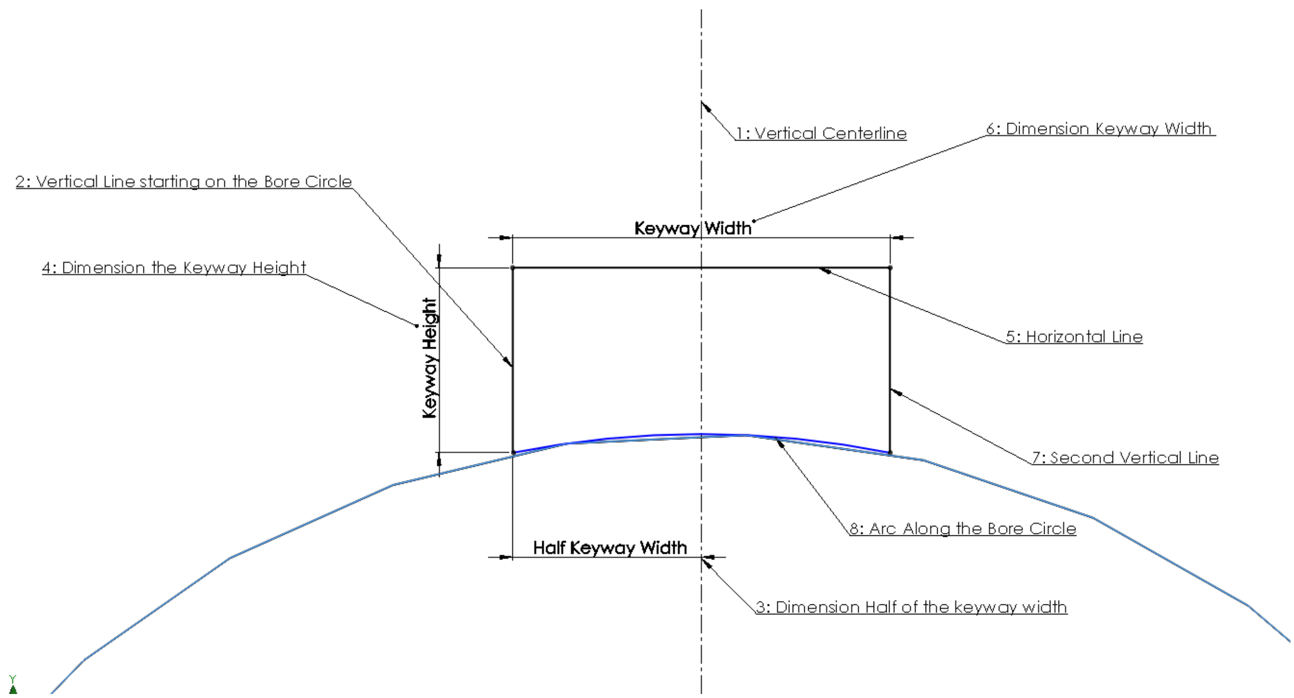


Figure 11: Process for Drawing the Keyway

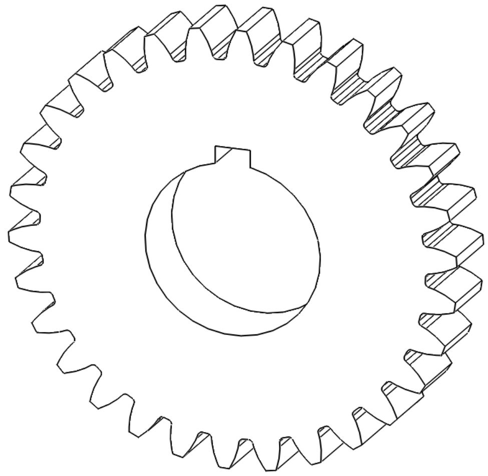


Figure 12: Final Spur Gear Model

5.1.3) SolidWorks Flowchart

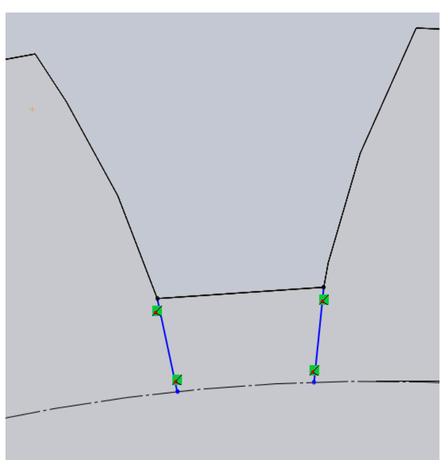
The process for creating the spur gear model in SolidWorks is described in detail in the following flowchart.

- Open a new Part in SolidWorks
- Begin by selecting one of the standard SolidWorks Planes
    - The drawing should be created in the plane best suited for how the gear will be manufactured
- Select the Sketch Circle tool
    - Sketch a circle centered at the origin
- Select the Smart Dimension tool
    - Add a dimension equal to the form diameter of the gear you are modeling to the circle you have just drawn
- Select the Extruded Boss/Base tool
    - Choose the form circle as the entity to extrude and enter the face width of the gear as the depth of extrusion
- Select the Sketch Circle tool
    - Sketch three new circles centered at the origin (concentric to the form diameter circle)
    - Select each circle you have just drawn and select the “For Construction” check box
- Select the Smart Dimension tool

- Add dimensions equal to the base diameter, pitch diameter and outside diameter to the three circles you have just drawn
- Select the Equation Driven Spline Tool
  - Select Parametric as the coordinate system used to define the spline
  - Enter the equations $x(t) = r_b(\cos(t) + t * \sin(t))$ and $y(t) = r_b(\sin(t) - t * \cos(t))$ in the appropriate areas. r_b represents the radius of the base circle (NOT the diameter)
  - Define and lock t1 to be 0
  - Define t2 to be approximately 0.5, but leave the value unlocked
  - Select the check mark to finish the sketch of the spline
- Click on the start point of the involute curve
- While holding down 'Ctrl' click on the base circle to select the circle and the start point simultaneously
  - Select Coincident as a relationship between these two objects
- Click and drag the end point of the involute curve until the outside diameter circle is highlighted, then release. (Or use the same process as in the previous step)
- Select the Centerline tool
  - Sketch a centerline, emanating from the center of the gear, that approximately bisects the tooth you are creating (does not need to be exact)
- Select the Point tool
  - Sketch two points, one on the intersection of the centerline and the pitch circle and the other on the intersection of the involute curve and the pitch circle
    - Make sure that the intersection relationship appears before sketching the points. This may require holding the mouse on the intersection for a second so that SolidWorks can recognize it.
- Select the Smart Dimension tool
  - Click on each of the two points you drew in the previous steps and then click on the portion of the pitch circle between the two points. This signifies that you are specifying an arc length as opposed to a linear dimension.
  - Add a dimension equal to ½ the circular tooth thickness of the gear teeth
- Select the Mirror Entities tool
  - Select the involute curve as the entity to mirror
  - Select the centerline as the entity to mirror about
  - Make sure that the "Copy" option is selected and click the check mark to draw the other edge of the tooth
- Select the Three-Point Arc tool
  - Select the points of each involute curve which lie on the base circle as the first two points and then select the base circle itself
  - Repeat with the points that lie on the outside diameter circle
- Select the Extruded Boss/Base tool

- Choose the sketch you just completed as the entity to extrude and use the face width of the gear as the depth of extrusion. Also make sure that the extrusion is going in the correct direction before clicking the check mark

- Turn on the “View Temporary Axes” option in the view menu
- Select the Circular Pattern tool
    - Select the center axis of the gear as the axis for the pattern
    - Input the number of gear teeth as the number of instances
    - Check the box for equal spacing
    - Select the extrusion of the gear tooth as the Feature to Pattern
    - Click the check mark to finish the pattern
- Begin a new sketch on the front face of the gear
- Select the Sketch Circle tool
    - Sketch a new circle centered at the center of the gear
    - Select the “For Construction” option
- Select the Smart Dimension tool
    - Add a dimension equal to the root diameter of the gear you are modeling to the circle you have just drawn
- Select the Line tool
    - Sketch two new lines
        - The first should start on the intersection of a tooth and the form circle extrusion and end on the root circle
        - The other line should start on the intersection of the next adjacent tooth and the form circle extrusion and also end on the root circle
        - The below picture is included to show what the result of this step should be:



- While holding down ‘Ctrl’ click on one of the lines you have just created and the involute curve it connects to (to select them simultaneously)
    - Add a tangent relation between the two shapes

    - Repeat this process with the other line and its respective involute curve
- Select the Three-Point Arc tool
    - Draw two, three-point arcs connecting the top and bottom of each line and tangent to the respective circles or arcs they naturally fall on
- Select the Sketch Fillet tool
    - Input the fillet radius of the gear in the fillet parameters section
    - Select either one of the lines you have just drawn
    - Select the arc closest to the inside of the gear
    - Repeat this process on the other line and click the check mark to add the fillets
- Select the Extruded Cut tool
    - Select the “Through All” option
    - Select the sketch you have just finished as the entity to cut
    - Click the check mark to finish the cut
- Select the Circular Pattern tool
    - Select the center axis of the gear as the axis for the pattern
    - Input the number of gear teeth as the number of instances
    - Check the box for equal spacing
    - Select the cut you have just made as the Feature to Pattern
    - Click the check mark to finish the pattern
- Begin a new sketch on the front face of the gear
- Select the Sketch Circle tool
    - Sketch a new circle centered at the center of the gear
- Select the Smart Dimension tool
    - Add a dimension equal to the bore diameter of the gear you are modeling to the circle you have just drawn
- Select the Extruded Cut tool
    - Select the “Through All” option
    - Select the sketch you have just finished as the entity to cut
    - Click the check mark to finish the cut
- Begin a new sketch on the front face of the gear
- Select the Centerline tool
    - Sketch a vertical centerline starting at the center of the gear and ending somewhere above the top of the bore
- Select the Line tool
    - Sketch a vertical line starting on the bore circle and to either side of the centerline
- Select the Smart Dimension tool
    - Add a dimension equal to half of the keyway width between the centerline and the vertical line you just drew
    - Dimension the length of the line as the keyway height
- Select the Line tool

    - Sketch a horizontal line starting on the free end of the line you just dimensioned and ending somewhere on the other side of the centerline
- Select the Smart Dimension tool
    - Dimension the length of the new line as the keyway width
- Select the Line tool
    - Sketch a vertical line starting on the free end of the line you just dimensioned and ending on the bore circle
- Select the Three-Point Arc tool
    - Sketch an arc between the two points of the vertical lines on the bore circle and also tangent to the bore circle
        - This arc simply closes the sketch of the keyway cut and could also be a line between the two points
- Select the Extruded Cut tool
    - Select the "Through All" option
    - Select the sketch you have just finished as the entity to cut
    - Click the check mark to finish the cut

5.2) Bevel Gear

5.2.1) Geometries

In modeling the bevel gear, the angles at which the teeth were created had to be calculated. The proper bevel gear dimensions were found by following the defining sets of equations. The tooth profile on the back face of the gear was defined using an equivalent spur gear as defined by Tregold's Approximation. The involute tooth profile was used to create this sketch.

5.2.2) Process

To make a bevel gear there are several steps that are important to follow. These steps ensure that the proper geometries are made following the governing equations. Once all the parameters have been calculated using the defining equations, the gear can be modeled. All of the parameters required to model the bevel gear are included in this section. Bevel gears are always designed in matching sets, therefore the equations for both the gear and the pinion are necessary. Only the equations of the gear are shown in this chapter. Equations for the pinion and all other geometric bevel gear parameters are included in Appendix 7.2.

To start the bevel gear, you will first make a sketch of the bevel gear profile. To start, draw a centerline through the origin. Off of the top point of the center line draw three construction lines. These lines will represent the pitch angle, face angle and root angle.

Pitch Angle: $\Gamma = \Sigma - \Upsilon$ (Eq. 10)

Face Angle: $\Gamma_o = \Gamma + \delta_G$ (Eq. 9)

Root Angle: $\Gamma_R = \Gamma - \delta_G$ (Eq. 11)

In the above equations, Σ represents the shaft angle, Υ represents the angle of the pinion, and δ_G represents the dedendum angle of the gear. Using the parameters defined by the governing equations, give each line the correct angle relative to the center line. Refer to figure 13 for how to fully define the bevel gear profile. The face width is defined by equation 12, where A_o represents the outer cone distance.

Face Width: $F = A_o/3$ (Eq.12)

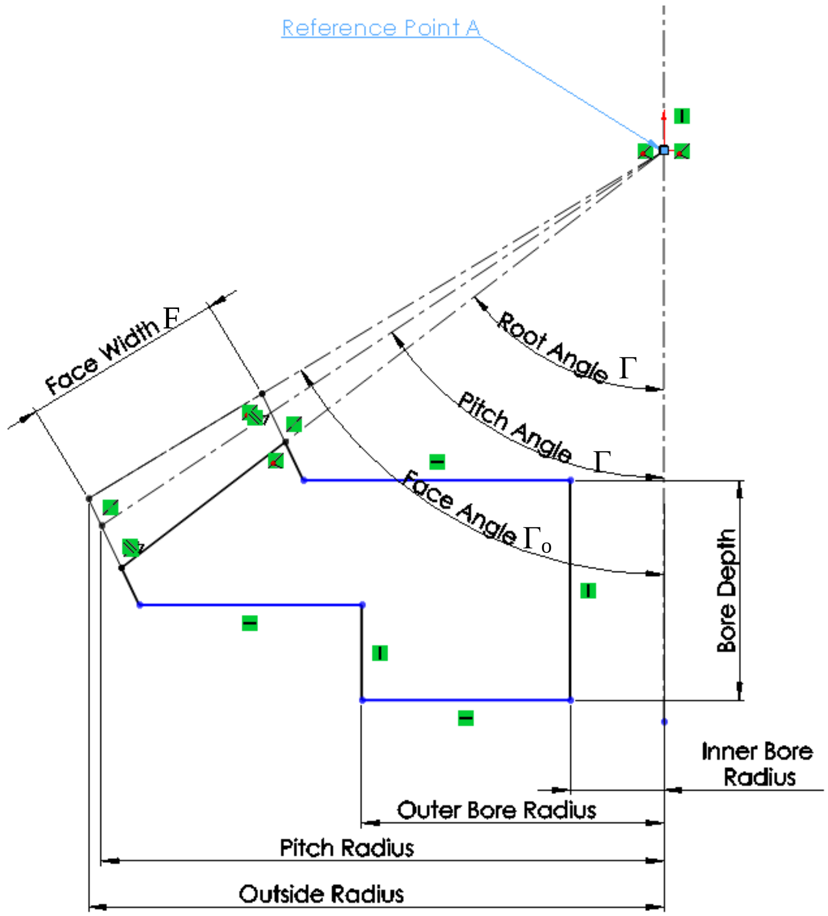


Figure 13: Bevel Gear Profile

Revolve the profile about the centerline to make the gear blank. A new sketch plane tangent to the outside face of the gear must be made next. This also allows for an easy reference to the outside edge of the gear profile.

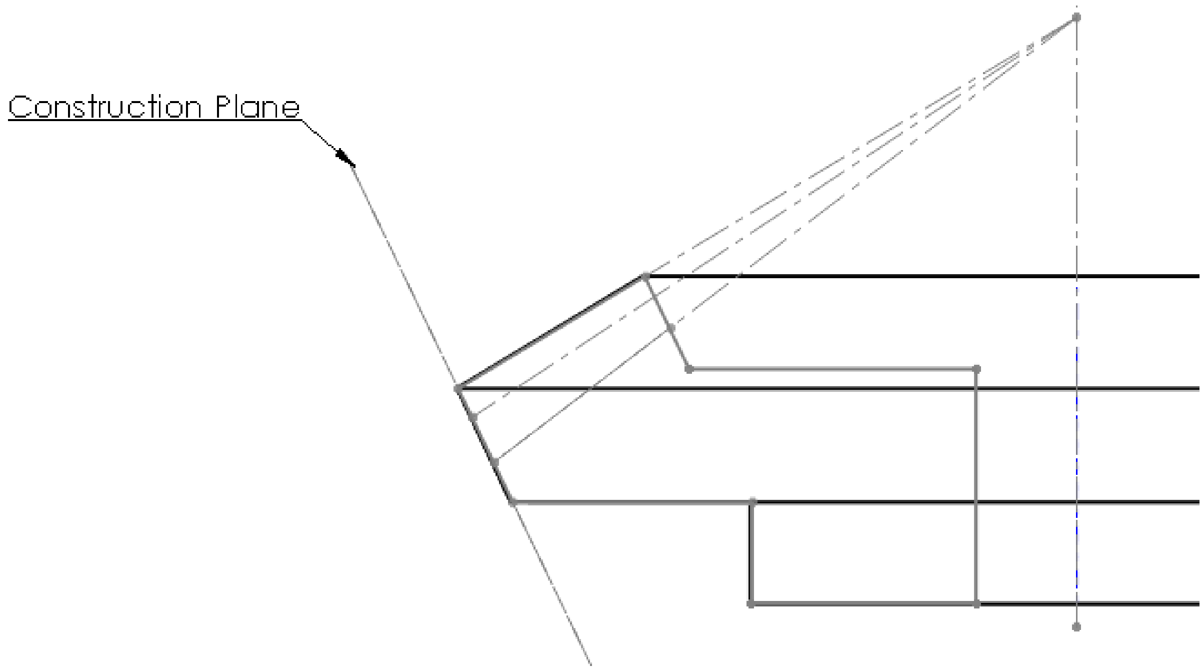


Figure 14: Construction Plane

On this new sketch plane an equivalent spur gear profile is made. This sketch is an approximation for the bevel gear teeth at the outer most part of the tooth. The process for making this profile is almost exactly the same as the spur gear. The first step is to draw a circle that represents the pitch circle making sure it lines up with the pitch angle line from the bevel gear profile. Then the base circle can be drawn and is constrained to the root angle. The equations used to find the equivalent spur gear geometries are the same as used for the spur gear, as described in the previous section. The pitch radius and number of teeth for the equivalent spur gear are based on the parameters of the bevel gear you are modeling and are given below.

Equivalent number of teeth: $N_{vg} = \frac{Ng}{\cos(\Gamma)}$ (Eq.13)

Equivalent pitch radius: $Rvg = \frac{Rg}{cos(\Gamma)}$ (Eq.14)

Where N_g is the number of teeth for the bevel gear and Γ is the pitch angle of the bevel gear. When using the equation for the equivalent number of teeth, round the number up in order to create a gap between teeth that is slightly smaller than would be created using the fractional number of teeth. By making the gap between teeth smaller, the teeth will be slightly larger than they are supposed to be and the gear can later be grinded down to its final dimensions. Create the involute using the same parametric equations used in the spur gear process. Remember that r_b is the radius of the equivalent base circle. Make the involute coincident with the base circle. Dimension the distance between the centerline and the involute to half the equivalent tooth thickness. Mirror the involute about the centerline, then do a circular sketch pattern to the attain the equivalent number of teeth. This will define the gap between the teeth, that can then be cut.

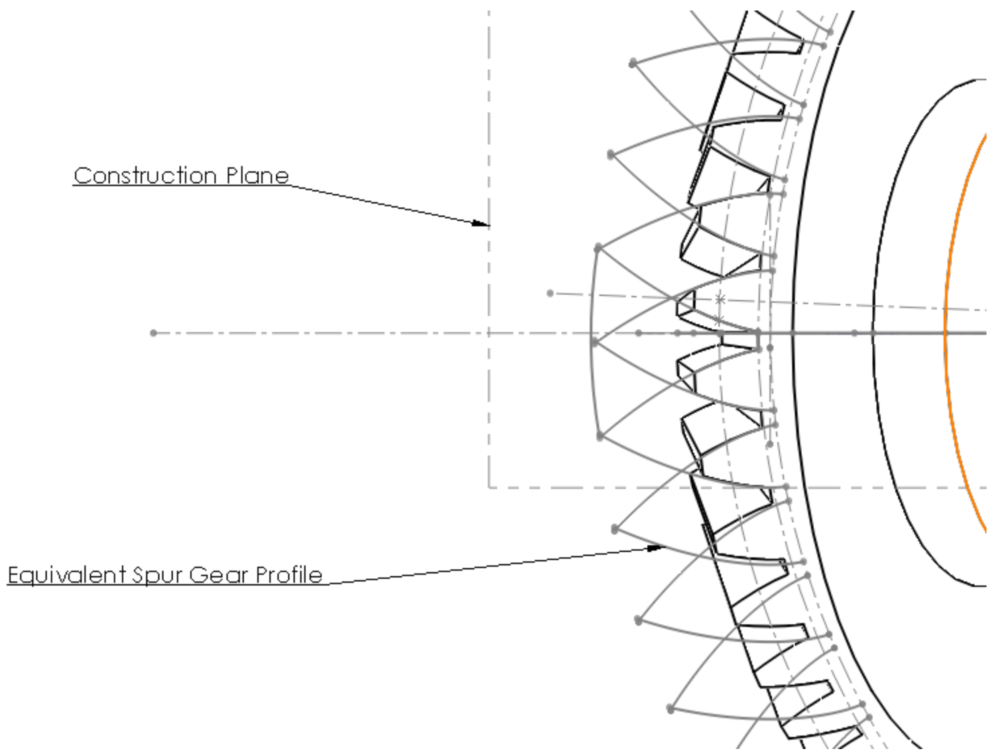


Figure 15: Equivalent Spur Gear Profile Sketch

Create the bottom of the profile by sketching an arc between the bottom of the involutes along the base circle. For the top of the profile a straight line can be used as long as the entire profile is taller than the outside face of the gear (see figure 16).

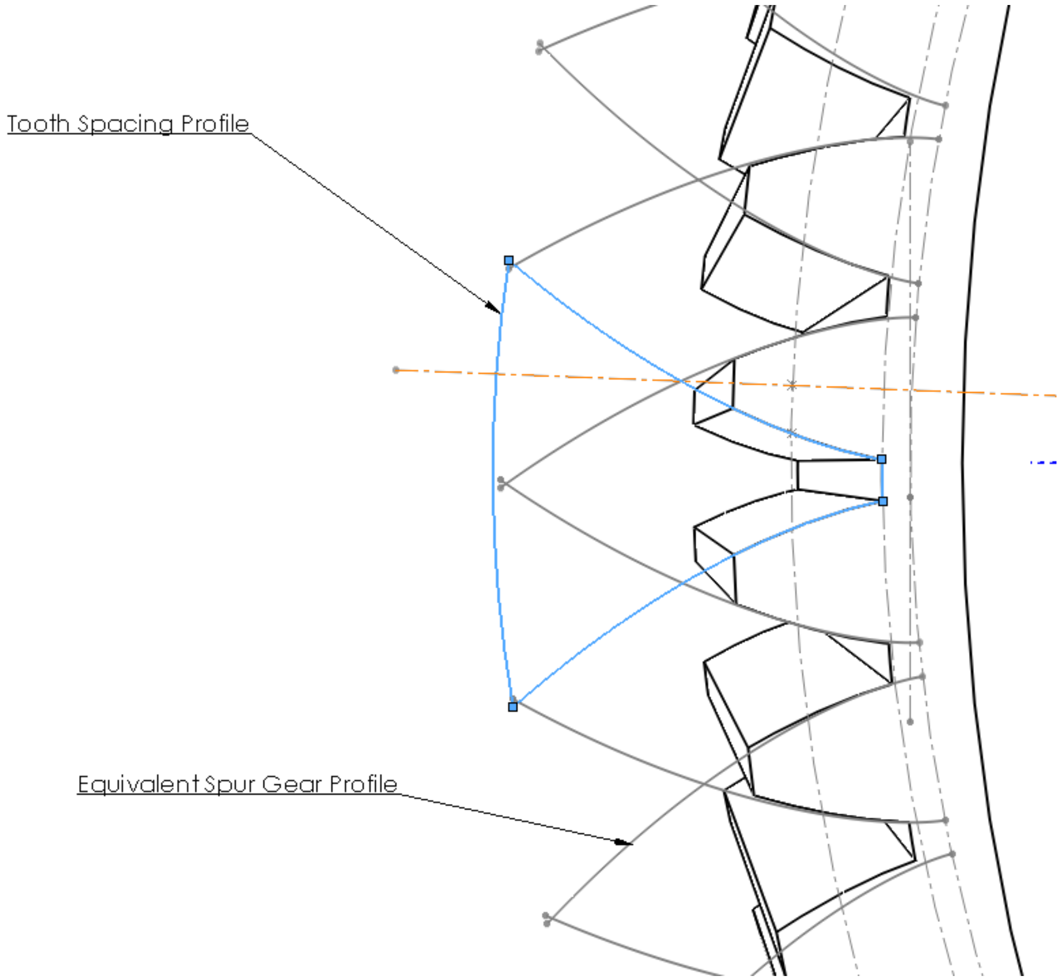


Figure 16: Teeth Spacing Profile

To remove the correct material a new sketch needs to be made at the apex of gear profile sketch. Create a 3D sketch and draw a point that is coincident with reference point A from figure 13.

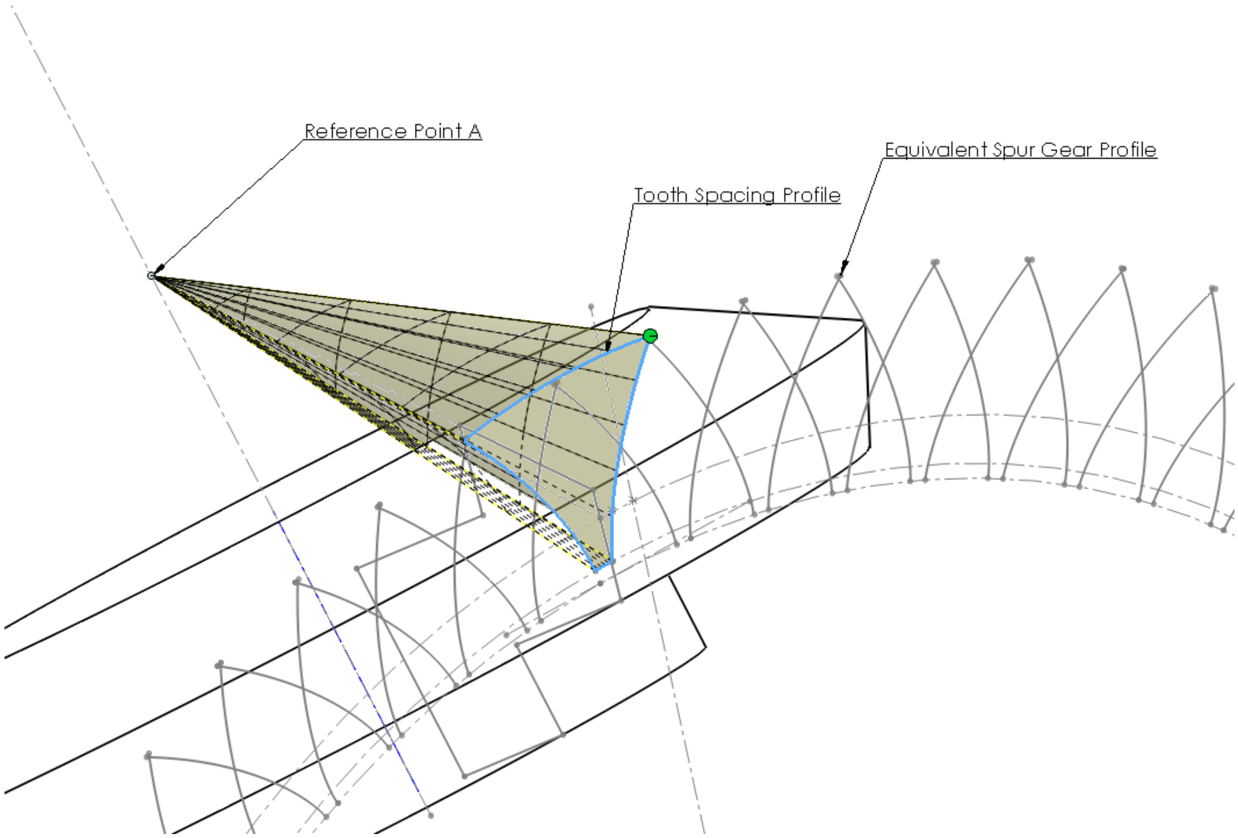


Figure 17: Profiles of the Cut

The last step is to use a lofted cut from the 3D sketch just created to the tooth space profile made on the construction plane. This creates one space in between two teeth. A circular pattern is then used to cut the space in between the rest of the teeth.

5.2.3) SolidWorks Flowchart

The process for creating the bevel gear model in SolidWorks is described in detail in the following flowchart.

- Step 1:
  - Create Profile of Bevel gear in the Front Plane.
    - All bevel gear and equivalent spur parameters are calculated prior to modeling the gear. The governing equations exist in most gear handbooks or can be calculated using the "Gear Parameter Calculator."
    - Select the Front Plane and Create a new Sketch (Front Plane png)

        - Create a center line to base your angles and profile on.
        - Draw three lines that represent the three different angles (root, face, and pitch).
            - Select “For Construction” under line Properties for each line.
        - Make each line the appropriate dimension
            - Select “Smart Dimension”
            - Click the center line then a construction line.
            - Enter each angle as shown in Figure…..
        - Make the dimensions for the outside diameter and the Pitch Diameter.
            - Select “Smart Dimension”
            - Click the end point of the face angle line and then the centerline
            - Enter the Dimension
            - Repeat for Pitch Diameter
        - Connect the ends of the Pitch line and Face line with a line.
        - Make a line from the Pitch line to the Root line
            - Start the line at the end of the pitch line and go past the root line
            - Left Click the line just created
            - While Holding shift Left Click the line connecting the Face and Pitch lines
            - Select “Parallel” under Relations in the Properties window
        - Create lines to finish the profile with dimensions
            - The Dimensions A, B, C, and D can be and dimension

- Step2:
    - Revolve Bevel gear profile.
        - Exit the Sketch
        - Select “Revolved Boss/Base”
        - In the Revolve Box
            - Highlight the first box under “Revolve Parameters” by left click
            - Select the center line of Sketch one
            - A preview of the revolved profile is shown
- Step3:
    - Create a plane tangent to the outside face of the teeth (Construction Plane 1).
        - A: Make the bevel gear profile visible in the solid model. This will help in the alignment of the Construction Plane 1.
            - See Figure 2.
- Step4:
    - On plane 1 create a profile of the equivalent spur gear.
        - A: Sketch circles which represent the base circle and pitch circle on Construction plane 1.

    - B: Sketch an involute curve which represents the correct profile of the tooth.
      - The equation of the involute should correspond with the base circle diameter of the gear. Its start point should be made coincident with a sketch of the base circle and its end point should be coincident with the outside diameter circle.
    - C: If the involute curve ends up being on the inside of the base circle draw a center line one the base circle and mirror the involute about the centerline.
    - D: Mirror the sketch of the involute curve to achieve the proper circular tooth thickness along the pitch circle of the gear.
    - E: Pattern the tooth profile about the base circle to get the appropriate spacing in between teeth.
      - To achieve the appropriate tooth spacing a circular pattern is used in SolidWorks.
        - See Figure 3.
- Step5:
  - Make a closed profile using the space between two teeth (See Figure 4).
    - A: Make arcs to close the two involute curves.
    - B: Make the bottom arc equal to that of the root angle on the bevel gear sketch.
- Step6:
  - Make a new point sketch in the Right Plane.
    - A: Make the sketch in the Right Plane to ensure the removed volume in Step 7 is correct.
    - B: This sketch is just one point.
    - C: Make the point directly over Reference Point A.
      - See Figure 1.
- Step7:
  - Remove the volume from the point sketch from step 6 to the Teeth Spacing Profile on Construction Plane 1.
    - A: Pattern the removed volume around the bevel gear for the calculated number of teeth.

5.3) Spiral Bevel Gear

5.3.1) Geometries

In making the spiral bevel gear the proper involute, angles, and spiral curve had to be used. Since the equivalent spur gear method could be used again the process is mostly the same as used for the straight bevel gear. The problem with the spiral bevel gears is the spiral itself. At first we used an equation for the spiral. The only problem was that it could not be confined in a way that made the method consistent. The spiral itself had the proper geometries, however. The next method was to use a spline guided by points that were predetermined to have the right curvature, but this method was also not fully resolved.

5.3.2) Process

To make a spiral bevel gear there are several steps that are important to follow. These steps ensure that the proper geometries are made following the governing equations. Once all the parameters have been calculated using the equations the gear can be made. The rest of the equations can be seen in Appendix 7.3.

The majority of the spiral bevel gear process is the same as the bevel gear process. The path that the lofted cut follows is where the real difference is. Our first method involved a parametric equation for a spiral:

Parametric Spiral Equations: $X(t)= D_g *t*\cos(t)$ (Eq.15)

$Y(t)= D_g*t*\sin(t)$ (Eq.16)

$Z(t)= K*t$ (Eq.17)

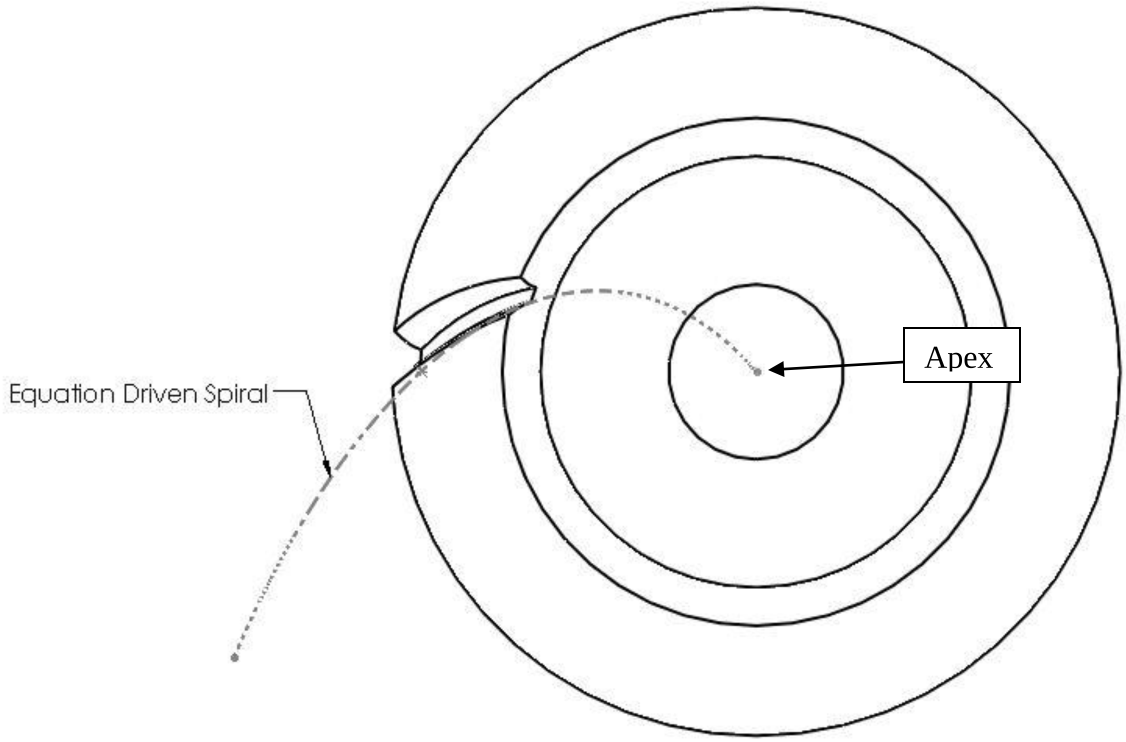


Figure 18: Top View of Spiral Gear with Parametric Spiral

Where D_g is the diameter of the gear and K is the root angle in terms of radians. These equations produce an accurate spiral curve. The problem was constraining the curve to the solid model to acquire the correct geometries every time. One end point could be constrained to the 3D sketch that represented the beginning of the path of the bevel gear tooth (the apex of the gear). The other end point was constrained to one of the bottom corners of the tooth space profile. The problem was having the middle of the curve constrained so that the path would follow the root angle through the entire face width.

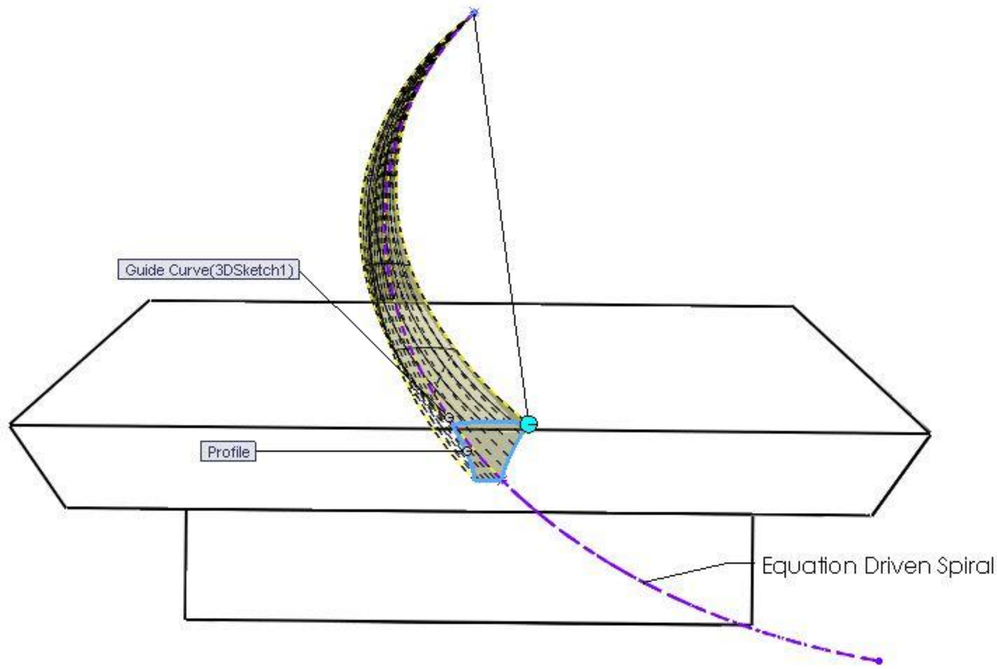


Figure 19: Lofted Cut of Spiral Gear Tooth Shape

The second process attempted was defining 3D points that we could constrain the path to. These points would be drawn on their x, y, and z coordinates in the SolidWorks space. Using these points, a spline would then be drawn from the apex to the tooth space profile while also being coincident with each of the 3D points. Defining the 3D coordinates would be based on the equation of the spiral angle, which is shown in equation 18.

$$\text{Spiral Angle: } \text{Sin } \gamma = (A_O/A)^* (\sin (\gamma_O) + ((A^2\text{-}A_O^2)/(2A^*r_C))) \qquad \text{(Eq.18)}$$

Where A_O is the mean cone distance, A is a general cone distance at which you want to find the spiral angle, γ_O is the spiral angle at the mean cone distance, and r_C is the radius of curvature of the spiral curve. Using these points as a reference geometry would ensured that the spiral curve had the proper spiral angle.

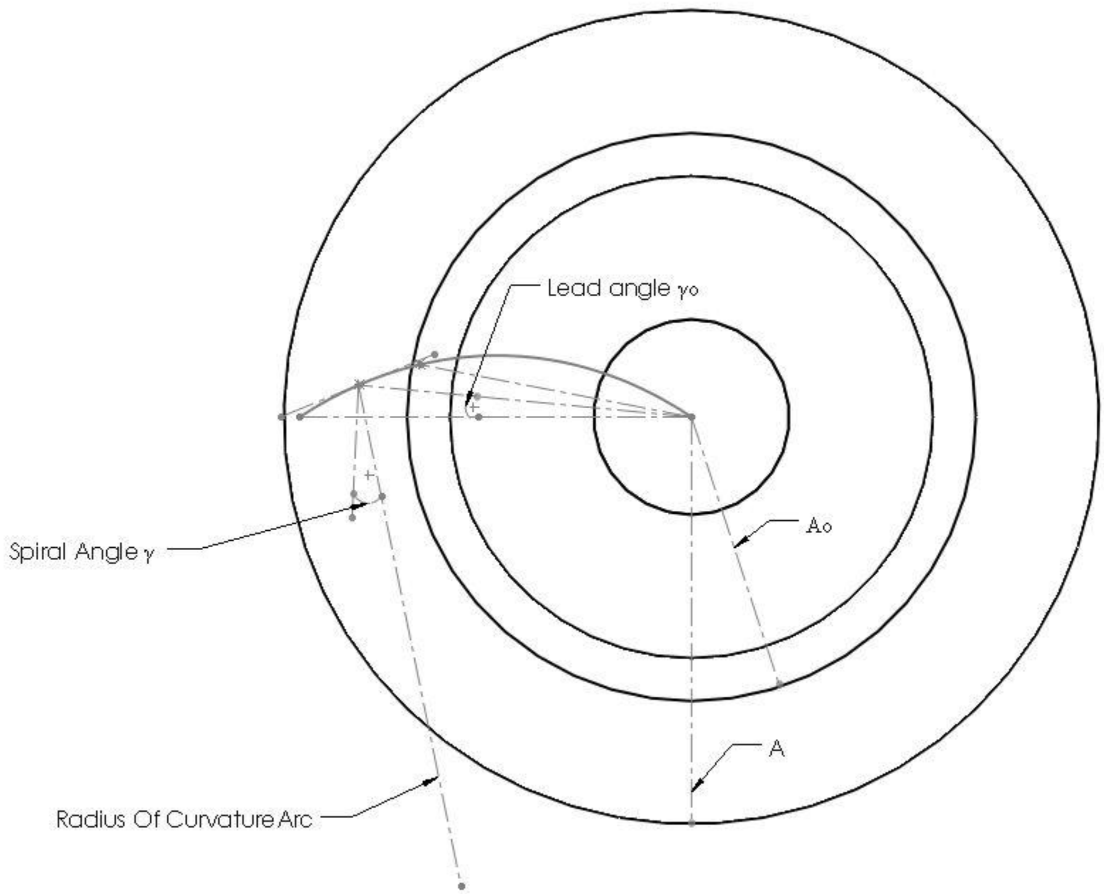


Figure 20: Top View of Spiral Gear with Spiral Angle Geometries

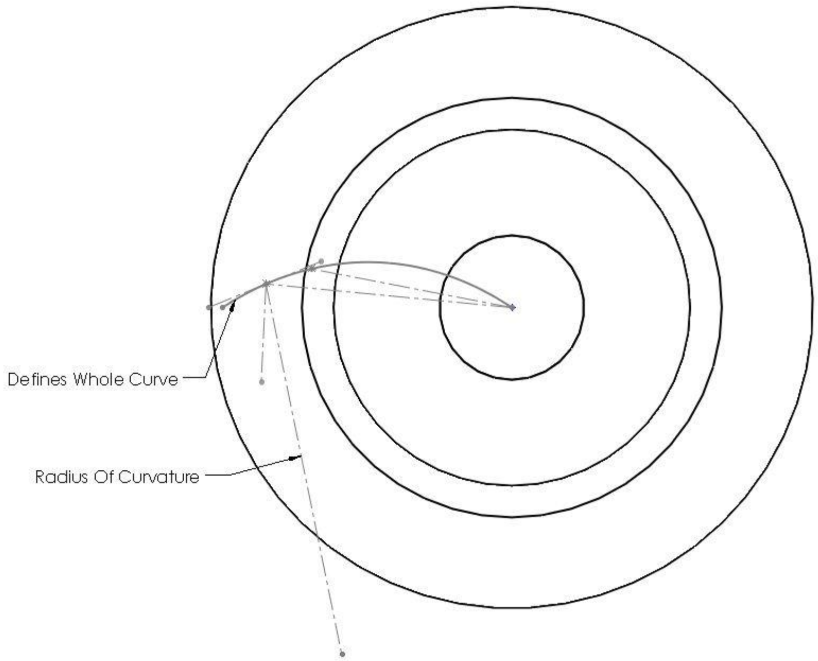


Figure 21: Radius of Curvature of the Spiral Arc

The problem with this equation is that the radius of curvature is not defined and is not an appropriate user-input. Unfortunately, no equation could be found to describe the radius of curvature of this curve. This leaves this method open for further research.

5.3.3) SolidWorks Flowchart

The process for creating the spiral bevel gear model in SolidWorks is described in detail in the following flowchart.

- Step 1:
  - Create Profile of Bevel gear in the Front Plane.
    - All bevel gear and equivalent spur parameters are calculated prior to modeling the gear. The governing equations exist in most gear handbooks or can be calculated using the "Gear Parameter Calculator."
    - Select the Front Plane and Create a new Sketch (Front Plane png)
    - Create a center line to base your angles and profile on.
    - Draw three lines that represent the three different angles (root, face, and pitch).

            - Select "For Construction" under line Properties for each line.
        - Make each line the appropriate dimension
            - Select "Smart Dimension"
            - Click the center line then a construction line.
            - Enter each angle as shown in Figure…..
        - Make the dimensions for the outside diameter and the Pitch Diameter.
            - Select "Smart Dimension"
            - Click the end point of the face angle line and then the centerline
            - Enter the Dimension
            - Repeat for Pitch Diameter
        - Connect the ends of the Pitch line and Face line with a line.
        - Make a line from the Pitch line to the Root line
            - Start the line at the end of the pitch line and go past the root line
            - Left Click the line just created
            - While Holding shift Left Click the line connecting the Face and Pitch lines
            - Select "Parallel" under Relations in the Properties window
        - Create lines to finish the profile with dimensions
            - The Dimensions A, B, C, and D can be and dimension
- Step2:
    - Revolve Bevel gear profile.
        - Exit the Sketch
        - Select "Revolved Boss/Base"
        - In the Revolve Box
            - Highlight the first box under "Revolve Parameters" by left click
            - Select the center line of Sketch one
            - A preview of the revolved profile is shown
- Step3:
    - Create a plane tangent to the outside face of the teeth (Construction Plane 1).
        - A: Make the bevel gear profile visible in the solid model. This will help in the alignment of the Construction Plane 1.
            - See Figure 2.
- Step4:
    - On plane 1 create a profile of the equivalent spur gear.
        - A: Sketch circles which represent the base circle and pitch circle on Construction plane 1.
        - B: Sketch an involute curve which represents the correct profile of the tooth.

            - The equation of the involute should correspond with the base circle diameter of the gear. Its start point should be made coincident with a sketch of the base circle and its end point should be coincident with the outside diameter circle.
        - C: If the involute curve ends up being on the inside of the base circle draw a center line one the base circle and mirror the involute about the centerline.
        - D: Mirror the sketch of the involute curve to achieve the proper circular tooth thickness along the pitch circle of the gear.
        - E: Pattern the tooth profile about the base circle to get the appropriate spacing in between teeth.
            - To achieve the appropriate tooth spacing a circular pattern is used in SolidWorks.
                - See Figure 3.
- Step5:
    - Make a closed profile using the space between two teeth (See Figure 4).
        - A: Make arcs to close the two involute curves.
        - B: Make the bottom arc equal to that of the root angle on the bevel gear sketch.
- Step6:
    - Make a new point sketch in the Right Plane.
        - A: Make the sketch in the Right Plane to ensure the removed volume in Step 8 is correct.
        - B: This sketch is just one point.
        - C: Make the point directly over Reference Point A.
            - See Figure 1.
- Step7:
    - Make a guide line using equation driven curve tool
        - A: Select 3D sketch
        - B: Enter the parametric equations for a spiral.
            - The z component is the root angle in radians times t
        - C: Make the beginning point of the curve coincident with Reference point A.
        - D: Make the end point of the curve coincident with the bottom corner of the tooth spacing profile.
- Step8:
    - Remove the volume from the point sketch from step 6 to the Teeth Spacing Profile on Construction Plane 1.
        - A: Make sure to use the guide curve when removing the volume

- B: Pattern the removed volume around the spiral bevel gear for the calculated number of teeth.

6) Conclusions and Recommendations

The defining equations for the spur, bevel, and spiral bevel gears are accurate. These equations helped to create the proper dimensions in SolidWorks either directly or indirectly. Once the dimensions were calculated the solid models could then be built with a trial and error approach. With this process an accurate spur gear and bevel gear were made. The method for making the spiral gear is still ongoing.

The spur gear model is complete and can be made in matching sets, with minimal interference. The successful mating of these gears proves that the involute equation is correct and that the governing equations used to design the gear are correct.

The bevel gear is a fairly accurate model. The process we used to create these models is well defined and makes consistently accurate gears. Using Tregold's Approximation for the equivalent spur gear does mean that the gears are somewhat inaccurate. However, due to the nature of the process of manufacturing bevel gears, the models that are created from our process are accurate enough to be machined. This is because the bevel gear goes through many procedures before it attains its final dimensions, such as heat treatments, grinding, and finishing. Using our method creates gear models that have slightly larger teeth than a finished bevel gear, meaning that they are a good enough estimation to be machined.

When mating our bevel gears in SolidWorks we ran into some alignment problems, but did not have the time to fully understand why. This is included in the following section on our recommendations.

The spiral bevel gear is not a completely accurate model. The spiral curvature is the biggest problem holding our gears back. The curves that we were able to define did not create a consistent spiral cut when used on different gears. These discrepancies are typically found between two gears which are designed as mating sets, which means they will not mate and thus are not fit for manufacturing. The biggest problem is how to constrain the spiral curve on gears of different sizes and orientations. We were not able to define the necessary relations to create a perfect spiral cut, but below are our recommendations for future work.

6.1) Recommendations

6.1.1) Automating the Modeling Process

The ultimate goal of this project is to automate the modeling process in order to reduce the design and manufacture time of these gears significantly. The next step towards this goal is to write a program that can import the modeling dimensions directly into a CAD software and instantly create a gear model or gear set. The program would have to tell the CAD software to complete each of the steps that we have defined for modeling the gears and also tell the CAD software where to find each of the dimensions and parameters. The biggest foreseeable problem will be in the use of the parametric equations in conjunction with the appropriate parameters.

6.1.2) Investigate the Mating of Bevel Gears

In order to fully validate the accuracy of the bevel gear modeling process, it would be useful to further investigate the gear mating tool in SolidWorks. The models that have been created using our method appear to have the correct final geometries, but do not mesh when imported into an assembly. Investigating the correct mates to use within an assembly is required to know whether or not the bevel gear pairs are accurate.

6.1.3) Defining the Spiral Angle

We believe that defining a series of points in terms of x, y, and z coordinates along the spiral curve would be the best way to define the path to cut along when modeling the spiral bevel gears. Since the path has not yet been fully defined, a gear set has not been made. Further research should be done on a spiral curve in three dimensions. With a better understanding of a spiral curve, a parametrically defined spiral bevel gear could be made. The second area of research would be gear mating the spiral bevel gears in SolidWorks. This would validate the design of the gears, making sure they mate correctly and have little to no interference.

7) Appendices

7.1) Appendix 1: Spur Gear Equations

Outside Diameter: D_O is a given value

Number of Teeth: N is a given value

Pitch Diameter: D is a given Value

Pressure Angle: φ is a given value (usually 20 degrees)

Diametral Pitch: $P = N/D$

Base Diameter: $D_b = D*\cos(\varphi)$

Root Diameter: $D_R = D_o\text{-}2h_t$

Whole Depth: $h_t = 2.2/P + .002$

Addendum: $a = 1/P$

Dedendum: $b = h_t – 2*a$

Auxiliary Angle: $\varphi_A = \cos^{-1}((D_b/2)/((D/2)+a))$ *(When calculating for the gear use pinion values for all variables and vice versa)*

Interval of Contact: $u = ((D/2)+a)*\sin(\varphi_A)-(D/2)*\sin(\varphi)$ *(When calculating for the gear use pinion value of D and vice versa)*

Roll Angle at Theoretical Limit Radius: $\epsilon_l = \tan^{-1}(((D/2)\sin(\varphi) - u)/(Db/2))$

Theoretical Limit Radius: $r_l = (D_b/2)/\cos(\epsilon_l)$

Form Diameter: $d_f = 2(r_l - .025*(.025/P))$

Circular Tooth Thickness: $t = \pi /2*P$

Fillet Radius: $r_f = .3/P$

7.2) Appendix 2: Bevel Gear Equations

(Symbols with the subscript "G" or "P" are for the gear or the pinion respectively)

Number of Teeth: N is a given value

Diametral Pitch: P_d is a given value

Shaft Angle: Σ is a given value

Pressure Angle: φ is a given value (usually 20 degrees)

Ratio: $m_G=N_G/N_P$

Pitch Diameter: $D = N/P$

Pitch Angle: Pinion: $\Upsilon = \tan^{-1}(N_P/N_G)$; Gear: $\Gamma = \Sigma-\Upsilon$

Outer Cone Distance: $A_o=1/2(D_P^2+D_G^2)^{1/2}$

Face Width: $F = A_o/3$ or $F = 10/P_d$ *(use the smaller value)*

Equivalent 90° Ratio: $m_{90}=m_G$

Working Depth: $h_k = 2.0/P_d$

Addendum (at heel of tooth): Gear: $a_{oG}=(.54/P_d)+(.460/P_d*m_{90}^2)$; Pinion: $a_{oP}=h_k-a_{oG}$

Whole Depth: $h_t=2.188/P_d+.002$

Dedendum (at heel of tooth): Gear: $b_{oG}=h_t-a_{oG}$; Pinion: $b_{oP}=h_t-a_{oP}$

Dedendum Angle: Gear: $\delta_G=\tan^{-1}(b_{oG}/A_o)$; Pinion: $\delta_P=\tan^{-1}(b_{oP}/A_o)$

Face Angle: Gear: $\Gamma_o=\Gamma+\delta_G$; Pinion: $\Upsilon_o=\Upsilon+\delta_P$

Outside Diameter: Gear: $D_{oG}=D_G+2a_{oG}*\cos(\Gamma)$; Pinion: $D_{oP}=D_P+2a_{oP}*\cos(\Upsilon)$

Pitch Cone Apex to Crown: Gear: $X_{oG}=D_P/2-a_{oG}*\sin(\Gamma)$; Pinion: $X_{oP}=D_G/2-a_{oP}*\sin(\Upsilon)$

Circular Pitch: $p=\pi/P_d$

Root Angle: Gear: $\Gamma_R=\Gamma-\delta_G$; Pinion: $\Upsilon_R=\Upsilon-\delta_P$

Back-Angle Distance = A_o

7.3) Appendix 3: Spiral Bevel Gear Equations

(Symbols with the subscript "G" or "P" are for the gear or the pinion respectively)

Number of Teeth: N is a given value

Diametral Pitch: P_d is a given value

Shaft Angle: Σ is a given value

Pressure Angle: φ is a given value (usually 20 degrees)

Ratio: $m_G=N_G/N_P$

Pitch Diameter: D = N/P

Pitch Angle: Pinion: $\Upsilon = \tan^{-1}(N_G/N_P)$; Gear: $\Gamma = \Sigma-\Upsilon$

Outer Cone Distance: $A_o=1/2(D_P^2+D_G^2)^{1/2}$

Face Width: $F = A_o/3$ or $F = 10/P_d$ *(use the smaller value)*

Equivalent 90° Ratio: $m_{90}=m_G$

Working Depth: $h_k = 1.7/P_d$

Addendum (at heel of tooth): Gear: $a_{oG}=(.46/P_d)+(.390/P_d*m_{90}^2)$; Pinion: $a_{oP}=h_k-a_{oG}$

Whole Depth: $h_t=1.888/P_d$

Dedendum (at heel of tooth): Gear: $b_{oG}=h_t-a_{oG}$; Pinion: $b_{oP}=h_t-a_{oP}$

Dedendum Angle: Gear: $\delta_G=\tan^{-1}(b_{oG}/A_o)$; Pinion: $\delta_P=\tan^{-1}(b_{oP}/A_o)$

Face Angle: Gear: $\Gamma_o=\Gamma+\delta_G$; Pinion: $\Upsilon_o=\Upsilon+\delta_P$

Outside Diameter: Gear: $D_{oG}=D_G+2a_{oG}*\cos(\Gamma)$; Pinion: $D_{oP}=D_P+2a_{oP}*\cos(\Upsilon)$

Pitch Cone Apex to Crown: Gear: $X_{oG}=D_P/2-a_{oG}*\sin(\Gamma)$; Pinion: $X_{oP}=D_G/2-a_{oP}*\sin(\Upsilon)$

Circular Pitch: $p=\pi/P_d$

Root Angle: Gear: $\Gamma_R=\Gamma-\delta_G$; Pinion: $\Upsilon_R=\Upsilon-\delta_P$

Back-Angle Distance = A_o

8) References

All Gear Equations:

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