

Issues of Gear Design Using 3D Solid Modeling Systems

Charles Cooper

More and more gear shops are wrestling with the issue of whether or not solid modeling is right for their gear design work. The Q & A Page of The Gear Industry Home Page™ has had numerous questions asking how to model gears in solid modeling applications such as AutoCAD, Solidworks and Pro/Engineer. Given the problems people have been having, we are presenting the step-by-step process for modeling gears in Pro/Engineer, but first we thought it would be a good idea to explore the question of whether or not you should even try to design gears using Pro/Engineer or any other 3D solid modeling program.

"From a design point of view, we've gotten along for hundreds of years without solid modeling—we don't need it!" These are the words of Gulliver Silvagi, a product design engineer with Ford Motor Co., but they reflect the general opinion among gear designers as to the application of 3-D solid modeling programs like Pro/Engineer, Solidworks and Autocad to their work: Such programs have their uses in both design and manufacturing processes, but initial gear design is not one of them. The issue really comes down to the way gear design programs work versus the way 3D solid modeling programs work, and how they each model gears.



Helical Gear Model. Courtesy of Performance Gear Systems

Gear Design Software. Dedicated gear design software is mathematical in nature. It has to be in order to properly model the involute curve and the tooth profile generated from that curve. Dedicated gear design programs do the calculations necessary to create the true geometry of the gear tooth. This used to be a tedious and time-consuming operation, often taking the gear designer 50 hours or more to perform the calculations. These programs do the same calculations in seconds and so produce a true involute tooth profile quickly and easily.

However, due to their graphical nature, CAD/CAM systems can not do this. "They are graphical modeling tools and there is a finite number of calculations they can perform, a finite number of points they can plot along the involute curve," said Universal Technical Systems President Jack Marathe. "Because of this, the best they can do on their own is approximate the involute tooth profile."

Pat O'Donnell, President of Performance Gear Systems and a consultant who works extensively with Pro/Engineer said, "Pro/Engineer is a great CAD/CAM system, but it was not written to design gears. That's not what it does."

CAD systems approximate shapes such as the involute tooth profile by defining points along a curve and then simply connecting those points with straight lines. The more points you can plot, the smaller the lines are that draw the curve. While they can plot many of the points along the curve, coming close to the involute profile, there is always an error due to the need for the software to approximate using points and lines.

Another area where the mathematical model used by gear design software is preferable is in the design of the gear itself. Taking a helical gear and designing

it in Pro/Engineer as an example, one sees that the method of constructing the three dimensional solid model provides a good approximation, but not a precise duplication, of the true geometry of the gear being designed.

Frank DeSimone, the Product Line Manager for Geometry at Parametric Technology Corporation, the makers of Pro/Engineer, explains that "a helical gear is nothing more than a constant cross-section [of a spur gear] rotated by a helix angle as the profile is swept across the gear width. Since a profile (sketch) is changing in rotary position as it is being swept, it naturally translates into a Pro/Engineer feature, the Variable Section Sweep." According to O'Donnell, this means that there is a cut at the center of the gear's face width and that the two resulting halves are rotated in accordance with the helix angle, slanting the teeth into the characteristic helical shape, transforming the spur gear into a helical. Because the spur gear design was based on the involute tooth design which was, itself, only an approximation of the true involute geometry (see sidebar), this model of a helical gear can only be as good and as accurate as the original approximation.

Gear ratings. Dedicated gear design programs allow you to make a gear that is within the AGMA or ISO quality rating you are designing for. In fact, the standards are already incorporated into many gear design programs. According to Robert Errichello, President of GearTech, a gear design consulting firm, "For rating gears, for bending fatigue and pitting, you want software that will allow you to stick to industrial quality codes. That way you can compare the gear you just designed to the accepted industry standards." Solid modeling programs can't do

MODELING GEARS IN PRO/ENGINEER

On the Q & A Page of **The Gear Industry Home Page™**, we have had several questions about modeling gears in Pro/Engineer. These questions describe problems with modeling the gear geometry, especially involute profiles, helicals, spurs and other forms. For example: "I have an application that calls for a molded sector gear. I will need to create part geometry for this design. I need help in generating an involute tooth profile. Can someone help me with suggestions how to model this in Pro/Engineer?" and "I am trying to model gears using Pro/E and am having a hard time with it..."

With the help of Frank DeSimone, Product Line Manager for Geometry at Parametric Technology Corporation, the authors of Pro/Engineer, and Daniel Gratten, a gear designer and technical specialist at Meritor Automotive, we will endeavor to answer some of these questions.

The Steps Involved. The first step in modeling a gear in Pro/Engineer is to define the involute curve. Once this is done, the gear itself, whether a spur or helical, is simply extruded from the tooth form. In Pro/Engineer, there are two ways to develop this involute tooth profile: You can do it mathematically or you can do it graphically.

The Mathematical Model. According to DeSimone, this is where most people get into trouble. However, he says that the following steps in Pro/Engineer will mathematically define an involute curve. The lines preceded by a /* are embedded notations to guide the designer. The Layout mentioned is Pro/Engineer's version of a template.

/*This first group of relations sets Feature Parameters to values in the Layout "gear_calc_sm.lay"

```
n = num_teeth      md = 1.25/Pd
Pd = diametral_pitch  cp = pi/Pd
a = pressure_angle  ts = cp-tt
Dr = root_diam      fr = fillet_rad
ad = addendum       rll = relief_diam
tt = tooth_thick    D_o = Dp+2*(ad)
Dp = n/Pd           r_b = .5*Dp*cos(a)
```

/*This group of relations is composed of a start angle (alpha) and three simultaneous equations for r, theta, and z, in cylindrical coordinates. /*(alpha) is calculated directly from the geometry defining the involute curve.

$$\alpha = t \cdot \sqrt{D_o^2 / (4r_b^2) - 1}$$

/*(r) is simply the changing length of a string that defines the involute.

$$r = r_b \cdot \sqrt{1 + \alpha^2}$$

/*(theta) is the angle created by the changing length of r, given that the line must always be held tangent to the base circle at (r_b).

$$\theta = 180/\pi \cdot (\alpha - \pi/180 \cdot \text{atan}(\alpha))$$

/*and we want the curve to stay in the same plane, so z = 0

According to DeSimone, this mathematical procedure creates a datum curve, the basis for the involute tooth profile, using an equation. It references the Layout for key parameters but, in the absence of geometry it has to explicitly calculate alpha to give the involute a starting point. The variable t varies from 0 to 1 over the length of the curve and is used as a time variable.

The Graphical Method. What follows is the step-by-step process to create a spur or helical gear ring graphically that was worked-out by Dan Gratten of Meritor Automotive.

The Gear Ring. This ring can be placed on any gear blank and work. Also this helical gear ring could be used on multiple gear blanks. The following describes the process used to create this helical gear ring:

1. Begin by creating datum curves that will define the attributes of the gear. Use Feature, Create, Datum, Curve. Create the four following circles as shown in Fig. 1.

Base Circle Diameter (BASE_CIRCLE_DIA).
Pitch Diameter (PITCH_DIA).
Minor Diameter (MINOR_DIA).
Major Diameter (MAJOR_DIA).

Set all four of these diameters to the values on your gear spec sheet.

2. Select Modify, Dim Cosmetics, Symbol to name these circles as shown in Fig. 1.

3. Create the Parameters:
NUMBER_OF_TEETH.
PRESSURE_ANGLE.
HELIX_ANGLE.
FACE_WIDTH.

Using: SetUp, Parameters, Create, Number. (Enter in values from your gear summary or spec sheet.)

4. Define the gear tooth itself. Begin by creating a datum curve, and define it as shown in Fig. 2. Note that it is not a true involute profile. Using this simpler method, you will obtain about a 98% accurate representation of the tooth without creating an involute.

Tooth Definition: The tooth radius is defined by a radius that has its center lying on the base circle diameter (it is aligned to the base circle diameter). The tooth thickness is defined by two points that intersect the pitch diameter and the tooth radius. The outside of the tooth edge is used from the datum curve MAJOR_DIA and the root of the tooth edge is used from the datum curve MINOR_DIA. The fillet radius at the bottom of the tooth to the root or minor diameter of the tooth is defined in Fig. 2 as sd14 & sd15 and is defined in our gear summaries as the tip radius on the hob. This is not the exact radius.

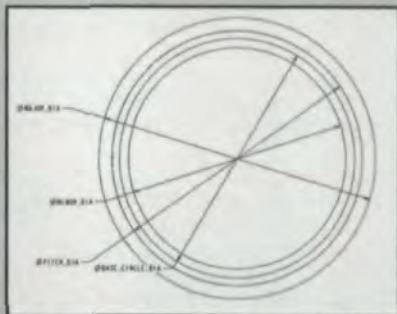


Fig. 1—Gear ring datum curves. Courtesy of Meritor Automotive.

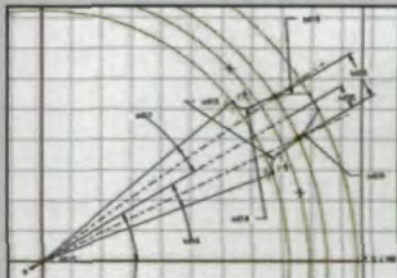


Fig. 2—Gear tooth definition. Courtesy of Meritor Automotive.



Fig. 3—Gear tooth copies. Courtesy of Meritor Automotive.



Fig. 4—Single helical gear tooth. Courtesy of Meritor Automotive.

The radius on the tooth will actually be greater than the tip radius on the hob, but once again we are about 98% accurate. Complete the curve by sketching lines to the center, forming a single tooth using a datum curve. Regenerate the curve and Select OK. Select Modify, Dim Cosmetics, Symbol and add names to the tooth thickness dimension (shown as *sd36* in Fig. 2) and one of the tooth radius dimensions (*sd18*) and the tip radius (*sd14*). Call the tooth thickness **TOOTH_THICKNESS**, the tooth radius **TOOTH_RADIUS** and the tip radius **TIP_RADIUS**.

- Write the following relations into the part:
 $TOOTH_RADIUS = PITCH_DIA / 2 * SIN(PRESSURE_ANGLE)$
 $D35 = TOOTH_THICKNESS / 2$
 $D19 = TOOTH_RADIUS$
 $D15 = TIP_RADIUS$
 $D17 = 180 / NUMBER_OF_TEETH$
 $D16 = 180 / NUMBER_OF_TEETH$

Make sure that you have a datum axis through the center of the diameters before you begin the next step. Then do a dependent copy of this curve and translate/rotate this curve. To do this use: Select, Feature, Copy, Move, Dependent, Done, then Select the Datum Curve, Done, Translate, Crv/Edg/Axis, and Select your center axis, Select OK, enter 1.00, Select Rotate, Crv/Edg/Axis, Select your center axis, Select OK, enter 10.0 for the angle, Select Done, Move, Done and OK.

It is best to create a relation driving these 2 Dimensions at this point. Simply Select Modify and then Select the copied Curve, Find the 10 Degree and the 1.00 Dimensions and then Select Relations and Note the system name for both dimensions. Select Add from the relations menu and type in the following:

$$DXX = FACE_WIDTH / 3$$

$$D?? = 2 * ASIN(DXX * TAN(HELIX_ANGLE) / BASE_CIRCLE_DIA)$$

where D?? is the system name for the angled Dim in the copy and DXX is the system name for the depth of the copy. Regenerate your part.

- Create three copies by patterning the copied tooth using the two dimensions mentioned above as the driving dimensions for the pattern. This will give you a smooth translation to your helical gear. You will now have something that looks like Fig. 3. Then add relations to define the pattern which sets the patterned depth and angle = the DXX and the D?? mentioned above. Regenerate your part.

- Create a datum curve that is aligned to the center axis of the four Diameters at a length of the face width of the gear (This dimension should now be driven by a relation $D?? = FACE_WIDTH$).

- Create your protrusion using Advanced, Swept Blend, Select Sec, Normal to Spline. The trajectory is the straight line curve created above along the center line of the diameters. The sections are the four datum curves created and shown in Fig. 3. Simply Use Geom Tools. Use Edge, Sel Loop on all four datum curves to define your four sections.

Note: Make sure that your start points are the same on all four sections.

- You have now created a single helical tooth for your helical gear ring. This tooth should look something like Fig. 4. Next create a copy of the first tooth using Feature, Copy, Move, Dependent, Done, Select the Protrusion, Done, Rotate, Crv/Edg/Axis and Select the center line axis as your rotating axis and then Select OK and enter your angle (use $360 / NUMBER\ of\ Teeth$), then Done, Move, Done, and OK. Add the relation $D?? = 360 / NUMBER_OF_TEETH$ (where D?? = the system name of the dimension angle just entered above). Regenerate your part.

You can now pattern the copied tooth by using Feature Pattern and then Select the copied tooth and use the angle from the previous step to drive the pattern angle and the number of instances should be your Number of Teeth-1. Add the relations:

$$D?? = 360 / NUMBER_OF_TEETH$$

$$P1 = NUMBER_OF_TEETH - 1$$

Where D?? and P1 are the system names that are given to the dimensions generated by the pattern creation. Regenerate your part. Once you are complete you should have something similar to Fig. 5.

- Lastly, create a coaxial hole at a blind depth of the face width of the gear to turn it into a ring (Fig. 6). Set this blind depth dimension $DXX = FACE_WIDTH$ using a relation and set the diameter of the hole $D?? = MINOR_DIA - 100$ using a relation. Regenerate your part. This we use so that this gear ring can be merged into any gear blank that you have developed. Add the relations to the gear part to drive all the geometry.

Automation. You can automate these steps for your users to simplify this process considerably. Create a Pro/Program out of this gear ring. Add to the gear ring program the information contained in Fig. 7. This can be done by selecting Program from the main menu and then Edit Design. From there simply add the information shown in Fig. 7 in between the input and end input lines. The user merely has to regenerate the gear ring. When he does, the system will prompt him to either use current values or to enter new ones. Simply select Enter, Select All, and then answer the questions that are asked. Once all the information is answered, the part regenerates and the gear is created.

It may be useful to create four such templates: A millimeter and inch version of both a right and left hand helical gear.

Once this gear ring is completed, you can then use the Advance Utilities Function to Merge it into your gear blank. The result is shown in Fig. 8. Also merged into the gear shown in Fig. 8 is an internal spline similar to the helical gear defined above. The helical gear can be converted into a spur gear simply by entering 0 for the helix angle.

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Fig. 5—Helical gear pattern. Courtesy of Meritor Automotive.



Fig. 6—Helical gear ring. Courtesy of Meritor Automotive.

```

INPUT PITCH_DIA NUMBER "WHAT IS THE
GENERATING PITCH DIAMETER?"
NUMBER_OF_TEETH NUMBER "WHAT IS THE
NUMBER OF TEETH?" MAJOR_DIA NUMBER
"WHAT IS THE NOMINAL MAJOR DIAMETER?"
MINOR_DIA NUMBER "WHAT IS THE NOMI-
NAL MINOR DIAMETER?" FACE_WIDTH NUM-
BER "WHAT IS THE FACE WIDTH?"
PRESSURE_ANGLE NUMBER "WHAT IS THE
GENERATING TRANSV. PRESSURE ANGLE?"
HELIX_ANGLE NUMBER "WHAT IS THE GENER-
ATING PITCH HELIX ANGLE?"
TIP_RADIUS NUMBER "WHAT IS THE TIP
RADIUS ON HOB?"
BASE_CIRCLE_DIA NUMBER "WHAT IS THE
BASE CIRCLE DIAMETER?" TOOTH_THICKNESS
NUMBER "WHAT IS THE GENERATING TRANS-
VERSE CIRCULAR TOOTH THICKNESS?"
END INPUT
    
```

Fig. 7—Information for gear ring design automation. Courtesy of Meritor Automotive.



Fig. 8—Helical gear with internal spline. Courtesy of Meritor Automotive.

that for you and so force the designer to go back to the AGMA and/or ISO calculations to get the standards.

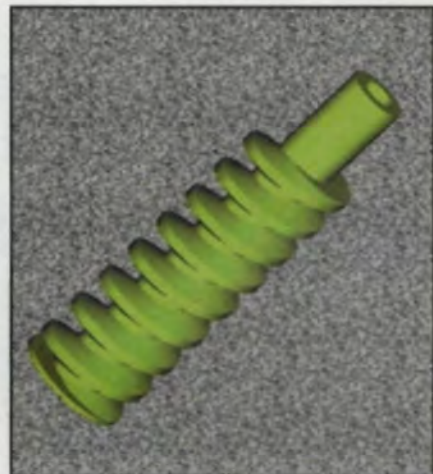
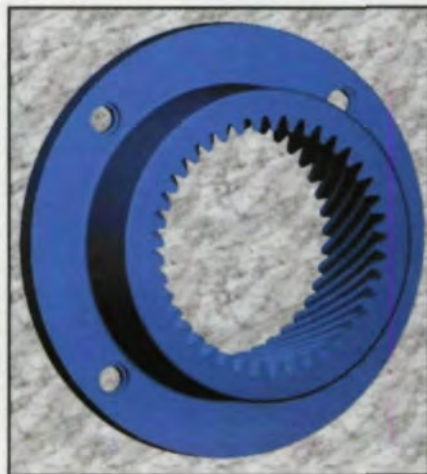
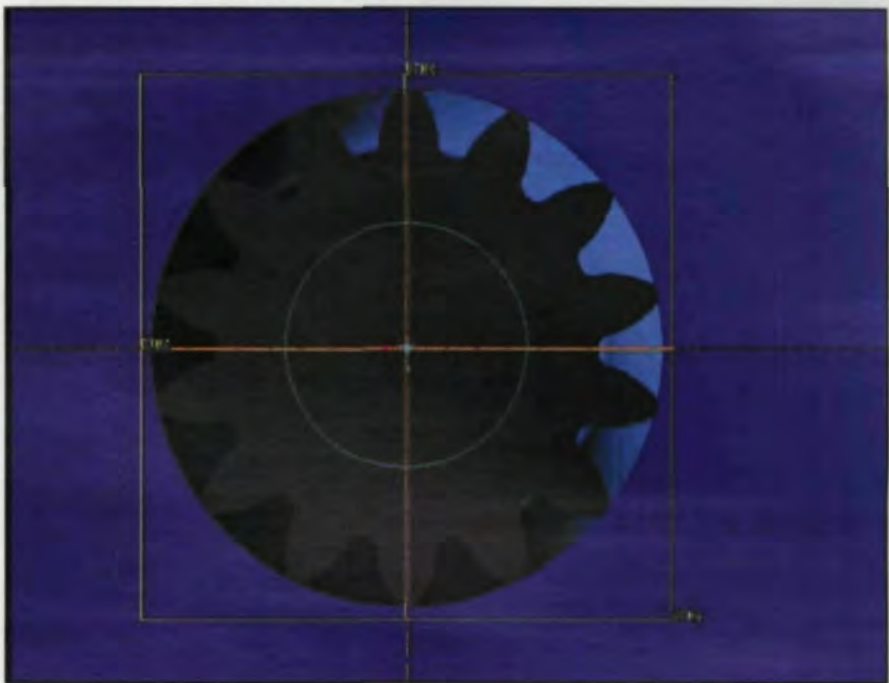
Strengths of 3-D Solid Modeling. One shouldn't believe that three dimensional solid modeling programs don't have a place in the gear industry. They clearly do, but not as gear design tools. According to Errichello, the place for solid modeling is in system design and analysis. "System dynamics analysis is helped by 3D solid modeling." This is especially true when trying to find the torsional natural frequencies of a particular system as well as in the aerospace industry, where gears are very thin and light-weight and have problems with resonance.

For applications like these, solid modeling programs are very useful. Silvagi adds that solid modeling is a good downstream tool, good for defining tool paths for EDMs, lasers and other systems that can draw data from a CAD system. Solid modeling is also the basis for stereolithography and other rapid prototyping systems.

These abilities and applications make modern CAD/CAM systems such as Pro/Engineer powerfully versatile engineering design tools with a great deal to offer the designer. All of this flexibility is made possible by a process called parametric modeling.

Parametric Modeling. Prior to the introduction of parametric modeling, most programs created engineering models via 2-D drawings, 3-D wireframes or 3-D surface models. In each case, full product descriptions, down to the proper dimensioning scheme and tolerances, were impossible. This changed with the introduction of parametric modeling.

Parametric modeling allows the design engineer to let the characteristic parameters of a product drive the design of that product. In the case of gear designers, key dimensions that would describe the gear being designed such as diametral pitch, pressure angle, root radius, web thickness, etc. could be used as the parameters to define the gear. But, the parameters do not have to be geometric. They can also capture key process information such as case hardening specifications, quality grades, metallurgical



Examples of Gears Rendered in Pro/Engineer. Courtesy of Performance Gear Systems

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properties and even load classifications for the gear being designed.

Programs like Pro/Engineer use these parameters, in conjunction with the program's features, to generate the shape of the gear and to add in all the essential information needed to create a product model. In creating a tooth profile, for example, the parametric dimensional information drives the shape of the tooth and non-geometric parametric information specifies things like the required case hardening depth or the nondestructive test requirements.

Since many gears are similar in many respects, Pro/Engineer can capture the differences within a family of parts very easily. For instance, two gears may be identical except for the web thickness and material. With a single product model, both gears are completely described because Pro/Engineer models the baseline design ("generic") and iterations on that design ("instances") via a spreadsheet. Differentiating parameters are characterized in the spreadsheet.

While all of these features and abilities are very useful to the designer, they do not deal with the question of the complexity of the calculations needed to define a gear tooth, nor do they address the tendency toward approximation inherent in CAD/CAM programs. Clearly, something more is needed.

The Synthesis of Gear Design Software and CAD: One Solution.

Given the limitations imposed by most gear design software on visual rendering and by CAD systems on the accuracy of the models, what is the gear designer to do?

Link the Systems. Most gear design software on the market today will export DXF files or X-Y coordinates that can be used by mainstream CAD/CAM software to draw the part. There are also programs that act as bridges between the gear design application and the CAD/CAM system. One such product is DesignLink by UTS.

Developed in conjunction with Jerand Technical Services, DesignLink bridges the gap between UTS gear design and engineering programs that run on its TK Solver platform and PTC's Pro/Engineer. "The combination of the mathematical

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modeling power of TK Solver and 3D graphical modeling power of Pro/Engineer provides a whole new capability for Pro/Engineer users," said Jerand President Bob Monat. "It's a new paradigm, with a unique productivity-boosting capability."

The geometry of the gear is designed with TK Solver and UTS gear software. This includes optimizing the design to increase both the life of the gear and the horsepower rating, reduce noise and cost, and so on. For plastic or powdered metal gears, the UTS software also helps you design molds by adjusting for shrinkage (plastic gears) or growth (powdered metal gears) when the gears come out of the mold.

DesignLink transfers the numeric information from TK Solver to Pro/Engineer, which reads the data as parameters of the model. This includes the number of teeth, face width, outside diameter, helix angle, pitch, pressure angle and coordinates of the tooth profiles for all the teeth. Once the transfer is complete, the 3D model is easily rendered. Since they are changeable parameters in the solid model, users may control the size of the Pro/Engineer model according to the number of points on the tooth profiles. Fewer points may lower precision somewhat, but that will not be a problem unless you are designing tooling such as a mold cavity.

According to Marathe, this combination of gear design and solid modeling software is the only way to accurately create gears in a 3D solid modeling environment. "Gear design involves complex mathematical calculations to get the geometry correct. It is not as simple as drawing an involute and a fillet and joining a few curves together. Accurate calculation of gear profile coordinates makes it easy to use CAD/CAM systems to make molds and other tooling for plastic and powdered metal gears. The whole job then becomes automated." Ⓞ

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