

# Computer Assisted 3 - D Molding Design of Gear Based on SolidWorks Software

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**Abstract:** Based on the need of computer aided design and manufacturing, the method and technique of the parametric order for drawing the plane profile of straight tooth column gear was developed with visual in SolidWorks. Parametric order 3 - D molding design of gear was completed using this method.

**Keywords:** Solidworks, Gear, 3 - D Molding

## 1. Introduction

SolidWorks is a 3-D modeling software, which uses an easy-to-learn Windows user interface to realize parametric design, and can use the combination and transformation of different parameters to achieve 3-D modeling design. SolidWorks has a wealth of part entity modeling functions, it can perform variable sketch outline drawing, and can automatically perform dynamic constraint checking. Design products more easily with operations such as extruding, rotating, thin-walled features, shelling, feature arrays, punching and so on.

Computer-aided modeling technology has been widely used in product design, engineering analysis, rapid prototyping and other technical fields. When applying CAD/CAM technology to design and manufacture gear products, the 3-D modeling of gears is a technical problem that needs to be solved urgently, such as the low precision of gear modeling will directly affect the finite element analysis, the simulation results of virtual prototype design, and the CAM manufacturing precision of gear products. This paper takes the cylindrical helical gear with more complex structure as an example to illustrate the three-dimensional modeling method of the involute gear[1].

## 2. Gear 3-D modeling design method

The key of gear 3-D dimensional modeling is to generate involutes and gear teeth that meet the requirements. There is no direct command for drawing involutes, so the involute tooth profile for parametric design must be generated with the help of a corresponding program.

The rectangular coordinate parametric equation of the involute is:

$$\begin{cases} X = R_b (\cos \theta + \theta \cdot \sin \theta) \\ Y = R_b (\sin \theta - \theta \cdot \cos \theta) \end{cases}$$

In formula, X, Y- rectangular coordinate value of any point of involute;  $R_b$ - radius of involute base circle;  $\theta$ - the rolling angle of the involute on the base circle.

$$\theta_{\max} = \sqrt{R_a^2 - R_b^2} / R_b$$

The range of the rolling angle in the parameter equation of the involute of the gear segment teeth is  $0 \leq \theta \leq \theta_{\max}$ . According to the parameter equation of the involute, the coordinate values of a series of points on the involute are calculated, and then the command of drawing a straight line is quoted to replace the involute by drawing a straight line through the points on the required involute. When enough points are found on the involute, an exact involute profile curve is drawn[2].

As long as one involute profile curve is drawn, the profile curve of the other half of the gear can be mirrored on the base circle, and then all the profile curves can be drawn according to the z-circle array of teeth.

The teeth are generated by the involute. To mirror the symmetrical tooth profile curve of the other side, the position of the mirror axis OP must be determined. Set the angle between radial axis OP and X to be  $\lambda$ , The arc length of the AC point is the tooth thickness  $S_b$  of the gear base circle, then the calculation formula of the tooth thickness of the base circle is:

$$S_b = \frac{\pi}{2} \cdot R_b + 2R_b \cdot \sin \alpha$$

$$\lambda = \frac{S_b}{2R_b} = \frac{\pi}{2z} + \operatorname{tg} \alpha - \alpha$$

In formula: z-number of teeth;  $\alpha$ - pressure angle

### 3. Curve generation program

Make a program in computer language to generate each point on the involute and save it as a data file with the extension .txt. This article takes C language as an example to illustrate the program for generating the curve.

```
#include <stdio.h>
#include <stdlib.h>
#include <math.h>
#include <conio.h>
#include <graphics.h>
main()
{ FILE *fp;
char name[20],buff[20];
int gd=DETECT,gm,i,zn;
float mn,rb;
float r,x,y,rk,thk,alphk;
printf("Input File name:"); /*Prompt for generate the data file name and extension, eg,
X50.txt*/
scanf("%12s",name); /*Wait to enter the file name*/
printf("\nInput Mn:"); /*Prompt for input the gear module, such as 3*/
scanf("%12s",buff); mn=atof(buff); /*Wait to input the gear modulus*/
printf("\nInput Zn:"); /*Prompts for input the gear teeth number */
scanf("%12s",buff); zn=atoi(buff); /*Wait to input the gear teeth number */
rb=mn*zn*cos(20*M_PI/180)/2; /*Calculate the involute base circle*/
initgraph(&gd,&gm,"");
fp=fopen(name,"w");
setcolor(15);
circle(320,240,(int)(rb*2+0.5)); /*Draw the base circle*/
for(alphk=0,i=0;alphk<M_PI/2;alphk+=0.02,i++)
{ thk=tan(alphk)-alphk;
```

```

rk=rb/cos(alphk);
x=rk*cos(thk);y=rk*sin(thk);    /*Calculate X and Y coordinates*/
r=sqrt(x*x+y*y);
putpixel(320+x*2+0.5,240-y*2-0.5,15);
fprintf(fp,"%f\t%f\t%d\n",x,y,0);
if(r>(zn+2)*mn/2) break;
}
fclose(fp);
getch();
restorecrtmode();
}

```

#### 4. Design example of cylindrical helical gear

Cylindrical helical gear: module  $m=80$ , number of teeth  $Z=3$ , helix angle  $\beta=15^\circ$ . The design process is described below.

##### 4.1 Draw gear main body

First the sketch drawn on the right-view datum is shown in Figure 1, and the gear body is generated with the "rotating convex/base" function, then the draft angle, cut spoke holes and chamfers are drawn, as shown in Figure 2.

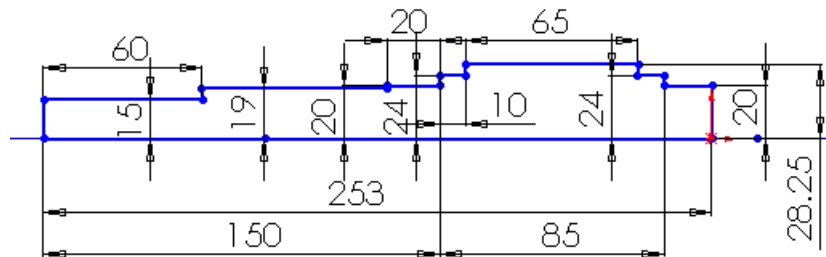


Figure 1: Sketch of the gear

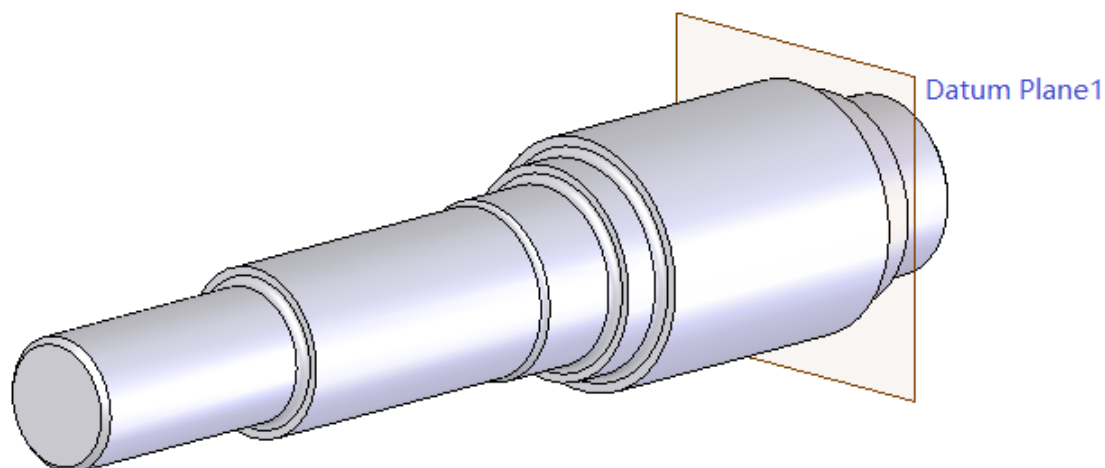


Figure 2: Main body of the gear

##### 4.2 Generate involute data file

Execute the above program, input the file named Z\_3\_20.txt, the modulus is 3, and the number of

teeth is 20. Generate a data file named Z\_3\_20.txt.

Click the "Curve through xyz points" option in the "Curve" cascade menu in the "Insert" drop-down menu. The dialog box shown in Figure 3 will pop up → open the gear involute data file Z\_3\_20.txt through the "Browse" button → click the "OK" button to form an involute curve in the figure.

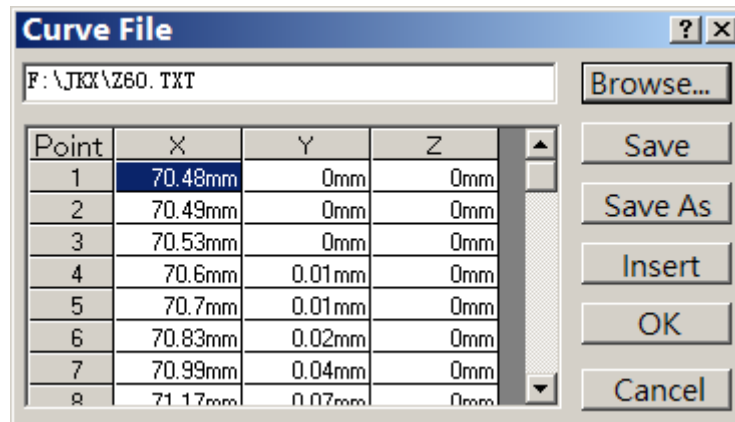


Figure 3: The inserted curvilinear dialog box getting through xyz-point

### 4.3 Draw gears

#### 4.3.1 Draw gear tooth curve

(1) Insert an involute curve. Click the "Curve through x yz point" option in the "Curve" cascade menu in the "Insert" drop-down menu, and the dialog box of Figure 3 will pop up. Click the "Browse" button to open the involute file, click the "OK" button in the dialog box, and the involute appears on the screen[3-4].

(2) Draw the gear tooth curve. Draw a sketch on the front view datum plane, use the Convert Entity function to convert the involute into a sketch curve, draw 2 centerline rays through the center, draw a centerline of the index circle, draw a point with the index circle and the involute. The line sketches are coincident, and the angle of the two center rays is marked as 4.5 degrees. Symmetric this curve with the centerline and draw an addendum circle with a diameter of 66. Use the Convert Solids function to convert cylindrical faces (root circles) to sketch curves. Use the crop function to cut the useless lines to form the outline of the gear teeth, as shown in Figure 4.

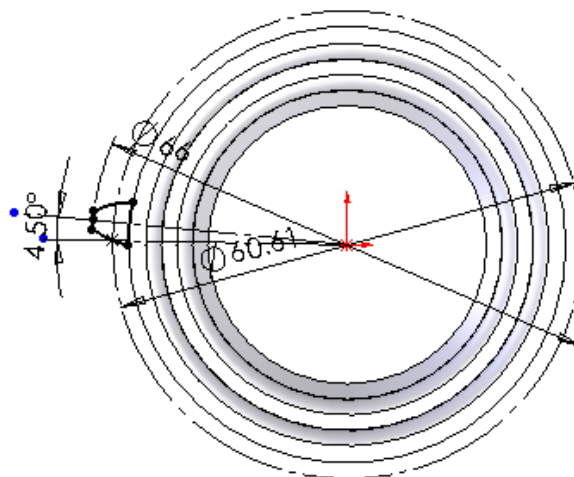


Figure 4: Sketch of the gear teet

#### 4.3.2 Draw gear helix line

Select the "Spiral/Swirl" option in the "Curve" cascade menu in the "Insert" drop-down menu, the dialog box shown in Figure 5 will pop up, and the helix will be established according to the requirements of the direction of rotation and lead.

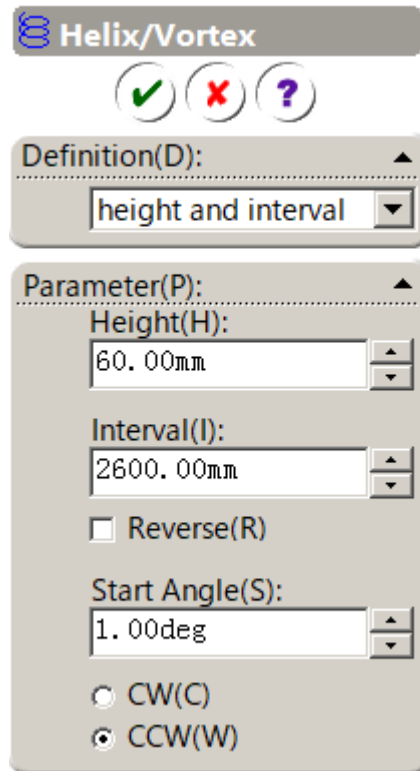


Figure 5: Dialog box of the helix line

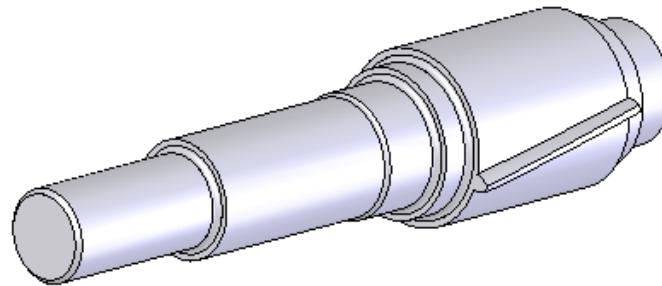


Figure 6: Entity of the single helical tooth

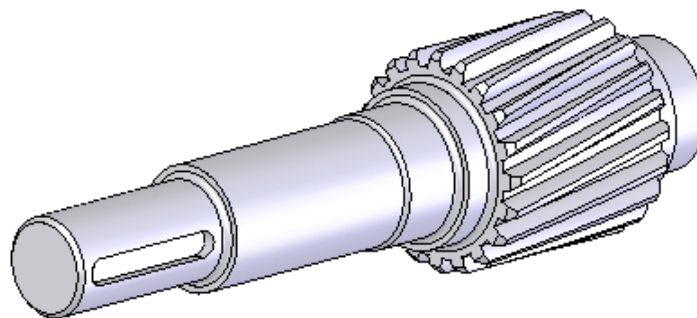


Figure 7: Entity of the gear

#### 4.3.3 Generate a single tooth entity

Use the "Extrude Boss/Base" function to extrude into gear teeth, and choose to shape to the next side in the termination condition box. The formed gear teeth are shown in Figure 6.

#### 4.3.4 Generate a complete teeth entity

All gear teeth are generated by the circular array method, and the generated gear is formed through the solid cutting of holes and keyways, and the chamfering of the gear teeth to form a complete teeth gear. As shown in Figure 7.

## 5. Conclusions

Taking cylindrical helical gears as an example, this paper introduces the process of parametric drawing of gears, which solves the problem of parametric design of three-dimensional modeling of gears, which can be applied to the design/manufacture of gears with CAD/CAM technology.

## Acknowledgements

Fund Project: 1) Engineering Research Center of Safe Exploitation and Comprehensive Utilization of Mineral Resources in Universities of Inner Mongolia Autonomous Region (2021KZYB02)

2) Scientific Research Key Project of Hulunbuir University (2022FZD05)

3) Teaching Research Project at the 8th session of Hulunbuir University (JYYB2021034)

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