Research on the Parametric Design of Involute Helical Gear Based on Pro/Program

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Abstract
Involute helical gear has advantages of larger overlap coefficient, steady transmission, lower impact and noise, higher load bearing capacity, longer life, so it is widely used in machinery, mining, metallurgy, aerospace and other fields. However, due to the involute helical gear tooth profile shape is more complex, and some low-level CAD software is difficult to directly generate a three-dimensional helical gear tooth profile. According to involute helical gear formation principle, this paper presents an effective method to establish the three-dimensional model for involute helical gear. In this paper, a parametric design method of realizing involute helical gear 3D model by using Pro/E modeling functions and Pro/Program programming is discussed. This method can enable designers to obtain helical gear 3D solid model quickly and accurately by inputting a little geometry characteristic parameters. It can also enables designers quickly update design based on the existing 3D gear model, so as to improve the design efficiency.

Keywords: Pro/Program, Helical Gear, Parametric Design

1. Introduction
Because of high transmission efficiency, compact structure, reliable working life, gear transmission is widely used in all kinds of mechanical drive system. However, many gear types, complex shape, cumbersome size gives designers a heavy duplication of work. Therefore, how to improve the design efficiency of the gears has become the primary solution to the CAD/CAM users[1].

Pro/E is a large engineering software which is marketed by Parametric Technology Corporation (Abbreviations PTC ), which integrate the part design and assembly of products, mold development, drawing production, sheet metal design, modeling design, CNC design, mechanism design/analysis, the dynamic simulation, structural analysis, and heat flow analysis, casting parts design, cooperative design development specially-featured avenue. Pro/E has many modules, and all modules are related, it can be described a full three-dimensional product development software. Since its inception in 1988, Pro/Engineer become the most popular three-dimensional CAD/CAM systems with standard software and widely used in electronics, communications, machinery, automotive, aerospace, home appliances, toys and other industries.

Designers can use CAD software to design the 3 d parametric model of gears. Parametric design [2-3] refers to automatically complete the corresponding modification of graphics by changing some sizes of the graphics, so as to achieve rapid design of similar structure mechanical products.

Pro/E is a widely used excellent CAD software, it has a powerful 3D modeling, parametric design of the product can be easily implemented, thus greatly improving the design efficiency[4-8]. When users design 3d gear model in Pro/E, the Program will record gear main design steps and the size parameter list in the form of a program. The user can edit and modify the program as desired. As long as the user re-run the program and change the parameters of the gear a new gear will be created, so that designers who not familiar with 3D modeling techniques can quickly update the design based on existing 3D gear model, these shorten the design cycle greatly, improve the design efficiency and calculation accuracy. This paper introduces a design method of realizing involute helical gear 3D model by using Pro/E modeling functions and Pro/Program programming.

2. The 3D-model design of helical gear
The main part of involute helical gear is a cylinder, the outline is involute tooth profile surface. These features are the main characteristics of the gear parts, such as the axle hole and keyway features
are dependent on these characteristics, known as additional features. When we design the 3D-model of helical gear Using Pro/E, we first customize the basic parameters of helical gear, which associated with gear size, shape, location, each parameter change will cause gear changed.

### 2.1 Establish the basic parameters of helical gear in Pro/E

The parameterization of helical gear basic parameters is to meet the demands of Pro/E editing program. In Pro/E for helical gear modeling, we should first set working directory into the part design mode, open the [Parameters] dialog box, and then click add as shown in Table 1 helical gear basic parameters and initial value.

<table>
<thead>
<tr>
<th>Parameter symbol</th>
<th>Parameter name</th>
<th>Parameter type</th>
<th>Initial value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>tooth number</td>
<td>Integer</td>
<td>20</td>
</tr>
<tr>
<td>MN</td>
<td>normal module</td>
<td>Real Number</td>
<td>2</td>
</tr>
<tr>
<td>ALPHA</td>
<td>normal pressure angle</td>
<td>Real Number</td>
<td>20</td>
</tr>
<tr>
<td>BETA</td>
<td>normal spiral angle</td>
<td>Real Number</td>
<td>16</td>
</tr>
<tr>
<td>HAX</td>
<td>normal addendum coefficient</td>
<td>Real Number</td>
<td>1</td>
</tr>
<tr>
<td>CX</td>
<td>normal clearance coefficient</td>
<td>Real Number</td>
<td>0.25</td>
</tr>
<tr>
<td>B</td>
<td>tooth width</td>
<td>Real Number</td>
<td>100</td>
</tr>
</tbody>
</table>

2.2 Draw the addendum circle, tooth root circle, pitch circle and base circle

Through the sketch tool in the toolbar, draw four concentric circles on the FRONT plane, as the basic circle of the gear (addendum circle, root circle, pitch circle and base circle), as shown in Figure 1. In the menu manager choose "program" / "edit design", open editor from model, between Relation and End Relation add the following equation (comments part do not enter)

- \(a=\text{atan} (\text{tan}(\text{ALPHA})/\cos(BETA))\) /* a is end face pressure angle
- \(d=mn*z/\cos(BETA)\) /* d is pitch circle diameter,d1=d
- \(da=d+2*mn\) /*da is addendum circle diameter,d0=da
- \(db=d*\cos(a)\) /*db is base circle diameter,d2=db
- \(df=d-2*mn*1.25\) /*df is root circle diameter,d3=df

---

**Fig. 1.** The basic circle of helical gear

**Fig. 2.** Involute tooth profile curve
2.3 Generated tooth profile datum curve curve

We click on the button (the inserting datum curve icon) in toolbar, select "fom equation → finish→ choose "Cartesian coordinate system“→open “rel.pptd” file→input the following involute curve’s parameters equation →finish, to create a involute curve.

\[
\begin{align*}
\text{ang} &= \text{t}*90 \\
\text{rb} &= \text{db}/2 \\
\text{s} &= \pi \text{rb}\text{t}/2 \\
\text{x0} &= \text{rb}\cos(\text{ang}) \\
\text{yo} &= \text{rb}\sin(\text{ang}) \\
\text{x} &= \text{x0}+\text{s}\sin(\text{ang}) \\
\text{y} &= \text{yo}-\text{s}\cos(\text{ang}) \\
\text{z} &= 0
\end{align*}
\]

On the basis of analyzing and calculating transverse tooth profile for geometry, we identify and draw its symmetrical line, obtain the other side of involute through mirror. In the menu manager, we select “feature”→duplicate→mirror image, the other side of involute has been drawn. Under "Sketch" model, by "through the side to create figure" and "round" tools we can generate a single tooth space profile section, as shown in Figure 2.

2.4 Three-dimensional modeling of the helical gear

3D modeling procedure of helical gear ‘gear tooth is as follows:
(1) Using “stretching” command to generate the addendum circle entity
(2) By “scanning mixed” command cut out the first tooth space, as shown in Figure 3 (We should choose "constant normal direction" option, draw out of the top end involute tooth slot section, and then create the other side of involute tooth slot section through the features operation "copy" command).
(3) Use the tool of “copy feature” to generate the second tooth slot.
(4) Use “array” tools array out all gear tooth (Figure 4). To facilitate modification of the model, the following relations will be added in the course of arraying

\[
\begin{align*}
\text{d#} &= 360/\text{z} \\
\text{d#} &= 360/\text{z} \\
\text{p#} &= \text{z} \\
\end{align*}
\]

2.5 The structure design of the helical gear

(1) The axle hole and keyway will be created by extrude command, the cross-section as shown in Figure 5, the finished cut features as shown in Figure 6.
(2) Editing program, so that the user can input shaft hole and keyway dimensions. Click Tools→Program→Edit Design→appear notepad editor, entering the following program between INPUT and END INPUT, save the document and close the editor.

```
INPUT
SHAFT_DIA NUMBER
"Enter the shaft diameter:"
KEY_WIDTH NUMBER
"Enter the key width:"
KEY_HEIGHT NUMBER
"Enter the key height:"
END INPUT
```

Information window will appear “Do you want to incorporate your changes into model:[yes]”, choose yes to program into the model.

(3) Adding relations to control axle hole and keyway dimensions

We selected the previously built keyway features in model tree, right-click→edit definition → into the sketch environment → tools → relations→open dialog box, for the keyway size sd0, sd1, sd2 add as below relationship in Figure 7.

Fig.5. The cross-section of axle hole and keyway  Fig.6. The finished cut features

Fig.7. Adding relations for the size of axle hole and keyway
Click regeneration command → input → Select all → finished selection, as shown in Figure 8(a), the keyway shaft hole diameter, keyway width and height can be changed by entering different SHAFT_DIA, KEY_WIDTH and KEY_HEIGHT value. By changing the initial value 60, 10, 7 to new value 80, 20, 10 in turn, the new keyway size will be produced as shown in Figure 8(b).

(a) The menu dialog box (b) The schematic diagram of new keyway size
Fig.8. Adding relations for the size of axle hole and keyway

3. Combined with Pro/Program realization gear parametric programs design

When the helical gear design requirements changes, the structure size is also should make a corresponding change to meet new needs. To meet the requirement, we will use the Pro/Program to realize parametric automatic modelling of the helical gear. As long as we modify the feature parameters, the entity model can be converted to the required new gear.

3.1 Edit parametric design programs of the gear

In the list of commands shown, we select menu commands “Program” → “Edit Design”, open the application editor, input the following program, then save and close the file to realize the inputs of gear’s feature parameters.

```
INPUT
SHAFT_DIA NUMBER
"Enter the shaft diameter:"
KEY_WIDTH NUMBER
"Enter the key width:"
KEY_HEIGHT NUMBER
"Enter the key height:"
Z NUMBER
"Enter the tooth number:"
MN NUMBER
"Enter the normal module:"
BETA NUMBER
"Enter the spiral angle:"
ALPHA NUMBER
"Enter the normal pressure angle:"
B NUMBER
"Enter the tooth width:"
END INPUT
```
3.2 Run the design program of helical gear

After you have finished editing the helical gear’s parameter design program, just call the “Regenerate” command, Pro/E will automatically execute this program. A dialog box displays, requesting the new value of the parameter. In this dialog, you have to provide a value. Though, of course, we can use the following method.

Click Tools→Program→Edit Design→Enter→Select all→finished selection, as shown in Figure 9.

If users input 60,10,7,20,16,20,100 in turn, it will automatically create a new helical gear(shaft diameter SHAFT_DIA=60, key width KEY_WIDTH=10, key height KEY_HEIGHT=7, tooth number Z =20, module MN = 2, spiral angle BETA = 16, pressure Angle ALPHA = 20 °, tooth width B=100),as shown in Figure 11 (a).

If users input new parameter value, Pro/E will according to the new data inputting generate new gear. For example, we select “Regenerate” tools and "enter" command, this launches a dialog that asks which parameter you would like to select,we select all, according to information window , we input 150,20,10,40,10,16,20,100 respectively(as shown in Figure 10), it will automatically create a new helical gear as shown in Figure 11 (b).

Fig.9. Adding relations for the size of axle hole and keyway

Fig.10. The schematic diagram of assigning new value for the parameters
4. Conclusions

In this paper, based on the Pro/E, we introduce the basic thought and realization method of 3D parameterization model. According to the research on the 3D parameterized solid model of involute helical gear, some conclusions were summarized:

(1) The 3D parameterized solid model of the involute helical gear can be achieved and updated easily based on Pro/E’s software module Pro/Program.

(2) This parameterized technology is realized by means of the program-parameterized and dimension-driven methods, and the 3D modeling by means of the Pro/E’s modeling function.

(3) This design method can make the helical gear structural form and design accuracy to achieve a satisfactory result, which is suitable for engineering application.

(4) Compared with conventional design method, this method can improve design efficiency and shorten the product development cycle.

5. Acknowledgements

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6. References