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Computer Aided Modelling and Meshing of Spur Gear (Stub Teeth) in a CAD Software IDEAS

Vijay Kumar Karma*
Department of Mechanical Engineering
Institute of Engineering & Technology
Devi Ahilya Vishwavidyalaya, Indore, MP, India

Govind Maheshwari
Department of Mechanical Engineering
Institute of Engineering & Technology
Devi Ahilya Vishwavidyalaya, Indore, MP, India

Abstract
Spur gears are most common power transmission elements used for various applications including gearboxes of machines and automobiles. For the purpose of analysis, it is now a common practice to use the CAD software to model the gears. This modeled gears are used further to perform the various tasks such as finite element analysis etc. This paper illustrates the modeling and meshing procedure of spur gear having stub teeth in a CAD Software IDEAS. The gear is model with standard dimensions of stub teeth having involute tooth form, since it is commonly used. The modelled gears are then assembled to form combination of gear pairs.

Keywords*: CAD, Spur gear, IDEAS, Sub teeth, Involute.*

****Author for correspondence*** vkarma@ietdavv.edu.in

1. Introduction

Gears are the main component of power transmission units such as automobiles, machines etc. They are classified in many ways such as parallel axis gears, inclined axis gears etc. Spur gear belongs to the category of parallel axis gears [1]. To improve the design and manufacturing of the gears research is going on to study it with the help of computer technology. That is various CAD software are used along with the different computer technology to model and analyse the gears, In these paper a procedure is discussed to model the spur gear and meshed to form a gear pair. Rathod et al. (2011) presented a procedure to make parametric model of spur gear using Pro-Engineer Wildfire 5.0. A finite element procedure was developed by Sfakiotakis & Anifantis (2002) to face the general computational design problem of a cracked gear currently engaged in a gear mesh. It was found that the incremental solution of the problem yields the fracture characterizing parameters with respect to the contact evolution. Parey & Tandon (2003) presented a review of the dynamic modeling of spur gears including defects. They have also given the classification of gear dynamic models by different authors, and explained the various terms used in gear dynamic modeling. Haraga (2006) presented the steps of 3D modeling of a spur gearing with straight teeth, using Solid Edge software. Huston et al. (1989) presented procedures for computer-modeling of spur gear tooth fabrication. They show that the standard involute tooth form results from a cutter with an involute shape rolling onto a gear blank. They also give examples and applications were discussed. Hefeng et al. (1985) presented a general method for describing external involute spur gears produced from the basic rack form. The description results in a computer graphic drawing of the cut tooth as an individual tooth, several teeth in a segment or a complete gear. It was also established that the surface normal vector at the cutting point must pass through the instant center. They derived the equations describing the tooth root, fillet, involute and top land based on this fact. The tooth description is based on the tooth addendum and dedendum, the number of teeth on the gear, the rack pressure angle, the diametral pitch and the rack tip radius. Singal (2002) developed a program for the Involute equation to develop the Spur Gears in Pro-Engineer software.

2. Modelling Procedure

The complete procedure, to model the spur gear, is described below. The gear is governed by the standard values of other related parameters of stub teeth, which are mentioned in Table 1. The fillet portion below the base circle is a standard fillet of radius of 0.4*m.

Table 1: Gear Parameters

Parameter	**Value**
Module	m
No. of teeth	Z
Pitch Circle Diameter (PCD)	m*Z
Pressure angle	φ
Addendum	0.8*m
Dedendum	m
Base Circle Diameter (BCD)	PCD * cos φ
Circular Pitch (CP)	π*m
Tooth Thickness	CP/2

The step-wise procedure for modelling the Master Gear (for a sample case with m=5, Z=20, phi=20) is as under.

1. First, the default units in I-DEAS were checked to be sure that **mm** was measuring unit of length.
2. Using line command a horizontal line was drawn with (0,0) as start point and of length 5 mm. This line was constrained to fix horizontally. It was also constrained to pivot at (0,0) point.
3. Another line was drawn using line command with (0,0) as start point (In I-DEAS, this point automatically get selected when cursor is brought near to the start point) and in vertical direction having endpoint as (0,20). This line was constrained as vertical and the endpoint (0,0) was constrained as X. Both these lines were dimensioned and the names of dimensions were given as “m” and “Z” for horizontal and vertical lines respectively.
4. An inclined line, about 100 mm length, was drawn from (0,0) as start point. This line was angularly dimensioned with vertical line and the angle was designated as “phi”.
5. These three dimensions were fundamental dimensions and all other dimensions were derived dimensions and hence by changing these dimensions the gear model was fully parameterised.
6. A circle was drawn with (0,0) as centre and of any suitable diameter/radius. The dimension (diameter/radius) was modified and named as “PCD”. In modify form, the value was modified and formula for pitch circle diameter in appropriate unit was written

as "m*Z".The centre of this circle was constrained at (0,0) point of horizontal (module) line.

7. Another circle was drawn with (0,0) centre and of any suitable diameter. The diameter was modified and designated as "DCD", its value was modified and formula "PCD – 2.5*m" was written. This dedendum circle was made concentric with PCD circle using the constraining.
8. In the same manner base circle diameter was drawn concentric with PCD circle.
9. Function spline was drawn from the base circle using parametric equations of the involute curve, as under:

$$x = rb\,(\sin\Theta - \Theta^*\cos\Theta) \quad (1)$$

$$y = rb\,(\cos\Theta + \Theta^*\sin\Theta) \quad (2)$$

$$z = 0 \quad (3)$$

where, x and y are co-ordinates of involute curve and rb is base circle radius, Θ is roll angle.

10. The x, y and z co-ordinates were taken from above equations. For this, u co-ordiante (as required in function spline form) was taken from 0 to 1 for right handed involute and -1 to 0 for left handed involute.
11. An inclined line was drawn from (0,0) to certain angle to vertical, its end point was constrained on the pitch circle diameter.
12. The angle from vertical line was taken as equal to 360/Z/2/2 i.e. 4.5° on right hand side. Similarly on the left hand side another line was drawn at the same angle and same constraining was done.
13. The involute curves were rotated by using the vector rotations, first vector was taken as the vertical line (drawn in step 3) and another vector was the inclined line at 360/Z/2/2 i.e. 4.5°.
14. In this way the two curves were now shifted so that the angle subtended at the centre from points on the involute curves intersecting the pitch circle diameter was equal to 360/Z/2° i.e. 9°.
15. The involute profile started from the base circle and extended above the addendum circle is shown in Figure 1.
16. For completing the wire frame of the complete tooth profile, fillet was required below the base circle.
17. A vertical straight line was drawn starting from end point of involute curve from base circle of a convenient length.
18. A circle was drawn of any radius. The dimension (diameter) was modified and designated as "Fr" and was given fillet radius value of "0.4*m".
19. The above circle was made tangentially constrained to line drawn in previous step and also to base circle. This ensured that the fillet always remained tangential to the base circle.
20. Similarly on the other side vertical line and circle were drawn.

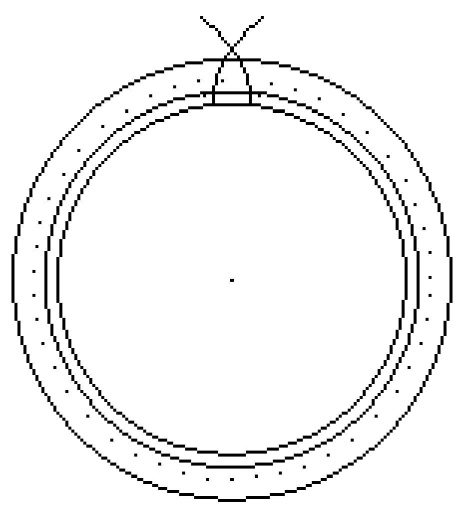
Fig. 1: Generation of Involute Profile

21. These were again made tangential as explained in the previous step.

22. The two fillets were ready.
23. In this way the complete wire frame for one tooth was ready as shown in Figure 2.
24. Using ***extrude*** command this wire frame was selected and extruded to a distance of 5 mm.
25. Single Tooth having all the standard proportions as mentioned in Table 1 was ready as shown in Figure 3.
26. This tooth was patterned, using ***pattern*** command.
27. The different values required in the patterning form were: center point (0,0), Radius (default), Number on circumference (number of teeth) and Total angle (360º).
28. A disc of base circle diameter was again drawn and extruded by a distance of 5 mm. It was joined to extruded and patterned teeth.
29. In this way the Gear was ready. Figure 4 and Figure 5 represent the gear in wire frame and filled mode respectively.

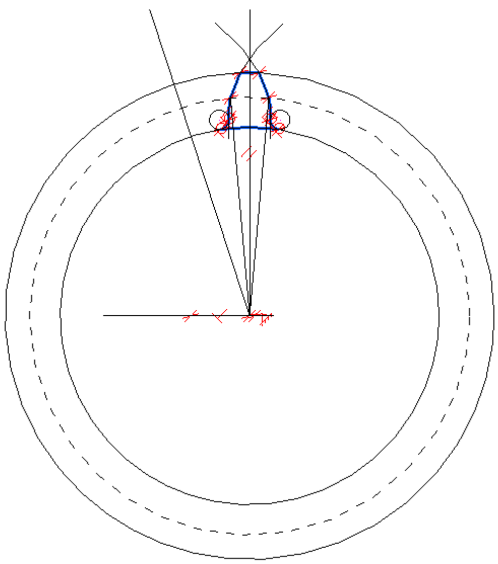

Fig. 2: Wire frame of tooth

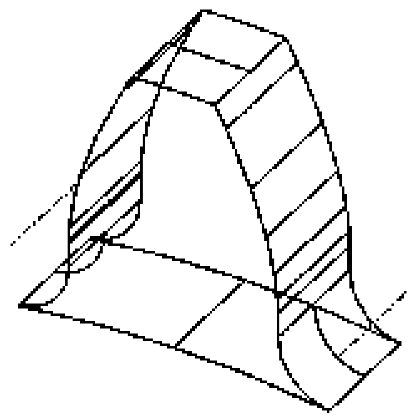

Fig. 3: Extruded tooth

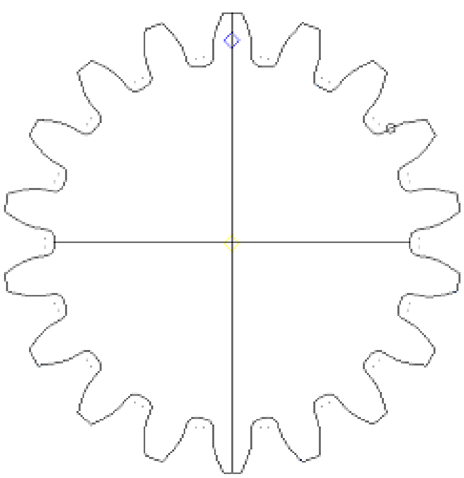

Fig. 4: Gear in wire frame mode

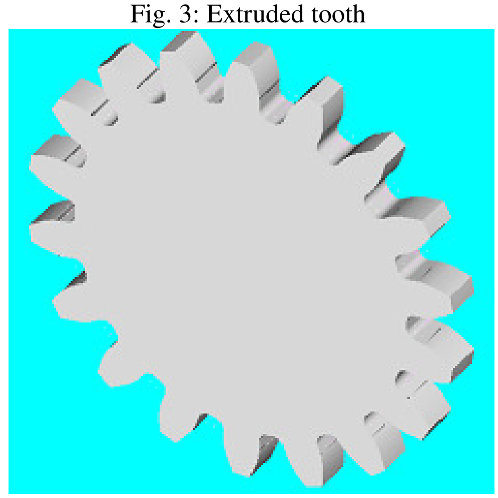

Fig. 5: Gear in solid filled mode

3. Meshing Technique

For having a gear pair, it was required to form the assembly of the modelled gear. The procedure for assembling and correct meshing of the *Gears* is summarised below.

1. The task was switched from the Master Modeler to the Master Assembly in the design application.
2. The *hierarchy* icon was clicked which opened the corresponding *hierarchy* form.
3. *Add parent* icon was clicked and the *Name* form was displayed. The corresponding name for the assembly was given, which is described in next points.
4. Sample nomenclature for different gears are as under:
5. *Add to* icon was used to add Gear (Pinion) and Gear in the assembly, which was selected from the form itself.
6. By clicking right button of mouse a menu was opened.
7. From the menu, *Get* was selected, which gave another menu, from where *From bin/Library* was selected.
8. The form *Select Part/Assembly* was opened.
9. The corresponding gear was picked from the form, where the gear is displayed as a list.
10. In the same manner, the other gear was selected and was added to the assembly.
11. Using *Move* command, the *Gear* was displaced from (0,0) position to a position depending upon their centre distances.
12. For putting the two gears in proper contact at the initial position of the engagement, it was necessary to rotate both the gears by certain degree, depending on their dimensions.
13. This was done by taking the rotations of the Left Gear by an amount of 10°, 9°, 7.5° and 6° for 18, 20, 24 and 30 number of teeth gears respectively.
14. Similarly the right gear was also rotated by 80°, 81°, 82.5° and 84° for 18, 20, 24 and 30 number of teeth gears respectively.
15. In case of the right gears, the pivot point for the rotation was taken as the centre point of the right gear and not the default point, which was (0,0).
16. Similarly, the corresponding directions of rotation were also considered.
17. The prepared assembly of gears was stored in the bin. Figure 6 shows the assembly of gear or the gears in mesh.

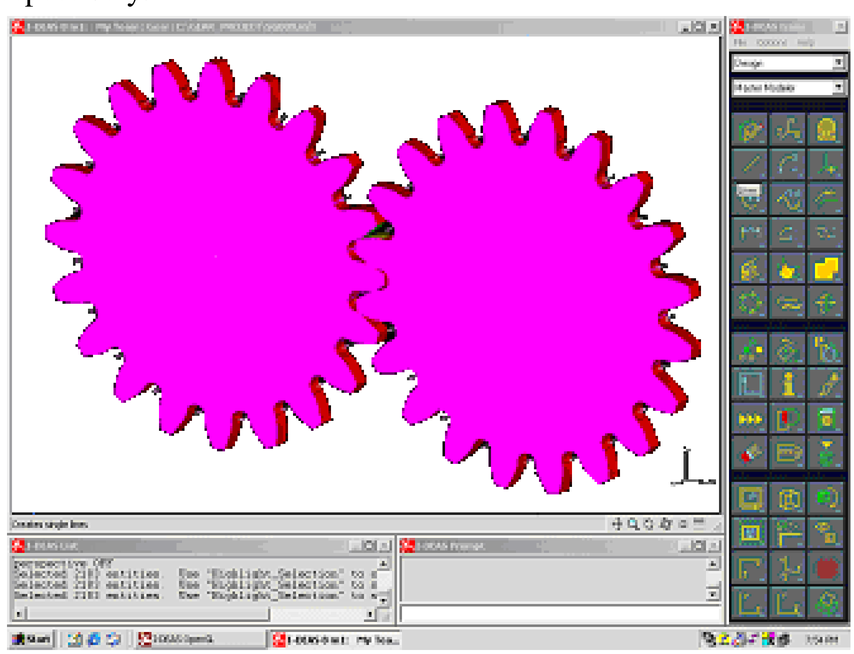

Fig. 6: Assembly of Master Gear and Erred Gear

4. Conclusion

In this paper, the detailed description of the modelling technique is elaborated. Starting from the basic drawing of all the circles, involute profile, fillets, construction of wire frame, extruding, patterning, joining etc. are suitably demonstrated. The meshing procedure is briefly explained for one case of the assembly. For correct engagements, it was necessary to rotate the gears, which is mentioned in the procedure. The same procedure may also be adapted to model and mesh the gear pair using other CAD software. These assemblies can further be used to do the various analyses.

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