Finite Element Analysis and Optimization of Helical Gear-pairs with Web based on SolidWorks

Jianhua Zhang\textsuperscript{1}, Qingping Zhang\textsuperscript{2}, Runze Shi\textsuperscript{3}

School of Mechanical Engineering, University of Jinan, shandong, China

1329893624@qq.com, me_zhangqp@ujn.edu.cn, 495093911@qq.com

Abstract. In this paper, Finite element analysis is implemented for helical gear-pairs by SolidWorks simulation. Firstly, GearTrax that is seamlessly integrated with SolidWorks is used to input the relevant parameters of helical gears. Then the gear entity is established in SolidWorks. Finally 3D models of gear-pairs are built up. Through the creation of auxiliary lines and auxiliary surfaces, the correct assembly of gear-pairs is completed, which guarantees the accuracy of subsequent finite element analysis. According to the results of finite element analysis, both the web thickness and the web hole diameter of the large gear are optimized to meet the maximum requirements. This work is able to provide reference for parameter design of helical gear-pairs with web.

1. Introduction

Helical gear-pairs is widely used to transmit force and motion through a typical tooth contact meshing. It has great advantages of good meshing performance, smooth transmission, big contact ratio, strong bearing capacity that other driving model cannot matched\textsuperscript{[1]}. Besides, web-shaped structure contributes to reduce their own weight, widely used in large-scale machinery and equipment. But in the process of meshing, there may be pitting, wear, plastic deformation, broken tooth or other damage on the contact tooth surface causing gear invalidation\textsuperscript{[2]}. Therefore, it is very meaningful to analyze contact stress problem in the meshing process of helical gear-pairs with web and improve the accuracy of the contact analysis of gears.

In this paper, GearTrax is seamlessly integrated with SolidWorks, which then is used to create 3D models. Then finite element analysis is implemented for this gear-pair by SolidWorks simulation, which avoids the conversion between softwares, and prevent the data loss and improve the accuracy of finite element analysis\textsuperscript{[3]}. According to the analysis results, the web thickness and the web hole diameter of the large gear are optimized.

2. Design parameters of Helical Gear-pairs with Web

Known conditions: In a factory, motor is used to transport power. Retarder is used to reduce speed. All equipment works two shifts a day, and the duration of work is 8 years. Belt pulley diameter \(d=300\text{mm}\), conveyor speed \(v=0.63\text{m/s}\). The output torque of the conveyor \(T=700\text{N\cdot m}\). The basic parameters of helical gear-pairs with web are shown in Table 1.

| \(Z_1\) | \(Z_2\) | \(m\) | \(m_0/\text{mm}\) | \(a\) | \(\beta\) |
|---|---|---|---|---|---|
| 25 | 82 | 3.26 | 3.5 | 20 | 9.76 |
When tip diameter is greater than 160mm and less than 500mm, it’s suitable for large gear to make gear with web. According to the empirical formula of gear with web, the gear parameters are calculated. At first the web thickness of 24mm and the web hole diameter of 42.5mm are selected. If the requirement of maximum stress is satisfied, optimize the web thickness and the web hole diameter may be optimized. The structure diagram is shown in figure 1.

Figure 1. The structure diagram of large gear with web.

3. Models Creating of gear-pairs and gear assembly
In this paper, GearTrax that seamlessly integrated with SolidWorks and SolidWorks are used to create 3D models. After modeling of gear-pairs, gear assembly is performed, as shown in Figure 2. Firstly, on the axial center plane of pinion and large gear, a line is created from the tooth top to the center and the root to the center respectively. It’s necessary to make the axial center of two gears on the same plane and make two lines coincident. Secondly, the center distance is set. Next, the command of gear assembly is executed to make gear indexing circle tangent. Finally, At this point this gear-pairs have been correctly meshed, but they are completely fixed, so it’s need to delete two lines of the first step. When the assembly is completed, the interference check is carried out to ensure the correct assembly of gear-pairs.

Figure 2. 3D models of gear meshing.

4. Static analysis
Finite element analysis of gears mostly selects one tooth or several teeth to analyze, which can greatly reduce the calculation amount and reduce the calculation precision as well. In order to improve the accuracy, this work selects the whole gear-pairs to carry on finite element analysis in SolidWorks Simulation.

In this paper, the material of large gear is 45# steel, the material of pinion is 40Cr. The basic parameters of the materials are shown in Table 2.

|                  | 45# steel | 40Cr   |
|------------------|-----------|--------|
| Elastic Modulus  | 2.05×10^5| 2.10×10^5|
| Poisson's Ratio  | 0.29      | 0.28   |
| Yield Strength   | 530MPa    | 620MPa |
| Tensile Strength | 625MPa    | 723MPa |

4.1. Constraints and load
For gear contact problem, the boundary conditions should describe the actual constraints of the model as realistic as possible. In the process of gear-pairs meshing, the driving gear is subjected to an additional torque to rotate at a certain speed, driving the driven gear to run. Torque is transmitted by the contact teeth between gear-pairs, so the driven gear is in equilibrium under the action of torque and resistance torque. In this paper, the influence of bearing deformation on gear deformation is neglected, and the gear shaft is assumed to be rigid. The pinion is the driving wheel and the large gear is the driven wheel. At any moment during the movement, the meshing transmission of gear-pairs can be regarded as quasi-static. The part of the driven wheel away from the meshing area can not feel the driving effect on
it. So it can be regarded as stationary and the displacement is zero. Therefore, the inner surface of large gear should be fixed \[6\]. After that, fixed hinge is used on the inner surface of the pinion holes, so that the pinion can only move in the circumferential direction. The pinion is the driving wheel, so the torque is applied to the pinion. The Torque is 229.81N m in Figure 3, the effect of the constraint and load are shown.

![Figure 3. The effect of the constraint and load.](image)

**4.2. Tooth contact**

As shown in Figure 4, the tooth profile surface 1 is set as the Initial contact surface, the tooth profile surface 2 is set as the target contact surface. Similarly, three pairs of teeth participating in the engagement are respectively set as contact pairs. Finally the friction coefficient of contact surfaces is set to 0.15.

![Figure 4. gear-pairs contact surface group.](image)

**Meshing and running**

The division of the grid has great influence on the calculation and accuracy of finite element analysis. The smaller grid is, the higher calculation precision is. In this paper, a good grid factor is used to generate 47,901 nodes and 26,743 cells, resulting in high-quality grids.

**4.3. Results analysis**

![Figure 5. The distribution diagram of contact stress.](image)

![Figure 6. The partial distribution diagram of pinion equivalent stress.](image)

It can be seen from Fig.5 that the maximum contact stress of gear-pairs is 365.4MPa, which is 8.3% different from the theoretical value. This is able to verify the rationality of finite element analysis. The maximum stress is lower than the yield strength of material. And there is still more room for
optimization. In order to better show the stress distribution, the partial distribution diagram was enlarged. On the meshing area, the maximum equivalent stress appears on the top edge of the pinion gear as shown in Figure 6, and several teeth participate in the meshing.

![Image](image.png)

Figure 7. The Integrated displacement diagram of gear-pair.

In the process of meshing, the contact area appear in the mesh line to the top of the tooth. The elastic deformation occurs on driving gear tooth root and driven gear tooth top. The maximum deformation is $4.082 \times 10^{-2} \text{mm}$.

5. Optimization
Helical gear-pairs with web is optimized based on SolidWorks. A new design study is generated from this statics analysis at first. And then variables, constraints, and targets are set in the variables view. In this paper, the equivalent stress $\sigma = 450 \text{MPa}$ is chosen as the constraint, both the web thickness and web hole diameter are taken as the design variables respectively. The objective is to minimize the mass of the gear. After setting the relevant parameters, the solution can be loaded. There are two optimization variables. According to empirical formula $C = (0.2\text{--}0.3)B$, the web thickness is 19.6 to 29.4. So the web thickness from 20mm to 29mm is chosen, the step length is 1mm. Another value surpasses the empirical formula and the thickness of 19mm is selected. In addition, the web hole diameter $d = (40\text{--}56) \text{mm}$, so the diameter of 40mm to 55mm is chosen, the step length is 2.5mm. To sum up, 77 optimization scenarios are formed.

![Image](image.png)

Figure 8. Equivalent stress diagram at different web thickness C and web hole diameter d.

![Image](image.png)

Figure 9. Gear quality diagram at different web thickness C and web hole diameter d.

After loading and solving, all the optimized data will be extracted and made into diagrams, as shown in Figure 9 and Figure10. Through the comparative analysis of the data, some laws can be obtained. In this optimization range, the maximum integrated stress gradually increases and the overall quality decreases with the decrease of the web thickness. Therefore, the optimal solution is to determine the web thickness of 19 mm and the web hole diameter of 55 mm. Compared with the initial case, the gear quality is reduced by 8.8%.

6. Conclusion
In this paper, GearTrax that is seamlessly integrated with Solidworks is used to input the relevant parameters of helical gears. Then the gear entity is established in SolidWorks. Finally 3D models of
Gear-pairs are built up. Through the creation of auxiliary lines and auxiliary surfaces, the correct assembly of gear pairs is completed, which guarantees the accuracy of subsequent finite element analysis. Finite element analysis is implemented for this gear-pairs by SolidWorks Simulation. The maximum contact stress of this gear-pairs is 365.4MPa, which is lower than the yield strength of the material. The elastic deformation occurs on driving gear tooth root and driven gear tooth top. The maximum deformation is 4.082×10^{-2}mm. According to the results of finite element analysis, the web thickness of large gear and the web hole diameter are optimized. The optimal solution was to determine the web thickness of 19 mm and the web hole diameter of 55 mm. Compared with the initial case, the gear quality was reduced by 8.8%. Optimization results is able to provide reference for parameter design of helical gear-pairs with web.

References
[1] Linlin Zhu, Baolin Wu, Yang Li. Contact Characteristic Analysis of Double Circular Arc Helical Gearing[J]. Journal of Mechanical Transmission2016, 40(02):146-149.
[2] Fei Guo. Research on the Methods of Loader Wheel Side Reducer Gears Contact Analysis[D]. Chang’an University, 2016.
[3] Aimei Gao, Jinliang Li, Yuan Zhang, Manshan Zhou, Qinghai Wang. Finite Element Analysis and Optimization of Meshing Spur Gears in Mine Reducer[J]. Hoisting and Conveying Machinery, 2014(04):83-85.
[4] Chunyi Zhang, Guangping Hao, Min Liu. Analysis of Design Example of Reducer[M].Beijing: China Machine Press, 2008.
[5] Lianggui Pu, Guoding Chen, Liyan Wu. Mechanical Design (9th Edition)[M].Beijing: Higher Education Press, 2013.
[6] Xiaoxi Gao. Finite Element Contact Analysis of Meshing Gears in Gearbox of Wind Turbine Generator[D]. Dalian University of Technology, 2008.
[7] S.Jyothirmai, R.Ramesh, T.Swarnalatha, D. Renuka. A Finite Element Approach to Bending, Contact and Fatigue Stress Distribution in Helical Gear Systems[J]. Procedia Materials Science, Volume 6, 2014, Pages 907-918.