Finite Element Analysis of Automotive Flange Stress

Xianwei Du [a], Xiang Shao [b] and Huazeng Hu [c]

School of Mechanical and Electrical Engineering, Shandong University of Science and Technology, Shandong 266590, China;

[a]893355652@qq.com,[b]1119061284@qq.com,[c]1046805687@qq.com

Abstract

Using the finite element analysis software ANSYS workbench, the number of the driven wheel hub bearing flanges of the car is counted.Value simulation, 3D modeling by Creo, generation of igs, introduction into ANSYSworkbench for analysis, the law of flange variation with temperature and its effect on relieving stress concentration. From the theoretical point of view, the reasonable temperature of the flange during the heat treatment process is laid, which lays a foundation for structural optimization.

Keywords

Flange, Creo,temperature field.

1. Introduction

Gears are closely related to our production and life today, and many of the products we use in our lives use flange drives. Gearbox modeling analysis is more complicated. Nowadays, there are few methods using parametric modeling[1]. The advantage of parametric modeling is that it can improve the accuracy of the model. It is not only convenient but also the cost of the analysis process. The time is small and the operation is simple and efficient.

This paper mainly uses Creo 3D software for parametric modeling to obtain a more accurate gear model, so that ANSYS analysis will not lead to data errors. The import process generates a file generated by the 3D software Creo in igs format and then imported into ANSYS for environment. Simulation analysis.

The car is one of the important tools for traffic on the road. The hub bearing is the core component of the car. Its main function is to bear the weight of the car and provide the correct guidance for the wheel. One of the most important parts of the wheel bearing is the flange, which is subjected to various super-carrier motions and impact forces during transportation. Its performance is related to the normal operation of the entire vehicle[2]. In harsh working environments, flanges are often hardened by high temperatures and stress concentrations, affecting their reliability and safety. Therefore, it is important to use the finite element analysis software to carry out numerical simulation calculation.

Numerical simulation of the flange at different temperatures. In order to more clearly represent the temperature distribution, a temperature distribution cloud map of the entire model is calculated. Through comparative analysis, the reasonable temperature of the flange during heat treatment is obtained, and the role of residual stress in relieving stress concentration is verified.

2. Finite element model establishment

2.1 3D solid model

ZG15Cr1Mo cast steel is widely used in the manufacture of flanges. Therefore, the wheel bearing flange material of this car is made of ZG15Cr1Mo cast steel with density $\rho=7.83\times10-9$ t/mm3 and yield limit $\sigma0.2=685$ MPa.

Elastic modulus E = 207 GPa, Poisson's ratio μ = 0.3, coefficient of linear expansion α = 1.35 × 10 - 5 m / ° C. The experimental value of the residual stress on the fillet surface of the hardened layer of the flange is about -420 MPa.

The finite element analysis model of the flanges constructed in this paper has a bore diameter of 40 mm[3,4]. Because the model and the stress field both have axis symmetry, and in order to save computational cost and improve the mesh quality of the model, the finite element analysis model is established, as shown in Figure 1.

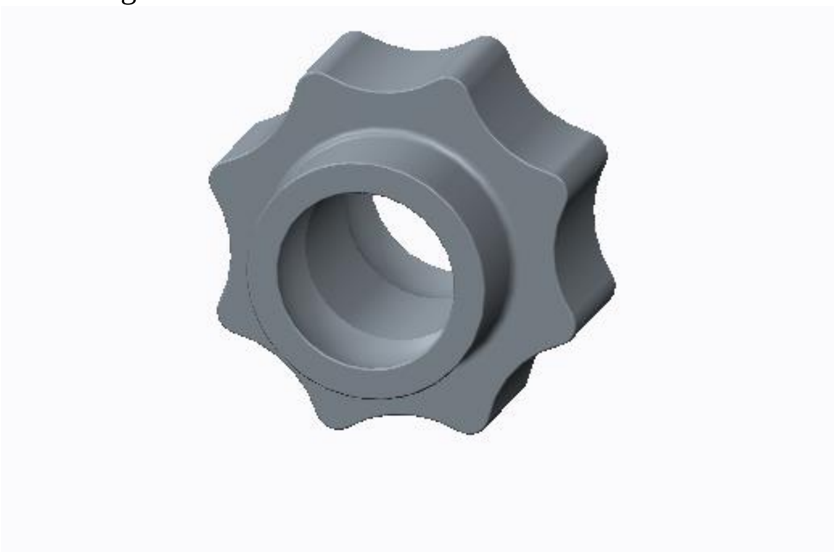

Fig. 1Flange 3D model

2.2 Dividing the grid

The 3D solid model in Figure 1 is generated by the 3D software Creo to generate a special format igs into ANSYSworkbench for finite element analysis. The generated grid book is shown in Figure 2.

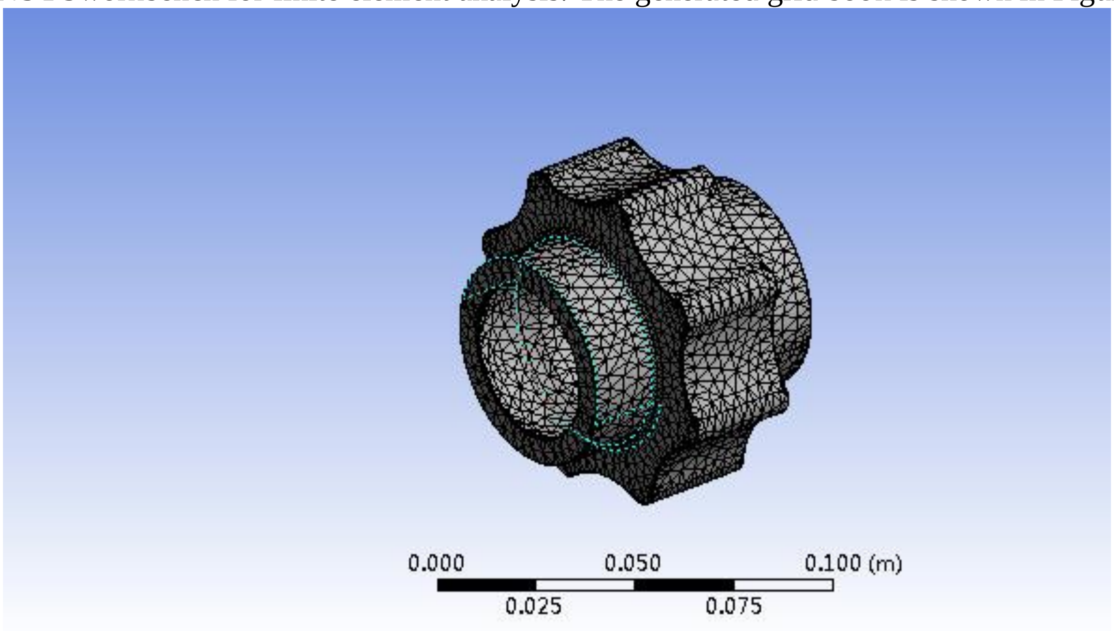

Fig. 2 Flange grid

2.3 Constraint and analysis step settings

According to the actual working conditions, the full constraint is added to the bottom end face; the symmetric cross section adds the symmetric constraint of the xoy face; the other end face applies the vertical downward face load p=100 MPa.

The stress field of the entire model is a static response, so the analysis step type is set to Static, General.

(1) In the first analysis step, the temperature of the hardened layer is raised to 25 °C, 35 °C, 45 °C, 55 °C, 65 °C, 75 °C, and the temperature of the remaining area is kept at room temperature of 20 °C. No external load is applied to simulate the workpiece. Residual stress field;

(2) The second analysis step keeps the above temperature field unchanged, and applies an external load p=90 MPa;

(3) The third analysis step The temperature of the whole model is changed to 20 °C, and the external load p=90 MPa is kept unchanged, so that the stress field without residual stress is obtained, which is used for comparison with the result of the second analysis step.

3. Analysis of finite element calculation results

(1) Temperature field distribution cloud map

(2) The residual stress cloud diagram of the flange surface obtained after calculation is shown in Fig. 3. The flange is heated to 85 ° C,

(3) Due to the temperature difference between the rounded surface and the core of the hardened layer, the volume expansion and shrinkage are uneven and a large compressive stress is generated, so the calculation result is consistent with the actual heat treatment process[5]. The resulting temperature distribution cloud map is shown below.

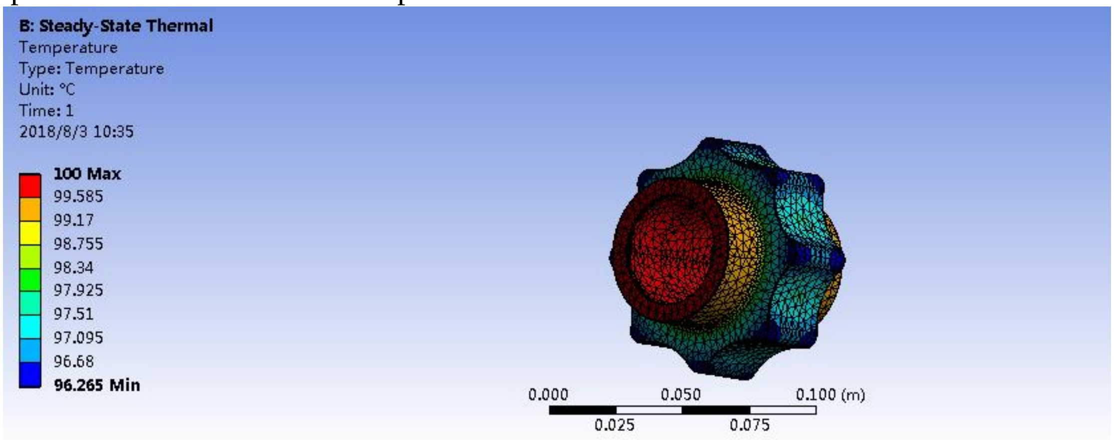

Fig. 3 Temperature distribution cloud

At the same time, through the similar operation, the heat flow cloud map of the flange is obtained, as shown in the following figure.

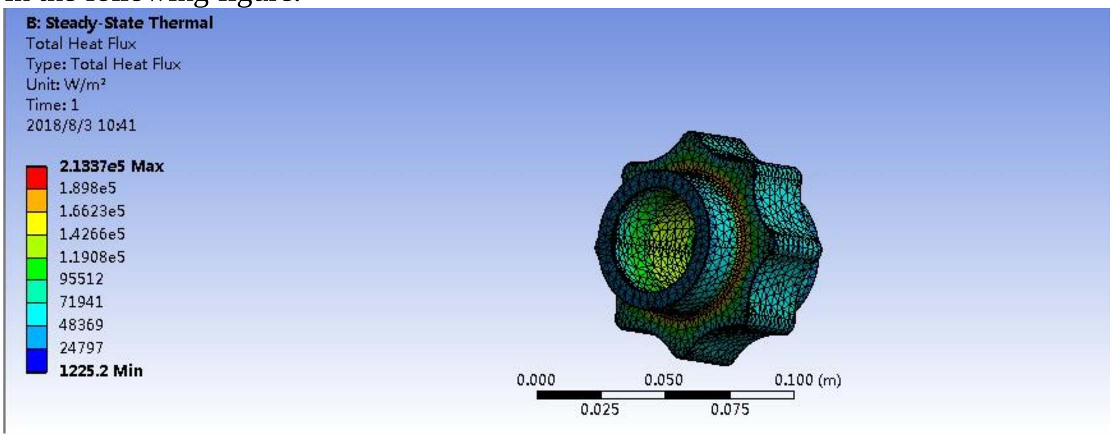

Fig. 4 Heat flow cloud map

(2) Effect of temperature on residual stress

It can be seen from the above simulation that with the different temperature fields, the influence on the flange is different. We can clearly know how to grasp the temperature range of this material. Only then can we Very good rationalization of the actual application of the flange[6]. The service life of the flange is improved.

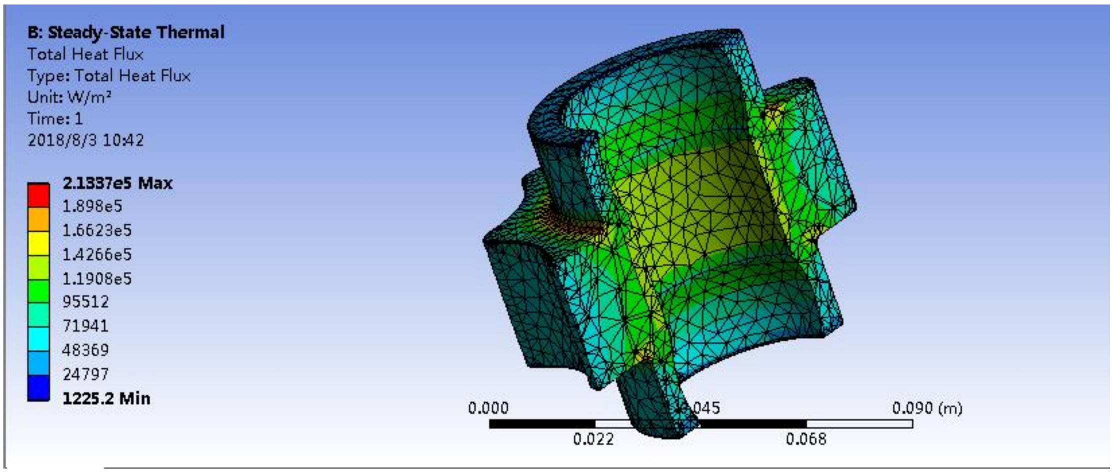

Fig. 5 Temperature field distribution inside the flange

As shown in the above figure, we have to segment the model analyzed by ANSYS workbench software. First, we need to select an observation datum. From this direction, we can divide it into half to observe the temperature field distribution inside the flange.

4. Conclusion

The material properties of the flange were simulated. The distribution of the quenching residual stress on the surface of the flange at different quenching temperatures and the stress distribution after the residual stress relief stress set were compared and analyzed. The following conclusions were drawn:

(1) Using Creo to complete the 3D modeling of the flange, the model generation efficiency is high, and the calculation result is accurate;

(2) Using the ANSYS structural analysis module to divide the finite element model of the generated three-dimensional flange, loading and meshing;

(3) The residual compressive stress generated at a reasonable quenching temperature reduces the stress concentration on the fillet surface of the flange, which plays a significant role in improving the life of the flange.

(4) The method of establishing product and performance analysis by using Creo and ANSYS software has certain use significance in current engineering design.

References

[1] Ren Zhongquan, Xin Xin, Li Dan, et al. Modal Analysis of Hub Bearing Flange Hub Based on ANSYS[J]. Coal Mine Machinery, 2014, 35(5): 97-98.

[2] Jia Hongling, Zhou Yanping. Analysis of axial load characteristics of car wheel bearings[J]. Axle, 2010(2):10-12.

[3] Lu Xiaohui, Xie Xiaopeng, Wang Wei, et al. Wheel bearing method based on ANSYS Workbench

[4] Shi Yiping, Zhou Yurong. Detailed Explanation of ABAQUS Finite Element Analysis [M]. Beijing: Mechanical Industry Press, 2012.4.

[5] Guo Qiuyan, Pang Hao, Deng Lei. Analysis of Torque Rigidity of Inner Flange of Car Wheel Bearing Based on ANSYS[J]. Mechanical Design & Manufacturing, 2010(7): 210-212.

[6]Finite Element Analysis of Lanpan Wheel[J]. Machine Tool & Hydraulics, 2012, 40(5): 129-131.