ATLANTIS PRESS
Advances in Computer Science Research, volume 75
7th International Conference on Mechatronics, Computer and Education Informationization (MCEI 2017)

Finite Element Analysis of Hook Structure Based on Autodesk Simulation

Chunming Xu[1, 2] and Bowen Luo [1, a *]

[1]Key Laboratory of Metallurgical Equipment and Control Technology, Ministry of Education, Wuhan University of Science and Technology, Wuhan 430081, China.

[2]Hubei Key Laboratory of Mechanical Transmission and Manufacturing Engineering, Wuhan University of Science and Technology, Wuhan 430081, China.

[a]378401869@qq.com

* please mark the corresponding author with an asterisk

Keywords: Hook; Finite element; Autodesk Simulation

Abstract. The generalized finite element software Autodesk Simulation is used to simulate and analyze the hooks, and the distribution of stress and displacement changes is obtained. The dangerous cross section of hook is analyzed and studied theoretically, which provides the necessary theoretical basis for structural design and optimization.

Introduction

The hook is an important component of hoisting machinery, and it is the part of the heavy load that directly bears the heavy load. If the crane hook is broken or damaged, it will cause major safety accidents. In order to ensure the safe and stable operation of hoisting machinery, the structure and working condition parameters selection of crane hook is very important. The static analysis and fatigue test of the ship hook are carried out by Toshihisa et al. It is found that the change of the arc transition angle has little influence on the strength of the hook, but the diameter of the shank has obvious influence on the strength of the hook. T.Muromakia et al believe that the hook design has a typical conical shape for optimum shape[1]. Y.Torres et al found that the fracture of the hook was caused by strain aging embrittlement. K.Easterling points out that the shape of the crane hook can be optimized, the service life can be prolonged and the failure rate can be reduced by predicting the concentrated area of the stress. In addition, the stress and strain distribution law of the hanger structure can be revealed by numerical simulation, and the dangerous section of the hook can be determined. The above research mainly focuses on the influence of the structural parameters of the hook on its strength, while the swing angle between the hanging wire and the hook also affects the safety condition of the hook[2].

Selection of Hook Model

The analysis object is a heavy lift hook made of 20# steel, and the specific size is shown in figure 1. The hook mechanism is complex, there are a lot of chamfer, while at the same time, the complex geometry of the hole, hook, in this case, whether the calculation mesh or final results will have difficulties, or even unable to obtain the numerical non convergence results, then you need to calculate the correctness of the results in the premise not under the influence of the simplified model structure[3], the model combined with the hook load and constraint conditions are analyzed, at the top of the hole and the hook can be part of a small fillet can be omitted, these small fillet and small hole will cause great difficulty in meshing, and will make a significant increase in calculation, and these small chamfer and a small hole on the final results is very small, therefore will omit the chamfer and small hole[4-5].

Establishment of Hook Model

Establishment of 3D Model of Hook. The 3D model of the hook is established in soildworks2013, and the hook model is drawn according to the size shown in the drawing[6]. The specific size of the hook is shown in Fig. 1. The main body of the hook is drawn out by lofting, and the drawing part of the guide line is completed as shown in Fig. 2.

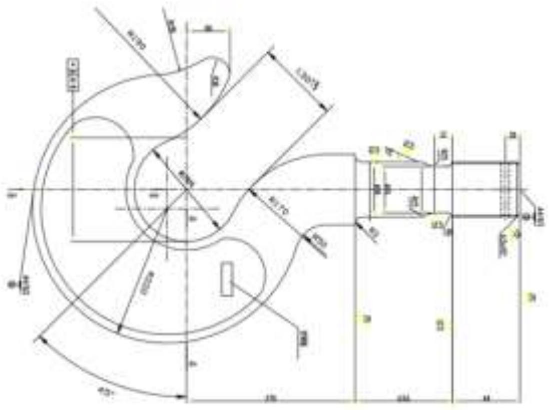

Figure 1. Finite Hook size

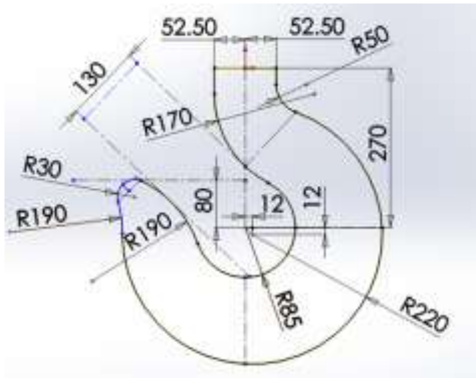

Figure 2. Finite Main body drawing of hook

After the guide line and contour drawing are finished, the lofting is started. Finally, the 3D model of the hook is drawn in SoildWorks, as shown in Fig. 3.

Figure 3. Finite Three dimensional model of hook

Establishment of Material Model

The plane problem is a simplification of the actual structure under special conditions. In practical problems, any object is strictly a space object, and its load is usually space. However, in engineering problems, the shape and load conditions of some structural or mechanical parts have certain characteristics. As long as they are properly simplified and abstracted, they can be reduced to plane problems. The characteristic of this problem is that all phenomena are considered to occur in a plane[7-8], and the model of plane problem can be greatly simplified without distortion.

Simulation of Hook

The preprocessed file into Autodesk Simulation, is a block element, then the material attribute, the hook is the use of the 20# steel hook rated load is 20T, according to the material properties, the elastic modulus is 2.13E+11N/m^2, the quality of density is 7.80E+03kg/m^3, Poisson's ratio of 0.282. Set it as a material property, and change the material properties[9]. Then the 3D mesh is set up. The Autodesk Simulation is used to divide the hook part into the grid, and the hook is divided into adaptive grids. Using 8 nodes and 183 units, a total of 5289 nodes and 1807 units are divided, and finally the partition of the hook grid is shown in Fig. 4.

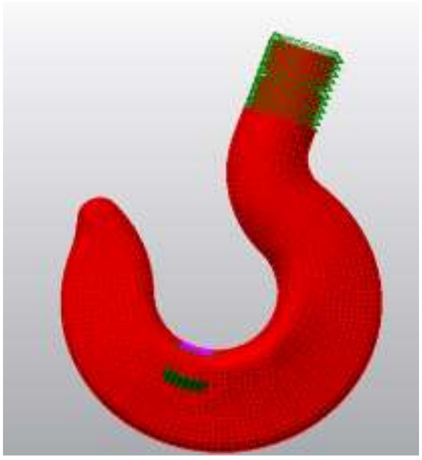

Figure 4. Finite Classification of hook grids Figure5. Finite Added finished hook model

After the grid settings are finished, the settings of the load and constraint groups are carried out. After the hook is fixed on the top surface, add force, as is the 20T hook, so the value is set to the -200000 direction of the Y axis, and then select the stress surface has been added on the hook, the force is loaded to force on the surface. The results of constraint and force addition are shown in Fig. 5. After the above work is completed, the simulation can be started[10].

After simulation, the displacement nephogram, stress nephogram and strain nephogram of the hook are obtained, which are shown in Fig. 6, Fig. 7 and Fig. 8, respectively.

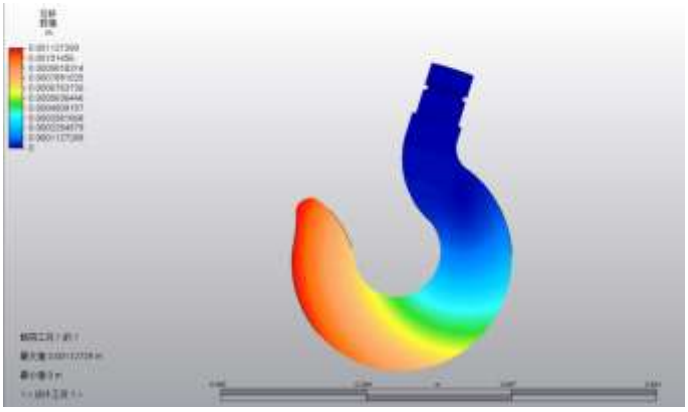

Figure 6. Finite Displacement nephogram of hook

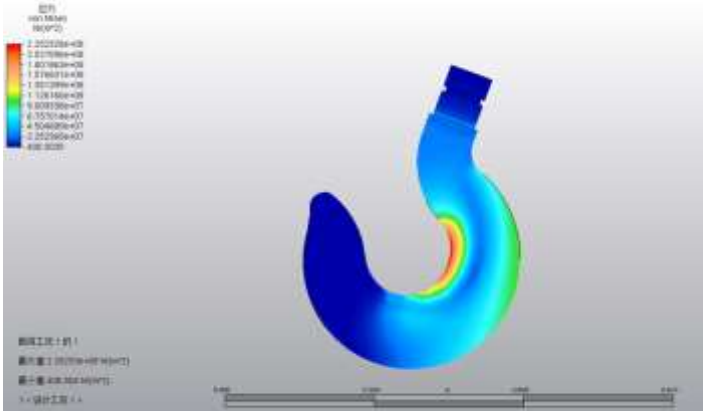

Figure 7. Finite Stress nephogram of hook

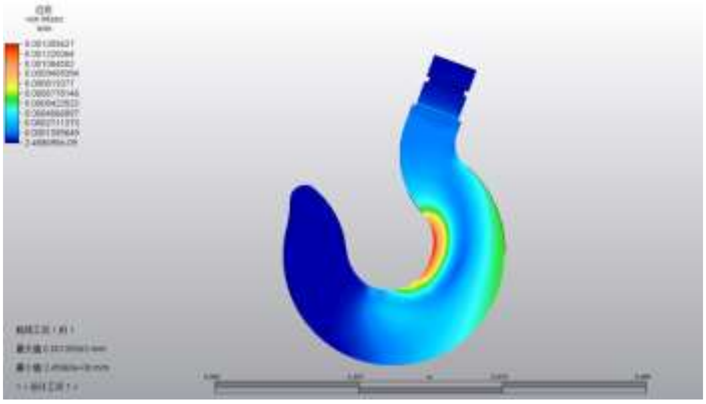

Figure 8. Finite Strain nephogram of hook

Conclusion

According to the relevant material properties, the yield limit of 20# steel is 2.45E+08Mpa, and the maximum stress is 2.25E+08MPa in this analysis. The maximum stress is smaller than the yield limit, the hook is safe, the maximum displacement is 0.00112729m, and the displacement and stress distribution show a strong correlation. The maximum stress and the maximum displacement appear near the B-B plane. The B-B surface is the dangerous section of the hook, and there is no stress distribution in the hook and the tip, and the hook handle also produces relatively small stress. From the overall displacement effect diagram can be seen: the overall deformation of the hook is still obvious. It can be seen from the simulation that the hook of 20# steel will not appear larger deformation or fracture under the action of 20T vertical load.

References

[1] Mirosław Galicki. Real-time Robotics and Autonomous Systems, (2016) No.23, P.13.
[2] Abdelfetah HENTOUT,Mohamed Ayoub MESSOUS,Brahim BOUZOUIA..IFAC Proceedings Vol (2014), p.47.
[3] M.H. Korayem,S.R. Nekoo. Robotics and Autonomous Systems, 2016 p.32.
[4] Y. Chen, F.Y. Xia. Journal of Chongqing University of Arts and Sciences, Vol. 35(2016) No.2, p.79. (In Chinese).
[5] Salima IFAC Proceedings Vol, 42(2009) No, 13.
[6] Grzegorz Pajak,Iwona Pajak. Archive of Mechanical Engineering, (2014) No. 61.
[7] Fu-Cai Liu,Li-Huan Liang,Juan-Juan Gao.International Journal of Automation & Computing,(2014) No.4,p.353.
[8]. T. Mouri, H. Kawasaki, K. Yoshikawa,et al:Chinese Journal of Mechanical Engineering,(2012) No.2,p.197.
[9] CHEN Kang,MA Chunxiang,ZHENG Maoqi,GAO Feng.Chinese Journal of Mechanical Engineering,Vol.2(2015),p.236.
[10] OTYC, ALBU—SCHAERA, KUGIA, et a1. IEEE Transactions on Robotics, Vol. 24(2008), No.2, p.416.