Finite Element Analysis of The Hoisting Process of A Lifting Device Based on ANSYS

Wanshan Su1* and Qunxian Qiu1
1Institute No.713, China Ship Industry Corporation, Zhengzhou 450052, Henan, China
*Corresponding author’s e-mail: sws1986@126.com

Abstract. This template use the finite element software ANSYS Workbench to analyze lifting process of the lifting device after debugging. The model is simplified on the basis of the calculation precision of guarantee and studies the deformation and stress distribution of components. Over these data, it can provides the basis for the transport and hoisting of the lifting device. By lifting the deflection of the various components in the process, put forward the corresponding improvement measures. Through the matching of the two driving functions, using the two braces, guarantee the reliability of hoisting, and give practice verification in the process of hoisting.

1. Introduction
A lifting device is an important component in marine supply equipment and the ammunition is mainly transported to the ship deck through chain drive system. Before the lifting device lifted in the shipyard, it usually completes assembly debugging in the debug platform. When transported to the shipyard, it must be considered a lifting mode to ensure the safety. Its length is 10.2m and runs through four decks, therefore it is necessary to verify it's lifting program again, and analyze the overall deformation and stress distribution in the lifting process[1]. It has a great significance for the construction site safety and subsequent optimization of the design.

2. Model processing
This lifting device is composed mainly of bottom cylinder, cylinderI, cylinderII, cylinderIII, middle cylinder, transmission parts assembly(including geared motors), upper cylinder. Flange connectionis adopted among the components. Because the mode is too complexly, it is necessary to simplify the model. In the process of the simplified model, the equipment which has little effect on the stiffness can be ignored. Meanwhile, the model edge chamfers and fillets also has little effect on the stiffness, so it can be removed too, then the model can be analyzed by finite element analysis[2-3]. The solid model is established by software Pro/Engineer, ANSYS-based finite element technique was used to model and analyze the structure. Using pach conforming algorithm to mesh the model, and the simplified finite element model is shown in figure 1.
Lifting device body’s material selection is Q235 steel over welding, its Young's modulus and Poisson's ratio were 200 Gpa and 0.3[4-5]. The finite element model totally have 307433 nodes and 142177 elements. In the process of the simplified model, it is necessary to improve the quality of equipment parts applied to real mass, so the structure of each part is used the simplified model of equivalent density. The density of the parts as shown in table 1.

| Parts          | Actual Volume $(mm^3)$ | Model Volume $(mm^3)$ | Equivalent Density $(kg/mm^3)$ | Mass $(kg)$ |
|----------------|------------------------|-----------------------|-------------------------------|------------|
| Bottom Cylinder| 2.1477474e7            | 1.5293481e7           | 11024.19                      | 168.598    |
| CylinderI      | 3.3210318e7            | 3.3210318e7           | 7850                          | 260.701    |
| CylinderII     | 3.3210318e7            | 3.3210318e7           | 7850                          | 260.701    |
| CylinderIII    | 2.2043056e7            | 2.0091074e7           | 8612.68                       | 137.038    |
| Transmission Part | 2.6096479e7          | 4.8907878e6           | 41886.37                      | 204.857    |
| Middle cylinder | 2.0877498e7           | 1.1132787e7           | 14721.23                      | 163.888    |
| Upper cylinder  | 2.0064185e7            | 1.4564656e7           | 14788.27                      | 157.504    |

After lifting device packaging transportation to shipyard, then it can go ahead lifting and installation. In the past lifting process, the equipment is firstly put on the horizontal support, then swing device into the vertical direction by lifting point of upper cylinder, lastly the equipment is attached to the hull. The whole process can be simplified further, the process of lifting one side supporting and the other lifting is a simply supported beam model[6]. With the equipment lifting, the centre of gravity is shift down, all of the bending stress also decreased, so lifting moment of the equipment lifting would be a dangerous working conditions in all process. Then we need to ensure the safety and reliable of the equipment hoisting working condition. Primary lifting point location is shown in figure 2.
As shown in figure 2, there is contact and welding between the parts. When the model is imported by the software, the ANSYS workbench can automatically detect the contact surface and generate the contact pairs between the parts each other. It can ensure that the force transmission between the various components, spot weld is used to connected deck sleeve and cylinder wall. It is a kind of discrete point contact assembly method.[7-8]. Spot welding assembly method is shown in figure 3. Model of the boundary conditions as follows: the bottom of cylinder flange is imposed by fixed support, Z direction fixed constraints is imposed on the side surface of upper cylinder, X, Y direction is free, then the simply supported beam model is simulated, at the same time, the gravity of the model is applied (by applying the inertial acceleration).

3. Analysis of the results
After the completion of the finite element analysis, the related data can be get[4]. Integral lifting device deformation and stress distribution are shown in figure 4 and figure 5.

As shown in the figure 4, large deformation zone is distributed in cylinder 2 area, and it is roughly occurred in the upper equipment. The result is due to the weight of the transmission part is too heavier, the part that the maximum deformation is 3.21 mm. The figure 6 shows that the maximum equivalent stress appears in the bottom of the barrel semicircle plate support position, the maximum stress value is 243.33 Mpa, this is a stress concentration area, mainly caused by coarse meshing. The stress concentration of the small scale will not cause the overall component failure, but in order to ensure the safety, we can increase thickness the semicircle plate or choose higher strength steel such as 40Cr material[9]. Around the wall repair square mouth between the cylinder and the central cylinder flange, it also have a large stress, but it is all between 81–135 Mpa, will not cause damage to the equipment.
4. Improved Design

On the previous analysis and the lifting device is improved, the quenching and tempering treatment is used on bottom cylinder in the production phase, its HRC hardness increased to 35~40, the strength of the corresponding limit has been increased[10]. Equipment of large deformation areas are mainly concentrated in the upper area, so in the process of device hoisting, we use another belt to increase a lifting point. Using two driving, we can complete hoisting equipment more safety. In the process of lifting at the scene, the equipment is also lifted smoothly.

5. Conclusion

(1) There exists a maximum instant stress about lifting device whose length is 10.2 m in the process of lifting. Large deformation area is mainly distributed in cylinder 2 area. The maximum deformation is 3.21 mm;

(2) There appears a larger stress value at the bottom of the barrel semicircle reinforcement plate in the selected lifting hoisting plan. At this time, in order to ensure the safety, it’s necessary to adopt a high strength and better material to replace the CylinderII;

(3) All kinds of unsafe factors should be considered in the process of hoisting. To further ensure safe and reliable operation, we can use another lifting scheme. Lifting position is moved around the flange area between upper cylinder and middle cylinder, which reduces the local stress maximum and ensures safe and reliable operation.

Acknowledgments

Furthermore, I’m grateful for naval gun project team of institute No.713 China ship industry corporation, thanks numerous individuals participated in this study.
References

[1] Ling G L, Ding J B, Wen Z. (2012) ANSYS Workbench 13.0 from entry to the master. Tsinghua Publishing House, Beijing.

[2] Yang T L. (1996) Basic theory of mechanical system structure-kinematics-dynamics. China Machine Press, Beijing.

[3] Cheng D X (2002) Mechanical design manual. Chemical Industry Press, Beijing.

[4] Pu L G, Ji M G. (2008) Mechanical design. Higher Education Press, Beijing.

[5] Yu F G, Wang Y. (2013) Optimization design of linkages for eight-links mechanical press. Forging & Stamping Technology, 38: 138-141.

[6] Wei M, Zhang H W. Zhu H. (2013) Strength analysis of the intermediate shaft of cotton picker gearbox based on Ansys simulation. Manufacturing automation, 35: 27-30.

[7] Wang K D, Han K K. (2018) Analysis and optimization design of grinder column based on ANSYS Workbench. Manufacturing automation, 40: 64-68.

[8] LI R Q. (2008) Mechanical Principle. National Defense Industry Press, Beijing.

[9] Guo Z W. (2015) Research on the method of certain medium calibre naval gun performance improvement. Ship Science And Technology, 37: 135-138.

[10] Duan J. (2018) Structural design and strength verification of surface ship based on finite element analysis. Ship Science And Technology, 40: 25-27.