

The Finite Element Analysis of Trolley Frame of Stacker-Reclaimer Running Mechanism Based on ANSYS

Yanjun Xiao, Diming Guo, Xinyu Liang and Yuming Guan

School of Mechanical Engineering, Hebei University of Technology, Tianjin 300130, China

Abstract: In order to design and optimize the trolley frame of stacker-reclaimer running mechanism, it is very convenient and efficient to take advantage of ANSYS, which can not only ensure security but also reduce time and cost. The stacker-reclaimer is a common continuous and efficient bulk materials stevedoring and transporting device in the world, running mechanism is located in bottom of the whole machine, which plays an important role of supporting the machine and the running function of the machine. Therefore, the security and reliability of the running mechanism is very significant and critical in the safe operation of the whole machine. To guarantee stiffness and hardness of the steel structure, it is carried out that the static analysis of the key components the trolley frame by using finite element analysis software. Through careful analysis and research, it proves that the design of the components meets completely requirements of real conditions.

Keywords: ANSYS, running mechanism, stacker-reclaime, trolley frame

INTRODUCTION

In recent years, the domestic development of bucket wheel stacker-reclaimer is quite rapid and the level of development has been raised greatly (Wang *et al.*, 1999). But compared with the developed countries such as America and Germany, there is still a big gap (Lv and Li, 2008). In the design of stacker-reclaimer running mechanism, most domestic research work is mainly focused on driving control and improving of research and study of complicated problems, which appear usually in the process of installation and debugging (Ying *et al.*, 2010). The study of the strength of the running mechanism, optimized design of system parameters and the static and dynamic analysis is not much. Besides, its analysis method and means is very backward.

Traditional design methods which relies to experience with too much emphasis on security often causes large dimensions and contour, so it is not easy to find vulnerabilities in the product (Mu, 1998). ANSYS is international popular software of finite element analysis which can study and analyze structure, thermal, fluid, electromagnetism, acoustics and architecture. It is widely used in industry, railway, transportation, automotive, aerospace, automotive, mechanical engineering and so on (Lan and Zhang, 2000). Using the finite element method, early in the production design, we will be able to find and solve problems and defects for the design and optimization of pivotal products with a lot of convenience (Yu, 2005).

THE INTRODUCTION OF TROLLEY FRAME OF STACKER-RECLAIMER RUNNING MECHANISM

The trolley frame of stacker-reclaimer running mechanism is one of key parts, which is a welded box-type structure made up of upper wing plates ,two side webs and stiffened plates (Liu *et al.*, 2011). Its strength and stiffness directly affects the security of stacker-reclaimer. In the working process of stacker-reclaimer, it mainly bears the gravity of the parts above it and lateral force of the wheels. Traditional design of trolley frame is dependent on previous experience and lessons, however, in order to ensure security, it often causes that the size is more and more enlarging and the weight is more and more increasing. This study applying the finite element method in the early check of the designed product figures out problems before manufacturing and production, which significantly reduces production costs and lays a strong foundation for future studies.

THE FINITE ELEMENT ANALYSIS OF THE TROLLEY FRAME

Reasonable assumptions and simplification of the model increases modeling speed and reduces the difficulty of solving problems (Kaveh and Tolou Kian, 2012). We made some assumption and simplification to the trolley frame: such as assuming uniform distribution of material, solid element in every sector of trolley frame and ignoring the influence of welding stress and bolts

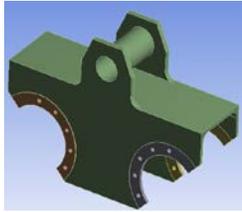


Fig. 1: 3D model of the trolley frame

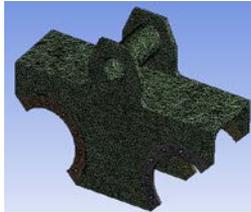


Fig. 2: The meshing of the trolley frame

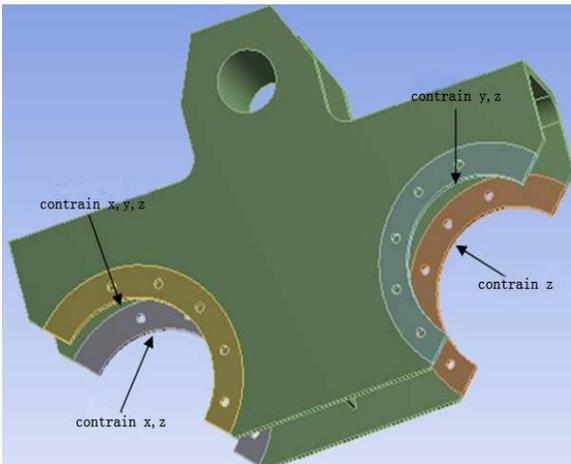


Fig. 3: The constraints of the trolley frame

on the frame. Now it is time to do static analysis of the trolley frame by ANSYS.

It contains five main parts:

- The establishment and import of 3D model
- Division of the finite element mesh
- Defining material properties
- Imposing constraints and loading
- Analysis of stiffness and strength

The establishment and import of 3d model: We use 3D software SolidWorks to model the trolley frame and import the designed frames to ANSYS environment through the seamless connection of SolidWorks software with ANSYS software. The model is shown in Fig. 1.

Division of the finite element mesh: There are many unit types of finite element analysis, these structures use unit type SOLID95, which is higher-order unit form

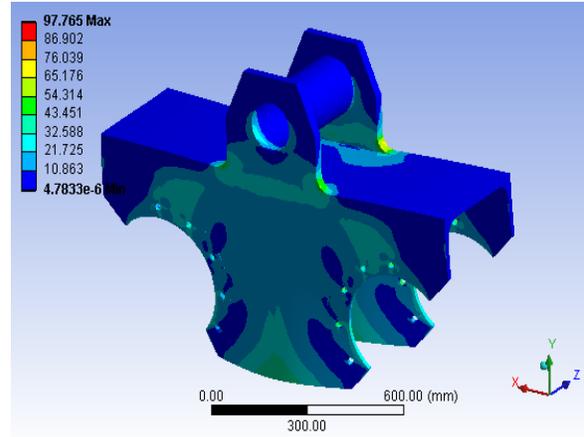


Fig. 4: Equivalent stress of trolley frame

of the SOLID45 (Chae and Bathe, 2000; Wang *et al.*, 2002). Each unit has translational degrees of freedom in three directions of x, y, z. This unit type allows irregular shape and will not reduce the accuracy with good compatibility. Due to the local stress and strain, refinement on the part of local grid in the mesh generation process will improve the accuracy of final solution. Mesh is shown in Fig. 2.

Defining material properties: We choose low-alloy steel Q345B of good welding properties for the trolley frame of running mechanism. Its performance parameters are as follows (Yu, 2008). Modulus of elasticity is $2.07 \times 10^{11} \text{N/m}^2$. Poisson's ratio is 0.29. And the density is $7.8 \times 10^3 \text{kg/m}^3$.

Imposing constraints and loading: The velocity of the running mechanism is very slow and uniform and it can only translate along the direction of predetermined orbit (Mile *et al.*, 2012), so we constraint other degree of freedom, as shown in Fig. 3.

The trolley frame is supported by four identical bearings and it also supports top balancing stand and bears lateral force wheels exerts. We take the three aspects of force into consideration when imposing loads and torque. In accordance with maximum wheel pressure, we calculate the magnitude of the pressure 250 kN, the support force the bearings exert the trolley frame is distributed relatively equally on the arc surface of bearing sleeve and the magnitude of support is 250000 N. The pressure the upper exerts the trolley frame is calculated accurately is 75000 N according to the formula that the pressure is 0.3 times than wheel pressure, which is also distributed relatively equally on the round surface of the top wing plate and the balancing stand. And the lateral force of the trolley frame is 30000 N, which acts on the ring round surface of the bolt hole.

Analysis of stiffness and strength: After imposing constraints and load, the results are shown in Fig. 4, 5.

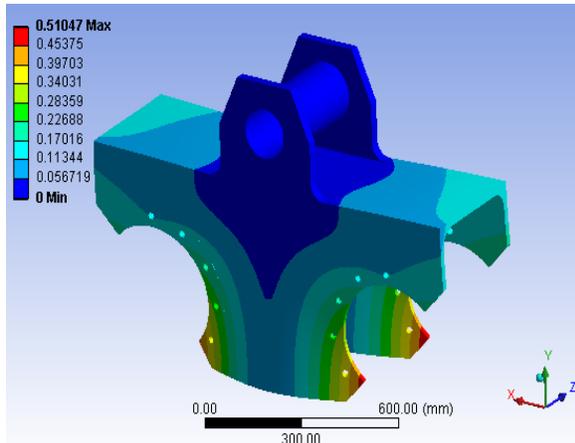


Fig. 5: Total deformation of trolley frame

Through the equivalent stress and total deformation nephogram, we know the value of the trolley frame stress is quite small and the maximum stress appears in the connecting area of the top wing and the semicircles of two webs, which are welded together. The maximum value is 97.765 MPa, far less than the maximum allowable stress. So maximum stress meets design requirements. Maximum deformation of the trolley frame is 0.51047 mm, which also meet the design requirements of the maximum deformation.

CONCLUSION

This study analyzes seriously the trolley frame of stacker-reclaimer running mechanism based on the static analysis. It constructs the 3D model of the trolley frame using SolidWorks software and also is imported into ANSYS by seamless connection. Through the selection of the unit, the division of the grid, constraints and imposing loads, the study proceeds to the analysis of the strength and stiffness. According to the results from analysis, deformation and stress of the trolley frame meets completely the design requirements and the design.

ACKNOWLEDGMENT

Thanks for Hebei province science support plan project, which supports this study. The funding project is 12227109D.

In the study, we got great help given by Bo Xu and other students in the lab research group. We express deep gratitude.

REFERENCES

- Chae, S. and K. Bathe, 2000. On automatic mesh construction and mesh refinement in finite element analysis. *Comput. Struct.*, 32(4): 911-936.
- Kaveh, A. and M.J. Tolou Kian, 2012. Efficient finite element analysis using graph-theoretical force method with brick element. *Finite Elem. Anal. Des.*, 54: 1-45.
- Lan, Y. and L.J. Zhang, 2000. Large-scale finite element analysis software. *Appl. Sci. Technol.*, 27(6): 11-12.
- Liu, S.Y., Z.X. Wang and Y.H. Xiao, 2011. Study on optimization of bucket wheel stacker/reclaimer driving room. *Comput. Simul.*, 28(10): 375-378.
- Lv, M.H. and Y.M. Li, 2008. The trend of bucket stacker and reclaimer. *Port Oper.*, 2: 6-9.
- Mile, S., G. Milomir and P. Dragan, 2012. Analysis of the drive shaft fracture of the bucket wheel excavator. *Eng. Failure Anal.*, 20: 105-107.
- Mu, P.F., 1998. The development trends of modern bucket wheel stacker-reclaimer. *Shanghai Electric Power*, 1: 4-6.
- Wang, Y.X., Y.Q. Tang and M.X. Wei, 1999. The development and research trends of cantilevered stacker-reclaimer. *Constr. Mach.*, 8: 25-29.
- Wang, R., H.X. Chen and G.F. Wang, 2002. Analysis of ANSYS finite element mesh dividing. *J. Tianjin Polytechnic Univ.*, 21(4): 8-11.
- Ying, G.W., B. Li and Q.G. Liu, 2010. The development and prospect of stacker-reclaimer. *Port Operation*, 4: 1-3.
- Yu, C.W., 2005. The finite element analysis of bucket wheel stacker-reclaimer metal structure based on ANSYS. *Port Operation*, 5: 24-26.
- Yu, M.Y., 2008. Research on Performance of the Working Device of Bucket Wheel Stackers-Reclaimer. Jinlin University Press, Changchun, pp: 22-48.