Finite Element Model Establishment and Strength Analysis of Crane Boom

Linyan Zhang[1], Hongliang Zhang[1, a, *] ,Changguo Lu[1]
[1]*Department of Computing,Yingkou Institute of Technology, Yingkou,China*

Keywords: Crane Boom, Finite element analysis, Strength Analysis.

Abstract: This paper is devoted to the strength analysis of crane boom with Ansys software. The results of strength analysis and theoretical calculation are compared and analyzed, and a method of using software to analyze the strength of crane boom is discussed. Firstly, using SolidWorks software, the three-dimensional model of each jib of the main arm of heavy crane is established by means of shell pulling, which is saved in x-t format. Secondly, the three-dimensional model is imported into Ansys software, and the finite element model of heavy crane boom is established by assigning attributes and meshing. Thirdly, the crane boom priority model is constrained and loaded, and the static simulation is carried out. Finally, the simulation results and theoretical results are compared and analyzed to verify the accuracy of the model (SINGH B, et.al, 2011; TOMASZ G, et.al, 2011).

1 ESTABLISHMENT OF FINITE ELEMENT MODEL OF CRANE BOOM

1.1 Applying SolidWorks Software to Build Three-Dimensional Model

Firstly, the three-dimensional model of crane boom is established by using SolidWorks software. In SolidWorks, all boards extract the middle surface, and all tubes extract the axis to build a three-dimensional model. The model of the crane boom needs to be established, including the boom root, the boom head, the completed 3m, 6m, 12m middle boom. According to the requirement of the subject, the structure size of the arm frame of the main arm is determined. According to the size requirement, the arm head and the arm root of the main arm are drawn. The model is shown in Figures 1 and 2.

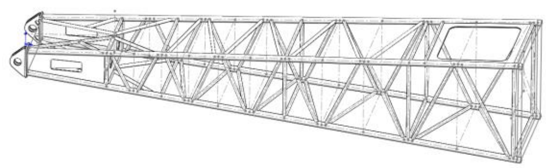

Fig 1. Arm Root.

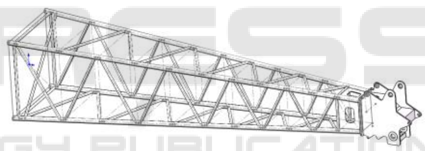

Fig 2. Arm Head.

1.2 Introduction to Ansys Software

Ansys software is developed by American Ansys Company. It is a finite element analysis software which integrates structure, fluid, electric field, magnetic field, sound field and thermal analysis. It has corresponding interfaces with most software (such as Pro/Engineer, Hypermesh, Adams, Nastran, Ideas, etc.), and can realize data sharing and exchange between them. The cell types used in this paper are shell 63 unit, beam 188 unit, link 180 unit and mass 21 unit.

1.3 A Simplified Scheme for Modeling Process Using Ansys Software

Beam 188 beam element is commonly used in Ansys to simulate the main chord and web members. This element can define the cross-section shape and also simulate the mechanics of the main chord and web

Zhang, L., Zhang, H. and Lu, C.
Finite Element Model Establishment and Strength Analysis of Crane Boom.
DOI: 10.5220/0008850001920196
In *Proceedings of 5th International Conference on Vehicle, Mechanical and Electrical Engineering (ICVMEE 2019)*, pages 192-196
ISBN: 978-989-758-412-1

members. The shell 63 element is used to simulate the plate of the arm head in the boom. The shell 63 element is commonly used in Ansys software. It is convenient to use. For other pull plates, it is simulated by link8 element, which is a three-dimensional bar element and is a tension and compression element in the direction of bar axis. Beam 188 beam element model is used to analyze the chord and web members.

1.4 The Finite Element Model of Heavy Crane Boom Is Established By Using Ansys Software

In SolidWorks, all the boards are extracted from the middle surface, and all the tube axes are extracted to build a three-dimensional model, which is saved in x-t format and imported into Ansys for modeling. In Ansys, shell 63 unit is used for board, beam 188 unit is used for tube, mass21 unit is used for key point and link 180 unit is used for pulling board.

In Ansysysy, the solid models established from Solidworks are imported to define their material properties, the real constants are given to the plates according to the thickness of the plates, and the cross-section properties of the tubes are defined according to the diameter and thickness of the tubes. Finite element mesh analysis is carried out on them, and mesh errors are checked and modified, and finite element models are established. Fig. 3 is the arm root finite element model, and Fig. 4 is the arm head finite element model.

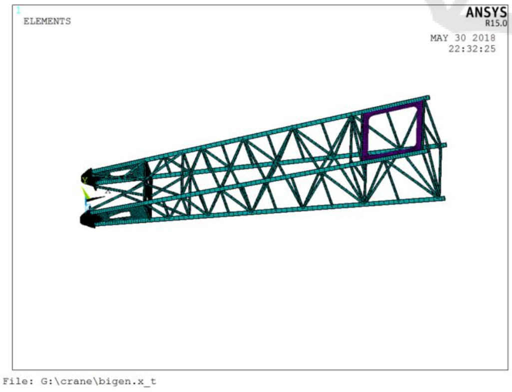

Fig 3. Arm Root Finite Element Model.

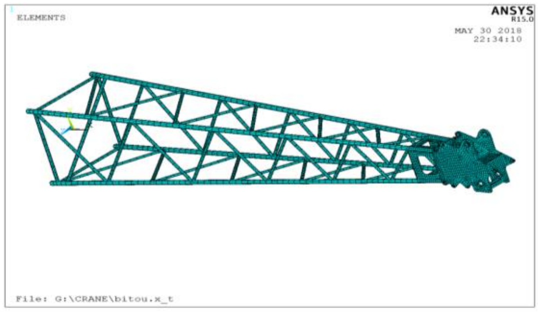

Fig 4. Finite Element Model of Arm Head.

The arm is connected with the arm root by moving, writing out and reading operation. The finite element model of the 13m basic main arm is established, as shown in Fig. 5.

Fig 5. Finite element model of 13m main arm.

2 STRENGTH ANALYSIS OF CRANE BOOM

As shown in Figure 6, the basic main arm of 13m is adjusted to the working state of minimum radius and maximum hoisting by moving and rotating. According to the position of the lower hinge point of the pulling plate in the working condition, the key points are established and the connecting arms are separately. For the two pulling plates, the link188 element can only be divided into one cell. The upper end of the drawing plate and the hole are connected by rigid area.

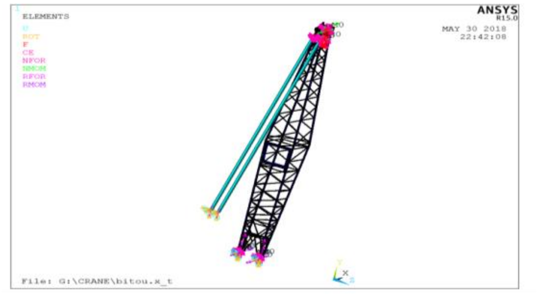

Fig 6. Force model of 13m main arm.

In the articulated position of the lower arm root and the articulated position of the upper arm head, the rigid region is established by mass21, and the rigid region is also established in the contact position of the tube and the plate. In the articulated position of the lower end of the arm root, the constraints of ux, uy and UZ displacement directions are added, and the lower end of the pull plate is fully constrained. Add 539 KN downward force to the arm lifting position, and add to the whole system.

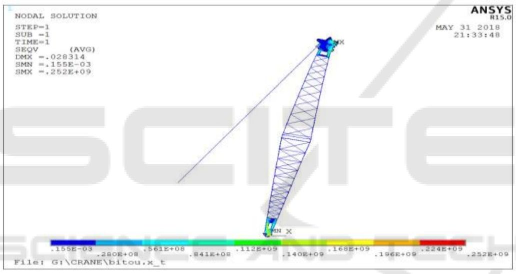

Fig 7. Total stress nephogram of 13m main arm.

From the total stress nephogram of the main arm, it can be seen that the maximum displacement of the main arm is 28.314 mm, the maximum stress is 252 MPa, and the position of the main arm appears in the arm head. See Fig. 7 for details.

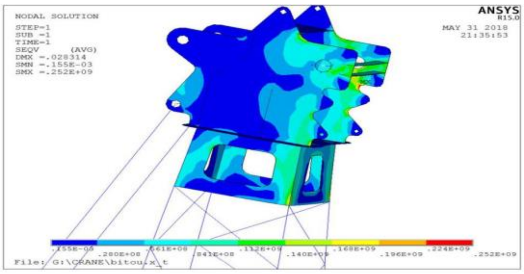

Fig 8. Maximum stress nephogram of Figure 8.

From Figure 8, it can be seen that the maximum stress position of the slab is in contact with the lifting load, and the stress near the slab is relatively large, averaging about 135 MPa. Q235 is adopted in this design, and its allowable stress is 213MPa, which is larger than the result of finite element calculation and meets the strength requirement. For the position where the maximum stress occurs due to the smaller chamfer, there is the possibility of stress concentration, so we can try to improve the design.

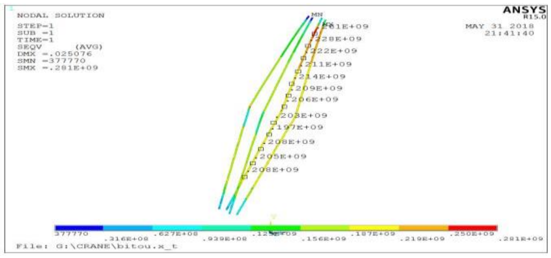

Fig 9. Maximum stress nephogram of Q345 main chord.

From the stress nephogram of the main chord, it can be seen that the maximum position of the actual stress appears on the lower main chord with the upper arm, and its maximum stress value is 281 MPa. The stress of the lower main chord is larger than that of the upper one, which accords with the fact that the upper main chord is under tension and the lower main chord is under compression. The maximum stress distribution of the lower main chord is about 220 MPa on average. The allowable stress of Q345 is 313 MPa, which is larger than the result of finite element calculation and meets the strength requirement.

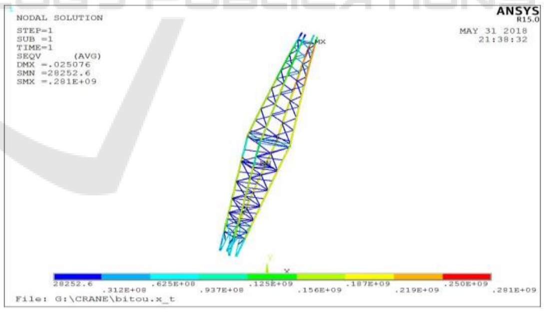

Fig 10. Stress nephogram of pipe.

Fig. 10 is the stress nephogram of the pipe. The maximum stress appears at the end of the arm. The material chosen is Q345. The maximum stress value is 281 MPa, which is greater than 213 MPa. Its strength meets the requirements.

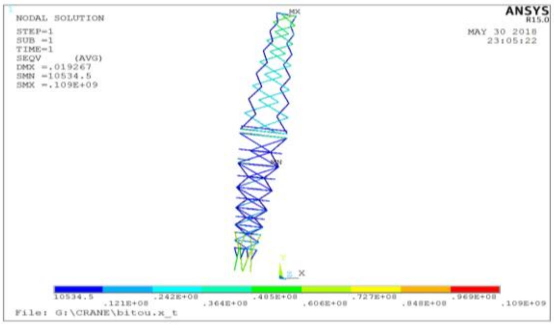

Fig 11. Nephogram of Abdominal Canal Force.

Figure 11 shows that the maximum stress of the abdominal canal occurs at the upper arm, followed by the root of the arm. The material used is Q235 and its allowable stress value is 109 MPa, which is larger than the result of finite element calculation and meets the strength requirement.

3 COMPARISON OF THEORETICAL CALCULATION AND SOFTWARE ANALYSIS RESULTS OF BOOM STRUCTURE STRENGTH

According to the theoretical calculation of plate strength and the strength analysis by ANSYS software, the results are compared. In the analysis results of Ansys software, the blue part accounts for most of the area, and the red part is the largest part of the plate. As shown in Fig. 12, when the dangerous section is 10 m away from the root of the boom:

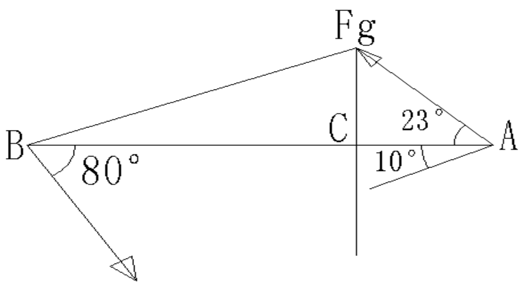

Figure 12. Hazardous Section Analysis C.

$$\sigma_1=\frac{Fa}{4\times A}=839.4/4\times 2093=100.26\text{MPa} \quad (1)$$

$$M(x)=\varphi_2 Q\cos\theta C\times AC-Fg\times AC\sin\theta_1=490\times 1.1\times \cos 80-218.21\times 3\times \sin 23$$

$$\approx 280.8-225.8=55\text{KN.m} \quad (2)$$

So

$$\sigma_2=\frac{M(x)}{Wx}=55\times 10^6/3.67\times 10^5=150\text{MPa} \quad (3)$$

Therefore, the stress of the upper and lower main chord is zero:

$$\sigma_{up}=\sigma_2-\sigma_1=150\text{ MPa}-100.26\text{MPa}=49.74\text{MPa} \quad (4)$$

$$\sigma_{down}=\sigma_2+\sigma_1=150\text{ MPa}+100.26\text{MPa}=250.26\text{MPa} \quad (5)$$

The stress of the main chord is 49.74 MPa on the dangerous section, which is 10 m away from the arm root of the boom. The stress of the lower main chord is 250.26 MPa. The result of Ansys software analysis is 242 MPa. The error is about 3.30%.

The theoretical calculation value regards the whole boom as a homogeneous rigid body, and simplifies the position of the pulling plate and the lifting load to the same point, which is different from the actual value and is not particularly accurate. Therefore, the theoretical calculation value and the results of finite element analysis have errors, but the error is within 5%. The finite element model and the calculation results can be basically considered. Correct.

4 CONCLUSION

In this paper, the three-dimensional model of crane boom is established by Solidworks software, and the finite element model of crane boom is established by introducing Ansys software. The strength analysis of crane boom is carried out by using finite element model. Finally, the strength analysis results of application software and theoretical calculation are compared. The accuracy of strength analysis results is verified. The research results of this paper have strong practical and theoretical significance for the application of crane boom strength analysis in engineering.

REFERENCES

KUMA N, PARVEZ M. Force distribution on telescopic boom of Crane [J].International Journal of Mechanical Engineering and Robotics Research, 2012, 1(2):97-104.

SINGH B, NAGAR B, KADAM B S, et al. Modeling and finite element analysis of crane boom [J]. International Journal ofAdvanced Engineering Research and Studies, 2011, 1(1): 51-55.

TOMASZ G, WOJCIECH S. Modeling and research into the vibrations of truck crane [J]. Scientific Research of the Institute of Mathematics and Computer Science, 2011, 1(10): 49-60.