Finite Element Analysis on Rig Boom Based on SolidWorks

Qiyuan Liu [1]*
[1]College of Mechanical Engineering, Hunan University of Arts and Science, Changde, 415000, China;
*Address correspondence to this author at Dingting Road, Changde, China. postcode: 415000;
Tel:13974281321;E-mail: lqy670@163.com

Youxin Luo[2]
[2]College of Mechanical Engineering, Hunan University of Arts and Science, Changde, 415000, China;

Huijun Wen[3]
[3]College of Mechanical Engineering, Hunan University of Arts and Science, Changde, 415000, China;

***Abstract*—Bracket rotary shaft, a major component of full-hydraulic drilling rig, is also a main force-bearing component in the conditions such as rock drilling and drill lifting. The work analyzed the intensity of bracket rotary shaft under the worst working condition of rock drilling rig and maximum load in each of the two working conditions. It provided theoretical basis for designing bracket pin.**

***Index Terms*—Racket rotary shaft; Finite element analysis; Solidworks**

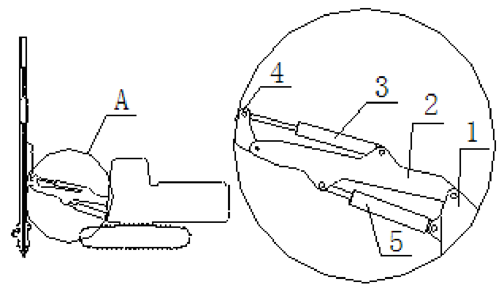
I. INTRODUCTION

As an important construction machine, full hydraulic drilling rig is widely used in mining, road and hydraulic engineering. By finite element method, drilling rig working device was analyzed to improve research efficiency and accelerate product update, thus meeting requirements of engineering construction [1-4].

After force analysis of rig working device, Solidworks was applied to establish 3D and finite element models of boom. The obtained stress and deformation nephogram provided theoretical basis for design of boom.

II. WORKING DEVICE MODEL ANALYSIS

For full hydraulic drilling rig, boom bore drill and move forces. Fig.(1) showed drilling posture of full hydraulic drilling rig. In drilling operation, working device moved by drilling and lifting, producing complicated force for boom. Boom only bore the gravity of the whole working device under moving condition.



1. Frame 2.Boom 3.Bracket cylinder 4.Bracket rotary structure 5.Boom cylinder

Fig. 1. Working device of full hydraulic drilling rig

A. Geometric Model

Research object was full hydraulic rotary drilling rig with maximum pushing and lifting forces of 15kN and 20kN in drilling operation.

Boom structure consisted of Q345D steel with yield, tensile strength and passion ratio of 345MPa, 470MPa~630MPa and 0.3. For fatigue load, safety factor was set as 1.5. Therefore, the minimum allowable strength [σ]=345/1.5=230MPa.

Boom was the main bearing structure of working device. Fig. (2) showed its finite element model.

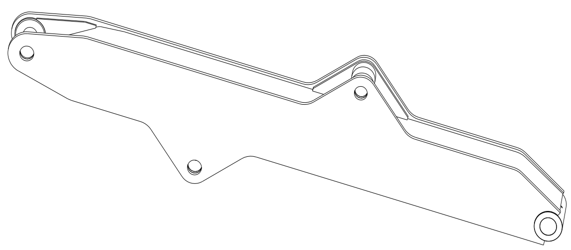


Fig. 2. Boom model

B. Load analysis under different working conditions

Full hydraulic drilling rig worked under conditions including drilling, lifting and moving. Boom bore complicated forces under drilling and lifting conditions.

Under moving condition, it only bore smaller gravity of the working device. The work used finite element analysis of boom under drilling and lifting conditions. Without structure deformation, boom cylinder was assumed to be rigidly connected. Fig. (3) showed boom force diagram.

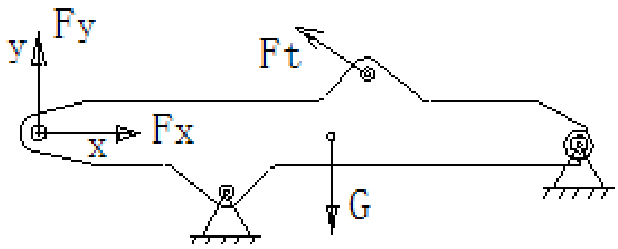


Fig. 3. Boom force diagram

(1) Drilling condition

In Fig.(1), boom cylinder applied a downward pressure to working device in drilling condition, making rig tend to pitch up for balance. Besides, pushing cylinder applied downward maximum pushing force of 15kN to drilling tool. The working device bore upward reaction force 15kN.

According to drilling rig parameters, The constraint reaction forces at hinge joints of boom were calculated and expressed (Fx=6350N, Fy=35860N, G=3300N, Ft=-4230N).

(2) Lifting condition

In Fig.(1), boom cylinder applied a downward pressure to working device in drilling condition, making rig tend to pitch up for balance. Besides, pushing cylinder applied downward maximum pushing force of 20kN to drilling tool. The working device bore upward reaction force 20kN.

According to drilling rig parameters, The constraint reaction forces at hinge joints of boom were calculated and expressed (Fx=3040N, Fy=17320N, G=3300N, Ft=-2370N).

III. STATIC ANALYSIS OF PROPELLING BEAM

A. Meshing

Meshing was the premise of finite element analysis. In solidworks, the default grid density indicator block was in the middle of the whole draw slip. In this example, the grid density indicator block was set as “fine”. The grid cell and common difference were 14.70mm and 0.735mm, respectively. Fig. (4) showed body mesh of bracket rotary pin with 213507 nodes and 122779 elements [5].

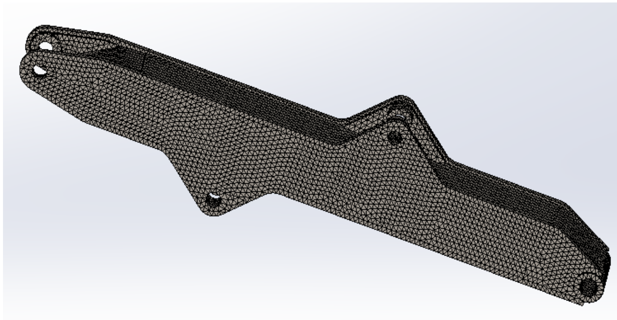


Fig. 4. Boom grids

B. Loading solution under drilling condition

(1) Fixed constraints were loaded at the hinge joints of boom cylinder and frame.

(2) External loads consisted of downward gravity load (g=9.81) and forces (Fx=6350N, Fy=35860N, G=3300N, Ft=-4230N).

(3) Fig.(5) showed boom stress distribution nephogram after finite element analysis under maximum load.

In Fig. (5), the maximum stress σ_{max} (σ_{max}=200.3MPa), produced in the front end of boom, was smaller than allowable stress of material [σ] ([σ]=230MPa).

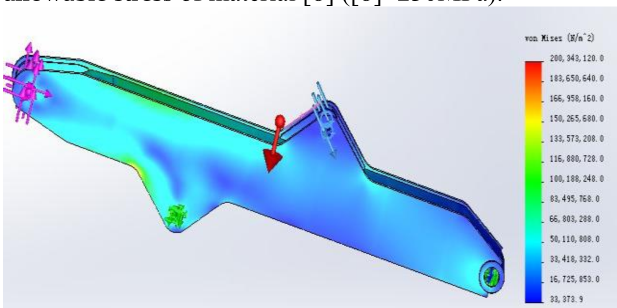


Fig. 5. Rotary shaft stress nephogram under 75kN load

C. Loading solution under lifting condition

(1) Fixed constraints were loaded at the hinge joints of boom cylinder and frame.

(2) External loads consisted of downward gravity load (g=9.81) and forces (Fx=3040N, Fy=17320N, G=3300N, Ft=-2370N).

(3) Fig.(6) showed boom stress distribution nephogram under maximum load.

In Fig. (6), the maximum stress σ_{max} (σ_{max}=96.6MPa), produced in the front end of boom, was smaller than allowable stress of material [σ] ([σ]=230MPa).

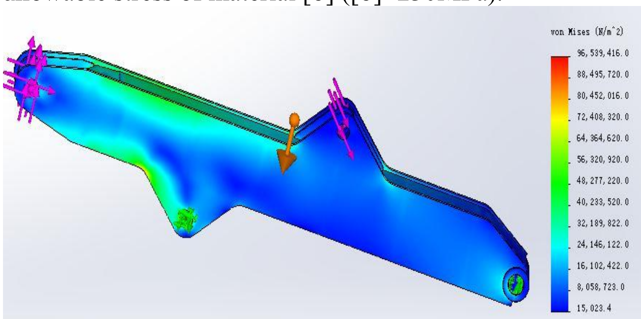


Fig.6. Rotary shaft stress nephogram under different loads

IV. CONCLUSIONS

The work used finite element analysis of boom based on two conditions of working device. Results showed that the strength of this boom accorded with the design requirements. Finite element analysis was used to check the strength of boom, thus effectively reducing design cost of drilling rig. The rig has been working in construction plant for 3 years.

ACKNOWLEDGEMENT

This research is supported by Hunan Major Special Projects of Science and Technology (2014GK1043), Industrialization Development Project of Technological Achievements of Universities in Hunan Province (15CY008), the grant of the 12th Five-Year Plan for the construct program of the key discipline (Mechanical Design and Theory) in Hunan province (XJF2011[76]), Cooperative Demonstration Base of Universities in Hunan, "R & D and Industrialization of Rock Drilling Machines" (XJT [2014] 239).

REFERENCES

[1]Z.F. Zhong. "Finite Element Analysis of Full Hydraulic Drilling Rig Based on Workbench". *Coal Mine Machinery*, vol.11, pp. 84-86, 2012.
[2]W.G. Cao, J. Song,Y.D. Dong. "Finite Element Analysis and Optimization of Rig Frame Based on HyperWorks", *Coal Mine Machinery*, vol.10, pp. 26-28, 2010.
[3]J. Liu. "Finite Element Analysis of Boom of Driving Drill". Ph. D. thesis, Jilin University, China. 2004.
[4]X.H. Guo, Y.H. Xu, H.M. Kang. "Strength Analysis of Rotary Drilling Rig Boom", *Machine Design*, vol.4, pp. 72-74, 2011.
[5]C.Q. Chen, X.Z. Ye. "Solidworks Simulation Elementary Tutorial, Beijing: China Machine Press", 2010.