Finite element analysis of working mechanism for full hydraulic backhoe loader

Li Xiaohuo[1, a], Chi Qingdi[2, b], Mu Yongcheng[3, c], Yao Tianyu[4, d]

[1,2,3,4] Mechanical Engineering College, Liaoning Technology University, Fuxin, China

[a]lixiaohuo@163.com, [b]qingdi724@163, [c]muyongcheng2006@sina.com, [d]yaotianyu85833@sina.com

Keywords: Full hydraulic backhoe loader; Working mechanism; Pro/E; ANSYS

Abstract. In order to study the stress and strain characteristics of working mechanism for a full hydraulic backhoe loader, a 3D entity model of the working mechanism was established by Pro/E, by importing the 3D model into ANSYS, the finite element model of organ was built by the interface of Pro/E, and three main parts of working mechanism were analyzed by FEM (finite element method).

Introduction

A full hydraulic backhoe loader is a new type mine excavating loading machine with high mechanization, it can be efficient and convenient to operate in harsh environment and narrow roadway. Due to poor underground environment and narrow working space underground, complex and changeable loads, forces on the working mechanism change greatly in the digging process, which will affect working performances of the working mechanism, loading capacity and service life, while the working mechanism has complex structure, large working range and complex working conditions, so it is difficult to determine loads and strength during working. However, studying force performances of a full hydraulic backhoe loader working mechanism has an important theory significance and a practical value for designing a new working mechanism, improving performances of a full hydraulic backhoe loader and extending service life of a bucket.

Model establishment

Solid model. A working mechanism of full hydraulic backhoe loader is composed of a bucket, Forearm, big arm, bucket cylinder, forearm cylinder, big arm cylinder, connecting pin shaft and other components. In order to ensure to analyze accurately and reliability a working mechanism under various conditions, a 3D model of a working mechanism of a type of full hydraulic backhoe loader is build by using Pro/E [1].

The working mechanism model in Pro/E is accurately imported into ANSYS by using interface between Pro/E and ANSYS after building the Pro/E model. In order to mesh quickly and analyze by finite element, reasonable simplifications are down under no influence on analysis results, parts like as turret cylinder, guard shield, cover, lift tank are neglected to analyze conveniently stresses of these components.

Finite element models. The material made the bucket and the arm is with 16 Mn, the performance parameters of the material are as follow: Poisson ratio is 0.3, density is 7800 kg/ m^3, yield strength is 345 MPa, tensile strength is 660 MPa, Young's modulus is 2.06×105 MPa [2]. Grids of these main parts are meshed by using adaptive meshing method, because the cross sections of the welding plates of bucket are rectangular and their lengths are much larger than the thickness, grids of bucket are meshed with 45 units of and 8 nodes [3], the bucket are divided into 8,907 units. The small arm and the big arm are welding with 10-15 mm steel plate, their thicknesses are less than 1/5 of the longitudinal length, the small arm is divided into 25,339 units, and the big arm is divided into 12,586 units.
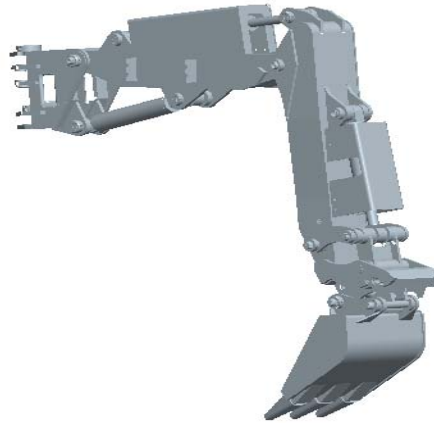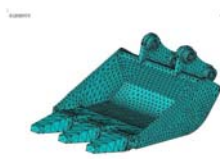
Fig.1 Solid model of working mechanism
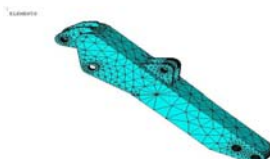

Fig.2 Mesh model of
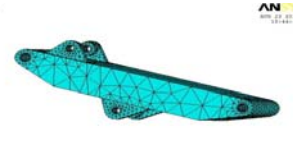bucket

Fig.3 Mesh model of
forearm

Fig.4 Mesh model of
big arm

Finite element analyses

Condition analyses. In the digging process, the bucket cylinder and the small arm cylinder are extended, the large arm cylinder is shortened, the tangential force is evenly distributed on the bucket teeth, and side direction forces are relatively evenly distributed in both sides of the bucket teeth and bucket side plates, and bear the action of gravity [4]. When the big arm cylinder reaches a limit position, digging force is supplied by the small arm cylinder and a bucket cylinder, mineral aggregate in bucket packe seriously, the force on the bucket is the maximum, which is to easy to overload, so that the working mechanism under the position is analyzed in the paper, ignoring the lateral force, taking the cylinder thrust force is 66,530 N, digging force FW =21,372.5 N, the quality of working mechanism m = 688 kg, wherein the bucket quality is 166 kg, the small arm quality is 251 kg, the big arm quality is 218 kg [5], the working condition under only positive loads is studied.

Constraints and load imposition and solution. In the process of static analysis, there is no relative move on the hinge, and it imposes the restriction on the two ear boards of bucket. According to the Saint-Venant principle, it does not affect the stress distribution that put applied loads on a bucket tooth tip. The deformation and stress cloud chart on bucket are obtained by the simulation results in the Current LS module, the maximum deformation is 6.114 mm, occurring in the bucket tooth point, and the maximum stress is 149.547 MPa, concentrating in the ear plate root.
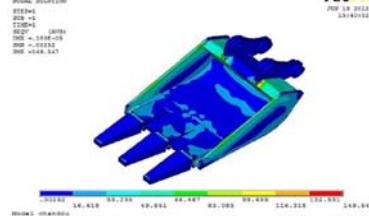

Fig.5 Figure of bucket deformation

Fig.6 Stress cloud of bucket

The arm should be bound with the hinge point node displacement in UX, UY, UZ, ROTX ROTY ROTZ, cylinder force as load. Forearm maximum deformation 3.056 mm is located in the front sleeve position, and the maximum stress concentrates on the ear plate and cylinder bearings roots, besides, the sleeve position of the cylinder block is the dangerous position and the maximum stress value is 151.922 MPa.

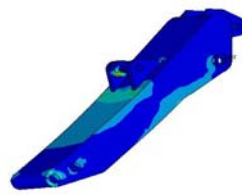Fig.7 Figure of forearm deformation

Fig.8 Stress cloud of forearm

The biggest deformation on big arm is located in the position of the front-end sleeve, and the amount of deformation is 1. 672 mm, the maximum stress (96.072 MPa) focuses on the root of the lower cylinder block.

Fig.9 Figures of big arm deformation

Fig.10 Stress cloud of big arm

Result analysis

In the digging process, because the speeds on tooth tips contacting with material surface are fast. The maximum stress is 149.547 MPa in the ear plate root, in the ear plate before and after the two periods of increased lateral ribs to reduce stress concentration, avoid plastic deformation. The forearm, as the actuating arm, deforms obviously (maximum deformation of 3.056 mm) on its front shaft sleeve, the ear plate and the cylinder seat and the shaft sleeve of the stress are concentrated; the maximum stress value is 151.922 MPa occurs in the ear plate root. The biggest deformation on big arm is 1 672 mm, stress concentration on upper and lower cylinder block is obvious; the maximum stress value on the cylinder block is 96.072 MPa. These research results are accord with the facts.

Various components of deformation are small; these do not affect the working mechanism of loader. The maximum actual stress is less than the allowable maximum stress value 345 MPa.

Acknowledgements

Conclusions

Analyses of strength and deformation of a working mechanism for full hydraulic excavating loader are done. The maximum stress concentration occurs in the bucket and arm. The maximum stress values of main components are less than the allowable stress values, the working mechanism meets the strength requirements, which have guiding significances for designing working mechanism and improving working performances of a full hydraulic backhoe loader.

References

[1] LI Xiaohuo, Wang Dan, Song Bo, He Yang. Finite element analysis of cutting mechanism of hard rock roadheader based on Pro/E and ANSYS [J]. China Mining Magazine, 2010, 19(5):

114-116.

[2] Cao Xuyang, Si Aiguo, Wang Guobiao, Xu Jinyong. Hydraulic-mechanical coupling simulation analysis of wheel loader operation mechanism [J]. China Mechanical Engineering, 2006, 17(23):2461-2464.

[3] Ren Huai wei, Wang Guofa, Fan Xun. Geometric parameters optimization of box cross section structure in hydraulic powered support based on finite element analysis [J]. Journal of China Coal Society, 2010, 35(4):680-685.

[4] Bao Jianwei, Li Jianping, Du Changlong, Hu Bin. Finite element analysis on work device of excavation type loader [J]. Coal Science and Technology, 2010, 38(8):111-114.

[5] Zhou Hongbing, Wang Huike, Guo Xinhua, Wang Shihuai. Static strength analysis of working device of hydraulic excavator based on MATLAB and ANSYS [J]. Journal of Guangxi University Nat Sci Ed, 2009, 34(6):774-779.