

Research on Optimization Design of Excavator Boom Structure Based on Finite Element Analysis

Bowen Sun¹, Bangsheng Xing^{1*}

¹School of Mechatronic Engineering, Jiangsu Normal University, Xuzhou, China

**Corresponding Author. Email: xbs138@jsnu.edu.cn*

Abstract: Excavators, as specialized equipment for energy extraction and construction in engineering machinery, play an important role in the quality and effectiveness of construction. With the gradual introduction of the concept of green environmental protection into society, energy conservation and material reduction have become one of the main development directions of excavators. This article takes a small excavator as an example to study the optimization design of its boom from the perspective of mechanical structure optimization design. SolidWorks is used to create a three-dimensional model of the boom in the excavator's working device. Ansys Workbench is used to perform finite element static analysis on the boom, obtaining the stress situation and hazardous conditions under various working conditions. The optimized position is determined through the topology optimization module. Design a response surface optimization simulation test group using Design Expert, and generate an optimization regression equation for the boom based on the results. Under the conditions of satisfying stiffness and strength, the optimal solution for boom optimization was obtained through this equation. A comprehensive and feasible method and approach are proposed for the optimization design of excavator boom by modeling the optimal solution and ultimately analyzing and verifying its correctness.

Keywords: excavator, finite element static analysis, topology optimization, response surface optimization.

1. Introduction

Since modern times, the world economy has developed rapidly, urban construction has been accelerating, and the demand for energy and resource extraction has gradually increased. To meet these development needs, construction machinery has also experienced rapid development. As a key specialized equipment in the fields of energy extraction and construction, the performance and reliability of engineering machinery have a decisive impact on engineering quality and construction efficiency. With the development of the times, the concepts of green mining and green construction have gradually entered the social vision, and environmental protection, energy conservation, and material saving are also one of the main development directions of future construction machinery. This article takes a small excavator as an example to study the optimization design of its boom, in order to reduce weight and save manufacturing materials while meeting safety requirements.

The boom is the core mechanical structural component of an excavator during excavation operations. The design of core mechanical structural components is crucial for excavators, as it directly affects the performance of excavator operations. However, it often leads to excessive design

due to the emphasis on safety performance during operation, resulting in safety performance far exceeding established goals while significantly increasing the weight of the structure and the consumption of raw materials during manufacturing. At the same time, the larger weight also requires excavators to consume more energy during operation, which is not in line with the current theme of green development and mining. Therefore, many scientists have conducted optimization design research on the boom of excavators.

Chen et al. [1] took the boom of a hydraulic excavator as the research object, optimized the thickness of each welding plate through Isight software on the premise of meeting the strength requirements, and reduced the weight of the boom by 8%. Zhao et al. [2] took the secondary arm of an excavator as the research object, and used Altair inspire software to optimize its topology. After optimization, the quality of the model was reduced by 33.6%, and the strength and stiffness met the safety requirements. Wang et al. [3] took the boom of a medium-sized excavator as the research object. The boom structure is optimized through the results of ANSYS software static analysis, and the strength and stiffness of the optimized model are improved. Pang et al. [4] took the boom of a hydraulic excavator as the research object, optimized the structure of high stress zone and reduced the thickness of welded plate in low stress zone through the finite element analysis results. After optimization, the boom mass and stress are reduced. Wang et al. [5] used the theory of controllable mechanism to carry out the lightweight design of excavator boom. After optimization, the boom mass is reduced and the performance parameters are improved on the premise of strength safety. Ning et al. [6] took the excavator stick as the research object and optimized the design of the stick through discrete element and topology optimization methods. The optimized model can reduce the weight and maintain the original level of strength. Li et al. [7] took the stick as the research object and optimized the design of the stick through the topology optimization method. After optimization, the stiffness of the stick model increased by 6.58% and the volume decreased by 18.22%. Huang et al. [8] took the boom of an excavator as the research object, obtained its stress distribution and modal shape through finite element analysis, and optimized the boom structure based on this, and the optimized structure met the safety requirements. Jiang et al. [9] took the boom of an excavator as the research object. Under the condition of meeting the stiffness and strength, the weight of the boom was reduced by 21.7% through the lightweight optimization method. Xiang et al. [10] took the boom of an excavator as the research object, and verified its lightweight model through the artificial neural network model. The results show that it is feasible to reduce the weight of the boom by the lightweight method. Bezkorovayny et al. [11] developed a method to determine the parameters of excavator working device based on mass and limit criteria, which can be used to determine the section size of boom and stick and the placement position of hydraulic cylinder. Yang et al. [12] constructed the optimization design coding system of the boom by using knowledge expression and coding processing technology, which realized the automatic optimization design of the boom, accelerated the optimization calculation speed and improved the optimization quality. Mohammad et al. [13] took an excavator stick as the research object and obtained the uncertain load acting on the stick by Monte Carlo method. The optimization results show that the weight of the stick obtained by this method is smaller and the stress value is lower. Sumar et al. [14] took the excavator arm as the research object and proposed a lightweight design scheme that still retains its strength. When the excavator arm meets the safety requirements, the mass is reduced from 4000kg to 3852kg.

Based on the analysis of the research status of the structural optimization design of the excavator working device, it can be seen that experts and scholars have conducted a lot of research on the excavator working device, used a variety of different methods to optimize the excavator boom, and achieved fruitful results. Therefore, this paper will take a small hydraulic excavator as an example, use the method of topology optimization and response surface optimization to optimize its boom, and put forward a complete set of excavator boom optimization process. Compared with the single use of

topology optimization, it is easier to obtain the optimization model, which can be used as a reference for the optimization of other working devices of the excavator.

2. Force analysis of excavator boom

In this paper, the boom of the small excavator is the integral bending boom. The advantages of the integral boom are simple structure, light weight and high stiffness. Its disadvantage is that it has few working devices to replace, and its versatility is poor. It is mostly used on excavators with similar long-term operating conditions. The three dimensional model is shown in Figure 1.



Figure 1: Three dimensional diagram of excavator boom

The typical excavation conditions of excavator excavation operation are divided into bucket excavation, stick excavation and compound excavation. Based on these three conditions, this paper will analyze the force of excavator boom under three conditions. Before analysis, it is necessary to name each hinge point and establish the X-Y axis coordinate system of the excavator, as shown in Figure 2.

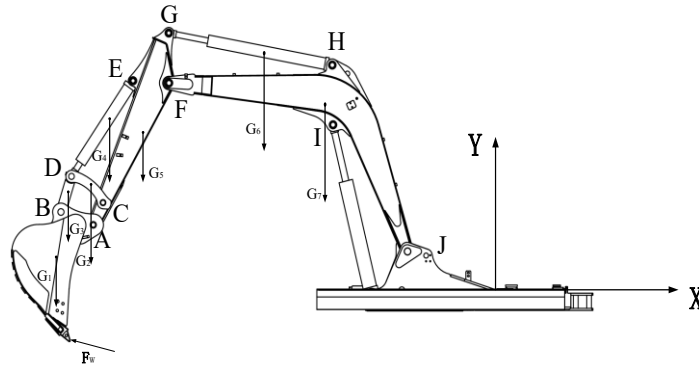


Figure 2: Coordinate system regulations for excavators

In the mechanical analysis of the boom, the first thing is to establish its force analysis model. The boom is connected to other components through four hinge points F, I, H and J. the interaction forces borne by these hinge points are located in the X-Y plane and can be characterized by their components in the X and Y directions. When carrying out finite element simulation and structural optimization of the boom, only the force conditions of its connection point F with the stick and the hinge point H with the stick cylinder need to be considered. Based on this simplification principle, the forces at F and H points on the boom can be derived from the interaction relationship between components after calculating the forces at F and G points on the stick. After the stress analysis, the stress results of each articulated point of the boom under three working conditions are shown in Table 1.

Table 1: The force distribution at each hinge point of the boom

Excavation conditions	Direction	F (kN)	H (kN)
Bucket excavation	X	-250	203
	Y	-187	183
	Z	0	0
Stick excavation	X	-165	141
	Y	-110	113
	Z	0	0
Compound excavation	X	-278	297
	Y	-172	160
	Z	0	0

3. Finite element static analysis of excavator boom

Taking the three dimensional model of the boom as the basis of the finite element analysis model, the finite element model of the boom obtained by importing the model established by SolidWorks into Ansys Workbench is shown in Figure 3.

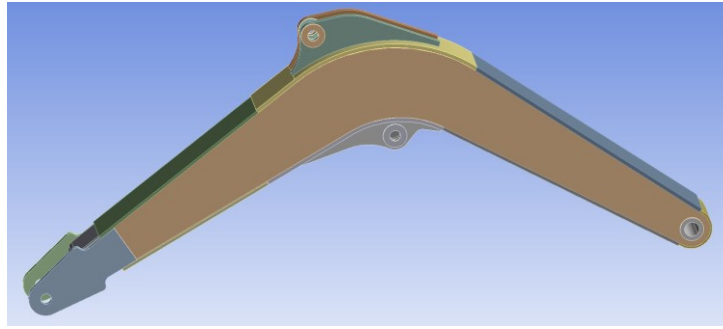


Figure 3: Finite element model of the boom

The materials used in the structural design of excavators are usually high-strength alloy plates of different specifications or precision cast steel parts. Set the default value in the Ansys Workbench material library according to its material parameters. The material data interface after setting is shown in Figure 4.

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m ⁻³		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's Mo...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0.3			
10	Bulk Modulus	1.6667E+11	Pa		
11	Shear Modulus	7.6923E+10	Pa		
12	Strain-Life Parameters				
20	S-N Curve	Tabular			
24	Tensile Yield Strength	2.5E+08	Pa		
25	Compressive Yield Strength	2.5E+08	Pa		
26	Tensile Ultimate Strength	4.6E+08	Pa		
27	Compressive Ultimate Strength	0	Pa		

Figure 4: Material data setting diagram

In the process of mesh generation, the default mesh size is set to 5mm, and sparse meshes can be used in non critical parts to reduce the computational pressure of simulation operation. Proper control

of grid density can make the simulation efficient while maintaining accuracy. The mesh division of the boom is shown in Figure 5.

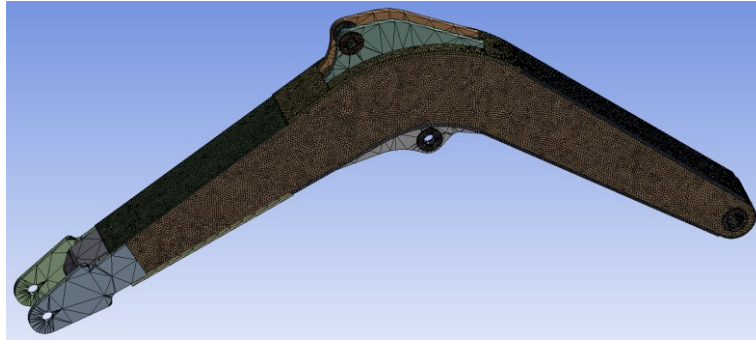


Figure 5: Boom network division diagram

The finite element simulation analysis is based on the modeling coordinate system of the model, while the stress analysis is based on the coordinate system with the gravity direction as the Y-axis. In order to ensure the consistency of the coordinate system during the simulation, the stress analysis results in the previous chapter need to be converted to the modeling coordinate system, and the transformed stress results are shown in Table 2.

Table 2: Force of each hinge point of the boom in modeling coordinate system

Excavation conditions	Point	X direction (kN)	Y direction (kN)	Z direction (kN)
Bucket excavation	F	-296	-99	0
	H	250	110	0
Stick excavation	F	-191	-52	0
	H	169	63	0
Compound excavation	F	-302	-125	0
	H	319	110	0

The working devices of excavators have large mass characteristics. In the process of finite element static analysis, it is necessary to decompose the gravity vector in X-axis and Y-axis. The effect of gravity field can be accurately simulated by applying acceleration which is opposite to the direction of gravity field and equal in size. The acceleration simulation of gravity in the modeling coordinate system under various working conditions is shown in Table 3.

Table 3: Gravity acceleration simulation value

Excavation conditions		Bucket excavation	Stick excavation	Compound excavation
Gravitational acceleration	Gx	3092	3092	1569
	Gy	9306	9306	9680

In the finite element analysis of the boom, the motion characteristics of the boom should be simulated first, and appropriate constraints should be imposed on the key nodes to ensure the accuracy of the motion behavior. For the boom structure, it is hinged to the body of the excavator and hinged to the boom hydraulic cylinder. These two places will not produce displacement, but will rotate around the hinge. Therefore, set its X, Y, Z, Rot X, Rot Y to 0, and its Rot Z to Free.

After adding materials, meshing, applying loads and constraints to the model, the finite model can be solved, and the stress nephogram and displacement nephogram of the boom model under three working conditions can be obtained.

(1) The stress nephogram and displacement nephogram of the boom under bucket excavation conditions are shown in Figure 6 and Figure 7.

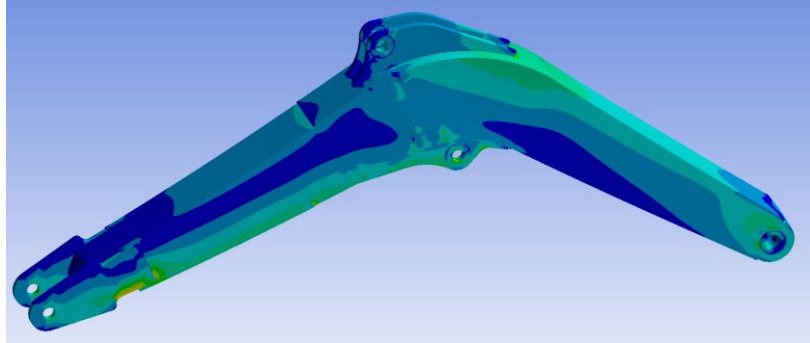


Figure 6: Stress nephogram of bucket excavation boom

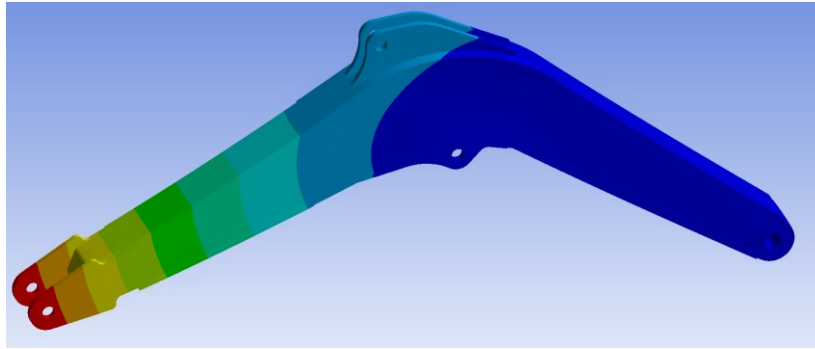


Figure 7: Displacement nephogram of bucket excavation boom

From Figure 6, it can be seen that the maximum stress on the boom is 197.56 MPa, located near the stick. Two low stress zones have appeared at the belly plate of the boom, which can be used as reference positions for optimization. From Figure 7, it can be seen that the maximum displacement is 5.9138 mm, located at the hinge between the boom and the stick.

(2) The stress nephogram and displacement nephogram of the boom under stick excavation conditions are shown in Figure 8 and Figure 9.

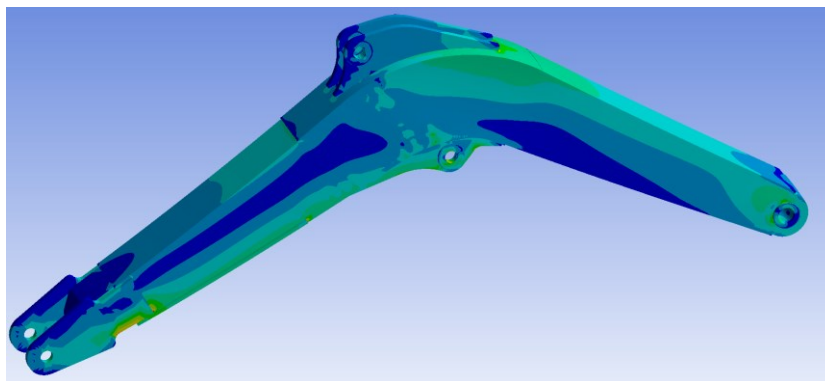


Figure 8: Stress nephogram of stick excavation boom

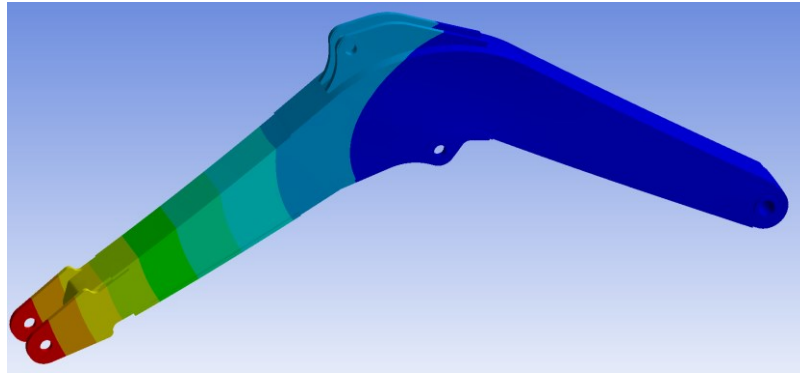


Figure 9: Displacement nephogram of stick excavation boom

From Figure 8, it can be seen that the maximum stress on the boom is 142.82 MPa, located near the stick. Two low stress zones have appeared at the belly plate of the boom, which can be used as reference positions for optimization. From Figure 9, it can be seen that the maximum displacement is 4.6828 mm, located at the hinge between the boom and the stick.

(3) The stress nephogram and displacement nephogram of the boom under compound excavation conditions are shown in Figure 10 and Figure 11.

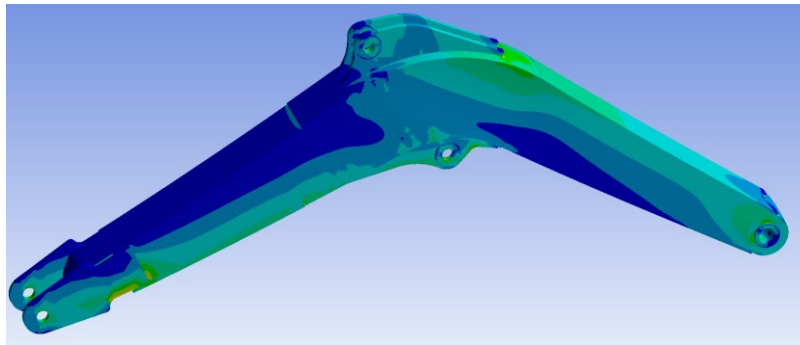


Figure 10: Stress nephogram of compound excavation boom

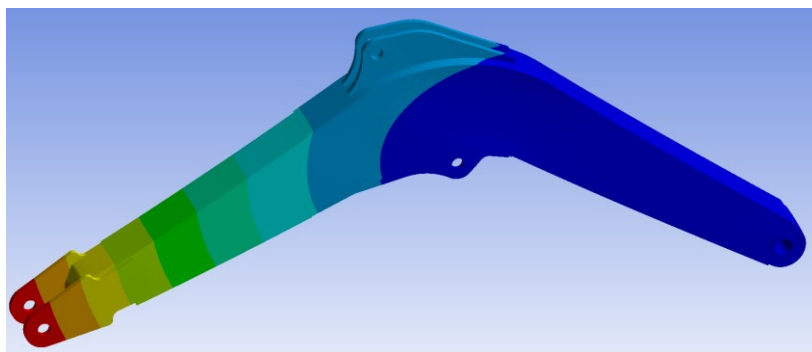


Figure 11: Displacement nephogram of compound excavation boom

From Figure 10, it can be seen that the maximum stress on the boom is 170.43 MPa, located near the stick. Two low stress zones have appeared at the belly plate of the boom, which can be used as reference positions for optimization. From Figure 11, it can be seen that the maximum displacement is 4.7345 mm, located at the hinge between the boom and the stick.

From the results of finite element analysis, it can be seen that the maximum stress of the boom under the three working conditions is 197.56 MPa under the bucket excavation condition, and the

maximum displacement is 5.9138 mm under the bucket excavation condition. According to the material properties of the boom, its yield strength can be determined to be 345 MPa. Because this article ignored the effects of bias and friction on the force analysis of the boom, a safety factor of 1.4 was selected. The allowable stress of the boom under this safety factor is 246.42 MPa, which is greater than the maximum stress under various working conditions. It is concluded that the structure meets the strength requirements. Because the excavator boom is a large-sized component, the maximum displacement values under various working conditions are much smaller than the overall size of the structure, so the structure meets safety requirements.

4. Optimization of excavator boom structure and dimensions

4.1. Topology optimization of the boom structure

Choose Ansys Workbench as the platform for optimizing the structure of the boom mechanism, establish a coupling optimization framework for the boom static simulation A module and topology optimization B module as shown in Figure 12, and use this optimization framework to solve the topology optimization results.

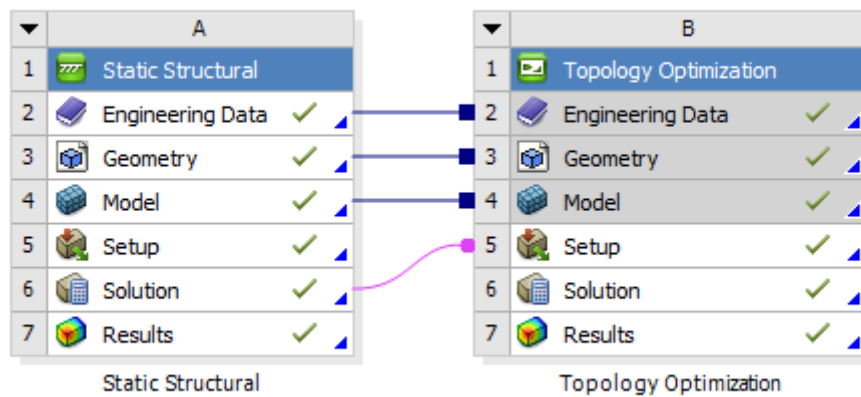


Figure 12: Coupling optimization framework

Topology optimization must clearly define the feasible region for optimization, which allows for material reduction and structural volume reduction operations. However, key areas in the structure, such as load-bearing surfaces, assembly holes, and load-bearing surfaces, usually need to maintain their integrity and are therefore not included in the optimization candidate range. Therefore, the optimization area for the boom is specified as shown in Figure 13.

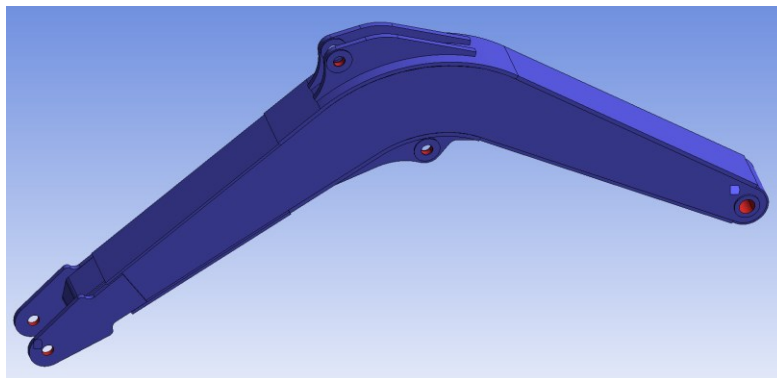


Figure 13: Optimization area of the boom

Set the number of iterations to 300, convergence accuracy to 0.1%, and quality retention to 80%, and perform iterative calculations. When solving, the working condition corresponding to the maximum stress of the boom is selected to obtain the topology optimization model of the boom as shown in Figure 14.

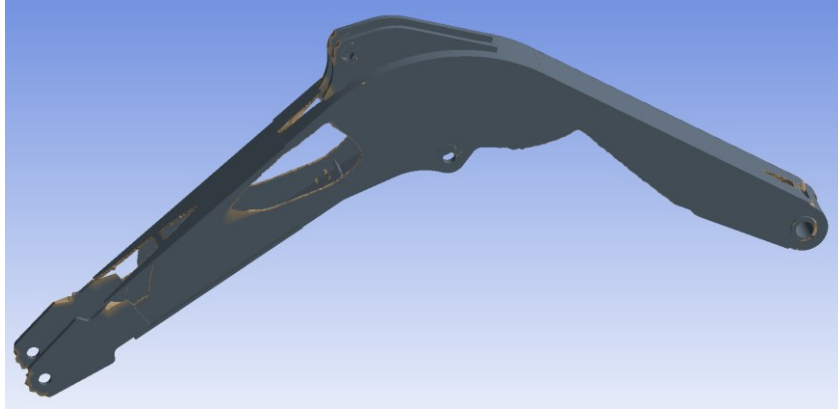


Figure 14: Results of the topology optimization model for the boom

According to the topology optimization model results of the boom shown in Figure 14, the boom optimization diagram is determined as shown in Figure 15.

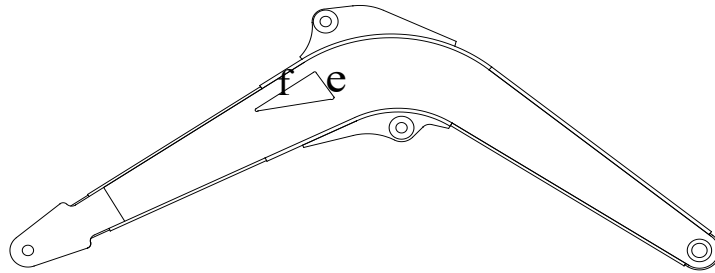


Figure 15: Schematic diagram of the boom resection model

In Figure 15, the cut portion is the lower area of the upper ear plate, which is a quasi right angled triangle area. The lengths of the two right angled sides are e and f , respectively, and the length side of f is parallel to the front upper wing plate of the boom.

4.2. Optimization of response surface for the boom size

The response surface methodology describes the relationship between variables by constructing a mathematical model, typically represented by polynomial equations. This method can continuously analyze various levels of experiments and obtain more reasonable and reliable results. This article uses a second-order polynomial model to fit the response surface.

When establishing the mathematical model of the boom, the lengths e and f of the two right angled edges in the cutting area in Figure 15, the thickness g of the boom web, the thickness h of the front section of the upper wing plate, the thickness i of the rear section of the upper wing plate, the thickness j of the front section of the lower wing plate, and the thickness k of the rear section of the lower wing plate are selected as independent variables. The range of size changes of the boom is based on the size range optimized by topology, and a seven factor three-level orthogonal simulation experiment code is set as shown in Table 4.

Table 4: Factors and levels of the boom test

Factors	Levels		
	1	2	3
e (mm)	110	165	220
f (mm)	380	490	600
g (mm)	7	10	13
h (mm)	6	9	12
i (mm)	8	13	18
j (mm)	6	10	14
k (mm)	6	10	14

Add the experimental factors and levels in Table 4 to the analysis software to generate a statistical table of response surface test results. Obtain the equivalent stress and mass results through finite element static analysis, and fill the results into the table. The results are shown in Table 5.

Table 5: Statistics of response surface test results for the boom

Number	e (mm)	f (mm)	g (mm)	h (mm)	i (mm)	j (mm)	k (mm)	P (MPa)	M (kg)
1	220	380	10	6	13	10	10	257.98	362.69
2	165	380	7	9	13	14	10	255.57	339.6
3	165	490	10	9	13	10	10	247.43	367.28
4	220	490	10	9	13	6	6	337.67	350.46
5	165	490	10	6	8	14	10	261.92	357.58
...									
55	165	490	7	12	13	10	6	305.13	328.46
56	165	490	7	6	13	10	14	311.33	335.94
57	165	490	10	9	13	10	10	247.43	367.28
58	220	490	7	9	18	10	10	307.94	341.39
59	165	600	10	9	18	10	6	250.92	367.48

Construct a linear regression model between response variables and multiple factors based on simulation experiment data using the analysis module. The results of the variance analysis of the mass and equivalent stress of the boom are shown in Tables 6 and 7.

Table 6: Analysis of variance of the boom mass

Source	Sum of squares	Free degree	Mean square error	F	P	
Model	35728.62	35	1020.82	2.254E+08	< 0.0001	significant
e	131.79	1	131.79	2.910E+07	< 0.0001	
f	105.50	1	105.50	2.330E+07	< 0.0001	
g	29522.63	1	29522.63	6.519E+09	< 0.0001	
...						
h ²	7.500E-06	1	7.500E-06	1.66	0.2109	
i ²	0.0000	1	0.0000	5.07	0.0342	
j ²	0.0000	1	0.0000	5.07	0.0342	
k ²	0.0000	1	0.0000	10.35	0.0038	

Table 7: Analysis of variance of the boom equivalent stress

Source	Sum of squares	Free degree	Mean square error	F	P	
Model	1.361E+05	35	3887.50	35.74	< 0.0001	significant
e	2474.98	1	2474.98	22.75	< 0.0001	
f	194.31	1	194.31	1.79	0.1945	
g	54341.12	1	54341.12	499.55	< 0.0001	
...						
h ²	70.15	1	70.15	0.6448	0.4302	
i ²	852.31	1	852.31	7.84	0.0102	
j ²	1900.85	1	1900.85	17.47	0.0004	
k ²	3.51	1	3.51	0.0322	0.8591	

As shown in Table 6 and Table 7, the evaluation of the mass model and equivalent stress model of the boom is significant, so the construction of the model is correct. The simulation item with P value < 0.05 in the figure is an important simulation item, which has a more significant impact on the results than the rest.

When using Design-Expert software to carry out multiple linear regression analysis on the boom, according to the generated response surface regression model and based on the simulation test data, the mathematical regression equation between the boom stress and mass and its various factors can be obtained. At the same time, the residual normal distribution of equivalent stress and mass of the boom is also generated, as shown in Figures 16 and 17.

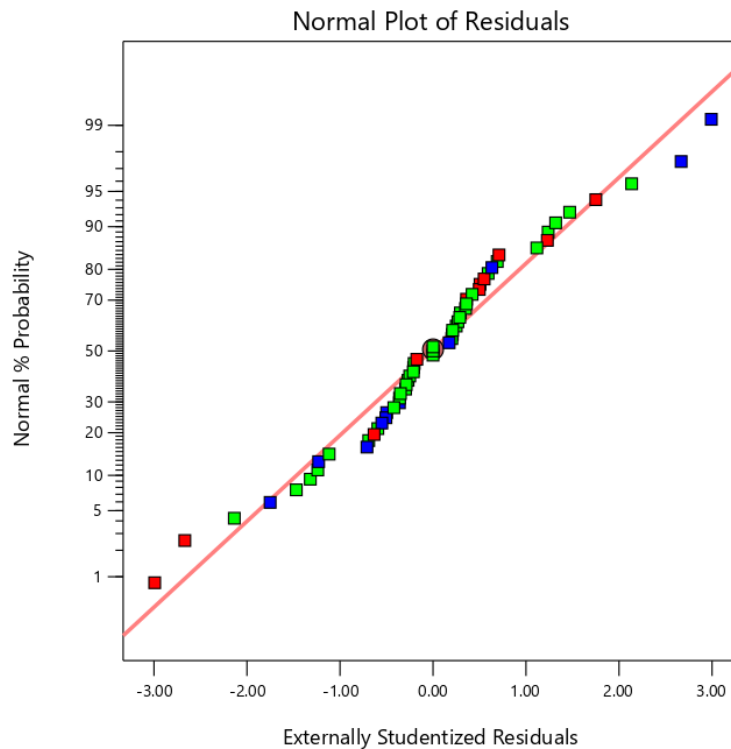


Figure 16: Normal distribution of residual equivalent stress of the boom

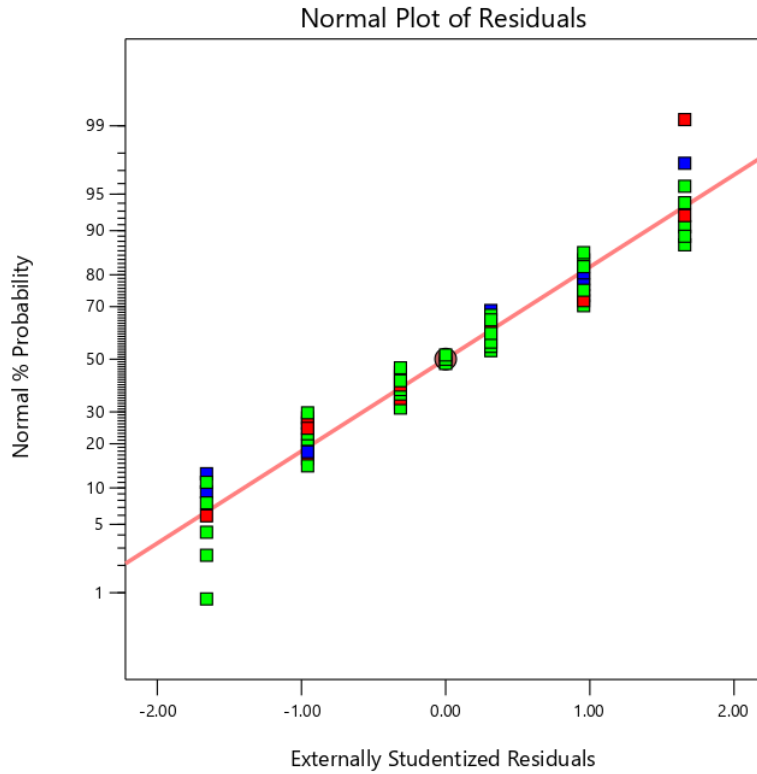


Figure 17: Normal distribution of residual mass of the boom

The linear distribution characteristics of the normal distribution of residuals can be used as an important index to evaluate the fitting degree of response surface model. It can be seen from Figures 16 and 17 that the residual data points of the boom show a good linear distribution trend, which fully shows that the established response surface model of the boom mass and equivalent stress has high fitting accuracy. Based on this, it can be considered that the calculation results of the regression equation have high reliability.

It can be concluded that the method of solving the optimal model parameters through the mathematical regression equation of the boom and obtaining the optimal parameter group is reasonable. In order to obtain the specific parameters of the optimized model, the Numerical module of Design-Expert software is used to solve the regression equation of the boom, and the optimal parameter solution of the boom can be calculated based on the objective function and constraints. The combination of optimal parameter solutions of the boom calculated by the mathematical regression model fitted by response surface software is shown in Table 8.

Objective function:

$$M = f(e, f, g, h, i, j, k) \rightarrow \min \quad (1)$$

Constraint condition:

$$\begin{cases} M \geq 0, 0 \leq P \leq 246 \\ 110 \leq e \leq 220, 380 \leq f \leq 600 \\ 7 \leq g \leq 13, 6 \leq h \leq 12 \\ 8 \leq i \leq 18, 6 \leq j \leq 14 \\ 6 \leq k \leq 14 \end{cases} \quad (2)$$

Table 8: Optimal parameter combination of the boom

Model	e (mm)	f (mm)	g (mm)	h (mm)	i (mm)	j (mm)	k (mm)	P (MPa)	M (kg)
Before optimization	0	0	10	12	18	14	14	197.56	407.6
After optimization	144.3	600	7	6	14.9	13.9	6.1	244.7	328.4
Optimized proportion %									19.31

5. Model performance evaluation of excavator boom after optimization

Through the optimal design of the boom in the previous chapter, the optimal design model of the boom is obtained. In the actual excavation operation of excavator, the working device will inevitably hit objects with high hardness during excavation, and the boom will produce certain vibration. Therefore, the optimized boom model needs not only finite element static analysis and verification, but also modal analysis to analyze its vibration characteristics.

5.1. Finite element static analysis of the boom

Because in the previous analysis, the working condition with the maximum equivalent stress among the three working conditions was selected as the dangerous working condition, and this working condition was used as the basis for optimization. Therefore, when evaluating the performance of the optimized model, the previously selected hazardous conditions are still used as the evaluation basis. The obtained boom stress nephogram and displacement nephogram are shown in Figure 18 and Figure 19.

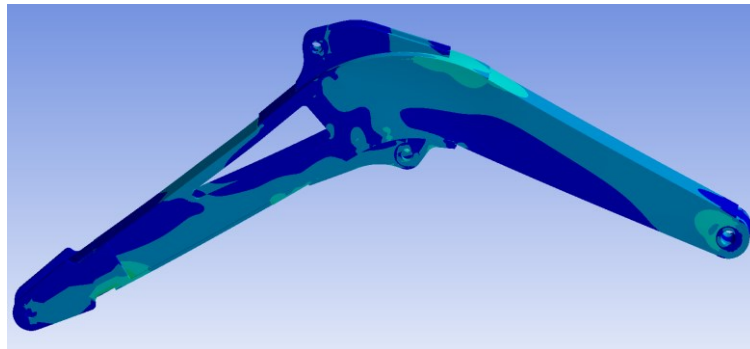


Figure 18: Stress nephogram of the boom

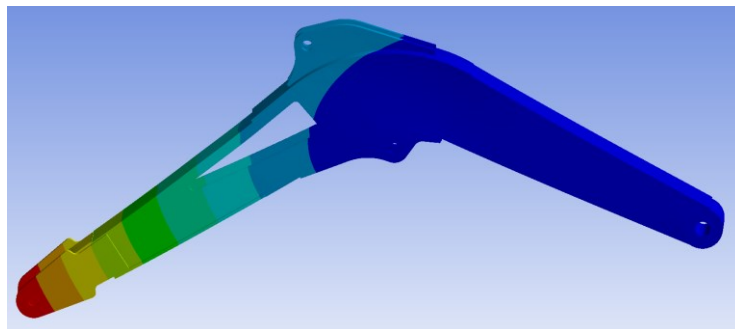


Figure 19: Displacement nephogram of the boom

According to Figures 18 and 19, the maximum stress value of the boom is located at the front of the boom, with a size of 387.2mpa, which is much larger than 246mpa under the safety factor. There is a problem here, so the image here is enlarged, as shown in Figure 20.

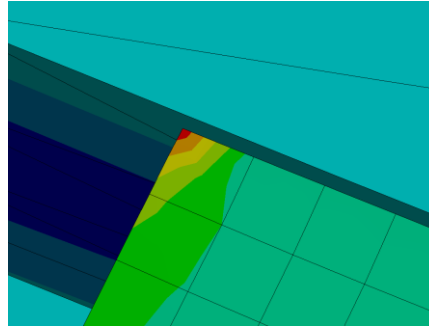


Figure 20: Location of maximum boom stress value

It can be seen from the figure that the area of maximum stress is very small, and the range is far less than one cell, and except here, the stress of the whole boom is in the normal range, so it is determined that there is a stress singularity during the simulation, which will not exist in reality, so it is excluded. The maximum displacement of the boom is at the hinge between the boom and the stick, and its value is 7.2793 mm, which meets the safety requirements. Therefore, the composite strength of the optimized boom model is required.

5.2. Modal analysis of the boom

Create a Modal module in Ansys Workbench, mesh the optimization model after importing the boom optimization model, and constrain the boom in order to simulate the impact. In the boom model, only Z-direction rotational degrees of freedom are reserved at the joints between the boom and the fuselage and between the boom cylinder and the boom, and the other five degrees of freedom are limited. No external load is set. The vibration mode diagrams as shown in Figures 21 to 26 are obtained.

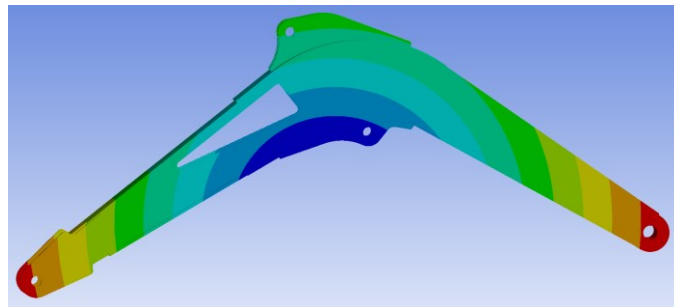


Figure 21: First order vibration mode of the boom

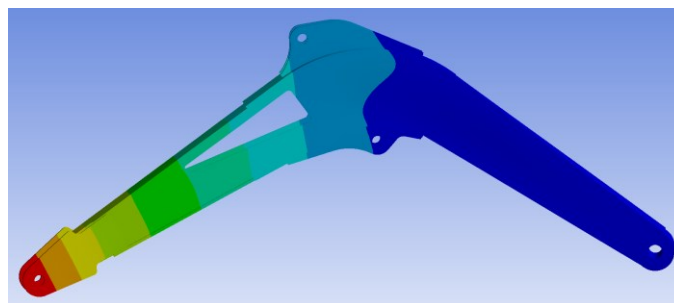


Figure 22: Second order vibration mode of the boom

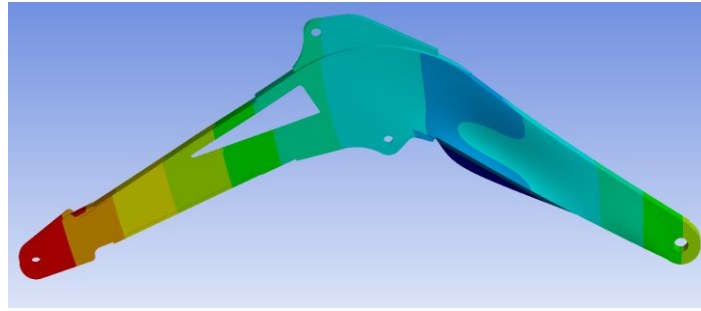


Figure 23: Third order vibration mode of the boom

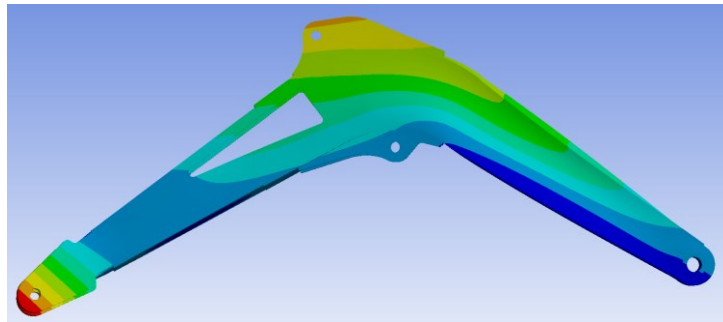


Figure 24: Fourth order vibration mode of the boom

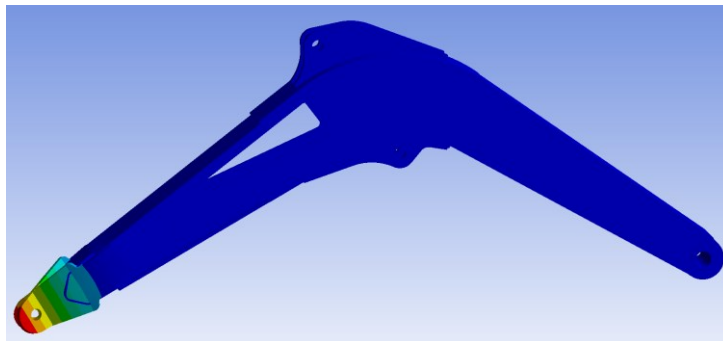


Figure 25: Fifth order vibration mode of the boom

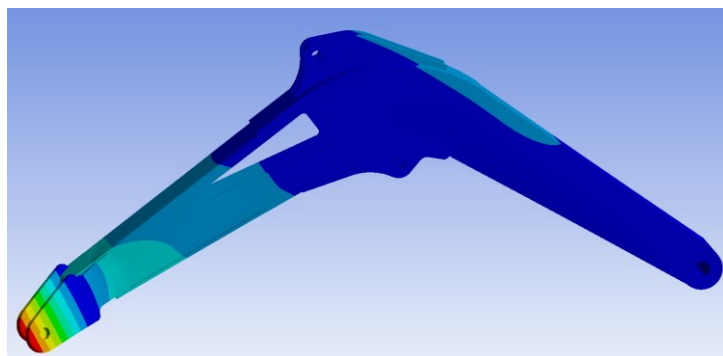


Figure 26: Sixth order vibration mode of the boom

After the boom is restrained, the first order vibration mode of the boom is the swing vibration at the hinge between the boom and the stick; The second order vibration mode is the left-right vibration of the first half of the boom; The third order vibration mode is the up and down vibration at the hinge

between the boom and the stick and at the hinge between the boom and the fuselage; The fourth order vibration mode is the left-right opposite direction vibration of the articulated part between the boom and the stick and the upper ear plate of the boom; The fifth order vibration mode is the left-right opening and closing vibration at the articulation of the boom and the stick; The sixth order vibration mode is left-right vibration at the junction of the boom and the stick and at the middle of boom. The high-order vibration mode diagrams of the boom are relatively complex. In the actual production process, appropriate additional reinforcement can be made at these positions to increase its service life.

6. Conclusion

In this paper, the excavator boom is optimized based on Ansys and Design-Expert software. The optimization conclusions are as follows:

(1) Through the results of the finite element analysis, it can be seen that the maximum stress of the boom under the three working conditions is 197.56 MPa under the bucket excavation working condition, the maximum displacement is 5.9138 mm under the bucket excavation working condition, and the bucket excavation working condition is the dangerous working condition under the three typical working conditions.

(2) Through the variance analysis table, we can know that the mathematical model is significant, which shows that the mathematical model is correct, and through the mathematical regression equation, we can get the optimal solution combination of the corresponding boom parameters. The optimized boom mass is reduced by 19.31%.

(3) The performance of the optimized boom model is evaluated, and the strength and stiffness of the optimized model are proved to meet the safety requirements through finite element static analysis. The modal analysis provides reference suggestions for the subsequent design.

(4) This paper has guiding significance for the design and performance improvement of excavator boom in the future, and also provides ideas and methods for the optimization of similar mechanical structures.

Funding

The paper is funded by the Jiangsu Normal University Graduate Research Innovation Program (2024XKT0642).

Author contributions

Xing proposed the concept and approach of research and guided its progress. Sun conducted literature search, experimental design, data collection, data analysis, and drafting of manuscripts. Xing has conducted multiple reviews of the manuscript and proposed important revision suggestions, guiding Sun to revise and improve this manuscript. All authors have read and agreed to the content of the manuscript.

References

- [1] Chen, X., Liu, Q. and Zhang, M. (2024) Multi Objective Optimization Design and Verification of Boom of Medium Hydraulic Excavator. *Construction Machinery and Equipmen*, 55, 127-130.
- [2] Zhao, Z.Y., Zhu, Y. and Cheng, X.Q. et al. (2024) Topology Optimization Design and Simulation Analysis of The Secondary Arm of Three Arm Excavator Based on Inspire. *Journal of Machine Design*, 2023, 1-6.
- [3] Wang, Z.S., Wang, Z.D. and Zhang, S.H. (2022) Optimization Design of Boom Structure of Hydraulic Excavator. *Chinese Journal of Construction Machinery*, 20, 75-80.
- [4] Pang, C.L. and Wu, S.J. (2022) Strength Check and Lightweight Design of Excavator Boom Based on Finite Element Analysis. *Equipment Manufacturing Technology*, 2, 47-49.

- [5] Wang, F.F. and Yang, L.H. (2024) *Optimization Design of Mechanical Structure of A Controllable Excavator. Construction Machinery Technology & Management*, 37, 38-41.
- [6] Ning, X.B., Liu, Y.R. and Li, J. et al. (2017) *New Structure of Large Hydraulic Excavator Stick Based on Topology Optimization Method. China Mechanical Engineering*, 28, 1936-1942.
- [7] Li, J.J. and Sui, Z. (2019) *Topology Optimization Design of Hydraulic Excavator Stick Structure. Group Technology & Production Modernization*, 36, 10-14.
- [8] Huang, J.N., Wang, X. and Liu, S.L. (2019) *Finite Element Analysis and Optimization Design of Hydraulic Excavator Boom. Tractor & Farm Transporter*, 46, 29-32.
- [9] Jiang, L.G., Chu, J.R. and Liu, P.C. (2017) *Research on Lightweight Design of Excavator Boom Structure. Agricultural Equipment & Vehicle Engineering*, 55, 79-84.
- [10] Xiang, Q., Zhang, H. and Hu, X.L. (2015) *Research on Lightweight Design Method of Hydraulic Excavator Boom Based on Ansys and Neural Network. Machine Tool & Hydraulics*, 43, 136-140.
- [11] Bezkorovayny, P.G., Shestakov, V.S. and Nesterov, V.I. (2021) *Optimization of Work Equipment Hydraulic Excavators. Mining Equipment and Electromechanics*, 158, 3-8.
- [12] Qiang, S.Y., Long, X.H. and Hui, Z.S. et al. (2021) *Knowledge-Based Structure Optimization Design for Boom of Excavator. Mathematical Problems in Engineering*, 2021, 1-14.
- [13] Jalal, M., Irina, O. and Vitaly, G. et al. (2021) *Optimization Design of The Stick of An Excavator Under Uncertain Loading (in the conditions of the Syrian Arab Republic). Matec Web of Conferences*, 341, 1-9.
- [14] Suryo, H.S., Harto, H. and Yunianto, B. (2020) *Optimization of Cat 374D L Arm Excavator Structure Topology Design Using Finite Element Methods. Rotasi*, 22, 79-86.