Design and optimization of elevator car bottom structure

Naiqiu Huang, Mengmeng Song *, Shungen Xiao, Zhipeng Zheng

School of information, Mechanical and Electrical Engineering, Ningde Normal University, Ningde, China

Abstract Taking the elevator car bottom structure as the research object, the car bottom sheet metal process structure model was established by the three-dimensional design software SolidWorks. The part connection and support were considered in the simulation, and the mesh division finite element analysis was carried out to study static stress characteristics of the car bottom structure under load. The car bottom structure is optimized by introducing crisscross weldments according to the characteristics. The simulation results show that all the performances are improved, so as to illustrate the effectiveness of structural optimization, which provides theoretical basis for car bottom structure design.

Key words: Car bottom; design; simulation; optimization.

1. Introduction

The car is an important part that passengers directly contact when riding the elevator. The bottom structure of the car carries the load in the car and the weight of the car. The design of the car bottom structure affects the safety performance, carrying capacity and riding experience. At present, there are only a few researches on the optimization of the elevator car bottom, mainly including the frame structure, material processing and force analysis of the elevator car carried out by major elevator manufacturers [1]. Li Yingjie studied th， e mechanical law of the car bottom platform under various working conditions by means of numerical analysis, and proposed that the quality of the car bottom can be reduced under the premise of satisfying safety and reliability [2].Liu Dazhu et al. used finite element and modal analysis methods to analyze key components such as the car, and proposed that the traditional car structure design could be optimized and improved and its rationality was verified [3]. Cai Jian et al. applied the research methods of structural statics and dynamics and proposed to reduce the material cost of the car frame to meet the overall safety under extreme working conditions [4].It can be seen from the above that the current industry research on the optimization of the car structure has achieved great results, but it mainly focuses on the analysis of the overall structure of the car and the optimization of the parameters of the mathematical model of the car bottom. There are few studies on the simulation optimization of the elevator car bottom. Therefore, in this paper, the model of the car bottom is established by modeling software, and the finite element static analysis method is used for analysis. According to the simulation analysis results of the car bottom structure, an optimization plan is proposed.

2. Car bottom design

In order to facilitate quantitative analysis, combined with the actual. In this paper, the research is carried out from the perspective of the actual production of elevators by manufacturing enterprises, taking the car bottom of an elevator with a main specification parameter of 1640 mm×1160 mm×3660 mm and a rated bearing capacity of 630 kg as an example. Referring to the conventional car bottom design scheme of the current manufacturing enterprises, refer to the various manufacturing parameters [5]. Based on the C-shaped steel shown in Figure 1, the car bottom design is carried out.

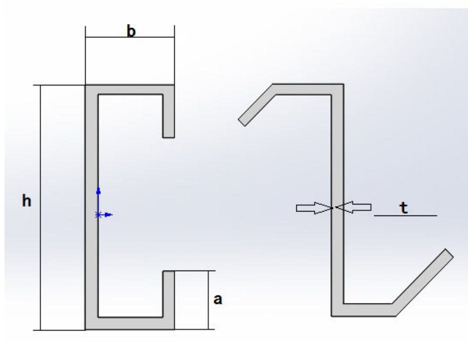


Figure 1. Outline drawing of C-shaped steel.

The car bottom is a rectangular structure, and the car support bracket and the car support C-shaped steel are used as the main load-bearing frame. Select the "Ω" type car bottom reinforcing rib with wall thickness t=2 mm[6], so that the bearing surface of the car bottom is as large as possible, and the two sides are fixed on the C-shaped groove of the bracket. The car trailer bracket adopts an

* Corresponding author: 544824964@qq.com

asymmetric C-shaped structure, one end is matched with the C-shaped groove, and the other end is matched with the reinforcing rib bottom to achieve the effect of supporting the bottom. The reinforcing rib bottom cooperates with the C-shaped steel of the car support to increase the bearing capacity of the car bottom surface. The third-layer steel structure is C-shaped steel for the car support, which increases the rigidity of the connection between the car support bracket and the bottom of the reinforcing rib. To sum up, in order to eliminate interference, all parts except the car bottom structure are compressed, and the intuitive car bottom structure as depicted in Figure 2 is obtained. In order to fit the frame, the dimensions of the parts of the car bottom and the C-shaped steel are defined in the same way: "h" is the height of the contour, "bL" and "bS" are the widths of both sides, "a" is the height of the folded edge of the parallel contour, "t" is the wall thickness of the part, and "l" is the part Length, specific dimensions are as follows:
C-shaped steel: h=110 mm, bL=67 mm, bS=47 mm, a=0 mm, t=3 mm, l=1636 mm; Car bottom reinforcement: h=85 mm, bL=73 mm, bS=73 mm, a=28 mm, t=2 mm, l=1050 mm; Car support bracket: h=100 mm, bL=105 mm, bS=30 mm, a=28 mm, t=5 mm, l=1630 mm; Rib bottom: h=108 mm, bL=37 mm, bS=37 mm, a=17 mm, t=3 mm, l=1640 mm; C-shaped steel for car support: h=120 mm, bL=50 mm, bS=50 mm, a=20 mm, t=3 mm, l=1130 mm.

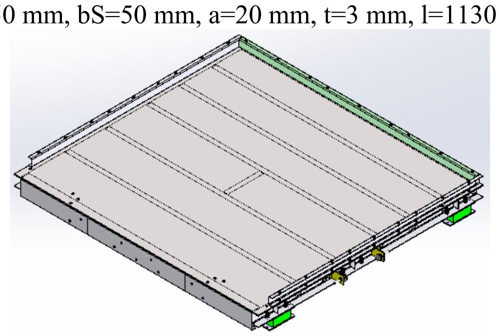


Figure 2. Structure of car bottom.

3. Simulation analysis of car bottom

3.1 Car bottom simulation

Simulation, as a simulation analysis plug-in in SolidWorks software, can perform static stress analysis on various components in mechanical design. Parts and assemblies to be analyzed can be selected locally or globally before performing a static stress analysis. To define component contact, contact sets must be defined to form connections before analysis. Fixtures describe how the model is supported, including fixed geometry, fixed hinges, elastic supports, and advanced fixtures. Generally a static stress analysis consists of at least one fixture and one external load. For the forces acting on the part, including various vector loads such as force, torque, pressure, etc. External loads should be added in the analysis process, the action analysis of no-load is invalid. The entire part must also be meshed before the static stress analysis, dividing the part into thousands of simple mesh elements. When performing control variable analysis, pay attention to the consistency of grid density, otherwise the analysis results will be affected. After running the static stress analysis, results options such as strain force, displacement, and strain appear to complete the finite element analysis of the part assembly [7-9].

3.2 Car bottom analysis

In the designed car bottom model, the structural model is simplified, and the selected Q235A material is applied to all. After the bottom plate is hidden, the rest of the parts are regarded as steel beams, and all the beam node groups are set, and the number of nodes is calculated to be 53. Use a global contact connection, use a fixed geometry fixture to make the joints hinged, and set all 53 joints to be hinged. The bearing capacity is 630 kg, that is, a force of 6174 N is applied to the bottom of the car as presented in Figure 3.Meshing using the default mesh density resulted in a total of 9105 meshes as shown in Figure 4.

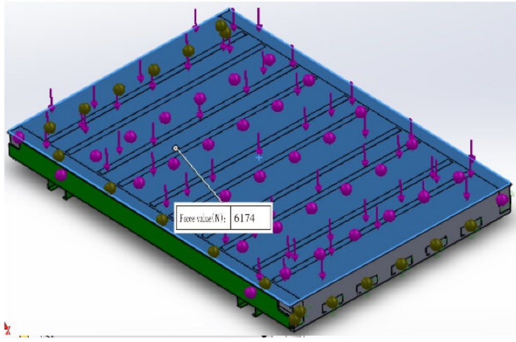


Figure 3. Load distribution diagram.

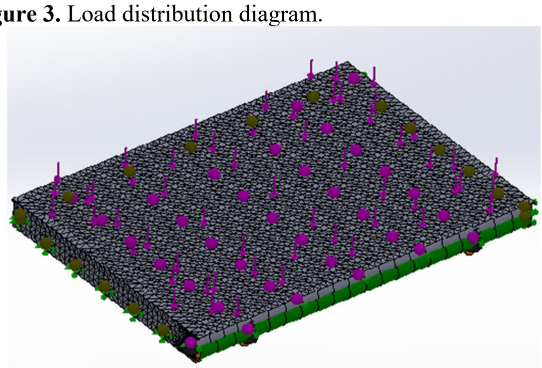
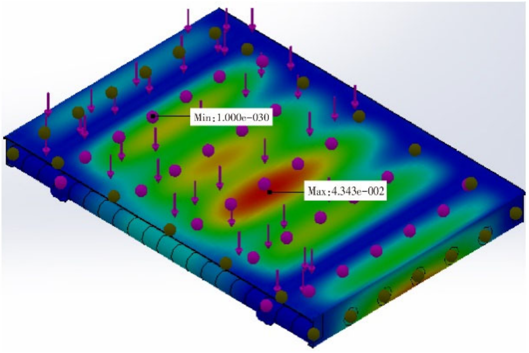
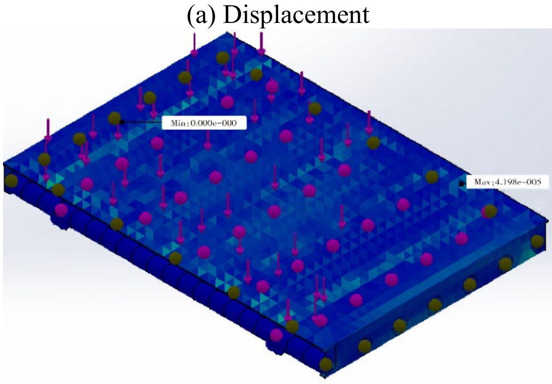


Figure 4. Mesh division diagram.

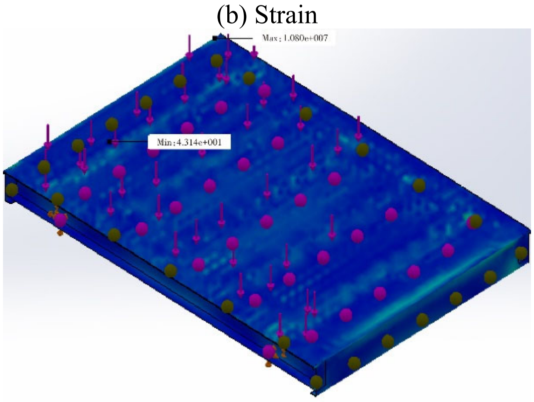
The running result is shown in Figure 5, and It is known that: The minimum displacement is 1.00e-30 mm, and the maximum displacement is 4.34e-02 mm; The minimum strain is 0.00e+00 mm/mm, and the maximum strain is 4.20e-05 mm/mm; Yield force is 2.35e+08 N/m2, minimum stress is 4.31e+01 N/m2, maximum stress is 1.08e+07 N/m2. The minimum safety factor (FOS) is 2.18e+01, and the maximum safety factor is 5.45e+06.



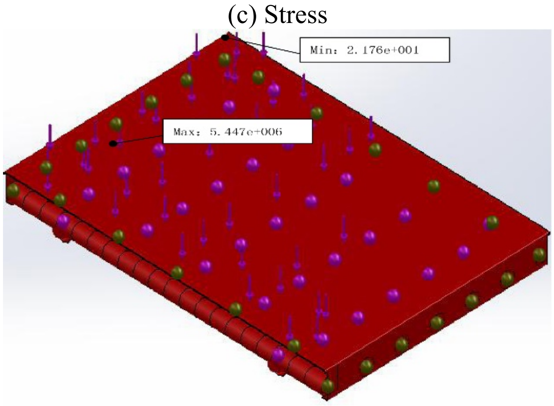
(a) Displacement



(b) Strain



(c) Stress



(d) Safety factor

Figure 5. Simulation and analysis results of the designed car bottom.

4. Car bottom optimization

4.1 Car Bottom Modeling Optimization

According to the above-mentioned static stress analysis of the designed car bottom, from the position where the maximum stress, strain and displacement occur, it is known that the optimized car bottom structure can be optimized from the structure and position of the car bottom reinforcing rib contacted by the car floor. In the case of ensuring that the weight of the car bottom structure does not increase, the welding structure is used to replace the car bottom reinforcing ribs to design a more stable load-bearing structure. According to the position distance of the original C-shaped steel, set the intersecting weldment at the diagonal of the two brackets. The midpoints of the two intersecting lines are connected to form a rectangle, and the components are square tubes with a size of 40 mm × 40 mm × 4 mm, which are matched with the end face of the C-shaped steel to generate a cross-shaped cross structure as shown in Figure 6. In the designed car bottom model, delete the car bottom reinforcing ribs, compress the shock-absorbing pad, insert the cross-shaped cross structure, and assemble as shown in Figure 7.

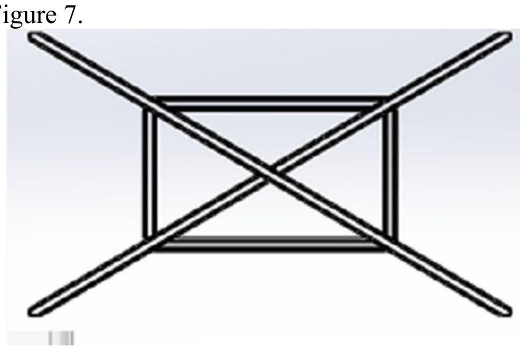


Figure 6. Cross-shaped structure.

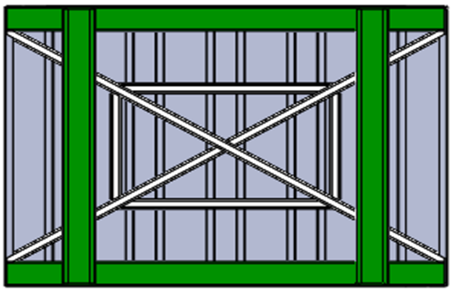
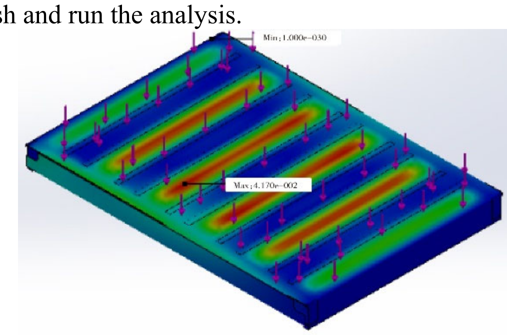


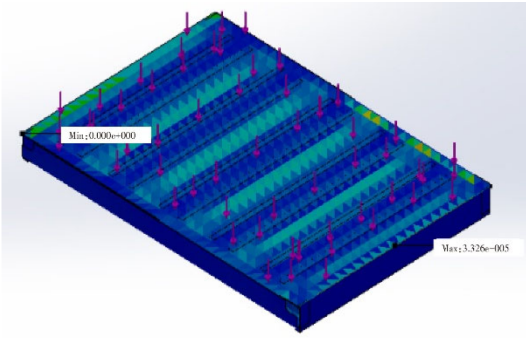
Figure 7. Optimized assembly drawing.

4.2 Optimization analysis of car bottom

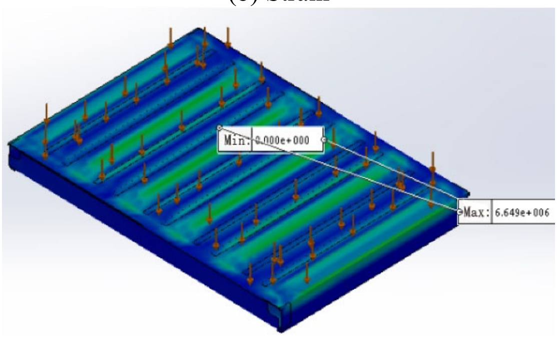
In the optimized car bottom assembly model, the parts are regarded as beams, Q235A material is selected, and fixtures and loads are added to the calculation nodes. Mesh and run the analysis.



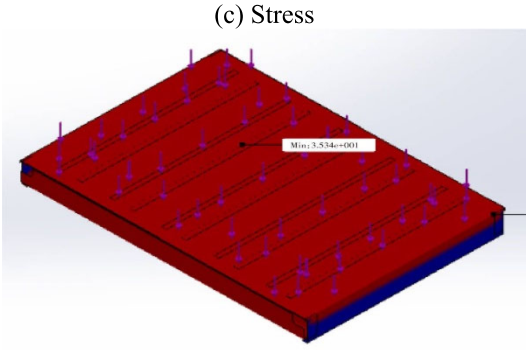
(a) Displacement



(b) Strain



(c) Stress



(d) Safety factor

Figure 8. Optimal car bottom simulation analysis result diagram.

The running result is shown in Figure 8, and It is known that: The minimum displacement is 1.00e-30 mm, and the maximum displacement is 4.17e-02 mm; The minimum strain is 0.00e+00 mm/mm, and the maximum strain is 3.33e-05 mm/mm; Yield force is 2.35e+08 N/m2, minimum stress is 0.00e+00 N/m2, maximum stress is 6.65e+06 N/m2. The minimum safety factor (FOS) is 3.53e+01, and the maximum safety factor is 1.00e+16.

4.3 Car bottom optimization comparison

Through the systematic simulation of the car bottom, the static stress analysis of the optimized car bottom structure is carried out, and the results of stress, strain, displacement, and safety factor are obtained. Table 1 shows the analysis results compared with the designed car bottom analysis results.

Table 1. Comparison of the analysis results between the optimized car bottom and the designed car bottom.

Project	Stress /（N/m^2）		Strain /（mm/mm）		Displacement /mm		FOS	
	Min	Max	Min	Max	Min	Max	Min	Max
Design	4.31 e+01	1.08 e+07	0.00 e+00	4.20 e-05	1.00 e-30	4.34 e-02	2.18 e+01	5.45 e+06
Optimization	0.00 e+00	6.65 e+06	0.00 e+00	3.33 e-05	1.00 e-30	4.17 e-02	3.53 e+01	+∞
Difference	-4.31 e+01	-4.15 e+06	0.00 e+00	-8.70 e-6	0.00 e+00	-1.70 e-03	1.35 e+01	+∞
Result	excellent	excellent	—	excellent	—	excellent	excellent	excellent

Notes: 1. Stress (N/m^2) is the internal force per unit area. The greater the stress at that point, the more dangerous. 2. Strain (mm/mm) is the amount of displacement per unit length. The greater the strain of a point along a certain direction, the greater the deformation of the point along that direction.3. Displacement (mm) is the distance that a point within a member moves in a certain direction. 4. FOS is the ratio of the yield strength of the material to the actual stress.

It can be seen from the comparison that the isosceles trapezoid structure with diagonal cross and midpoint reinforcement can enhance the stability of the car bottom and reduce the extrusion of the car bottom plate caused by the car load. The performance of the optimized car bottom in terms of stress, strain, displacement, FOS, etc. is obviously improved compared with the designed car bottom. Under the same conditions, the maximum stress is reduced by 38.4%, the maximum displacement is reduced by 3.9%, and the minimum safety factor is increased by 61.9%. At the same time, the base plate structural support adopts the national standard square tube to facilitate the production and assembly of manufacturing enterprises.

5. Conclusions

Build a model and simulate the structural design around the conventional car bottom of the manufacturing enterprise, and analyze the static state, then find out the characteristics of the car bottom structure under load. According to the characteristic research, it is shown that there is a structural change plan that increases the minimum safety factor of the car bottom by more than 50% without changing the factors that affect the simulation results of the car bottom such as quality, material, connection, etc.

Acknowledgments

This paper was supported by the following research projects: by the Natural Science Foundation of Fujian Province (Grant No. 2020J01432), Research and Innovation Team of of Ningde Normal University (Grant No. 2020T02), Ningde Normal University Serving Local Special Project(Grant No.2018ZX401).These supports are gratefully acknowledged.

References

1. Q.S.WANG, H.Y.ZHAO, Structural analysis of the elevator car frame and platform, Mechanical Research and Application. 1 (2015)133-134.
2. Y.J.LI, Strength analysis and optimization design of elevator platform based on ANSYS Workbench, Mechanical and Electrical Engineering Technology. 7(2021)227-230 .
3. D.Z.LIU, Research on structure design and analysis of a new type of household elevator.Suzhou:Suzhou university,2015.
4. J.CAI, Research and analysis of elevator structural mechanics base on ANSYS Workbench.Suzhou:Suzhou university,2016.
5. X.K.JIAN, L.WANG,Y.N.ZHAO, M.G.LI and H.C.LI, Preliminary study of lift car parametric design, one family redevelopment based on VB.NET Solidworks, China Elevator. 15(2020) 26-31.
6. Y.GAO, D.Q.JING, Y.F.KOU, Static analysis and fatigue damage research of elevator car reinforcement bar, China Elevator.1(2020):6-9.
7. Y.D.CHEN, Z.Q.BAO, H.J.REN and Y.X.WANG, Finite element analysis for product design based on SolidWorks Simulation, Computer Technology And Development.9(2012):177-180.
8. Q.S.LI, L.J.GUO,W.B.WANG, J.Z.FAN and R.R.SU. Fatigue analysis of head board of die castingmachine based on Solidworks Simulation, Journal of Mechanical & Electrical Engineering. 10(2010): 47-50.
9. H.B.JIN, X.JI Xin, Based on SolidWorks simulation finite element analysis of piston and rod of hydraulic cylinder, Journal of Machine Design.37(2020):176-179.