中國機械工程學刊第四十五卷第六期第 619~626 頁(民國一百一十三年)
Journal of the Chinese Society of Mechanical Engineers, Vol.45, No.6, pp619~626 (2024)

Optimization Simulation Analysis of Certain Electric Bus Body Structure Based on ANSYS

Yong-Gang Li*and Jian-Xin Xie**

Keywords：electric bus body framework, Static stiffness analysis, modal analysis, optimization design.

ABSTRACT

The body structure of electric bus is mainly welded by small section rectangular steel pipe, with strong bearing capacity. It has obvious advantages in structural safety, structural stability, riding comfort and other aspects, and is currently widely used in electric buses. This article takes the body framework of an electric bus as the research object, and based on the characteristics of body structure design, uses software such as SolidWorks and ANSYS Workbench to establish a parameterized bus body structure model. Taking the reliability of the electric bus body as the research objective, under the premise of meeting the overall strength and stiffness of the structure, local optimization was carried out on the electric bus body structure. Static mechanical performance analysis and modal analysis were conducted on the optimized body structure. Through comparative analysis of data before and after optimization, it was found that the optimized electric bus body skeleton met the requirements of stiffness and strength, indicating that the optimization of the body skeleton was relatively successful.

INTRODUCTION

At present, relatively mature experience has been gained in the research on car bodies.Wang et al. analyzed the lightweight research of new energy buses from four aspects: body, chassis, electrical system, and powertrain. The research results showed that replacing ordinary steel materials with high-strength materials can reduce the weight of the new energy bus body skeleton by 15.89%, and using aluminum alloy materials can reduce the weight of the body skeleton by 40.52%.(Wang et al., 2016) Wei Qingtan used CATIA software to create a three-dimensional model of the LCK6660EVG body of a light electric bus. He preprocessed the bus body using Hypermesh software and analyzed the stiffness and strength of the body under four working conditions, established an approximate model, and used the NSGA-II algorithm for lightweight design, achieving the goal of lightweight and reducing the weight of the body by 10%.(Wei., 2017) Fu et al. developed an aluminum steel composite material for the structure of electric bus bodies and carried out corresponding structural optimization while meeting the constraint conditions. (Fu et al., 2019)Wang Dengfeng, Mao Aihua, and others combined topology optimization and sensitivity analysis to achieve lightweight multi-objective optimization design of the body frame of pure electric buses. In terms of research and design, the use of topology optimization design to reduce the excess materials of various components and body frames and enhance local performance has achieved good lightweight effects. (Wang et al., 2017)Fuyuan Fang et al. established a rigid body topology optimization model for the body structure of a micro electric vehicle under various working conditions, with maximum weight stiffness as the optimization objective and node displacement and low order modal frequency as constraint conditions, and completed the topology optimization design of the body. (Fu et al., 2010)

The above research results provide a theoretical basis for the design of the body structure of electric buses, but currently, previous design experience and methods are mostly used for the design of electric body structures. Due to the significant differences in vibration frequency and other related characteristics between buses powered by internal combustion engines and electric buses powered by electric motors, simply using previous methods for body design will inevitably result in significant errors. Therefore, it is necessary to consider the power source characteristics of electric buses, summarize design methods, and achieve the goal of improving the level of electric body design and development.

Paper Received July, 2023. Revised January, 2024. Accepted April, 2024. Author for Correspondence:Yong-Gang Li.

* *Associate Professor,College of Intelligent Manufacturing, Qingdao Harbour Vocational & Technical College, Qingdao,Shandong 2666427,China.*

** *Associate Professor,College of Intelligent Manufacturing, Qingdao Harbour Vocational & Technical College, Qingdao,Shandong 2666427,China.*

OPTIMIZATION AND MODEL ESTABLISHMENT OF ELECTRIC BUS BODY FRAMEWORK

Body frame optimization

Based on the macro structural design principles and micro structural design principles of optimizing the structure of the electric bus body skeleton, the middle column of the side window, the side wall column of the vehicle body, and the lower beam of the side window are optimized. At the same time, the parts with high stress and severe deformation of the vehicle body skeleton are used as reference and optimization variables to optimize the vehicle body skeleton.(Liu et al., 2017) The optimization results are shown in Figure 1.

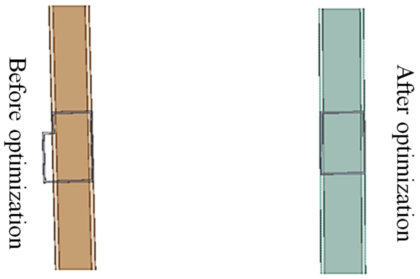


Fig. 1. Optimization results.

Establishment of an optimized model for electric buses

Build a three-dimensional model of an electric bus using SolidWorks software, as shown in Figure 2. In the process of establishing a physical model, some unsupported structures and microstructures on the bus body skeleton have little impact on the stress distribution and skeleton deformation. If it is established during the modeling process, it will only increase the complexity and computational complexity of the model, and in severe cases, it may even affect the accuracy of the calculation results, making it impossible to calculate the correct results. Therefore, during the modeling process, the model was simplified.(Yin et al., 2018)

The bus body skeleton is generally a rigid structure composed of hollow steel pipes and other components that are riveted or welded. When analyzing, it is generally assumed that the welding between the skeleton components is safe and reliable, and there will be no problem of unreliable connection between the skeleton components due to the weak connection between the welding points.(Fan et al., 2022) Therefore, when conducting body structure simulation, the connection method of the bus body skeleton structure can be treated as an integrated body equivalent method(Feng et al., 2020; Huang et al., 2018).

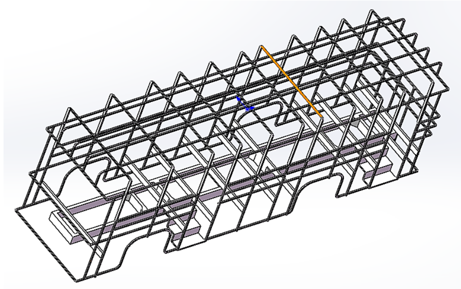


Fig. 2. Bus body skeleton model diagram.

STATICS ANALYSIS OF OPTIMIZED ELECTRIC BUS BODY FRAME

For the bus body skeleton, on the one hand, it is required that the bus should have a certain structural strength in a normal environment, and the skeleton should not break, meeting the basic strength, that is, meeting the requirements of strength verification; On the other hand, when a traffic accident occurs accidentally while the vehicle is driving, the body of the bus should have sufficient structural strength to ensure the safety of passengers, leaving enough space for personnel inside the bus, especially the space on the top of the vehicle, which is called "upper strength", that is, the body should meet the requirements of stiffness verification. So in static analysis, it is necessary to verify the stiffness and strength of the passenger car(Peng et al., 2019; Zhu et al., 2020.).During the daily operation of passenger cars, due to the large number of vehicles and the uncertainty of traffic conditions, the body skeleton may have different stress modes and deformation degrees. Therefore, four different working conditions of the passenger car were selected for simulation analysis.(Wang et al., 2019; Wang et al., 2018).

Analysis of Electric Bus Body Model

The finite element method discretizes the entire solution area into a finite number of interconnected elements, and then approximates each element to obtain an approximate solution for the entire solution area. And the elements can be divided into beam elements, shell elements, and solid elements. Both beam element and shell element are used in the finite element model of bus frame structure. The research objective is to use shell element modeling for the body of electric buses, and the thickness of the original bus skeleton structure square steel simulated by the shell element is small and consistent. Therefore, in the finite element solver, the position of the shell element is set at the middle thickness position of each square steel. Each square steel in the bus frame structure is extracted from the middle surface and given its original thickness properties to simulate the original square steel structure. The main square steel and other thin wall components of the body frame structure use shell elements, dumpling chains and other solid elements, and hooks and other components use beam elements. (Chen et al., 2022; Han et al., 2022)

The shape of the structural elements of the vehicle body skeleton is mainly composed of quadrilateral elements. Based on the size of the elements and the actual engineering situation, the finite element analysis model of the electric bus skeleton structure is divided into 10mm meshes. When dividing the mesh of the bus skeleton structure, it is necessary to conduct element quality checks at all times to ensure the accuracy of the finite element model.

There are generally two connection methods for the skeleton structure of electric buses, one is the welding seam connection between square steel; One way is through bolt connections between bolt holes. There are two connection methods for weld seam connection: one is to simulate the weld seam between two components through a common node of elements, and the other is to simulate the weld seam between two components through the connection of rbe2 elements. Due to the phenomenon of excessive local stress caused by simulating welding seams with rbe2 elements, it is advisable to use a common node method to simulate welding seams within each large section of the bus skeleton structure. Depending on the actual situation, the common node method or rbe2 method can be chosen to simulate welding seams between each large section. For bolted connections, at least 6 nodes and a layer of Washer simulated bolt holes are required. Bolted connections only consider the connecting components, without considering the influence of the bolts themselves. The rbe2 calculation method is used to connect the central node to the nodes around the first and second layers of bolt holes.

Load application of electric buses

When a passenger car is actually driving on the road, in addition to being subjected to the weight of its own skeleton, it also needs to bear the load of passengers, drive motors, air conditioning, batteries, dashboard, glass, doors, skin and other components. In the finite element analysis of buses, loads can generally be divided into concentrated loads and distributed loads based on their actual distribution. For heavy components such as drive motors, air conditioners, and batteries, a concentrated load method is used to load them, that is, a mass point located at the center of mass is used to replace the actual components. For components such as glass, skin, and doors, uniform loading is adopted, that is, mass points are used to simulate the uniform distribution of component mass on the corresponding nodes of the bus frame. The two loading methods for bus counterweights are shown in Table 1. It is worth noting that after the vehicle skeleton is weighted, it is necessary to ensure that the spring mass and center of mass coordinates of the passenger car are completely consistent with the actual vehicle.

Table 1. Electric Bus Counterweight Loading Method.

Loading method	Component	Weight（t）
Concentrated load	Passengers+Drivers	5.651
	Passenger seat+driver's seat	0.345
	Battery pack	1.888
	Storage battery	0.068
	Drive motor	0.296
	Air-conditioning	0.345
	Air storage tank	0.040
Uniformly distributed load	Glass	0.398
	Skin	0.120
	Car doors	0.170
	Floor	0.289
	Instrument panel	0.026

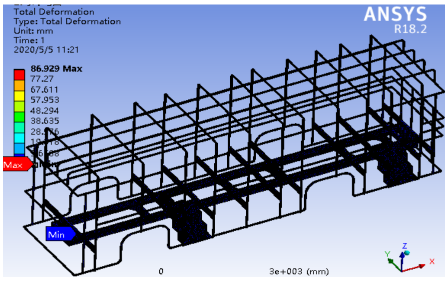
The skeleton weight of the fully loaded pure electric bus is 2.212 tons, and after counterweight, the total weight reaches 15.391 tons.

Horizontal bending conditions of passenger cars under full load

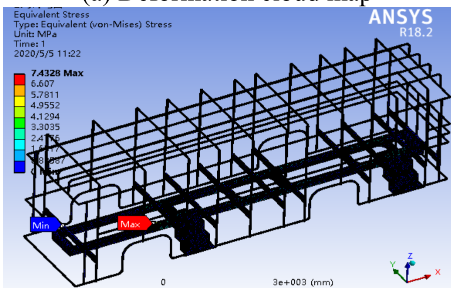
The horizontal bending condition refers to the symmetrical vertical load that a bus is subjected to while driving on a flat road surface. The various static loads and vertical accelerations that the bus body bears determine the degree of bending deformation of the bus body, which is a common driving condition during bus operation.

The load borne by the bus body skeleton is mainly generated by the mass of passengers, drivers, body, seats, etc. under the action of gravitational acceleration. When simulating horizontal bending conditions, the ANSYS Workbench is used to apply loads, and the static load can be multiplied by an appropriate dynamic load coefficient to equivalent the dynamic load. The dynamic load coefficient of vehicles varies on different road surfaces,This article takes the dynamic load coefficient k=1.3 based on the B-level road surface and simulates the horizontal bending condition.

In the simulation analysis, the degrees of freedom of the four support points are limited, and the degrees of freedom of the X, Y, and Z axes of the left and right suspensions of the front axle are also limited. The degrees of freedom of the X and Z axes of the left and right suspensions of the rear axle are also limited, and an acceleration of 2.5g in the negative direction of the Z-axis of the bus skeleton structure is given in ANSYS Workbench software. Under horizontal bending conditions, the optimized deformation and stress cloud map of the electric bus body frame is shown in Figure 3, and the optimized deformation and stress cloud map of the body frame is shown in Figure 4.

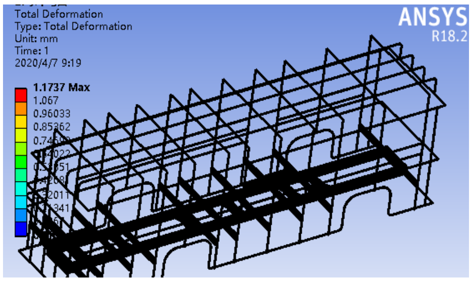


(a) Deformation cloud map

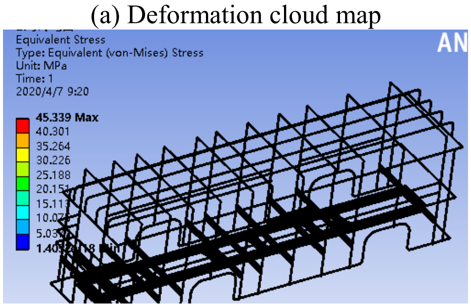


(b) Stress nephogram

Fig. 3. Cloud Chart of Deformation and Stress of the Bus Body Framework before Optimization under Horizontal Bending Condition.



(a) Deformation cloud map



(b) Stress nephogram

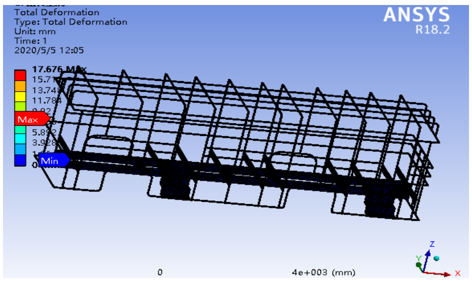
Fig. 4. Optimized deformation and stress cloud map of the bus body skeleton under horizontal bending conditions.

From images 3 and 4, it can be seen that the deformation of the upper part of the front end of the bus frame is the most obvious, with a deformation of 86.93mm, and the optimized deformation is 1.1737mm. The maximum stress of the bus body skeleton is 7.43Mpa, and the optimized one is 45.34Mpa, both lower than the yield strength value of material Q235, which is within the safe range and meets the strength requirements. According to the degree of deformation of the body skeleton, the optimized electric bus meets the stiffness requirements under this working condition.

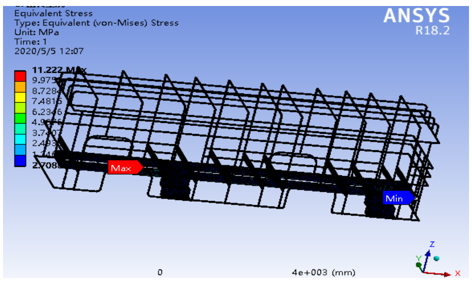
Torsional operating conditions when the passenger car is fully loaded

When a bus travels on uneven roads at a lower speed, the body frame is prone to deformation, and the deformation of the bus body is most severe under torsional conditions. Research has found that the inertia load gradually decreases with the increase of dynamic load under this working condition, and a series of other experiments have also verified this characteristic of inertia load. Therefore, under torsional working condition, dynamic load is unstable and cannot be directly used to evaluate the mechanical characteristics of the vehicle body skeleton. Instead, static torsional vehicle body strength needs to be used for evaluation.

In the simulation analysis, the degrees of freedom of four support points were limited and analyzed under the right torsion condition (right front wheel suspended). When the right front wheel is suspended, the left wheel of the rear axle sinks. The front axle right suspension does not limit the degrees of freedom, but limits the degrees of freedom of the X, Y, and Z axes of the rear axle left suspension, and limits the degrees of freedom of the X, Z axes of the front axle left suspension and the Y, Z axes of the rear axle right suspension. Under torsional working conditions, the deformation and stress cloud maps of the optimized electric bus body skeleton are shown in Figure 5, and the deformation and stress cloud maps of the optimized body skeleton are shown in Figure 6.

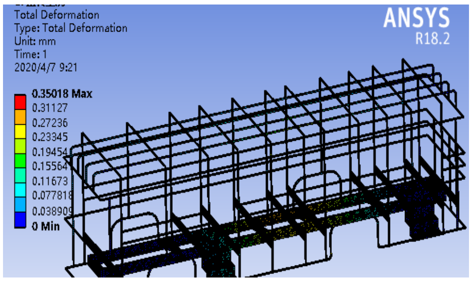


(a) Deformation cloud map

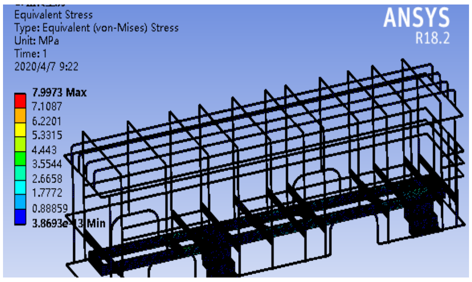


(b) Stress nephogram

Fig. 5. The deformation and stress cloud maps of the bus body skeleton before optimization under torsion conditions.



(a) Deformation cloud map



(b) Stress nephogram

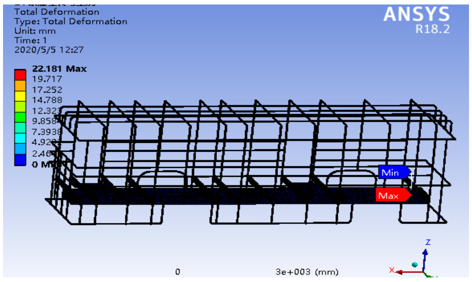
Fig. 6. Torsional working condition optimized deformation and stress cloud maps of the bus body skeleton.

From figures 5 and 6, it can be seen that the deformation of the upper right part of the front end of the vehicle body is the most obvious, with a deformation of 17.676mm. After optimization, the maximum deformation is 0.35mm. The deformation of the frame at this time is a joint result of the deformation of the leaf spring suspension under load and the loss of road support for the right front wheel. Overall, the body structure meets the requirements for strength and stiffness. The maximum stress of the frame is 11.222Mpa, and the optimized maximum stress is 7.99Mpa, both lower than the yield strength value of 65Mn material, which is within the safe range and meets the strength requirements. According to the degree of frame deformation, the optimized electric bus meets the stiffness requirements under this working condition.

Emergency left turn conditions when the bus is fully loaded

When the bus turns left, there will be lateral loads acting on the body itself. When turning, the magnitude of centrifugal acceleration is determined by the driving speed and turning radius. For the convenience of calculation, when simulating an emergency left turn condition, a lateral acceleration of 0.4g is applied to the right side of the bus, pointing towards the left side of the bus.

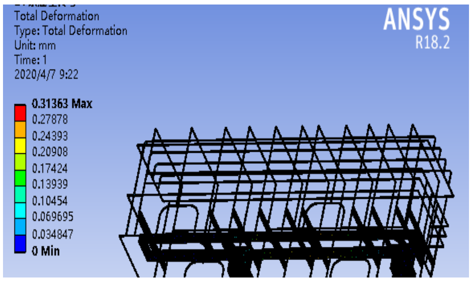
In the simulation analysis, the degrees of freedom of four support points were limited, including the degrees of freedom in the X, Y, and Z directions of the front axle left suspension, the degrees of freedom in the Y and Z axes of the front axle right suspension, and the degrees of freedom in the Y and Z axes of the rear axle left and right suspension. Under emergency left turn conditions, the deformation and stress cloud maps of the electric bus body skeleton before optimization are shown in Figure 7, and the deformation and stress cloud maps of the optimized body skeleton are shown in Figure 8.



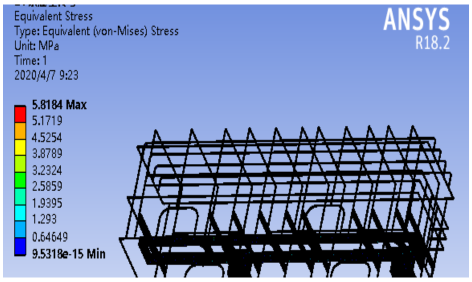
(a) Deformation cloud map

(b) Stress nephogram

Fig. 7. Deformation and stress cloud maps of the bus body skeleton before optimization under emergency left turn conditions.



(a) Deformation cloud map



(b) Stress nephogram

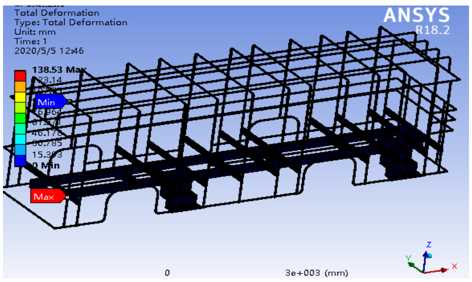
Fig. 8. Deformation and stress cloud maps of the bus body skeleton after optimization under emergency left turn conditions.

From images 7 and 8, it can be seen that the maximum deformation is 22.181mm. The main reason for the large deformation is the large force on the bottom, which leads to the large deformation. After optimization, the maximum deformation is 0.314mm. Overall, it can be inferred from the degree of deformation of the body skeleton that the passenger car meets the stiffness requirements under this working condition. The maximum stress of the bus skeleton is 7.608 Mpa, and the optimized maximum stress is 5.818 Mpa, which is within the safe range and meets the strength requirements. According to the degree of deformation of the body skeleton, the optimized electric bus meets the stiffness requirements under this working condition.

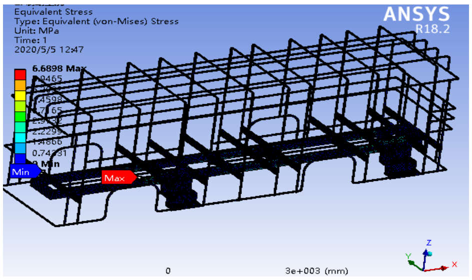
Braking conditions when the bus is fully loaded

During the operation of passenger cars, due to the complex road conditions and environment, emergency braking is often required. During emergency braking, the body skeleton will bear longitudinal loads. The size of the longitudinal load is determined by the braking deceleration and the mass of the vehicle, and the inertia force during braking also depends on the braking deceleration. In simulation analysis, it is assumed that the front and rear wheels lock simultaneously during emergency braking, and longitudinal acceleration is applied to the vehicle body skeleton to simulate emergency braking conditions. Normally, when a passenger car is driving on the road, 0.7 is taken as the maximum adhesion coefficient, so it is necessary to apply an inertial acceleration of 0.7g to the passenger car frame.

The simulation analysis limited the degrees of freedom of four support points. Limit the degrees of freedom of the X, Y, and Z axes of the left and right suspension of the front axle, and limit the degrees of freedom of the Y and Z axes of the left and right suspension of the rear axle. Under emergency braking conditions, the deformation and stress cloud maps of the electric bus body skeleton before optimization are shown in Figure 9, and the deformation and stress cloud maps of the optimized body skeleton are shown in Figure 10.

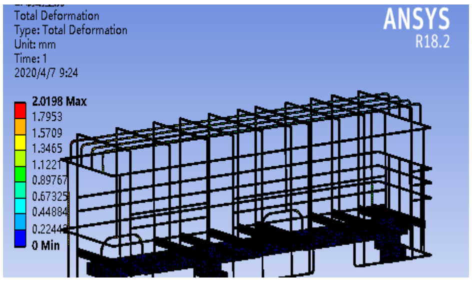


(a) Deformation cloud map

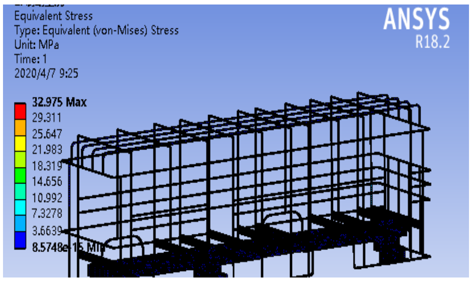


(b) Stress nephogram

Fig. 9. Deformation and stress cloud maps of the bus body skeleton before optimization under emergency braking conditions.



(a) Deformation cloud map



(b) Stress nephogram

Fig. 10. Deformation and stress cloud maps of the bus body skeleton after optimization under emergency braking conditions.

From Figures 9 and 10, it can be seen that when the bus is under braking conditions, the front and rear ends of the bus body will deform forward due to longitudinal acceleration. Therefore, the left front of the body frame and the left front top of the frame will undergo significant deformation, with a maximum value of 138.53mm. Compared with the horizontal bending condition, the deformation is larger, and the optimized maximum deformation is 2.019mm, which is much smaller than before optimization. The maximum stress of the bus body skeleton is 6.689 MPa, and the optimized stress is 32.975 MPa, which is lower than the yield strength value of material Q235 and within the safe range and meets the strength requirements. According to the degree of deformation of the body skeleton, the optimized electric bus meets the stiffness requirements under this working condition.

From the comparison of the deformation and stress cloud maps of the entire vehicle structure before and after optimization, it can be seen that the deformation resistance performance of the optimized bus body skeleton has been improved to varying degrees compared to the original body skeleton, and meets the stiffness and strength requirements of the material. The specific data is shown in Table 2.

Table 2. Comparison table of deformation of bus body frame before and after optimization（mm）.

Working condition (full load)	Deformation of the bus body skeleton before optimization	Deformation of the bus body skeleton after optimization
Horizontal bending conditions	86.929	1.173
Torsional operating conditions	17.676	0.350
Emergency left turn conditions	22.181	0.314
Braking conditions	138.530	2.010

MODAL ANALYSIS OF BUS BODY FRAMEWORK

Modal is the unique natural vibration of a component, with its fixed frequency. Modal analysis can identify the main modal features of components within a constant frequency range and predict the true vibration responses of different external or internal vibration sources within that frequency range.

The vibration of a car is caused by road excitation, engine vibration, and collisions. The modal analysis of the bus body skeleton, like other stress analyses, is to avoid resonance caused by fixed frequency co frequency with system components. Modal analysis was conducted on the vehicle body skeleton before and after optimization under no-load conditions, and the modal analysis data is shown in Table 3.

Table 3. Natural frequency data table before and after body frame optimization

Order	Natural frequency before body frame optimization	Natural frequency after optimizing the body frame
1	0	0
2	9.97×10^{-5}	0
3	4.20×10^{-4}	0
4	7.33×10^{-4}	0
5	1.32×10^{-3}	0
6	1.45×10^{-3}	1.84×10^{-5}
7	1.74	1.626
8	2.163	2.035
9	3.412	3.194
10	3.831	3.547

When the vehicle is running, the excitation is mainly from the road surface, the engine, the vehicle transmission system, the vehicle steering system and other factors. At idle speed, the excitation of the bus is mainly from the vibration of the engine, so the free mode analysis is mainly compared with the engine idle speed frequency. The modal analysis of the electric bus takes the six cylinder four stroke engine as the analysis object, and its speed at idle speed is 700r/min, The calculation of idle frequency f is 35Hz. The maximum natural frequency of the bus frame before optimization is 3.83Hz, while the maximum natural frequency of the bus frame after optimization is 3.547Hz, which is different from the engine idle frequency of 35Hz. Therefore, the optimized bus body frame effectively avoids the possibility of resonance with the engine.

Conclusion

Based on the finite element simulation analysis method, the structure of the electric bus body skeleton is simulated and analyzed. An optimization plan is obtained through data analysis,A model of the electric bus body skeleton is constructed using SolidWorks software, and four typical motion conditions of the electric bus are selected. ANSYS Workbench software is used to simulate and analyze the four different working conditions. Static mechanical performance analysis and modal analysis are performed on the optimized electric bus body skeleton, This verifies the stiffness and strength of the optimized bus body skeleton, as well as the reliability of the optimized electric bus body skeleton, and confirms the correctness of the optimization method.

REFERENCES

Wei, Q.T., “Design and lightweight research of load-bearing body for light electric buses,” *Liaocheng University*, (2017).

Wang, D. F., Niu, Y. Y., “Lightweight technology path analysis for new energy source electric bus,” *Singapore:World Scientific Publ Co Pte Ltd,* (2016).

Fu, C., Bai, Y., Lin, C., et al. “Design optimization of a newly developed aluminum-steel multi-material electric bus body structure,” *Structural and Multidisciplinary Optimization*, Vol.60,No.5,pp. 2177-2187 (2019).

Wang, D.F., Mao, A.H., Niu, Y.Y., etc “Multi objective optimization design for lightweight body frame of pure electric buses based on topology optimization,” *Chinese Journal of Highways*, Vol.30,No.2,pp. 136-143 (2017).

F, Y.F., J, D.F., Q, W.W., “Topological optimization design of micro electric vehicle body structure under multiple working conditions,” *Machine Mechanical Design*, Vol.32,No.2,pp.81-84 (2010).

Chen， T., He， F., and Jiang, X. S., “Lightweight design of the roof frame of all aluminum pure electric buses,” *Agricultural Equipment and Vehicle Engineering*, Vol.60,No.10,pp.52-55+108 (2022).

Fan, D., Yang, X. J., and Bian, X. H., “Optimization of cross-sectional dimensions of electric bus bodies considering collision safety,” *Science and Technology and Engineering*, Vol.22,No.15,pp.6331-6339 (2022).

Feng, G. S., Li, L., and Zhou, K., “Reliability evaluation of the body framework of articulated electric passenger cars with air suspension,”. *Journal of Shijiazhuang Tiedao University (Natural Science Edition)*, Vol.33,No.2,pp.52-56 (2020).

Han, G., “Structural Design and Strength Analysis of Pure Electric Bus Bodies,” *Heilongjiang Science.,*Vol.13,No.5,pp.1-3+6 (2022).

Huang, Y. H., and Wang, J. W., “Application analysis of full load bearing technology on pure electric bus,” *Science and Technology Vision*, No.11,pp.89-90 (2018).

Liu, X. C., “Optimization of body frame structure of a pure electric bus based on sensitivity analysis,” *BAIC Motor*, No.4,pp.16-18+21 (2017).

Peng, Y. N., “Bai Wenlong. Design and optimization of pure electric bus body skeleton,” *Journal of Shandong Jiaotong University*, Vol.27,No.3,pp.7-12 (2019).

Wang, Y. M., Gu, T. Y., and Dai, Q, Y., “Optimization design of pure electric bus body structure based on improved response surface methodology,” *Mechanical Design*, Vol.36,No.S1,pp.97-103 (2019).

Wang, Y. K., Wang, Y. M., and Huang, M. Y., “Lightweight design of bus structures based on ANSYS and sensitivity analysis,” *Highway Transportation Technology*, Vol.35,No.8,pp.144-151 (2018).

Yin, J. C., Liu, Z. J., and Qin, H., “A stiffness chain mathematical model for forward conceptual design of electric bus bodies” *Journal of Automotive Safety and Energy Conservation*, Vol.9,No.3,pp.325-332 (2018).

Zhu, M., “Research on the Structural Design of Fully Loaded Electric Bus Bodies,” *Science and Technology Innovation and Application*, No.28,pp.84-85 (2020).

基於 ANSYS 某電動客車車身結構優化模擬分析

李永剛　　謝建新
青島港灣職業技術學院智慧製造學院

摘 要

電動客車車身結構主要是由小斷面矩形鋼管焊接而成，承載能力強, 在結構安全性、結構穩定性、乘坐舒適性等方面優勢明顯, 目前普遍應用於電動客車。本文是以某電動客車車身骨架為研究物件，針對車身結構設計的特點, 運用 SolidWorks 和 ANSYS Workbench 等軟體，建立參數化的客車車身結構模型。以電動客車車身的可靠性為研究目標，在滿足結構總體強度和剛度的前提下，對電動客車車身結構進行局部優化, 並對優化後的車身結構進行靜態力學性能分析、模態分析, 通過優化前後資料對比分析，發現優化後的電動客車車身骨架滿足剛度、強度的要求，說明車身骨架的優化較為成功。