Finite Element Analysis and Topology Optimization Design of Seat Bracket for New Energy Bus

Enguang Zhang \textsuperscript{1,}\textsuperscript{a} and Li Wang \textsuperscript{1,}\textsuperscript{b},
\textsuperscript{1}Zhuhai College of Jilin University, Zhuhai, 519041, China.
\textsuperscript{a}teamcenter@163.com \textsuperscript{b}wangli00019@163.com

Abstract. This paper used NX8.0 to carry out the finite element analysis and topology optimization design of the electric bus seat bracket and the mechanical properties before and after optimization are compared. The results show that when the volume is reduced by 8.9%, the maximum deformation is reduced by 47.8%, the maximum stress is reduced by 66%, the average deformation is reduced by 50%, and the average stress is reduced by 54%. In this process, light weight is achieved, and the mechanical properties of the product are improved significantly. All processes are completed under the same software, which ensures the data consistency and editability. As the case involves new energy vehicles, parametric design, finite element analysis and topology optimization design, it is introduced into the CAE course as a project case under the background of "new engineering" and achieves good results.

1. Introduction
According to incomplete statistics, in 2019, the sales volume of new energy buses with the length of more than 6 meters from major domestic manufacturers reached 76278. New energy buses are becoming more and more popular in the field of public transportation in China. Take the seat bracket of new energy bus as an example, 20 to 50 seat brackets are used on each 6 to 10 meters long new energy bus. As the key parts of automobile interior, the optimal layout of materials is particularly important in cost control, energy saving and reliability design. Therefore, the strength and stiffness must be better guaranteed to avoid large deformation and fatigue fracture. The seat bracket is usually made of Q235. Because of the simple structure of parts, the material layout is not the best in the structural design based on experience. In this paper, NX8.0 is used to carry out the finite element analysis and topology optimization design for the seat bracket of a new energy bus, which improves its mechanical properties significantly. At present, many researchers at home and abroad use variable density method to carry out finite element analysis and topology optimization design \cite{1-5}, but there is few application in the structural optimization design of automotive interior parts. The design data and analysis data of almost all of the above research are not unified, and the design efficiency is not high.

2. On nx8.0 and topology optimization
Nx8.0 is a powerful three-dimensional digital software that can provide CAD / CAE / CAM solutions. It can be used in the whole life cycle development process of products. From product design, assembly, drawing to advanced simulation, dynamic simulation and manufacturing can be completed under the software, effectively ensuring the unity and editing of data. The advanced simulation module of NX can realize the finite element analysis and topology optimization analysis of the structure. The default solver for NX8.0 is NX Nastran8.1. The steps and processes of finite element analysis and topology optimization using NX 8.0 are as follows: Firstly, carry out finite element analysis of the parts to get
the analysis results. Secondly, according to the analysis results of the working condition, create a topology optimization analysis scheme. Thirdly, define the design area of topology optimization. The density of each element in the design area is a variable scalar, and its value varies between 0-1. Fourthly, define machining and geometric constraints. This step ensures that the parts can be machined or cast. Using this step, you can also freeze some areas that you do not want to optimize. Fifthly, create the design response of topology optimization. The design response is the displacement, strain energy and reaction force of the previous finite element analysis results. Sixthly, the design goal is to maximize or minimize the functions during the optimization, such as the maximum of stiffness, the minimum of volume, etc. Seventhly, design constraints are defined to limit the value of the design response, such as the absolute value of the maximum displacement or the percentage of the maximum volume. Eighthly, define the smoothness parameter, which refers to the smoothness of the generated reference STL file.

3. The finite element analysis of the original new energy bus seat bracket
The physical drawing and parametric design model of a new energy bus seat bracket are shown in Figure 1. The finite element analysis is carried out in the advanced simulation module of NX 8.0. The mesh type used is 4-node 3D tetrahedral mesh. The load is 2000N for vertical downward pressure and 300N for horizontal tangential force, and the bottom is fixed. The finite element model consists of 693689 elements and 153817 nodes. The original new energy bus seat bracket is 475281 mm³. The maximum deformation is 3.599mm. The average displacement is 1.316mm. The maximum Von Mises is 188.576 Mpa. And the average stress is 14.972 Mpa, as shown in Figure 2.

![Figure 1 Seat bracket of new energy bus](image)
4. Topology optimization design of seat bracket of new energy bus

As shown in Figure 3, the design area of topology optimization is indicated in green, and the density of the area unit changes between 0-1. The design response is the maximum displacement, the sum of the strain energy in the design area and the total volume. The design objective is to minimize the strain energy. The design constraint is 55% of the current total volume and the maximum displacement is not more than 3.2mm. And then the topology optimization analysis is carried out. As shown in Figure 4, after 30 topology optimization cycles, the density scalar diagram of the area unit is designed about 4 hours later. The density ranges from 0 to 1. The topology optimization results can be seen more clearly after filtering out the cells less than 0.8.

Figure 3 Define green area as design area
5. Finite element analysis of optimized seat bracket of new energy bus

As shown in Figure 5, the parameterized design of the new energy bus seat bracket is carried out by using NX8.0, and the advanced simulation module is used for the finite element analysis of the optimized bracket. In the middle part of the mesh type, 8-node 3D skim mesh is used, and the rest is 4-node 3D tetrahedral mesh. The finite element model includes 286457 elements and 153552 nodes. The optimized volume is 433098 mm$^3$. The maximum deformation is 1.879mm. The average displacement is 0.658mm. The maximum Von Mises is 63.97 Mpa, and the average stress is 6.866 Mpa, as shown in Figure 6.

Figure 5 Optimized three-dimensional model of new energy bus seat bracket
Fig. 6 Deformation and von Mises of new bracket

6. Conclusion
The mechanical properties of the optimized bracket are improved as shown in Table 1.

|          | deformation (mm) | stress (Mpa) | Mean deformation (mm) | Mean stress (Mpa) | volume (mm³) |
|----------|------------------|--------------|-----------------------|-------------------|--------------|
| Original | 3.599            | 188.58       | 1.316                 | 14.972            | 475281       |
| New      | 1.879            | 63.97        | 0.658                 | 6.866             | 433098       |
| comparison | ↓ 47.8%         | ↓ 66%        | ↓ 50%                 | ↓ 54%             | ↓ 8.9        |

This paper takes a new energy bus seat bracket as the research object, uses NX8.0 to carry on the finite element analysis and the topology optimization design, and carries on the finite element analysis to the optimized bracket. Through comparison and verification, its mechanical properties have been significantly improved at the same time of lightweight. All processes have been completed in NX8.0, which ensures the unity of data, improves the design efficiency, and has a certain pertinence and universality. Because this case involves the knowledge content of new energy vehicles, parametric design, finite element analysis and topology optimization design, it has been introduced into CAE courses as a project case under the background of "new engineering", and achieved good results.

Acknowledgement
The authors acknowledge the financial supports from key scientific research platforms and research projects of Guangdong universities in 2018 project “Design and reliability analysis of key parts in automobile interior”(2018KTSCX310).

References
[1] Shu Huang Sun. Optimum topology design for the stationary platen of a plastic injection machine. 2004 J. Comput Ind 55 147-158
[2] Zhu Jun-xia, Wu Jia-meng. Topology Optimization of Transverse Web Frame in Cargo Hold of VLCC Based on Variable Density Method 2019 J.Ship & Boat 06123-132

[3] Sun Bin, Wu Chang-feng, Yu Guo-fei, Shen Si-zhe and Ye Song-kui. Optimization Design of Chassis-frame Topology for a Pure Electric Bus 2019 J. Bus & Coach Technology and Research 41 27-30

[4] Zhou Si, Wu Xiao-dong and Pan Song. Lightweight topology optimization design of aluminum alloy for front steering knuckle. 2019 J. Automobile Applied Technology 19 149-151

[5] Zhao Dong-wei, Wang Zheng-chao and Yin Hua-ixi. Topology Optimization Design of City Bus Chassis Frame Based on Variable Density Method 2019 J. Bus & Coach Technology and Research 41 1-3.