Finite element analysis and Topology optimization design for automobile door-handle

Enguang Zhang [1, a] and Xin Zhang[1,2,b],

[1]Zhuhai College of Jilin University, Zhuhai, 519041, China. [2]School of Mechanical Science and Engineering, Jilin University, Changchun, 130022, China.

[a] teamcenter@163.com [b]zhang862679893@163.com

Abstract. We use NX10.0 to perform finite element analysis, fatigue life analysis, and topology optimization design on the door handle base. The mechanical properties before and after optimization were compared. With no increase in mass, the maximum deformation is reduced by 13%, the maximum stress is reduced by 31%, the fatigue life is increased by 8%, the average deformation is reduced by 8%, and the average stress is reduced by 5%. All processes are completed under the same software, ensuring the consistency and modification of data.

1. Introduction

According to incomplete statistics, the sales volume of domestic passenger vehicles in 2017 has reached more than 20 million vehicles, and cars are becoming more and more popular in China. Taking the door handles as an example, the door handle assembly is the most touch-sensitive products besides the steering wheel. People have more beautiful and higher requirements for aesthetics and strength. Therefore, better protection must be achieved in terms of strength and rigidity, and problems such as breakage, large deformation and fatigue fracture should be avoided. The material of the inner door handle is usually PC+ABS plating. The most common method is direct injection molding because the parts themselves are not complicated. As a result, past experience is often used in structural design, and the material layout is not optimal. In this study, the finite element analysis, fatigue life analysis, and topological optimization of the base of a car door handle were performed under NX 10.0, and its mechanical properties were significantly improved. At present, many researchers at home and abroad utilize ANSYS software or Hyperworks software to use the variable density method for finite element analysis and topology optimization design [1-4]. However, almost no structural optimization is applied to automotive interior parts, especially door handles.

2. About NX10.0 and topology optimization

NX10.0 is a powerful 3D digitizing software integrating CAD/CAE/CAM. Its functions cover the entire life cycle of the product, including design, assembly, drafting, advanced simulation, motion simulation and processing modules. The advanced simulation module gradually absorbed and integrated many functions and advantages of the world's excellent finite element software over the years [1]. The advanced simulation module of NX can realize the finite element analysis and topology optimization analysis of the structure. The advanced simulation module is fully integrated in NX and shares many basic functions with many NX application modules. The advanced simulation module is seamlessly connected to many mainstream solvers. Such solvers include NX Nastran, MSC Nastran,

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

ANSYS, and ABAQUS. Without having to export the solver file or import the results, users can view the results of the solution directly in NX Advanced Simulation. The default solver for NX10.0 is NX Nastran [5].

Through the help file of NX10.0, we can understand the steps and processes of its topology optimization. 1.Define the design variables in the topology optimization of NX10.0. The design variables are the density of each element in the design area. The value can be modified through optimization. Software modifies cell density during optimization to achieve optimization goals. 2. Define the design area for topology optimization, and the density of each cell in the design area is a design variable. Optimize cell density to achieve design goals. 3.Define machining and geometry limits. This step ensures that parts can be machined or injection molded. 4.Create a design goal for topology optimization that is a function that can be maximized or minimized during optimization. This function depends on the results of the finite element analysis. The result of the analysis (displacement, strain energy, etc.) used to determine the objective function and constraints is called the design response. 5. Define design constraints that limit the value of the design response or a linear combination of design responses, such as volume. 6. Define fairing parameters, indicating the smoothness of the generated reference STL file.

3. Finite Element and Fatigue Life Analysis of Original Door Handle

The finite element analysis of the original door handle base was performed in the NX10.0 advanced simulation module using a 4-node 3D tetrahedral mesh. The finite element model contains 580511 elements and 134711 nodes. The volume of the original door handle base is 67110mm^3, the maximum deformation is 0.0611mm, the average displacement is 0.025mm, the maximum Von Mises is 2.255Mpa, and the average stress is 0.083Mpa,as shown in Figure 1. The minimum fatigue life cycle is 878,000 cycles, as shown in Figure 2.

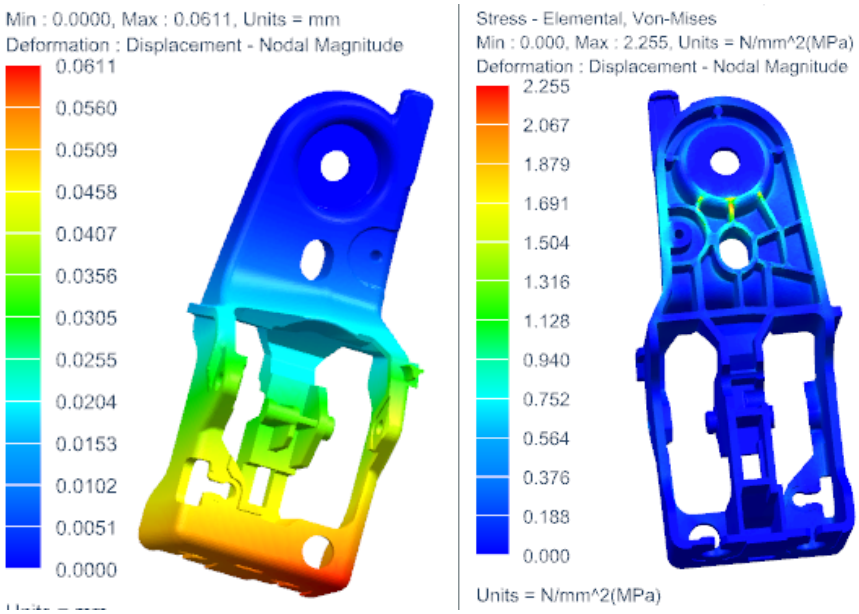


Figure 1. Deformation of the original door handle base and Von Mises

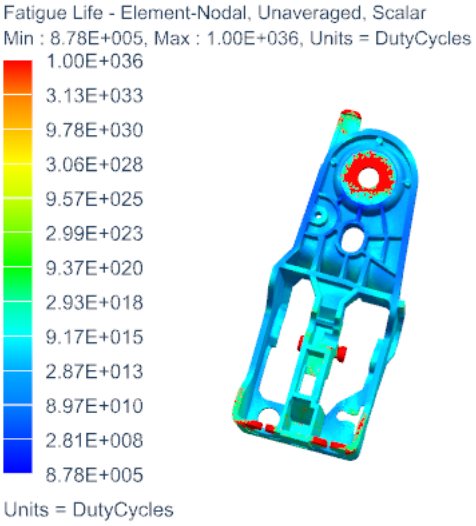


Figure 2. Fatigue life cycle of the pedestal

4. Topological optimization design of door handle base

First, as shown in Figure 3, the green area is defined as the design area, and the cells in this area are variable density. Then define the maximum displacement as the design response and the volume as the design constraint. Perform topology optimization design. Figure 4 shows the density scalar map of the design area unit. The density ranges from 0 to 1. Filtering out units less than 0.9 provides a clearer view of topology optimization results.

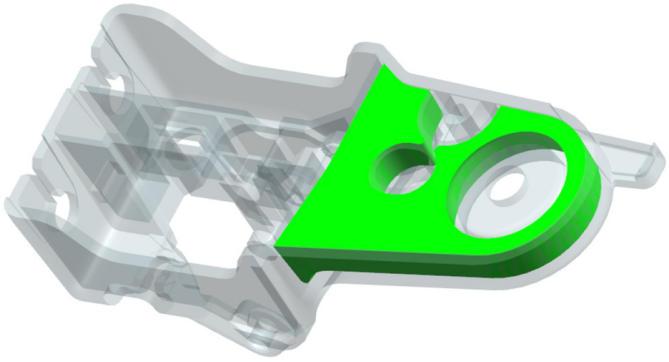
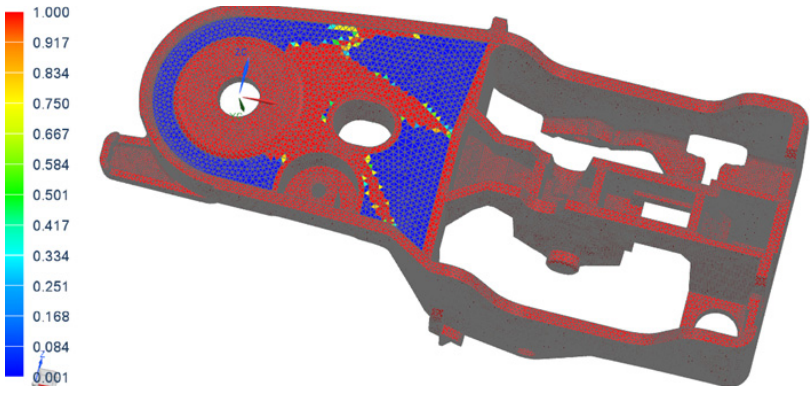


Figure 3. Define the green area as the design area



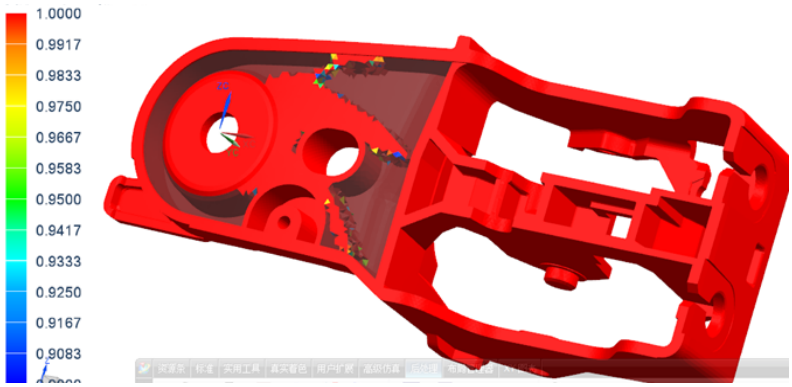


Figure 4. Design the density scalar map of the area unit

5. Finite element and fatigue life analysis of door handle base

The NX10.0 advanced simulation module performs a finite element analysis on the optimized door handle base, using a 4-node 3D tetrahedral mesh type. The finite element model contains 554,048 cells and 128,920 nodes. The volume of the door handle base is 67093mm^3, the maximum deformation is 0.0533mm, the average displacement is 0.023mm, the maximum Von Mises is 1.566Mpa, and the average stress is 0.079Mpa,as shown in Figure 5.The minimum fatigue life cycle is 954000 cycles.

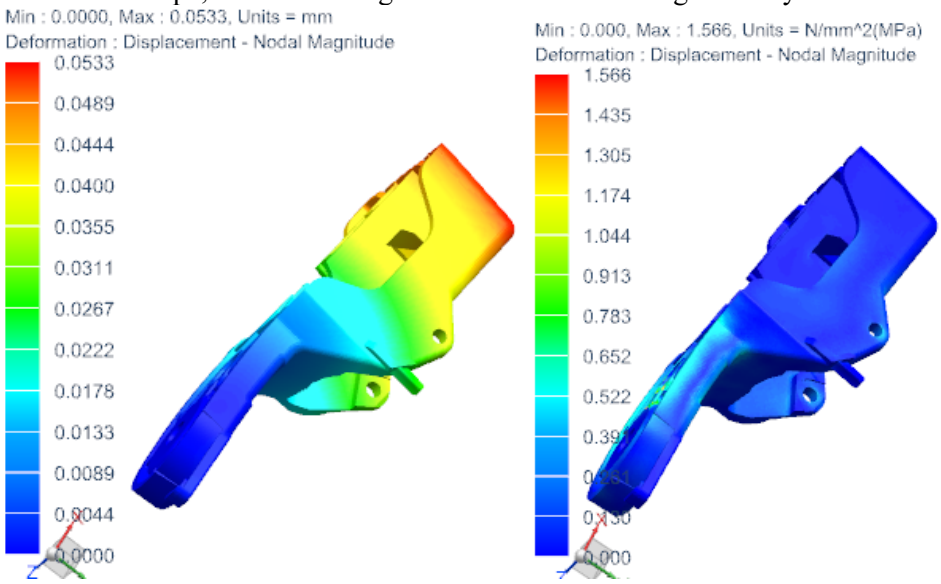


Figure 5. New door handle base deformation and Von Mises

6. Conclusion

The optimized mechanical performance of the door handle base has the following improvements, as shown in Table 1.

Table 1. Comparison of mechanical properties before and after optimization

	Deformation（mm）	Stress（Mpa）	Average deformation（mm）	Average stress（Mpa）	Fatigue life	Volume
Original	0.0611	2.255	0.025	0.083	878000	67110
New base	0.0533	1.566	0.023	0.079	954000	67093
Compared	↓13%	↓31%	↓8%	↓5%	↑8%	Almost

						unchanged

This article takes a certain door handle base as the research object, uses NX10.0 to carry on its finite element analysis and the topological optimization design, and carries on the finite element analysis and the comparison verification to the optimized base. All processes are completed in NX10.0, ensuring data integrity. The selection of content has certain engineering application value and can help some related companies to have less design cost and production cost, and has certain pertinence and universality.

Acknowledgement
This paper is supported by the project "Design and reliability analysis of key parts in automobile interior"（2017XJCQ006）and the "Three Levels" teacher talent construction project funding of Zhuhai College of Jilin University.

References
[1] Shu Huang Sun . Optimum topology design for the stationary platen of a plastic injection machine . [J].Computers in Industry,55(2004): 147-158 .
[2] M. Cavazzuti, A. Baldini, E. Bertocchi, D. Costi, E. Torricelli, P. Moruzzi. High performance automotive chassis design: a topology optimization based approach. Struct. [J].Multidiscip. Optim, 44 (1) (2011), pp. 45-56
[3] X. Huang, Y.M. Xie, G. Lu. Topology optimization of energy-absorbing structures. [J]. Int. J. Crashworthiness, 12 (6) (2007), pp. 663-675
[4] E. Norberg, S. Lovgren. Topology Optimization of Vehicle Body Structure for Improved Ride & Handling. [D].
[5] Siemens PLM Software, NX10.0 Help Library, 2015.