Finite element analysis of passenger car transverse stabilizer bar based on NX8.0

Enguang Zhang1*, Shuai Hui1
School of mechanical engineering, Zhuhai College of Jilin University, Zhuhai, Guangdong, 519041, China
*Corresponding author’s e-mail: teamcenter@163.com

Abstract. The parametric design and finite element analysis of the lateral stabilizer bar of the new energy bus were carried out by using NX8.0, and the structural strength and stiffness of the stabilizer bar were qualitatively analyzed. The simplified methods for different boundary conditions, solving types and finite element models are compared. All the processes are completed under the same software, which ensures the uniformity and editability of the data. Finally, the subject is introduced into the student's computer-aided engineering analysis training program, so that the students' comprehensive ability is improved.

1. Introduction
The function of the stabilizer bar is to prevent the car from excessive lateral roll when the car turning, and to keep the body as balanced as possible. The aim is to reduce the degree of lateral roll of the car and improve ride comfort [1]. Vehicles that do not have a stabilizer bar have a much higher probability of rollover, especially in the case of large mass, high center of mass, and variable road conditions. The transverse stabilizer bar is an auxiliary elastic element that is connected at both ends by a tie rod and a suspension (shock absorber or lower control arm); the shaft is fixed to the part of sub-frame or sub-frame by rubber bushings and clamps [2]. The digital model of the transverse stabilizer bar structure and the actual product structure are shown in Figure 1. At present, many scholars have studied it [3].

2. About NX8.0 and its finite element analysis
NX8.0 is a digital software integrated with CAD/CAE/CAM function of Siemens. Its function covers almost the whole life cycle of the product. Finite element analysis, optimization analysis, fatigue analysis, and Response Simulation of the structure can be achieved using NX's Advanced Simulation...
module. The Advanced Simulation module seamlessly interfaces with NX's modeling and assembly modules. The solvers available for the Advanced Simulation module include NX Nastran, MSC Nastran, ANSYS, and ABAQUS for linear and nonlinear analysis. The default solver for NX8.0 is NX Nastran 8.1.

The process of nx8.0 finite element analysis is as follows: 1. Establish the 3D digital model of parts or assembly body, and carry out parametric design when necessary, so as to facilitate the later optimization. 2. Enter the Advanced Simulation module to create an idealized component, a finite element model file (FEM), and a solution plan file (SIM). 3. Select the solver and solution type, such as NX NASTRAN solver by default. Select SESTATIC 101 as the linear static solution and choose NLSTATIC 106 as the solution for nonlinear statics. 4. If necessary, ideally operate the components, such as rounding, small holes and facets that have little effect on the mechanical properties of the components. 5. In the finite element model file (FEM), the model is divided into cells, such as 1D, 2D, 3D cells, and materials are added. If necessary, the connection grid and contact grid are created. 6. Add boundary conditions, such as load, constraint, and final solution, in the solution plan file (SIM). 7. After the solution is successful, enter the post-processing and read the finite element analysis data.

3. Finite element analysis of the transverse stabilizer bar
The main structure of the stabilizer bar can be regarded as a U-shaped rod with a diameter of 50 mm, so it can be theoretically simplified into a 1D Element for analysis, or it can be analyzed with a 3D Element. The load is a 35 mm forced displacement load at both ends of the U-shaped bar along the positive and negative directions of the Z-axis. The constraint is a user-defined constraint at the rubber bushing that only releases the Y-axis rotational freedom here.

Common materials for the transverse stabilizer are 60Si2CrA, 60Si2MnA and 55Cr3. The parameters selected in this case are as follows: material as 55cr3, Young's modulus as 206gpa, Poisson's ratio as 0.29, density as 7850kg / m3, ultimate tensile strength as 1225Mpa, yield limit as 1080Mpa [4]. The rubber bushings are mostly made of solid rubber and are common elastomers. The material has significant nonlinear characteristics and is capable of withstanding extreme strains [5].

3.1. 1D Element simplified analysis
As shown in Figure 2, the centerline of the stabilizer bar is created. Create a 1D mesh in the finite element model. The Element section is shown in Figure 2. Create a user-defined constraint at the end of the tie rod to release only the y-axis rotational degrees of freedom here. A 1D connection point to point is created at the end of the pull rod and the center of the rubber bushing. The ReB2 rigid rod is used to connect, and the rubber bushing is ignored. At this time, there are 3594 elements and 3595 nodes in the finite element model.

Figure 2. Creating a finite element model of a 1D mesh
The simulation scheme has 21,546 degrees of freedom and the solution takes about 5 seconds. The maximum deformation and maximum stress are shown in Figure 3. The maximum deformation is
35.62mm and the maximum Von Mises is 913.9Mpa. The 1D Element has fewer finite element model elements, fewer nodes, and less freedom. Therefore, the solution is faster. For a simple and constant section model, a 1D Element can be used to create a finite element model for qualitative analysis. However, this method has a large deviation.

3.2. 3D Tetrahedral Element single linear static finite element analysis
As shown in Figure 4, a 4-node 3D Tetrahedral grid of size 3 mm is used for the model division Elements. Ignoring the effect of the rubber bushing, the finite element model contains 302,199 Elements and 65,265 nodes with 195,165 degrees of freedom. Create 1D Connection-Point to Edge at the center of the rubber bushing, create user-defined constraint at the connection center, release only the Y-axis rotational freedom here, and add the load as before. Select the SESTATIC 101 linear static solution, as shown in Figure 5. The maximum deformation is 35.68mm, and the maximum Von Mises is 803.17Mpa. Since the rubber bushing is neglected, and the actual rubber bushing has significant nonlinear characteristics, there is a certain deviation between the solution and the actual one.

Figure 3. Maximum deformation and maximum stress

Figure 4. Add meshing and boundary conditions
3.3. 3D Tetrahedral Element multi-body nonlinear static finite element analysis

As shown in Figure 6, cells are divided for the stabilizer bar, rubber bushing and pull rod, and mesh matching condition is added at the assembly contact surface by using 4-node 3D tetrahedral grid with size of 2mm. The finite element model contains 2,770,498 Elements and 561,301 nodes with 195,165 degrees of freedom. Create a 1D Connection-Point to Face at the center of the tie rod on the rubber bushing, create a user-defined constraint in the connection center, and release only the Y-axis rotational freedom here. The load is added as before, as shown in Figure 7. Select NLSTATIC 106 as the solution type of nonlinear statics. As shown in Figure 8, the maximum deformation is 22.45mm and the maximum Von Mises is 678.78Mpa.
4. Conclusion
In this paper, the lateral stabilizer bar of passenger car is taken as the research object, and finite element analysis is carried out by using NX8.0. The different simplified methods, mesh types, working conditions and solution types are compared and verified. This topic is used as a teaching case for undergraduate CAD/CAE in teaching, so that students can understand the process of the entire finite element analysis and the role of key options. The selection of content has certain engineering application value, and it has certain pertinence and universality.

Acknowledgments
The authors would like to express their sincere thanks to the editor and the reviewers for their constructive suggestions, which are very important for the improvement of the paper quality. They also acknowledge the financial supports from teaching quality engineering project (Project No. ZLGC20180816, 2018010) of Zuhua College of Jilin University of China.

References
[1] Yan Z.B., Chen H.B., He H.M. (2019), Pre-tightening fracture of 60si2MnA automobile stabilizing bars [J]. Journal of Shijiazhuang University of Applied Technology, 31(04): pp.13-16.
[2] Shan H.Y., Yao B. (2011), Studying of rubber bushings in anti-roll bar system of passenger car [J]. Machinery Design & Manufacture, (11): pp.95-97.
[3] Wang G.L., Huang X.H., Liu S.H., Li Q.W. (2013), Fatigue simulation and structure optimization of suspension stabilizer link bracket [J]. Chinese Journal of Engineering Design, 20(01): pp.18-21.
[4] Ge Q. (2015), Finite Element Analysis And Design Research of Vehicle Stabilizer Bar [D]. Hebei University of Technology.
[5] Song J., Xing R.F. (2005), Finite Element Analysis on Stabilizer-bar with Rubber Bush [J]. Automotive Engineering, (05): pp.89-91.