Acta Technica 62 (2017), No. 4A, 547–554

Fatigue life analysis of automotive stabilizer bar based on fea.

Xuewen Liu[1], Shengwei Zhang[1], Hao Chen[1,4], Yali Yang[1,3], Kun Gao[1], Yipeng Wang[1], Wen Qi[2]

Abstract. Due to the lack of scientific basis in fatigue design, the traditional fatigue reliability design of automotive stabilizer bar is rather conservative, which results in longer development period, cumbersome components and low fatigue reliability, etc. A combination method of FEA calculation and fatigue life analysis of stabilizer bar was proposed in this paper. Based on real CAD model, the FEA model was established. During model establishment of stabilizer bar, two problems were countered, bushing contract area constraints definition and stress extraction accuracy. For bushing constraints, two represent methods were used, in which one is zero width RBE2 with linear solver, and the other one is RBE3 with nonlinear solver for the bushing contact area constraint. Stiffness was calculated for each method. Simulation was conducted in NX NASTRAN to determine bushing constraint and stress extraction. For bushing constraints, stiffness comparison was also conducted between RBE2 and RBE3 with real testing stiffness curve data. Simulation results showed that the solid Von Mises stress of RBE3 constrains with bush element was decreased by 4% than that of RBE2 constrains. And the simulation stiffness with RBE3 constraint was basically the same as the test curve. It was indicated that RBE3 constraint was more effective than RBE2. For stress extraction, the surface stress of skin elements is relatively less than solid stress, and the plate top stress and the plate bottom stress is roughly equal. Thus, the stress of skin elements was used to describe the surface stress of stabilizer bar and can also maintain the accuracy. Then, fatigue analysis was conducted in ANSYS NcodeDesignlife based on skin stress and strain under RBE3 constraint. It was shown that the fatigue life of the stabilizer bar can be up to 530000 times. This fatigue life completely meets the minimum fatigue requirements of the new product development. It was illustrated that the combined FEA calculation and fatigue life method can provides a reference for the fatigue design of the stabilizer bar

Key words. Stabilizer bar, FEA, bushing constraint, fatigue design.

[1]Workshop 1 - College of Automotive Engineering, Shanghai University of Engineering Science, Shanghai, China

[2]Workshop 2 - China Spring Corporation Limited, Shanghai, China

[3]Workshop 3 - Department of Mechanical Engineering, University of Ulsan, Ulsan, Korea

[4]Corresponding author: Hao Chen; e-mail: pschenhao@sues.edu.cn

1. Introduction

Stabilizer bar is an important security component of automobile suspension system. The function of stabilizer bar is to keep the stability and balance of the body, reduce vehicle's lateral tilt, and improve automobile smoothness when faced with a sharp turn[1]. Horizontal stabilizer bar is actually a transverse torsion bar spring, and can be regarded as a special kind of elastic element in function. Stabilizer bar doesn't work when both sides of suspension have the same deformation. When turning, beating different on both sides of suspension caused rolling of the car body, and distort of the Stabilizer bar. At the same time, bar's elasticity will stop the wheels up, keep the body balance thus to keep the lateral stability[2]. Stabilizer bar is typically affected by cyclic fatigue loading during running. So, it is quite essential to study its fatigue characteristics, and predict its fatigue life, to achieve optimal design performance. Thus, FEA and fatigue life design analysis was proposed in this paper.

2. Finite element analysis of the stabilizer bar

2.1. FEA model establishment

CAD model was provided by a company and can be imported into NX to construct the FEM with NX data interface. The whole stabilizer bar assembly consists of a solid rod, bushing and clamp.

The material of stabilizer bar was 55CrVA, a highstrengthspringsteel. The elastic modulus was 2.09*$10^5 MPa$ the Poisson ratio was 0.3.

During simulation, bushing and clamps were simplified by using an bush element with bushing stiffness[1]. The structure of stabilizer bar is very simple, a curved cylinder with simplified represent bushing. In order to obtain a better calculation accuracy, all solid elements models were built based on the second-ordertetrahedron element (Tetra10),which defined by 10 nodes and each node of Tetra10 has 3 degrees of freedom (displacement of node X, Y and Z direction). This Tetra10 element has the ability of plasticity, expansion, stress stiffening, large deformation and displacement, which was more suitable to simulate surface boundary. The finite element model with Tetra10 in NX was shown in Fig.1.

(1) Displacement Loads

When the stabilizerbar is subjected to torsion, the simplified force diagram of the stabilizer bar is shown in Fig.2.In the fatigue test, the stabilizer bar is not directly applied to the hole center of the end head, but applied by drop-link center. Linear load of the stabilizer bar with avertical force or displacement in the opposite directions should be applied to the drop-link center. The drop-link center node and the surface node of the end head hole are connected with the RBE2 element, a rigid connectionelement, which will transfer the force and torque to the end head of the stabilizer bar. The calculated displacement load in NX NASTRAN is ??40mm, which determined by switching the value of wheel beat to the displacement value of stabilizer bar.

Fig. 1. Tetra10finite element model

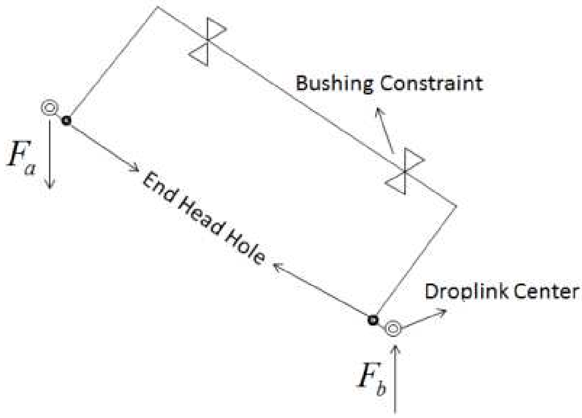

Fig. 2. The simplified forcediagramof the stabilizer bar

(2) Constraint

Two different constraint ways were built to compare the difference of Von Mises. Comparison of the stiffness between FEA and test was conducted to confirm which method is more suitable for the simulation of bushing connection.

A. Zero width RBE2: In creating FEM, rigid element was used to represent the bushing. Independent node was arranged at the center of the bushing where loads are applied. The independent nodes should be attached to the edge of the bushing hole edges, and constrained in degrees of freedom 123. Using of an RBE2 element on bushing contact area would make this area perfectly rigid, which produces unrealistic analysis results. The FEM of zero width RBE2 constraint was shown in Fig.3.

B.RBE3 modeling of bushing contact area: The bushing of the stabilizer bar is fixed on the frame through the clamping hoop, which plays the role of supporting and locating the stabilizer bar. At the same time, the bushing must have enough rigidity to limit the movement of the stabilizer bar in each direction[5].But if the bushing is established in finite element model, material nonlinearity and contact nonlinearity have to be introduced, which will result in bad effect for convergence. And the material parameterof the rubber was not easy to obtain, since the bushing

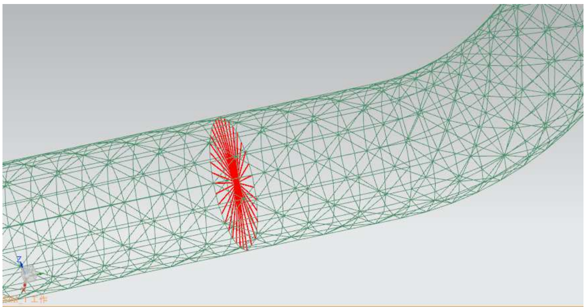

Fig. 3. The FEM of zero width RBE2 constrain

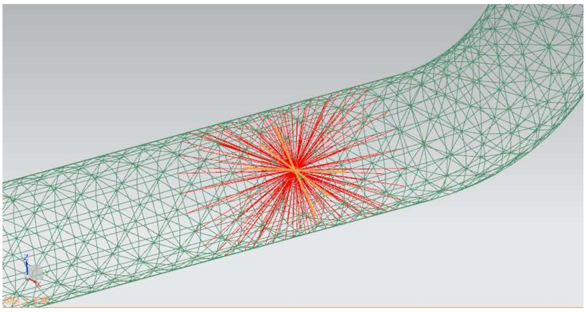

Fig. 4. RBE3 constraint of bushing contact area with bush stiffness

elastic center is frequently used to attach 1-dimensional bush elementsto constrain the part[6], some method must be used to connect this point to the node of the contact area of the bar. In the case of NASTRAN, one solution would be using an RBE3 element, with the elasticcenter as the dependent node constrained in all 6 degrees of freedom. The nodes on theouter metal would be the independent nodes, constrained in degrees of freedom 123 only. The bushing stiffness would be obtained from the stiffness test of the bushing. Stiffness value of the bushing test was shown in Table1, and constraints model of FEM was shown in Fig.4.

Table1. Static stiffness characteristics for each bushing in each direction

STAB BAR	001
Radial Stiffness [KN/mm]	8
Axial Stiffness [KN/mm]	0.4
Torsional Stiffness [Nm/]	1

There are many ways to process stresses in FEA. For 3-D solid elements, stresses should be recovered on the surface. In order to recover surface stresses and use these results in fatigue analysis, very thin skin elements, will need to be added to the model. These elements should be extremely thin so as not to add any additional structure to the model. Local FEM of the skin elements were shown in Fig.5.

Fig. 5. skin elements with 0.001mm

2.2. Analysis and solve

NX NASTRAN was used to solve these two different cases. The first case, zero width RBE2, used the linear solver, and the second case, RBE3 constraint of bushing contact area, used the nonlinear solver. At the same time, in order to improve the convergence character and effectiveness of calculation, it is necessary to open large displacement and applied load by dividing the whole displacement into 10 steps.

The Von Mises stress was calculated by NX NASTRAN. and make a post processing in FEMAP showed in Fig.6 to Fig.9.

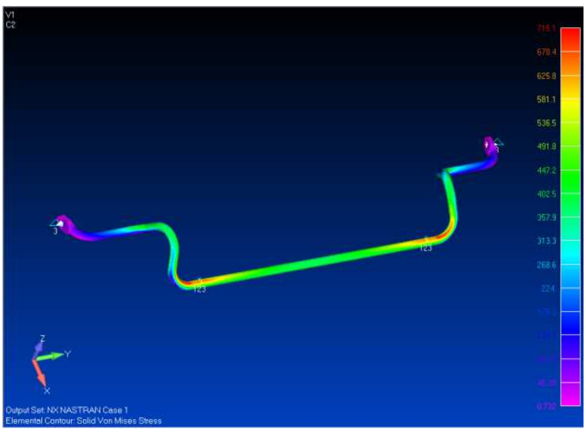

Fig. 6. Solid stress distribution of RBE2 constraint

The results showed that the maximum solid Von Mises stress of RBE2 constraints and RBE3 was 715.1MPa and 686.1MPa, respectively. The maximum plate top stress and bottom stress of RBE3 constraints were almost the same, 679.6MPa, and 679.9MPa, respectively.

By the analysis of the simulation results, it can be seen that the solid Von Mises stress of RBE3 constrains with bush element was decreased by four percent than that of RBE2 constrains. This is due to generate a perfectly rigid area by RBE2

Fig. 7. Solid stressdistribution of RBE3 constraint

Fig. 8. Plate top stressdistributionof RBE3

element, which cause the stress to increase. The Von Mises stress of skin elements was relatively less than solid stress. This was because there was no average for nodes stress. The plate top stress and the plate bottom stress were almost equal, which indicated that the skin elements had no influence on FEA of stabilizer bar. So using stress of skin elements to describe the surface stress of stabilizer bar will be more accurate.

2.3. Stiffness analysis of stabilizer bar

Stiffness and fatigue life are the two most important parameters considered in the design of a stabilizer bar. By comparing the stiffness of the finite element analysis with the experimental measurement, it can be verified that whether the finite element model is more accurate or not, and can provide reference basis for the reasonable finite element analysis. Stiffness test used the horizontal fatigue test machine was

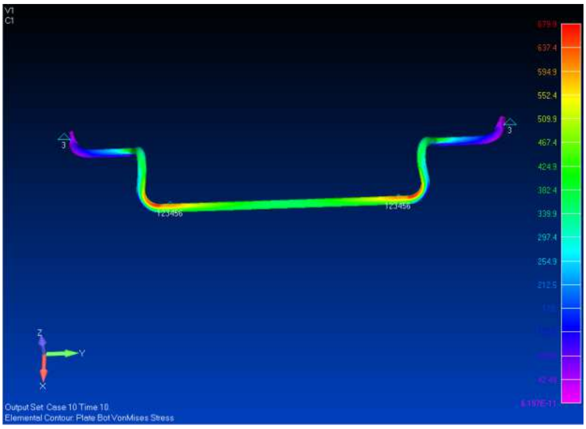

Fig. 9. Plate bottom stressdistributionof RBE3

shown in Fig.10. The comparison results of the test stiffness and the simulation stiffness was shown in Fig.11.

Fig. 10. Horizontal Fatigue MachineFig

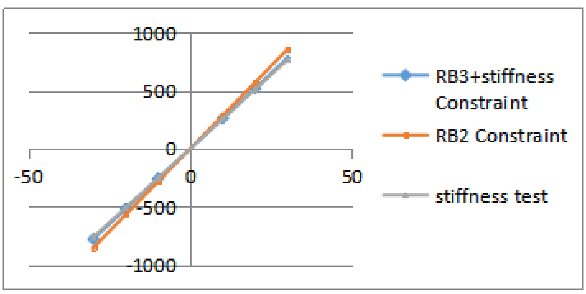

Fig. 11. Test stiffness and simulation stiffness curve

According to the stiffness curve, it can be found that the test stiffness is basically the same as the simulation stiffness of RBE3 constrains with bush stiffness, but the stiffness value of RBE2 constrains is too large, which is not suitable for the finite element analysis of stabilizer bar.

3. Conclusion

The equivalent stress value of RBE3 constrains with bushing stiffness is smaller 4% than that of RBE2 constrains, and it is more suitable for the FEA of stabilizer bar.

The stress of the skin elements is more accurate than that of the solid element for the surface stress distribution of the stabilizer bar.

The proposed method that combined the FEA results of RBE3 constrains with bushing stiffness and the NcodeDesignlife to calculate the fatigue life of the stabilizer bar is fully satisfied with the engineering requirements and provides a reference for the fatigue design of the stabilizer bar.

References

[1] Huang. Kang, Yang. Rongde: *Fatigue analysis on the automobile transverse stability rod based on ANSYS.*Automotive Engineering *27* (2005), No. 5, 59–594.

[2] NI. Yun, CHEN. Wei: *Strain Fatigue Analysis on Front Transverse Stabilizer Bar Based on nCode.*Hot Working Technology *2* (2014),82–58.

[3] P. H. Cronje. RS: *Improving off-road vehicle handling using an active antirollbar.*Journal of Terramechanics *47* (2010), 179–189.

[4] M. Cerit, E. Nart: *Investigation into effect of rubber bushing on stress distribution andfatigue behaviour of anti-roll bar.* Engineering Failure Analysis *17* (2010),1019–1027.

[5] Fan. X, Cai. H,Lin. S,Gummadi. LNB, Cao. K: *Non-linear analysis of tunable compression bushing for stabilizer bars.*SAE technical paper series (2004),01–1548.

[6] Gummadi. LNB,Cai. H,Lin . S, Fan. X, Cao. K: *Bushing characteristics of stabilizer BARS.* SAE technical paper series (2003) 01–0239.

[7] Uygun. EM: *Structural Analysis of stabilizer bar under torsion. MSc thesis.* İTÜ Institute of Science and Technology (2007).

[8] Tasatar. C: *Structural analysis of stabilizer bar. MSc thesis.*SAU Institute of Science and Technology (2006).

Received November 16, 2016