International Journal of u- and e- Service, Science and Technology
Vol.6, No.5 (2013), pp.39-48
http://dx.doi.org/10.14257/ijunesst.2013.6.5.04

Optimization Design of Rear Stabilizer Bar for WF-1 Model Based on CAE

Jinlong Wang[1], Aiping Zhang [2] and Liang Zhu[3]
[1]*College of Civil Engineering and Architecture of Weifang University,*
Weifang 261061, China
[2]*College of Civil Engineering and Architecture, Southwest Petroleum University,*
Chengdu 610500, China
[3]*Pan Asia Technical Automotive Center Co., LTD of Shanghai GM,*
Liuzhou 545001, China
wfxyjinlong@163.com, zhangaiping620@163.com,
a511752282@163.com

Abstract

Considering WF-1 model automobile as the research object, establish the geometry entity model of rear stabilizer bar through CATIA software, and use ANSYS software to carry out the mechanic analysis and fatigue lifetime. Then the measures for improving its configuration are put forward as well as some reasonable designs and structural improvements about rear stabilizer bar. Then the rear stabilizer bar can achieve the goals including lightening, miniaturization and durability by optimization design.

Keywords: *CAE Technology; Rear Stabilizer Bar; Fatigue Analysis; Optimization Design*

1. Introduction

As a auxiliary component of suspension, Lateral stabilizer bar play the role in preventing excessive lateral tilt at turn and other conditions. When bodywork only bear symmetrical load and vertical movement, it will happen equal suspension deformation on both sides, then stabilizer bar will not have any forces and not work. When the body heel, longitudinal parts on both sides of the stabilizer bar occurs deflection in different directions, then the lateral stabilizer bar is in torsion. Elasticity restored moment of the shaft produce resistance for the torsion deformation, thus reducing the body tilt for the inertia force when turning and then making it possible to be back to the center position [1].Automobile rear stabilizer bar is made by spring steel, which hangs on the ends of vehicle by U shape .The sleeve on both ends is fixed on the axle by bolts [2].

Many scholars at home and abroad have do some research about auto stabilizer bar. Ye Zhang *etc.*, [3] adopts the electrical measuring method to determine the actual stiffness of stabilizer bar. Shengliang Dai *etc.*, [4] put the stabilizer bar on the monoclinic arm rear suspension of a light bus and analyze that horizontal stabilizer bar's sports coordination and effectiveness in the suspension system. Yizheng Luo [5] takes the chemical composition analysis, metallographic examination and hardness testing on the stabilizer bar which is out of work.

As an effective numerical computation method, finite element analysis plays a major role in the optimization design of stabilizer bar. FEM about the stabilizer bar can quickly determine the weak position of the stabilizer bar, also judge the reliability and fatigue life of

ISSN: 2005-4246 IJUNESST

mechanical products [6-8].At home and abroad, it has occur a large number of reports. These cases are mainly concentrated on the analysis of the auto front stabilizer bar structure, while the research about the rear stabilizer bar is less. Therefore, this paper, taking a supplier's rear stabilizer bar as an example, establishes finite element model, analyses the stress distribution of the rear stabilizer bar and predicts the fatigue life of the rear stabilizer bar. Through the analysis of the data, this paper presents rear stabilizer bar structure optimization scheme. Effect is proved to be good by optimization design of lightweight, high strength, durability with WF-1 rear stabilizer bar.

2. Stress Analysis on Rear Stabilizer Bar

2.1. Geometric Model and Finite Element Model

WF-1 rear stabilizer bar is one U shape structure stem, which in the middle is U shape hollow tube. The rear stabilizer bar is fastened on the rear axle through the M12 the rear axle bolt and it's outside diameter Φ is 20 mm, and thickness of 3.5 mm; Sleeve outside diameter Φ of 28 mm, its thickness is 5.75 mm, height of 48 mm. Accuracy of stabilizer bar model is directly related to simulation results, and 3-D CAD software provides the possibility for the accurate establishment of the mathematical model[9-10].In CATIA, according to the relevant size parameters, this paper establish 3-D entity model of the rear stabilizer bar. As is shown in Figure 1.

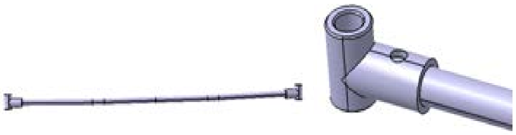
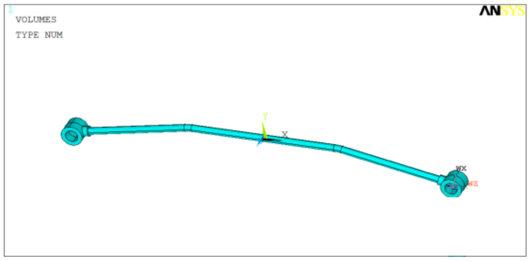


Figure 1. Rear Stabilizer Bar Model in CATIA Software

With the function improvement of the application software platform, interface configuration between FEM analysis software ANSYS and the 3-D modeling software CATIA develop quickly. By using data interface configuration between CATIA software and ANSYS software, it can realize CATIA 3-D the precise import of mathematical model. According to the specific geometry parameters, this paper establish 3-D entity model of stabilizer bar in CATIA, and then imports it into ANSYS. The model of rear stabilizer bar that established in the ANSYS is shown in Figure 2.



(a)

(b)

Figure 2. Rear Stabilizer Bar Model in ANSYS Software

2.2. Material Parameters

Rear stabilizer bar uses the materials of 40Cr and the sleeve uses material of 45# steel, the material parameters are shown in Table 1.

Table 1. Material Parameters

Name	MOE/GPa	PR	density/Kg□m^{-3}
Rear stabilizer bar	201	0.3	7850
Sleeve	206	0.27	7820

2.3. Element Selection and Meshing of FEM

SOLID92 Element in the ANSYS is defined with 10 points. Every node has three degrees of freedom: Node displacement of X, Y, and Z direction. Element has some characters inculding the plasticity, creep, expansion, toughened-stress, large-deformation and large strain capacity, which is fully afford the needs of rear stabilizer bar analysis.

2.4. Imposing Constraints, Loading, Solving and Checking the Results

Finite element model of stabilizer bar imposed constraints is shown as in Figure 3, the stress cloud picture is shown as in Figure 4[11].

Figure 3. Finite Element Model of after the Imposed Constraints

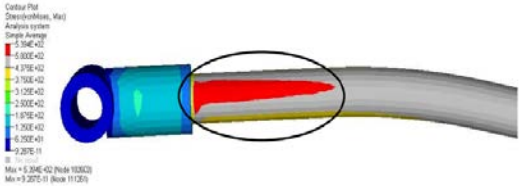


Figure 4. Effective Stress Nephogram of the Rear Stabilizer Bar before Improvement

As is shown in Figure 4, we can see that the stress mainly concentrates at the welds between sleeve and stabilizer bar ends. The maximum stress is 539.4MPa. Because of stress concentration, fracture mainly happens at this position. The fracture from the market feedback also proved that the accuracy of the analysis results. Fault fracture situation is shown in Figure 5.

Figure 5. The Unserviceable Component

3. Fatigue Analysis of Stabilizer Bar

3.1. Fatigue Analysis Overview

3.1.1. Fatigue Analysis Definition

Fatigue refers to one of static fracture damage phenomena that structures bear repeat load under the action of ultimate strength. One steel bar that can bear 300KN tension force, for instance, will be destroyed after bearing 200KN cycle loading by 1,000,000 times.

3.1.2. Main Factors about Fatigue Failure

There are many factors that lead to fatigue damage including load cycles, stress amplitude, mean stress and local stress concentration.

3.1.3. Fatigue Calculation Function about ANSYS

1) Determine the fatigue life consumption coefficient (fatigue usage factors) about solid element or shell element model by stress results post-processing. The stress of line element model must be entered manually in the fatigue calculation.

2) Determine a certain number of events and the load of these events that can be pick up on the pre-arrange location, and then put away the stress.

3) Define the stress concentration coefficient of every location and the ratio for each event.

3.2. Basic Steps about Fatigue Calculation

General postprocessor (POST1) of ANSYS can carry out the fatigue calculation after the stress calculation. Main steps are as follows:

3.2.1. General Postprocessor (POST1) Entrance and Database Recovery

Read the database file (Jobname.DB) into memory. If the fatigue calculation is the continuation of the ongoing ANSYS calculation process, the file (Jobname.DB) has been already in the memory. Node stress results file (Jobname.RST) must have existed and read it into the memory.

3.2.2. Establishments about the Size, Material Properties and Calculation Position about Fatigue Calculation

(1)Definitions of the locations, events, and the maximum number of loads

By default, the fatigue calculation at most includes five node locations, ten events and three loads in each event.

(2) Definition of the material properties about fatigue

In order to calculate all kinds of consumption coefficients and simplified elastic-plastic effect, we must define the material properties about fatigue. The material properties considered in the fatigue calculation are as follows:

1) S-N curve: curve: one curve about stress amplitude [(Smax - Smin)/2] and the curve of fatigue cycles. If necessary, we should adjust S-N curve considering the average stress intensity effect. If not, stress amplitude will adopt descending order for all possible combinations of stress, and not calculate cost coefficients.

2) Sm-t curve: one curve about stress intensity value changing with the temperature. Whether or not the check stress ranges into the plastic, If you want to consider it, then you must define the curve.

3) Elastic-plastic material parameter (M) and stress intensification index (N). Input M and N only considering the simplified elastic-plastic code.

(3) Definition of the stress position and stress concentration coefficient.

3.2.3. Storing Stress, Setting Events Cycles and the Scaling Factor

(1) Storing stress

For fatigue calculation, the program must know the stress about every event in different positions and loads, also the cycle times of each event.

1) Artificial storing stress

We can store stress and temperature artificially, not input from the Jobname.RST result file. In this case, POST1 fatigue module plays the role as the fatigue calculator not as the processor. The line units’ stress (such as beam elements) must be entered manually, because the fatigue module can't read line units stress data from the results file.

2) Node stress from the Jobname.RST file

The method stores the node stress vector that includes six components directly in the database. Then we can use the FS to revise the stress components.

3) Cross-section stress

Calculate and storage the linear stress of section path endpoints. The linear stress calculation is usually carried on the surface of the most short line segment. Therefore, we need take one point on every surface to describe PPATH path. In this step we can obtain the stress from the calculation results, so we must use the FSSECT before use the SET. The stress components that are stored by FSSECT can be revised by FS.

(2) Event repetitions and scaling factor definition

Define the event repetitions, and also specify stress scale factor of all loads that constitute the event.

3.2.4. Fatigue Calculation Activation

Base on the location, stress, events, and all assigned material parameters, the fatigue calculation can be performed in the specified location. We can define the positions by nodes themselves as well as e the location numbers.

3.2.5. Checking by the Calculation Results

Results about fatigue calculation can be seen from the output window. If you export the results in the OUTPUT file (such as Jobname.OUT), you can open the file to check the results.

Fatigue is calculated by using the stress results of post-processing from ANSYS and determined by element life consumption coefficient [1].After completing the simulation calculation, stress concentration parts are determined by the stress nephogram. Compared stress concentration areas from figure 4 of effective stress nephogram with damaged parts from Figure 5 actual of failure parts, the welding between sleeve and the ends of rear stabilizer bar is the position which is prone to occur fatigue damage. So this paper chooses nodes nearby this place as a fatigue analysis object.

The fatigue life calculation of the stabilizer bar is imposing rotational displacement at one end, which makes the stabilizer bar rotating around the center axis condition within ±12 °.Meanwhile the fatigue life values is at least 200,000 times. According to the fatigue calculation, this paper get the results which is shown in Figure 6.The analysis result shows that fatigue life value is more than 350,000 times, which basically meets the requirements. But requirements for some special vehicle parts fatigue property is very high, meanwhile the structure fatigue life value is close to the required value. The rear stabilizer bar exist so many potential risks. Therefore, its structure optimization is needed.

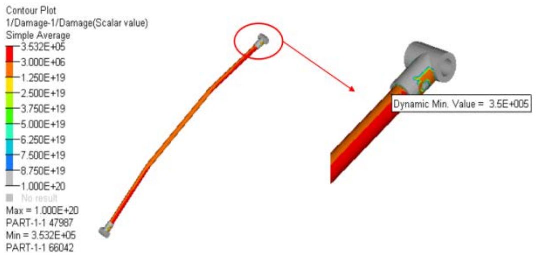


Figure 6. Fatigue Analysis Results of the Rear Stabilizer Bar before Improvement

4. Optimization of the Stabilizer Bar Structure

Optimization Design Theory

The basic principle of optimization problem is to establish optimization model and use various kinds optimization methods, in order to calculate objective function extremum and obtain optimum program through iterative computations based on the design requirements are fulfilled.

The mathematical model of optimization problem is expressed as [12]

$$
\begin{aligned}
&\min F(X) = F(x_1, x_2, \dots\dots, x_n) \\
&\text{s.t } g_i(X) = g_i(x_1, x_2, \dots\dots, x_n) \leq 0 \quad (i = 1,2, \dots\dots, n) \\
&\quad h_j(X) = h_j(x_1, x_2, \dots\dots, x_n) \leq 0 \quad (j = 1,2, \dots\dots, m) \\
&\quad X = [x_1, x_2, \dots\dots, x_n]^T
\end{aligned}
$$

Here, $F(X)$ is objective function, it is the function of design variables that is used to evaluate good and bad of design proposal, optimization problem refers to calculating objective function extremum; $g_i(X)$, $h_j(X)$is constraint condition, it is the function of design variables and it is limiting conditions of the design variables range and state variables range;

X is design vectors, it consists of design variables, it is design parameters of optimization in design, each design vector is a design proposal.

The method for solving optimization problem is generally numeric iteration. Finite element method and optimized method are the main mathematical tools.

Based on the above analysis, this paper takes the optimization of the structure for the weak links in order to achieve the optimization goals of rear stabilizer bar about lightweight, high strength, durability. This paper redesigns the stabilizer bar sleeve structure. The end of sleeve stabilizer bar is perpendicular to the surface of the stabilizer bar. This is stress concentration area which makes the force transfer non-uniformly. Considering the demands of welding technology, the model sets up a 11 ° slope between its face and stabilizer bar surface. Meanwhile, optimize the intersection line that is formed by the junction between end fixed sleeve and the stabilizer bar sleeve. Then stress distribution at the end become uniform, also the method can save materials and reduce costs by stabilizer bar lighting. The improved structure of the stabilizer bar by redesigning is shown in Figure 7. The stress cloud picture is shown in Figure 8 by finite element calculation.

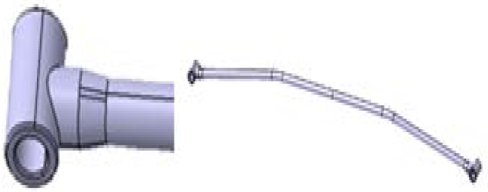


Figure 7. The Rear Stabilizer Bar Structure after Improvement

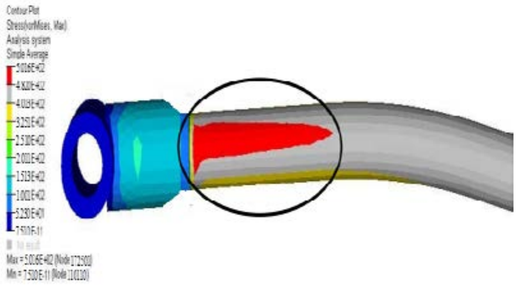


Figure 8. Effective Stress Nephogram of the Rear stabilizer Bar after Improvement

From the Figure 8, it is obtained that rear stabilizer bar's maximum effective stress is still largely focused on the welds between sleeve and the stabilizer bar ends. The maximum effective stress becomes 501.6MPa, which is less than the maximum of 539.4MPa before improvement. We can draw the conclusion that stabilizer bar can effectively avoid stress concentration after redesign.

Taking fatigue analysis for the optimized structure of rear stabilizer bar again, it obtain that the value of fatigue life is more than 1 million times, and its structural stability performance is much higher than the structure of the stabilizer bar before improved. The improved product's fatigue analysis results are shown in Figure 9.

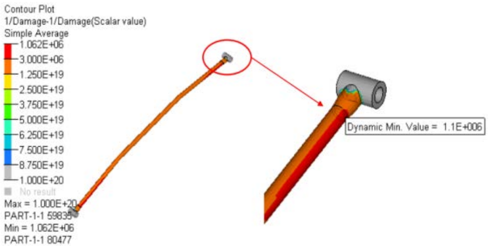


Figure 9. Fatigue Analysis Results of the Rear stabilizer Bar after Improvement

5. Conclusions

(1) By using the software of finite element analysis including CATIA and ANSYS, it can determine the loading and fatigue life of stabilizer bar quickly and accurately;

(2) Through taking optimization design for the rear stabilizer bar structure, it makes the structure of the rear stabilizer bar become reliability, and anti-fatigue performance is improved remarkably;

(3) By the optimization designs about lightweight, high strength, durability for rear stabilizer bar of WF-1, the performances improves remarkably and we also obtain the affirmation from manufacturers and users.

Acknowledgements

The work in this paper has been supported by funding from Outstanding Young and middle-aged scientists Foundation of Shandong(BS2010CL046)and from National Science Foundation of China(51004077).

References

[1] S. Guang-lie, “Modern Manufacturing Engineering”, vol. 2, **(2005)**, pp. 123-125.
[2] Y. Rong-de Huang-Kang, Journal of Machine Design, vol. 25, no. 12, **(2008)**, pp. 66-68.
[3] G. Shao-de Zhang-Ye, Journal of Experimental Mechanics, vol. 18, no. 2, **(2003)**, pp. 271-277.
[4] D. Sheng-liang, T. Xue-dong, W. Chang-wen, L.-H. Li-Lei, Journal of Hefei University of Technology, vol. 32, **(2009)**, pp. 123-138.
[5] L. Yi-Zheng, L. Ji-lin and X. Chun-jun, “Hot Working Technology”, vol. 42, no. 2, **(2013)**, pp. 213-215.
[6] S. Jia, X. Ru-fei and W. Chang-wen, “Automotive Engineering”, vol. 27, no. 5, **(2005)**, pp. 592-594.
[7] Z. Dong, H. Hu, L. Xin-tian and L. Ting, Journal of Shanghai University of Eengineering Science, vol. 22, no. 3, **(2008)**, pp. 215-217.
[8] L. Hong-yan and G. Xian-yue, Machinery Design ＆ Manufacture, vol. 9, **(2010)**, pp. 191-192.
[9] L. Fang and W. Cheng, Automobile Techonlogy, vol. 7, **(2006)**, pp. 5-8.
[10] D. Neng-gen, Z. Hong-bing, R. Xiao-feng and B. Ying, Automobile Techonlogy, vol. 2, **(2007)**, pp. 19-22.
[11] Y. Peng, “Research on the Fatigue Reliability of Automotive Stabilizer Bar”, HeFei: HeFei University of Technology, **(2010)**.
[12] S. Jing-min, “Mechanical Optimum Design”, Beijing: Mechanic Industry Press, **(2005)**.
[13] L. Hong-qi, L. Li, T.-H. Kim and X. Shao-long, “An Improved PSO-based of Harmony Search for Complicated Optimization Problems”, IJHIT, vol. 1, no. 3, **(2008)** July, pp. 91-98.
[14] S. Ram Kotra, A. Kumar, K. R. S. Sambasiva Rao and K. K. Pulicherla, “Statistical Optimization of Media Components for Enhanced Production of the Recombinant Staphylokinase Variant from Salt Inducible E.Coli GJ1158”, IJBSBT, vol. 4, no. 4, **(2012)** December, pp. 27-40.

Authors

Jinlong Wang received his doctor's degree from China University of Petroleum.He is currently an associate professor in College of Civil Engineering and Architecture,Weifang university,China.and he has published over 30 research papers in scholarly journals and international conferences.

Aiping Zhang received her bachelor's degree from Weifang University. She is currently a post-graduate student in College of Civil Engineering and Architecture, Southwest Petroleum University, China.

Liang Zhu received his bachelor's degree from Weifang University. He is a Production Engineer from Liuzhou and currently work in Pan Asia Technical Automotive Center Co., LTD of Shanghai GM, China.