Advances in Engineering Software 91 (2016) 63–68

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

A research on fatigue life of front axle beam for heavy-duty truck

Min Zhang, Xiangfei Ji, Lijun Li*

School of Transportation and Vehicle Engineering, Shandong University of Technology, No. 12, Zhangzhou Road, Zhangdian District, Zibo City, Shandong Province 255049, China

ARTICLE INFO

Article history:
Received 13 August 2015
Revised 10 October 2015
Accepted 25 October 2015
Available online 12 November 2015

Keywords:
Front beam
Strength
Fatigue life
Multi-conditions
Crack
Finite element

ABSTRACT

Front axle is one of the important parts of vehicle. The Fatigue Tool in ANSYS Workbench software is used to analyze its strength situations and fatigue life. The dangerous area of beam is determined, which provides a certain basis for structure improvement. In addition, in view of three kinds of working condition often used, a combined fatigue simulation of multiworking conditions is carried out. The influence of different conditions on fatigue life is studied. Finally, fatigue life of front axle containing crack is investigated to analyze the effect of crack parameters such as length and depth on fatigue life.

1. Introduction

Corrosion, wear and fatigue are the main causes of failure of mechanical parts. Main failure form of front beam (also known as front axle) is fatigue damage, as shown in Fig. 1. Therefore, the research on the fatigue life has important value.

With the development of computer technology and finite element numerical method, the fatigue analysis method based finite element method is adopted gradually. Topaç et al. predicted the fatigue life of a heavy vehicle steel wheel under radial loads by using finite element analysis. When *S–N* curve of the wheels material is established, they considered the surface roughness, the size, stress concentration and other factors [1]. Hui studied some driving bridge and considered the main factors influencing fatigue strength, established the *S–N* curve. At last the dangerous position and the fatigue life were obtained [2]. Lu established the fatigue life prediction model of hub bearing based on multi-operating modes under different operating modes [3]. Xinliang did a research to the fatigue damage which was mainly caused by the durability test under multiple cyclic loading condition of some hydraulic support test rig [4].

This paper studies the fatigue life of steering bridge of some heavy truck by ANSYS Workbench software. Fatigue life is calculated using mean stress correction method [5], and the simulation results are compared with the experimental results. The danger zone of beam axle is determined and the structure is optimized. These researches play a guiding role for predicting its fatigue failure and fracture prevention.

2. The establishment of finite element model

The research object of this paper is the front beam of some heavy duty truck. Its rated load is 4.5 t. The material is 50 steel [6], and its main parameters are shown in Table 1.

2.1. Meshing

In order to make the boundary conditions more suit with the actual situation, the steering knuckle and master pin are still retained. According to the overall size of the model, the grid size of front beam is set up to 8 mm. The finite element model is obtained, as shown in Fig. 2.

2.2. Load and boundary constraints

The rated load that is known as the uniformly distributed load is applied to plate spring seats, which is 22,500 N.

In order to meet the actual situation, the auxiliary supports [7] are used. Movement constraints of *X*, *Y*, *Z* directions are applied on the left support arch and movement constraints of *Y*, *Z* directions are applied on the right support arch, as shown in Fig. 3.

2.3. S–N *curve*

The material of front beam is 50 steel. Its *S–N* curve can be obtained by fatigue test.

* Corresponding author. Tel: +86 13864419622.
E-mail addresses: zmin2924@163.com (M. Zhang), jixiangfei@outlook.com (X. Ji), lilijun@sdut.edu.cn, 546271281@qq.com (L. Li).

http://dx.doi.org/10.1016/j.advengsoft.2015.10.006
0965-9978/

Fig. 1. Fatigue damage cases of axle.

Table 1
Materials data sheet of front beam.

Material properties	Specific values
Density (kg/m^3)	7860
Elastic modulus (GPa)	207
Poisson ratio	0.276
Ultimate strength (MPa)	661
Yield limit (MPa)	418

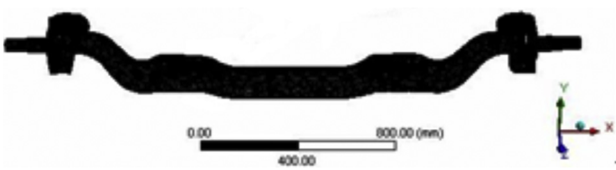

Fig. 2. The finite element model of front axle.

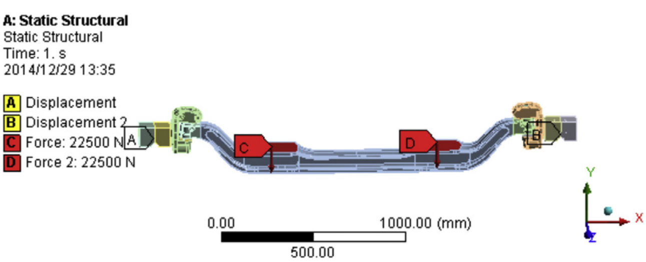

Fig. 3. Boundary conditions of front beam.

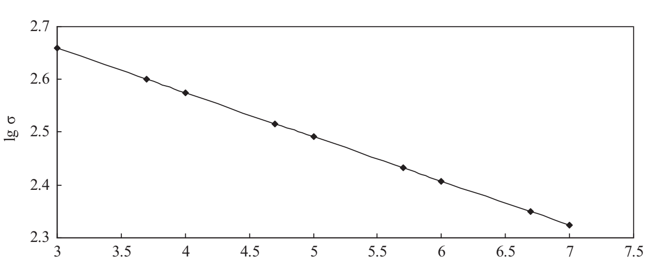

Fig. 4. *S–N* curve of 50 steel.

This article refers to the fatigue design manual. Its *S–N* curve [8] is established by the simplified method, as shown in Fig. 4.

2.4. Fatigue load

According to *Automobile front axle fatigue life test method* which is an automobile industry standard, the lower limit is 0.5 full load and the upper limit is 3.5 full load in fatigue test. Therefore, the fatigue load is calculated as shown in Fig. 5.

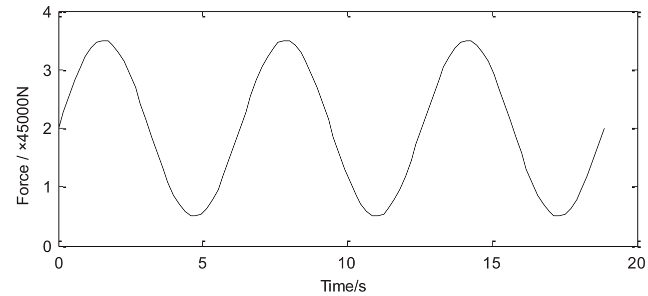

Fig. 5. The curve of fatigue load.

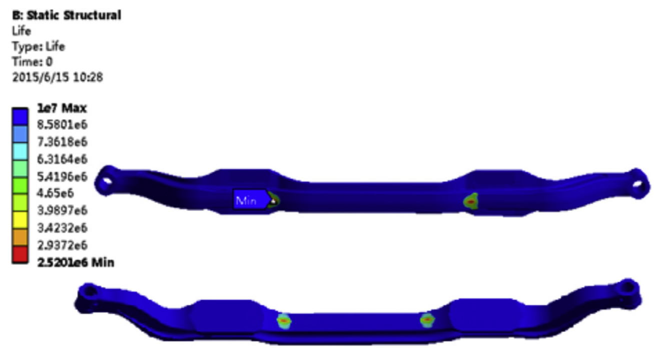

Fig. 6. Fatigue life of front beam after *Gerber* correction.

3. The results of fatigue analysis

3.1. Finite element results

Based on the first principal stress, fatigue analysis is carried out. And the *Gerber* correction method is applied according to its material. The result is as shown in Fig. 6.

The minimum life beam is under the plate spring seat. That is because this place is always under tensile stress, and tensile stress has larger effect on the fatigue life than compressive stress. In addition, this place has stress concentration.

3.2. The test results

Vertical bending fatigue test of front beam was carried out at the technical inspection center. These lives of front beam were obtained as 172,000 times, 190,000 times and 290,000 times. Failure results are as shown in Fig. 7. The position of crack is in the circle in Fig. 7, which other location is not damaged.

The finite element calculation results are compared with the test results. Fatigue trend of front beam is in line with the actual. The results with Gerber correction method are close to test results. And both the simulation results and experiment results show that the dangerous area appears in the variable cross section under plate spring seat of front beam. Fatigue damage is under tensile stress all the time, and the tensile stress has larger effects on fatigue than compressive stress. Therefore, this location needs to be optimized.

4. Improvement measure

According to the analysis, the danger zone is improved. Front beam's structure before improving is shown in Fig. 8(a). The surface in the circle is not level, and has a transition zone. Keep its cross section unchanged. The web is stretched to make sure that the lower surface has no bending, as shown in Fig. 8(b). After optimization, the neutral axis of cross section moves toward the middle. Tensile stress of the lower surface is reduced, when decreasing the gap between tension and compression stress. This can enhance the fatigue strength.