Presented at TopTECH SAE and ARAI, Feb, 2009. Due for publication in Journal of Fatigue.

Endurance testing and FE analysis of four wheeler automobile stabilizer bar

P.Senapathi, S.Shamasundar, G. Venugopala Rao and B.M.Sachin
ProSIM R&D Center,Bangalore.

Krishna Srinivas, N.Dasari, Cheryl and Karthikeyan
Tube Investments of India Ltd, Chennai.

Ponkshe Shripadraj and Basava Raju
Mahindra and Mahindra Ltd, Nasik.

ABSTRACT

Durability of automobile stabilizer bar is assessed in the present study. Finite element analysis of the stabilizer bar is performed based on the experimental observations. Fatigue analysis is performed under cyclic loading. Effect of shot peening have been taken into account. Fatigue cycles are defined for the maximum loading conditions. Custom built fatigue testing equipment is used to simulate the fatigue life of stabilizer bar. Commercial software **ABAQUS** is used for numerical simulation. Computational fatigue simulation software, **fe-safe** is used for the fatigue analysis to predict the fatigue life (number of cycles for crack initiation) and crack site location. Simulated results were compared with that of the physical test results for validation.

INTRODUCTION

Stabilizer bars are part of the cars suspension system. They are also referred to as anti-sway bar or anti-roll bars or torsion bar [1]. It is a U-shaped metal bar connects opposite (left/right) wheels together through short lever arms as shown in figure 1. It is clamped to the chassis of the vehicle with a rubber bush. The function of vehicle stabilizer bar is to reduce body roll when cornering. Due to the resulting shift in wheel load and change in the camber angle, body roll has a decisive effect on steering behavior. Stabilizer bar can be designed to counter this effect through under steer or oversteer.They thus enhance ride comfort and, to a great extent, driving safety. Stabilizers generally do not lie a single plane, but are bent, offset and cranked in sometimes remarkable fashion in order to fit around other chassis components.

STABILIZER BAR WORKING Principle - The location of the stabilizer bar have been chosen, so that the roll stiffness becomes stiffer - making it more difficult for the body to roll around the roll axis - without any influence on the vertical suspension. To accomplish this, the stabilizer bar is arranged in such a way in the axle assembly. The central section comes to rest approximately on a level with the wheel centers and transverse to the direction of travel. The rubber bushes of the stabilizer bar are connected to the body, while the arms are linked directly to the struts or the control arms as shown in figure 2. As a result, stabilizer bars have no contribution on vehicle weight support in static condition and remain unloaded during equal wheel deflection. When the body tilts due to the action of lateral centrifugal forces, the sides of the suspension react in opposite ways. The suspension springs to the outside of the curve compress and the inside springs extend [2]. As a result, the stabilizer arms are pulled in opposite directions and the central section is twisted.

When cornering, body tilt could be reduced by selecting stiffer rate of the suspension springs, but this would have a negative effect on ride comfort. Stabilizer bar therefore, considerably improves ride comfort [2]. The tuning of vertical and roll stiffness on the one hand, and stabilization of the front and rear axle on the other, depends on the individual carmaker's philosophy.

Due to high level of competition, manufacturers are trying to reduce the weight of the individual components by optimizing the design with expected life. Now a days manufacturers are going for hollow stabilizer bar instead of solid. Configuration of the stabilizer bar used for the investigation is shown in figure 3.

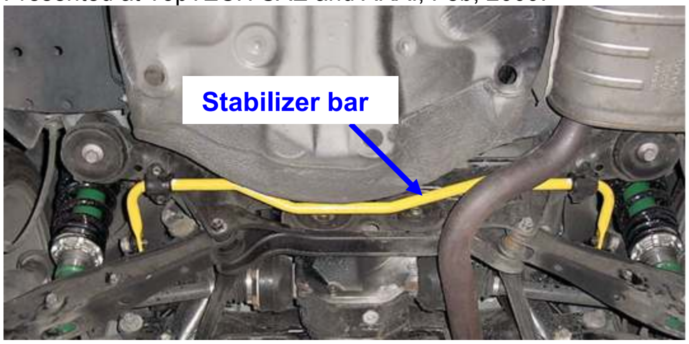


Figure 1. Typical stabilizer bar attached in the vehicle

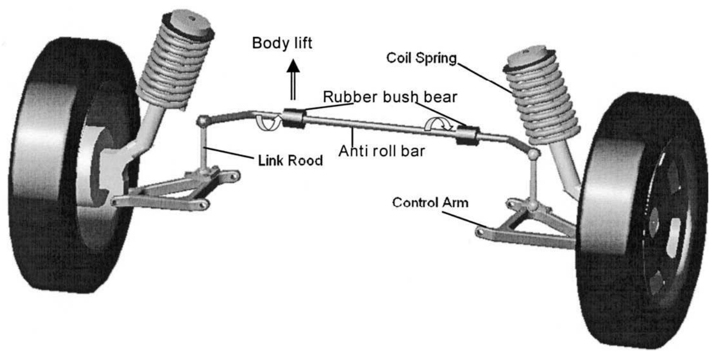


Figure 2. Stabilizer bar in action

Depends upon the road condition the loads acting on the ends of the stabilizer bar will vary in vehicle. But for our investigation maximum loading is considered on the both ends as per durability testing.

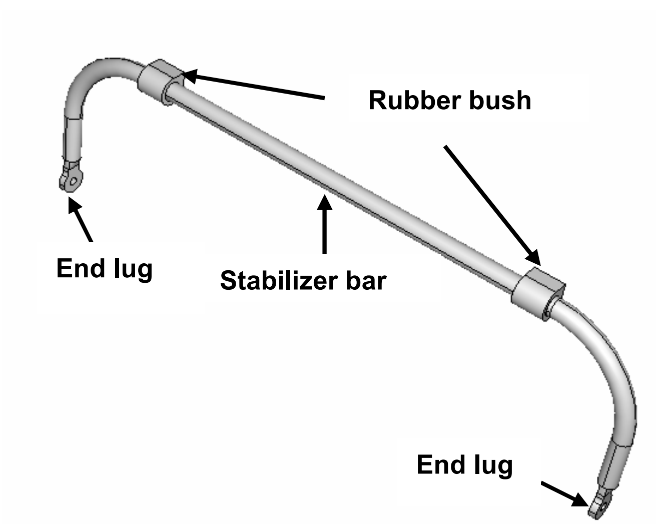


Figure 3. Configuration of stabilizer bar with rubber bush

This is a U-shaped metal bar with two end lugs. End lugs have a hole which is used to link other suspension components with stabilizer bar as shown in figure 3.

METHODOLOGY

Straight tube is taken and is bended to the required shape. Finite Element (FE) analysis is done using the bended configuration. Stress and strain from the FE analysis is taken to fatigue analysis and fatigue analysis is done for the following cases; Case 1: Without shot peening effect Case 2: Considering the shotpeening effect. Strains are measured at two different locations using strain gauges. Durability test is performed using special purpose machine under two conditions: 1. Before shot peening 2. After shot peening. Results at different stages are validated with experimental observations. Complete methodology flow chart for this study is shown in figure 4.

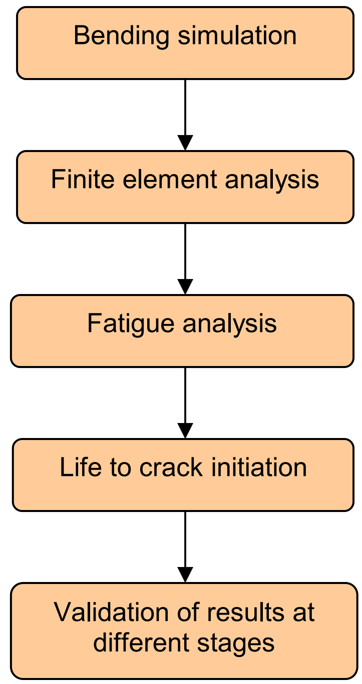


Figure 4. Methodology flow chart

ANALYSIS PROCESS

BENDING SIMULATION - Bending simulation is done using Ls-Dyna. Straight tube is taken and bend to the required shape. Bending simulation is done using shell and solid elements, solid element configuration is taken for further investigation along with thickness variation across bend section. Bended stabilizer bar consists of 74241 brick elements.

FE ANALYSIS - Bended solid element configuration is taken to ABAQUS for FE analysis. End lugs are not considered for FE analysis. Rubber bushes are inserted at the appropriate locations and are assumed to be an hyper elastic material. Elastic plastic analysis is carried out using true stress strain curves.

Loading and boundary condition - Analysis is carried out for a completely reversed cycle. Amplitude curve as shown in figure 5 is defined in the ABAQUS. 39mm displacement is applied at ends in opposite direction.

Outer surface of the rubber bush is completely constrained. Surface to surface contact is defined between stabilizer bar outer surface and rubber bush inner surface. To constraint the axial movement of the bar, set of center nodes at middle of stabilizer bar are coupled to a common reference point and axial movement is constrained then analysis is carried out for one complete cycle.

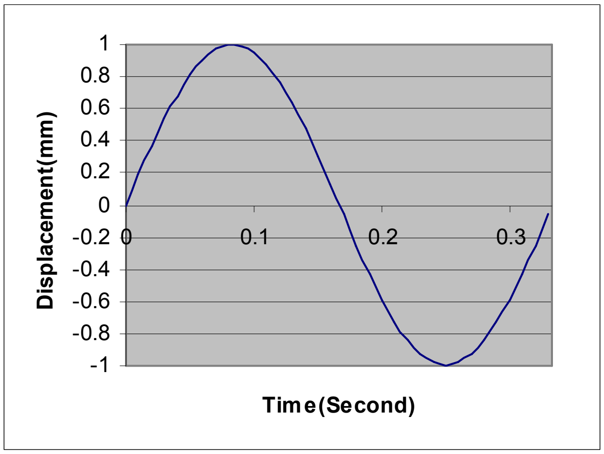


Figure 5. Completely reversed cycle

FATIGUE SIMULATION – Fatigue simulation is carried out using fe-safe. fe-safe is a highly effective tool for fatigue calculation based on the stress and strain results from FE analysis. Strain life approach is preferred for the fatigue analysis, because this is best suitable for low cycle fatigue (LCF) and it will be taken care of elastic as well as plastic strain in the model [5]. Strain life analysis uses the following governing equation

$$\Delta\gamma_{max}/2 + \Delta\epsilon_n/2 = C_1\sigma'_f/E\,(2N_f)^b + C_2\varepsilon'_f\,(2N_f)^c$$

where, $\Delta\gamma_{max}/2$ is the shear strain amplitude and $\Delta\epsilon_n/2$ is the normal strain amplitude. C_1=1.65 and C_2=1.75 are the constants derived based on the assumption that cracks are initiate on the plane of maximum shear strain. σ'_f is the fatigue strength co-efficient, E is the Young's modulus, b is the fatigue strength exponent, $2N_f$ is the number of reversals, ε'f is the fatigue ductility co-efficient and c is the fatigue ductility exponent.

Sequence of stress and strain is imported to fe-safe and appropriate surface finish conditions are taken. Fatigue properties for the specific material are created in fe-safe using uniform material law or Seeger's approximation by providing Young's modulus and ultimate tensile strength of the material. Fatigue contours are exported to output database file.

RESULTS AND DISCUSSION

FINITE ELEMENT ANALYSIS RESULTS – Elasto plastic FE analysis is carried out for one complete cycle. Stress and strains are of our interest is discussed here. Maximum stress and strain are observed at maximum displacement (39mm) as shown in figure 6 and 7.

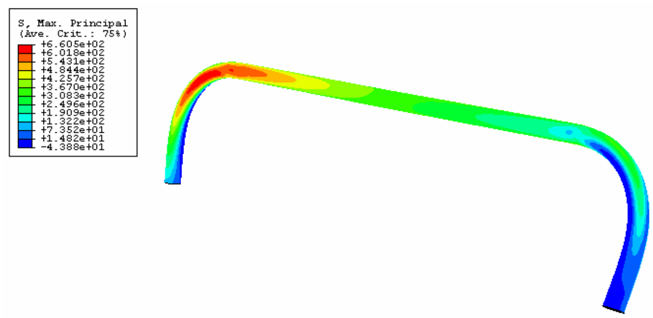


Figure 6. Maximum principal stress (MPa)

Maximum principal stress of 660 MPa is observed in the bend region as shown in figure 6. This location is near to the rubber bush mounting location.

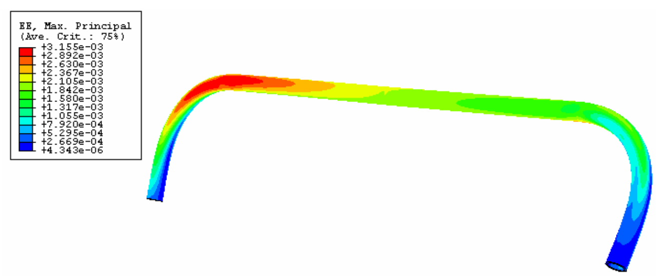
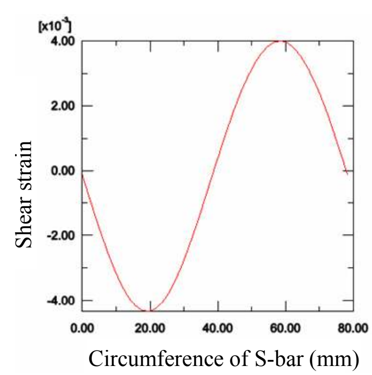


Figure 7. Maximum principal elastic strain

Maximum principal elastic strain of 0.003155 is observed near the bend region. There is no plastic strain in the stabilizer bar.

The variation of shear strain in the circumference (360°) of the stabilizer bar at the center location as shown in figure 8.



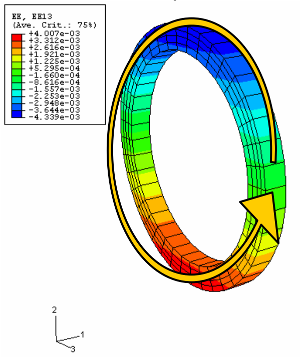


Figure 8. Shear strain variation in circumference of stabilizer bar at the center location

FATGUE SIMULATION RESULTS – Finite element stress and strain for the complete cycle is imported to the fe-safe. Stress and strain are paired in fe-safe appropriate surface finish conditions and materials are assigned, fatigue simulation is carried out. Rubber bushes are not taken for fatigue analysis. For the shot peening case, compressive residual stress of around 500MPa is assigned (measured by X-ray diffraction technique) and analysis is carried out. Fatigue simulation results are shown in the table below.

Results	Without shot peening	With shot peening
Life to crack initiation	17540 cycles	67712 cycles

Table 1. Fatigue simulation results comparison

VALIDATION OF RESULTS

FINITE ELEMENT ANALYSIS RESULTS – Two different strain gauges were fixed at two locations on the stabilizer bar and strain measured at maximum loading. Nodal strain (direct) are measured where the strain gauge is pasted, four effective nodes were taken into consideration on these surface, finally averaged them to compare with the experimental observations. To measure the shear strains, strain gauges were fixed at 45º with respect to the axis of stabilizer bar.

Location on stabilizer bar	Experimental shear strain	FE simulation shear strain
Center	1100×10^{-6}	1040×10^{-6}
Near bush	900×10^{-6}	737×10^{-6}

Table 2. Comparison of experimental and FE results: Shear Strain Measurement

FATIGUE RESULTS – Fatigue test was performed using special purpose machine, this is an Accelerated Fatigue Testing (AFT). During the fatigue testing rubber bushes were changed for 3 to 5 times. There is a scatter in fatigue life because of material and loading variation from specimen to specimen, this is an inherent characteristic of the fatigue life calculation. From the present study, the fatigue results found to be well matches with experiment results.

Fatigue life results	Experimental life (AFT), Cycles	FESAFE life prediction, Cycle
Without shot peening	11000 – 20000	17540
With shot peening	60000- 78000	67712

Table 3. Comparison of fatigue life results

CONCLUSION

In the light of the results and discussions presented above, the following conclusions made

1. The strain on the stabilizer bar is determined by two different methodologies: FEM, experimentally using strain gauges. Strain values are almost similar from both experiments and simulation.

2. Durability tests are performed using special purpose fatigue testing machine and fatigue analysis software fe-safe and results are well correlating each other.

3. Using commercially available software packages a methodology is formed for stabilizer bar fatigue analysis and that can be used for any stabilizer bar.

ACKNOWLEDGMENTS

ProSIM wish to thank Tubes investments of India limited, Chennai for provision of laboratory facilities and Mahindera and Mahindera limited, Nasik for giving permission to publish this work.

REFERENCES

1. Fan, H. Cai, S. Lin, L. N. B. Gummadi and K. Cao, Visteon corporation, "Non-Linear Analysis of Tunable Compression Bushing for Stabilizer Bars" ,SAE technical papers 2004-01-1548.

2. Rer. Nat. Lutz manke, Advanced Engineering ThyssenKrupp Federn, Hagen, "Weight saving through high stressed tubular stabilizer bars", techno forum December 2004.

3. H. Bayrakceken, S. Tasgetiren, K. Aslantas, "Fracture of an automobile anti-roll bar", Engineering Failure Analysis 13 (2006) 732–738.

4. E S Palma and E S dos Santos, "Fatigue damage analysis in an automobile stabilizer bar ", Proceedings Instn Mech Engineers Volume 216 Part D: J Automobile Engineering, 2002.

5. John draper, safe technology limited, UK "Modern metal fatigue" Volume1, 2008.