Strength evaluation of truck bumper for peak load by numerical method

A. T. Kamble[1], P. A. Harari[2], S. R. Marajakke[3], A. J. Deokar[4]

[1,2,3,4]*Assistant Professor, Department of Mechanical Engineering, SGMCOE Mahagaon, Gadhinglaj, Kolhapur, Maharashtra, India*

Abstract

In this study, steel bumper is considered and its maximum stress, strain and displacement are compared with cast iron and aluminium materials. Furthermore, nonlinear finite element simulations are performed using ABAQUS, in order to know the maximum nonlinear conditionals results and fracture life for point load at various load cases. The main aim of the study is to modify the design of the bumper and also calculate the life of the bumper for various loading conditions. The truck bumper made of steel of thickness 1.6 mm is taken as base modal, modeling is done using software CATIA and this model is exported to HYPERMESH for meshing.

Keywords – Truck, Bumper, Peak load, Numerical method.

Introduction

Bumpers are the protective structures which are used to protect the vehicle and its occupants from minor impacts. Generally bumpers are placed front end and rear end of the vehicles. But they are not capable of protecting the vehicle and its occupants at high speed impacts so it only protecting from low speed impact. The vehicle bumper is designed to prevent or reduce physical damage to the front and rear ends of passenger motor vehicles in low speed collisions. Automobile bumper are not typically designed to be structural components that will significantly contribute to vehicle occupant protection during front or rear collisions. Bumper is also protecting the cars or small vehicles under passing the truck. Generally bumpers are made of metal because easily available of metal. Metal bumpers are heavy in weight. So designers are design the bumpers with low weight to high strength ratio considering geometry and load on bumper. But this load not gives the life of component. How far it's give life prediction of bumper. The stress analysis is important in fatigue and its life prediction for bumper. To find the maximum stress at a point or critical point with maximum sustainable load which is gives the failure of bumper. This critical point is causes the fatigue failure. The magnitude of the stress is used to forecast the life of the component. Static or dynamic analysis is gives us about stress, displacement, acceleration etc. But not how long is component survive. Nonlinear static analysis is gives the how long component is survive at maximum sustainable load. Maximum displacement, true stress and true strain these parameters are given the life of component. The main aim of this study is to modify the design of the bumper and also calculate the life of the bumper for various loading condition.

Objectives

- To make the modal analysis of truck bumper for different materials and geometric shapes or changes.
- To make the nonlinear static analysis of truck bumper for different materials and geometrical shapes by considering the different loading conditions.
- To calculate the deflection of truck bumper by numerical method.

Project methodology

- CAD modeling of truck bumper is done with the help of CATIA-V5 R20 software.
- With the help of finite element method the truck bumper is meshed using HYPERMESH software.
- Modal analysis is done considering natural frequency of the materials properties and geometric changes in designs using ABAQUS.

Geometric modeling of truck bumper

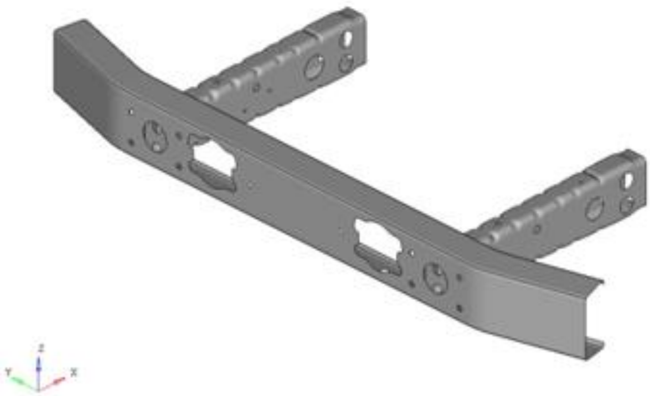

Fig 1. Isometric view of bumper

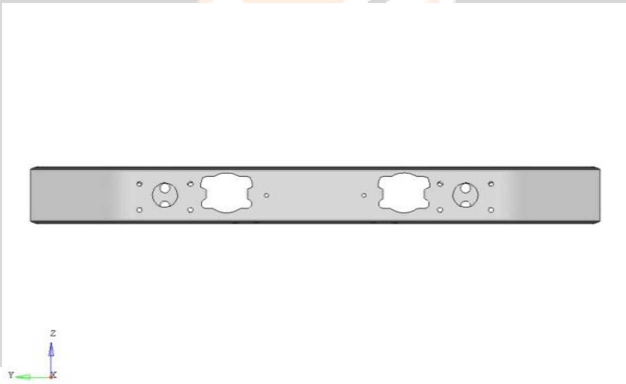

Fig 2. Front view of bumper

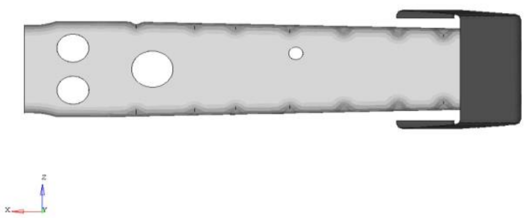

Fig 3. Side view of bumper

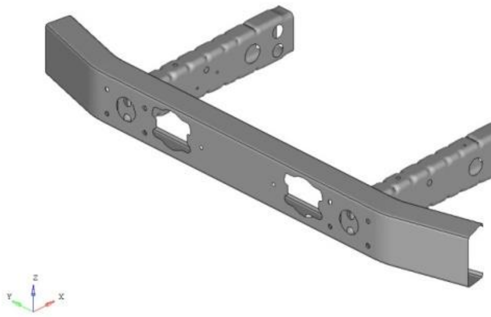

Fig 4. Base model of bumper with thickness 1.6 mm

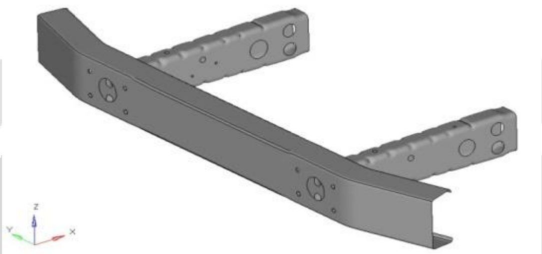

Fig 5. Modified model of bumper with thickness 1.6 mm.

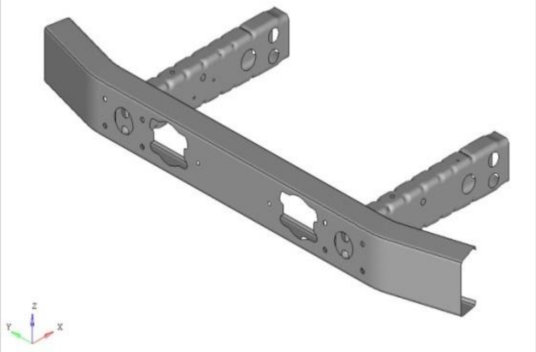

Fig 6. Modified model of bumper with thickness 2.4 mm.

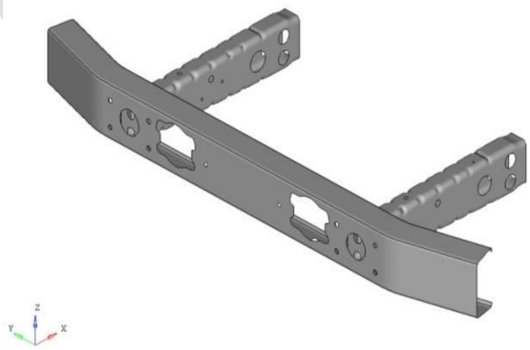

Fig 7. Modified model of bumper with thickness 3 mm.

Meshing of bumper

Meshed model is done by using HYPERMESH. Here shell mesh is selected for meshing the bumper. In shell mesh the thickness of the component is to be mentioned, so thickness of bumper is 1.6 mm, 2.4 mm and 3 mm for steel, cast iron and aluminium. Here element size of mesh is selected as 6 mm and thickness of shell mesh for steel bumper is 1.6 mm. Meshing is done using HYPERMESH software. The number of elements used for the meshing of the bumper is 27224 and numbers of nodes are 27463.

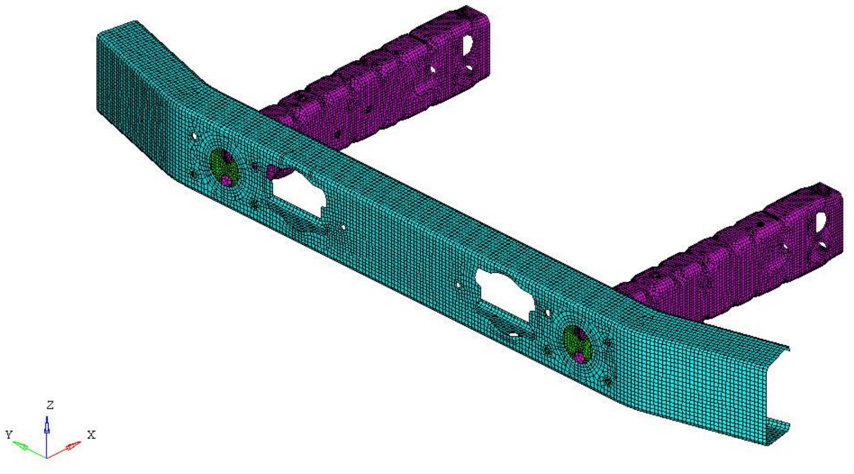

Fig 8. Shell meshed steel bumper

Results and discussion

Boundary conditions of bumper for normal modal analysis

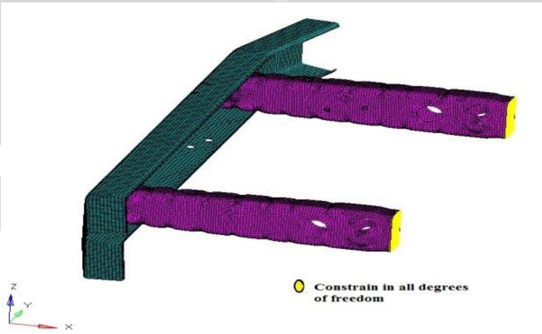

Fig 9. Boundary condition of bumper

The above model has five parts these are two long members and two supporting plate and bumper. Bumper is connected to long members with supporting plates.

Modal analysis of bumper for geometry shapes

Base model design

Base model is the truck bumper model with material is steel. For this bumper model analysis is done with four natural frequencies.

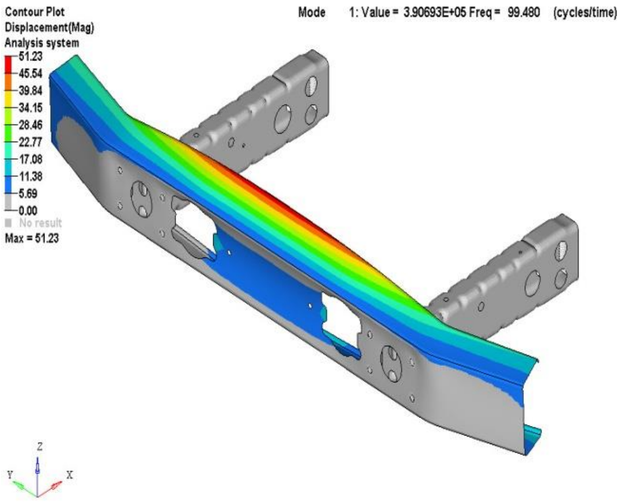

Fig 10. Base model design

Modified material design-2

In this modified material design-2 the material is cast iron with base model. For this type of material fist four modal natural frequencies are found out.

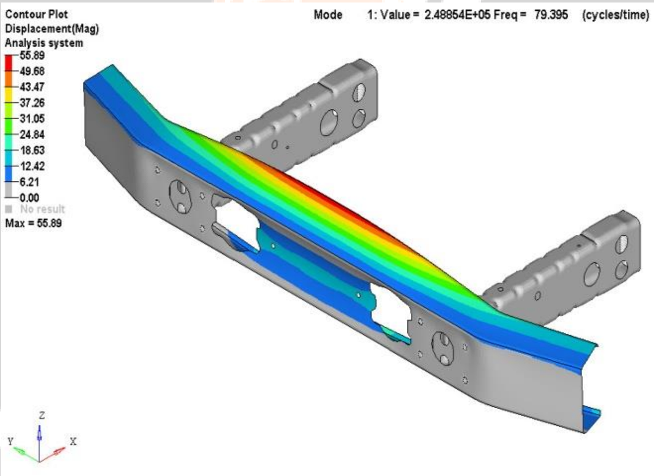

Fig 11. Modified material design-2

Modified material design-3

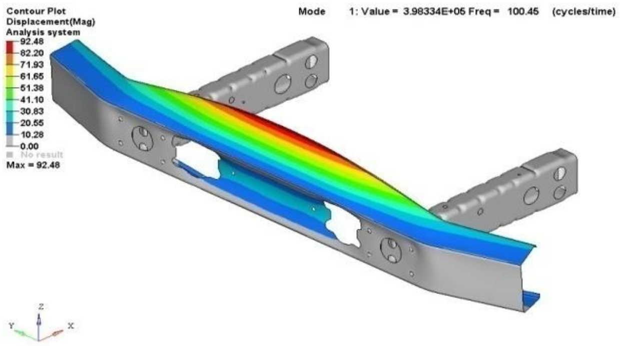

Fig 12. Modified material design-3

Conclusion

In this project material and geometry change for bumper was selected to carry out the modal and nonlinear static analysis to find out the best one and life of bumper. According to the series of analysis considering the materials and geometry change are respectively steel and design-1 is best one and gives better life than other materials and geometry change designs. modal analysis for bumper made of geometry design-1 is better and considering nonlinear static analysis for bumper made of steel is better.

References

1. Prabhakaran, Chinnarasu, Senthil, "Design and fabrication of composite bumper for light passenger vehicle", International Journal of Modern Engineering Research, 2(4), 2012.
2. Shahril, Shazwan, Othman, "Stress distribution behavior of car bumper bracket", 2013.
3. Alijani, Amabili, "Non-linear static bending and forced vibrations of rectangular plates retaining non-linearities in rotations and thickness deformation", International Journal of Non-Linear Mechanics, 2014.
4. Marzbanrad, Alijanpour, Kiasat, "Design and analysis of an automotive bumper beam in low-speed front all crashes" Thin Walled Structures, 47, 2009, 902-911.
5. Ayhan, Geneland, "Simulation of non-linear bending behavior and geometric sensitivities for tubular beams with fixed supports", Thin Walled Structures, 51, 2012, 1-9.
6. Naderi, Iyyer, "Fatigue Life Prediction of Cracked Attachment Lugs Using XFEM", International Journal of Fatigue, 2015.
7. Celik, Altunel, "The fatigue life estimation of a product having elasto-plastic behavior", Engineering Failure Analysis 26 (2012) 100-107.
8. Belingardi, Beyene, Koricho, "Geometrical optimization of bumper beam profile made of pultruded composite by numerical simulation", Composite Structures 102 (2013) 217-225.
9. Belingardin, Scattina, "Experimental investigation on the bending behaviour of hybrid and steel thin walled box beams – the role of adhesive joints", International Journal of Adhesion & Adhesives 40 (2013) 31-37.
10. Cheon, Choi, Lee, "Development of the composite bumper beam for passenger cars", Composite Structures 32 (1995) 491-499.