Structural Durability Analysis of Powertrain Mounting Bracket

Sameer U. Kolte[*1],David Neihguk [2], and Abhinav Prasad[3]
Mahindra and Mahindra Ltd.
* Mahindra Research Valley, Mahindra World City, Chennai, India
KOLTE.SAMEER@mahindra.com

Abstract: Structural analysis is performed to check durability of specified part for a given load and support conditions. For the component to be safe structurally, in any domain, the stresses generated should not exceed the yield strength of the material. However, considering possibility of fatigue failure, the component is optimized such that stresses generated do not exceed the endurance strength of the material. The engine mount bracket is subjected to loads primarily due to weight of the powertrain, the unbalanced torque. In this analysis, structural mechanics library of Comsol is used. The CAD model is imported from CATIA®V5R21. Tetrahedron is selected as the meshing element. The material is selected from 'built in' materials in Comsol library. The bracket is subjected to different load conditions like 3-2-1g weights, 3g downwards unbalanced torque etc and is optimized so that stresses generated and deflection of the bracket is within permissible limit. Effect of bolt preloads is also considered. The results of the analysis are compared with Hypermesh® results and are found to be in good correlation.

Keywords: Powertrain mounts, Engine mounts, Powertrain NVH, Stress analysis

1. Introduction

Powertrain mounts have great effect on the noise, vibration and harshness characteristics of the vehicle. They are prone to failure since they have to withstand heavy dynamic loads of the powertrain in the operating conditions. To ensure that the design is foolproof, it is necessary to bring down the stress levels to a permissible limit. In the presented analysis, COMSOL is used to analyze an Engine mounting bracket to evaluate its structural performance and design optimization.

1.1 Model

Figure 1 shows the powertrain mounting bracket assembly. The bracket serves as the connector between the powertrain and the rubber mount.

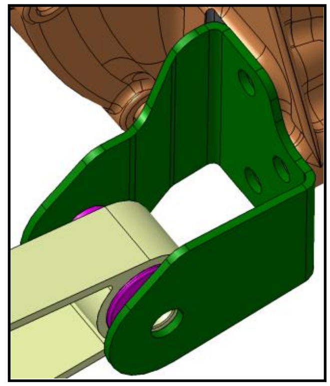

Figure 1: Powertrain mount assembly

2. Pre-Processing

2.1 CAD Model

The CAD model of the powertrain mount bracket is imported from CATIA®V5 into COMSOL.

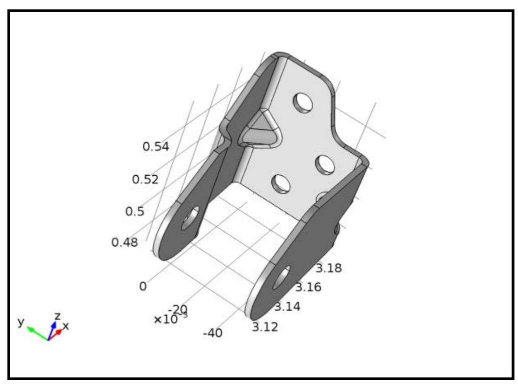

Figure 2: Imported CAD model in COMSOL

2.2 Material

Material was applied to the model from 'built in' library of COMSOL. Structural Steel' was selected since its material properties were close to that of actual material.

Excerpt from the Proceedings of the 2013 COMSOL Conference in Bangalore

Table 1: Material Properties

Material Property	Value	Unit
Density	7850	kg/m^3
Young's modulus	210	GPa
Poisson's ratio	0.33	–
Ultimate Tensile Strength	410	MPa
Yield Strength	270	MPa
Endurance Strength	210	MPa

2.3 Meshing

Once the material was selected, mesh was generated for bracket. 'Free tetrahedral' mesh element was selected for meshing with 'extra fine' element size for greater accuracy.

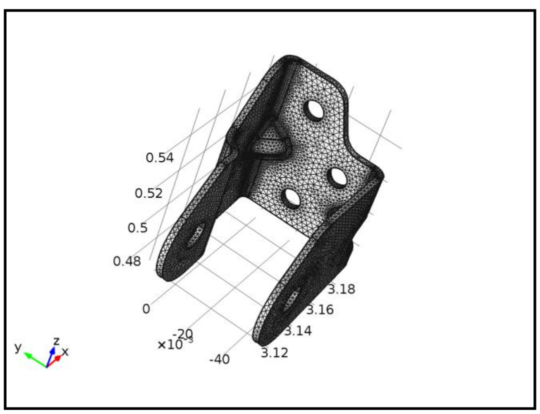

Figure 3: Meshed Bracket

Table 2: Mesh Properties

Mesh Property	Value
Maximum element size	0.00316
Minimum element size	1.36E-4
Resolution of curvature	0.3
Resolution of narrow regions	0.85
Maximum element growth rate	1.35
Predefined size	Extra fine

2.4 Boundary Conditions

The mounting locations of the bracket on the powertrain were fixed. The boundary loads were applied on locations where Rubber mount is bolted to the bracket.

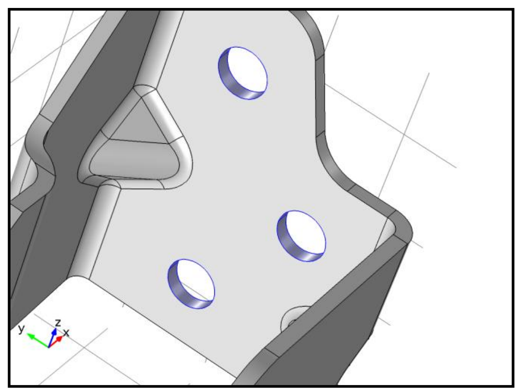

Figure 4: Fixed constraints

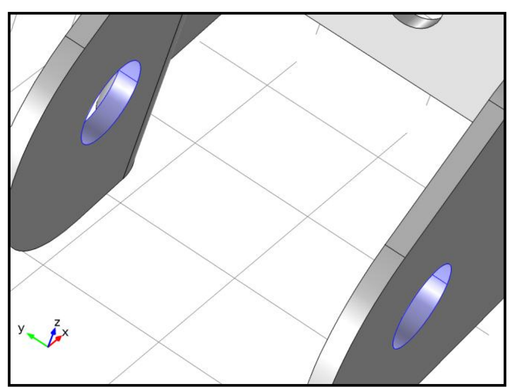

Figure 5: Boundary Load

Table 3: Applied Loads

Force Direction	Force Value
X	-3000 N
Y	100 N
Z	3000 N

The values of loads were obtained by performing dynamic simulation of powertrain which is out of the scope of this paper.

3. Processing

The processing stage involves stiffness generation, stiffness modification, and solution of equations, resulting in the evaluation of nodal variables. Following is the element matrix equation.

$$[K^e]\{u^e\} = \{f^e\}$$

4. Post-processing

4.1 Plotting the results

Von-mises stresses and displacement results were plotted.

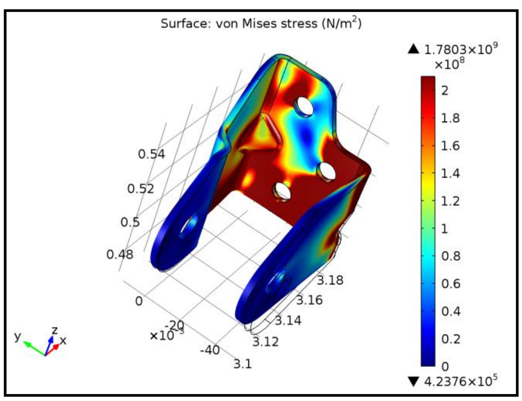

Figure 6: Stress Plot

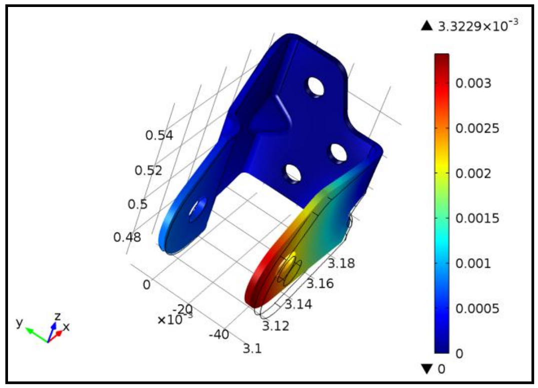

Figure 7: Displacement plot

4.2 Analysis of results

From obtained results it was observed that the stresses induced are above the permissible endurance stress. Hence it is required to do design modifications in order to reduce the induced stress level. The bracket is weaker at base section hence it needs to be strengthened.

4.3 Design modifications

To improve strength, additional member of less thickness was inserted inside main bracket. Flanges were provided to this new member in order to increase stiffness of the base plate.

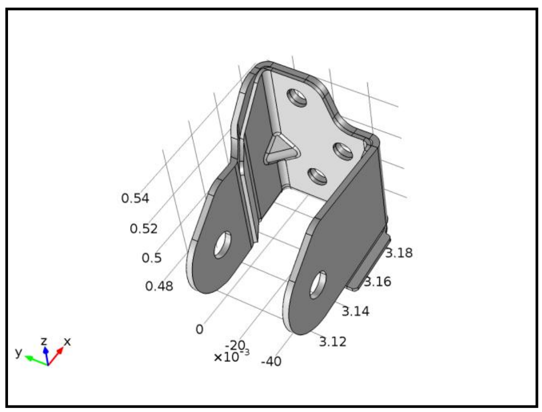

Figure 8: Modified bracket

Above analysis was once again performed. To represent the welding between two brackets 'contact constraint' was added at welding locations.

Following results were obtained-

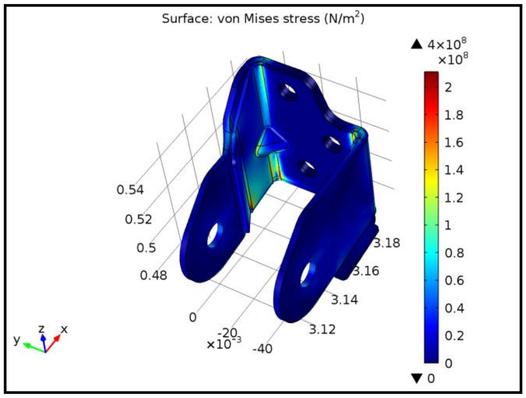

Figure 9: Stress plot for modified bracket

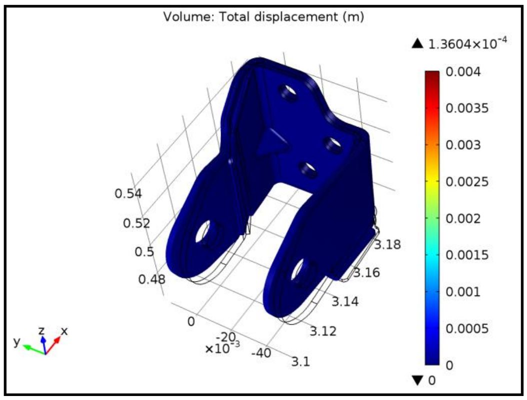

Figure 10: Displacement plot for modified bracket

5. Correlations

Results obtained from COMSOL were validated with the results from Hypermesh® analysis. The correlation was extremely good in terms of values of stresses, stress pattern and displacement.

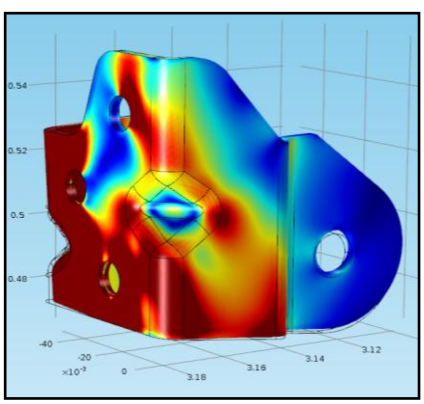

Figure 11: COMSOL Stress plot

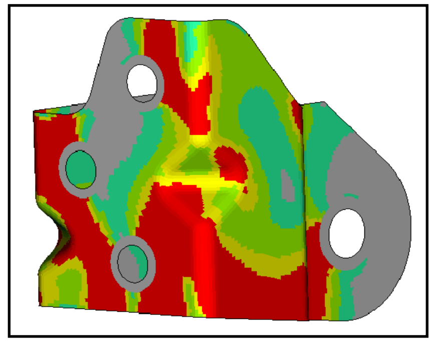
Figure 12: Hypermesh® stress plot

6. Conclusions

Structural analysis of powertrain mounting bracket was successfully performed using COMSOL Multiphysics. The initial results obtained were used as a guide to do necessary design modifications in bracket to meet the strength criteria. The modified bracket was analyzed again. It was observed that the new bracket passes the strength criteria.

7. References

1. J.N.Reddy, 'An Introduction to Finite Element analysis', McGraw-Hill Inc.1993.
2. Taylan Altan and A. Erman Tekkaya, 'Sheet Metal Forming - Processes and Applications', ASM International.
3. 'Automotive Handbook', BOSCH , 8th Edition.