Stress analysis of engine mounting assembly of a three wheeler

Deepak Singh¹ and Rajeev Kumar²
¹Research Scholar, Department of Mechanical Engineering, Institute of Engineering & Technology, Lucknow,
²Professor, Department of Mechanical Engineering, Institute of Engineering & Technology, Lucknow

Email: deepakx2005@gmail.com

Abstract- One of the essential parts of an auto vehicle is an engine mounting bracket, which helps in mounting the engine on the chassis. This paper analyses and compares the existing bracket design of one of the famous three-wheeler of Scooter India limited named as Vikram 750 D with five new alternative models using finite element analysis. In the first part, linear structural finite element analysis of engine mounting bracket of the existing design was carried out to determine the maximum stresses and deformations in the existing model and then identical analysis was conducted for five proposed models of the mounting brackets. The maximum stresses and deformations of the proposed models were compared with that of the existing model to identify a better design. The weight of the existing model and five proposed model were also compared to obtain an optimal design.

Keywords: FEA, ANSYS, Structural Analysis, Vikram 750D

1. Introduction-
Engine is one of the most vital components of a transport vehicle. It is generally supported by engine mounting bracket system. The improvement in the design of the engine bracket system has been the subject of great interest for many years. As suggested by Ghorpade et al, [1] the automotive engine mounting systems are very important due to different aspects of performance. They presented a finite element (FE) analysis of a basic engine mounting bracket of a car to determine the natural frequency of the bracket.

Adkine and Kathavate [2] have performed static as well as modal analysis of engine mounting bracket by using ANSYS for investigating whether the current natural frequency of engine mounting bracket is lower than that of self-excitation frequency of bracket.

Dhillon et al, [3] proposed that the exact geometry and positioning of engine mounting brackets on the chassis ensures a good riding quality and performance of the vehicle. They described the solid modelling, finite element analysis, modal analysis and mass optimization of engine mount brackets for an FSAE car. Sebastian et al, [4] suggested that the analysis of a jet engine mounts by finite element method usually deals with the stress analysis of the mount. It is pointless to believe that only stress analysis is essential, as the displacement of the mount (elongation) is also pivotal factor when real-life scenarios come across. Kolte et al, [5] performed structural analysis to check the durability of specified part for a given loading 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.

In an automobile, the engine mounting brackets are connected to the main-frame of the vehicle and support the engine. While in operation, the unsought stresses and vibrations generated by the engine and road roughness can transmitted to the frame directly through the brackets. These vibrations may
cause discomfort to the passenger sitting inside the vehicle or might even damage the chassis of the vehicle [6].

The main purpose of an engine mounting bracket is to support the engine therefore it is required to design it properly. The stress and vibrations produced in an engine mounting bracket have been continuously a matter of great concern and may lead to the failure of structure. If the vibration in the engine mounting further exceeds it may cause fatigue and sometimes it damages the vehicle [1].

In this paper first, the existing design of engine mounting of three-wheeler Vikram 750 D is analysed using finite element analysis on the three-dimensional model of the bracket to determine the stress distribution & displacements at various points of mounting. In the second part, five alternative designs of the bracket were proposed and were analysed using finite element analysis by subjecting to identical loading conditions on their respective three-dimensional models. The maximum stresses & displacements of each proposed designs are calculated and compared with the results obtained from the present design.

2. Materials & Methods

2.1 Engine Mounting

The engine mounting assembly of an existing model of Vikram 750D consists of main three parts

1. Bracket
2. Channel
3. Square bar

The manufacturing of each part is discussed as follows.

Bracket

The bracket is fabricated by shearing off a 5 mm thick sheet according to dimensions shown in the bracket drawing in fig 1 and then by bending the sheet at 130°. Finally, two holes and two slots are also pierced off from the sheet.

![Figure 1. 2-D Drawing of Bracket](image-url)
Channel
The channel is fabricated using a 1 mm thick sheet cut off using a shearing machine, and then notching operation is performed according to the drawing given in figure 2. Finally, it is bent according to the drawing. The two brackets fabricated earlier are welded at the two ends of the channel through MIG welding.

Square Bar
This component is made from standard 16 x16 mm² bar. A square bar of 325 mm length is cut off using a press machine as per the drawing is shown in figure 3. This square bar is welded inside the U-shape channel for increasing the strength of the channel.

Figure 2. 2-D Drawing of Channel

Figure 3. 2-D Drawing of Square Bar

The dimensional details of the bracket geometry are listed in Table 1.
Table 1. Dimensions of the main parts of the engine mounting assembly.

| Assembly Parts        | Length (mm) | Width (mm) | Thickness (mm) |
|-----------------------|-------------|------------|----------------|
| Bracket               | 225         | 144.22     | 5              |
| U-shape Channel       | 511.83      | 14         | 1              |
| Square Bar            | 325         | 12         | 12             |

Finally, all three parts of engine mounting described above are assembled and joined as per the drawing using MIG welding. The real image of engine mounting bracket assembly is shown in figure 4 and 5.

**Figure 4.** Rear view of engine mounting bracket with the engine mounted on it.

**Figure 5.** Picture of engine mounting bracket with the engine mounted on it.
2.2 Static Structural Analysis

In static structural analysis, displacements, stresses are determined on the structure due to load distribution. The steps used in the analysis are shown in fig. 2.6.

![Flow Chart for Structural Analysis](image)

The basic steps involved in the development and analysis of existing and proposed designs are discussed in the following subsections.

2.2.1 Modelling

A three-dimensional solid model of the engine mounting bracket was developed using Solidworks solid modelling software. The dimensions of the assembly as mentioned in Table 2.1 were taken from the actual mounting bracket used in Vikram 750D by Scooter’s India Limited, Lucknow. Figure 7 shows the three-dimensional solid model of the actual bracket assembly used in the three-wheeler.

![Three dimensional solid models of engine mounting assembly developed in Solidworks software](image)
2.2.2 Meshing
The solid model of the engine mounting assembly was then imported to the ANSYS Workbench environment. The materials properties of the material as given in Table 2 used in the fabrication of assembly were assigned.

Table 2. Properties of engine mounting bracket material

| Property            | Unit                  |
|---------------------|-----------------------|
| Elastic modulus     | $2 \times 10^{11}$ N/m$^2$ |
| Poisson’s ratio     | 0.29                  |
| Shear modulus       | $8 \times 10^{10}$ N/m$^2$ |
| Density             | 7870 Kg/m$^3$         |
| Tensile strength    | 325 MPa               |
| Yield strength      | 180 MPa               |

The model was then meshed using tetrahedral elements with the help of the mesh tool provided in the ANSYS workbench. This tetrahedral element consists of four nodes, with each node having three degrees of freedom in X, Y and Z directions. The meshed model of the existing mounting assembly has 17119 nodes and 8767 elements.

Figure 8. Meshed model of existing design

2.2.3 Boundary Conditions
In an automobile, the engine is mounted on the mounting bracket by tightening four bolts on the tapered side walls provided at both the sides of the bracket. For ease of mounting four elliptical slots (30 x 9 mm) are provided, two slots of on the each of the tapered wall. To apply the boundary conditions of the model, these four elliptical slots were fixed assigning all degrees of freedoms to zero.
Figure 9 and 10 display the left and the right tapered walls of the bracket having elliptical slots fixed in the model.

**Figure 9.** Fixed Elliptical slot on the left tapered wall of the existing model

**Figure 10.** Fixed Elliptical slot on the right tapered wall of the existing model

### 2.2.4 Loading

The weight of the existing engine block of Vikram 750D is 703 N. This is distributed over the two horizontal flanks provided in the bracket to support the load of the engine block. In the FE model of the bracket, the entire load was divided into half and applied as uniformly distributed over the horizontal flanks in vertically downwards direction. Figure 11 displays the FE model of the bracket with applied load.
2.2.5 Solution

After the loads and boundary conditions were specified, the model was solved for linear, static structural analysis. Once the solution is obtained, Von-Mises stress and maximum deformation contour plots for the model were generated. Figure 12 and figure 13 displays the contour plots for Von-Mises stress distribution and maximum deformation respectively. Table 3 lists the maximum values of Von-Mises stress and maximum deformation obtained in the analysis for the existing design.

Table 3. Values of maximum Von-mises stress and maximum deformation for the existing design

| Von-Mises Stress (MPa) | Deformation(mm) |
|-----------------------|-----------------|
| 8.309                 | 0.025166        |
Figure 12. Von-Mises Stress distribution on the existing design

Figure 13. Maximum Deformation in the existing design

From the figure 12 it is seen that the Von-Mises stress (maximum) of the value 8.309 MPa is induced at the bend of the bracket plate and slightly at the rear portion of the two holes of 11 mm diameter to which the engine is fastened. Similarly, figure 13 displays that the total maximum deformation of $2.5166 \times 10^{-2}$ mm occurred at the ends of the horizontal flank of the bracket plate.

3. ANALYSIS OF PROPOSED DESIGNS
The proposed designs were analyzed by the method described in the previous section under the identical load and boundary conditions and the results of stresses & displacements developed at various points are compared with that of the existing model. Total of five proposed designs were studied in this work. Following are the details of each proposed model:
Model-1
In model-1 “L” shaped channel is used instead of “U” shaped channel of the existing model as shown in fig 14. The geometry of the bracket remains the same as in the present design however the square bar is not used in this design.

![Figure 14. Three-dimensional model of engine mounting assembly with L-channel throughout](image)

Model-2
For model-2 shown in fig 15 the thickness of the “U” shaped channel was increased to 3 mm instead of 1 mm of the present design, while the remaining geometry was same as the present design.

![Figure 15. Three-dimensional model of engine mounting assembly having 3 mm thickened channel](image)

Model-3
In the third proposed design (model-3) the bracket and the square bar were removed and the entire engine mounting assembly was made as a single component of thickness 3 mm with the flange as shown in fig 16.
**Figure 16.** Three-dimensional model of engine mounting assembly as a single component of 3mm with flange

**Model-4**
In model-4, as shown in fig 17 the bracket and the square bar were removed and the entire engine mounting assembly was made as a single component of thickness 6 mm without flange.

**Figure 17.** Three-dimensional model of engine mounting assembly as a single component of 6 mm without flange

**Model-5**
For model-5, the square bar of the existing design was removed whereas the shape of the channel is in the form of a square bar of cross section 16x16 itself was used shown in fig 18. The geometry of the bracket was the same as used in the present design.
4. Result & Discussion

The finite element analysis of the existing bracket was conducted in two parts. In the first part, a solid model of the existing bracket used in Vikram 750D was developed and a linear, static FE analysis was carried out to determine the base values of stresses and deformation in the bracket.

In the second part, five new designs of the bracket were proposed and finite element analysis was performed on each using the method discussed in section 2.2 under identical boundary conditions. Figure 19 to figure 28 depicts the Von-mises stress distribution and maximum deformation in all proposed designs.

**Figure 18.** Three-dimensional model of engine mounting assembly with square bar channel throughout

**Figure 19.** Von-Mises Stress distribution on proposed model-1
Figure 20. Maximum Deformation in proposed model-1

Figure 21. Von-Mises Stress distribution on proposed model-2

Figure 22. Maximum Deformation in proposed model-2
**Figure 23.** Von-Mises Stress distribution on proposed model-3

**Figure 24.** Maximum Deformation in proposed model-3

**Figure 25.** Von-Mises Stress distribution on proposed model-4
Figure 26. Maximum Deformation in proposed model-4

Figure 27. Von-Mises Stress distribution on proposed model-5

Figure 28. Maximum Deformation in proposed model-5
The values of max stresses and deformations for each are tabulated in table 4.

**Table 4.** Solutions of existing as well as proposed models

| Design        | Max. Von Mises Stress (MPa) | Max. Deformation (mm) |
|---------------|-----------------------------|-----------------------|
| Existing Design | 8.309                       | 0.02516               |
| Model-1       | 8.648                       | 0.03959               |
| Model-2       | 7.084                       | 0.02735               |
| Model-3       | 19.745                      | 0.06075               |
| Model-4       | 22.1                        | 0.01                  |
| Model-5       | 6.3                         | 0.02617               |

Mass of the engine mounting also plays a very critical role in designing of engine mounting assembly. Low mass is always preferable for engine parts. Therefore, to achieve this, the mass of the existing design as well as five proposed designs are calculated and tabulated in table 5.

**Table 5.** Mass of Existing & Proposed Models

| Design        | Mass (Kg) |
|---------------|-----------|
| Existing Design | 3.4754    |
| Model-1       | 3.2932    |
| Model-2       | 3.3842    |
| Model-3       | 2.4271    |
| Model-4       | 5.1644    |
| Model-5       | 3.5739    |

Figure 29 and fig 30 display the comparison of Von-mises stresses in the existing and proposed designs and comparison between the maximum deformation in the existing and proposed design respectively.
Figure 29. Comparison of Von-Mises stress produced in different designs

Figure 30. Comparison of deformation produced in various designs

Figure-31 displays the mass comparison of each model of the engine mounting bracket.
From Table 4, figure 28 and 29 it is evident that the minimum stress was induced in Model-2 and Model-5 but the deformation in these models is slightly more than the existing model. The minimum deformation was observed in the Model-4 but the induced stress in this model is very large. Model-1 and Model-3 show both stress and deformation is more in these models as compared to the existing model.

Table 5 and fig 31 shows that the Model-1, Model-2 and Model-3 are lighter than the existing model whereas Model-4 and Model-5 are heavier than the existing model.

5. Conclusion
The finite element analysis of the existing model and five proposed model of engine mounting bracket were conducted and the values of stress and deformations are compared. It can be concluded that the two models which may be considered for the existing model are Model-2 and Model-5 as under working condition the maximum stress developed in them is less than the maximum stress developed in the existing design. However, on comparing these two models, it was concluded that the better choice for designing engine mounting bracket is Model-5 (Design with square bar channel throughout), as the maximum stress in this model is less than the Model-2. As far as maximum deformation and mass are concerned these parameters are slightly more in Model-5 but this increment of deformation and weight has not much impact on the design. Another reason for choosing Model-5 is that the manufacturing of Model-2 involves two additional processes viz. shearing of sheet & two bending processes which will be requiring additional operation cost as well as time, whereas the manufacturing of Model-5 is simple, as it requires only the process of part off and bending which is very simple & less time taking as compared with the other design.

6. Acknowledgement
The authors would like to thank the management and the staff of Scooter India Ltd., Lucknow especially Mr Anil Kumar, (Retd. Chief Engineer) for their steady support in the execution of this
research. We would also like to acknowledge the role of Mr Gaurav Seikri for his guidance in understanding the software like Solidworks & ANSYS.

7. Conflict of Interest
This research work carried out by me has not received any grant from any funding agency of commercial or public sector. It is also not for profit sectors.

8. References

[1] Ghorpade, U. S., Chavan, D. S., Patil, V., Gaikwad, M., 2012. “Finite element Analysis and Natural Frequency Optimization of Engine Bracket” International Journal of Mechanical and Industrial Engineering (IJMIE), Vol. 2, Issue 3, pp. 1-6.

[2] Adkine, A. S. and Kathavate, V. S., 2015. “Design and Analysis of Engine Mounting Bracket using ANSYS Tool”. International Journal of Innovation in Engineering, Research and Technology [IJIERT], ICITDCEME’15 Conference Proceedings, April 2015, pp. 1-6.

[3] Dhillon, J. S., Rao, P., Sawant, V. P., 2014. “Design of Engine Mount Bracket for an FSAE Car Using Finite Element Analysis”. International Journal of Engineering Research and Applications, Vol. 4, Issue 9, pp. 74-81.

[4] Sebastian, C. S., Baby, S., Tony, A., 2016. “Design and Optimization of Engine Mount Bracket”, International Journal of Application or Innovation in Engineering & Management (IJAITEM), Vol. 5, Issue 2, pp. 168-172.

[5] Kolte, S. U., Neihguk, D., Prasad, A., 2013. “Structural Durability Analysis of Power-train Mounting Bracket”, Excerpt from the Proceedings of the COMSOL Conference in Bangalore, 2013, pp. 1-4.

[6] Adkine, A. S., Kathavate, V. S., Overikar, G. P., Doijode, S. N., 2005. “Static Behaviour of Engine Mounting Bracket”. International Advanced Research Journal in Science, Engineering and Technology, Vol. 2, Issue 4, pp. 68-73.

[7] Maski, S., and Basavaraj, Y., 2015. “Finite Element Analysis of Engine Mounting Bracket by Considering Pretension Effect and Service Load”. International Journal of Research in Engineering and Technology (IJERT), Vol. 04, Issue 08, pp. 327-333.

[8] Deshmukh, M. and Sontakke, K. R., 2015. “Analysis and Optimization of Engine Mounting Bracket”. International Journal of Scientific Engineering and Research (IJSER), Vol. 3, Issue 5, pp. 131-136.

[9] Jadhav, P. D. and Ramakrishna, 2004. “Finite Element Analysis of Engine Mount Bracket”. International Journal of Advancement in Engineering and Technology, Management and Applied Science, Vol. 1, Issue 4, pp. 1-10.

[10] Ghorpade, U. S., Chavan, D. S., Kavade, M. V., 2012. “Static Structural and Modal Analysis of Engine Bracket Using Finite Element Analysis”. International Journal of Engineering Research & Technology (IJERT), Vol. 1, Issue 10, pp. 1-6.