International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 07 | July 2018 www.irjet.net p-ISSN: 2395-0072

Design and Analysis of Foot Mounting Bracket of Seating System for Passenger Vehicle

Mr. Suraj N. Mahadik[1], Prof. Vikram S. Mahajan[2]

[1]PG Scholar, Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune- 411033
[2]Associate Professor, Department of Mechanical Engineering, Rajarshi Shahu College of Engineering, Pune-411033

Abstract - *Foot mounting bracket is the bracket used to mount the seating system in the vehicle. These brackets support to the seating system and hold the seat rigidly in place and also prevent it from shifting around. Keeping this in mind the current paper discusses static analysis of foot mounting bracket by using Hypermesh and Abaqus. Static analysis of foot mounting bracket was done for checking design of existing and modified bracket. The results were analyzed for stresses and displacements. From the design and analysis, the stress and displacement of modified bracket were 111.2 MPa and 4.928 mm which is less than existing design so the design is safe. After experimentation, results were validated with FEA results.*

Key Words: **Foot Mounting Bracket, FEA, Static Analysis, Experimental Analysis.**

1. INTRODUCTION

"Seat mounts" or "seat bases", these are basic support plates that bolt to the vehicle floor and serve as a mounting point for other seat items that are placed on top of them. Seats with front-to-rear adjustment will have seat slider bars that bolt to the top of these brackets. In some cases, seat brackets may come complete with integrated slider rails already attached. Depending on vehicle application, seat brackets may be one piece, or multiple individual pieces.

Foot mounting bracket is the bracket used to mount the seating system in the vehicle. Seat assembly is mounted on this bracket. The driver seat bracket of Indica Vista X601 passenger vehicle is taken for study. The existing bracket is made of D513 steel grade. The existing bracket faces the problem of high stress and displacement. Considering stress and displacement, the existing bracket should be designed for reduction of stress and displacement.

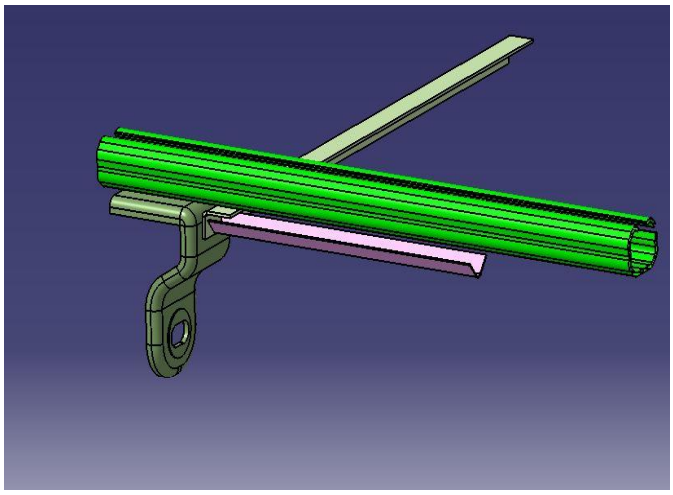

Fig-1: Existing Bracket

Fig.1 shows the existing foot mounting bracket which is present below the seat assembly carries overall weight on the seat. Seats with front to rear adjustment have seat sliding bars that bolt to the top of these brackets. Seat bracket carries overall weight from the seat. The material of the bracket is D513 and the thickness of the bracket is 3mm and it is spot welded to the bar. Manufacturing of the bracket requires various die operations such as draw die, piercing die, stamping die etc.

The modelling of the existing bracket is done in CatiaV5 where meshing is done in Hypermesh and Abaqus solver is used for analysis. The results of existing bracket obtained with high stress and displacement. So, there is necessity to design the new bracket which can withstand for high stress and displacement. The modelling of the modified bracket is done in CatiaV5 while the meshing is done in Hypermesh and analysis is done in Abaqus. Comparing the results of existing bracket and modified bracket, best is selected for the desired application.

2. LITERATURE REVIEW

Dhillon et al. discusses the modeling, Finite Element Analysis, Modal Analysis and mass optimization of engine mount brackets for a FSAE car. As the brackets tend to undergo continuous vibrations and varying stresses, the fatigue strength and durability calculations also have been done to ensure the engine safety [1]. M. V. Srinivasan et al. in this paper they design the car seat mounting bracket with seat arrangement by using Pro/E software for Automobile Applications. They had conducted structural analysis by varying the seat bracket material. By obtaining the results the comparison is done for three materials to validate better material for seat mounting bracket to find out which material is best to withstand in loading condition. ANSYS software is used for analysis [2].Sanket Vinchurkar et al. states in this paper that the function of an engine mounting bracket is to safely support the power-train system in all of the conditions. Since it is very difficult to change the supporting locations and types of support after the engine is built, the mounting brackets must be verified in the design stage. This paper includes study of design and analysis of engine mounting bracket. CAD model is generated through reverse engineering. Engine mount bracket of Mahindra Scorpio has been taken into study. After analyzing it, scope for optimization of engine bracket is suggested [3].

3. PROBLEM DEFINITION AND OBJECTIVES

In recent studies some problems are observed and they are

- Existing bracket have facing the problem of high stress and displacement
- Tooling cost of existing bracket is more

The main aim of this paper is

- The main objective of this project is to do static analysis of Foot mounting bracket.
- To design the new bracket for ease of manufacturing

4. METHODOLOGY

- The identification of problem and work area
- Collect research paper for literature review
- The benchmarking of existing bracket is the first step to consider
- Modelling of existing bracket is done in CatiaV5
- Meshing and analysis of bracket is done in Hypermesh and Abaqus.
- Applying preprocessing and post-processing i.e. applying properties and constraints and analyzing results on the basis of stress and displacement
- Modification of bracket by changing design based on post processing
- Accepted results if analysis results within desirable range and if not then change variables i.e. design
- Manufacturing of bracket is done on the basis of the design and experimentation of the bracket need to be done for checking design values.
- Validate experimentation and FEA result

5. FINITE ELEMENT ANALYSIS

Finite element method is the process of solving the complex and simple experimental methodology by converting them into simulation. Hypermesh is the product of Altair Engineering is the preprocessor which is widely used for finite element modeling. FEA consists of a computer model of a material which is stressed and analyzed for specific results. It is not only used in new product design but also existing product refinement.

5.1 Static Analysis of Existing Bracket

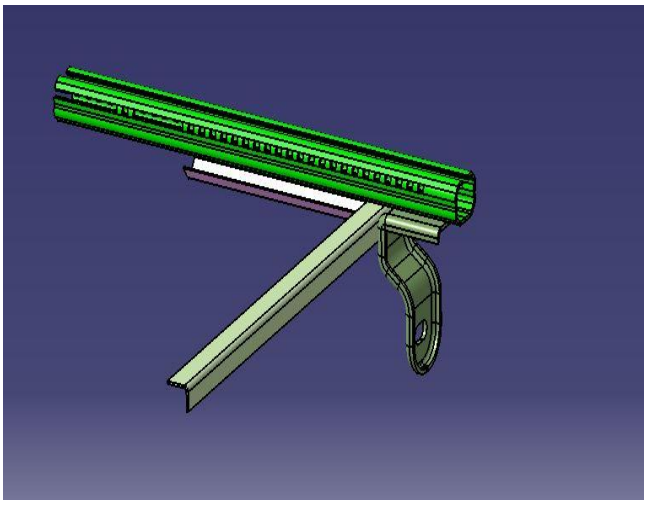

Fig.2 Modelling of Existing Bracket

It was decided to do 2D mesh for the CAD model. The modelling is done in CatiaV5 and Hypermesh software is used for meshing and analysis.

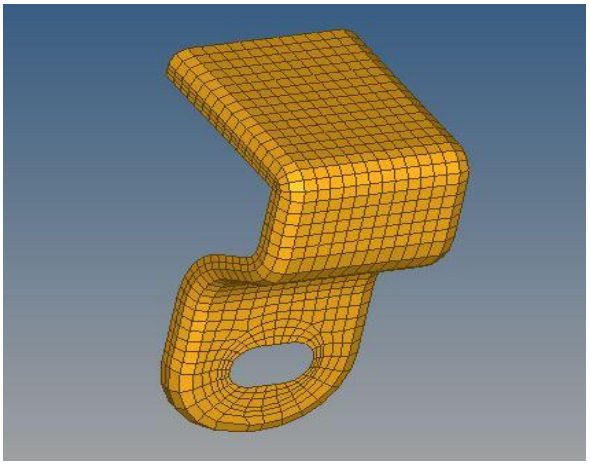

Fig.3 Meshing of Existing Bracket

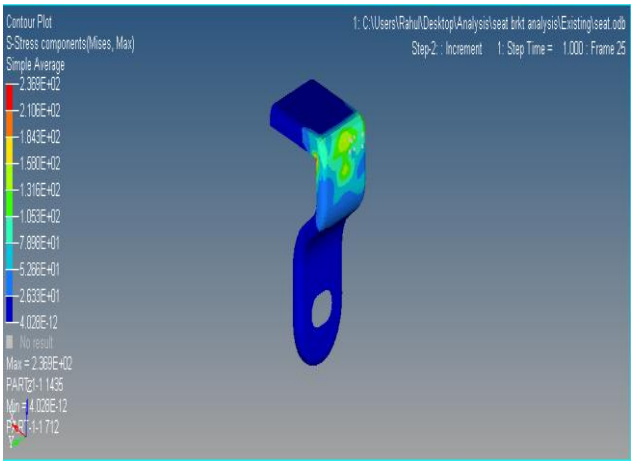

Fig.4 Stress of Existing Bracket

Boundary Condition

- Only seat weight and external applied load is considered for static analysis
- Weight of the seat = 15 Kg
- Average weight of person = 80 Kg
- Total weight on seat = 95 Kg = 95 × 9.81 = 932 N
- No. of seat bracket on which seat mounted = 4
- Force exerted on the bracket = 932 / 4 = 233 N

Here we considered 250N load on a single bracket

Material Properties

Material of existing bracket is D513 steel grade

- Yield strength = 220 MPa
- Young's modulus, E = 2.1×10^5 MPa
- Poisson's ratio, μ = 0.3
- Density, ρ = 7.8×10^{-9} tonnes / mm^3

Meshing is done in Hypermesh and the meshing criteria is

- Element size = 5
- Mesh type = mixed (quad+ tria)
- No of nodes = 1080
- No of elements = 1016

Weight of the bracket = 282 gram

Stress generated in the existing bracket is 236.9 MPa which is more than yield strength of material

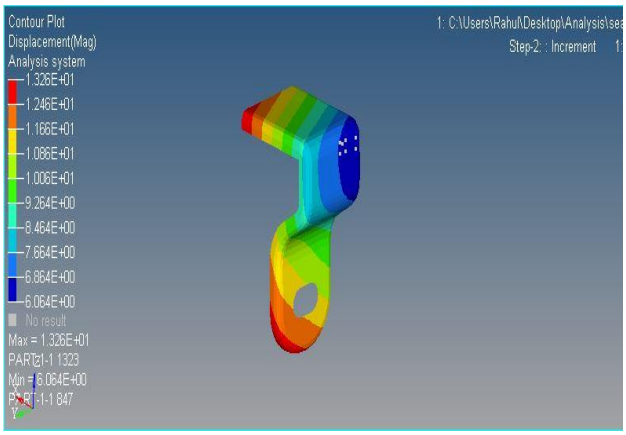

Fig.5 Displacement of Existing Bracket

The displacement of existing bracket is 13.26 mm and the direction of displacement is in z direction.

5.2 Design Modification

By studying of the results of analysis of existing bracket, certain design changes have been made here such as changing the shape of bracket for ease of manufacturing and addition of bead in the bracket is done for modified bracket. Shape is changed due to tooling feasibility. Tooling cost of existing bracket is more due to various die operations. The existing bracket requires draw die, piercing die, stamping die, bending die, and flanging die for manufacturing of the bracket but the modified bracket is designed in such a way that it reduces die operation i.e. it requires draw die, piercing die, forming die and bending die. So the cost of manufacturing of modified bracket is less than existing

bracket. Material of the existing and modified bracket is same.

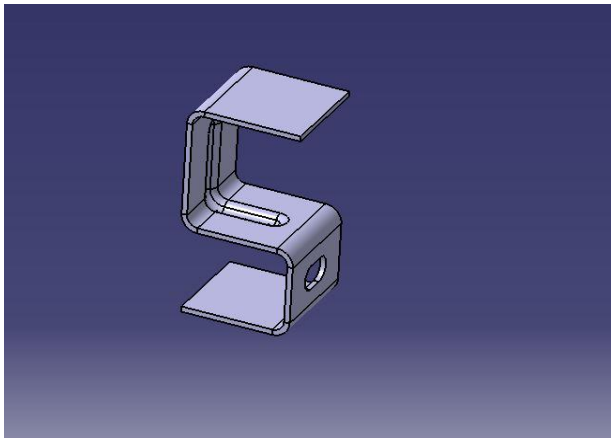

Fig.6 Modelling of Modified Bracket

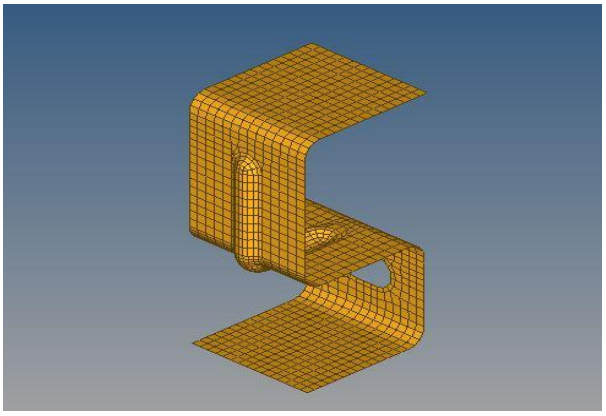

Fig.7 Meshing of Modified Bracket

Meshing is done in Hypermesh and the meshing criteria is

- Element size = 5
- Mesh type = mixed (quad + tria)
- No of nodes = 1537
- No of element = 1431

Weight of the modified bracket = 365 gram

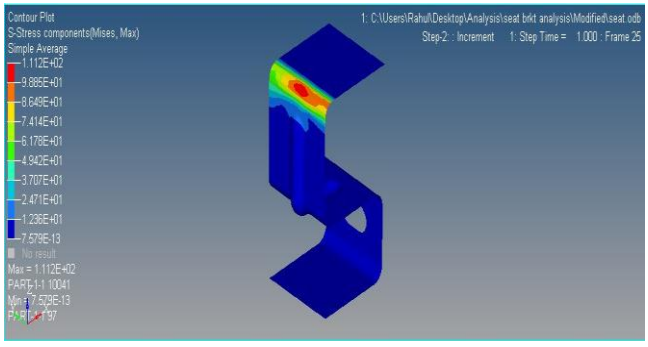

Fig.8 Stress of Modified Bracket

Stress generated in the modified bracket is 111.2 MPa

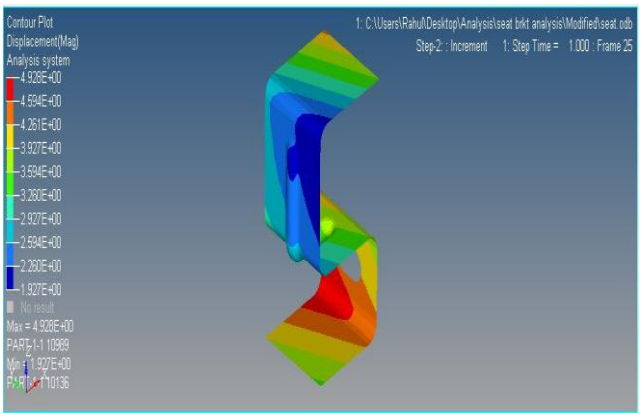

Fig.9 Displacement of Modified Bracket

The displacement of modified bracket is 4.928 mm and the direction of displacement is in z direction

6. EXPERIMENTAL ANALYSIS

The experimentation is done on the Universal testing machine. A universal testing machine is a machine that is used to perform standard tensile and compressive tests on materials, components and structures. It works on the principal of elongation and deformation.

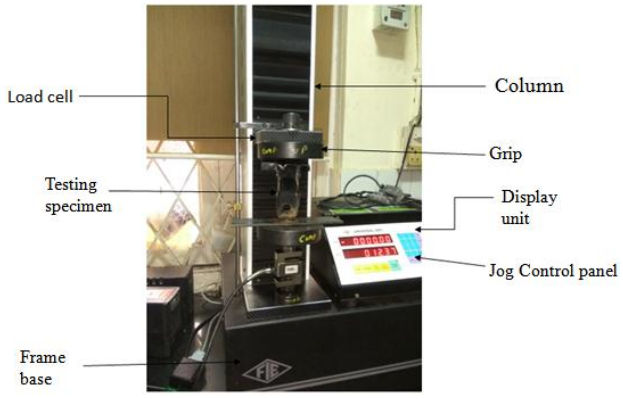

Fig.10 Experimental set up for compression test

The existing bracket is placed in between two plates, there is no need of special fixture for the bracket mounting. The load is applied on the bracket gradually by the speed of 2mm/min. By applying constantly load on existing bracket up to required loading condition i.e. 250 N. The displacement obtained in the bracket is shown in the display panel. In this case for 250 N load we get the displacement of 6.820 mm. We increased the load up to 330 N and the displacement obtained is 8.840 mm. The graph of load versus

displacement is displayed in the computer screen. Finally we observed that, load applied 250 N and found no cracks or deformation observed. Load applied to sample to 330 N and found free from deformation with no cracks

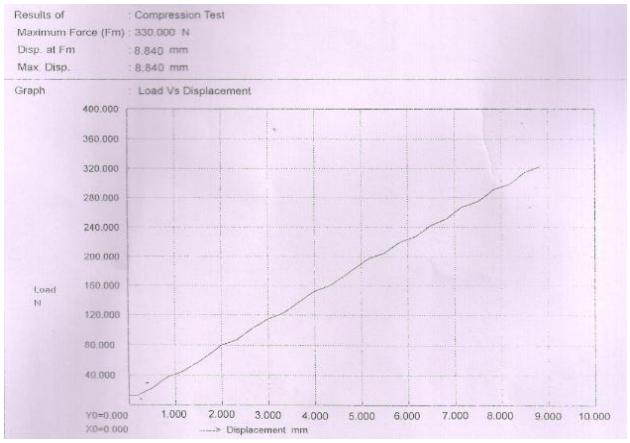

Fig.11 Graph of load Vs Displacement of Existing Bracket

The above graph shows load versus displacement values such that at 330 N load the displacement obtained is 8.840 mm.

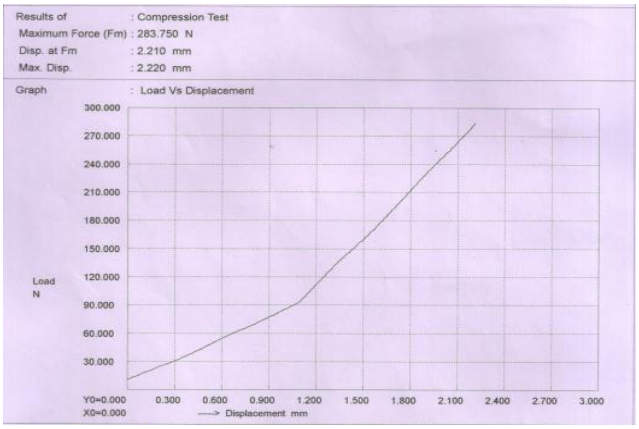

Fig.12 Graph of load Vs Displacement of Modified Bracket

The above graph shows load versus displacement values such that at 283 N load the displacement obtained is 2.210 mm.

7. RESULT AND DISCUSSION

In the static analysis generally two parameters are considered i.e. stress and displacement. The table shows the stress and displacement of existing bracket and modified bracket following

Table.1 Static Analysis results

Parameter	Existing Bracket	Modified Bracket
Stress (MPa)	236.9	111.2
Displacement (mm)	13.26	4.928

From experimental analysis, it is found that the displacement in the foot mounting bracket is

Table.2 Experimental results

Parameter	Existing bracket	Modified bracket
Load (N)	250	250
Displacement (mm)	6.820	2

From this testing it is observed that load applied to 250 N and found no crack and deformation.

By comparing the results of finite element analysis and experimental analysis, the displacement of existing bracket is 13.26mm and 6.82 mm respectively which close to each other while the displacement of modified bracket is 4.928 mm and 2 mm which close to each other.

8. CONCLUSION

The design had been modified from its preliminary stage. The addition of bead in the modified design increases the stiffness of the bracket. Static analysis is carried out for existing and modified bracket under same loading condition which shows modified bracket have less stress and displacement so design for modified bracket is safe.

From experimental testing, it is found that no crack or deformation observed for given loading condition and displacement for modified bracket is less than existing bracket so we can replace existing bracket by modified bracket.

ACKNOWLEDGEMENT

I would like to express my sense of gratitude to my guide, **Prof. Vikram S. Mahajan** for his valuable guidance at every stage of this project. This paper helped me to learn new things. I am really thankful to him.

REFERENCES

[1] Jasvir Singh Dhillon, Priyanka Rao, V.P. Sawant, "Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis" International Journal of Engineering Research and Applications, ISSN: 2248- 9622 Volume: 4, Issue: 9 September 2014

[2] M. V. Srinivasan, M. Lava Kumar, "Modal and Analysis on Car Seat Mounting Bracket" International Journal Of computer Trends and Technology (IJCTT), Volume: 4 Issue: 8 August-2016

[3] Sanket Vinchurkar, Prof. P. M. Khanwalkar, "A review on Optimization of Engine Mounting Bracket" International Journal of Engineering Trends and Technology (IJETT), Volume: 35, Number I –May-2016

[4] A P Khode, K Senthilkumar, Bhushan S Patil. Nagaraj Kulkarni, M W Trikande, "Shape Optimization and Weight Reduction of seat structure for wheeled Armoured amphibious Combat vehicle", Materials Today: Proceedings 4 (2017) 1917- 1926.

[5] P. Meghana, Y. Vijayakumar, Dr P. Ravinder Reddy, P.Seema Rani, "Analysis of Cabin Mounting Bracket of Truck Using ANSYS" International Journal of Engineering Science Invention, ISSN: 2319 – 6734, Volume: 5 Issue:11 November 2016 PP. 82-89

[6] K Sagar N. Narute, Prof. Dr. Santosh G.Taji, Prof. Prabhuling G. Sarasambi, "Design and Analysis of AC Mounting bracket using Composite material" International Journal for Science and advanced Research in Technology, ISSN: 2395-1052 Volume: 3 Issue: 9 September-2017

[7] Sandeep Maski, Yadavalli Basavaraj, "Finite Element Analysis of Engine Mounting Bracket by Considering Pretension Effect and Service Load" International Journal of Research in Engineering and Technology eISSN: 2319-1163 ISSN: 2321-7308