Design and Analysis of Foot Mounting Bracket using Finite Element Analysis

Mr. Suraj N. Mahadik[1] Prof. Vikram S. Mahajan[2]
[1]PG Scholar [2]Associate Professor
[1,2]Department of Mechanical Engineering
[1,2]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.

Key words: Foot Mounting Bracket, FEA, Static Analysis, Modal Analysis

I. INTRODUCTION

Driver seat is an inseparable part of an automobile. Its main function is to provide seating space to the driver and gives support, protect and to provide comfortable seating posture to the occupants. The seating system is made up of various components such as arm rest, head rest, back rest, tracks. These have their own natural frequency and in practice seat gets excited due to vehicle vibration like engine vibration and external road condition. Mounting bracket goes under various problems like design space issue, material used, weight of the bracket which affects the performance. The choice of a passenger automobile depends on factors such as vehicle type, brand, trend, security, its performance, interior pace, interior design, additional equipment offered etc. 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

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.

II. LITERATURE REVIEW

Jasvir Singh Dhillon et al. discuss 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]. A. P. Khode et al. studied eight different combinations for different size, shape and distance between structure members, 1st combination is manufactured and tested after that modal analysis, linear static FEA analysis has been done. Its results are examined, tested and after that aluminium alloy is used. Best optimized combination has been suggested with different thickness of aluminium alloy and 75% weight reduction has been obtained with comfort for user [4].

III. PROBLEM DEFINATION 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

In order above mensioned problems, main aim of the project is

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

IV. METHODOLOGY

The study of existing bracket is done for the modification in the bracket. The main aim of this study is to do static analysis of the bracket and to reduce stress and displacement for mounting bracket. The 3-D Dimensional model is designed for bracket. Different modifications have done in shape and design of the bracket. The static analysis is carried out in the Hypermesh. FEA analysis is used for finding out the results of existing bracket and modified bracket. The flowchart shows the step by step procedure to do the project.

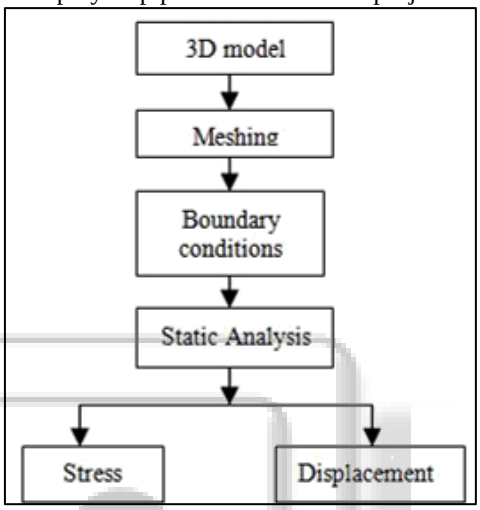

Fig. 2: Flow chart

A. Material Properties

Mechanical properties of D513 steel grade is
Yield strength = 220 MPa
Young's modulus, $E = 2.1 \times 10^5$ MPa
Poisson's ratio, $\mu = 0.3$
Density, $\rho = 7.8 \times 10^{-9}$ tonnes / mm^3

B. 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 single bracket
Dimensions of the existing bracket are

- Length of the existing bracket = 68 mm
- Height of the existing bracket = 114.5 mm
- Thickness of the bracket = 3 mm

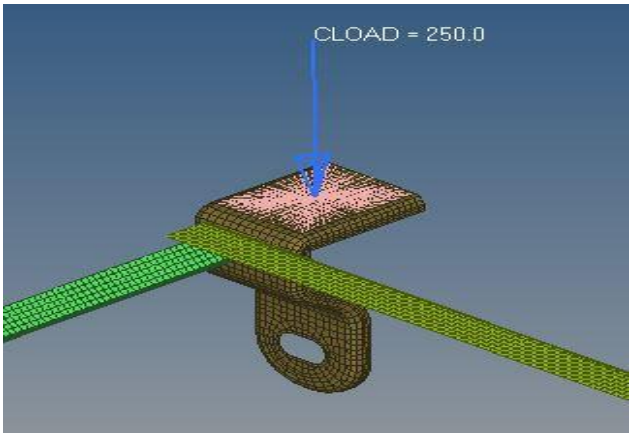

Fig. 3: Boundary Condition

C. Acceptance Criteria

The maximum von-mises stress under given loading condition is obtained from FEA results. If it is more than the yield strength of the material then the design is unsafe where von-mises stress is less than yield strength of the material then the design is safe.

V. 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 modelling. 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. In structural failure, to meet new condition FEA used to determine the design modifications. There are two types analysis used in industry that is 2D meshing and 3D meshing. 2D modelling keeps simplicity and allows the analysis runs on a normal computer while the 3D modelling requires fastest computer to run.

A. Static Analysis of Existing Foot mounting 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. Meshing is done in Hypermesh by considering the criteria given by R&D team.

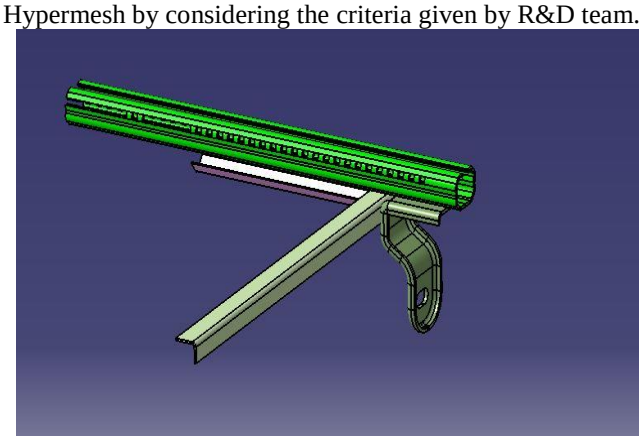

Fig. 4: Modelling of Existing Bracket

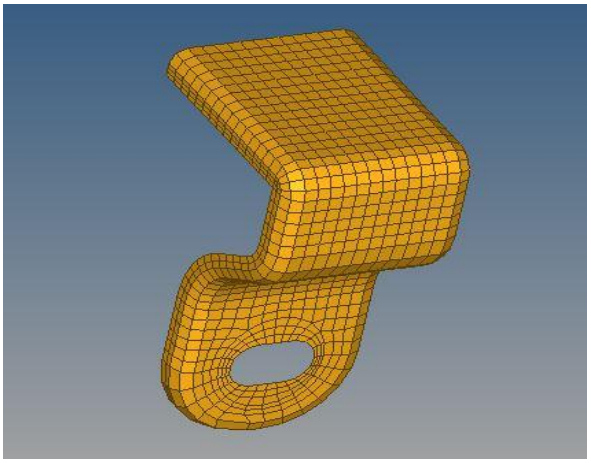

Fig. 5: Meshing of Existing Bracket

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

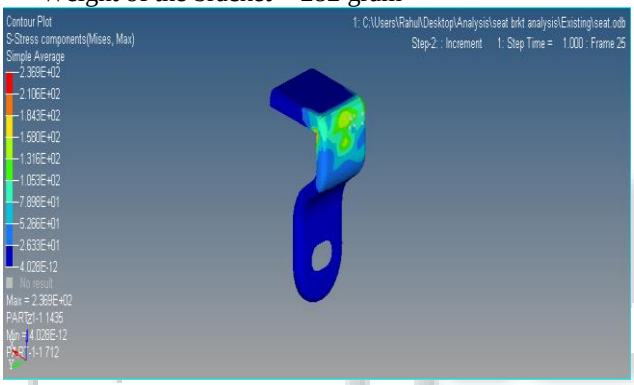

Fig. 6: Stress of Existing Bracket

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

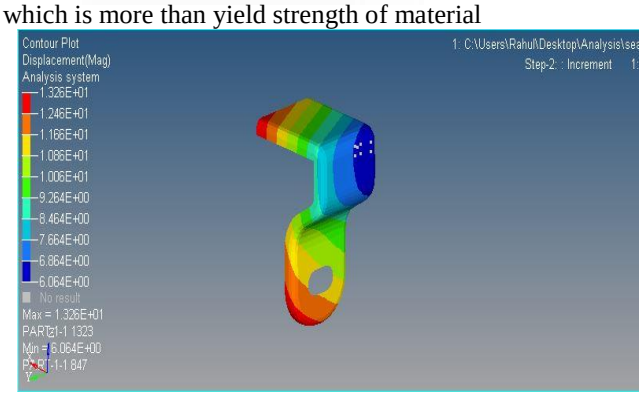

Fig. 7: Displacement of Existing Bracket

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

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

A. Static Analysis of Modified Bracket

It was decided to do 2D mesh for the CAD model. The modelling of the modified bracket is done in CatiaV5 and Hypermesh software is used for meshing and analysis. Meshing criteria for the modified bracket is given by R&D team.

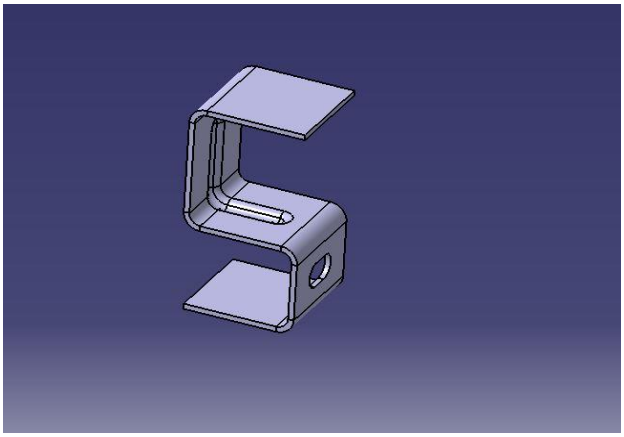

Fig. 8: Modelling of Modified Bracket

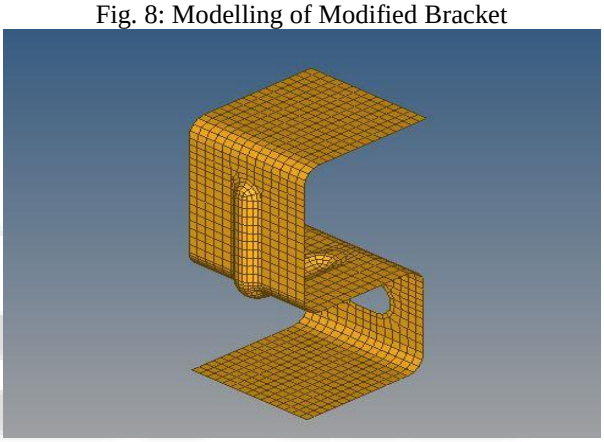

Fig. 9: Meshing of Modified Bracket

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

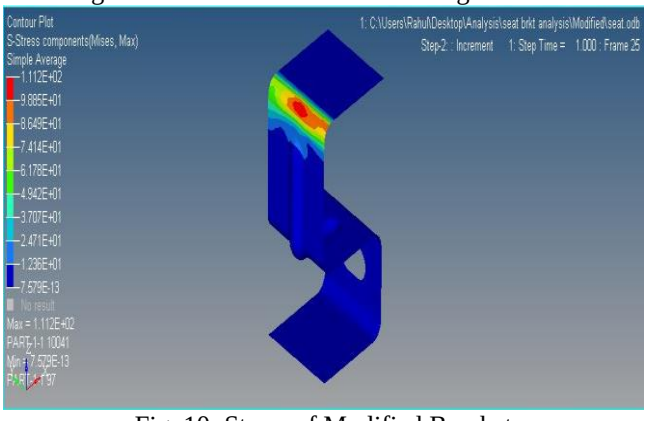

Fig. 10: Stress of Modified Bracket

Stress generated in the modified bracket is 111.2 MPa

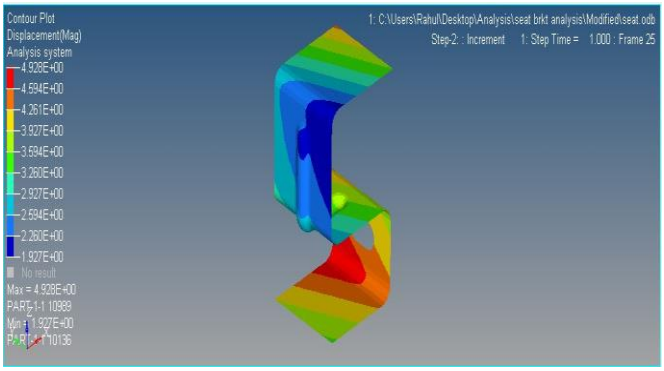

Fig. 11: Displacement of Modified Bracket

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

B. Modal Analysis

Modal analysis gives us the mode shapes corresponding to natural frequencies without considering any applied forces.

Mode shapes of Existing bracket are

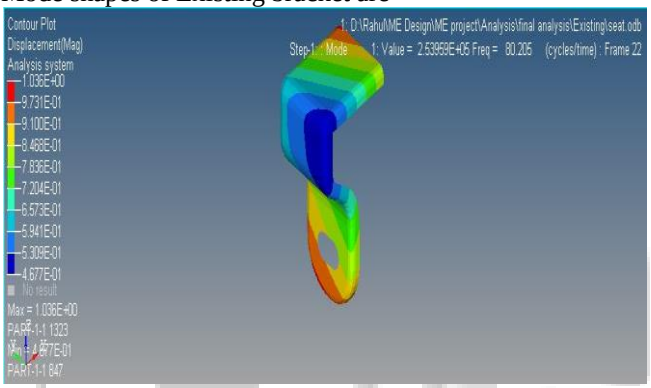

Fig. 12: 1st mode shape

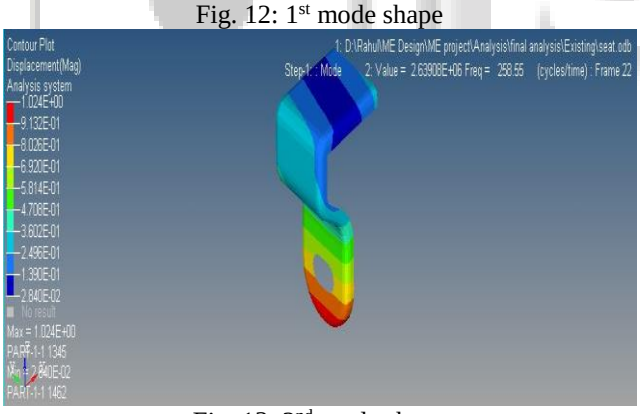

Fig. 13: 2nd mode shape

Mode shapes of Modified bracket are

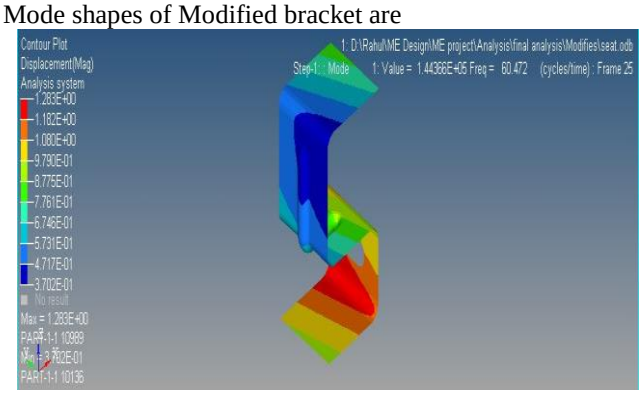

Fig. 14: 1st mode shape

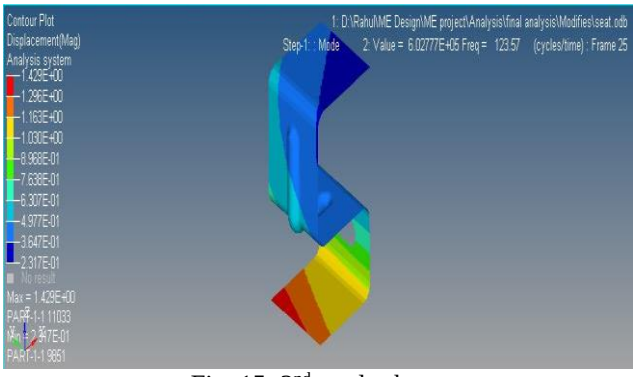

Fig. 15: 2nd mode shape

Mode Shape	Existing Bracket (Hz)	Modified Bracket (Hz)
1st	80.20	60.47
2nd	258.55	123.57
3rd	431.68	206.67
4th	602.24	321.99
5th	712.47	323.16
6th	750.48	443.16
7th	865.45	602.64
8th	1050.5	632.19
9th	1251.4	769.35
10th	1341.4	829.65

Table 1: Mode shapes of Foot Mounting Bracket

From the above table, first frequency is called as natural frequency.

VII. RESULT AND DISCUSSION

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

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

Table 2:

From the result table, it is seen that stress and displacement of modified bracket is 111.2 MPa and 4.928 mm respectively is below the yield strength of the material.

VIII. 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.

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] 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
[6] Rajeshkumar U. More, Dr. (Mrs.) R. S. Bindu, "Comfort Analysis of passenger Car Vehicle Seat" International Journal of Engineering science and Innovative Technology (IJESIT), ISSN: 2329-5967 Volume: 4 Issue: 4 July-2015
[7] Gaurav Bhusari, Prof. M. Sohail Pervez, "Design and Analysis of Fixture Bracket for Engine Assembly line" International Journal for Innovative Research in Science and Technology, ISSN: 2349-6010 Volume: 2 Issue: 05 October 2015

