International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

Field Problem of Cabin Mounting Bracket of Load King Pride

[1]M MEGHANA,[2]K VINOD REDDY

1 Pg Scholar,Department of Mechanical Engineering, elleniki college of enginnering and technology

2.Asst.Prof, Department of Mechanical Engineering, elleniki college of enginnering and technology

ABSTRACT

In an automobile industry while designing the components, the most critical aspect considered is the compactness and the weight of the component. According to the Newton's 2 law of motion the energy required to accelerate the vehicle depends upon the mass of the automobile. In the structural point of view the automotive materials should have more strength to weight ratio. One of the structural automotive vehicles and which are more in number are the mounting brackets. Mounting brackets are generally used support the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passengers. Since it is very much needed to reduce the weight and therefore it is to be redesigned or optimized for minimum weight without sacrificing the functionality. The most common material used for the structural components is structural steel. The bracket is designed according to the specifications of the mountings without considering any other factors. Analysis will be performed for Existing and New modified designs. The most important factors that are concentrated are stress distribution and deflection. The design structure is optimized for its topology. The reviewed design is analyzed and the better design for the minimum weight is considered, according to the optimized design parameters the structure is rebuilt. Modeling is the representation of the system or a part of the system in a physical or a mathematical model that

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

suitable for demonstrating behavior of the system. Simulation involves subjecting the models to various inputs or environmental conditions to observe the behavior and thus explore the nature of the results. That might be obtained from the real world system. Simulation is the manipulation of the model. It may involve mathematical model subjected to mathematical disturbance functions that simulate expected service conditions or it may involves software which are built on the basis of the finite element techniques like Ansys, Nastran, Hypermesh, LS-Dyna etc… loads and constraints are used as boundary conditions and required results are extracted from the Software tools. In this project, an attempt has been made to produce optimized design of a mounting bracket.

1.INTRODUCTION:

This bracket that's getting used within the vehicle weights regarding 1KG. Four mounting brackets area unit wont to support the complete cabin assembly. Total weight of the cabin assembly is 650 kilogram with four passengers. 2 brackets area unit placed at the front facet of the vehicle cabin and remaining 2 area unit placed at the rear facet. During this Project work static and dynamic analysis of the mounting bracket was carried out. Within the static analysis solely the load of the cabin with passenger's weight, vertical load, cornering load and acceleration load was taken. In dynamic analysis natural frequencies premeditated. Brackets area unit meant for carrying hundreds, supports structures, bearings that support the shaft.

1.1. STYLES OF BRACKETS

Wall Brackets

Wall brackets area unit mounted to The wall for the aim supporting bearings, which either

could also be solid with the brackets.

Pillar Brackets

These area unit wont to support a horizontal shaft from a pillar wherever there's no enclose

the method of wheels or pulleys on the shaft

Mounting Bracket

These area unit principally employed in automotive and part industries for supporting the structures like electronic product like batteries, sensors in automobile for fixing the body to

the chassis, for fixing the auxiliaries and for fixing the Cabin to the Chassis.

2.LITERATURE REVIEW:

Balamurugan [1] used shell elements and finite element simulation method with beam for modeling of military-tracked vehicle. An Eigen value analysis has been done to estimate natural modes of vibration of the vehicle. The dynamics response of such important location is obtained by carried out by a a transient dynamic analysis using implicit new marl beta method.

Curtis .F. Vail [2], this paper illustrates the use of F.E.Methods for modeling automotive structure for their dynamic characteristics. The results obtained using F.E.Computer models were within 10% of test results. Data reparation typical F.E.Model, outline of the analysis and the display of the data in the form of movie is covered.

Dr. Pawlowski [3] suggestions from his book Design Engineering and body construction has been used in this for designing the floor.

J.L.Hedges, C.C.Norville, O.Gurdogan[4], in these paper coarse and refined idealizations of the structure was analyzed by considering the effect of manufacturing tolerances. Stresses wee predicted under bending and torsion loads. Predictions follows the measured deflection curve but and sensitive to the idealization of the beam.

Karuppaiah [5] had done vibration analysis of a light passenger vehicle using a half-car rigid model and a finite element model. It has found that the results from the rigid body model are slightly in the lower side as compared to those from finite element model. Parametric study has also been carried out to study the effect of different parameters on vibration levels in the vehicle. The advantages and disadvantages of the non-contact method for road profile measurement have also been brought out.

Karuppaiah [6] applied for a passenger vehicle, finite element method for model and vibration/stresses analysis. The block lancozs method has been successfully used for the model analysis. The experiments were carried out using piezo-electric accelerometers and strain gauges to measure the vibration and strain gauges to measure the vibration and strain levels at critical points of vehicle. The prediction values through F.E.M were match well with experimental result. And also he has been

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

carried out with a view to find the optimum suspension/tyre characteristics for maximum ride comfort in the vehicle.

Kiyoshi Miki [7]. This paper presents the outline of a theoretical analysis of bending & torsional vibration of passenger car bodies. Body structure is simulated by a framework with tension rigs & additional panel stiffness's. The framework is a three-dimensional model for the bending and torsional vibration,or two-dimensional for the bending vibration, and is analyzed by the lumped mass system. All input data are calculated from drawings, and therefore characteristics of body structure are forecast &controlled in the design process.The analysis is applicable to coupled vibration & forced vibration problems.

Mukul Shukla [8], the work done is finite element analysis of vikram chassis frame under critical loads, simulating Indian load conditions. Modeling is done using deem, shell and rod and then analysis simple beam and rod elements are used, but analysis using 3 – D tetrahedral element is better and realistic approach as stated by the author. In his paper Mukul Shukla, in static analysis maximum stress is with in the specified limits and a factor of safety of 2-3 is obtained and the maximum stress occurs in the leaf spring joints and author says that present analysis is inadequate for leaf springs, as springs, as springs an more susceptible to fatigue failure than static and dynamic failure.

R.Ali, J.L.Hedges, B.Mills [9], in this paper Finite element techniques are applied to determine the static properties of and automobile body floor,

Robert .J. Melson[10], This paper reviews the success of efforts to predict linear static dynamic and non-linear transient behaviors of car components and structural systems. It relates the analysis accuracy to essential features of justify boarder use of numerical methods in the industry. Modeling from engineering drawings and coupon test data cannot be expected to yield behavior pre dictions with an error of less than five percent.

3.IMPLEMENTATION:

".In this Project work, It got to style the mounting bracket in such manner that it ought to face up to for road load conditions. each thickness and therefore the material area unit studied for this style. because the project deals with the look of mounting bracket, the prevailing style is analyzed and therefore the results area unit taken because

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

the reference for the current project work. the most downside is that, the mounting bracket thickness ought to be optimized for its minimum weight and even be taken care that stress ought to be below yield stresses.

The main aim of this project work is to form 3d model of the prevailing bracket as per second measurements and Hyper Mesh & Ansys has got to be performed on this model to watch stresses developed at bolt regions area unit like sensible testing.Similerly New style one & New style two has got to be created and meshing and analysis has got to be performed finally the optimisation got to performed to cut back the stresses at weak components within the mounting bracket and overall weight of the mounting bracket. this can be the goal of the project work.

Following area unit the image of the sensible mounting bracket failure underneath high stresses and geometric models of existing new style 1&2has been created.

- The field issues of mounting bracket image area unit shown within the chapter2. Refer Plate one.1, 1.2 and 1.3.

- The elaborated drawings of the prevailing and changed styles area unit shown within the following figures three.1, 3.2 and 3.3 severally.

- Geometric models of the prevailing and changed styles area unit shown within the following figures three.4, 3.5 and 3.6 severally.

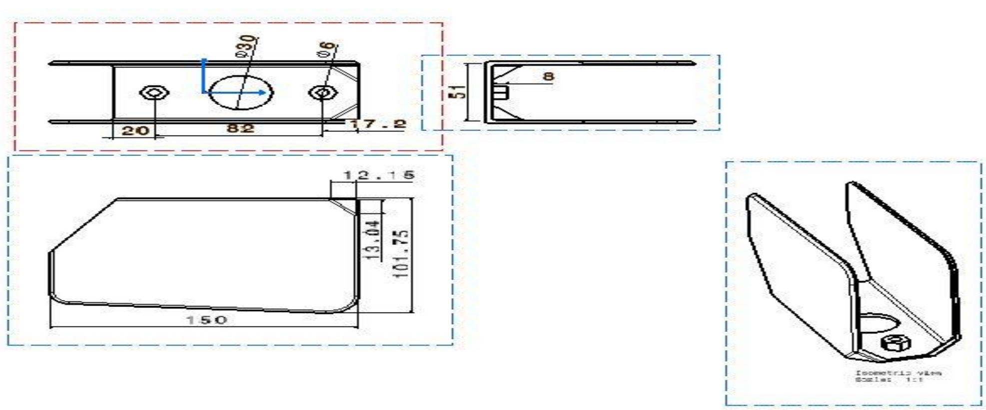

Fig 3.1 Existing Design of the mounting bracket (All Dimensions are in mm)

International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

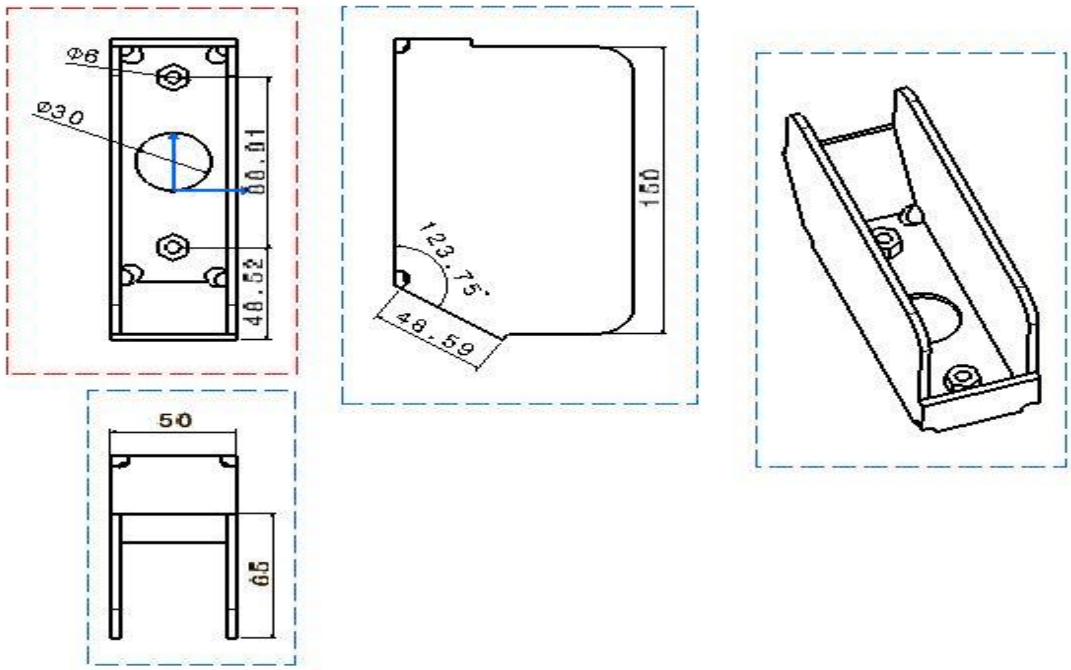

Fig 3.1 shows the detail description of the mounting bracket of the existing design.

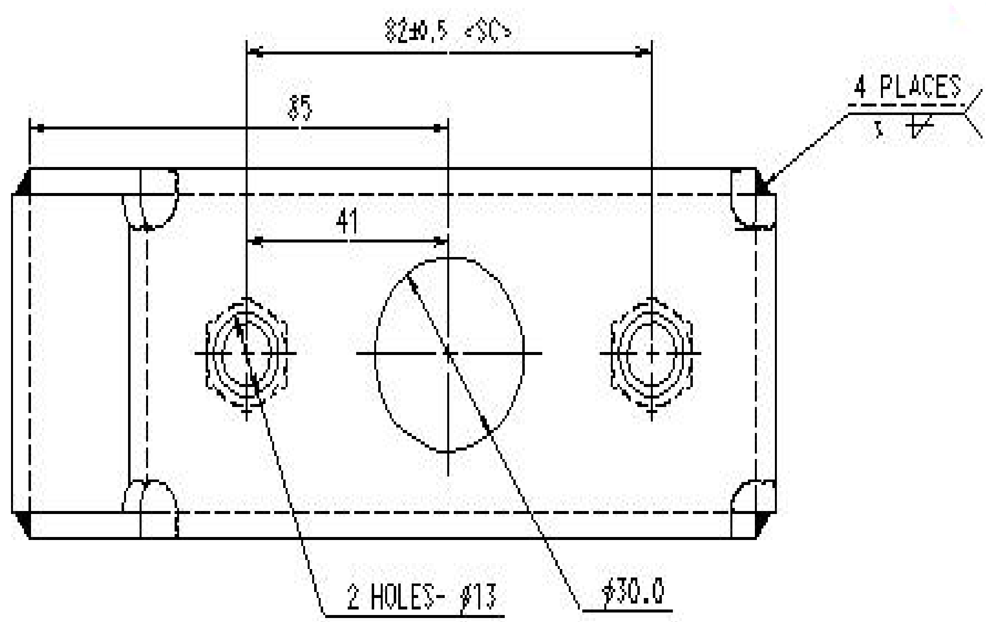

Fig 3.2 first modified design of the mounting bracket (All Dimensions are in mm)

Fig 3.2 shows the detail Description of the new design-mounting Bracket, the modifications are

1. Straighten the two legs of the bracket.
2. Adding the extra plates at front and rear faces of the mounting bracket.
3. Increased the thickness from 3.2 mm to 4 mm.
4. Weight is increased from 1 Kg to 1.2 Kg.

And the modifications has been done in the new drawing 2 are similar to above drawing 1 but additionally fillets were added at the corners of the bracket as shown in the following figure.

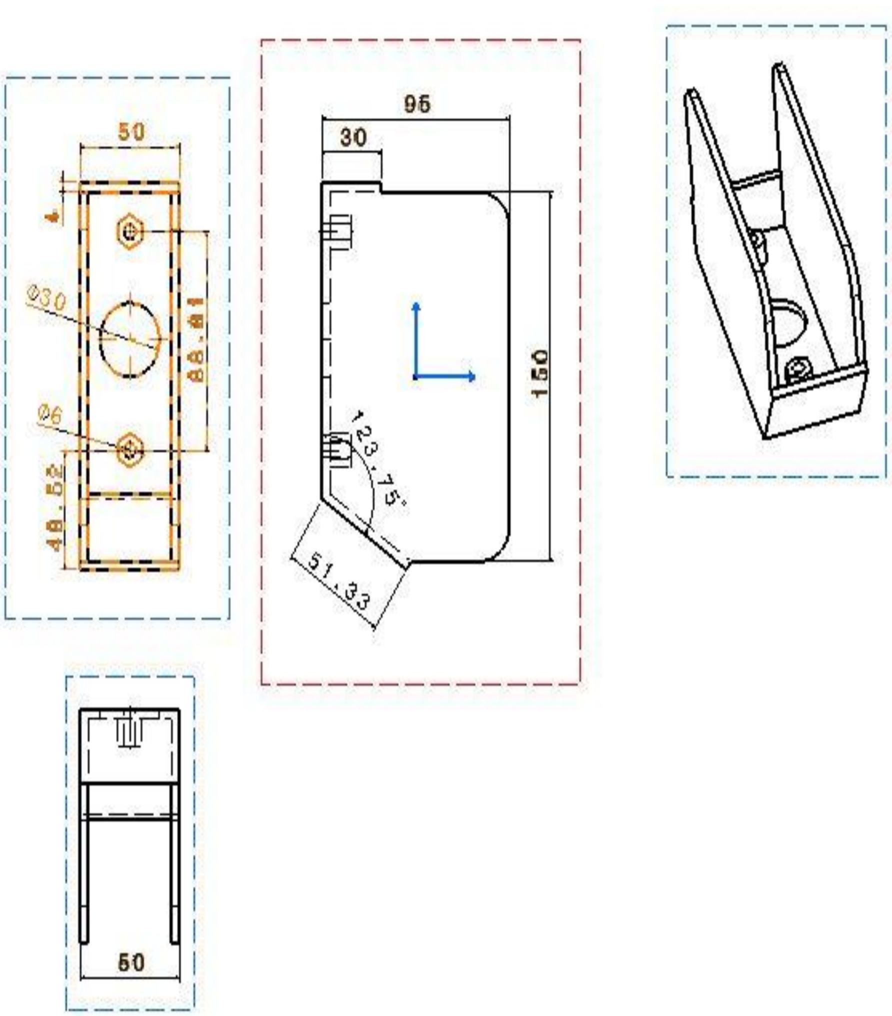

Fig 3.3 New designs (2) of the mounting bracket (All Dimensions are in mm)

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

Fig 3.3 shows the detail description of the new modified design of the mounting bracket.

4.RESULT:

The finite element modeling and analysis is used to study the stress variation at different points of the mounting bracket and also the deflection at various locations of the mounting bracket having various thicknesses with single material. Topology optimization is also carried out to extract the best design and to reduce the weight of the mounting bracket.

Two parameters studied in this Project work are one is the thickness and the other one is the weight of the mounting bracket to rectify the field problem.

4.1 STATIC ANALYSIS RESULTS

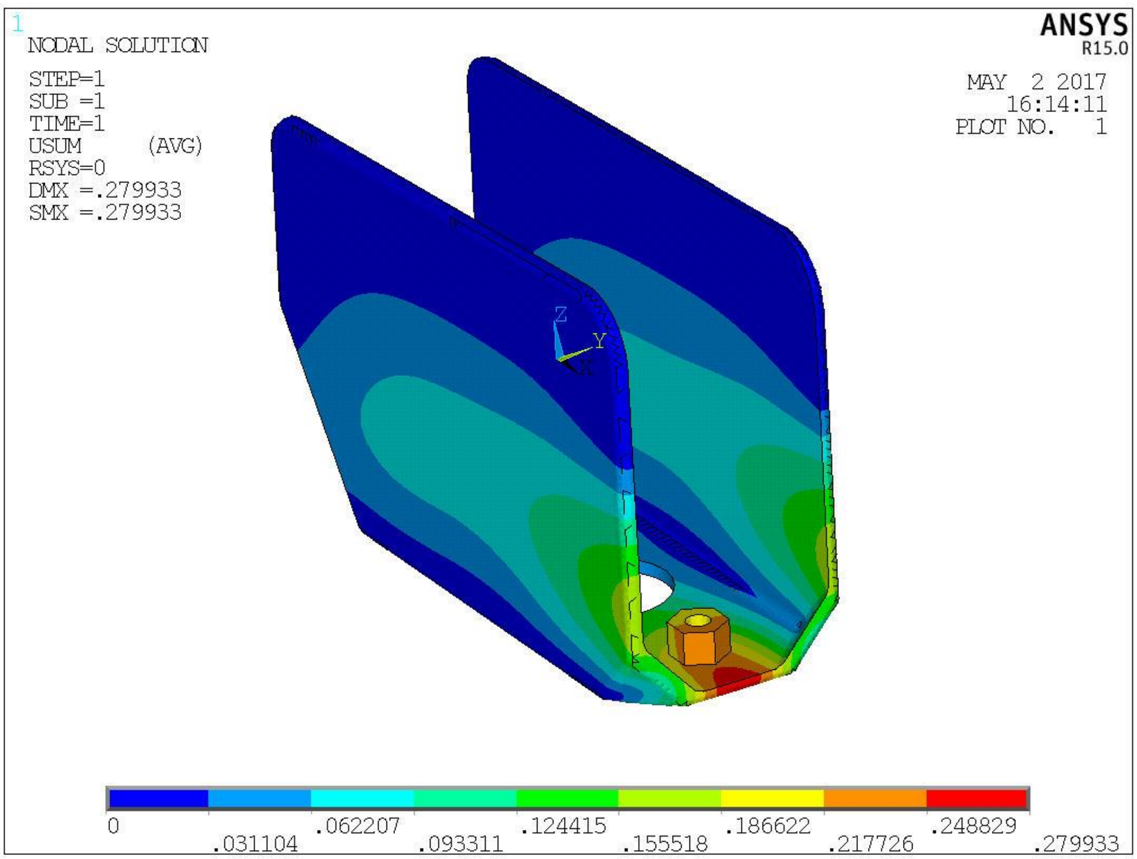

Fig.4.1. Deformation induced in the Mounting Bracket of Existed design made of STEEL-HR-1DD with a load of 995 N on each bolt is 0.27993 mm. From the **figure 5.1** shows the variation of

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 04 Issue 13
October 2017

deformation in the Mounting Bracket due to with a load of 750 N on each bolt is applied, the maximum deformation is observed is 0.27993 mm for the STEEL-HR-1DD material.

5.THE CONCLUSION THAT ARE DRAWN FROM THE PRESENT WORK ARE

- From the analysis the bracket with that of 3.2 mm thickness weighing 1 Kg and 4 mm thick bracket weighing 1.2 Kg and the final optimized bracket with 4 mm thick weighing 1 Kg. The more strength is gained for 4 mm thick bracket after its optimization.
- The induced deflections of different thickness in all the brackets are very small and are less than 0.1 mm for all the cases and hence these are all rigid.
- The stresses induced are well under the allowable stresses and maintaining high factor of safety.
- The thickness of the optimized bracket 4mm is reduced from 1.2 Kg to 1 Kg.
- Cost of the Mounting bracket remains same when the optimized design compared with Existing Design. And the safety of optimized design is increased.
- Finally the weight of the both the Optimized design and Existing Design is same but the deformation and stresses were better than the Modified Design as well as Existing Design and therefore the Optimized design should be implemented.
- From the dynamic point of view the amplitudes were well within its limits, Hence the design is safe for the dynamic stability. Compared to other brackets, brackets that are made of CFRC material had superior perforamance.

FUTURE SCOPE

It is suggested to run the fatigue analysis on the modified design to further Optimize the shape and size.

REFERENCES:

1. Alf Samuelsson and Nils-Erik Wiberg: Finite element adaptively in dynamics and elasto-plasticity". Volume "The finite element method in the 1990's, Springer-Verlag, 1 pp. (1991).

2. C.C. Menzemer, L. Fei, T.S. Srivatsan, Design criteria for bolted

connection elements in Aluminum Alloy 6061, Transactions of ASME, Vol.121, Sep 1991, p348-357.

3. Fernandes, Cmtt, Leitao, Vma, Vasarhelyi, a., Thin Plate Bending Analysis Using An Indirect Trafftz Collocation Method, computer Assisted Mechanics and Engineering Sciences, 8, 1-16, 2001.

4. Hao Jin, Kjell Mattiasson, Kenneth Runesson and Alf Samuelsson: The boundary element method for elasto-plastic large deformation problems, International Journal of Numerical Methods in Engineering, vol. 25, pp. 165-175, (1987).

5. J. Newnham, L. Curley, Bolted joints under tensile loads: The effect of torsion in the Bolt on in-service clamping loads, Transactions of ASME, vol.15, March 1993, p36-40.

6. Kjell Mattiasson, Lars Bernspong and Alf Samuelsson: Solution of quasi-static, force-driven problems by means of a dynamic-explicit approach and an adaptive loading procedure, Engineering Computations, vol.13, pp. 172-189, (1996).

7. Mats Ander and Alf Samuelsson: Finite Element analysis of Geometrically Non-Linear Structures Using Translational Variables, International Journal for Numerical Methods in Engineering, Vol. 46, pp.1367-1383, (November 1999).

8. Magnus Kjellman and Alf Samuelsson: A high-precision adaptive procedure for solving Kirchhoff plates, in Discretization Methods in Structural Dynamics, (Eds. Kuhh, G. and Mang H.), Springer, pp. 125-133, (1990).

9. Nina Lauterstajn-S and Alf Samuelsson: Further discussion on four-node isoperimetric quadrilateral elements in plane bending, International Journal for Numerical Methods in Engineering, vol. 47, pp. 129-140, (2000).

10. Ansys and Hyper mesh user manuals.