International Journal of Engineering Science Invention
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 5 Issue 11|| November 2016 || PP. 82-89

Analysis of Cabin Mounting Bracket of Truck Using ANSYS

P. Meghana [1], Y. Vijayakumar[2], Dr P. Ravinder Reddy[3], P.Seema Rani[4]
[1,2,4]Departmentof Mechanical Engineering, J B Institute Of Engineering &Technology, Hyderabad, Telangana State, India [3]Departmentof Mechanical Engineering, Chaitanya Bharathi Institute Of Technology, Hyderabad, Telangana State, India

Abstract : *In an automobile industry while designing the components, the most critical aspect considered is the compactness and the weight of the component. The mounting brackets are meant for supporting the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passenger's weight. In the initial stage the bracket is designed according to the specifications of the mountings without considering any other factors. Analysis is performed for Existing and New modified designs. The design structure is optimized for its topology and topography. In the present work an attempt has been made to produce optimized design of a mounting bracket. The modeling is carried out in CATIA and meshing with quality is ensured through Hyper Mesh. The analysis is carried out using ANSYS by the objective function as shape and topology and the weighting function as weight, and constraints are on deflection and stresses induced.*

Keywords : *Catiav5 R20, Bracket, Static Loads, Dynamic Loads, Optimization*

I. Introduction

The present bracket that is being used in the vehicle weights about 1KG. Four mounting brackets are used to support the entire cabin assembly. Total weight of the cabin assembly is 510 kg including three passengers. Two brackets are positioned at the front side of the cabin and remaining two are positioned at the rear side. In this Project work static and dynamic analysis of the bracket was carried out. In the static analysis only the weight of the cabin including passenger's weight was considered. In dynamic analysis vertical load, cornering load and acceleration load with some time period were considered. Balamurugan [1] used finite element simulation method with beam and shell elements 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 certain salient location is obtained by carrying out 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 movie form is covered. Dr. Pawlowski [3] suggestions from his book Body Construction and Design Engineering has been used in this paper 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] has 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 finite element method for model and vibration/stresses analysis of a passenger vehicle. 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 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 and torsional vibration of passenger car bodies. Body structure is simulated by a framework with tension rigs and 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 and controlled in the design process.

The analysis is also applicable to coupled vibration and 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.

II. Objectives And Goals

In this work, It need to design the mounting bracket in such way that it should withstand for road load conditions. Both thickness and the material are studied for this design. As the work deals with the design of mounting bracket, the Existing Design is analyzed and the results are taken as the reference for the present work. The main problem is that, the mounting bracket thickness should be optimized for its minimum weight and also be taken care that stress should be below yield stresses. The main aim of this work is to create 3d model of the existing bracket as per 2d measurements and Hyper Mesh & Ansys is performed on this model to observe stresses developed at bolt regions are similar to practical testing. Similarly New Design 1 & New Design 2 has been created and meshing and analysis is performed, finally the optimization is carried out to reduce the stresses at weak parts in the mounting bracket and overall weight of the mounting bracket. Following are the picture of the practical mounting bracket failure under high stresses and geometric models of existing new design 1&2has been created. The detailed drawings of the existing and modified designs are shown in the Figures 1, 2 and 3 respectively. Geometric models of the existing and modified designs are shown in the Figure 4.

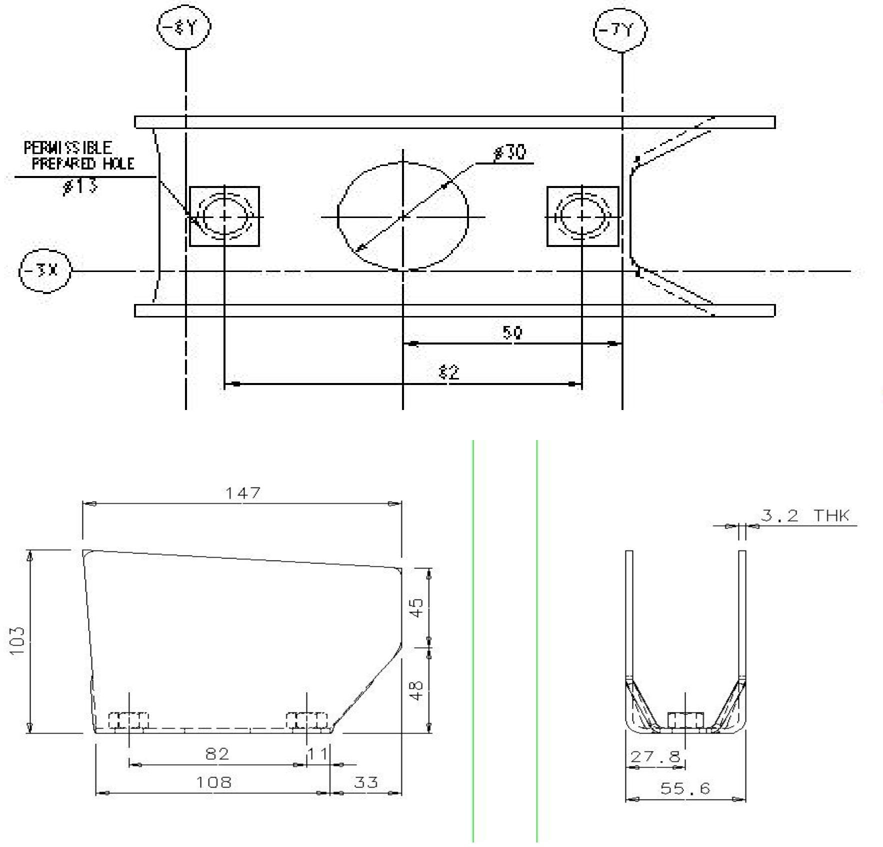


Fig.1: Existing Design of the mounting bracket (All Dimensions are in mm)

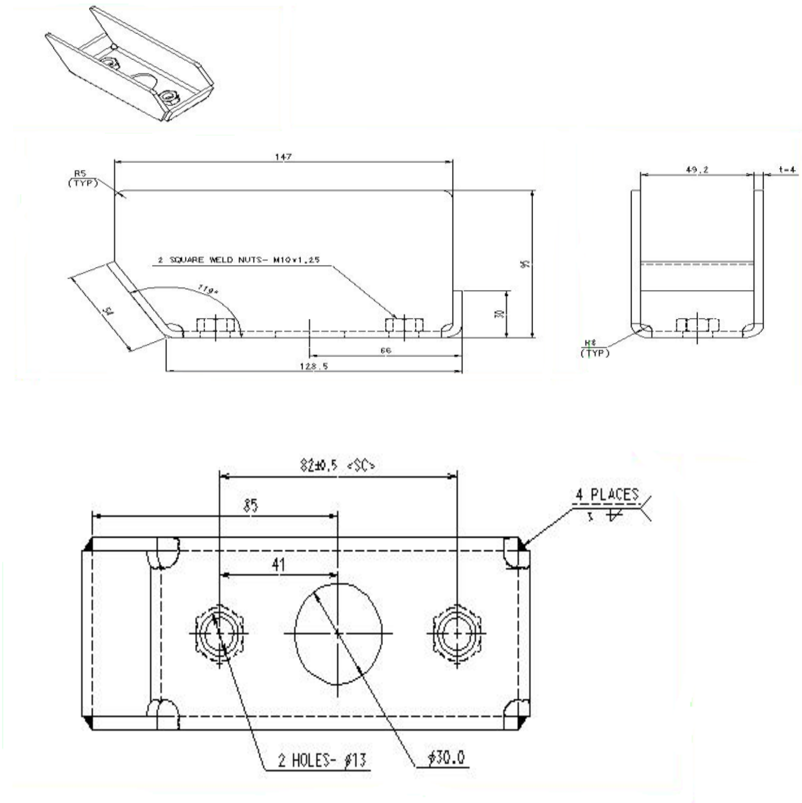


Fig.2: First modified design of the mounting bracket (All Dimensions are in mm)

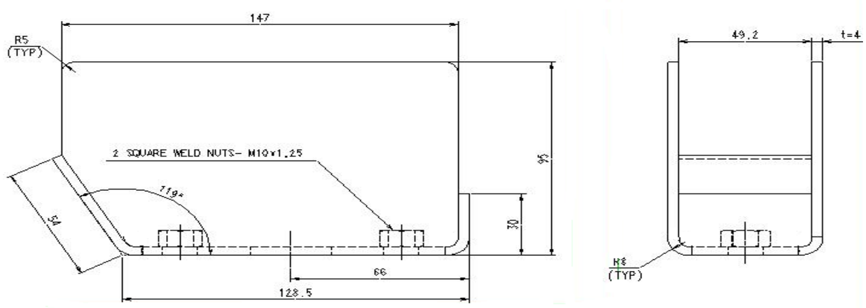
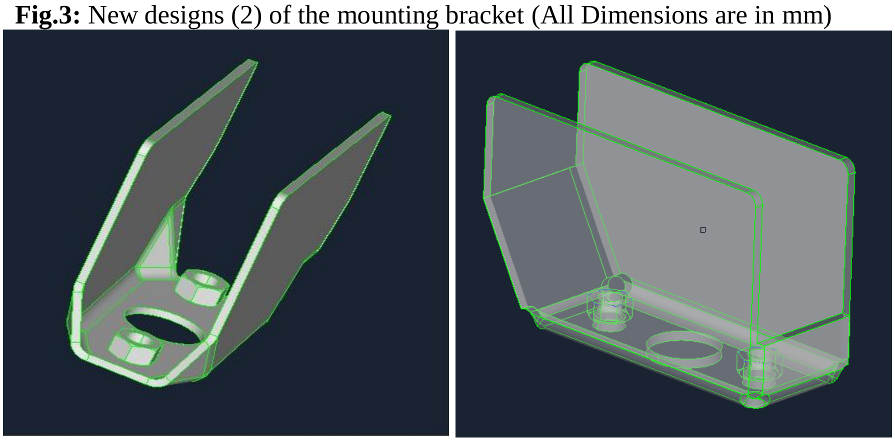
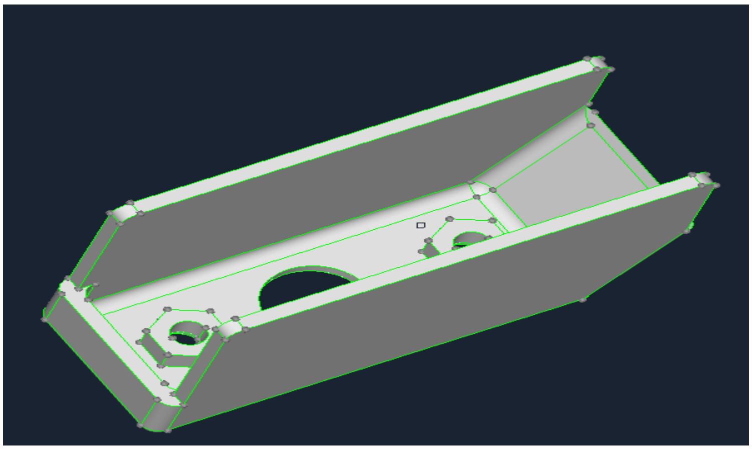


Fig.3: New designs (2) of the mounting bracket (All Dimensions are in mm)



a. Existing design **b. New design1**



c. New Design2

Fig.4: Geometric model of the existing and new designs of mounting bracket

Finite Element Modeling Of Gyroscope

The model created in CATIA software is imported through the IGES file in to Hypermesh software then geometry clean up is carried out. Using the ANSYS library of elements the cards are prepared. The element selected for meshing the bracket is a 8 nodded 3D sold element which is having 3dof/node and 24dof/element, the element has shown in Figure5. After checking the convergence norms the meshed model is shown in Figure6.

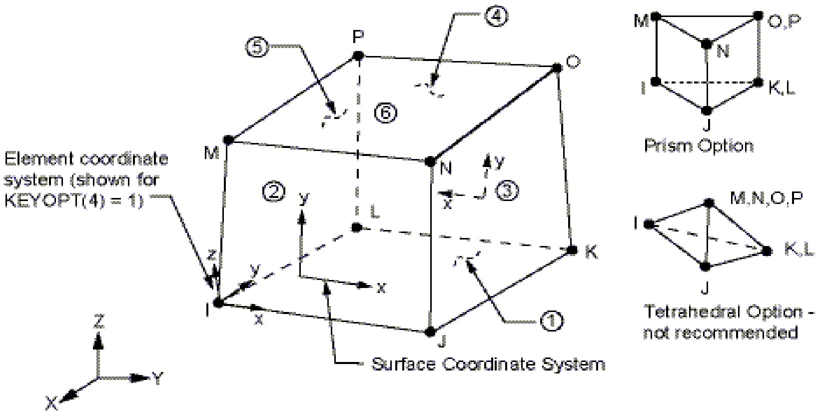


Fig.5: SOLID45 Geometry

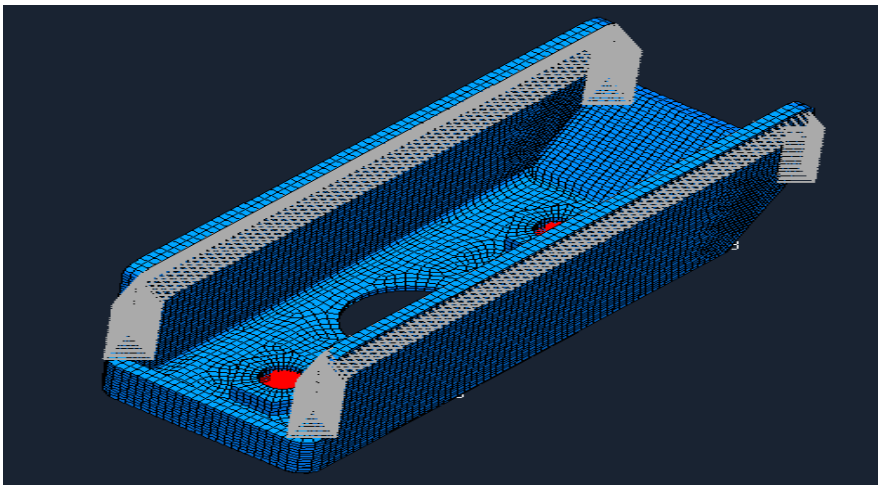


Fig.6: Meshed Model of Mounting Bracket of Cabin

Material Properties:

The materials selected for analysis are steel and CFRP Composites. The properties of these materials are given as here under. Mechanical Properties of Steel: Young's modulus: 210GPa, Poisson's ratio: 0.30 and desnsity=7850 kg/m^3 . Mechanical Properties of CFRP Composites are Young's modulus =E_x = 180GPa, E_y = l0Gpa =E_z , Poisson's ratio (nu_{xy}) = 0.28 , Shear modulus =G_{xy} = 7. 1 GPa, Mass density =1600 kg/m^3 , Damping co-efficient = 0.018.

III. Load Calculations

5.1 Static Load Calculations: The static load is calculated by using the weight of the cabin is 285 kgf and Weight of Three passengers (Each Passenger weight 75kg)= 3x75 =225 kgf, Total weight acting on the four mounting brackets= 285+225 = 510 kgf. When the vehicle is on the rest position 40 % of the total weight is going to act on the two front mounting brackets and 60% of the total weight is going to act on the rear two mounting brackets. Therefore 60% of the total weight on each bracket = 153 kgf. Total force acting on the each bracket= 153x9.8 = 1500 N. This load is acting along the vertical Y direction. Each bracket consists of two bolts. Total load is going to act on these two bolts. So each bolt takes half amount of the total load. Constraints were considered at the welding region.

5.2 Dynamic Load Calculations: The transient dynamic loads are extracted from the PAVE test. PAVE is the practical automotive vehicle-engineering test where vehicle is traveled on the worst road conditions. In this test, by using sensors it is measured the total weight which is going to act on the mounting bracket at different times. The dynamic load data measured in the PAVE test is shown in the Figure 7.

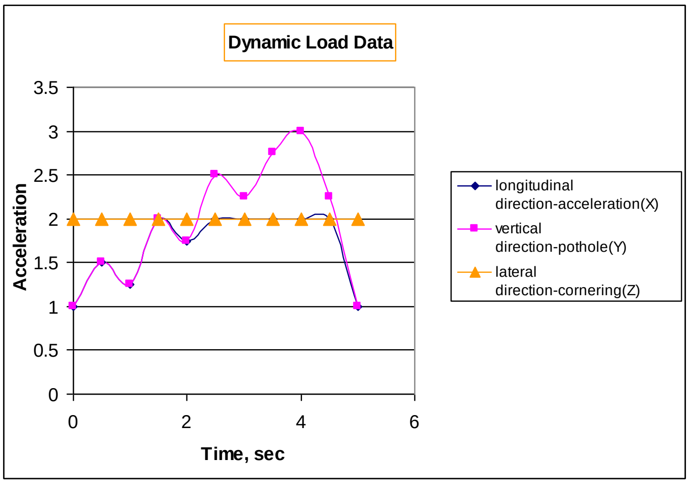


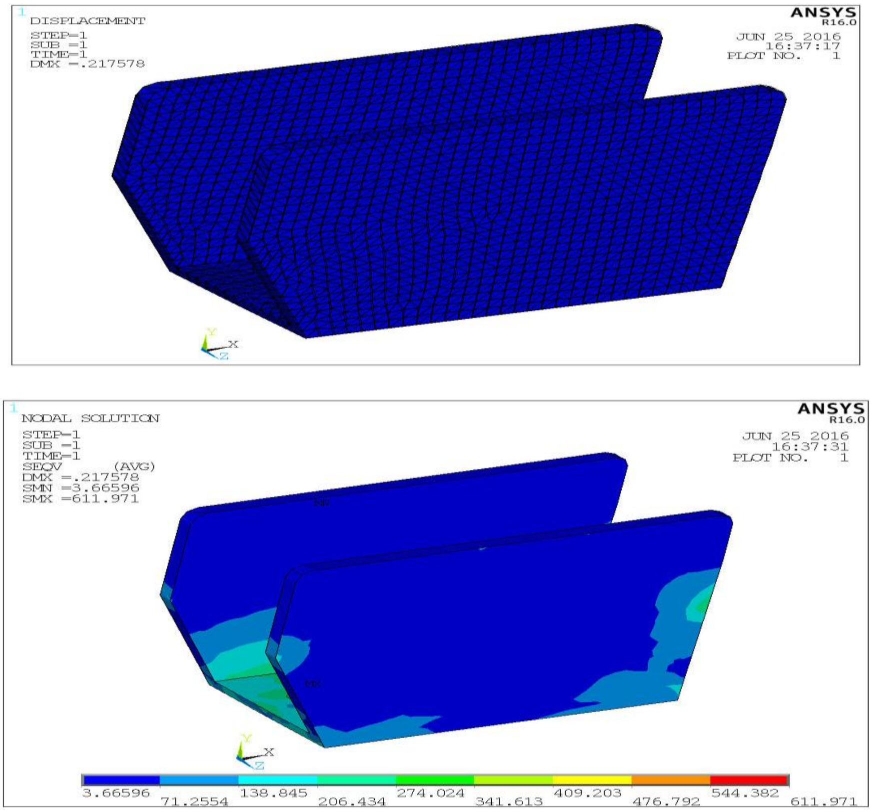
Fig.7: The dynamic load curve

Steps in Analysis:

As an input to the problem density, modulus of elasticity and poisons ratio is given. Creation of the Finite Element Model is carried out in the solution. Load and displacements are applied, at the area of welding are constrained in all degrees of freedom. Vehicle load of 60 percent in 510 kg is applied at the two bolts. In post processor the required results are obtained as Resultant deformations, Stress distribution, von-misses stresses and Deformation in static and dynamic load direction. The finite element modeling and analysis is used to study the stress variation at different locations 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.

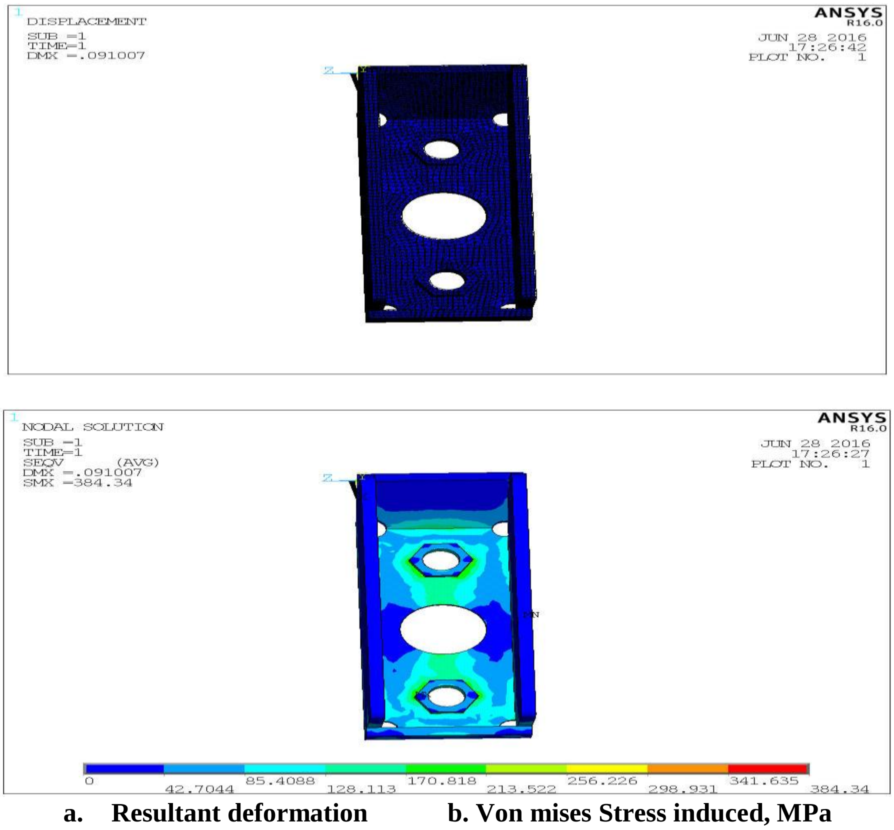
IV. Results and Discussions

From the Figure 8a shows the variation of deformation in the Mounting Bracket due to a load of 750 N on each bolt is applied, the maximum deformation is observed is 0.217578 mm for the STEEL-HR-1DD material. The Vonmises stress induced in the Mounting Bracket made of STEEL-HR-1DD with a due to a with a load of 750 N on each bolt is 611.971 MPa is shown in Figure8b.



a. Resultant deformation b. Von mises Stress induced, MPa

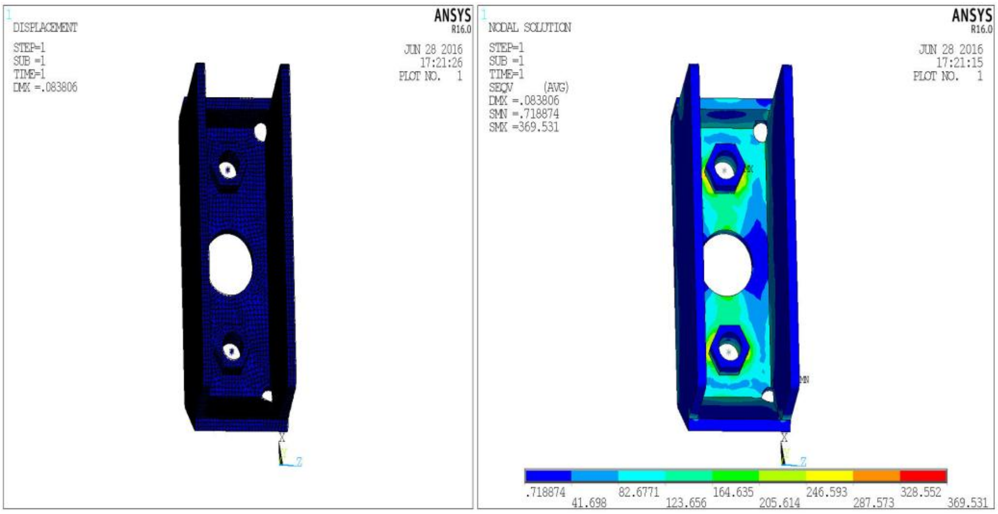
Fig8. The resultant deformation and vonmises stress induced in the cabin bracket



a. Resultant deformation b. Von mises Stress induced, MPa

Fig.9: The resultant deformation and Vonmises stress induced in the Mounting Bracket of design 1 made of STEEL-HR-1DD due to a with a load of 750 N on each bolt

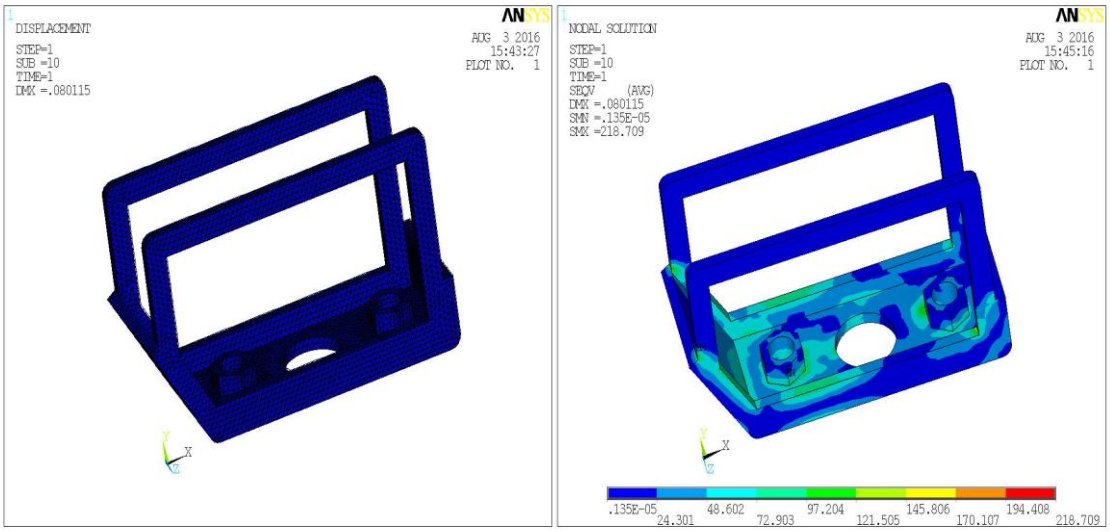
Figure9a shows the deformation induced in the mounting bracket of design 1 made of STEEL-HR-1DD with a load of 750 N on each bolt is 0.091007 mm. Figure 9b shows the Vonmises stress induced in the Mounting Bracket of design 1 made of STEEL-HR-1DD with a due to a with a load of 750 N on each bolt is 384.34 MPa.



a. Resultant deformation **b. Von mises Stress induced, MPa**

Fig.10: The resultant deformation and Vonmises stress induced in the Mounting Bracket of design made of STEEL-HR-1DD due to a with a load of 750 N on each bolt

Figure10a shows the Deformation induced in the Mounting Bracket of design 2 made of STEEL-HR-1DD with a load of 750 N on each bolt is 0.083806 mm. The Vonmises stress induced in the Mounting Bracket of design 2 made of STEEL-HR-1DD with a due to a with a load of 750 N on each bolt is 369.531 MPa is shown in Figure10b.



a. Resultant deformation **b. Von mises Stress induced, MPa**

Fig.11: The resultant deformation and Vonmises stress induced in the Mounting Bracket of optimized model made of STEEL-HR-1DD due to a with a load of 750 N on each bolt

Figure11a shows the Deformation induced in the Mounting Bracket of optimized model made of STEEL-HR-1DD with a load of 750 N on each bolt is 0.080115 mm. The Vonmises stress induced in the Mounting Bracket of optimized model made of STEEL-HR-1DD with a due to a with a load of 750 N on each bolt is 218.709 MPa is shown in Figure11b. Results of Existing design, new design (1), new design (2) and Optimized Design are shown in Table 1.

Table1: Deformations and stresses induced in the existing and modified brackets with different thickness

S.No.	Design	Deformation, mm	Von-mises stress, MPa	Thickness, mm	Weight, Kg
1	Existing Design	0.2175	611.971	3.2	1
2	New design (1)	0.091007	384.34	4	1.2
3	New design (2)	0.083806	369.531	4	1.2
4	Optimized Design	0.080115	218.709	4	1

From the table1 it is observed that the deformation in optimized design is much smaller than the existing design and stresses induced are 1/3rd that of existing design and the weight is maintained same. From Table 2 it is observed that the natural frequencies of each material and the natural frequencies are very high for CFRC in all the modes.

Table2: Natural Frequencies of mounting bracket with different materials (HZ)

	Natural Frequencies of mounting bracket with different materials (HZ)		
	Mild steel	CFRC	GFRC
First mode	59.609	984.106	205.507
Second mode	69.074	1077	211.554
Third mode	71.534	1386	286.85
Fourth mode	3610	1866	374.251
Fifth mode	4181	2309	474.744
Sixth mode	4316	2457	535.404

V. Conclusions

From the analysis the bracket with that of 3.2 mm thickness weighing 1 Kg and 4 mm thickness bracket weighing 1.2 Kg and the final optimized bracket with 4 mm thickness weighing 1 Kg. The more strength is achieved for 4 mm thick bracket after optimization. The induced deflections in all the brackets of different thickness 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 below the allowable stresses and maintaining high factor of safety. Weight of optimized bracket thickness 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. However the safety of optimized design is increased. 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 hence the Optimized design should be used in practice. From the dynamic point of view the amplitudes are well within limits, hence the design is safe for the dynamic stability. The bracket made of CFRC material is superior performance as compared other brackets.

References

[1]. Alf Samuelsson, Some aspects on non-conforming plate bending elements, computer and Structure, 19(1-2) ,1984, pp.193-197

[2]. Alf Samuelsson and Nils-Erik Wiberg, Finite element adaptively in dynamics and elasto-plasticity, 1(2), The finite element method in the 1990's, Springer-Verlag, 1991, pp.21-27

[3]. C.C. Menzemer, L. Fei, T.S. Srivatsan, Design criteria for bolted connection elements in Aluminum Alloy 6061, Transactions of ASME, 121, Sep 1991, p348-357.

[4]. Fernandes, Cmtt, Leitao, Vma, Vasarhelyi, a., Thin Plate Bending Analysis Using An Indirect Trafftz Collocation Method, Computer Assisted Mechanics and Engineering Sciences, 8(2), 2001, pp.1-16

[5]. Hao Jin, Kjell Mattiasson, Kenneth Runesson and Alf Samuelsson, On the use of the boundary element method for elasto-plastic large deformation problems, International Journal of Numerical Methods in Engineering, 25, 1987, pp. 165-175

[6]. J. Newnham, L. Curley, Bolted joints under tensile loads:The effect of torsion in the Bolt on in-service clamping loads, Transactions of ASME,15, March 1993, pp.36-40

[7]. 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,13, 1996, pp. 172-189

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

[9]. 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, 1990, pp. 125-133

[10]. Nina Lauterstajn-S and Alf Samuelsson, Further discussion on four-node isoperimetric quadrilateral elements in plane bending, International Journal for Numerical Methods in Engineering, 47, 2000, pp. 129-140,

[11]. P. Ravinder Reddy and J. Pradeep Kumar,Vibration Response of Composite Plate with Delamination using FEA, International Journal of Applied Engineering Research, 2295) , 2010,3633–3643.

[12]. S. Khadar Vali, Dr P Ravinder Reddy, Dr P Ram Reddy, Impact Testing of Advanced Ceramic Composites Using Experimental & Numerical Analysis, International Journal of Technology Spectrum, Research Communication, 2(1), 2008, 22-25

[13]. S. Khadar Vali., Dr P. Ravinder Reddy, Dr P. Ram Reddy, Modeling and Analysis of Three Dimensional Carbon Fiber/SiC Composites prepared by Liquid Silicon Infiltration Using FEA, Global Journal of Engineering and Applied Sciences (GJEAS),3(1), 2011,133-139.

[14]. Ansys and Hyper mesh user manuals.