iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 7 **Issue:** IX **Month of publication:** September 2019

DOI: http://doi.org/10.22214/ijraset.2019.9098

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

Design of HCV Super Bracket by Topology Optimization Technique for Weight Reduction and Strength Enhancement

Rajani R. Mhatre[1], R. B. Gunale[2]
[1]Student of ME (Design), [2]Assistant Professor, Mechanical Engineering Department, JSPM'S ICOER Wagholi

Abstract: A commercial vehicle is any type of motor vehicle used for transporting goods or paying passengers. Body and Chassis are the two major assemblies in any vehicle either car or truck. As competition is growing rapidly in automotive market it is important to reduce the time required for production and cost of manufacturing process. Nowadays Automobile industry has become one of the biggest domain of mechanical engineering industries. Automobile vehicle requires lot of fuel while transporting people goods from one to another place. Heavy commercial vehicle consumes more fuel and hence it has become necessary to reduce the weight of vehicle for optimum consumption of fuel and thus it is one of the biggest challenge in automobile industries. However overall weight of the HCV trucks is not acting on a single component, rather it is acting on all the components which are being part of automobile system. In consideration of same we have chosen three basic components acted upon by heavy loading effects. Those components can be named as, front cabin mounting bracket, suspension mounting bracket and towing pin. In these paper we have integrated all these three component into single component with an aim to reduce weight, material saving, strength enhancement and improved operational efficiency. Further topology optimization techniques have been used to remove surplus material and thus to stimulate component geometry integration to serve above objective. Finally weight of individual basic components to final optimize integrated model reduces from 12.3 kg to 9.42 which is 24% reduction.

Index Terms: HCV, CAD, FEA, Ansys, Topology Optimization etc.

I. INTRODUCTION

A vehicle used for transportation purpose of goods and passengers which is self-propelled is known as automobile . Body and Chassis are the two major assemblies of which both trucks and passenger cars are generally made up of. Chassis is the component of vehicle on which total load of vehicle is act. Chassis and the main body these are the two main parts which helps to complete automobile vehicle. [5] For smooth performance of their functions the whole truck consists of various assemblies. Though there are so many various essential parts, the place where driver and co-driver seat is cabin. Their weights mainly act on the cabin floor where it have to withstand many other loads which are coming from other ways in various directions. Due to this the driver remains seated without any distractions and vibrations. Floor panel sheet made up of thin material is flexible for loads which are out of plane. The function of the floor is to receive all the loads from the point of application to the main components of the automobile vehicle, like side frames. Floors are subjected to the loads which are normal to their planes. So under these circumstances floors do not behave as simple structural surfaces. The floor usually made stiff for loads which are out of plane by arranging beam members in planar framework. Advantage of tilting the cabin over rigid cabin becomes easy for servicing, less weight, easy for design modification and less vibrations. Growing competition in automotive industry demands to reduce the development time and cost of the product development process. Also, repetitive ultterance for designing, prototyping and testing are expensive for the time and cost restriction for development of a products. Today, computer simulation by using analytical software reliably predict performance. [14]

Cab mounting system is generally used in Heavy Commercial Vehicle (HCV) to apart driver from vibrations which are generated from road. The vehicle's cabin is always mounted on the chassis of vehicle by using four supports. At the rear end of the cab is placed on Cab cross member centre channel. Generally isolators are used for mounting the cabin on this centre channel and this centre channel is attached with the frame through cross member end bracket. By using the assembly of cab side mounting bracket and frame side mounting bracket these are mounted on frame and on that frame front cabin is mounted. For this attachment of front cabin and frame bushing is use to provide segregation or keeping apart from vibration.

The frame side cab mount is attached to frame rail. Front cab mounting bracket give an advantage for tilting the cab for purpose of inspection and maintenance of the under-cab systems. The following Figure 1.1 shows cab mounting system with its components.

714

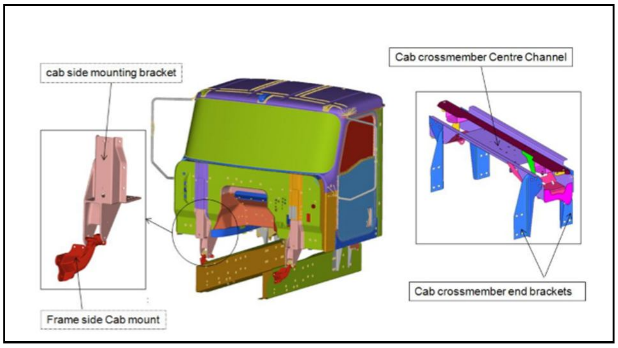


Fig.(1.1): cab mounting system [2]

Objectives and Methodology of Sub-systems are like cab mounting, fuel tank mounting, battery box mounting are subjected to failure due to heavy dynamic loads occurred during vehicle operation. For evaluation of the structural performance of these sub-systems of automobile vehicle it is important to determine “g” levels of acceleration at these systems. The methodology helps us moving towards virtual testing of components to avoid physical testing of the sub-systems to reduce time and cost. [9]

The researchers concerning that comfort in the cabins of truck implies several aspects like: vibration reduction and noise generated because of the structure, decreasing vibrations transmissibility from road and the power unit to the cabin, and keeping always up the parameters of climate inside the cabin. Vibrations caused by the loads due number of excitation may create lot of problems regarding comfort of passenger and to it effect on reliability of the body structure, too. Dynamic behaviour of the structures can be studied through: with the use of modal analysis dynamic behaviour of structure is calculated, and frequency response analysis also done . By the analytical studies of the structure vibration behaviours of the mounting systems can be predicted, analysed and can be improved which uses modelling systems and structures with Finite Element method. The truck cabin is connected to the frame by mounting system. This gives significant effect on the comfort in the cabin of truck.

II. LITERATURE SURVEY

“Durability Analysis Of HCV Chassis Using Fpm Approach” published by Shailesh Kadre, Shreyas Shingavi et.al The goal of the structural design is to reduce component weight and satisfying requirements of loads (stresses), stiffness, etc. This study focused on the durability analysis and fatigue process manager. In this study durability analysis was performed for historical loads such as racking, twists, and inertial loads. Also, for component level analysis, full frame model was used by them to retain accuracy by considering boundary conditions. by using the Hyper Works tool stress analysis was perform. In this tool Fatigue Process Manager (FPM), which help to use strain - life method. They observed that realistic results were produced by this approach with saving cost on pre and post processing efforts by reducing the time for solution. [13]

“Study Concerning the Optimization of the Mounting System of the Truck Cab” by Cornelia Stan, Daniel Iozsa et.al presents analytical approach concerning the mounting system optimization of truck cabin using FE model. among various structural elements this model is very useful to study vibration transmissibility, for the purpose of improvement in the comfort of the truck cab. Analytical study carried validated with the experimental study and in conclusion the study of optimization of the component mounting systems, for finding an optimal solution for, the damping and elastic characteristics easily can be changed in the model. The model presented in the paper is useful for analysing the behaviour of vibration which is generated in the automobile to increase the passenger’s comfort. In the body vibration behaviour analysis this model is very useful over various hurdles and turning on different types of road. [9]

“Optimization of Cabin Mounting” by Richard Ambroz. Composition of the FEM model is described in this study for anti-vibration mounting of tractor cabin. The main task here was to find a usable mathematical model which could really explain the behaviour of anti-vibrational mountings involved in static tests. Next it was suggested that change of structure of anti-vibration mounting and its effect on vibrations. To find out parameters the important thing is to prepare the set of material measuring. The next work was to create the dynamic model for anti-vibrational moan whereby the trans-missing of vibrations into the cabin would be possible to count. Then the anti-vibration moan static model was used by carrying the impact test on the cabin. [3]

“Design and Analysis of a Tractor-Trailer Cabin Suspension” by Paras Jain. The work done in this paper for overcoming the riding problem of Tractor or trailer vehicle. To improve the ride Low frequency of suspension can be adjusted.. An important parameter of HCV truck is the load carrying capacity beyond a certain limit it does not allow the softening of suspension . This research describes some insight and knowledge required for the cabin suspension design for a HCV. By using low frequency for a system at cabin mounting ride can be improved. for any commercial vehicle this methodology is much more effective and useful to improve comfort of ride for other passenger vehicles. For prediction of dynamic behaviour of any vehicle method of Numerical simulation is used for prediction of dynamic behaviour of vehicle in various condition.[1]

"Design and Analysis of Tiltable Truck Cabin Floor" by D Murali Krishna. Here In this study author have focused on load carrying capacity of the truck cabin generated from the roads, assemblies . The floor should be designed by ensuring fatigue life of it and should not get fail in service for instantaneous over-load. It should be able to resist vibrations from the bulkheads and engine assembly. To get the requirements, designed and analysis is carried on floor to know the behavior of the floor to the applied loads. By using CAD tool CATIA and ANSYS. Hence the design was safe based on strength analysis. The fundamental frequency from is obtained 6.288Hz from modal analysis and Resonance condition exists. By the addition cross members resonance was decreased. the maximum displacement as 0.3884mm and the velocity of 2.2839mm/sec by transient analysis. As per the analysis Transient displacements are little high in the front portion. [5]

III. PROBLEM DEFINITION

A. The overall weight of the HCV trucks is not acting on a single component; rather it is acting on all the components which are being part of automobile system.
B. The main three components for carrying the load on chassis are cabin mounting bracket, suspension mounting bracket and towing pin.
C. Research work basically is focusing on integrating these three components namely, front cabin mounting bracket, suspension mounting bracket and towing pin into single integrated component to achieve weight reduction, material saving, strength enhancement and improved operational efficiency by using topology optimization techniques.

IV. OBJECTIVES

A. To achieve weight reduction, basic three components are integrated into single model.
B. To achieve material saving Topology optimization technique is used.
C. To perform topology optimization on new design bracket with objective of reducing weight without affecting its strength and improve operational efficiency.

V. FINITE ELEMENT ANALYSIS

Different forces acting on different bracket are as follows:

1) Cabin Load: TATA 1613 truck is a vehicle with cabin weight of 2 ton in completely loaded conditions. Total of 2 ton loading gets distributed in 4 different brackets on which isolators are mounted. The product we are studying is utilized to connect these isolators to the vehicle body. So we can assume that 4 brackets of the cabin can each support around 500 kg of loading at fully loaded condition.

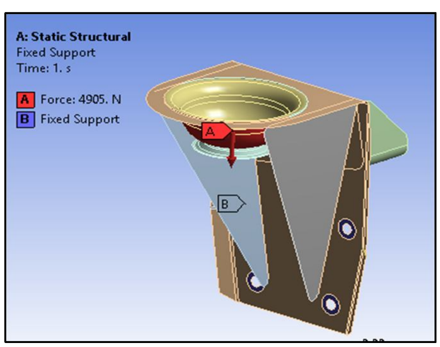


Fig.(5.1): Boundary condition of cabin mounting bracket

2) Load on Leaf Spring: TATA 1613 truck is a vehicle with gross weight of 16 ton in completely loaded conditions. When\ discussed about the weight distributions with the experts in the field of truck design and truck component sales and marketing technical team, inputs has been recorded as 8 ton load will be shared by the front axle and 8 ton loading will be shared by the back side axle, as loading happens at the back end of the vehicle.

716

So total of 8 ton loading gets distributed in 2 different leaf springs on the front axle those who isolate the shocks of the roads and goods loaded in the vehicle as well as vehicle body parts. The product we are studying is utilized to connect these leaf springs to the vehicle body. So we can assume that 4 ends of the leaf spring can each support around 2 ton of loading at fully loaded condition.
So loading on the single support bracket of the suspension mounting can be calculated as below
Total weight supported by each front leaf spring bracket - 2000 kg
Earth gravity acceleration: - 9.81 m/s ^{2}
Total load acting on suspension mounting bracket W_1 can be given as

$$W_1 = Weight \times g \qquad \text{.........} (1)$$

$$W_1 = 2000 \times 9.81$$

$$\boldsymbol{W_1 = 19620\ N}$$

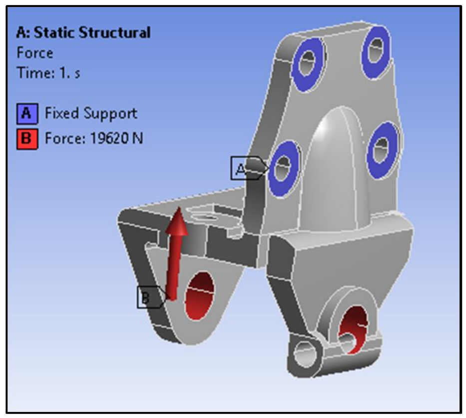


Fig. (5.2): Suspension mounting bracket boundary conditions

3) Towing load: Towing force is force that is required to tow the vehicle after is failure or accident situations.
So the total towing force required to pull the vehicle can be calculated as follows:

A. Rolling Resistance

Rolling resistance Composed primarily of

1) Tire deformation Resistance (~90%)
2) Surface compression and Tire penetration (~ 4%)
3) Air circulation and Tire slippage around wheel (~ 6%)

This resistance of a vehicle is directly proportional to the component of weight and normal to the surface of travel
F_{rr} α N,

$$F_{rr} = C_{rr} \times N \qquad \text{.........} (2)$$

Where,
N=Force Normal to Surface N
C_{rr}=Coefficient of rolling friction

$$F_n = 0.01 \times 16000$$

$$F_{rr} = 160 kg$$

$$F_n = 160 \times 9.81$$

$$F_{rr} = 1569.3\ N$$

B. Aerodynamic Resistance

The force which the oncoming air applies on a moving body is Aerodynamic drag. It is the resistance which is offered by the air to the movement of the body. So, when a car is moving; it displaces the air. However, it affects the car's speed and performance.
The drag force in given by,

$$F_d = \frac{\rho}{2} \times C_d \times A \times V^2 \qquad \text{.........} (3)$$

Where, F_d= Drag force, is the force component in the direction of the flow velocity N
ρ = Mass density of the fluid kg/m^3

717

V = Velocity relative to the object m/s
A =Reference area m^2
C_d=Drag Coefficient

$$F_d = \frac{1.2245}{2} \times 0.45 \times 3 \times 4 \times 5.5^2$$

F_d=100 N

C. Acceleration Force

The net force act on object is equal to the mass of the object multiplied by the acceleration of the object.

$$F_a = M \times A \qquad \text{……… (4)}$$

Where,
F_a= Acceleration force or Pulling force N
M = Mass Kg
A = Acceleration m/s^2
For acceleration consider,
Speed =20 km/hr.=5.5m/s
Time =1 min=60sec

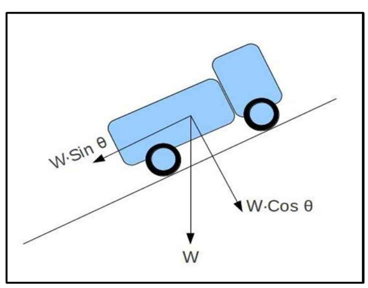
$$\text{Acceleration} = \frac{\text{Speed}}{\text{time}}$$
$$= \frac{5.5}{60}$$

=0.09166

Pulling force required=F_a = Mass × Acceleration
=160000×. 09166
=1466.56 N

D. Gradeability Requirement

A vehicle's component weight acts in a direction opposite to its motion when vehicle is travelling uphill.. To overcome this backward force if some energy is not supplied to overcome this backward force, then the vehicle may slow down and roll backwards. If the vehicle is going uphill at a slope of θ, weight of the vehicle, W has two components: one which is perpendicular to the road surface and the other along the road surface. The component along the road surface is tries to restrict the motion. The gradient resistance is given by: F_G = W·Sin θ



In our case θ=0
Therefore,

$$F_G=0$$

Total Resistance on vehicle

$T = F_{rr} + F_d + F_a + F_G$
T =1569.3+100+1466.56+0
T =3135.86 N

Therefore total amount of force required for towing = 3135.36 N which can be taken approximately as 3200 N for more safety purpose. This force will act in horizontal direction.

718

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

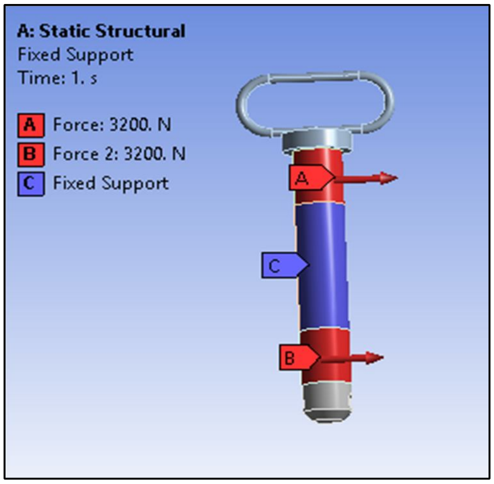


Fig. (5.3): Boundary conditions for towing pin

E. Stress in Different Brackets

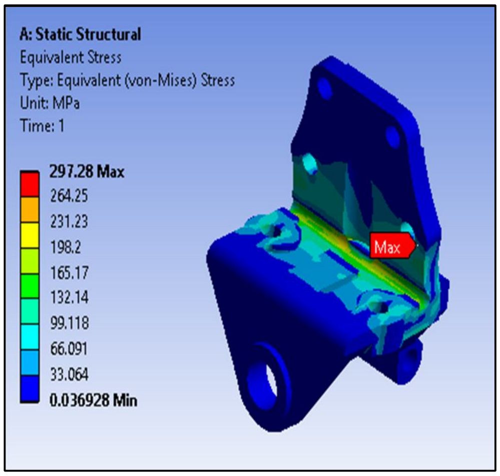


Fig.(5.4): Von Mises stress on suspension mounting bracket

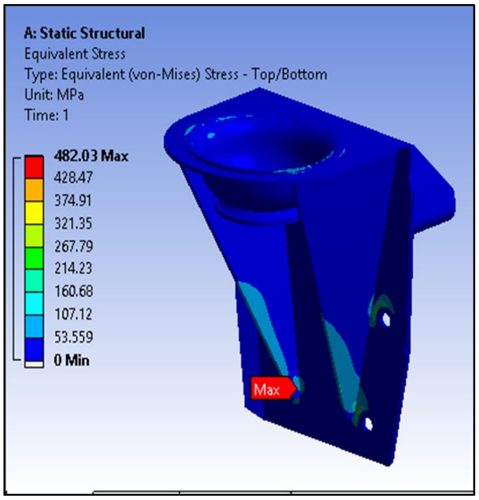


Fig.(5.5): Von Mises stress on cabin mounting bracket

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

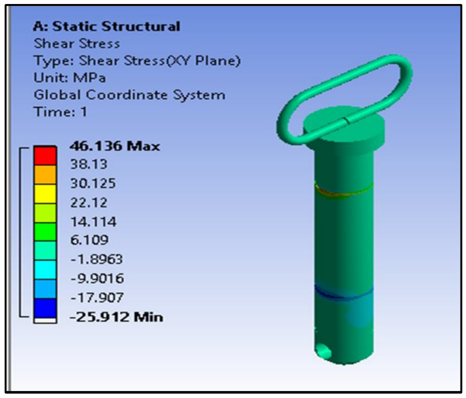


Fig.(5.6): Towing pin geometry for truck shear stress plot

F. New Design

We have integrated all three parts into single component and optimized the same for weight reduction. That new design model is call as Super bracket.

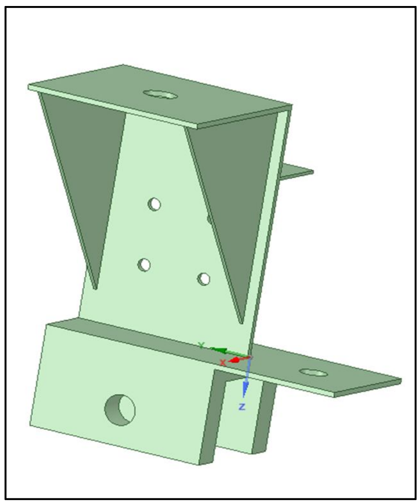


Fig.(4.7): Geometry of Super Bracket in Space Claim

G. FEA Analysis for New Part

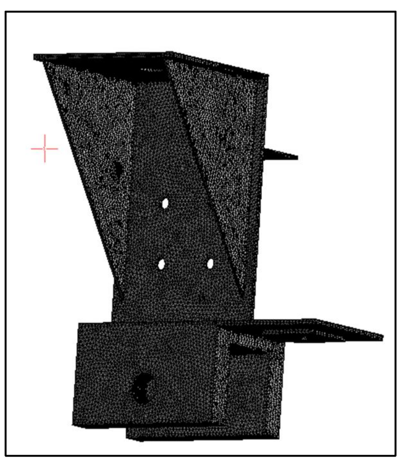


Fig.(4.8): Mesh model of Super Bracket

720

Statistics	
Nodes	269823
Elements	150685

Fig.(5.9):Number of nodes and elements

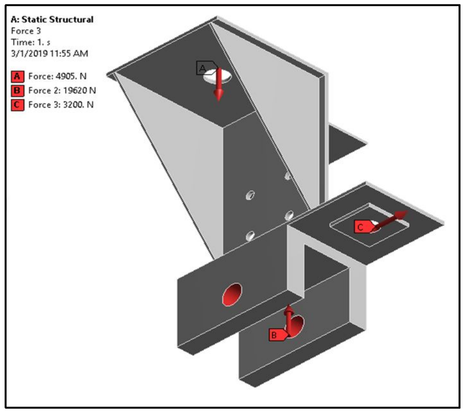


Fig.(5.10): Force acting on super bracket

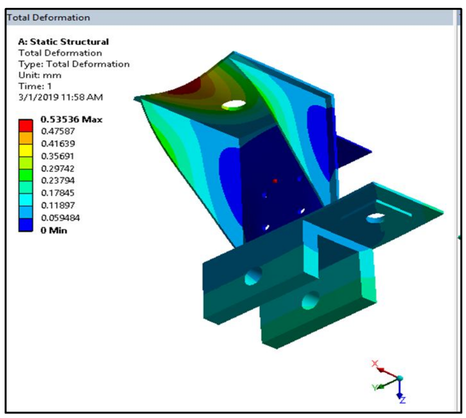


Fig.(5.11):Total deformation on Super Bracket

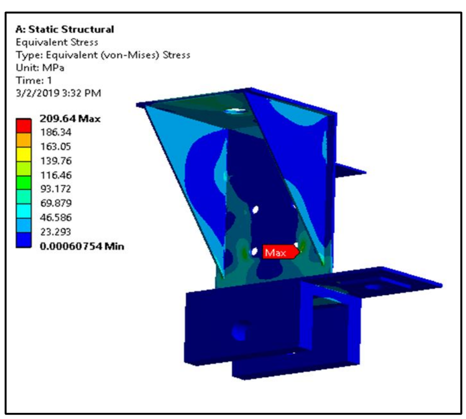


Fig.(5.12):Equivalent Stress on Super Bracket

The Maximum equivalent stress is 209.64 MPA. Again if we see, the maximum stress is observed near the edge of fixed support only. If we move two elements away the Equivalent stress decreases from 209.64 MPA to 96.033 MPA.

721

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

The detailed view near fixed edge is shown below:

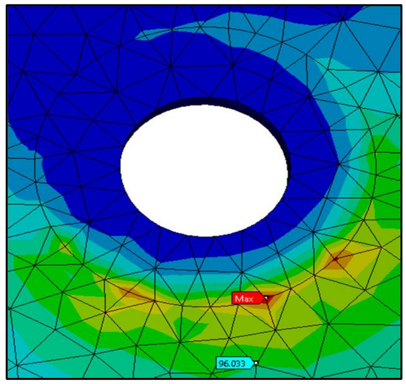


Fig (5.13): Maximum Stress on Super Bracket

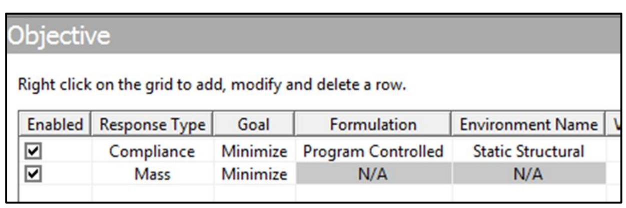
So as we can see that as the Total deformation and Equivalent stress are on the lower side there is further scope for optimization so as to reduce the weight of bracket and increase its stiffness.
So now we will perform Topology optimization on Super bracket with objective of minimizing weight and compliance of super bracket.

H. Topology Optimization

Optimization technique in Ansys that optimizes material layout within a given design space, for a given set of load, boundary condition and constraint is known as Topology Optimization.
In our case the objective and constraint in doing topology optimization are as follows
Objective
1) Minimize Mass
2) Minimize compliance
Response Constrain
Global von-Mises Stress =146.66 MPA
The maximum allowable stress is calculated based on fatigue life

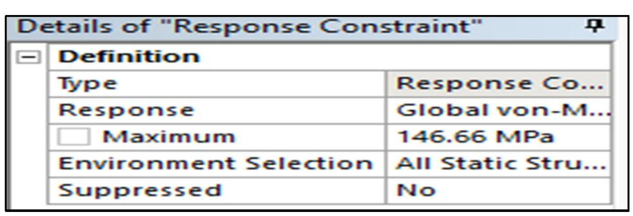
$$\sigma_{all} = \frac{0.5 * Sut}{FOS}$$

Where,
Sut=Ultimate tensile strength=440 MPA
FOS=Factor of safety=1.5
Therefore, the maximum allowable stress =146.66 MPA


Objective
Right click on the grid to add, modify and delete a row.

Enabled	Response Type	Goal	Formulation	Environment Name
☑	Compliance	Minimize	Program Controlled	Static Structural
☑	Mass	Minimize	N/A	N/A

Fig.(5.14):Topology objectives


Details of "Response Constraint"

Definition	
Type	Response Co...
Response	Global von-M...
Maximum	146.66 MPa
Environment Selection	All Static Stru...
Suppressed	No

Fig.(5.15):Topology Response constrain

722

The result of topology optimization is shown below:

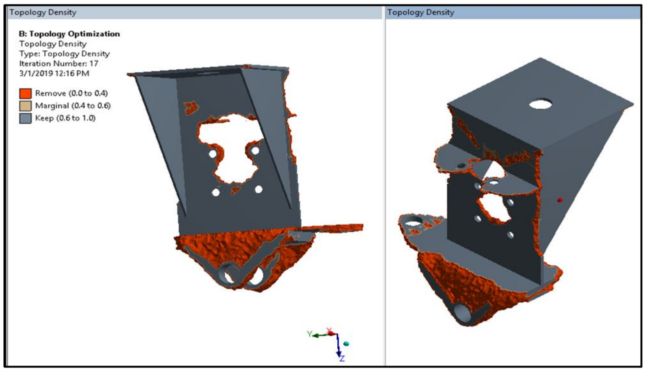


Fig.(4.16):Topology density

The red area is area from where we can remove the material for optimizing our design.

Taking optimization result in consideration now we can create new geometry that will fulfill our objective of minimizing weight and compliance of Super Bracket.

The Topology optimization has given us the result by using which we can further optimize our geometry in term of mass. Therefore the new geometry of our super bracket after removal of material is design in Space Claim as shown below. The weight of new geometry is 9.428 kg.

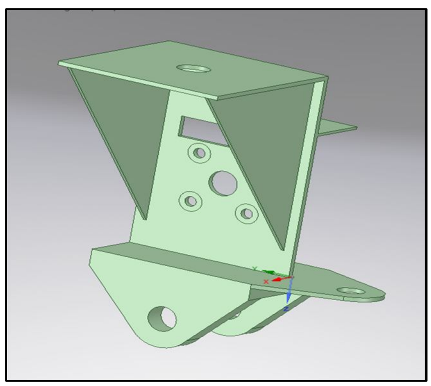


Fig.(5.17):Geometry of Super Bracket in Space Claim

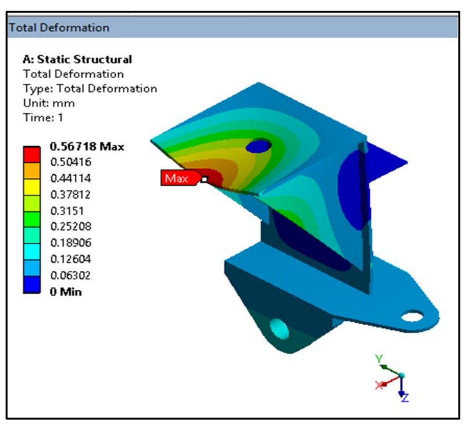


Fig.(5.18): Total Deformation of Super Bracket

The Total Deformation is shown in scale of (2.e +002) to show the pattern of deformation that occurs. In True scale the deformation is negligible. The Maximum Deformation in bracket is 0.56718mm, which is just 0.03182mm more than old design.

723

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

Equivalent (von-mises) Stress on New design is as shown

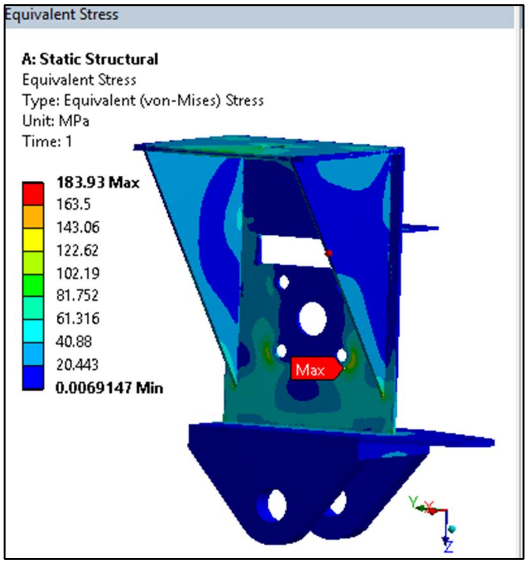


Fig.(5.19): Total Deformation of Super Bracket

The Maximum equivalent stress is 183.93 MPA. Again if we see, the maximum stress is observed near the edge of fixed support only. If we move two elements away the Equivalent stress decreases from 183.93 MPA to 83.73 MPA.
The detail view of maximum Equivalent Stress near the fixed support is as shown below:

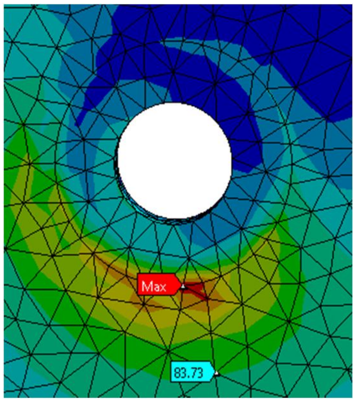


Fig.(5.20): Detail view of maximum Equivalent Stress

VI. MANUFACTURING AND TESTING

The optimized super bracket is manufactured by laser cutting operation. The sheets of required thickness are cut as per the requirement by laser cutting operation. The small pieces cut are welded together to form one single piece known as our super bracket

Fig.(6.1): Laser cutting operation for manufacturing super bracket

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IX, Sep 2019- Available at www.ijraset.com

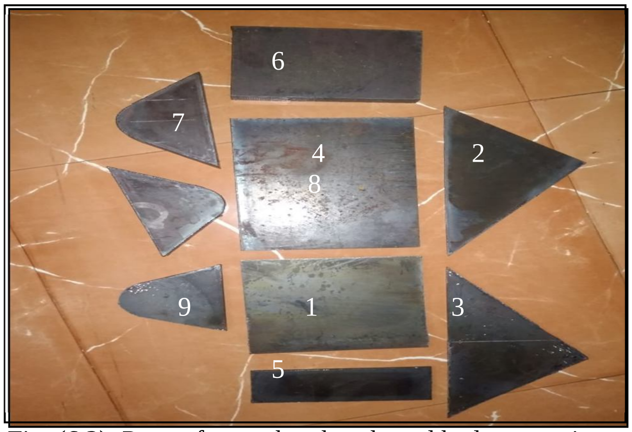


Fig. (6.2): Parts of super bracket shaped by laser cutting

1- Back plate 2 and 3- side plate
4 -Top plate 5 -support back plate
6- bottom plate 7 and 8- suspension mounting plates
9 - towing plate

Fig.(6.3): Manufactured model.

Figure shows the actual assemblage of laser cutting shaped pieces. This is just a representation of the how pieces will be welded together. currently held together with spot welds. After drilling and slotting operations are performed on the plates then first back plate will be welded with side wedges and back side support plate with back plate. Then Top plate will be welded to the assemblage of welded plates. Later on suspension mounting section will be welded together separately. Then cab mounting bracket will be welded to suspension mounting bracket on the top of it. All components are welded with continuous welds.
The Two types of loads where applied on component: Tensile and compressive load is applied on bracket to study of tensile and compressive deformation of bracket. Appling load on bracket are as following table.

Testing	Load
Tensile	3200N
Compressive	14715N

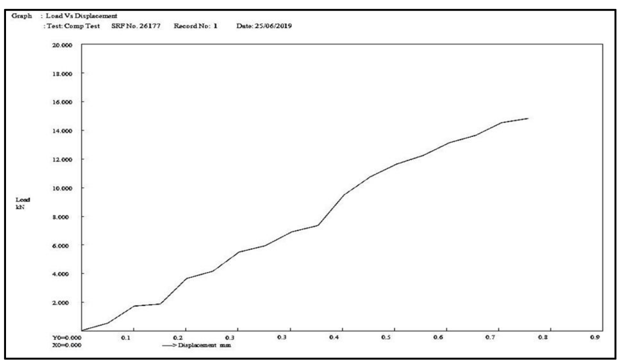
Table 1: Load table

725

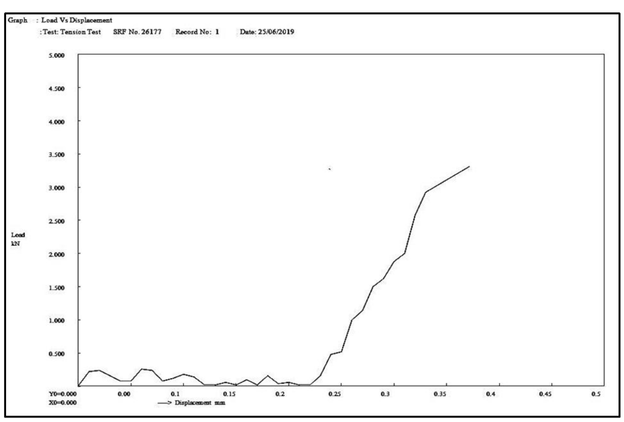
Fig.(6.4): Testing of Manufactured Bracket on UTM

Type of loading	Max. Deformation observed (mm)
Compressive	0.77
Tensile	0.37

Table 2: Results from Experimentation for final optimize integrated model.



Graph 1: Graph for Compression Test.



Graph 2: Graph for Tensile Test.

726

VII. RESULT AND CONCLUSION

	model	Load (N)	Max Stress (Mpa)	Max Def (mm)	Weight (kg)	Cost (Rs)
1	Suspension mounting bracket	19620	297.28	0.544	7.42	1200
2	Cabin mounting bracket	4905	482.02	0.297	3.92	1700
3	Towing pin	3200	46.13	0.144	0.96	700
				Total	12.3	3600

Table 3: Result summary for three basic components.

Model	Load (N)	Max Stress (Mpa)	Max Def. (mm)	Weight (kg)	Cost (Rs)
Integrated model	A-4905 B- 19620 C-3200	96	0.54	12.72	3120

Table 4 : Result summary for integrated model.

model	Load (N)	Max Stress (Mpa)	Max Def. (mm)	Weight (kg)	Cost (Rs)
Final optimize Integrated model	A-4905 B- 19620 C-3200	83.7	0.57	9.42	2450

Table 5: Result summary for final optimize integrated model.

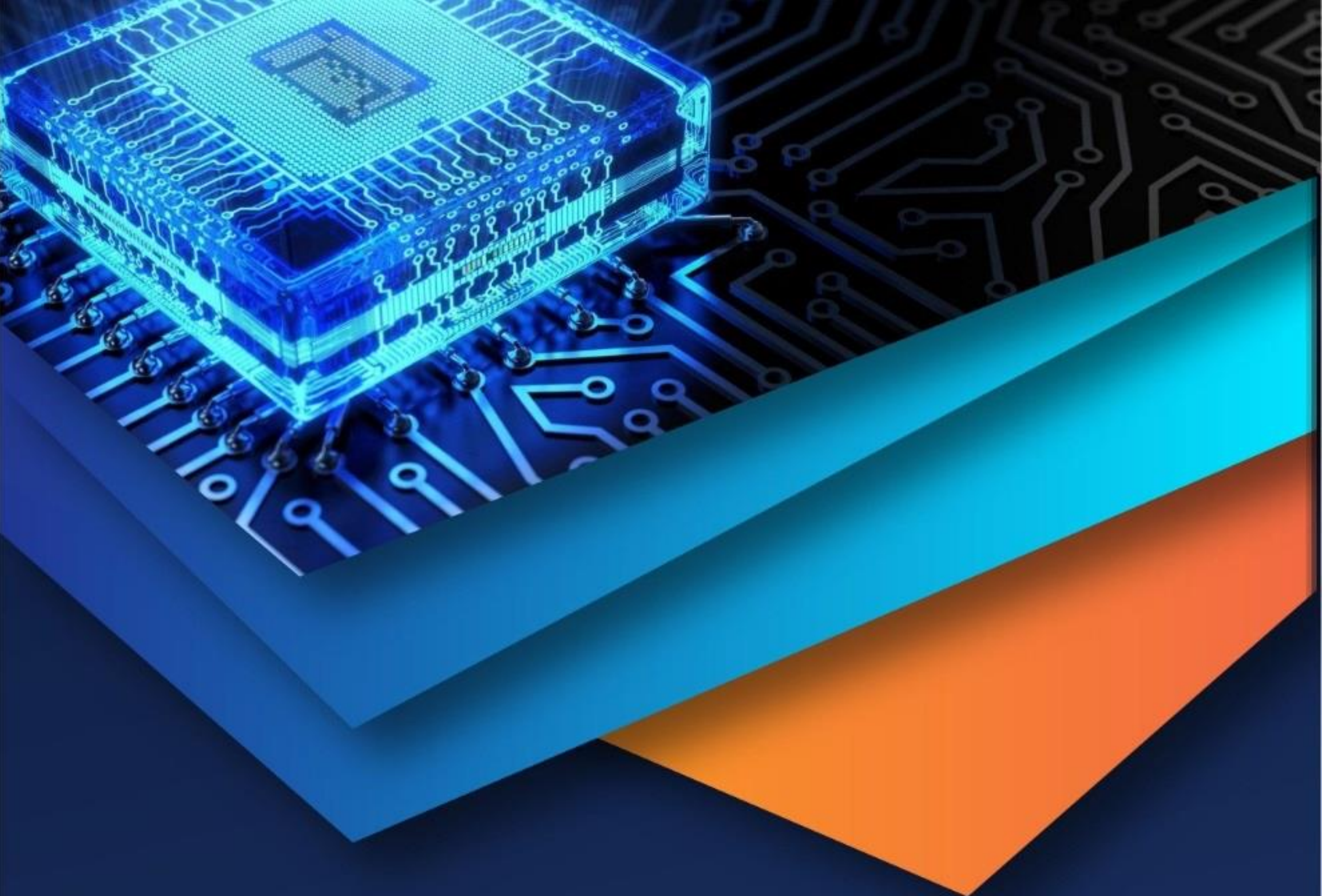
A. Reverse engineering of the selected components and creation of CAD models for the same is done and FEA is perform on the prepared CAD models.
B. Designed an alternative super bracket for the combination of three brackets and successfully analyzed the bracket while saving weight of the bracket.
C. From FEA analysis of integrated model the maximum value of stress is 96 MPa and total deformation is 0.54 mm and topology optimized integrated model maximum stress value is 83.7 MPa and deformation is 0.57 mm.
D. Basic three component stress value and optimized integrated model stress values are well within acceptance criteria.
E. Weight of individual basic components and final optimized model reduce from 12.3 kg to 9.42 kg, which is 24% reduction.
F. Basic three component total cost was 3600 Rs and our final optimized integrated model cost is 2450 Rs. Cost reduction achieved by 32%
G. The experimental Testing of model shows the maximum deflection of 0.7732 mm for compressive load and 0.3798 mm for tensile loading.

727

REFERENCES

[1] Hui wang ,zheng-Dong Ma (2003), “Numerical and experimentation verification of optimum design obtained from Topology Optimisation”.
[2] Caner Demirdogen ,Jim Ridge (2004), “Weight optimized design of front suspension component for commercial heavy trucks”.Vol.1.
[3] Hong suk chang (2006), “A study on analysis method for optimizing mounting bracket”.Vol.1.
[4] Paras Jain,(2007), “Design and Analysis of a Tractor-Trailer Cabin Suspension”,Vol.1, Issue 2.
[5] Teo Han Fui, Roslan Abd. Rahman, (2007), “Statics and Dynamics Structural Analysis of A 4.5 Ton Truck Chassis”, Vol.3, Issue 7, PP (24, 56 – 67).
[6] Richard Ambroz, David kollhammer, David palousek, (2007) “Optimization of cabin mounting” Vol. 2, issue 3.
[7] R. Prakash V. Kavinraj, A. Anandajayakumar S. Karthik (2013) “Design and Analysis of Leaf Spring Bracket in Air Suspension Vehicle Using FEA”, Vol. 2, Issue 11.
[8] D. Murali Krishna, S. Madhu, (2014), “Design and Analysis of Tiltable Truck Cabin Floor”, Vol 2, Issue 7, PP (69-73).
[9] Dattatray Gavade, Satej Kelkar (2015), “Study & Analysis of Structural Behaviour of Front Suspension Rear Shackle Bracket”,Vol. 2, Issue 12.
[10] Pushpendra Mahajan, Abhijit L. Dandavate (2015), “Analysis and Optimization of Compressor Mounting Plate of Refrigerator using FEA.” Vol. 5, Issue 5.
[11] Vyankatesh D. Pawade , Pushkaraj D. Sonawane (2015) “Study of Design and Analysis of Air Conditioner Compressor Mounting Bracket.” Vol.2, Issue 5.
[12] Cornelia Stan, Daniel Iozsa, Razvan Oprea (2016), “Study Concerning the Optimization of the Mounting System of the Truck Cab”, Vol.1, Issue 2
[13] P. Meghana , Y. Vijayakumar (2016) , “Analysis of cabin mounting bracket of truck using Ansys”.Vol.5 , Issue 11.
[14] Jadhav Shashikant, Madki S (2017) “Study and Analysis of Front Suspension Shackle Bracket for Commercial Vehicle”, Vol. 5, Issue 4.
[15] Rajkumar Ghadge, Pankaj Desle (2017) “Design and weight optimization of cabin mounting bracket for hcv”, Vol. 2, Issue 7 .
[16] Shashikant Jadhav, S.J. Madki (2017) “Optimization of Front Suspension Shackle Support using Finite Element Analysis”, Vol. 4, Issue: 08.
[17] Shailesh Kadre, Shreyas Shingavi, Manoj Purohit, (2017), “Durability Analysis of HCV Chassis Using Fpm Approach.” Vol.1, Issue 2.
[18] Suvarna, M Shirsath (2018) “Design & Weight Optimization of The Front Cab Mounting Bracket Of Truck”, Vol.2, Issue 8.

728



10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY