Design and Weight Optimization of Integrated Super Bracket According To Stress Analysis

Mr. Admane Anant Dnyaneshwar[1], Mr. Satej Kelkar[2]

[1]PG Scholar, JSPM's Jaywantrao Sawant College of Engineering, SPPU, India
[2]Professor, JSPM's Jaywantrao Sawant College of Engineering, SPPU, India

---***---

Abstract - *An Automobile is a self-propelled vehicle which is used for transportation of goods and passengers. The motor vehicles, both passenger car and trucks are generally considered to be made up of two major assemblies: Body and Chassis. Chassis is a frame or main structure of a vehicle. Growing competition in automotive market makes it more and more necessary to reduce the development time and cost of the product development process. Hardware prototypes cannot be made in early design phase, however, today with the use of CAE tools virtual models can be created to accurately represent physical models and to take right decisions at the right time. The vehicle designer needs to know the most damaging loads to which the structure is likely subjected. If the structure can resist the worst possible loading which can be encountered then it is likely to have sufficient fatigue strength. Nowadays Automobile industry has become one of the largest branches of mechanical engineering industry. Automobile vehicle consumes a lot of fuel while transporting goods and people from one place to other by road. Heavy automobiles consume more fuel for its operation hence reducing automobile vehicle's weight for better economy is the challenge in front of automobile industries right now. Project work is focused on design and shape optimization of HCV truck's integrated front Cab mounting bracket, suspension mount and tow hook. The overall weight of the HCV trucks is not under single scope, so we will be selecting the components cab mounting bracket, towing pin and suspension mounting bracket so as to reduce the weight. Study will be focused on finding alternative design for cab mounting bracket of the truck.*

Keywords: *HCV, Vibration, CAD, FEA, ANSYS.*

1. INTRODUCTION

Strength is an important consideration in the damper mount design and to increase the stiffness (bending and torsion) characteristics. Adequate torsion stiffness is required to have good handling characteristics. Generally brackets are designed on the basis of strength and stiffness. In the conventional design procedure the design is based on the strength and emphasis is then given to increase the stiffness of the mounts, with very little consideration to the weight. One such design procedure involves optimizing the periphery of bracket to the existing design to increase its torsion stiffness. As a result weight of the bracket decreases. This decrease in weight increases partly the fuel efficiency and helps in judicious use of material. The design of the bracket with adequate stiffness, strength and lower weight provides the scope for this project. The goal of the structural design is to obtain minimum component weight and satisfying requirements of loads (stresses), stiffness, etc. The process of producing a best structure having optimum structural performance is termed as structural optimization.

1.1 STRUCTURAL ARRANGEMENT OF TRUCK

In contrast to private cars where a structural part is the body frame, by trucks performs chassis this function. All mechanism which allows ride and manipulation of trucks are its components. Some parts of undercarriage have to assign further function.

Chassis:

Chassis especially of tractor with the performance of lower-class are frameless structures. This solid structure is form from the motor on which transmission is screwing on and then on the transmission final gear. Considering relaying performances are both relatively massive castings. From side to final drive housing there are screwing on another casts in which are located mostly speed reducing gearbox.

Cabin:

The tractor cabin is a specific structural element. It makes work environment of operation and quality of its variant impacts work performance of driver and his health as well. The cabin foundation is welded steel frame, usually with six or four narrow columns which are connecting the top and the lower part of structure. In view of safety at work the cabin protects driver in case of jolt or overturn.

Anti-vibrations mounting:

The cabin is placed on 4 anti-vibration mountings. They are bolting to brackets. Since used bracket is frameless as anti-vibration mountings are fixed to casts. The anti-vibration mounting is usable for flexible conjunction of two structural elements. It is made from an outer casing which is made most often from structural steel. The attachment of anti-vibration mounting is made from screw joint. The screw crosses through an internal pipe of mount and the tightening torque of the screw is 80 – 100 Nm. A space between the outer casing and the internal pipe is filled by rubber. Rubber is vulcanization on the outer casing and the internal pipe. Between the internal pipe of anti-vibration mounting and the connected structural there are inserted washers even.

2. LITERATURE REVIEW

Ms.Suvarna M Shirsath (2018) [1] (Design & Weight Optimization of The Front Cab Mounting Bracket of Truck) in this project, mounting bracket design optimization will be performed by changing from conventional steel to composite material. Consequently, there will be tremendous saving in material in mounting brackets manufacturing industries as a result of optimization.

Mr. Rajkumar Ghadge, Mr. Pankaj Desle (2017) [2] (DESIGN AND WEIGHT OPTIMIZATION OF CABIN MOUNTING BRACKET FOR HCV) Project work is focused on design and weight optimization of HCV truck's front Cab mounting bracket. Study is focused on finding alternative design or material for cab mounting bracket of the truck.

Jadhav Shashikant, Madki S. (2017) [3] (Study and Analysis of Front Suspension Shackle Bracket for Commercial Vehicle) This paper reviewed work done in the area of Optimization & Design of Rear Suspension Shackle Bracket by FEM. Shackle Bracket is part of leaf spring assembly, which accommodates leaf deformation, when subjected to operational load.

Shashikant Jadhav and S. J. Madki.(2017) [4] (Optimization of Front Suspension Shackle Support using Finite Element Analysis) presented by Design of suspension systems for Heavy Trucks is always challenging due to the heavy loads the system is exposed to and the long life requirements for the total system. Topology optimization is used at the concept level of the design process to arrive at a conceptual design proposal that is then fine-tuned for performance and manufacturability. Due to this avoids costly design iterations and time consuming. Engineers can find the best design concept that meets the design requirements by using topology optimization. Application of topology optimization has been done with finite elements methods.

Pushpendra Mahajan and Prof. Abhijit L. Dandavate (2015) [5] (Analysis and Optimization of Compressor Mounting Plate of Refrigerator using FEA.) In this research, the researchers have said that NVH is one of the major factors impacting quality for household appliances like refrigerators. In refrigerators, compressor is the main source for vibrations and noise. If compressor operating frequency matches with natural frequency of plate then resonance would occur leading to excessive vibrations and noise. In this paper, natural frequency and static state deflection of a compressor mounting plate are analysed using FEA software, ANSYS. Further two methods of improving and optimizing the design to increase the natural frequency are illustrated and analysed.

3. PROJECT OUTLINE

A. NEED OF RESEARCH

Nowadays Automobile industry has become one of the largest branches of mechanical engineering industry. Automobile vehicle consumes a lot of fuel while transporting goods and people from one place to other by road. Heavy automobiles consume more fuel for its operation hence reducing automobile vehicle's weight for better economy is the challenge in front of automobile industries right now.

B. OBJECTIVES

1. To study the background of currently available cabin mounting bracket, Suspension mounting bracket and towing pin.
2. To create 3D models for the selected components using CAD software by reverse engineering the selected components and perform static and modal analysis on the 3D models using FEA software
3. Integrate the selected components to create a single alternative design (super bracket) and create 3D model for the same.
4. Perform static and modal analysis on the super bracket model using FEA to find stress, deformation and natural frequencies of the component and optimization of the super bracket.

C. METHODOLOGY

1. Reverse engineering of selected components:
Selected currently available components will be purchased from market. By reverse engineering the purchased components we will find out the actual dimensions of the components.

2. Creation of CAD models:
Once the dimensions are finalized we will create the 3D models of the selected components. For creation of the 3D models CAD software will be used. First the components are bought for reverse engineering from aftermarket shops of truck components so that detailed modelling of the component can be performed using AUTODESK FUSION 360 software. Images below show the actual components purchased for the reverse engineering.

3. Perform FEA:
CAD model created for bracket will be imported in to FEA software (ANSYS). Boundary conditions will be applied to the model as per the loads and constrains that are physically experience by the components. Static and modal analysis will be carried out on the 3D CAD models and results will be obtained.

4. Design Integration:
In this step all the three components cabin mounting bracket, suspension mounting bracket and towing pin will be integrated and a new single alternate design will be created by using design integration method. Also create the CAD model for the same.

5. Finalization of block dimensions:
The block dimensions will be finalized by using space constraint method.

6. Topology Optimization:
By using FEA optimization module in ANSYS software shape optimization of the integrated design will be performed. Topology optimization finds the best distribution of material given an optimization goal and a set of constraints. It works by taking a solid block of material in any shape and removes material from it to minimize or maximize an optimization objective such as mass, displacement, or compliance while satisfying a set of constraints such as maximum stress or displacement

7. Perform FEA on Integrated design:
Static analysis and modal analysis on the optimized integrated model will be performed and the results will be obtained in ANSYS.

8. Comparison of FEA results:
FEA results of both selected components and optimized

4. FEA WORK

First the components are bought for reverse engineering from aftermarket shops of truck components so that detailed modelling of the component can be performed using AUTODESK FUSION 360 software. Images below show the actual components purchased for the reverse engineering.

Fig. 1 Suspension Mounting Bracket

Fig. 2 Cabin Mounting Bracket

Fig. 3 Pin for Towing

By measuring the components bought using measuring instruments like measuring tape, venire calliper and scale 3 dimensional models for the all three components are created using FUSION 360. Models created are shown in the images below. Model module of Fusion 360 is used to create the object shown in the image below. Series of sketching and modelling commands are used to create the object.

Meshing:

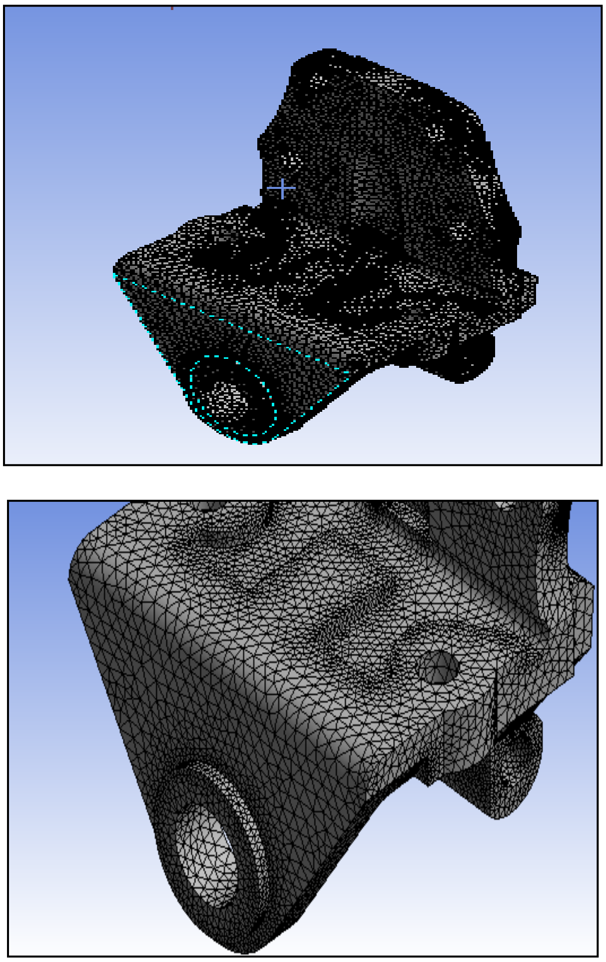

Fig. 4 Meshing Of Suspension Mounting Bracket.

Solid component is meshed using solid 185 elements of tetrahedral elements. 5 mm average and 0.14 mm minimum edge size for the meshing of the component is used. Higher order elements with mid side nodes are used for better accuracy of results.

Total of 3.68 lakh elements and 5.49 lakh nodes are used to mesh the single components. So smooth results can be obtained for the same.

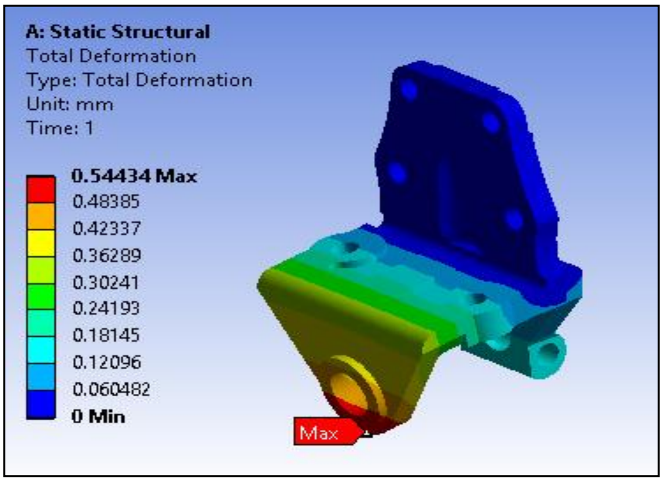

Fig. 5Deformation Plot Suspension Mounting Bracket

Suspension mounting bracket with load conditions mentions shows maximum deformation of 0.5443mm at free mounting end. Stress or von mises stress plot of the component shows the maximum stress of 297 MPa which is again restricted to very small area, due to stress concentration. Most of the geometry has stresses below 100 MPa. This is well within the acceptance criteria of high carbon steel material from which the components of suspension mounting bracket are made.

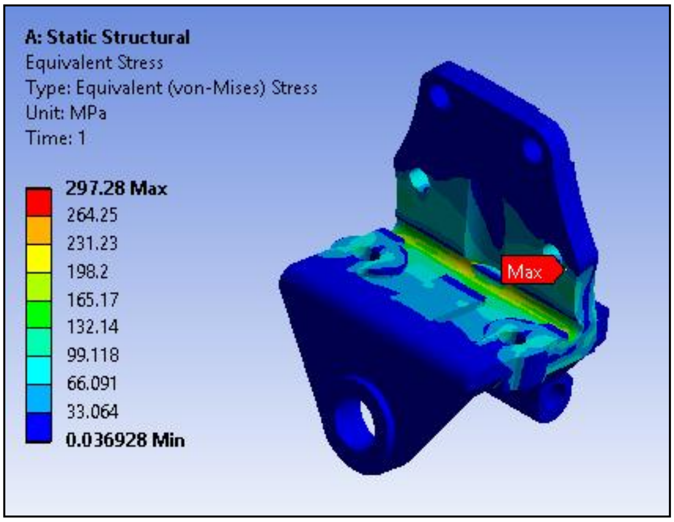

Fig.6 Von Mises Stress Plot Suspension Mounting Bracket

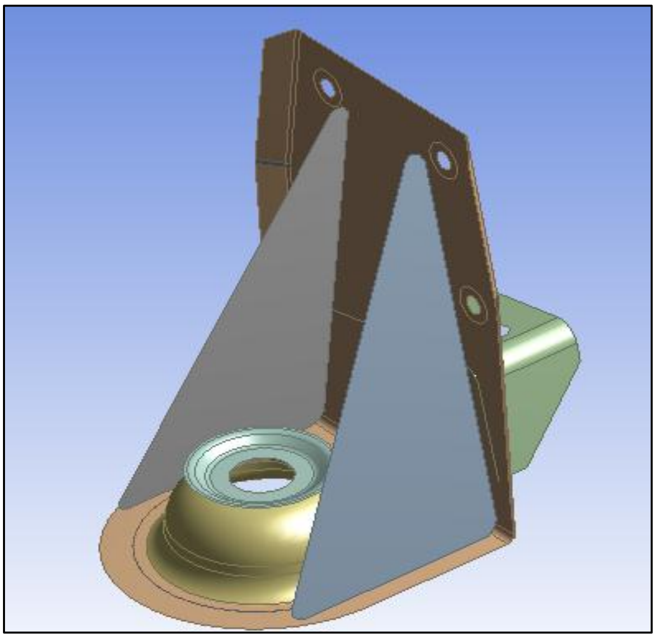

Fig. 7 Sheet metal extracted from iges cabin mounting bracket

TATA 1613 truck is a vehicle with cabin weight of 1.5 ton in completely loaded conditions. Total of 1.5 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 375 kg 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 cabin mounting bracket - 375 kg

Earth gravity acceleration: - 9.81 m/s ^{2}
Total load acting on cabin mounting bracket W_2 can be given as

$$W_2 = Weight \times g$$
$$W_2 = 375 \times 9.81$$
$$W_2 = 3678.75 \ N$$

All the components are made using 3.05 mm thick sheet metal of cold rolled steel. Boundary conditions as per calculated load and fixed to bolted holes is shown in the figure below.

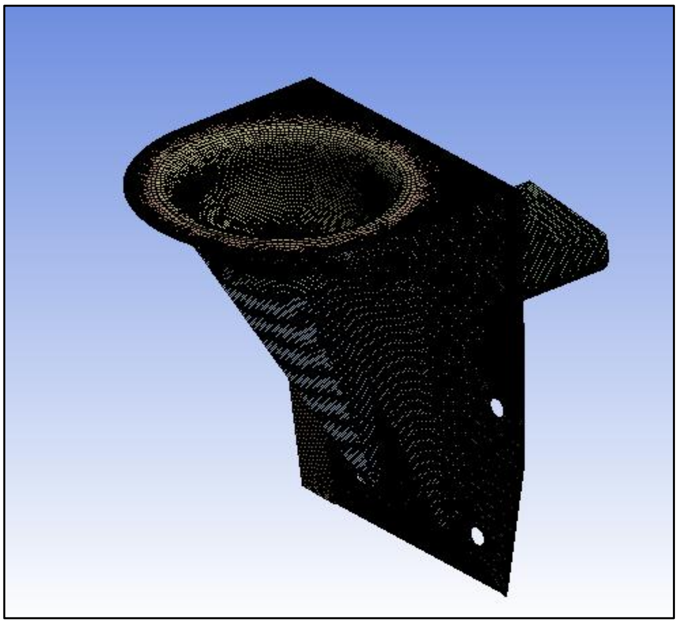

Fig. 8. Meshing of cabin mounting bracket

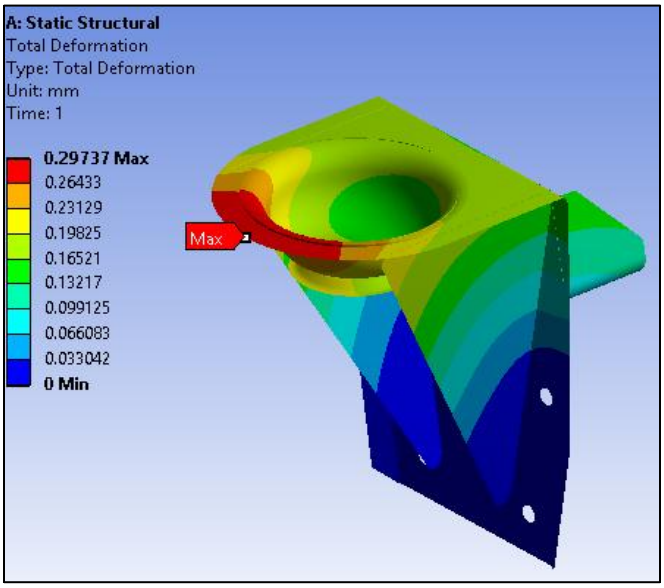

Fig. 9 Deformation plot cabin mounting bracket

Cabin mounting bracket with load conditions mentions shows maximum deformation of 0.29 mm at free mounting end. Stress or von mises stress plot of the component shows the maximum stress of 361 MPa which is again restricted to very small area, due to stress concentration. Most of the geometry has stresses below 100 MPa. This is well within the acceptance criteria of high carbon steel material from which the components of suspension mounting bracket are made.

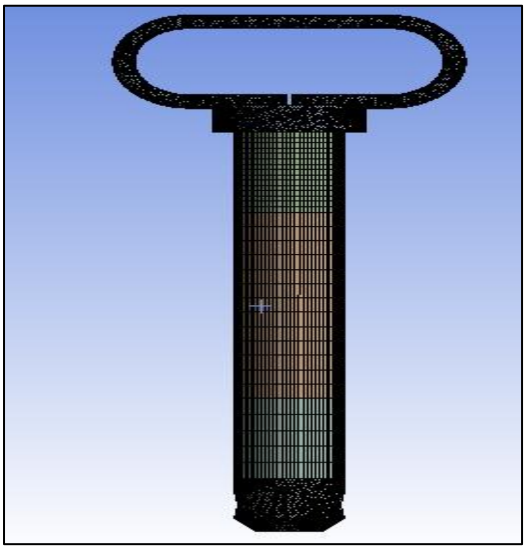

Fig. 10: Meshing for pin geometry

Towing is application when truck experiences down time of engine while transport and it is rolled on the road with complete weight with connecting it to another vehicle using pin and brackets or shackles. So 2 pins will take up the complete load of 16 ton in the scenario. Dividing between both the pins equal loading boundary conditions are calculated for the pin.

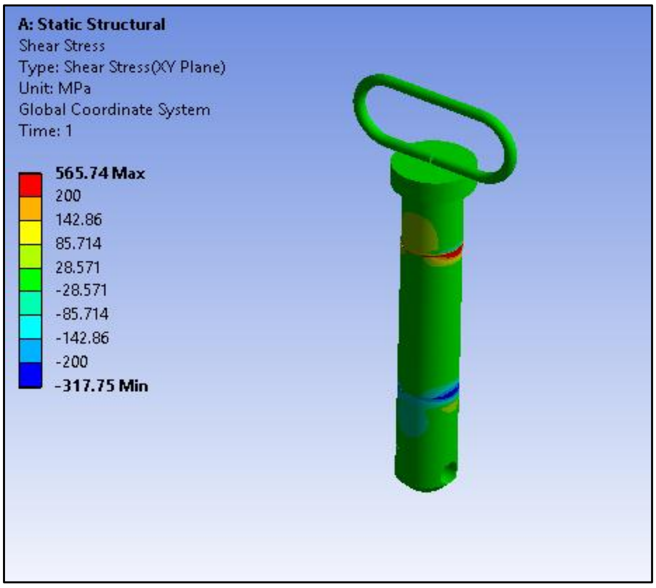

Fig. 11 Towing pin geometry for truck shear stress plot

Most of the are shows stresses less than 200 MPa and between 28 MPa in both directional shear, high concentration region shows maximum of 565 MPa stresses.

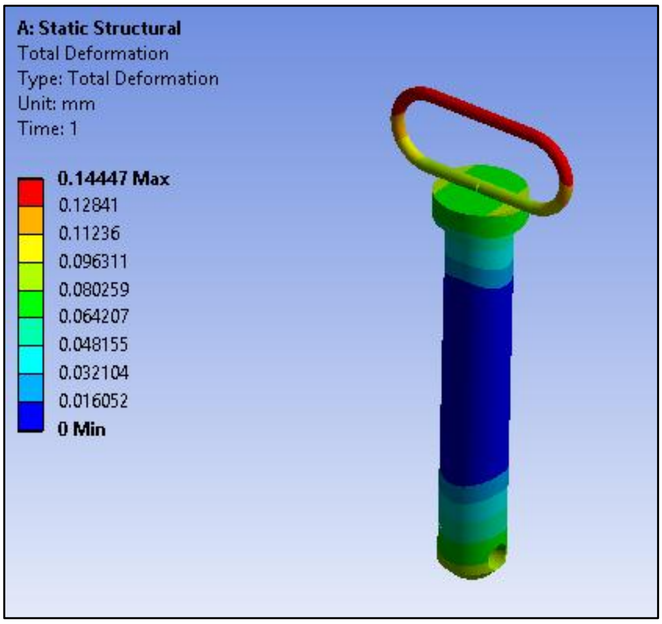

Fig. 12 Total deformation plot

Maximum total deformation of 0.14 mm is shown in the analysis.

5. SUPER BRACKET FEA:

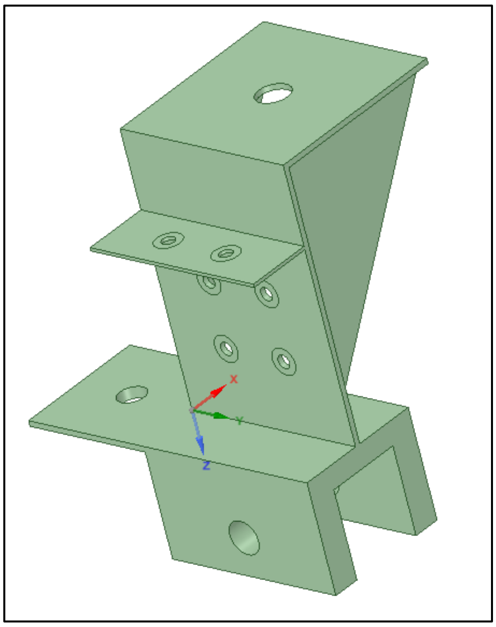

Fig.13 Geometry of Super Bracket in Space Claim

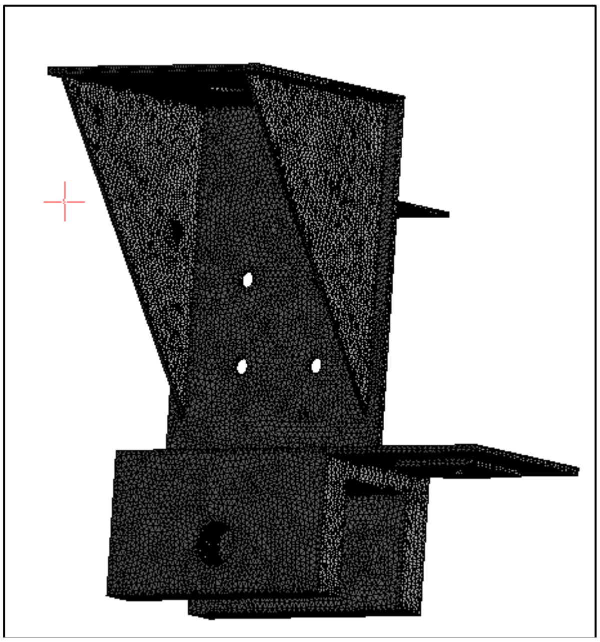

Fig.14 Mesh model of Super Bracket

Statistics	
Nodes	269823
Elements	150685

Fig15. Number of nodes and elements

Forces acting on Super Bracket:

1) Cabin load: The load acting on cabin mounting bracket due to front cabin is around 1000kg. Therefore the load acting on the single mounting bracket will be 500 kg. i.e. 4905 N. It will act in vertically downward direction.

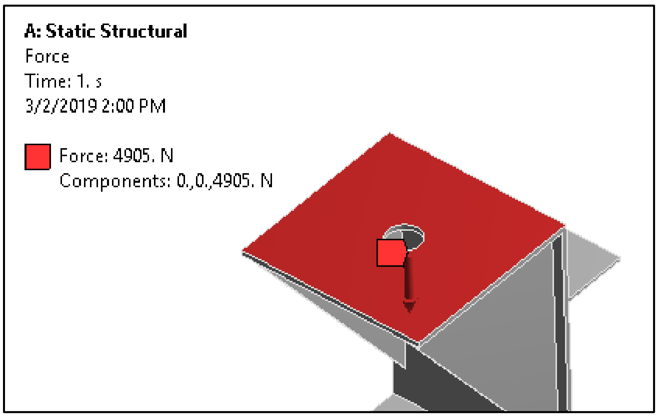

Fig16. Cabin load on Super Bracket

2) Load on Leaf Spring: The total Gross load on HCV is around 16 Ton i.e. 16000kg. So the total load on leaf spring that will act on single Super bracket will be 2 Ton i.e. 2000kg.It will act in vertically upward direction.

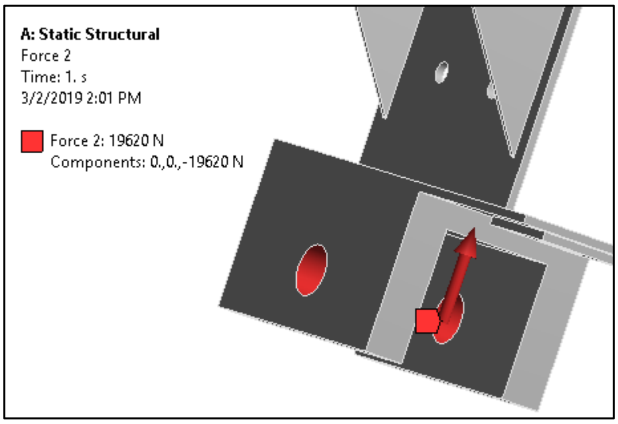

Fig.17 Suspension load on Super Bracket

3) Towing load: Towing force is force that is required to tow the vehicle after is failure or accident situations.

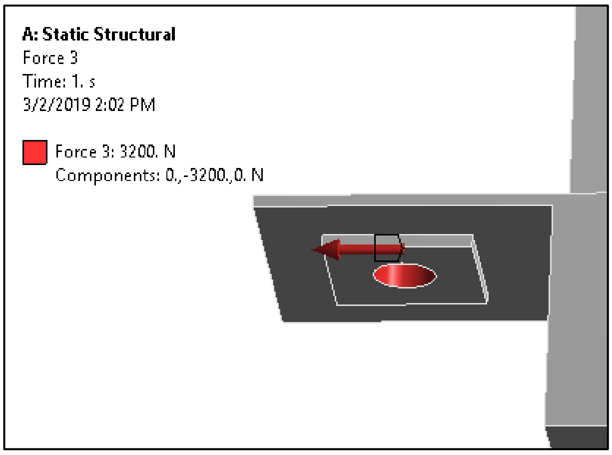

Fig.18 Towing load on Super Bracket

The fixed support in our super bracket geometry will be as shown:
The region in blue shows fixed support when load is acting on it.

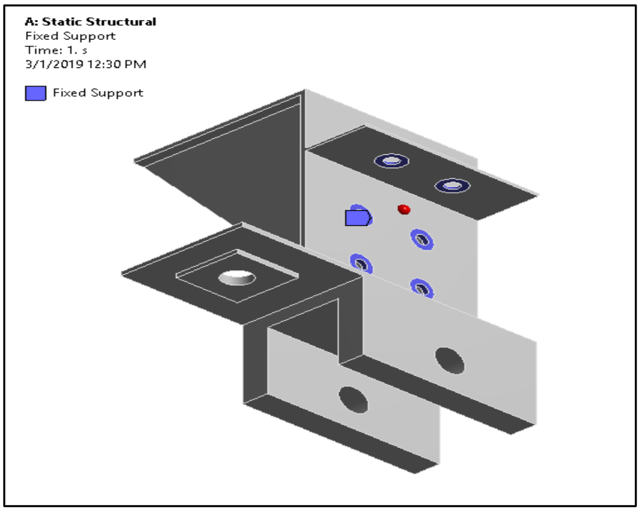

Fig19. Fixed support of Super Bracket

Total deformation is shown below:

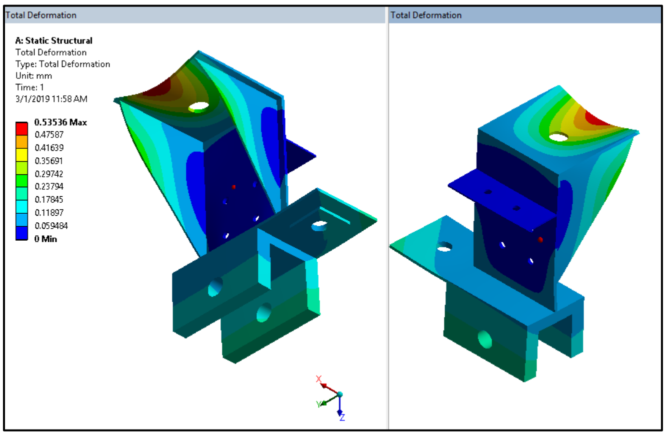

Fig.20 Total deformation of Super Bracket

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 0.53536mm, which is very lot

Equivalent (Von-mises) Stress:

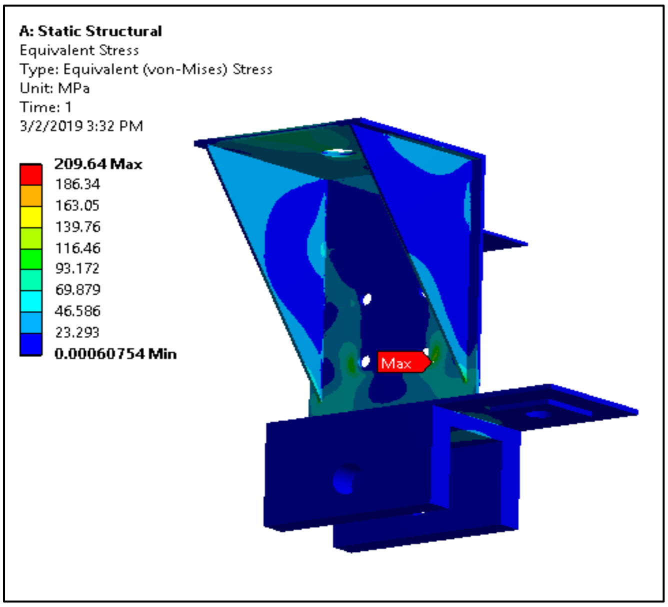

Fig.21 Equivalent Stress on Super Bracket

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 element away the Equivalent stress decreases from 209.64 MPA to 96.033 MPA

The detailed view near fixed edge is shown below:

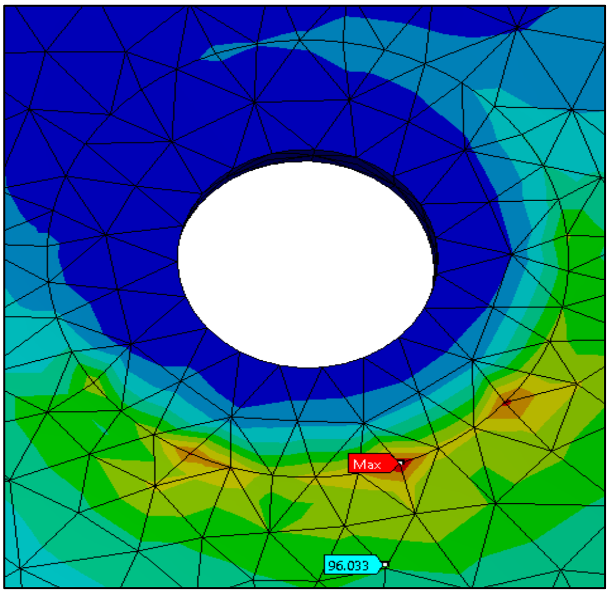

Fig22 Maximum Stress on Super Bracket

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 mass of bracket and increase its stiffness. So now we will perform Topology optimization on Super bracket with objective of minimizing mass and compliance of super bracket.

6. TOPOLOGY OPTIMIZATION

The conventional Topology Optimization formulation uses a finite element method (FEM) to evaluate the design performance. The design is optimized using either gradient-based mathematical programming techniques such as the optimality criteria algorithm and the method of moving asymptotes or non-gradient-based algorithms such as genetic algorithms.

Topology Optimization has a wide range of applications in aerospace, mechanical, bio-chemical and civil engineering. Currently, engineers mostly use TO at the concept level of a design process. Due to the free forms that naturally occur, the result is often difficult to manufacture. For that reason the result emerging from TO is often fine-tuned for manufacturability. Adding constraints to the formulation in order to increase the manufacturability is an active field of research. In some cases results from TO can be directly manufactured using additive manufacturing; TO is thus a key part of design for additive manufacturing.

In our case the objective and constraint in doing topology optimization are as follows
Objective:

- Minimize Mass
- Minimize compliance

Response Constraint:

Global von-Mises Stress =146.66 MPA

The maximum allowable stress is calculated based on fatigue life

$6_{all}=\frac{0.5*Sut}{FOS}$

Where,

Sut=Ultimate tensile strength=440 MPA

FOS=Factor of safety=1.5

Therefore, the maximum allowable stress =146.66 MPA

The result of topology optimization is shown below:

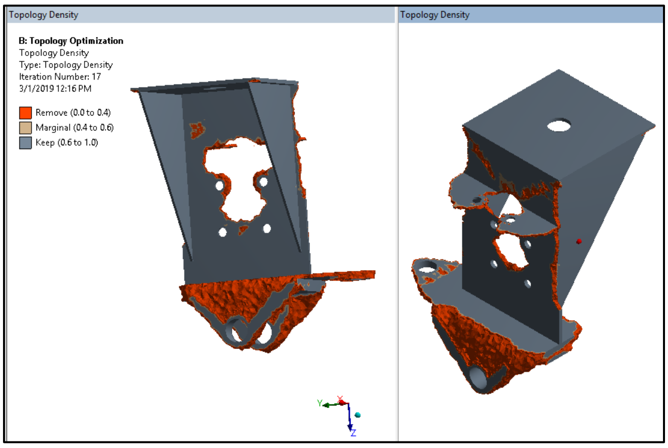

Fig.23 Topology Optimization

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 mass of new geometry is 9.428kg.

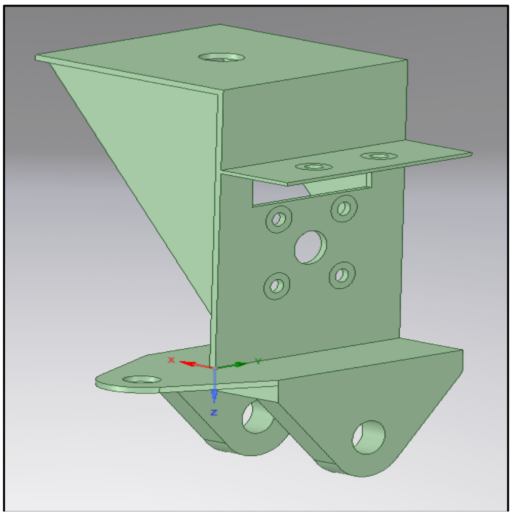

Fig.24 Geometry of Super Bracket in Space Claim

For FEA analysis of our Super Bracket the same procedure is applied as for old design. The new geometry of Super Bracket is meshed with the element size of 2mm and their number of nodes and element are as shown below:

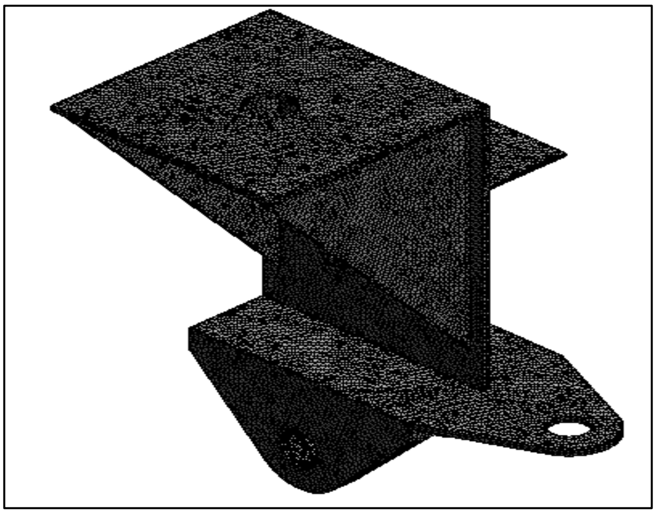

Fig.25 Mesh model Super Bracket

Statistics	
Nodes	236168
Elements	130950

Fig.26 Number of nodes and element

Forces acting on Support Bracket and their fixed support will be same as for old design as shown below:

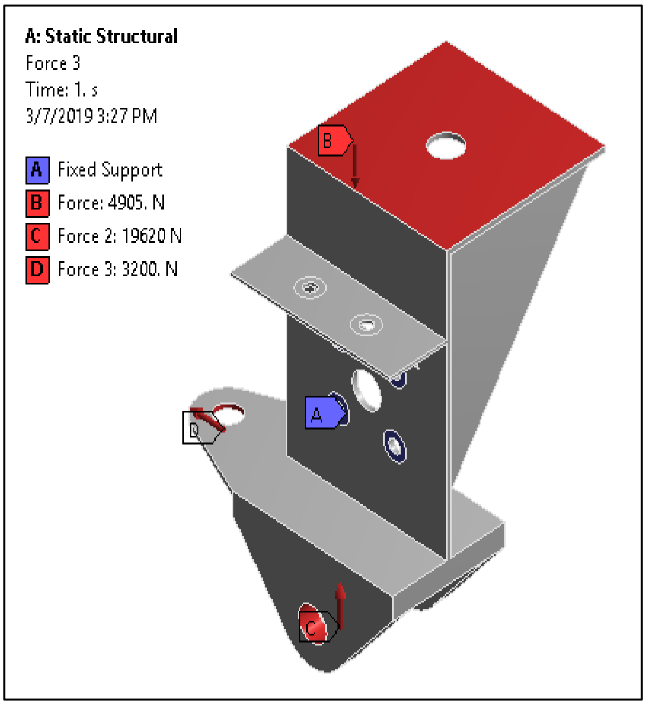

Fig.27 Constraints on Super Bracket

Due to application of load, the super bracket will go under deformation which is shown by Total deformation plot below:

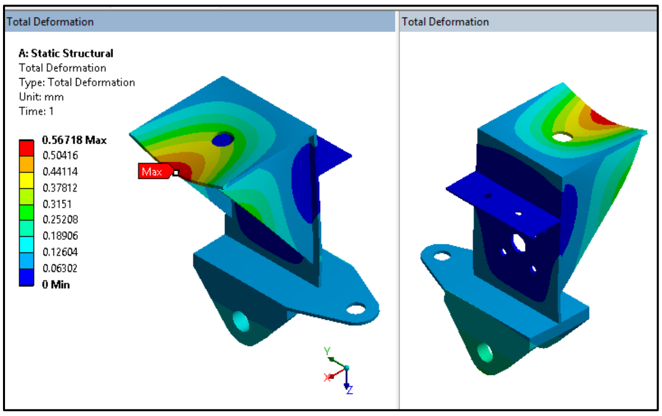

Fig.28 Total Deformation of Super Bracket

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

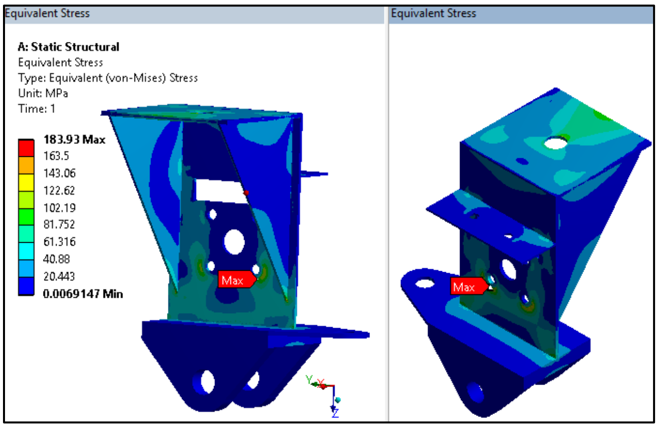

Fig.29 Total Deformation of Super Bracket

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 element away the Equivalent stress decreases from 183.93 MPA to 83.73 MPA.

7. RESULT SUMMARY

Sr No	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

8. FEA RESULTS:

Iteration	Stress(Mpa)	Deformation(mm)	Mass(Kg)
Baseline Super bracket	96.033	0.53536	12.721
Topology optimized Super bracket	83.73	0.56718	9.428

9. CONCLUSION COMMENTS

1. Maximum stress Observed for Suspension mounting bracket is 297.28 MPa. And maximum deformation is 0.544mm.
2. Maximum stress Observed for Cabin mounting bracket is 482.02MPa. And maximum deformation is 0.297mm.
3. Maximum stress Observed for Towing pin is 46.13 MPa. And maximum deformation is 0.144mm.
4. Total weight of Suspension mounting bracket, Cabin mounting bracket & Towing pin is 12.3 kg.
5. Stress produce in Baseline Super bracket is 96.033. Deformation is 0.53mm and mass is 12.72 kg
6. Whereas in Topology optimized Super bracket stress is 83.73Mpa. Deformation is 0.56mm and mass is 9.428 kg.
7. So we conclude that Topology optimized Super bracket is best in all aspect.

REFERENCES

1) Ms.Suvarna M Shirsath "Design & Weight Optimization of The Front Cab Mounting Bracket Of Truck" Resincap Journal of Science and Engineering Volume 2, Issue 8 August 2018ISSN: 2456-9976
2) Mr. Rajkumar Ghadge, Mr. Pankaj Desle "DESIGN AND WEIGHT OPTIMIZATION OF CABIN MOUNTING BRACKET FOR HCV" 2017 IJNRD Volume 2, Issue 7 July 2017 | ISSN: 2456-4184 IJNRD1707008 International Journal of Novel Research and Development
3) Jadhav Shashikant, Madki S. "Study and Analysis of Front Suspension Shackle Bracket for Commercial Vehicle" 2017 IJEDR | Volume 5, Issue 4 | ISSN: 2321-9939 IJEDR1704222 International Journal of Engineering Development and Research
4) Shashikant Jadhav, S. J. Madki "Optimization of Front Suspension Shackle Support using Finite Element Analysis" International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056 Volume: 04 Issue: 08 | Aug -2017 www.irjet.net p-ISSN: 2395-0072
5) Pushpendra Mahajan and Prof. Abhijit L. Dandavate submitted their research on "Analysis and Optimization of Compressor Mounting Plate of Refrigerator using FEA." International Journal of

Emerging Technology and Advanced Engineering (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 5, Issue 5, May 2015)

6) Shailesh Kadre, Shreyas Shingavi, Manoj Purohit, "Durability Analysis of HCV Chassis Using Fpm Approach."HTC 2011.
7) Cornelia Stan, Daniel Iozsa, Razvan Oprea "Study Concerning the Optimization of the Mounting System of the Truck Cab", ISBN: 978-960-474-383-4
8) Richard Ambroz, Ing. David Kollhammer, Ing. David Palousek. "OPTIMALIZATION OF CABIN MOUNTING" Conference of Diploma Thesis 2007 Institute of Machine design, Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology June 5-6, 2007, Brno, Czech Republic.
9) Paras Jain, "Design and Analysis of a Tractor-Trailer Cabin Suspension", SAE Paper No. 2007-26-047.
10) D. Murali Krishna, Dr. S. Madhu, "Design and Analysis of Tiltable Truck Cabin Floor", International Journal of Research Studies in Science, Engineering and Technology Volume 2, Issue 7, July 2014, PP 69-73.
11) Teo Han Fui, Roslan Abd. Rahman, "Statics And Dynamics Structural Analysis Of A 4.5 Ton Truck Chassis", Jurnal Mekanikal December 2007, No. 24, 56 – 67.
12) R. Prakash V. Kavinraj, A. Anandajayakumar S. Karthik "Design and Analysis of Leaf Spring Bracket in Air Suspension Vehicle Using FEA" International Journal of Innovative Research in Science, Engineering and Technology(An ISO 3297: 2007 Certified Organization) Vol. 2, Issue 11, November 2013 Copyright to IJIRSET
13) Vyankatesh D. Pawade and Pushkaraj D. Sonawane "Study of Design and Analysis of Air Conditioner Compressor Mounting Bracket." International Engineering Research Journal (IERJ) Special Issue 2 Page 4825-4829, 2015, ISSN 2395-1621
14) Mr. Dattatray Gavade, Prof. Satej Kelkar "Study & Analysis of Structural Behavior of Front Suspension Rear Shackle Bracket" IJSRD - International Journal for Scientific Research & Development| Vol. 2, Issue 12, 2015 | ISSN (online): 2321-0613