Design and Optimization of Integrated Super Bracket According to Stress and Vibration 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 - *Automobile vehicle is consisting of various components and assemblies. Study is focused on replacement of cab mounting bracket, suspension mounting bracket and tow hook with single component. New integrated design is then optimized through topology optimization to improve shape and reduce weight and cost by removing unnecessary material. Static analysis is followed by dynamic analysis to understand dynamic characteristics of component. Behavior of this integrated bracket for compression and tensile load is studied. Experimental modal analysis performed to compare numerical and experimental natural frequency. Dynamic analysis is an important tool for understanding the vibration characteristics of mechanical structures. It will help the researchers and engineers for the better design and development of engineering components. The experimental and numerical modal analysis of engineering component always provides an extreme contribution to our effort for better understanding and to control many vibration problems encountered in practice. Modal analysis, Harmonic Response analysis, Random Vibration Analysis performed to understand the vibration characteristics of integrated bracket using CAE tool. By performing these analyses, we found different natural frequencies and how the structure behaves with certain natural frequency. Only a few frequencies resonance can occur. While designing component we need to find response of these frequencies.*

Key Words: *CAE, Optimization, Integrated Design, Frequency.*

1. INTRODUCTION

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.

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 analyzed using FEA software, ANSYS. Further two methods of improving and optimizing the design to increase the natural frequency are illustrated and analyzed.

3. PILOT STUDY

Stress and Deformation data related to Suspension Mounting Bracket, Cabin Mounting Bracket, Towing Hook collected from various research available and reverse engineering experts. Tata 1613 vehicle is chosen as an area of research. All the data is gathered and analyzed thoroughly with the help of reverse engineering experts, research and development faculties. Outcomes are discussed below:

3.1. Suspension Mounting Bracket

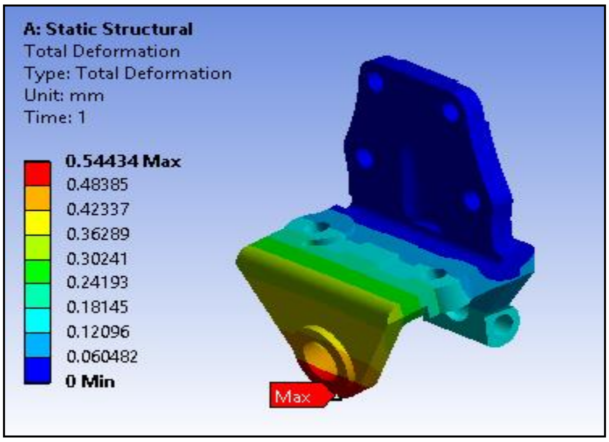


Fig. 3.1: Deformation of Suspension Mounting Bracket

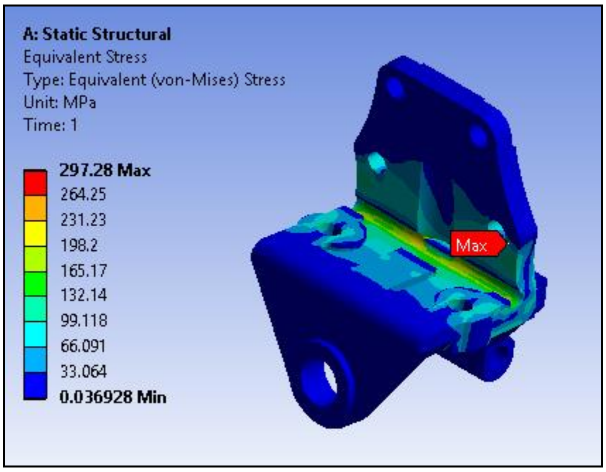


Fig. 3.2: Stress on Suspension Mounting Bracket

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 have been recorded as 6-ton load will be shared by the front axle and 10-ton loading will be shared by the back-side axle, as loading happens at the back end of the vehicle.

So, total of 6-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

$$W_1 \ = \ Weight \ \times g$$

$$W_1 \ = \ 2000 \times 9.81$$

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

Suspension mounting bracket with fully loaded condition shows maximum deformation of 0.5443mm at free mounting end. Stress or von mises stress 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.

3.2. Cabin Mounting Bracket

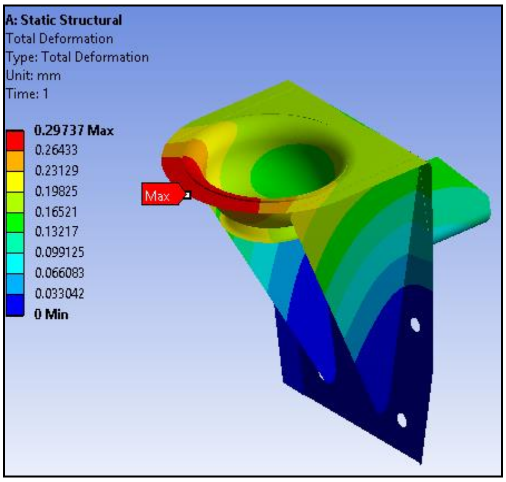


Fig. 3.3: Deformation of Cabin Mounting Bracket

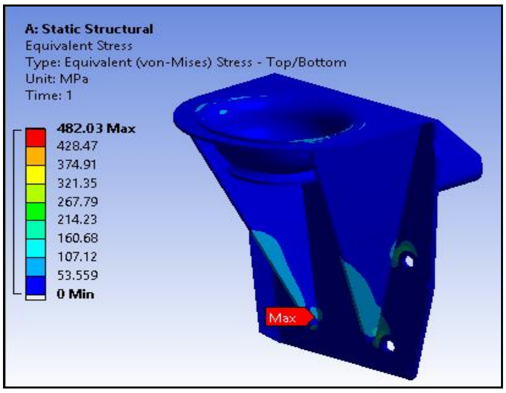


Fig. 3.4: Stress on Cabin Mounting Bracket

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.

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.

3.3. Towing Hook

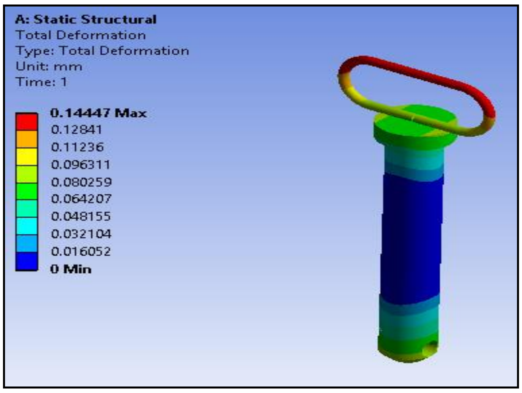


Fig. 3.5: Deformation of Towing Hook

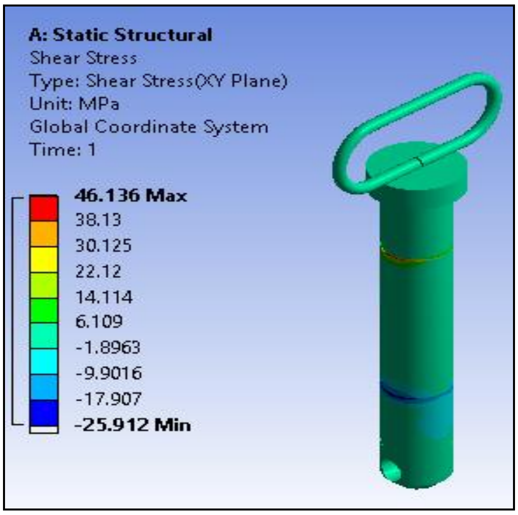


Fig. 3.6: Stress on Towing Hook

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.

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

4. CAD MODELLING OF BASELINE SUPER BRACKET

The modelling of super bracket is done in CAD software. The material properties used for super bracket is Mild Steel with following properties:

Density =7850 kg/m^3

Mass = 12.721 kg

Syt=370 MPA

Sut=440 MPA

Poison Ratio=0.3

Modulus of elasticity E = 2.1e5 MPA

The geometry of super bracket is shown below, which is derived from the space available at the location in truck chassis to combine these brackets together. Thicknesses and dimensions from the purchased and measured geometries area used to create many concepts and finally single manufacturable concept is finalized as shown below.

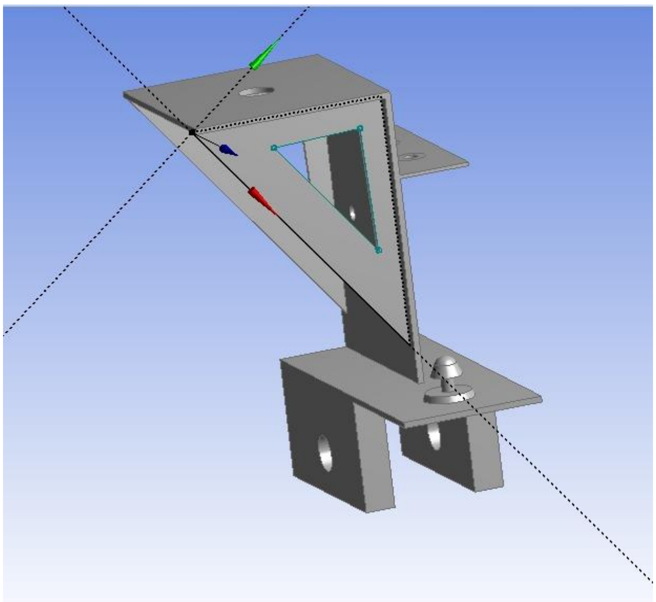


Fig. 4.1: Geometry of Super Bracket in Space Claim

4.1. Mesh Generation:

For analysis purpose meshing of super Bracket is done on Ansys workbench. Solid model is meshed with 3 D solid elements. The mesh size is used as 2mm which is based up on mesh sensitivity analysis performed on the previous analyzed components of similar style. The meshed Super Bracket appears as shown in the fig 4.2. Total number of nodes and element observed are recorded.

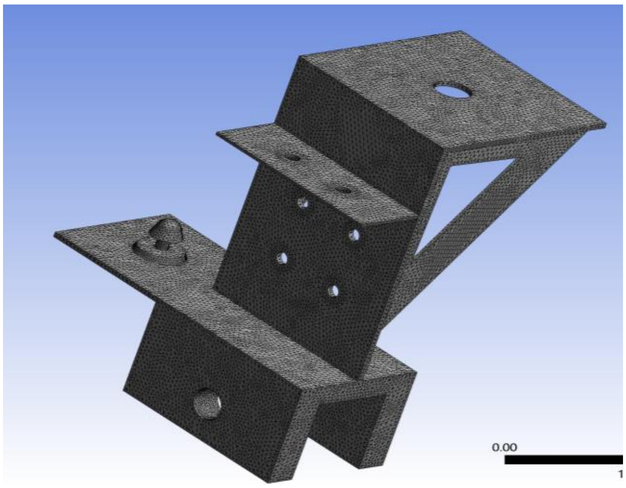


Fig. 4.2: Meshing of Super Bracket

Statistics	
Nodes	267219
Elements	149019

Fig. 4.3: Nodes and Element

4.2. Static Structural Analysis:

Static Structural analysis is done to find the Total deformation and Equivalent (von-mises) Stress in Super Bracket after the application of load on it.

The forces acting on Super bracket are as follows:
The fixed support in our super bracket geometry will be as shown:

- 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.

- 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.

- Towing load: Towing force is force that is required to tow the vehicle after is failure or accident situations. i.e., 3200N.

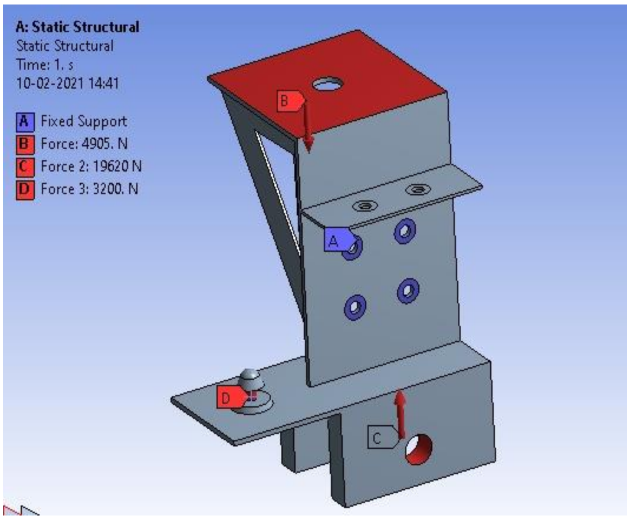


Fig. 4.4: Fixed support and Loads

4.2.1. Total deformation:

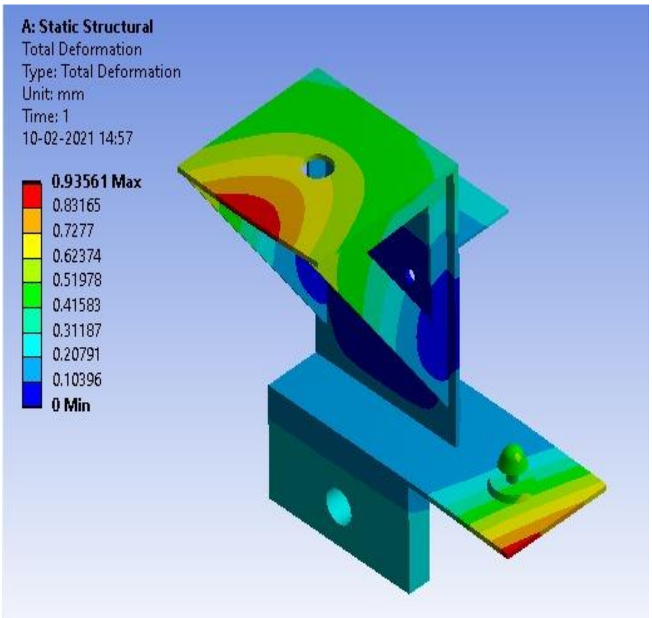


Fig. 4.5: Total deformation

The Total Deformation is shown the pattern of deformation that occurs. In True scale the deformation is negligible. The Maximum Deformation in bracket is 0.93561mm.

5.2.3. Equivalent (Von-mises) Stress:

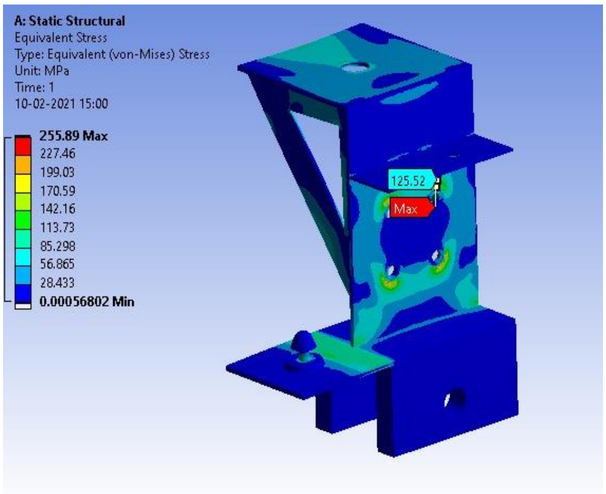


Fig. 4.6: Equivalent Stress

The Maximum equivalent stress is 255.89 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 255.89 MPA to 125.52 MPA. Static structural damage can be observed at fixed support where maximum stress observed.

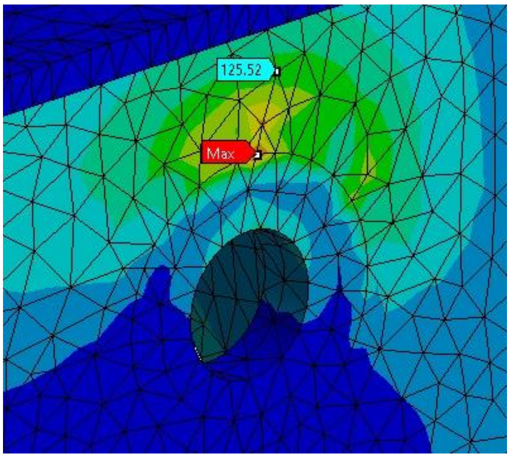


Fig. 4.7: Maximum Stress on Super Bracket

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.

4.3. Topology optimization:

Topology Optimization is optimization technique in Ansys that optimizes material layout within a given design space, for a given set of loads, boundary condition and constraint. 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 =150 MPA

The maximum allowable stress is calculated based on fatigue life

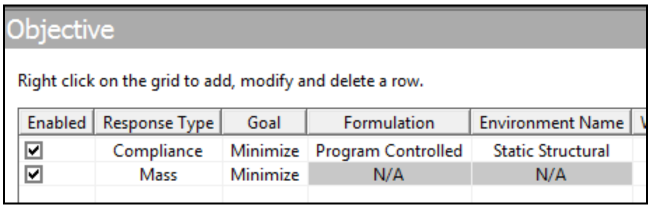
$$6_{all}=\frac{0.5*Sut}{FOS}$$

Where,

Sut=Ultimate tensile strength=440 MPA

FOS=Factor of safety=1.5

Therefore, the maximum allowable stress =150 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. 4.8: Topology objectives

4.3.1. Topology objectives:

Topology optimization removes the elements which has very less stress concentrations. For this purpose, few inputs such as boundary conditions, magnitude of forces are given as an input. Static structural analysis is used as a starting point for topology optimization. As explained the top part of bracket which is cabin mounting bracket is considered as an exclusive region. Fixed support regions are exclusion region for topology optimization.

Response	Global von-Mises Stress
Maximum	150 MPa
Environment Selection	All Static Structural
Suppressed	No

Table 1: Details of Response Constraint

The result of topology optimization is shown below:

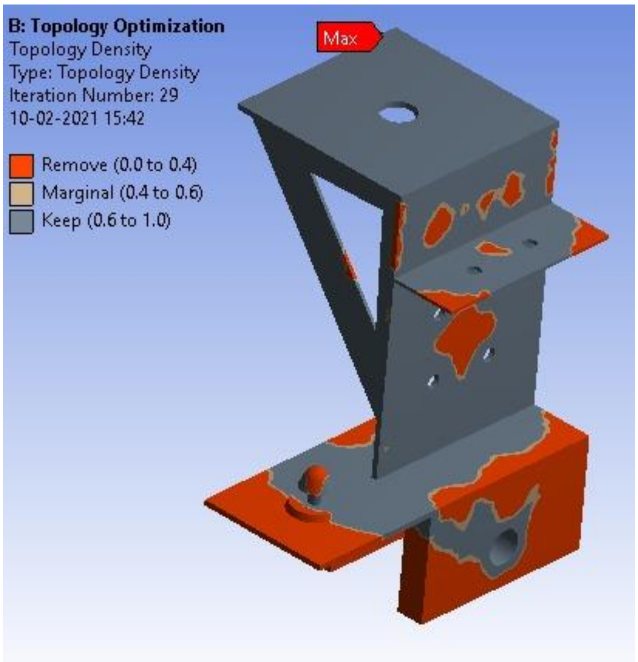


Fig. 4.9: Topology Density

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 fulfil our objective of minimizing mass and compliance of Super Bracket.

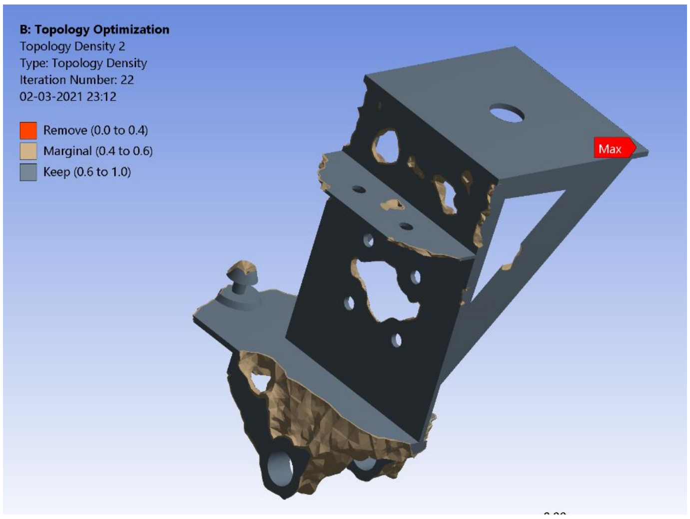


Fig. 4.10: Topology density

As shown in fig. 4.9 the red shaded elements are accordingly less stressed. This area is underutilization for topology optimization, after 22nd iterations approximate geometry is shown in fig. 4.10. It is not possible to manufacture exact same design suggested by topology optimization because of special purpose machining and costly operations involve, we can redesign it by considering ease of manufacturing.

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.

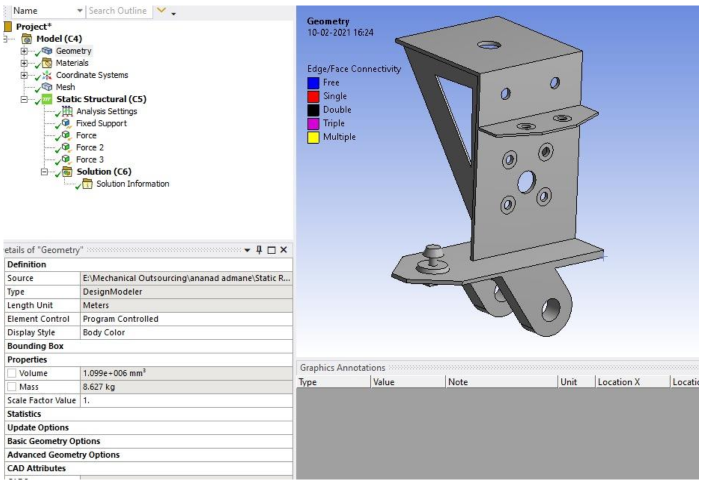


Fig. 4.11: Geometry of Super Bracket in Space Claim

For FEA analysis of our Super Bracket the same procedure is applied as for old design. In fig. 4.11. clearly mansions mass 8.627 kg for optimized version of super bracket.

4.4. Static Analysis of topology optimized Super Bracket

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:

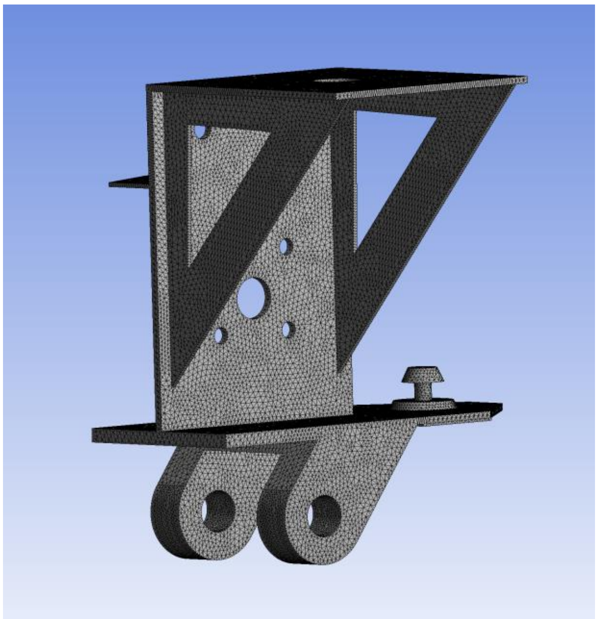


Fig. 4.12: Meshing of Super Bracket

Statistics	
Nodes	282727
Elements	157199

Fig. 4.13: Number of nodes and element

4.4.1. Static Structural- Total Deformation

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

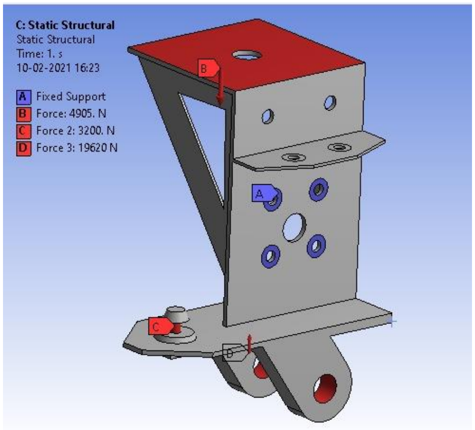


Fig. 4.14: Constraints on Super Bracket

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

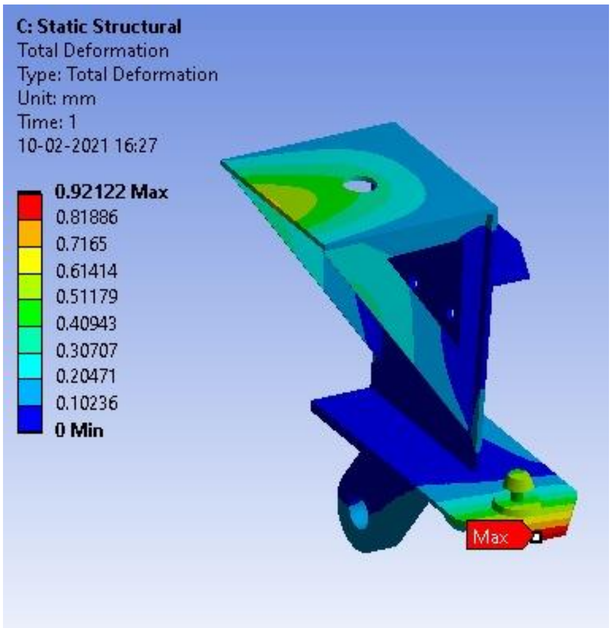


Fig. 4.15: Total Deformation

The Maximum Deformation in bracket is 0.92122 mm, which is 0.01394 mm less than baseline design.

4.4.2. Equivalent (Von-Mises) Stress on New Design:

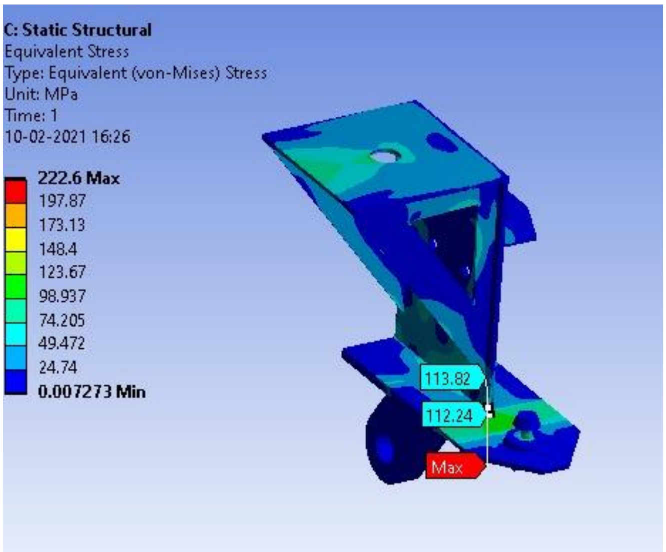


Fig. 4.16: Equivalent Stress

The detail view of maximum Equivalent Stress near the fixed support is as shown below;

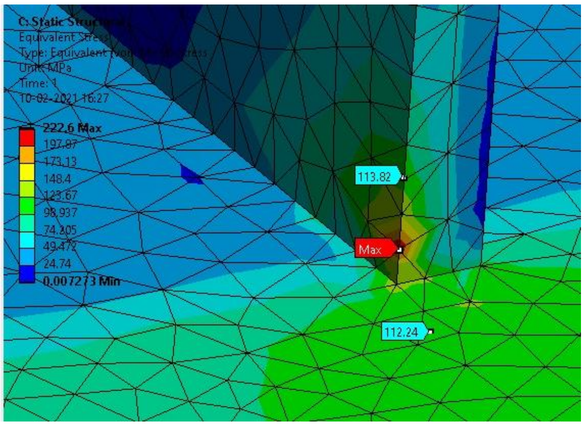


Fig. 4.17: Detail view of maximum Equivalent Stress

The Maximum equivalent stress is 222.6 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 222.6 MPA to 112.24 MPA.

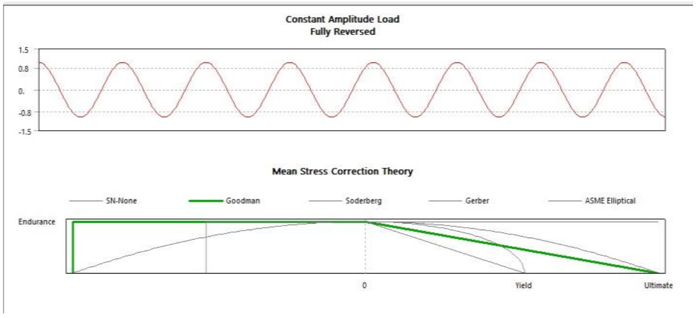


Fig. 4.18: Mean Stress Correction Theory

In fatigue tool by applying fully reversed type of loading and according to Goodman's Criteria area under the curve indicate that the material won't fail for given stress cycle and area above curve represents likely failure.

4.4.3. Safety Factor

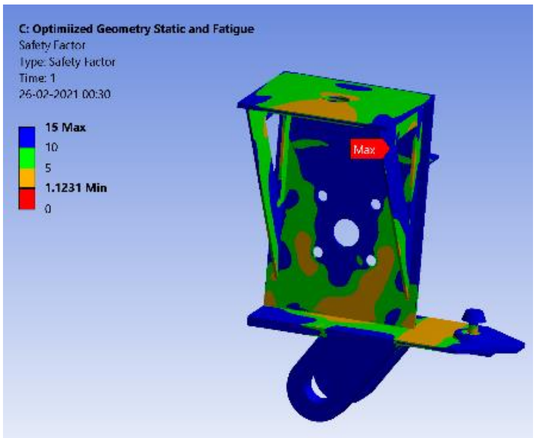


Fig. 4.19: Safety factor

The factor of safety is representation of how component will behave if subjected to maximum load.It should not less than 1. The blue region has factor of safety 10-15 ,green region 5-10 and orange region indicates that factor of safety is 1.1231 to 5. As per the results it is clear that optimized super bracket not having factor of safety less than 1 at any region .

4.5 Modal Analysis

Modal analysis helps to calculate the natural frequencies of system so we know which frequencies can be destructive and dangerous for system as component behaves more aggressively at natural frequencies that causes resonance and damage entire system. It helps to identify various modes of vibrations as well as the frequencies at which those modes are created. In modal analysis we need not to apply any external load but it is first step of dynamic analysis. Modal analysis determines the vibration characteristics (natural frequencies and mode shapes) of the structure or a machine component.

It can also serve as a starting point for another analysis, such as harmonic response analysis, random vibration analysis spectrum analysis. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. Structures vibrate or deform in particular shapes called mode shapes when being excited at their natural frequencies.

Modal testing is a common method of characterizing the vibrations of a structure by imparting a known force and measuring the response of the structure. By measuring both the input to the structure and the response, the frequency response of the structure can be calculated. Calculating the frequency response over multiple locations, either simultaneously or individually, will yield data that can be used to estimate the dynamic response of the structure.

4.5.1. Free - free boundary condition

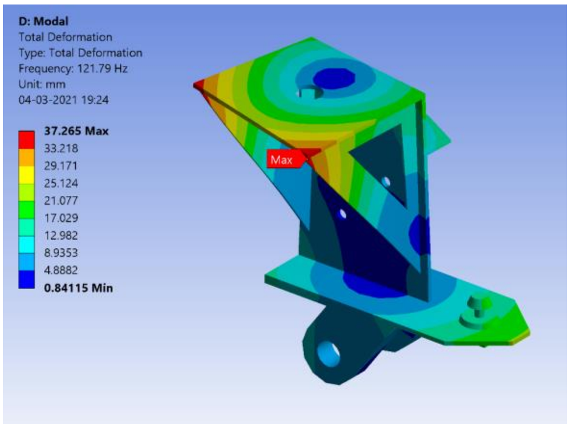


Fig. 4.20: Free Free Modal Analysis

Modal analysis performed by free free boundary condition and at 121.79 Hz maximum total deformation is 37.26 mm. This is considered as a natural frequency at free boundary condition.

4.5.2. Boundary Condition - Fixed support

Under typical operation conditions a structure will vibrate in a complex combination which consists of all mode shapes. However, by understanding each mode shape the we can then understand all the types of vibrations that are possible. Modal analysis also transfers a complex structure that is not easy to perceive, into a set of decoupled single degree of freedom systems that are simple to understand. Identification of natural frequencies, modal damping, and mode shapes of a structure. Now we perform further process to find six natural frequencies and mode shape for each frequency.

By applying fixed supports now we can find number of natural frequencies. Here we are considering six number of frequencies and for each frequency componant shows six different modes of deformations as given below .

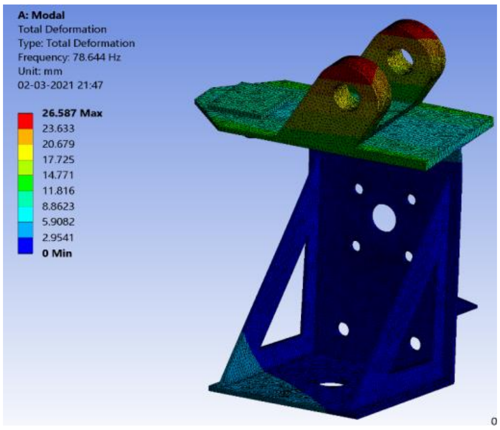


Fig. 4.21: Mode Shape 1

At the frequency 78.644 Hz , at the top section of super bracket which is suspension mounting region, here mode of failure with maximum deformation of 26.58 mm is noted at mode shape 1.

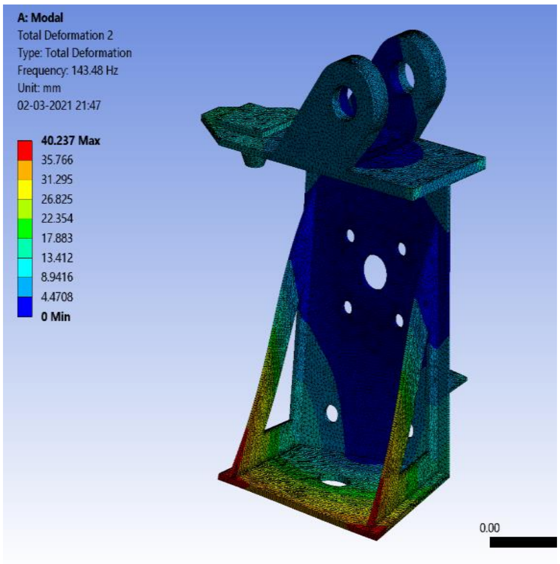


Fig. 4.22: Mode Shape 2

Mode shape 2 shows that the bottom part of super bracket which is designed to replace cabin mounting bracket, this part shows twisting with maximum deformation of 40.237 mm at frequency 143.48 Hz.

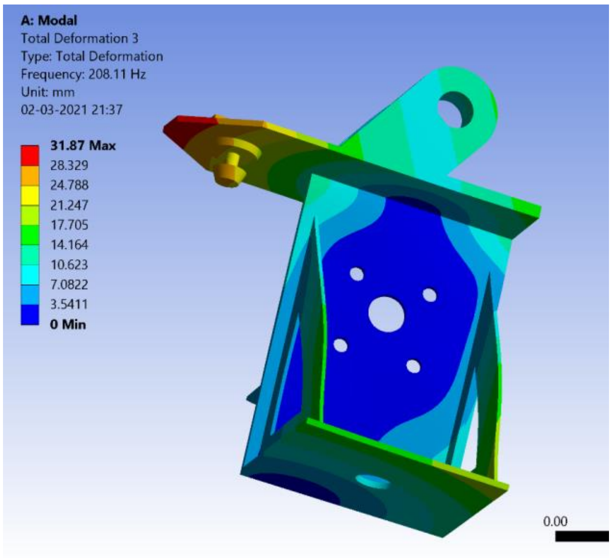


Fig. 4.23: Mode Shape 3

At ftequency 208.11 Hz third mode shape is formed which indicates possibility of failure at the towing pin of the super bracket. The mode of failure is tesile in nature with maximum deformation of 31.87 mm.

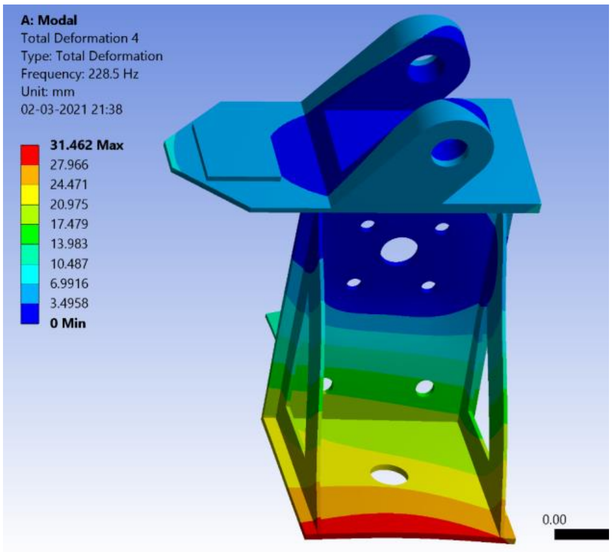


Fig. 4.24: Mode Shape 4

At 228.5 Hz maximum deformation is occur at the bottom of superbracket which is cabin mounting bracket. Maximum total deformation 31.462 mm observed in this region.The upside section of super bracket there is not significant deformation. i.e. 0 to 13.983 mm.

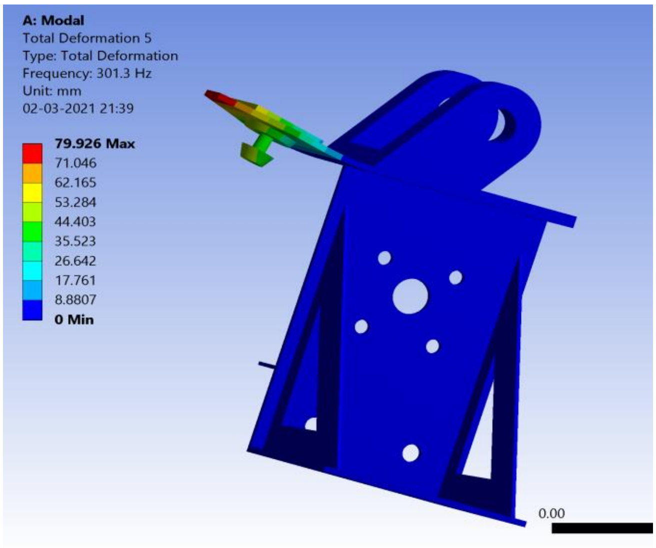


Fig. 4.25: Mode Shape 5

Mode shape 5 again shows the failure possibility at the towing pin section of super bracket as shown in mode shape 3 but here failure is bending in nature rather than tesile. This mode shape is at 301.3 Hz. frequency with maximum deformation 79.926 mm.

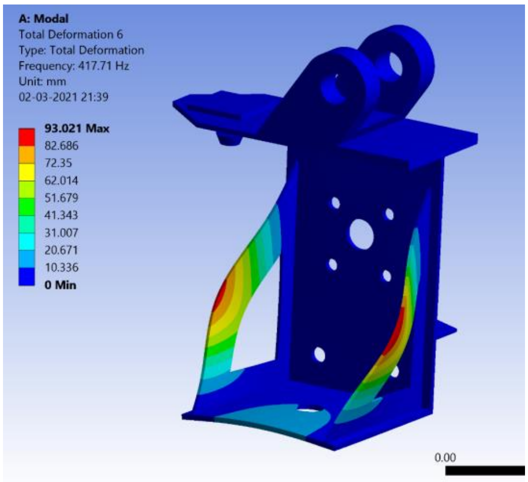


Fig. 4.26: Mode Shape 6

Sixth mode shape is obtained at the 417.71 Hz. frequency and shows the bending failure of ribs of superbracket in outward direction and maximum deformation observed is 93.021 mm. Table 2 represents the magnitude six frequencies and maximum total deformation obtained at it.

Mode	Frequency [Hz]	Total deformation(Max) (mm)
1	78.644	26.587
2	143.48	40.237
3	208.11	31.87
4	228.5	31.462
5	301.3	79.926
6	417.71	93.021

Table 2: Mode Shape Frequency and Total Deformation

4.6. Harmonic Response Analysis

In harmonic response analysis load applied is well deterministic as we know the value of load. Its behavior is harmonic in nature. Harmonic response analysis is used to determine the steady-state response of a linear structures to loads that vary sinusoidally(harmonically) with time, thus enabling to verify whether or not our design will successfully overcome resonance, fatigue and other harmful effects of forced vibrations.

Purpose of harmonic response analysis is to find magnitude of frequencies where higher magnitude of displacement or amplitude can occur and can generate resonance.

Frequency Spacing	Linear
Range Minimum	0 Hz
Range Maximum	500 Hz
Solution Intervals	50

Table 3: Harmonic Response Range

4.6.1. Harmonic Response Analysis -Total Deformation

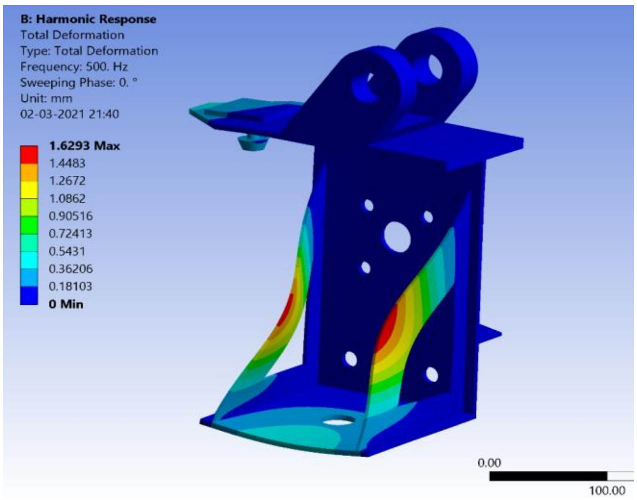


Fig. 4.27: Harmonic Response (total desformation) at 500 Hz

4.6.2. Harmonic Response Analysis - Equivalent Stress

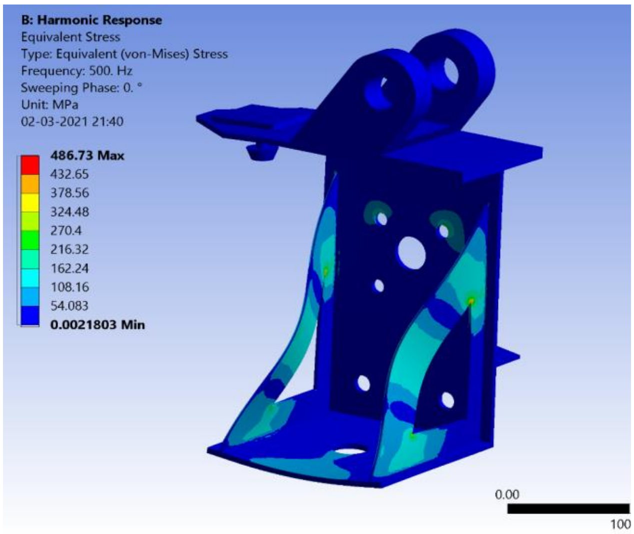


Fig. 4.28: Harmonic Response (Equivalent stress) at 500 Hz

By performing modal analysis, we found different natural frequencies and how the structure behaves with certain natural frequency. Only a few frequencies resonance can occur. While designing component we need to find response of these frequencies. Modal analysis used as a starting point of harmonic response analysis. At 500Hz equivalent stress 486.73 MPa and total deformation 1.629 mm. is observed.

4.6.3. Frequency Response Curve

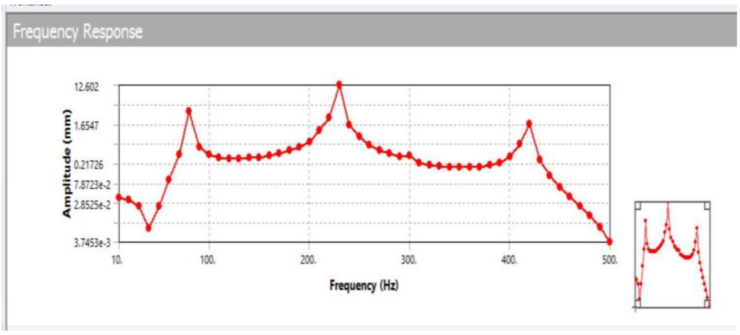


Fig. 4.29: Frequency Response Curve

Tabular Data

	Frequency [Hz]	Amplitude [mm]	Phase Angle [°]
1	10.	3.7503e-002	0.
2	20.	3.2919e-002	0.
3	30.	2.3916e-002	0.
4	40.	7.5038e-003	0.
5	50.	2.3895e-002	180.
6	60.	9.4578e-002	180.
7	70.	0.34214	180.
8	80.	3.1587	0.
9	90.	0.50824	0.
10	100.	0.34911	0.
11	110.	0.29938	0.
12	120.	0.28143	0.
13	130.	0.27873	0.
14	140.	0.28727	0.
15	150.	0.30064	0.
16	160.	0.32721	0.
17	170.	0.36626	0.
18	180.	0.42451	0.
19	190.	0.51484	0.
20	200.	0.66152	0.
21	210.	1.2214	0.
22	220.	2.2796	0.
23	230.	12.602	180.
24	240.	1.613	180.

Fig. 4.30: Frequency response Tabular data

Fig. 4.29 shows frequency response curve which is graph generated Amplitude (mm) vs Frequency (Hz) by taking range of 0 Hz to 500 Hz between 50 solution intervals. While fig.4.30 shows tabular data, which represents frequencies and amplitude with respect to it. From the curve it is clear that maximum amplitude 12.602 mm occur at the frequency 230 Hz. At this frequency maximum chances of resonance predicted by harmonic response analysis.

4.7. Random Vibration

In random vibration analysis load is unpredictable because it is based on probability concept and may differ with time and direction. This analysis enables to determine the response of structures to vibration loads that are random in nature. Time history of load is unique every time the vehicle runs over the same road.

Hence it is not possible to predict precisely the value of the load at a point in its time history. Such load histories however can be characterized statistically (mean, root mean square, standard deviation.) The frequencies content of the time history(spectrum) is captured along with the statistics and used as the load in the random vibration analysis.

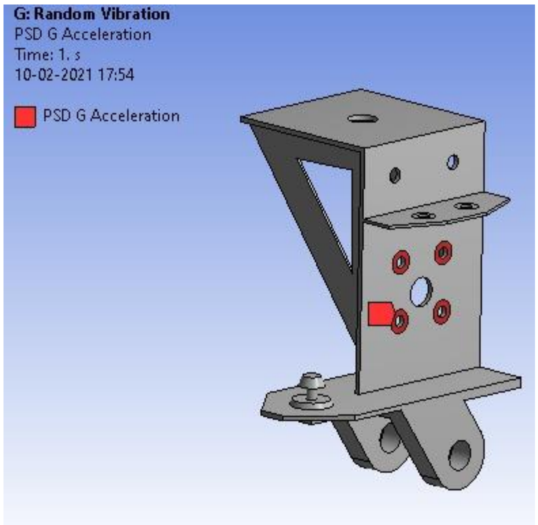


Fig. 4.31: Random Vibration PSD G Acceleration fixed supports

The spectrum for historical reasons is called Power Spectral Density or PSD. In random vibration analysis since the input excitations are statistical in nature so, output response such as displacements stress and so on are not an accurate value but the probability based.

The excitation is applied in the form of power spectral density. The PSD is a table of spectral values vs frequency. These analyses based on the mode superposition method. Hence a modal analysis that extracts the natural frequencies and mode shapes is a prerequisite.

4.7.1. Random Vibration Analysis – Directional Deformation

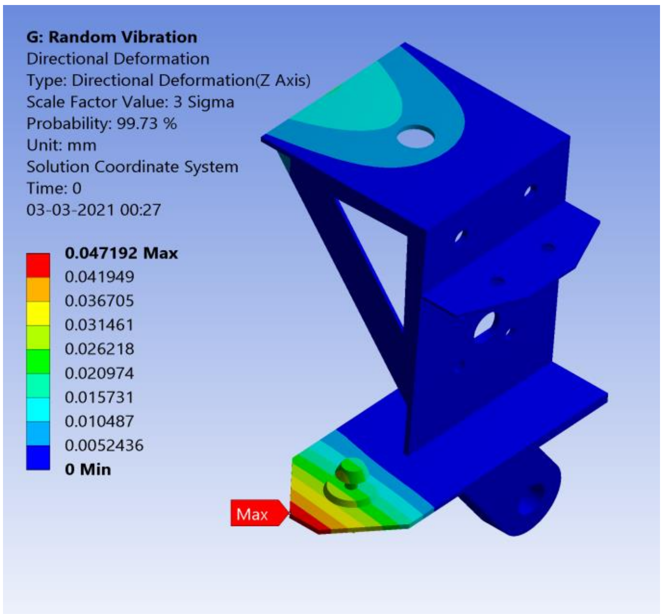


Fig. 4.32: Random Vibration Directional Deformation

Fixed supports taken for random vibration analysis we observed probability that at 99.73% of time the result in term of deformation along z axis is less than 0.047192 at the scale factor value 3 sigma.

4.7.2. Random Vibration Analysis – Equivalent Stress

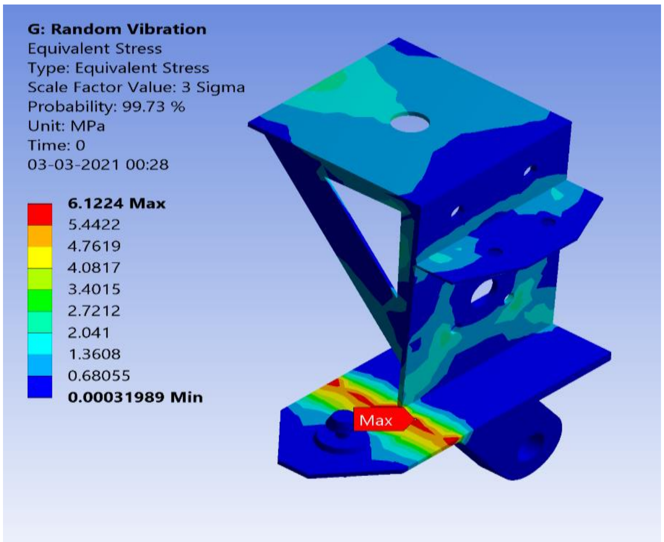


Fig. 4.33: Random Vibration Equivalent Stress

Result of random vibration equivalent stress can be expressed as probability that at 99.73% of time the result in term of equivalent stress at scale factor 3 sigma is less than 6.1224 MPa. Failure can be occurred at the region of towing pin section of super bracket at which maximum amount of stress concertation seen.

4.8. Few techniques to prevent resonance

- When the frequency of the excitation coincides with a natural frequency of the structure, the structure may exhibit very high level of vibrations that can lead to structure fatigue and failure.
- When it is known that the excitation force coincides with one of the natural frequencies found in the modal analysis, the structure can be redesigned or modified to shift the natural frequency away from the excitation frequency.
- Structural elements can be added so as to increase the stiffness of the structure or mass can be increased or decreased. By doing that, the excitation frequency will no longer fall on the natural frequency of the structure. These techniques can be applied to move the natural frequencies away from the excitation force frequency.
- Other techniques of vibration suppression include increasing the damping of the structure by changing the material or adding viscoelastic material to the surface of the structure.
- Vibration absorbers can be added which are tuned to the excitation force frequency to yield large vibrations in the absorber and reduce the vibration in the structure.

5. EXPERIMENTAL VALIDATION

5.1 Manufacturing

The optimized super bracket is manufactured by cutting operations, joined together by welding. The sheets of required thickness are cut as per the requirement by laser cutting operation. The small pieces welded together to form one single piece known as our super bracket. The pin part is made on lathe. Basic three view dewing is shown below.

Fig. 5.1: Manufactured model

5.2 Experimental Strength Test of Super Bracket

The Two types of loads were applied on component by UTM:

- The Axial load along the vertical axis (Resultant load due to Cab mounting loading and Suspension loading): 14715 N
- The Tensile load for towing: 3200 N

Fig.5.2. UTM Test Setup

UTM Experiment Results are as follows:

Type of loading	Maximum Deformation observed (mm)
Compressive	0.858
Tensile	0.36

Table 4: Results of UTM Test

5.3 Experimental Modal Analysis Using FFT

Experimental modal analysis (EMA) enables engineers and researchers to get a well understanding of dynamic properties of structures. DEWEsoft X2 SP7 software is used to perform EMA. In the past two decades, methodology based on FFT approach has received a lot of attention, here we performed EMA to find out natural frequency of super bracket. The finite element modelling software, ANSYS workbench, Finite element analysis based modal analysis of Super Bracket is discussed in previous section 4.5. Thus, the obtained natural frequency at the free-free boundary condition compared with result obtained by experimental method.

The experimental setup was prepared to get natural frequency at free-free boundary condition, the bracket is placed freely to carry out the experiment. The care has been taken that the support does not offer any resistance to the motion providing virtually free boundary conditions. The time domain signal is converted into the frequency domain using Fast Fourier Transformation with the help of MATLAB program. The bracket was excited with the help of impact hammer and response was captured using prob.

Fig. 5.3: Experimental setup for FFT test

Fig. 5.4: FFT Test Results

Method	Natural Frequency
FEM	121.79 Hz
EMA	126.95 Hz

Table 5: FEM and EMA Frequency

The experimentally obtained natural frequency 126.95 Hz indicated in Fig. 5.4. and natural frequency determined by FEM is 121.79 Hz. The comparison of numerical results, and experimental results shows that the results are all well within the reasonable error margin that is 4.23%.

The modal analysis may be helpful in dynamic analysis, developing and optimizing the design of complex engineering structure and component. It will also help the researchers and engineers for the better design and development of engineering components. The experimental and numerical modal analysis of engineering component always provides an extreme contribution to the effort for better understanding and to control many vibration problems encountered in practice.

6. RESULT

Sr No.	model	Load (N)	Max Stress (MPa)	FEA Max Def (mm)	Weight (kg)	Cost (Rs)
1	Suspension mounting bracket	19620	297.28	0.544	7.57	1500
2	Cabin mounting bracket	4905	482.02	0.297	4.21	1850
3	Towing pin	3200	46.13	0.144	0.98	850
				Total	12.76	4200

Table 6: Result Summary for Three Basic Components.

The cost estimation for three basic components, suspension mounting bracket has cost 1500Rs. Cabin mounting bracket has cost 1850 Rs, and towing pin has cost 850 Rs. Total cost of three component will be 4200 Rs.

Iteration	Load (N)	Stress (MPa)	FEA Max Deform. (mm)	Mass (Kg)	Cost (Rs)
Baseline Super bracket	A-4905 B-19620 C-3200	255.8	0.9356	12.721	3240
Topology optimized Super bracket	A-4905 B-19620 C-3200	222.6	0.9212	8.627	2590

Table 7: Result summery for Super bracket

7. CONCLUSIONS

- Design integration performed on three basic components by using reverse engineering and creation of CAD models.
- The alternative design named as a baseline super bracket is designed by combining basic components and successfully analysed using Ansys workbench.
- Topology optimization performed on the super bracket for weight optimization.
- FEA analysis performed on this topologically optimized super bracket shows maximum value of stress 222.6 MPa and total deformation 0.9212 mm.
- All the stress values are well within the acceptance criteria.
- Total weight of Suspension mounting bracket, Cabin mounting bracket & Towing pin is 12.76 kg and Topology optimized Super bracket is 8.627 kg. Weight is reduced from 12.76 kg to 8.627 kg which is 32.39 % reduction.
- Basic three component total cost was 4200 Rs. and cost of our topology optimized super bracket is 2590 Rs. Which is 38.33% reduction.
- The experimental strength testing of super bracket shows the maximum deflection of 0.85 mm for compression load of 14.71 KN and 0.36 mm for tensile load of 3.2 KN.
- The experimental modal analysis (Free free) also performed which concludes that the natural frequency of super bracket is 126.95 Hz and natural frequency obtained by FEM is 121.79 Hz., the results are well within acceptable error margin 4.23%.
- So, we conclude that Topology optimized Super bracket is best in all aspect.
- Further scope for optimization is to do the more detailed dynamic analysis. It will improve the design as well as lifecycle of component.

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, “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. Rajani R. Mhatre1, R. B. Gunale, ("Design of HCV Super Bracket by Topology Optimization Technique for Weight Reduction and Strength Enhancement"), International Journal for Research in Applied Science & Engineering Technology (IJRASET) ISSN: 2321-9653 Volume 7 Issue IX, Sep 2019.
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
15. Mr. Admane Anant Dnyaneshwar, Mr. Satej Kelkar, ("Design and Weight Optimization of Integrated Super Bracket According to Stress Analysis.") International Research Journal of Engineering and Technology (IRJET) ISSN: 2395-0056 Volume: 07 Issue: 08 | Aug 2020
16. 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