International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 05 Issue: 07 | July - 2021 ISSN: 2582-3930

DYNAMIC ANALYSIS AND OPTIMIZATION OF TRUCK STEPNEY BRACKET

*Vivek Joshi and Guide Prof Desale D.C

Padmabhooshan Vasantdada patil Institute of Technology-[PVPIT] Bavdhan,Pune

---***---

Abstract- The advancement of Computer Aided Engineering (CAE) makes industry to perform dynamic/static analysis of vehicle components in actual working environment. In the present work, A chassis mounted bracket of Eicher truck has been considered for dynamic analysis using the actual load vibration excitation. The bracket is used as a foundation to support heavy spare tyre and mounted on the chassis. The dynamic forces subjected on the bracket can generate high stresses in the bracket, which also influences by the speed and different road conditions. High stresses may lead to structure failure or fatigue failure of the bracket. Generally, the components are overdesign to avoid such failures due to dynamic loading, which in turn increases the overall weight of component/vehicle and affect the performance of the vehicle. In this project, a methodology is developed to dynamically analyze the mounting bracket using real time vibration excitation induced on the support structure of bracket. The end effect of the excitation on the excitation on the bracket is further used to optimized the bracket. The mounting bracket has been designed and analyzed using commercial packages CREO and ANSYS Workbench respectively. Real time vibration signal is acquire in the form of acceleration on the base of bracket and used as an input dynamic force to the support of the bracket to perform transient dynamic analysis. From the different, it is noted that the chassis mounted bracket is overdesigned and, hence, optimization of bracket is performed to reduce its mass, while keeping the strength unaltered.

Keywords— A chassis mounted bracket , ANSYS, Strain gauge.

I. INTRODUCTION

Vibrations An automotive heavy vehicle like truck requires a number of structural parts to hold various components. Engine mounting brackets, suspension bracket, chassis mounted bracket etc. are some of the example of such structural parts. These structural brackets are subjected to various kind of excitation originated from various sources such as periodic excitation from the engine vibration, unbalance in the tyre assembly, random loads generated from the road profile etc. Designing of these structural components against the dynamic loading is an important aspect in overall design of the vehicle. Dynamic loading forces structural member to vibrate, which may lead to fatigue failure of the component. It also influences the performance of the vehicle. In many cases, the components are overdesign to avoid failure due to dynamic loading, which in turn increases overall weight of the vehicle and affects the performance of the vehicle. Selection of a heavy bracket also increases the cost. A significant research is underway to optimize the weight of such structural component. The current investigation is also related to the optimization of an automotive bracket against the dynamic loading measured from the actual load.

VIBRATION IN MECHANICAL SYSTEM

Vibration is an oscillating motion of a system or a body or an assembly of bodies abut its mean or equilibrium position. Vibration occurs in the structure or any system when it is displayed from its equilibrium position by an external excitation. A typical vibrating system is consist of an inertial mass and an element that provide restoring force. In most of the application, vibration are an undesirable, it may influence in many ways as:

- Increase Stress level
- Decrease Fatigue life
- It leads to the discomfort to passengers in vehicles.

What is fundamental frequency:

The fundamental frequency is just the lowest possible frequency among all the natural frequencies of vibration of an object. In a natural vibration, you just excite the object once. Then it will vibrate for a while. Those are the “natural” vibrations of the object.

Random Vibration Testing Vibration can produce objectionable operating characteristics, noise, wear and physical distortion by loosening parts or by causing motion between parts in a specimen, which can result in fatigue and failure of mechanical parts.

Fig 1 Truck stepney bracket

II. LITERATURE REVIEW

Sagar Inglea, Nikhil Tilakpurea, V.Narayanamurthyb, S.M.Hussaini et.al[1], Interface structures in shuttle's and flight vehicles experience mechanical stuns from an assortment of sources, for example, stage detachment, discharge activities and abrupt outer unsettling influences. The flight interface sections should be planned, examined and tried to withstand these stun loads representing brief terms. Because of high solidarity to-weight proportion and protection from consumption carbon fiber built up polymer (CFRP) is famously embraced as flight interface sections and are supplanting metals. The reaction of CFRP flight interface sections under stun stacking isn't completely perceived. This paper presents transient stun investigations performed on normal flight interface sections, for example, level rectangular and L-shape CFRP sections with fiber directions in 33 diverse stacking groupings for fixed-fixed and cantilever limit conditions; and exposed to drive stacking. Results showed that the stacking succession [90/0/0/90]s demonstrated best for rectangular fixed and cantilever conditions while stacking arrangement [90/90/90/90]s yielded best outcomes for L-formed cantilever and fixed-fixed limit conditions with least response load at fixed end.

Mr. Kaustubh Shete. et.al [2], In the traditional system spare tires in transports are frequently put away in an extra tire well – a rooftop transporter region over the rooftop a vehicle, generally in the middle, where the extra tire is put away while not being used. In many transports, the extra tire isn't gotten with a screw and wing-nut style clasp. While this is as yet the most normally utilized strategy for putting away the extra wheel in a business vehicle like transport, trucks and so on There are numerous weaknesses of this technique. To defeat the burdens of this customary instrument we have planned and fostered an 'Progressed spare wheel transporter' which can be utilized to store the extra wheel(s) in a vehicle. This system utilizes electrically controlled engine to lift and lower the extra haggle a reasonable securing instrument to secure the extra wheel in the extra wheel section after the difference in tire; it is additionally deliberately situated at the lower part of the transport to work with simple and more secure tire evolving.

Tamzid Ahmed ,Norma Ab Rahman , et.al [3], To look at the orthodontic section debonding constrain and evaluate the section disappointment design clinically between various teeth by an approved model deboning gadget. Materials and Method. Thirteen (13) patients toward the finish of complete fixed orthodontic treatment, anticipating for section expulsion, were chosen from the rundown. An aggregate of 260 sections from the focal incisor to the second premolar in the two jaws were debonded by a solitary clinician utilizing an approved model holding gadget furnished with a power delicate resistor (FSR). Mean section holding powers were specified to ten (10) gatherings of teeth. The investigation prevailing with regards to revealing clinical information on orthodontic section debonding power with a positive section disappointment design (i.e., less finish harm) on various teeth with critical results. As the section bond strength shifts with tooth types, it is more legitimate to concentrate on a similar tooth tests either from the upper or lower curve.

Ovundah King Wofuru-Nyenke. et.al [4], In this paper, a proficient, moderate, and compact manual vehicle tire changing instrument was planned. The device works on the rule of below average switches, where the heap to be defeated is arranged between the support and the exertion point. The mechanical benefit (MA) of the globule breaker arm and pry bar gathering of the apparatus was resolved to be 11.5. The plan of a compact manual vehicle tire transformer has been created in this paper. Burdens, stresses, shear powers, twisting minutes and torsional minutes were considered in choosing the segment portions of the apparatus. The bill of materials in Fig. 13 shows that it costs just ₦21,000 to buy the materials expected to manufacture the instrument. Thusly, the plan gives an elective convenient and generally reasonable tire changing device that can be managed by tire professionals across Nigeria, and other creating or immature nations.

Basem Alzahabi, et.al [5], The transmission mount in a force train mounting framework is regularly an essential way for clamor. This is particularly the situation when the transmission is mounted straightforwardly to the body without the advantage of an underlying casing. The vehicle's construction at the mounting area is essential as to clamor transmission, solidness and crash value. The upper section connects to the transmission by utilization of two even latches. A restricted vertical space is accessible for both the elastic mount and section. With restricted elastic volume it's anything but conceivable to permit the elastic to be however delicate as it seems to be wanted for most extreme detachment of vibration. Accordingly the section should be intended to be just about as hardened as could really be expected. Using Computer Aided Engineering, an improved section plan for solidness, strength and mass is gotten while as yet supporting an abbreviated advancement cycle. This paper centers around the streamlining of transmission cross part for modular prerequisites utilizing MSC.NASTRAN and ANSYS Design space programs. The outcomes are likewise contrasted and test information.

Jong Jae Kim and Heon Young Kim et.al [6], In request to acquire a consequently planned state of motor mount, an ideal shape configuration cycle of motor mounting elastic utilizing a parametric methodology is presented. The enhancement code is created to decide the shape to meet the firmness prerequisites of motor mounts, combined with a business nonlinear limited component program. A bramble type motor mount being utilized in a traveler vehicle is picked for an application model. The shape from the aftereffect of the boundary streamlining is resolved as a last model for certain alterations. The shape and firmness of every advancement stage are shown and the solidness of the improved model along the primary heading is contrasted and the plan determination of the current model. At long last, an outline of the current status and future works for the motor mount configuration are talked about.

S.K. Loh a, W.M. China, Waleed F. Faris et.al [7],

Engine section has been planned as a structure to help engine and fan in cooling items. Vibration and weakness of engine section has been constantly a worry which may prompt underlying disappointment if the subsequent vibration and stresses are serious and unreasonable. It's anything but a huge report which needs inside and out examination to comprehend the primary qualities and its dynamic conduct. This paper presents and spotlights on some Finite Element (FE) investigations performed, for example, recurrence examination to decide the primary reaction because of symphonious excitation over a recurrence range. The full recurrence can be anticipated dependent on the reactions in recurrence space. Other than that, the static and dynamic vibration investigations give the most extreme underlying pressure condition under static stacking and dynamic condition.

LITERATURE GAP

From the literature survey in the board field of "Designing of automotive components" mainly mounted brackets, it is noted that significant work has been done in the area of Finite element analysis (FEA) using static structural and dynamics analysis of automotive components. Some of the gaps have been found during literature survey are as follows:
Ample research is dedicated to the analysis of engine mounting bracket, suspension bracket. However, it is noted that less work has been reported in the area of static and dynamic analysis of chassis mounted bracket.
For most of the dynamic load investigations, mainly periodic excitation has been considered on the structural component. In limited studies, random vibration analysis of the mounting brackets of automotive vehicles is reported.

III. PROBLEM STATEMENT

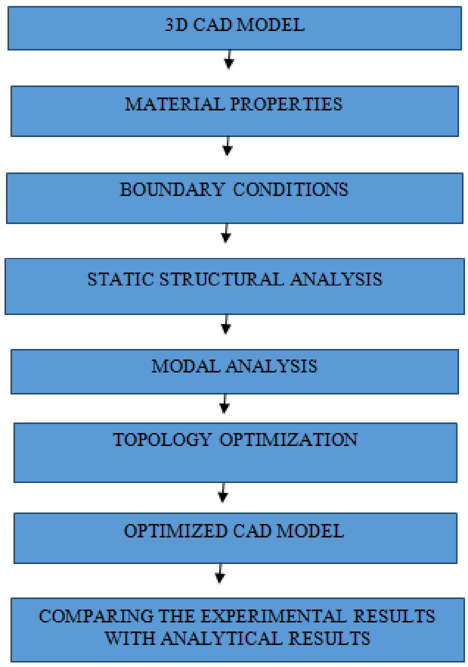
Efforts to reduce the weights of automobile components are commonly pursued in the automobile industry because weight reductions can reduce the production costs and enhance the fuel efficiency. In particular, enhancing the fuel efficiency has become increasingly important because of market demands and environmental regulations.
With the rising cost of fuels in addition to stricter emission standards, modern vehicles ought to be more fuel efficient. The best approach to increase fuel efficiency is to reduce the mass of vehicles. In order to produce light weight components for vehicles, topology optimization (TO) is now widely used by designers. However, the raw results obtained from TO cannot be manufactured directly and require significant reinterpretation to be able to be manufactured using traditional manufacturing processes. By considering the manufacturing process outside of TO, a sub-optimal design is obtained. The consideration of process specific manufacturing constraints within the TO ensures that a more optimal design will be produced. Previously the complex designs produced by TO have been a barrier to its implementation as the components cannot be produced without excessive costs. By coupling manufacturing constraints with TO more optimal designs can be obtained.

IV. OBJECTIVES

By studying the gaps in the reviewed literature, following objectives have been formulated:

1. Designing a digital model of the Truck stepney bracket using mechanical design software by reverse engineering method.
2. Analytical measurement of Random vibration acted on the bracket by doing Modal analysis and plotting the Fundamental frequency plots.
3. To do Static structural analysis of Stepney bracket by real life static load conditions.
4. Perform Optimization techniques on the bracket and optimizing the bracket for the reduction in weight.
5. The validation of analytical results is done through FFT analyzer method.

V. METHODOLOGY



EXISTING CAD MODEL OF TRUCK STEPNEY BRACKET

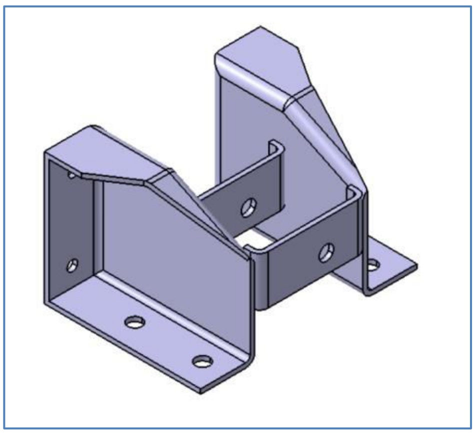


Fig. CAD model of truck stepney bracket

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 05 Issue: 07 | July - 2021 ISSN: 2582-3930

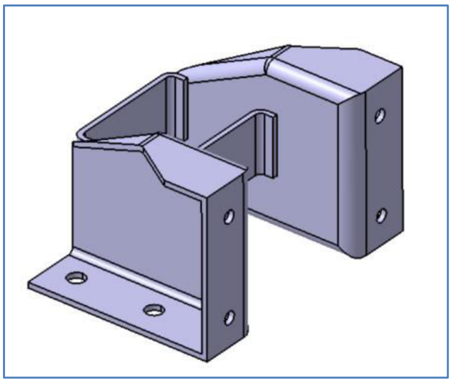


Fig. CAD model of truck stepney bracket

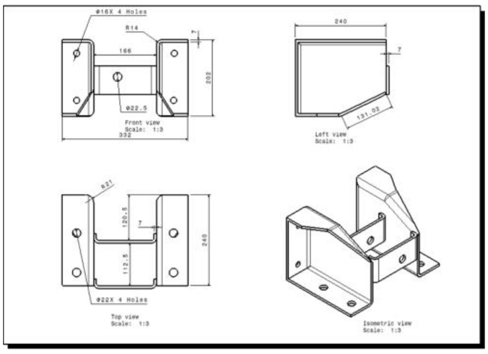
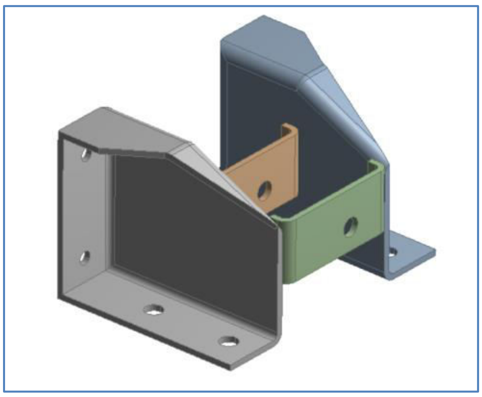


Fig. Drafting of truck stepney bracket

FEA OF TRUCK STEPNEY BRACKET

GEOMETRY



Geometry imported in ANSYS software

MATERIAL PROPERTIES

Properties of Outline Row 3: Structural Steel

	A	B	C	D
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Density	7850	kg m^-3	
4	Isotropic Secant Coefficient of Thermal Expansion			
6	Isotropic Elasticity			
7	Derive from	Young...		
8	Young's Modulus	2E+11	Pa	
9	Poisson's Ratio	0.3		

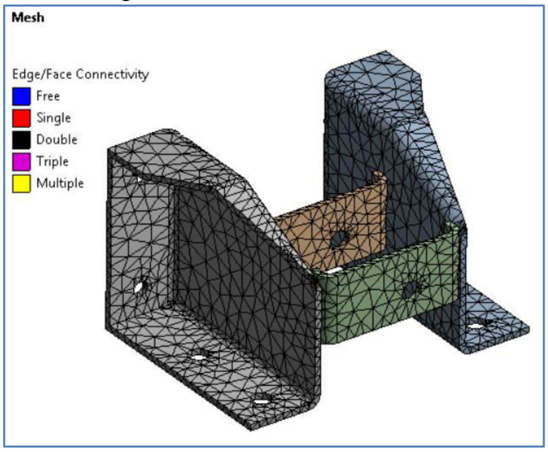
Fig material properties of truck stepney bracket

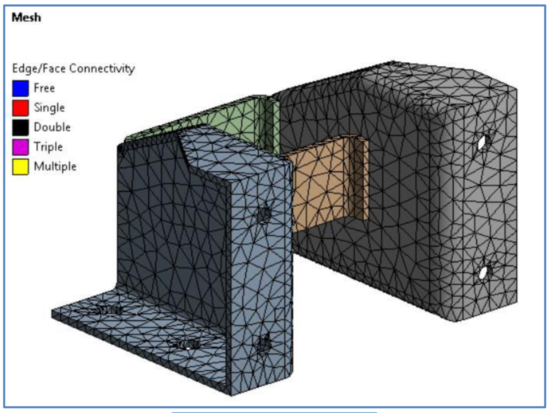
WEIGHT

Properties	
Volume	1.3724e+006 mm^3
Mass	10.774 kg

Mesh:

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation.





Statistics	
Nodes	14716
Elements	6994

Fig. Meshing details of existing truck stepney bracket

MODAL ANALYSIS

Modal analysis is the fundamental dynamic analysis type, providing the natural frequencies at which a structure will resonate. These natural frequencies are of paramount importance in various engineering fields. Suspensions are usually tuned to have different natural frequencies for passenger cars and race cars. Structural engineers need to calculate the natural frequency of buildings so that the seismic waves produced during earthquakes do not match the natural frequencies of the buildings. This course focuses on the basic theories and concepts, as well as the application of modal analysis in engineering.

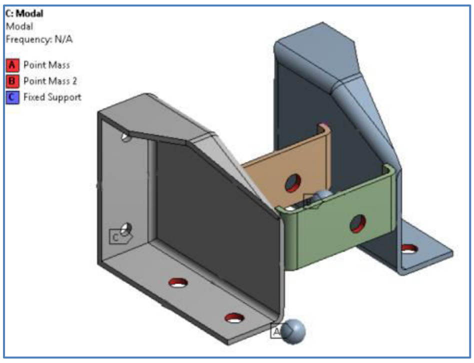


Fig. Boundary condition of existing truck stepney bracket

RESULTS

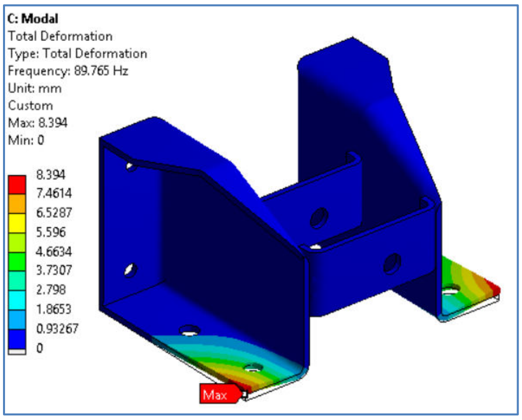


Fig. Mode shape 1 with natural frequency 89.765 Hz

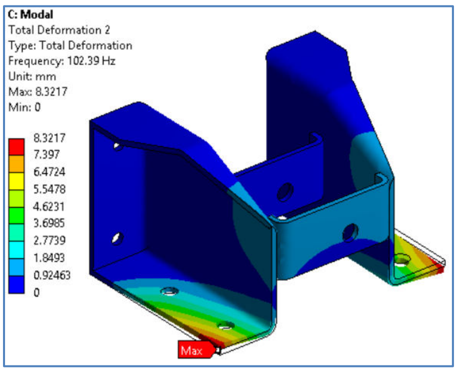


Fig. Mode shape 2 with natural frequency 102.39 Hz

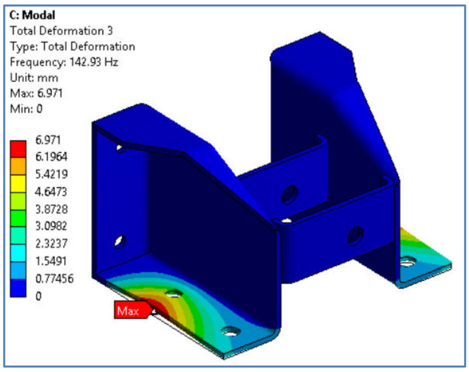


Fig. Mode shape 3 with natural frequency 142.93 Hz

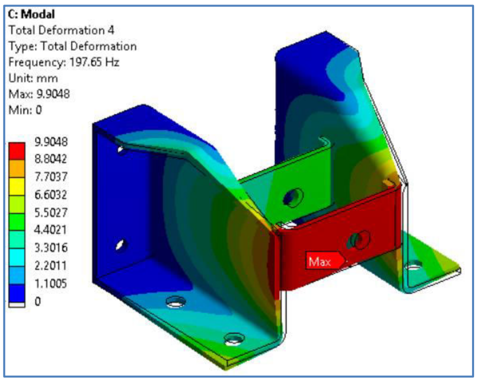


Fig. Mode shape 4 with natural frequency 197.65 Hz

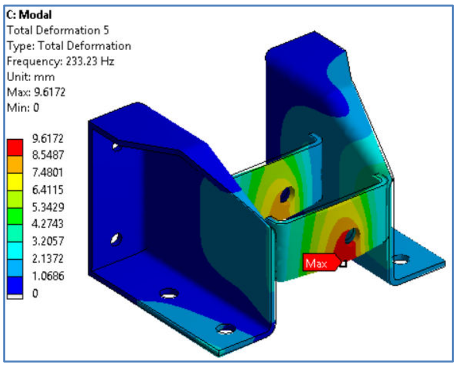


Fig. Mode shape 5 with natural frequency 233.23 Hz

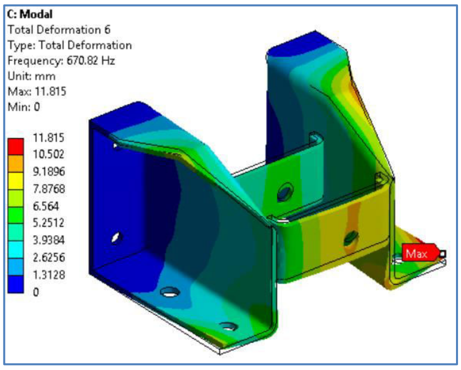


Fig. Mode shape 6 with natural frequency 670.82 Hz

MADAL ANALYSIS RESULT

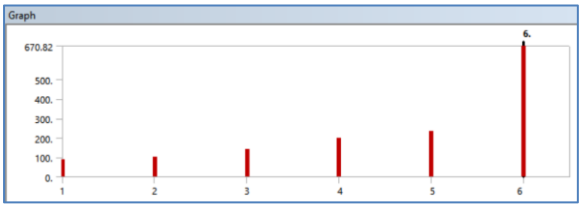


Fig. Result in graphical form

Tabular Data

	Mode	Frequency [Hz]
1	1.	89.765
2	2.	102.39
3	3.	142.93
4	4.	197.65
5	5.	233.23
6	6.	670.82

Fig. Result in tabular form

STATIC STRUCTURAL ANALYSIS

Boundary Conditions:

A boundary condition for the model is the setting of a known value for a displacement or an associated load. For a particular node you can set either the load or the displacement but not both. The main types of loading available in FEA include force, pressure and temperature. These can be applied to points, surfaces, edges, nodes and elements or remotely offset from a feature. The way that the model is constrained can significantly affect the results and requires special consideration. Over or under constrained models can give stress that is so inaccurate that it is worthless to the engineer. In an ideal world we could have massive assemblies of components all connected to each other with contact elements but this is beyond the budget and resource of most people. We can however, use the computing hardware we have available to its full potential and this means understanding how to apply realistic boundary conditions.

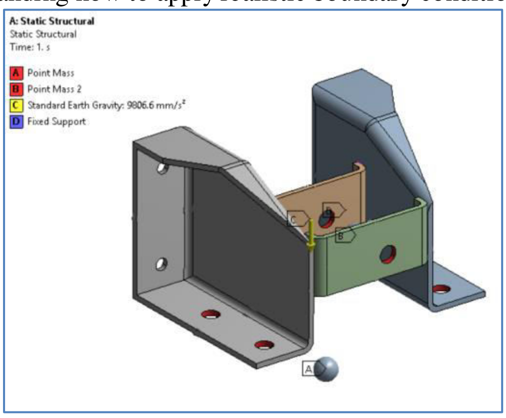


Fig. Boundary condition of existing truck stepney bracket

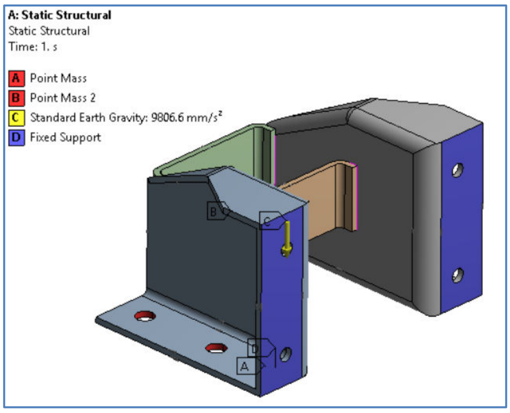


Fig. Boundary condition

Results

Total deformation

The total deformation & directional deformation are general terms in finite element methods irrespective of software being used. Directional deformation can be put as the displacement of the system in a particular axis or user defined direction. Total deformation is the vector sum all directional displacements of the systems.

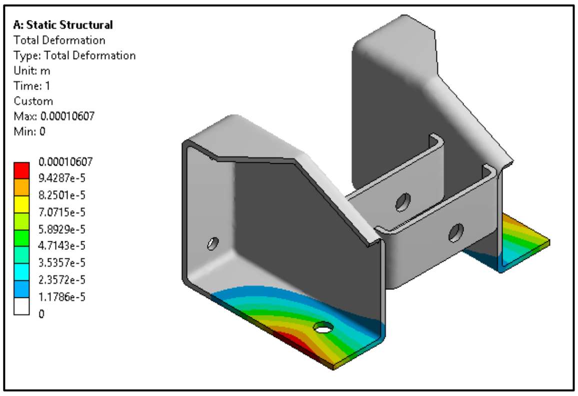


Fig. Total Deformation of existing truck stepney bracket

Equivalent Stress

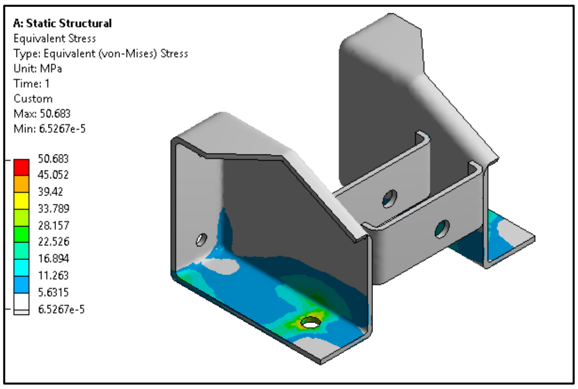


Fig. Equivalent Stress of existing truck stepney bracket

Maximum Principal stress

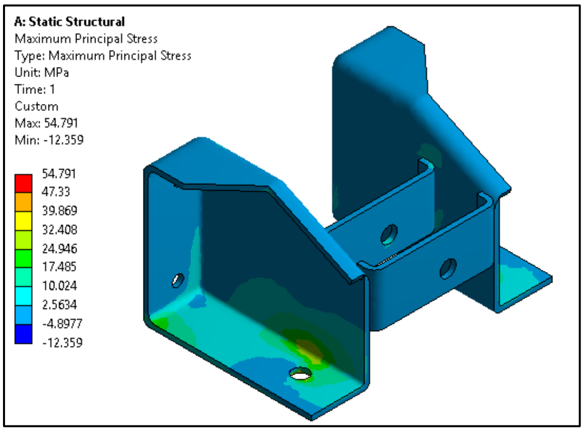


Fig. Maximum Principal stress of existing truck stepney bracket

TOPOLOGY OPTIMIZATION:

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

Basic Theory:

There are three kinds of structure optimization,

- Size Optimization
- Shape Optimization
- Topology Optimization

Optimization Boundary Condition:

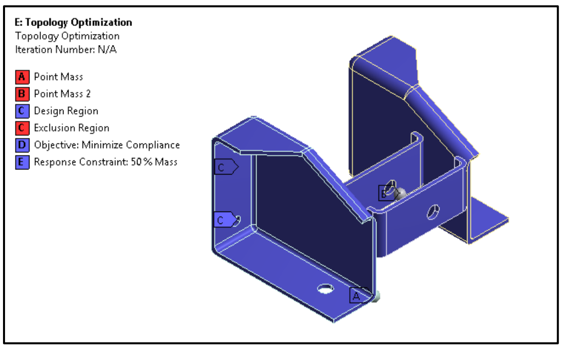
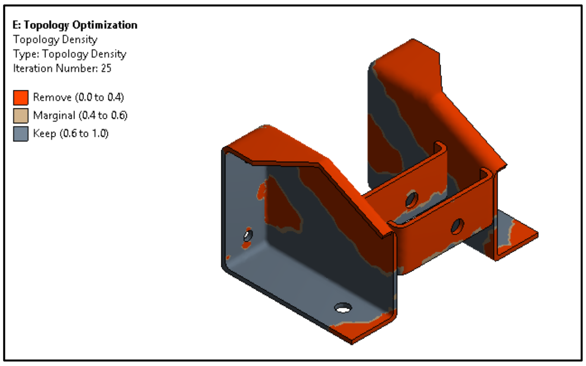


Fig. Topology optimization boundary condition

The regions which are being excluded are the same which have been selected for static structural analysis regions from which the material can be removed:



OPTIMIZED GEOMETRY:

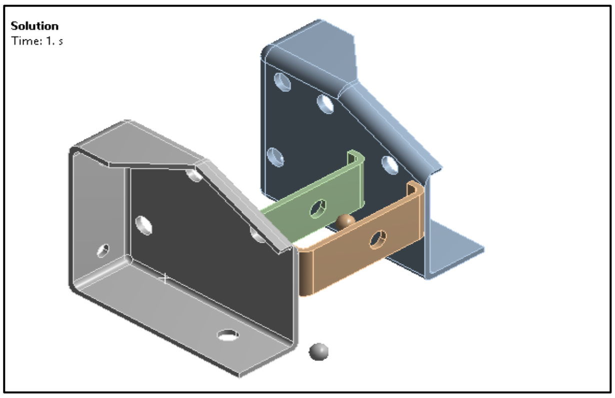


Fig. optimized model of stepney bracket

FEA of optimize truck stepney bracket

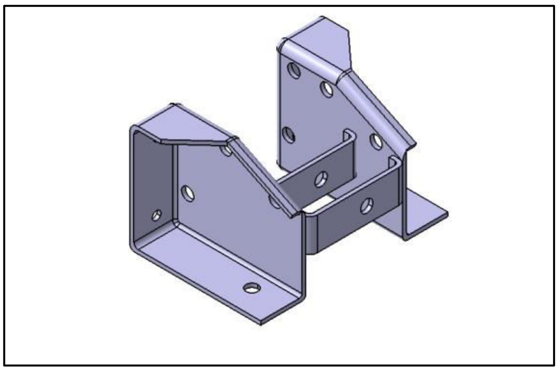


Fig. Geometry of optimize truck stepney bracket

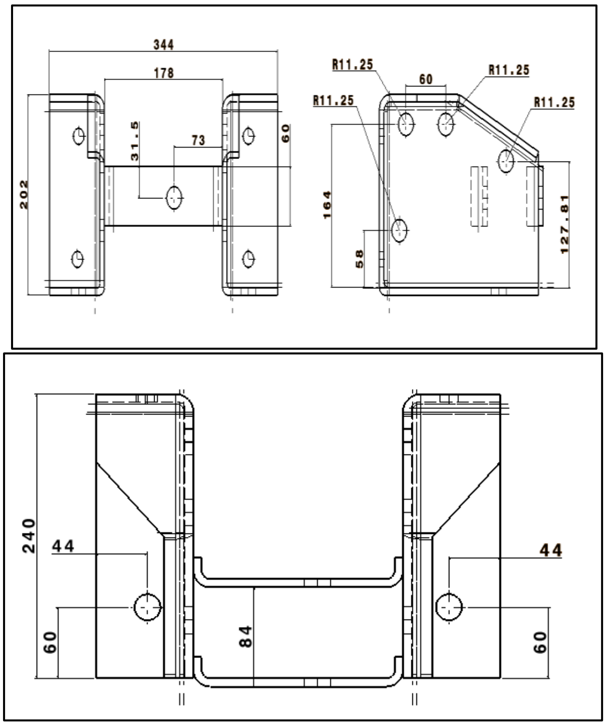


Fig. Drafting of optimize truck stepney bracket

Weight Of The Optimized Model:

Properties	
Volume	1.3071e-003 m³
Mass	10.261 kg

MATERIAL:

Properties of Outline Row 3: Structural Steel

	A	B	C	D
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Density	7850	kg m^-3	
4	Isotropic Secant Coefficient of Thermal Expansion			
6	Isotropic Elasticity			
7	Derive from	Young...		
8	Young's Modulus	2E+11	Pa	
9	Poisson's Ratio	0.3		

Fig. optimized model of stepney bracket

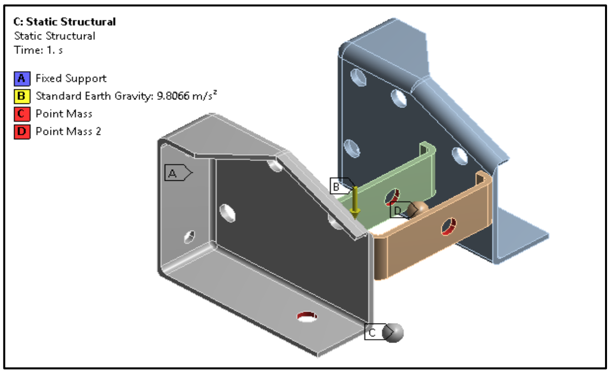


Fig. optimized model boundary condition

POINT MASS OF 200 KG AND 10 KG IS APPLIED ON THE STEPNEY BRACKET AT C AND D RESPECTIVELY AND THE BACK PORTION OF STEPNEY BRACET IS CONSIDERED AS FIXED SUPPORT. STANDARD EARTH GRAVITY IS ALSO APPLIED.

RESULTS AND PLOTS:

TOTAL DEFORMATION PLOT:

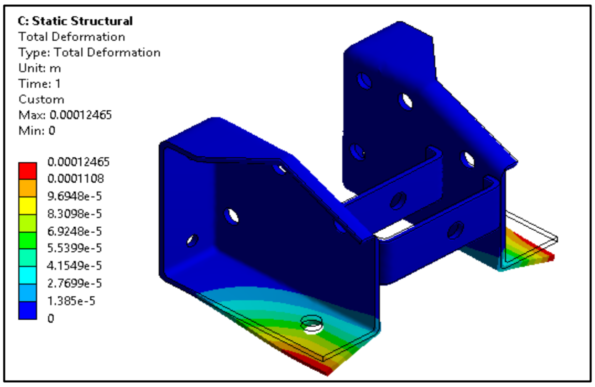


Fig. Total deformation is 0.00012465mm

EQUIVALENT STRESS PLOT:

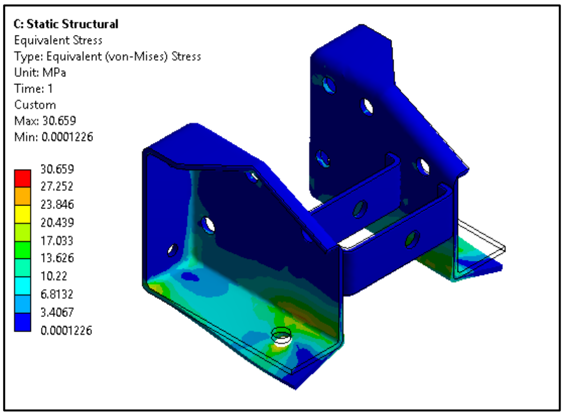


Fig. Equivalent stress generated 30.659 MPa

MAXIMUM PRINCIPLE STRESS PLOT:

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 05 Issue: 07 | July - 2021 ISSN: 2582-3930

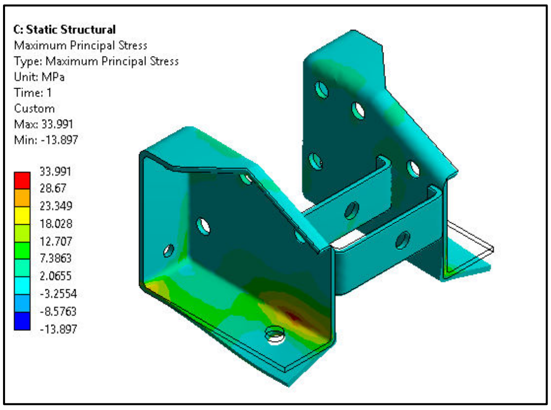
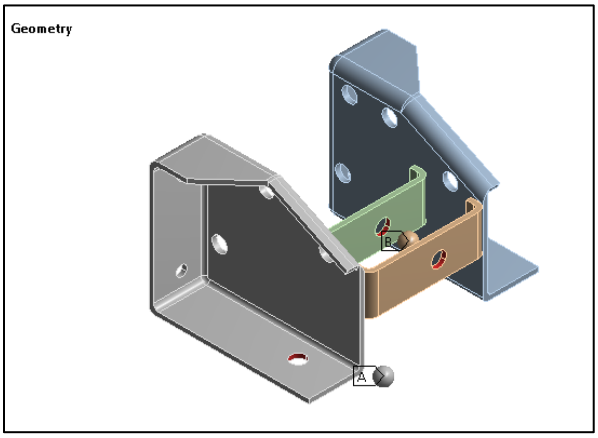


Fig. Maximum principle stress is 33.99 MPa

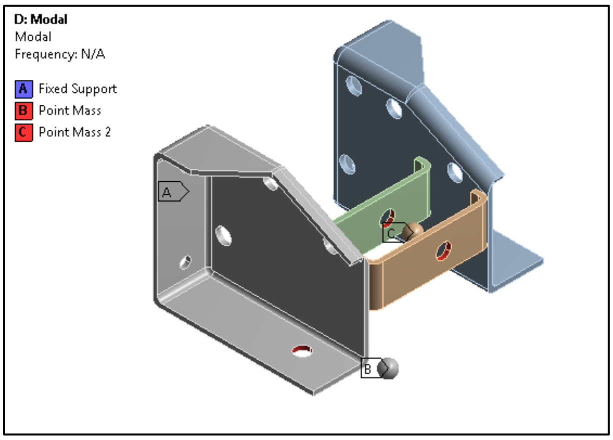
As the stresses induced inside the optimized model of stepney bracket is less than the yield strength of the material Hence the optimization done is safe.

MODAL ANALYSIS OF OPTIMIZED MODEL:

GEOMETRY:



BOUNDAR CONDITION FOR MODAL ANALYSIS:



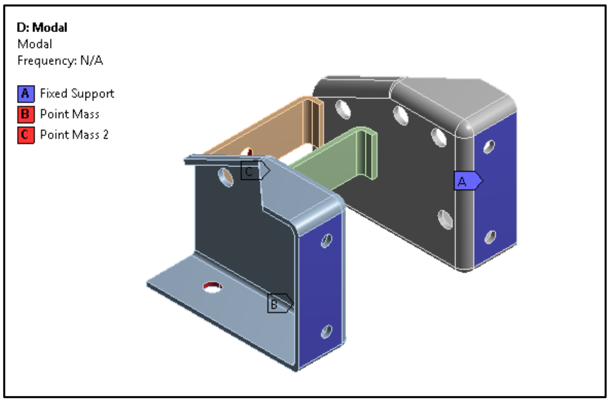
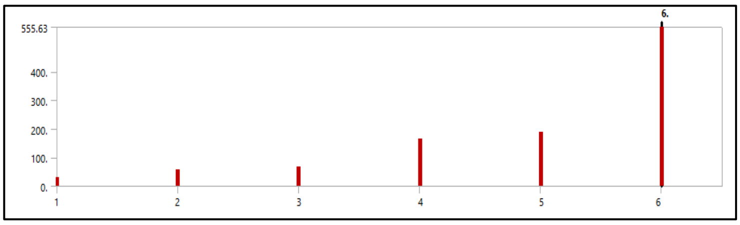


Fig Boundary condition optimize truck stepney bracket

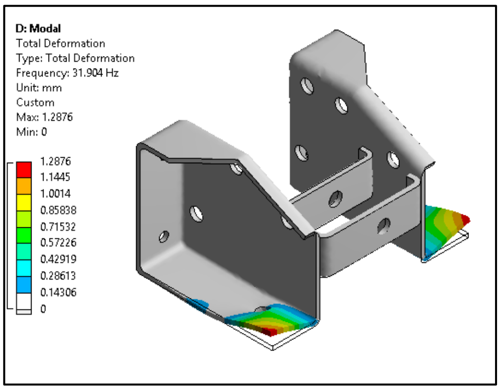
RESULTS AND PLOTS:



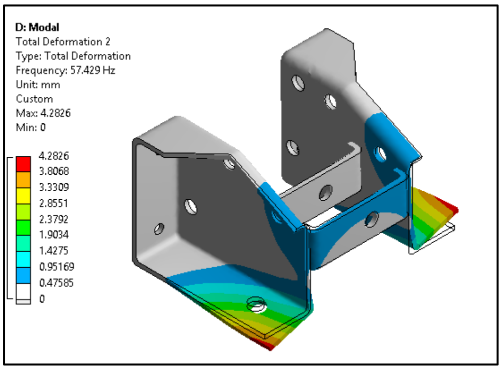
Tabular Data

	Mode	Frequency [Hz]
1	1.	31.904
2	2.	57.429
3	3.	67.999
4	4.	165.48
5	5.	188.13
6	6.	555.63

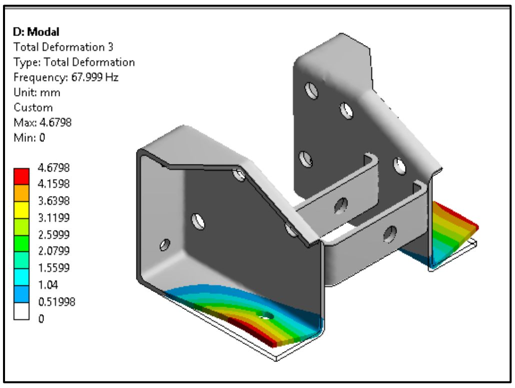
MODE SHAPE 01:



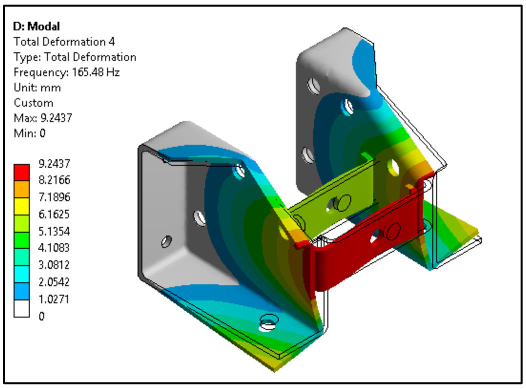
MODE SHAPE 02:



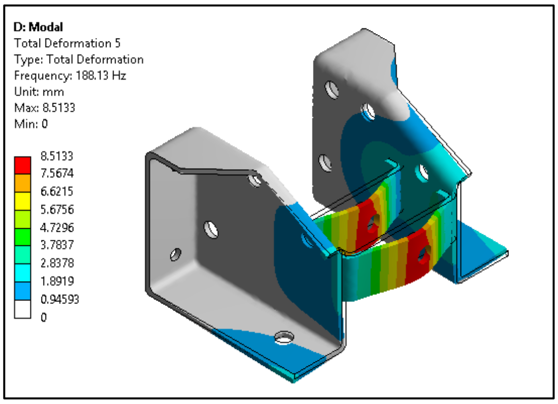
MODE SHAPE 03:



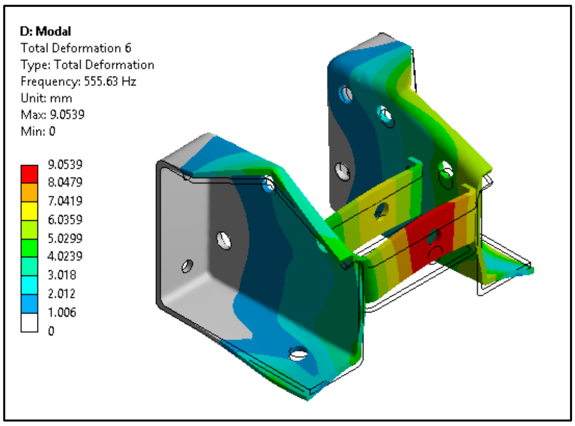
MODE SHAPE 04:



MODE SHAPE 05:



MODE SHAPE 06:



EXPERIMENTAL VALIDATION:

IN THIS PROJECT WE ARE GOING TO VALIDATE THE STATIC STRUCTURAL RESULTS OF STEPNEY BRACKET BY USING STRAIN GUAGE TECHNIQUE.

1) Stepney bracket is subjected to UTM machine, In this procedure we have welded a bracket at the mounting location of stepney and the vertical rod is pushed against the bracket which will act as a weight of Tyre itself.
2) After the engagement of UTM the results of strain produced inside the bracket is plotted.
3) The strain developed while experiment are being compared with the analytical results that we have plotted in ANSYS Workbench.
4) After the validation of strain values strain guage is being installed on the region where maximum strain is developed.

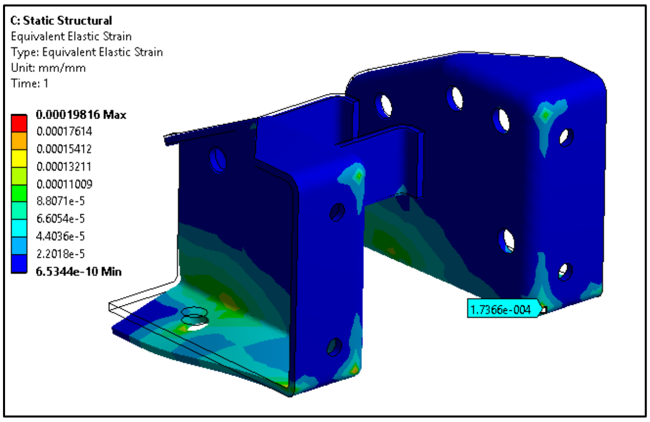


Fig. Analytical value of Strain is 173 Micron

A Universal Testing Machine (UTM) is used to test both the tensile and compressive strength of materials. Universal Testing Machines are named as such because they can perform many different varieties of tests on an equally diverse range of materials, components, and structures.

Universal Testing Machines can accommodate many kinds of materials, ranging from hard samples, such as metals and concrete, to flexible samples, such as rubber and textiles. This diversity makes the Universal Testing Machine equally applicable to virtually any manufacturing industry.

The UTM is a versatile and valuable piece of testing equipment that can evaluate materials properties such as tensile strength, elasticity, compression, yield strength, elastic and plastic deformation, bend compression, and strain hardening. Different models of Universal Testing Machines have different load capacities, some as low as 5kN and others as high as 2,000kN.

SPECIFICATION OF UTM

1	Max Capacity	400KN
2	Measuring range	0-400KN
3	Least Count	0.04KN
4	Clearance for Tensile Test	50-700 mm
5	Clearance for Compression Test	0- 700 mm
6	Clearance Between column	500 mm
7	Ram stroke	200 mm
8	Power supply	3 Phase , 440Volts , 50 cycle. A.C
9	Overall dimension of machine (L*W*H)	2100*800*2060
10	Weight	2300Kg

- Fixture is manufactured according to component designed.
- Single force is applied as per FEA analysis and reanalysis is performed to determine strain by numerical and experimental testing.
- Strain guage is applied as per FEA results to maximum strained region and during experimental testing force is applied as per numerical analysis to check the strain obtained by numerical and experimental results.
- During strain gage experiment two wires connected to strain gage is connected to micro controller through the data acquisition system and DAQ is connected to laptop. Strain gage value are displayed on laptop using DEWESOFT software.

UTM test result:
Strain value on the UTM machine

CONCLUSION:

- In this project we design stepney bracket with the help of CATIA software by reverse engineering method and research papers.
- We perform modal analysis and static analysis on existing tire bracket with the help of ANSYS software.
- In modal analysis find out natural frequency of existing stepney bracket. The natural frequency of existing bracket is 89.76 Hz
- In static analysis we find out total deformation and equivalent stress generated on bracket, The maximum deformation and equivalent stress obtained in bracket is 0.0001226 mm and 34 MPa respectively.
- Next step is to perform topology optimization on existing component to reduce weight and re-analysis using ANSYS software.
- The weight of the stepney bracket is reduced from 10.774 to 10.261 Kg. which is about 5% of weight is reduced from the stepney bracket.

REFERENCES

1) “Analysis of CFRP Flight Interface Brackets under Shock Loads” by Sagar Inglea, NikhilTilakpurea, V.Narayanamurthyb, S.M.Hussaini.

2) “Advanced Spare Wheel Carrier” by Mr. Kaustubh Shete. International Journal of Scientific & Engineering Research, Volume 6, Issue 7, July-2015, ISSN 2229-5518.

3) "Comparison of Orthodontic Bracket Debonding Force and Bracket Failure Pattern on Different Teeth In Vivo by a Prototype Debonding Device” by Tamzid Ahmed ,Norma Ab Rahman ,and Mohammad Khursheed Alam. Hindawi BioMed Research International Volume 2021, Article ID 6663683.

4) “Design Analysis of a Portable Manual Tyre Changer” by Ovundah King Wofuru-Nyenke. European Journal of Engineering Research and Science · November 2020.

5) "Optimization of Transmission Mount Bracket." by Basem Alzahabi, Scott C. Simon, Logesh Kumar Natarajan.

6) "Shape design of an engine mount by a method of parameter optimization." by Jong Jae Kim and Heon Young Kim.

7) "Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading" by S.K. Loh a, W.M. China, Waleed F. Faris.