Design and Optimization of Hydraulic Cylinder Mounting Bracket

Komal Sawant[1], Sainand Jadhav[2]

[1]*M.E. Student, Department of Mechanical Engineering, NBN Sinhgad School of Engineering, Pune, India*
[2]*Professor, Department of Mechanical Engineering, NBN Sinhgad School of Engineering, Pune, India*

---***---

Abstract - *In present study weight optimization, vibration analysis is considered as a major factor to design an optimized model for the existing design of hydraulic cylinder mounting bracket. So, experimental and numerical analysis of existing hydraulic cylinder mounting bracket is to performed to study the effect of deformation, mode shapes, weight optimization and to obtain optimized model using ANSYS software (Topology optimization). Static structural and free vibration analysis are to be performed to determine best optimized model results with existing design. From the static structural analysis, total deformation, von Mises are to be calculated and natural frequencies are to be obtained from modal analysis. Results and conclusions will be drawn. Suitable future scope will be suggested.*

Key Words: *FEA, UTM, Topology optimization, Hydraulic bracket*

1. INTRODUCTION

Current advancements in the water power industry have reformed our capacity to mechanize forms in the horticultural, development, and assembling areas. To accomplish these advances, computerized machines must have pressure driven systems that can give exact and dependable movement control. Water driven systems can control both the rotational and straight movement used in these procedures. Water powered engines are utilized to actualize rotational movement, while direct control is performed by pressure driven chambers. Water driven chambers are accessible in an assortment of styles and can be mounted from multiple points of view. They are effective and dependable, and albeit little of their fundamental structure has changed altogether in ages, they are as yet pertinent to the ventures to which they provide food. Albeit a few chambers are made with a by and large poor form quality, for example, with cast iron tops and heads, greater chambers are normally built with produced steel. Water powered chambers get their capacity from pressurized water powered liquid, which is regularly oil. The water driven chamber comprises of a chamber barrel, in which a cylinder associated with a cylinder bar moves to and fro. The barrel is shut toward one side by the chamber base (likewise called the top) and the opposite end by the chamber head (additionally called the organ) where the cylinder bar comes out of the chamber. The cylinder has sliding rings and seals. The cylinder separates within the chamber into two chambers, the base chamber (top end) and the cylinder bar side chamber (bar end/head end). Ribs, trunnions, clevises, and drags are basic chamber mounting alternatives. The cylinder bar likewise has mounting connections to interface the chamber to the item or machine segment that it is pushing or pulling. A water driven chamber is the actuator or "engine" side of this framework. The cylinder pushes the oil in the other chamber back to the supply. On the off chance that we accept that the oil enters from top end, during expansion stroke, and the oil pressure in the pole end/head end is around zero, the power F on the cylinder bar rises to the weight P in the chamber times the cylinder zone A. The welded chamber is essentially a barrel with a top welded to the base, and afterward with the mounting treatment welded to that top, commonly a cross cylinder or double tangs to imitate a clevis. The cylinder and pole are introduced into the chamber, and afterward a strung head is slid over the pole and torque onto the barrel.

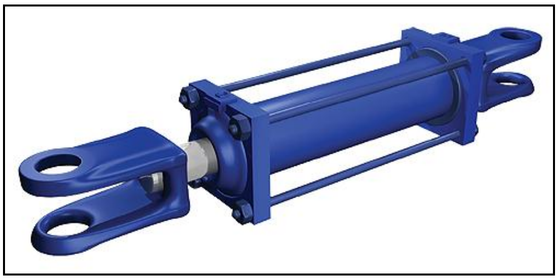

Fig -1: Hydraulic cylinder bracket

2. LITERATURE REVIEW

Priyanka S. Dahaleaet al. [1] in this paper it presents experimental and Finite Element examination of a run of the mill motor mounting bracket. It likewise introduced the Modal Analysis in FEA to decide the frequency band and check the bracket for security. The exploratory examinations for co-connection to build up variety of rate and in this manner decide the idea of Boundary conditions to be utilized in FEA for progressively precise investigation. The Automobile motor case framework may encounter undesirable vibrations brought about by impedance between the street and the motor. Motor bracket has been planned as a structure to help motor. Because of vibrations of motor the openings on the motor Bracket get extended which prompts the failure of bracket. The outcomes got for the static basic and modular investigation have demonstrated that the subsequent limit condition for example one fixed and one pressure support are increasingly precise with less level of deviation 18.6 % with exploratory outcomes, which can be utilized for additional investigation for wellbeing of Engine mounting bracket.

Mohammed Khaja Nizamuddinet al. [2]This paper manages the topology optimization of motor mounting bracket of 'Chevrolet beat' utilizing the instruments CATIA V5R20 for demonstrating and Hyper works for limited component investigation. The utilization of motor mounts is the best answer for dampening the impact of vibrations and transmitting powers between the motor and the car body structure. The principle goal of the work is to limit the heaviness of the motor mounting bracket by thinking about the structure and material format. For various material format and various structures, the anxieties and loads are registered and contrasted with show up at the best model under recommended conditions. It is seen that the most extreme worry for all plans didn't surpass a definitive rigidity for the relating materials. In light of the examination of the weight decrease accomplished in the three streamlined structures, it tends to be reasoned that the most elevated weight decrease has been acquired in plan.

Pooja Moreaet al. [3] this paper centers on the deduction of move capacity of radiator brackets to communicate their dynamic vibration attributes. The way to deal with determine the equivalent involves limited component demonstrating of the radiator bracket, model approval, parameter recognizable proof dependent on affectability study and structure of tests to infer the exchange work. Extreme vibration brings about auxiliary harm. The structure itself has certain inside properties and it is essential to comprehend its attributes. In this paper, move work for the initial two bending frequencies of the bracket is created with parameters that would essentially influence the major frequency as information sources. The initial two bending frequencies acquired from diagnostic technique are contrasted and that from the limited component models and found to have great understanding. The exchange capacities to acquire the initial two twisting frequencies of radiator mounting bracket are inferred dependent on parametric examination and plan of investigations. The seven basic parameters influencing the elements of the bracket are radiator mass, motor mass, number of jolts, generally speaking thickness of bracket, bracket length, tie bar length and nearness or nonattendance of AVMs among radiator and bracket.

Lan, Jet al. [4], In this paper, rope-wheel type is contemplated. Rope-wheel glass lifter as a rule comprises of slider, manage rail, direct wheel, engine, the upper and lower wire ropes, and so forth. Two distinct techniques are applied to the topology improvement of guide rail and a weighted trade off programming approach for illuminating multi-target topology enhancement of numerous stacking cases and normal frequencies is proposed in this paper. Initially, the plan area is resolved by the mechanical structure and working conditions. Furthermore, the single objective continuum structure topology streamlining mathematic models of guide rail are fabricated and the investigations of multi-firmness topology improvement of various volume parts are done appropriately. Thirdly, a weighted trade off programming approach is proposed to take care of multi-targets advancement issues and multi-target topology improvement are performed dependent on this strategy in which volume portion is set as 0.4,0.5 and 0.6. The estimations of advanced consistence are lower than that of the single objective topology improvement and the initial three request frequencies of multi-target topology enhancement are higher than that of single objective.

Sreekanth Dondapatib et.al [5] in the current work, experimental examination on the failure of a suppressor mounting bracket connected to business vehicle is finished. Splits are distinguished at the welded area of suppressor mount which shows that weld joint has preferred quality over the suppressor/bracket body. To comprehend the conceivable underlying drivers of the failure, fishbone chart was utilized, which helped in deciding the significant reasons for the failure by a graphical portrayal. Further, the three parameter Weibull dispersion was likewise evolved to decide the Mean Time to Failure (MTTF) life which was seen as 15,172 km. Also, elastic testing of sheet metal was performed on the sheets which were utilized in the assembling of Muffler. Besides, a Thermo-Mechanical coupled examination was completed utilizing business code, ANSYS 16.0, which adjusts Finite Element Analysis (FEA) definition. The warm loads on suppressor were imported to basic investigation alongside a static heap of 4 g quickening were forced on the suppressor body to reenact the impacts of high effect loads.

3. PROBLEM STATEMENT

Optimization of weight has been very critical aspects of any design. It has substantial impact on vehicle performance, and in spin minimizes the emissions. For Experimental and numerical analysis of existing hydraulic cylinder mounting bracket to study the effect of deformation, mode shapes, weight optimization and to obtain optimized model using ANSYS software.

4. OBJECTIVES

- To develop optimized hydraulic cylinder mounting bracket for existing model in CATIA software.
- To perform static and vibration analysis (modal analysis) of hydraulic cylinder mounting bracket using ANSYS 19 software. To determine mode shape, natural frequency, deformation, von misses, stresses and strain.
- To perform stain gauge test on UTM for extracting of stain in hydraulic cylinder mounting bracket.
- Validation of experimental and numerical analysis.

5. METHODOLOGY

Step 1:-Initially research papers are studied to find out research gap for project then necessary parameters are studied in detail. After going through these papers, we learnt about optimization of hydraulic cylinder bracket mounting.

Step2:-Research gap is studied to understand new objectives for project.
Step 3: - After deciding the components, the 3 D Model and drafting will be done with the help of software.
Step 4: - The components will be manufactured and then testing will be performed on UTM using strain guage.
Step 5: -The testing will be carried out and then the result and conclusion will be drawn.

6. DESIGN AND ANALYSIS

6.1 CAD Design

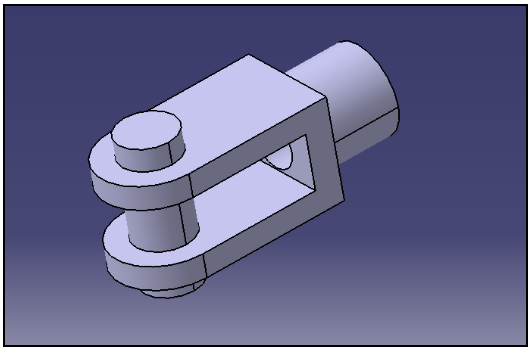

Fig -2: CATIA model of hydraulic cylinder bracket

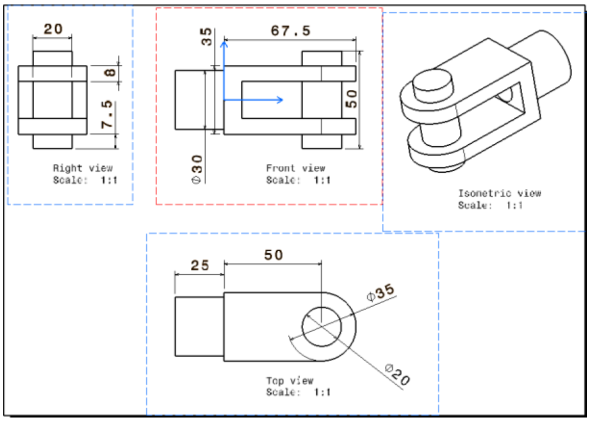

Fig -3: drafting of hydraulic cylinder bracket

Material Properties

Properties of Outline Row 3: Structural Steel

	A	B	C
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's Modulus and Pois...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa

Table -1: Material properties of S.S

6.2 Analysis

Geometry

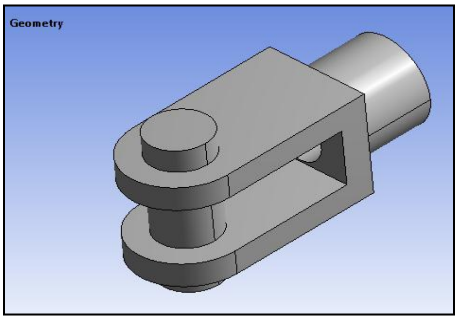

Fig -4: Geometry of actual hydraulic cylinder bracket

Mesh

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient metaphysics 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

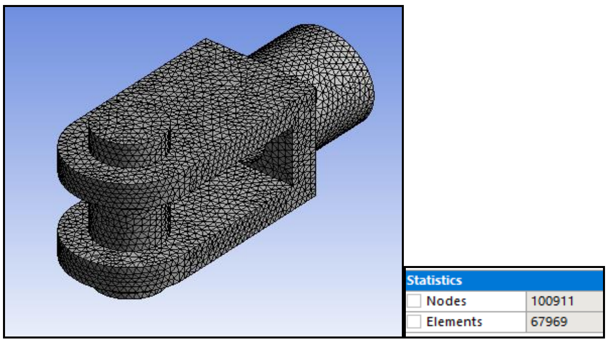

Fig -6: meshing of actual hydraulic cylinder bracket

After meshing of hydraulic cylinder bracket are 100911 and elements 67969.

Standard Stroke

Bore size (mm)	Standard stroke (mm)
40	25 to 800
50	25 to 800
63	25 to 800
80	25 to 800
100	25 to 1000

Theoretical Output (OUT / IN)

Unit: N

Bore size (mm)	Rod size (mm)	Operating direction	Piston area (mm^2)	Operating pressure (MPa)		
				3.5	7	10
40	22	OUT	1256	4396	8792	12560
		IN	876	3066	6132	8760
50	28	OUT	1963	6871	13741	19630
		IN	1347	4715	9429	13470
63	36	OUT	3117	10910	21819	31170
		IN	2099	7346	14693	20990
80	45	OUT	5026	17591	35182	50260
		IN	3436	12026	24052	34360
100	56	OUT	7853	27486	57971	78530
		IN	5390	18865	37730	53900

Theoretical output (N) = Pressure (MPa) x Piston area (mm^2)

Table -2: Strandard stroke and Theoritical output

Boundary condition

In present investigation existing hydraulic cylinder bracket of standard size of bore size 63 mm is selected. So, in boundary condition following in and out for 3.5 MPa operating pressure FEA analysis have been performed to determine stress, deformation and topology optimization.

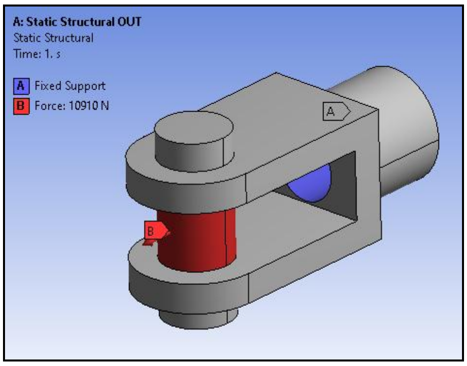

Fig -7: Boundary condition of actual hydraulic cylinder bracket

Deformation Results

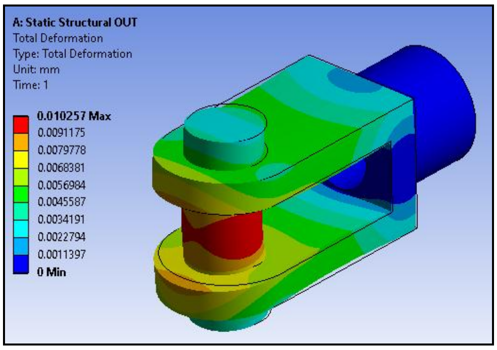

Fig -8: Deformation result of actual hydraulic cylinder bracket

Maximum deformation under static condition of actual hydraulic cylinder bracket was 0.01025 mm.

Equivalent stress

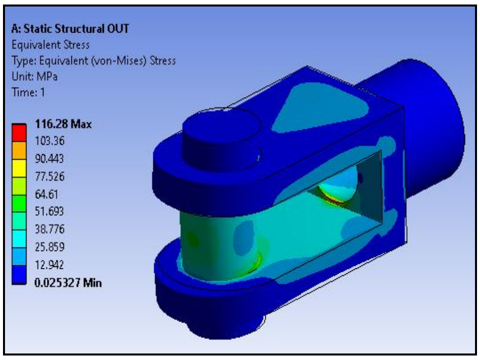

Fig -9: Equivalent stress result of actual hydraulic cylinder bracket

Maximum Equivalent stress of actual hydraulic cylinder bracket was 116.28 MPa.

MODAL ANALYSIS

To perform modal analysis fixed support is applied at inner side of cylinder bracket to determine mode shape with natural frequency.

At mode1

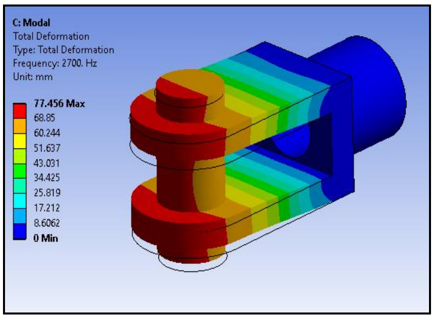

Fig -10: Mode shape 1

Natural frequency of actual hydraulic cylinder bracket at mode shape 1 was 2700 Hz.

At mode 5

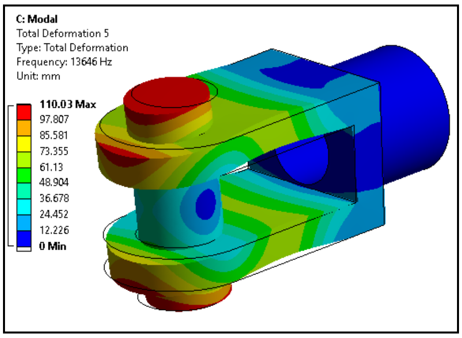

Fig -11: Mode shape 5

Natural frequency of actual hydraulic cylinder bracket at mode shape 5 was 13646 Hz.

Tabular Data

	Mode	Frequency [Hz]
1	1.	2700.
2	2.	3767.7
3	3.	5372.5
4	4.	13384
5	5.	13646

Table -3: Tabular representation of the mode shapes with respective frequency

Static analysis Under inword boundary condition

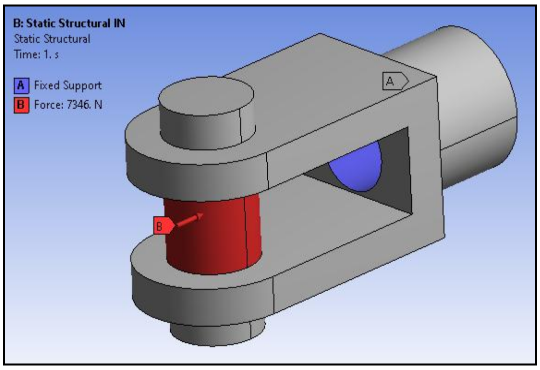

Fig -12: Static analysis for IN boundary condition

Deformation

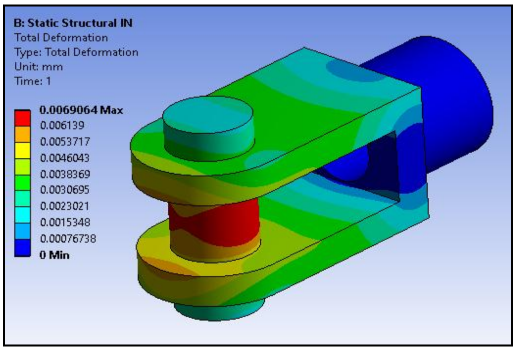

Fig -13: Deformation result of hydraulic cylinder bracket

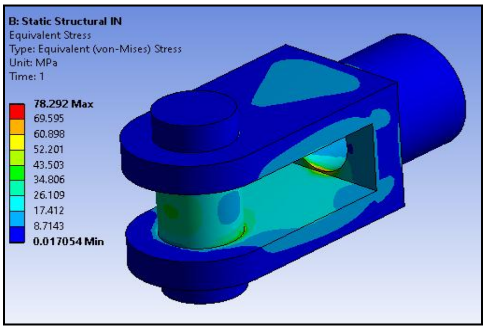

Fig -14: Equivalent stress result of hydraulic cylinder bracket

7. TOPOLOGY OPTIMIZATION OF HYDRAULIC CYLINDER BRACKET

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

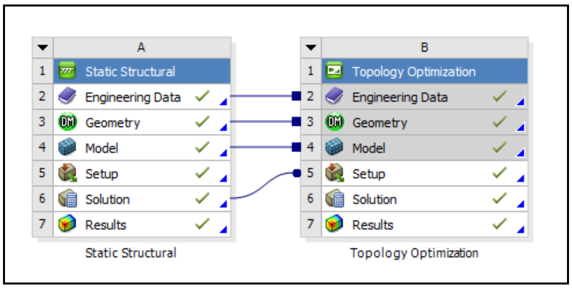

Fig -15: Flow of process in ANSYS

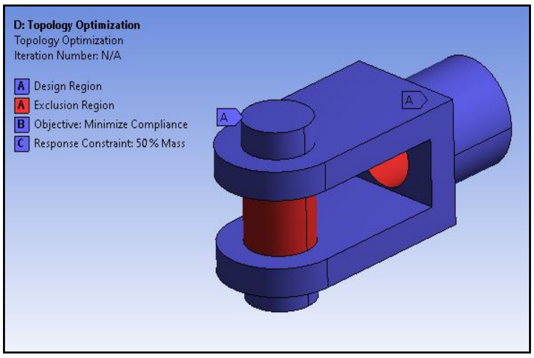

Fig -16: Topology optimization of the hydraulic bracket showing the Exclusion region

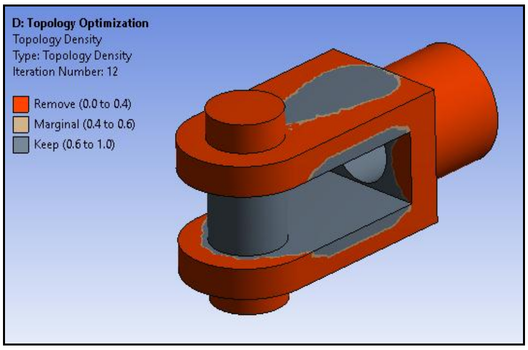

Fig -15: Optimized region in topology optimization

8. FEA ANALYSIS OF NEW OPTIMIZED HYDRAULIC BRACKET IN ANSYS

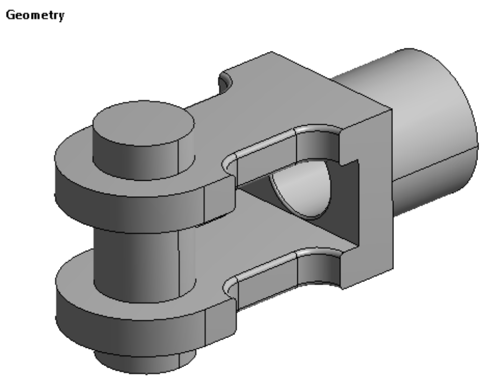

Fig -16: Optimized design

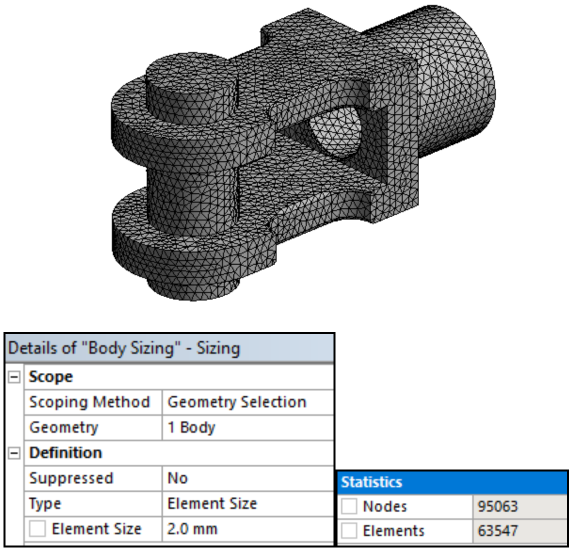

Fig -17: Optimized design meshing

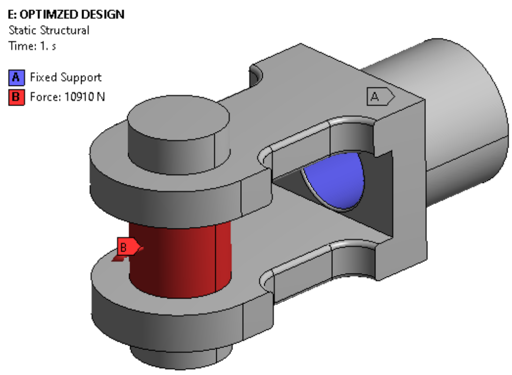

Fig -18: Optimized design boundary condition

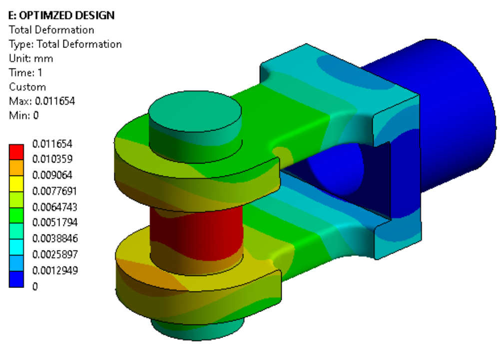

Fig -19: Optimized design deformation result

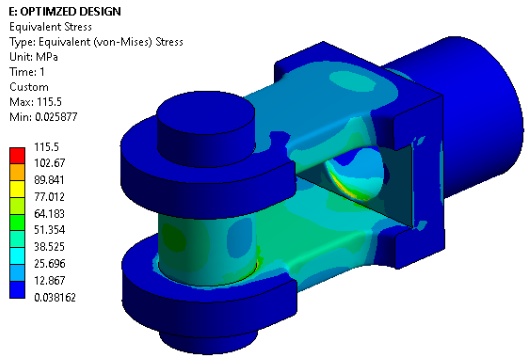

Fig – 20: Optimized design equivalent stress result

9. EXPERIMENTAL TEST

A universal testing machine (UTM), also known as a universal tester, materials testing machine or materials test frame, is used to test the tensile strength and compressive strength of materials. An earlier name for a tensile testing machine is a tensometer. The "universal" part of the name reflects that it can perform many standard tensile and compression tests on materials, components, and structures (in other words, that it is versatile). The set-up and usage are detailed in a test method, often published by a standards organization. This specifies the sample preparation, fixturing, gauge length (the length which is under study or observation), analysis, etc.The specimen is placed in the machine between the grips and an extensometer if required can automatically record the change in gauge length during the test. If an extensometer is not fitted, the machine itself can record the displacement between its cross heads on which the specimen is held. However, this method not only records the change in length of the specimen but also all other extending elastic components of the testing machine and its drive systems including any slipping of the specimen in the grips. Once the machine is started it begins to apply an increasing load on specimen. Throughout the tests the control system and its associated software record the load and extension or compression of the specimen.

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

Table -3: Specification of UTM

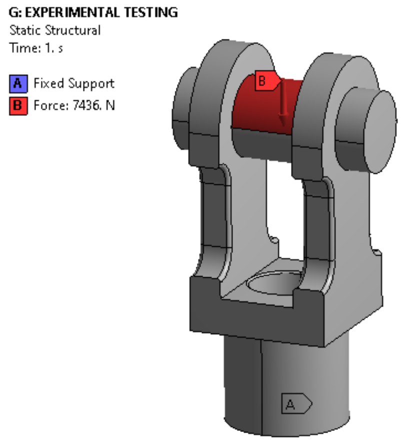

Fig -21: Experimental testing boundary condition

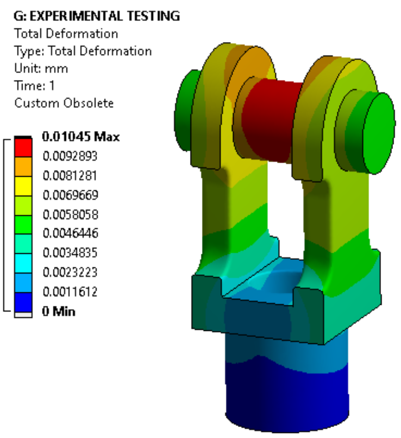

Fig -22: Experimental testing deformation results

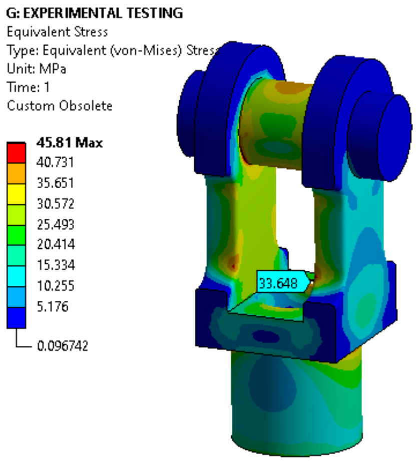

Fig -23: Experimental testing stress results

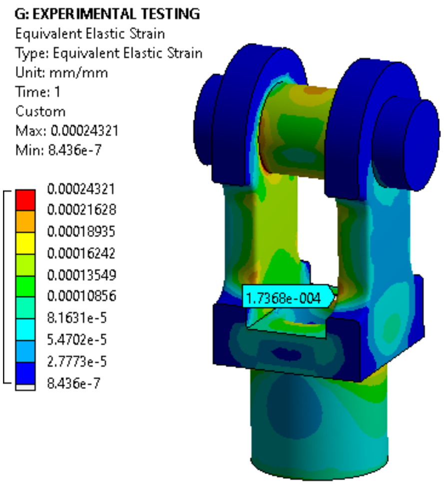

Fig -24: Experimental testing strain results by FEA analysis strain is observed around 173 micron

Experimental procedure

- 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 gauge 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 values are displayed on laptop using DEWESOFT software.

The load pointer is set at zero by adjusting the initial setting knob. The dial gauge is fixed and the optimize hydraulic cylinder bracket for measuring elongation of small amounts. Measuring the diameter of the optimize hydraulic cylinder bracket by vernier caliper at least at three places and determine the mean value also mark the gauge length. Now the optimize hydraulic cylinder bracket is gripped between upper and middle cross head jaws of the machine. Set the automatic graph recording system. Start the machine and take the reading.

10. CONCLUSIONS

In present study existing hydraulic cylinder mounting bracket is studied under static and modal analysis technique to determine stress, deformation and natural frequency.In topology optimization technique area indicated in red region assist to remove the material from that area and reanalysis is to performed to check the sustainability of element under existing boundary condition. Modal analysis of hydraulic cylinder mounting bracket is performed to obtain different mode shapes and natural frequency of existing hydraulic cylinder mounting bracket and also it is observed that maximum frequency is around 13646 Hz. Weight reduction of around 14% is observed along with strain measurement of microns 173 and 185 microns by numerical and experimental testing respectively.

REFERENCES

[1] Priyanka S. Dahalea, Pallavi D. Pawarb, Vinaay G. Patilc, "Assessment of Boundary Conditions for FEA of Mounting Bracket Using Co-relation with Experimental Results." Materials Today: Proceedings 5 (2018) 13290–13300

[2] S. Kirthanaa, Mohammed Khaja Nizamuddin,"Finite Element Analysis and Topology Optimization of Engine Mounting Bracket." Materials Today: Proceedings 5 (2018) 19277–19283

[3] Pooja Morea, B Santhosha, Amol Jain, "Analysis of Radiator Mounting Bracket Design and Derivation of Transfer Function to Predict Modal Frequencies Based on Parametric Study" Materials Today: Proceedings 5 (2018) 23945–23954

[4] Sreekanth Dondapatib, Mudit Trivedi, "Investigation on the mechanical stresses in a muffler mounting bracket using Root Cause Failure Analysis (RCFA), finite element analysis and experimental validation," Engineering Failure Analysis 81 (2017) 145–154

[5] An, B., Yang, Y., Huang, J., Liu, W, "Structure Optimization Method for Heavy Truck Propeller Shaft Bracket," T2015 SAE International, 2015-01-0638, 04/14/2015

[6] eng, M. and Lan, J., "The Topology Optimization Analysis on Rope-Wheel Glass Lifter," by SAE International by University of Minnesota, Thursday, August 02, 2018