Optimization and FEA of Leaf Spring Mounting Bracket

Mr.Amit J. Patil[1], Prof. G.E.Kondhalkar[2]
[1, 2] Department of Mechanical Engineering
[1, 2] AnantraoPawar college of Engineering, Parvati.

Abstract- *One of the main causes of vibration that produce by a car is engine and transmission. Leaf spring mounting bracket are used to hold the leaf spring firmly with chassis. From last so many years Automobile industry continuously growing with different idea & technology for change in existing parts of vehicles. The most effective way of increasing automobiles mileage while decreasing emission is to reduce vehicle weight. Existing bracket has scope of mass optimization in current design .finite element analysis of bracket will be done using hyper mesh and ansys. Optistruct software will be used for topology optimization. Experimental stress of bracket will be done using strain gauge and applying corresponding loading through UTM. Validation for strain vector from FEA & Experimental results*

Keywords- Hybrid joint, Adhesive, FEA, Reaction Force

I. INTRODUCTION

The suspension system bears all the weight of the vehicle and provides for a comfort for the driver and passengers. The Advancement of Vehicle travel and control can be done by placingunspring mass as low as possible. When loaded wheel assemblies gone through a bump or pothole, they exert a larger reaction force. So for this, a suspension system should be capable enough to withstand loads imposed by vehicle mass during cornering, accelerating, braking, and uneven road surfaces. Heavy parts such as the differential can be made part of the sprung weight by connecting them directly to the body. Forces is going to be considered while designing of leaf spring bracket are spring rate, roll couple percentage, weight transfer, unspring weight transfer, damping, camber control, roll centre height, jounce, rebound, wheel hop, ride steer, spring oscillation, instantaneous centre, anti-drive and anti-squat, wheel unit location, the roll of axles, beam axle, dead axle and plain axle.

Automotive industry is trying to keep vehicles part weight as possible as and considered the factors such as safety, fuel efficiency, government regulation & industry norms. In the today's service conditions, the determination of mechanical behaviour of the bracket is important. For monetary reason it is essential to decrease the development & testing time. A 3–D stress analysis of bracket includes little complex geometry. Therefore, it is difficult to estimate the stresses by using elementary mechanical approximations. So that's why finite element analysis (FEA) software is used. FEA simulation of the bracket tests can remarkably reduce the time and cost required to finalize the bracket design. Thus, the design changes could be manage on a component to examine how the change would affect its performance, without making costly modification to tooling and equipment in real production. Therefore, in order to replace the physical test, the FEA simulation for load should supply reliable results and sufficient information.

II. LITERATURE REVIEW

1. R. Prakash [1] In this paper explains the idea of the fatigue analysis of spot-weld joints to forecast the lifetime and location of the weakest spot-welds due to the imposed loading conditions. A simple model was used to explain the idea of spot-weld fatigue analysis. FEM and analysis were carried out utilizing the finite element analysis commercial codes.
2. S. Chaitanya [2] this paper, an try is made to decrease the weight of the wheel by exchange the aluminium alloy with composites..weight can be reduced by exchanging material and manufacturing process with optimization of design. From the FE calculations it is conclude that weight reduced to 50% from the existing alloy wheels..for this work modelling software CATIA V5 R20 is used and analysis is made by using ANSYS15.0.
3. P.D. Jadhav [3] for improving vehicle comfort engine mounting plays anvital role in decreasing the noise, vibrations and harshness. Mainwork of an engine mounting bracket is to properly balance (mount) the power pack (engine &transmission) on the vehicle chassis for good motion control & good isolation. this work enclosed FEA of engine mounting bracket.
4. Joong Jae Kim [4] toachieve an automatically designed shape of engine mount, an optimum shape design process of engine mounting rubber using a parametric approach is explained. The optimization code is prepared to determine the shape to meet the stiffness need of engine mounts, connected with a commercial nonlinear FE program.

5. B. Sreedhar [5] In today's situation the competitive automotive world a light weight component is playing a important role in fuel efficiency and economy point of a vehicle. This initiates to design a light weight component for the desired safety standards.
6. MonaliDeshmukh [6] the engine mounting plays avital role in reducing the noise, vibrations for improving car comfort. The brackets gone through high static and dynamic stresses as well as large amount of vibrations. Hence, minimizing the vibrations and placing the stresses under a pre-determined level of safety should be gained by neat designing and analysis of the mount brackets. FEA software package ANSYS 15.0 is used.

III. OBJECTIVES

1. Modeling existing leaf spring bracket.
2. Analyzing for stresses and deformation.
3. Topological optimization for the model.
4. Analyzing for stresses and deformation on optimized model.
5. Machining existing leaf spring bracket as per optimized model.
6. Preparing fixture to hold leaf spring bracket firmly in testing.
7. Experimental testing and correlating results.

IV. METHODOLOGY

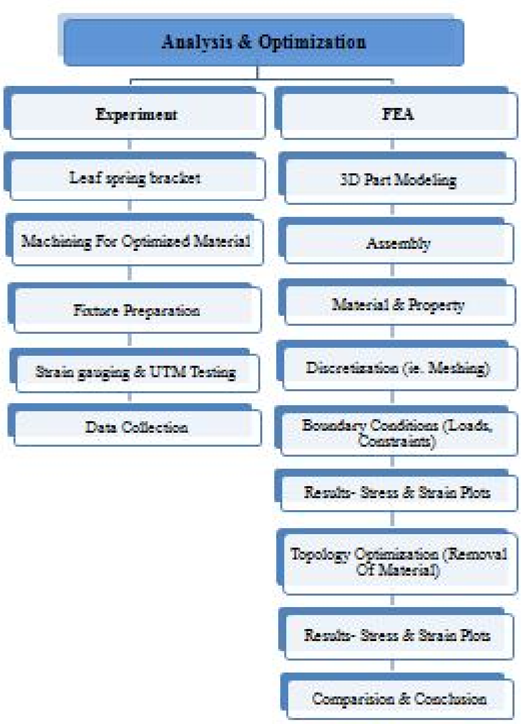

Figure 1.

V. FE ANALYSIS

1. CAD Model of leaf Spring Mounting Bracket

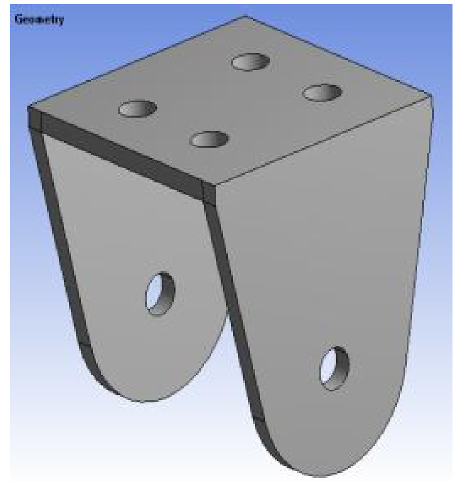

Figure 2. CAD Model of leaf spring mounting bracket

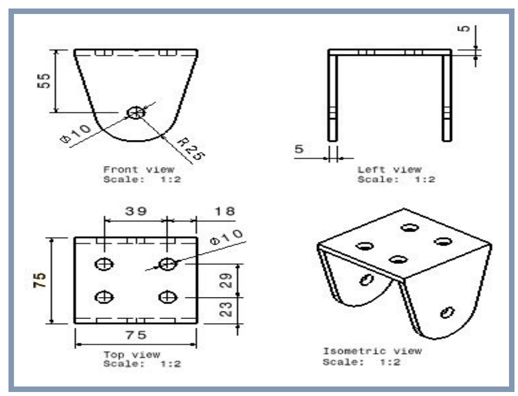

Figure 3. CAD Drafting of leaf spring mounting bracket

2. Calculations:

Assumptions:

- Mass of the vehicle (m) = 3000kg
- Maximum force acting on the spring is 8,976 N
- Force acting on bracket = maximum force/2
  = 8,976/2
  = 4,488 ~ 5000 N

3. Mesh Generation

Element Type:- Hexahedron
No. of Elements:- 6898
No. of Nodes=10668

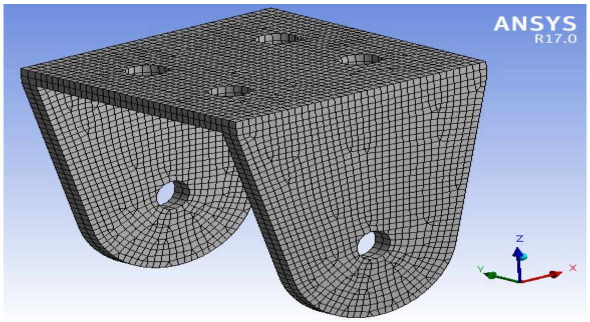

Figure 4. Discretized Model

4. Static Structural Analysis

The Finite Element Method is a numerical approximation method, in which the complex structure is divided into number of small parts that is pieces and these small parts are called as finite elements. These small elements are attached to each other by means of small points known as nodes. As the FEM uses matrix algebra to solve the simultaneous equations, so it is also called as analysis of structure and it's going to be primary analysis tool for designers and analysts.

The three basic FEA process are

a) Pre-processing phase
b) Processing or solution phase
c) Post processing phase:

- **Material properties:**

1) Material- Steel
2) Young's Modulus- 200 GPa
3) Poisons Ratio- 0.3
4) Density- 7850 kg/m3
5) Yield Strength- 520 MPa

Constraints:

All degree of freedom are fixed at top face and load is applied on holes at bottom. In the fig, 'A' defines fixed support and 'B' defines direction of applied load.

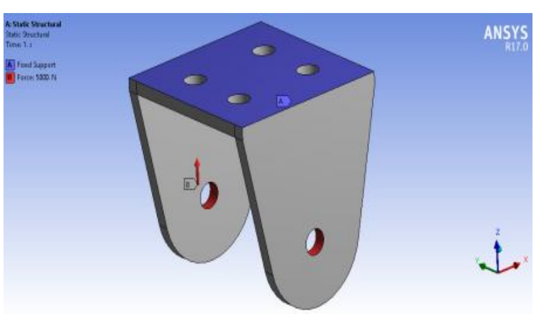

Figure 5. Boundary conditions

- Post Processing

a) Von- Mises

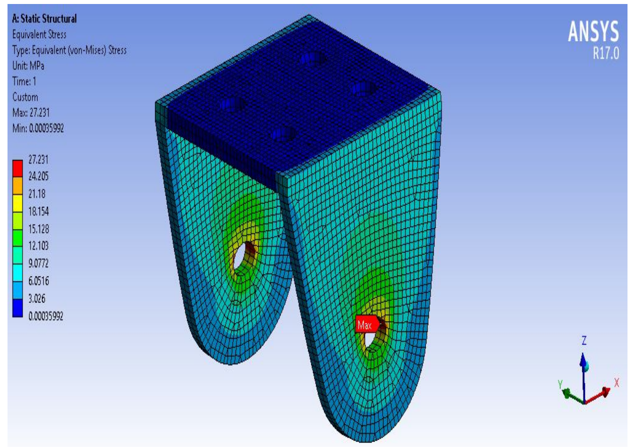

Figure 6. Von- Mises stress of leaf spring mounting bracket Maximum von-mises stress is 27.231MPa.

b) Deformation

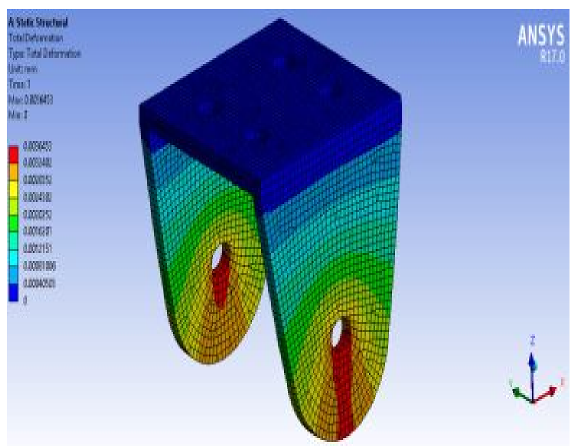

Figure 7. Deformation of leaf spring mounting bracket

VI. OPTIMIZATION

Engineering is a field where concept of nature is applied to build useful part. A mechanical engineer designs a new machine, or a vehicles shock absorbers or a robot. A civil engineer designs of hydro project or a nuclear plant. A chemical engineer designs a milk product plant , sugar industry or a chemical process. An electrical engineer designs a transformer or an integrated circuit. For lots of reasons, not the least of which is the competitive world, an engineer may not only be interested in a design which works at some sort of nominal level, but is the better design in some way. The process of choosing the better design is called optimization. Often engineering optimization is done absolutely. Using a integration of perception, skill, modelling, advice of others, etc. the engineer create design conclusion which, he or she expect, lead to an optimal design. However, if there are many variables to be managed with several objectives and constraints, this type of experience-based optimization can fall short of recognised the best design. The interactions are too critical and the variables too countless to intuitively measurethe optimum design. In this paper we discuss a design optimizationon computer-based. With this approach, we use the computer to research for the best model according to criteria that we mentioned. The computer's enormous processing capacity allows us to discover many more design integration than we could do manually. Further, we employ detailed algorithms that enable the computer to systematically search for the optimum. We can then see if any development can be made. In order to apply this type of optimization, several qualifications must be encounter. First, we must have a quantitative model available to calculate the responses of interest. If we want to minimize cost, we must be able to determine cost. Sometimes achieve such quantitative models is not easy. Obtaining a valid, perfect model of the design problem is the most important part in optimization. It is common for 90% of the performance in optimizing a design to be spent on developing and validating the quantitative model. Once a good model is achieved, optimization results can often be noticedquickly. Although engineering models are usually physical in nature we can also use actual models (based on the results of test). It is also perfectly sufficient for models to be solved numerically (using, for example, the FEM). that are devoted to applications of these method to hard optimization problems in other areas of engineering.

1. OPTIMIZED MODEL

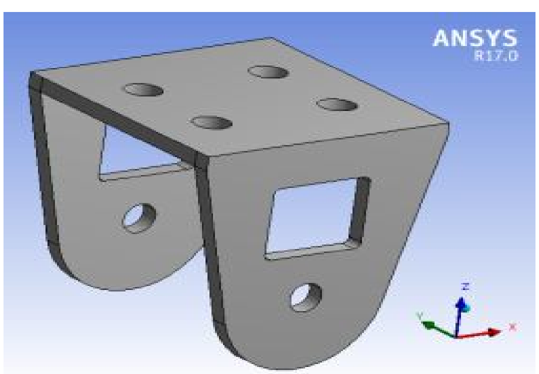

Figure 8. CAD Geometry of Optimized model

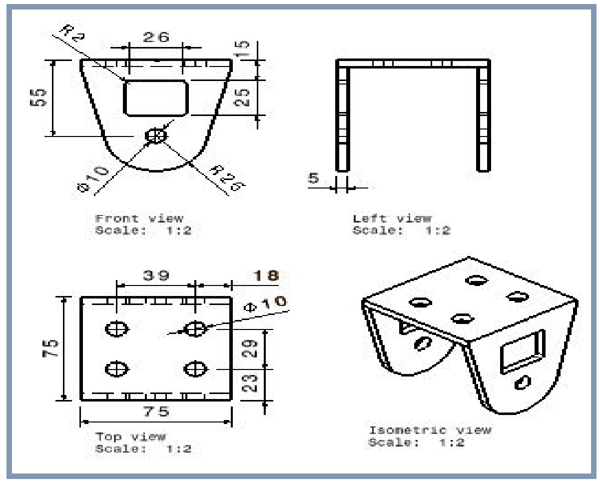

Figure 9. CAD Drafting of Optimized model

2. Boundary Conditions:

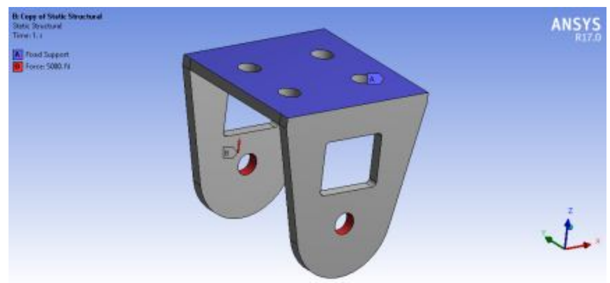

Figure 10. Boundary conditions on optimized model

3. Von-Mises:

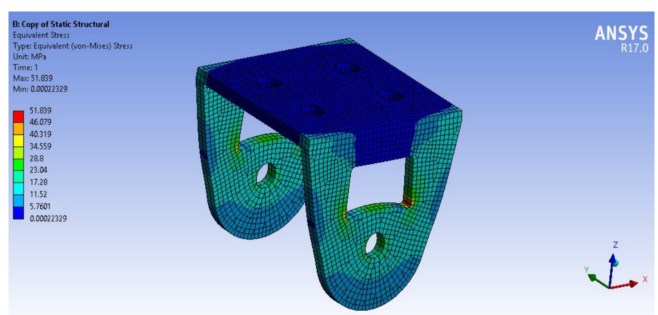

Figure 11. Von- Mises stress of Optimized model

Maximum von-mises stress is 51.839 MPa

4. Deformation:

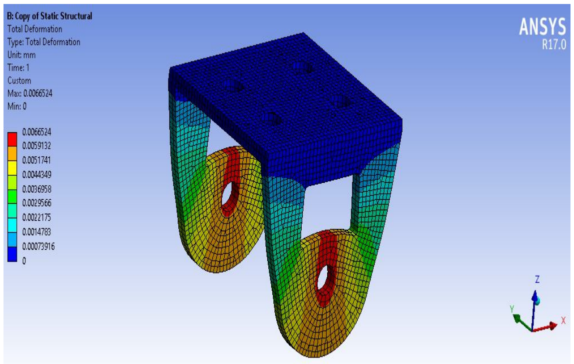

Figure 12. 0Deformation of Optimized model

Maximum Deformation is 0.007mm.

VII. EXPERIMENTAL STRESS ANALYSIS

1. Strain Gauge:

Mounting Strain Gages

1. Lightly etch a line over the side of the load cell body at the exact height of the hole centre, by using awl
2. mounting area should be cleared with lacquer thinner, acetone or methyl alcohol.
3. Place the strain gauge in such position FOIL SIDE UP Using tweezers; line up the longitudinal gauge(s) such that the grid centre alignment marks line up with the etched line. Don't touch the gauges with your fingers.
4. The proper location of these gages is not important, but it is essential that they are aligned in the transverse direction.
5. Carefully attach the tape over the gauges. Push the tape down onto the load cell body; make sure that the gauges do not become dislocated.
6. Next,. Leave the one end attached to the load cell body, take off other end of the tape such that the gages are exposed
7. Using a toothpick, put a small amount of epoxy adhesive to the underside of each gauge.
8. Once again put carefully masking tape to the load cell body. Push firmly down over each strain gauge
9. Permit the adhesive to cure fully, and then remove the masking tape.
10. Remove 2 mm of insulation from the lead wires. retain stripped ends of lead wires onto the solder pads, then use a piece of cellophane tape to carry lead wires in place.
11. Align edge of tape to edge of gage Lead wires Epoxy
12. Apply a small amount of epoxy to bond the lead wires to the body of the load cell
13. Wire movement should be such that it will not produce strain (stress) to the strain gauge solder pads.
14. Make sure that epoxy does not go along the lead wires such that it reaches the solder pads. permit the epoxy to fully cure before going to the next step.
15. Carefully detach the cellophane tape. Make sure that lead wire ends stand in contact with the solder pads.
16. Permit the soldering gun to attain operating temperature. Make sure the gun tip is clear (no oxide).
17. Touch the gun tip opposite to the wire. Solder will quicklygo into the wire and solder pads, forming a tiny solder mound.
18. Verify the resistance for each gage is 350±1 ohm.By the use of ohmmeter,
19. Neatlyclear excess solder flux using alcohol and a cloth or paper towel.
20. put a thin layer of epoxy on the entire strain gage to give environmental protection.

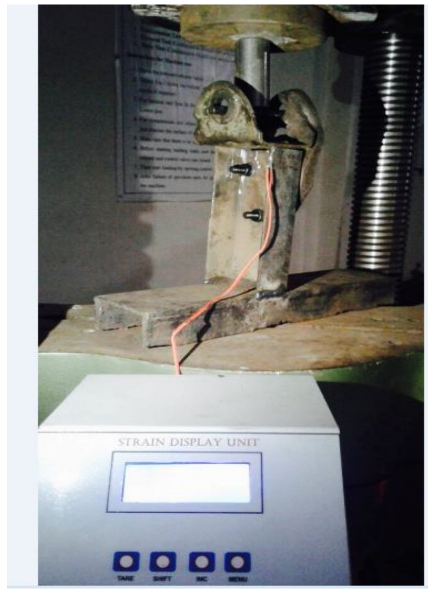

Figure 13. Experimental setup of UTM

2. Experimental Strain Gauge setup:

Figure 14. Experimental Strain Gauge Reading.

VIII. COMMENT ON MANUFACTURING

For basic model their are milling, drilling, VMC, grinding(if necessary) operation involved. For the basic model the costing goes to 2691.0Rs (with considering all allowances) And for same optimised model machining area increased thats why side by side cost also increased which is near about 2961.0 Rs only.

COMPARISON

Table 1.

	FEA	EXPERIMENTAL
STRAIN	259µS	266µS

Above table gives cpmaparison between strain values what we get from FEA and by Actual Experiment.

FATIGUE LIFE ANALYSIS:-

Table 2. Actual results of no. of cycles & stress from analysis

	B	C
1	Cycles	Alternating Stress (MPa)
2	10	3999
3	20	2827
4	50	1896
5	100	1413
6	200	1069
7	2000	441
8	10000	262
9	20000	214
10	1E+05	138
11	2E+05	114
12	1E+06	86.2

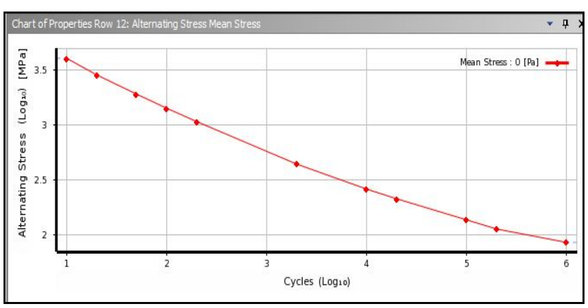

Figure 15. SN Curve and result table for stress and no. of cycles

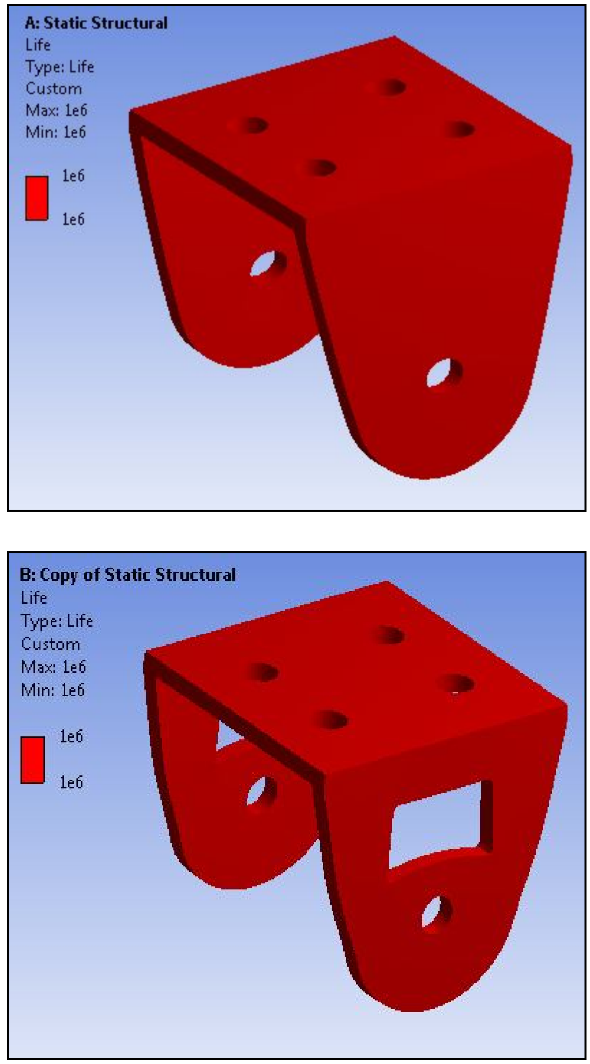

Figure 16. Comparison of fatigue results of the actual model and the optimized model.

IX. RESULT &CONCLUSION

From results of finite element analysis it is observed that the maximum stress value is within the safety limit. There is a great potential to optimize, this safety limit which can be done by removing material from low stressed region thus optimizing its weight without affecting its structural behavior. The maximum displacement value is also very less.

Von-mises stress found on existing (27.231 MPa) and optimized (51.839 MPa) components are within the material yield strength.

Deflection measured and found on existing (0.004mm) and optimized (0.007mm) model is very less.

With this topological optimization a weight saving of 11.2% from existing (0.527kg) to optimized (0.468kg) component.

Both components (before optimization & after optimization) have infinite life (from ANSYS Fatigue life Analysis.)

X. ACKNOWLEDGEMENT

I wish to express my sincere thanks to Prof. Thakre S.B.(Principal of APCOER, Parvati) for providing me with all the necessary facilities for the research. I place on record, my sincere thank you to Prof. K.H. Munde (PG Cordinator) for the continues encouragement.Iamextremly thankful and indebted to Prof. G.E. Kondhalkar (HOD Mechanical Departmentas well as Project Guide) for sharing expertise, and sincere and valuable guidance and encouragement extended to me.I take this opportunity to express gratitude to all of the Department faculty members for their help and support.

REFERENCES

[1] Sourav Das, “Design and Weight Optimization of Aluminium Alloy Wheel” International Journal of Scientific and Research Publications, Volume 4, Issue 6, June 2014 ISSN 2250-3153.

[2] S. Chaitanya, and B.V.Ramana Murty, “Mass Optimization of Automobile Wheel Rim” International Journal of Engineering Trends and Technology (IJETT) – Volume 26 Number 3- August 2015.

[3] Ch. P. V. Ravi Kumar, and R. Satya Meher, “Topology Optimization of Aluminium Alloy Wheel”, International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue. 3, May.-June. 2013 pp-1548-1553 ISSN: 2249-6645

[4] Turaka Venkateswara Rao, and Kandula Deepthi, “Design & Optimization of a Rim Using Finite Element Analysis”, International Journal of Computational Engineering Research (IJCER), ISSN (e): 2250 – 3005, Vol, 04, Issue, 10, October – 2014.

[5] Mr. Sushant K. Bawne, and Prof. Y. L. Yenarkar, “Optimization Of Car Rim”, International Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol. 5, Issue 10, (Part - 2) October 2015.

[6] BGN Satya prasad, M Anil kumar, “Topology Optimization of Alloy Wheel”, Altair Technology Conference, India-2013.

[7] D. H. Burande, and T. N. Kazi, “Fatigue Analysis of Alloy Wheel for Passenger Car under Radial Load”, International Journal of Engineering Research and General Science Volume 4, Issue 2, March-April, 2016, ISSN 2091-2730.

[8] H. N. Kale, and Dr. C. L. Dhamejani, “COMPARATIVE STUDY OF WHEEL RIM MATERIALS”, IJARIIE-ISSN(O)-2395-4396, Vol-1 Issue-5 2015.