International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 07, July-2020

Structural Analysis and Topology Optimization of Leaf Spring Bracket

Ms. Shraddha R. Gunjawate
ME Design Student, Department of Mechanical Engineering
Sinhgad Academy of Engineering,
Pune, India.

Dr. Shrishail B. Solapur
Assistant Professor, Department of Mechanical Engineering,
Sinhgad Academy of Engineering,
Pune, India.

***Abstract*— Leaf spring is a basic form of suspension, widely used in commercial vehicles like tempo, truck etc . It is made up of layers of long steel plates of varying lengths, sandwiched one upon the other. Leaf spring is attached to chassis with the help of leaf spring mounting bracket. Shape and size of the bracket differs from vehicle to vehicle. As bracket acts as a link between chassis and suspension, it can be redesigned in optimum manner, such that it will have low weight and can efficiently transfer stresses without failure. In this paper, we have done weight optimization of leaf spring bracket of LCV Diesel truck, using Topology optimization method. Leaf spring bracket is modeled using CREO 2.0 software. Real condition forces and constrains are applied on bracket. Static and fatigue analysis is done using Ansys software. Topology optimization of leaf spring bracket is done in Ansys 19. Final optimized bracket is redesigned as per result generated by topology optimization in Ansys and manufacturing feasibility. Static and fatigue analysis is carried out on an optimized bracket design. It is observed that stress value in original and optimized bracket remains same. Strain gauge testing is conducted on optimized bracket. Strain value measured on strain gauge at 6000N load is 153.2 microstrain and strain value measured in FEA at same load and location of strain gauge is 180 microstrain. Weight of the bracket is reduced from 1.8Kg to 1.6 Kg i.e. 200 grams, which is equal to 12% of original bracket weight. Hence, Weight optimization of Leaf spring bracket is done using Topology optimization method, without degrading the bracket performance with respect to original bracket design.**

Keywords—Leaf Spring Bracket, Sptrain Guage, Topology Optimization.

I. INTRODUCTION

In recent years automotive industries are emphasizing on increasing vehicle efficiency by optimizing part weight without compromising on part quality. Optimization is mainly divided into three types. Shape optimization, Size optimization and Topology optimization. Shape and size optimization is the traditional method of part optimization, followed in industries from many years. Topology optimization is upcoming way of optimization, which consist of mathematical method of optimizing material layout within a given design space, with pre-defined loads and constrains, with the objective of minimizing weight, volume or maximizing performance. Topology optimization is done using FEA software. There are various algorithms used for topology optimization. Most of the optimization software use density based approach, in which each element in the design space will have its own stress level and strain energy. Elements having higher stress level (one) are retained and elements with zero stress level are eliminated unless overall structure is not affected. Topology optimization targets number of elements to get rid of, which depends on input values given by designer.

In this paper, Topology optimization method is used for weight optimization of leaf spring bracket. Leaf spring bracket is the connecting link between the leaf spring and chassis. It is non-movable part and hence has good scope of optimization. Leaf Spring bracket of LCV Diesel truck has been analyzed and optimized to reduce its weight without compromising the strength. 3-D modeling of part is done in Creo 2.0 software. Analysis and topology optimization is done in Ansys 19 software.

Problem Statement is to optimize weight of leaf spring bracket using mathematical optimization technique i.e. Topology optimization using Ansys. Compare FEA results of original and optimized bracket. Perform experiment validation using strain gauge testing.

Figure 1 gives the methodology used for Topology optimization.

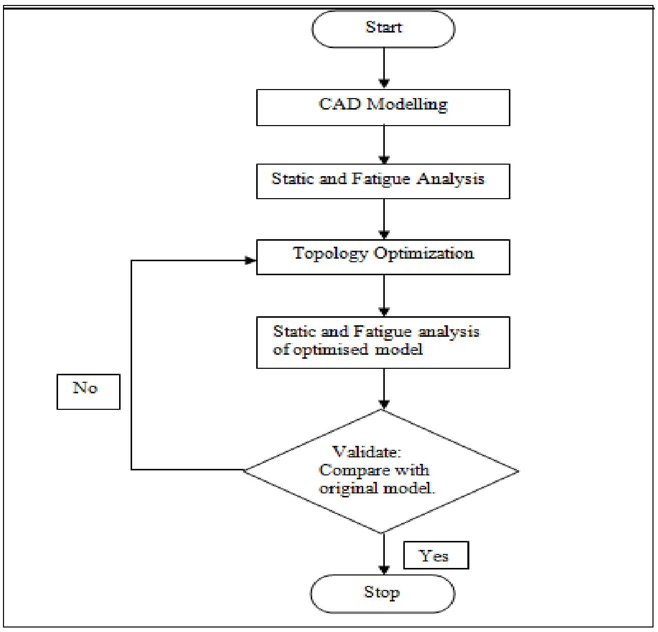

Fig.1. Methodology of Topology Optimization

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 07, July-2020

II. ANALYSIS OF LEAF SPRING BRACKET

Leaf spring bracket specifications:
1. Vehicle – LCV Diesel truck
2. Material - SS4012a-E34 [5]. Bracket Material specifications are given in table 1.

TABLE I. Material Specifications

Sr. No.	Parameters	Value
1	Young Modulus	210 GPa
2	Poisson ratio	0.3
3	Endurance limit	170 MPa
4	Yield stress	340 MPa
5	Ultimate tensile stress	400 MPa

A. Force Calculation on Each Leaf Spring Bracket

1. Gross weight of the vehicle [9]: 7490Kg
2. Front axle weight [9]: 2400Kg
3. Brackets per axle : 4 (2 for left and 2 for right wheel)
4. Weight on each front wheel bracket: 2400/4= 600Kg
5. Force on each bracket: 5886 N ~ 6 KN

B. CAD model of hanger

Figure 2 shows the original and CAD model of leaf spring bracket.

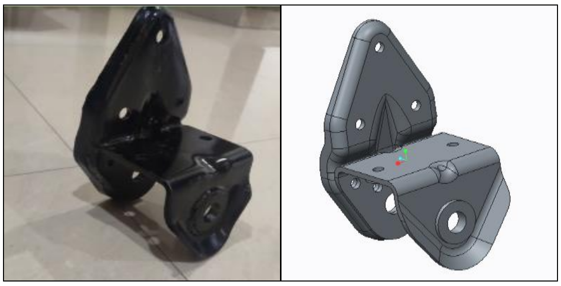

Fig.2. Original and Optimized Leaf Spring Hanger

C. Load and Constrains

After studying original bracket assembly in vehicle, constrains and loads are applied in following manner:

1. 6000 N vertical load is applied on big holes, through which pin carrying one end of leaf spring passes. Vertical load is applied, as force is experienced in vertical direction when vehicle passes from pothole.
2. 3 holes on vertical triangular plane and 2 holes on horizontal plane of bracket are bolted to chassis. Hence they are fixed.
3. Displacement of bracket in Y direction is zero. Hence horizontal plane is given zero displacement in Y direction.

Refer Figure 3.showing loads and constrains applied in leaf spring hanger.

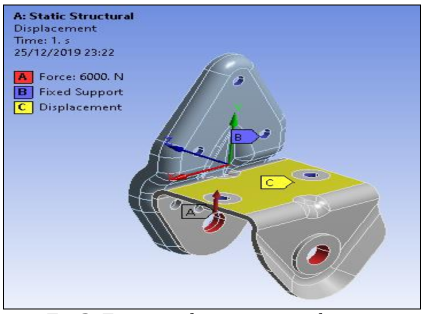

Fig.3. Forces and constrains on hanger

D. Statistic Analysis of Original Bracket

After applying constrains and forces on bracket, static analysis is done in Ansys. Total deformation, stress and strain are measured on original bracket. Refer Figure 4, Figure 5, and Figure 6 for the total deformation, equivalent stress and equivalent strain.

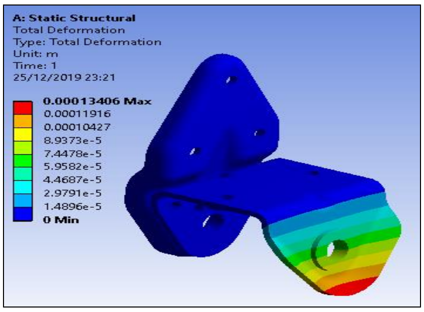

Fig.4. Total Deformation on Original Bracket

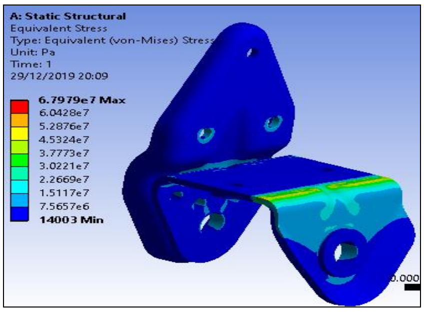

Fig.5. Equivalent Stress in Original Bracket

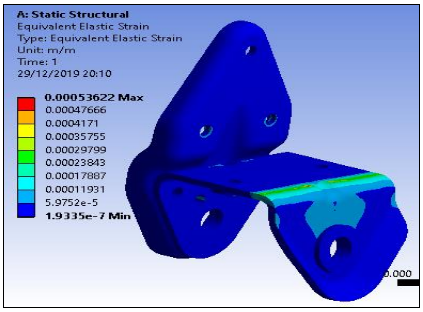

Fig.6. Equivalent Strain in Original Bracket

Total deformation in Original bracket is 0.13mm.
Equivalent Stress in Original bracket is 67.9 MPa.
Equivalent Strain in Original bracket is 536 micronstrain

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 07, July-2020

E. Fatigue Analysis of Bracket

1) Theoritical Calculation:

1. Ultimate stress of bracket= Sut = 400 MPa
2. Endurance limit of bracket= Se = 170MPa.
3. Equivalent stress on bracket= S = 68 MPa

Refer Figure 7 in which Log S vs. Log N curve is plotted. Equivalent stress in bracket is less than endurance limit. Hence life of bracket is more than 1 million cycles. In below diagram it can be seen that at 68MPa can bracket can sustain up to 10^{12} cycles.

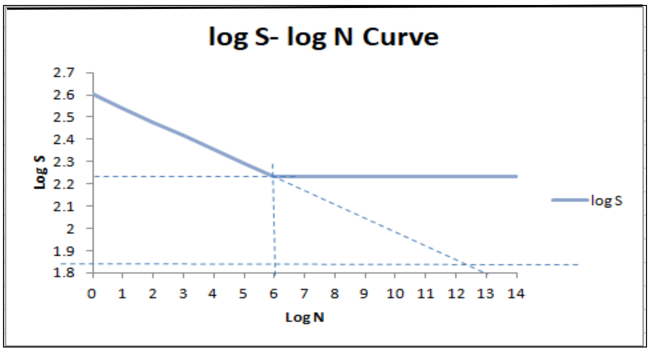

Fig.7. Log S vs. Log N Curve

2) Ansys results

Fatigue life of bracket is more than one million cycles, both theoretically as well as in FEA. Refer Figure 8, for fatigue life of bracket. Also it is observed that Factor of safety for most of the bracket is 15, whereas in automobile factor of safety is maintained up to 2. Hence there is large scope of bracket optimization. Minimum factor of safety is 2.018. Refer Figure 9 for factor of safety.

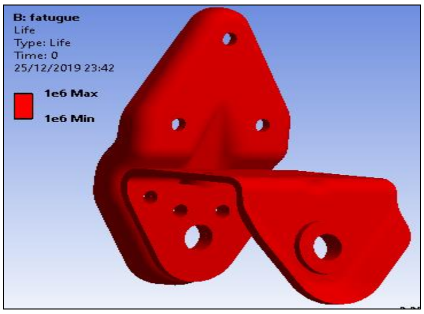

Fig.8. Fatigue Life of Original Bracket

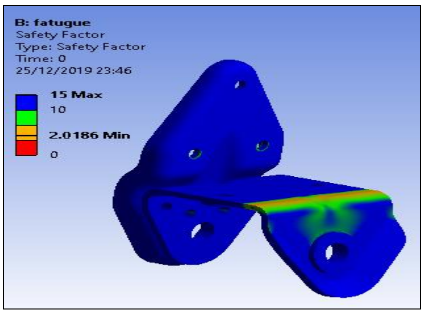

Fig 9. Factor of Safety of Bracket

III. TOPOLOGY OPTIMIZATION

Optimization of leaf spring bracket is done using Topology optimization module in Ansys workbench 19.

1) Step 1: Define Optimization Region

Define design and non-design region. Non-design area is that part of the bracket which is attached to other assembly parts, whereas design area is that part of the bracket which is not in touch with any peripheral parts. Hence all bolted joints and loaded area is non-design region and remaining area of bracket is considered as design region. Figure 10 shows the design and non-design region defined for bracket under study.

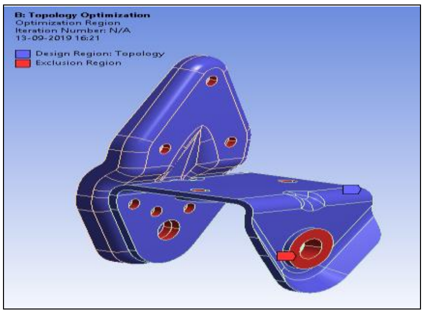

Fig.10. Design and Non-Design Region

2) Step 2: Define Objective

Objective is to minimize the compliance.

3) Step 3: Give Response Constrain

We want to optimize 20% of bracket mass, hence our response constrain is to retain 80% of mass.

4) Step 4: Solution

Topology optimization run's various iterations to reach to final solution. Below Figure 11 shows the topology optimized bracket result generated by Ansys.

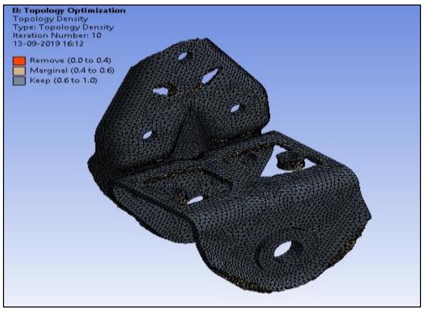

Fig.12. Topology Optimization Result from Ansys

After getting topology optimization result, 3D model of optimized bracket is created in CREO. Final bracket is designed considering manufacturing feasibility, hence does not match exactly with the topology optimization result.

From the result obtained in topology optimization, horizontal plane of bracket have square shape cut-out which can be manufactured, but vertical triangular plane cutout need to be redefined based on manufacturing feasibility. Hence we studied carried out shape optimization here, to select best shape from 3 basic shapes i.e. Triangle, Rectangle and Circle.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 07, July-2020

Shape	Max equivalent stress	Stress at local area of shape	Stress at shape corners
Original	67.97 MPa	NA	NA
Square	66.8MPa	9.8 MPa	10.17 MPa
Circle	66.7 MPa	9.3 MPa	NA
Triangle	64.6 MPa	9.7 MPa	12.02 MPa

TABLE II. Comparison of cutouts made in vertical portion of hanger

Below Figure 13, Figure 14, Figure 15 shows basic shape cut-stresses generated due to cutout.

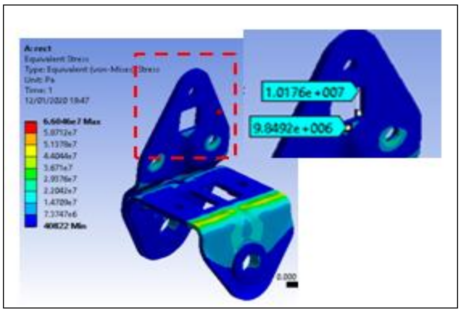

Fig.13. Square cutout- Equivalent stress

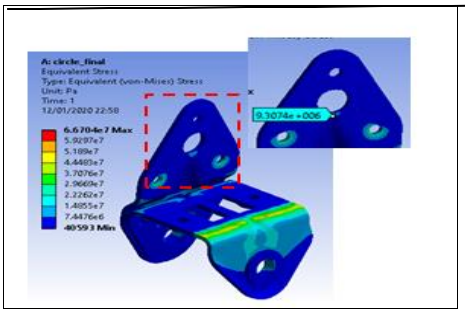

Fig.14. Circular cutout- Equivalent stress

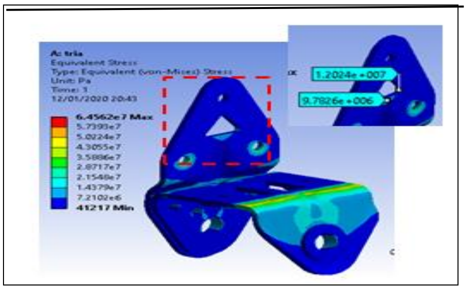

Fig.15. Triangular cutout- Equivalent stress

Refer below table 2, which gives comparison between basic shape cutouts made in vertical triangular portion of bracket. From column 2, it can be observed that maximum equivalent stress value in all 3 geometries is equal and same as original bracket. Column 3 gives stress generated at local area due to cutouts and is not applicable for Original bracket design, as it does not have any cutout. But for remaining three shapes, stress value is almost same and very less compared to yield stress of bracket. Column 4 in the table, gives stress generated at corner of various shapes, it can be seen that circle shape cutout does not have corner stresses. Hence we selected circular shape.

Thus, considering both topology optimized results and manufacturing feasibility final optimized bracket is made in CAD model. Refer Figure 16 for final optimized bracket.

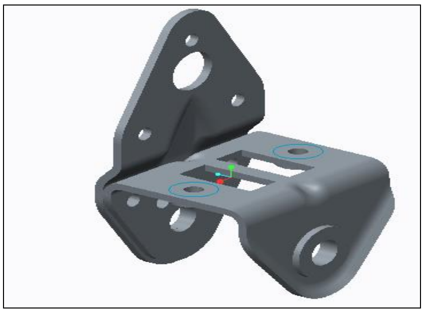

Fig.16. CAD Model of Topology Optimized Bracket

IV. ANALYSIS OF OPTIMIZED BRACKET

A. Static Analysis of Optimized Bracket

Force and constrains, similar to original bracket are applied on optimized bracket. Static analysis is done in Ansys. Total deformation, stress and strain are measured on optimized bracket. Refer Figure 17, Figure 18 and Figure 19 for the total deformation, equivalent stress and equivalent strain.

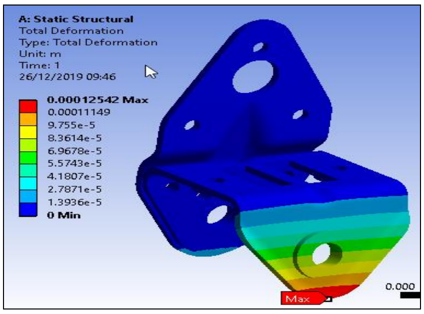

Fig.17. Total Deformation in Optimized Hanger

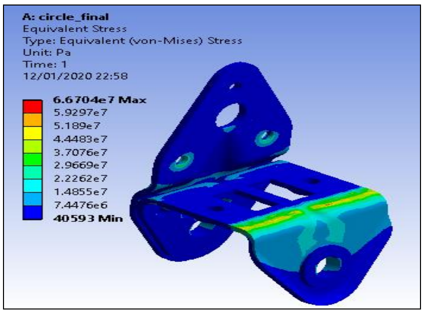

Fig.18. Equivalent Stress in Optimized Hanger

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 07, July-2020

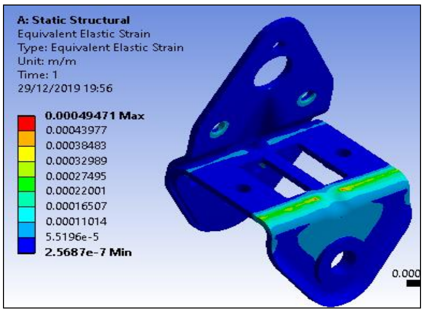

Fig.19. Equivalent Strain in Optimized Hanger

Total deformation of original bracket is = 0.125mm.
Maximum equivalent stress of original bracket is = 66.7 MPa.
Maximum equivalent strain of original bracket is = 490 microstrain

B. Fatigue Analysis of Optimized Hanger

Fatigue life of Optimized bracket is more than one million cycles in FEA. Refer Figure 20, for fatigue life of bracket. Also it is observed that Factor of safety for most part of the bracket is 15. Minimum factor of safety is 2.02. Refer Figure 21 for factor of safety. Hence optimized bracket is also robust.

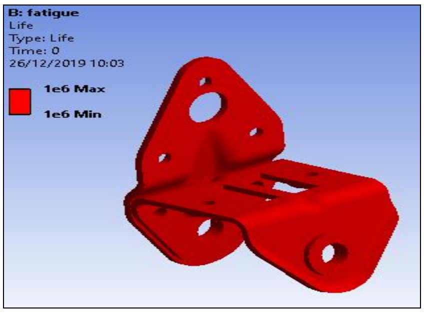

Fig.20. Fatigue life of Optimized Bracket

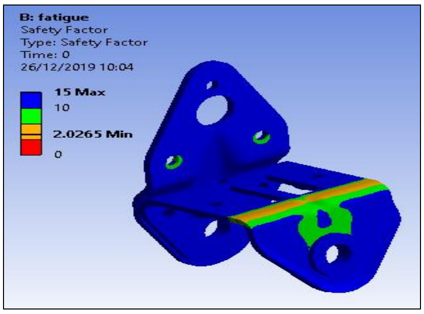

Fig.21. Factor of Safety of Optimized Bracket

V. Manufacturing and Experimental Validation

A. Manufacturing

Optimized leaf spring bracket is manufactured using three machining operations which include wire cutting, vertical drilling and milling. Below figure 22, figure 23 and figure 24 shows operations used for making of optimized bracket. Figure 25 shows the final optimized bracket.

Fig.22. Wire Cutting for cutting square profile

Fig.23. Vertical Drilling for Making Circular Profile

Fig.24. Milling for Cutting Flanges

Fig.25. Optimized Bracket

Leaf spring bracket of LCV Diesel Truck is made up of sheet metal hence horizontal slot and hole can be made using punching operation in actual part.

Further, for experimental validation boundary conditions need to apply. Hence fixture is manufactured using welding process as shown in figure 26.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 07, July-2020

Fig.26. Wire Cutting for cutting square profile

B. Experimental Validation

For experimental testing, Strain gauge testing is done using UTM machine which is used to apply load of 6000 N on Leaf spring bracket [1]. Refer figure 27 showing strain gauge testing setup. Strain gauge should be mounted at the location on bracket having maximum stress value. But in leaf spring bracket, maximum stress is obtained on bend. It is difficult to apply strain gauge on bend portion, as it may not adhere properly to the surface. Hence strain gauge is mounted on horizontal surface of the bracket, near to maximum stress location. Refer Figure 28 showing strain gauge location. Load is gradually increased from 0 to 6000N and strain value is recorded using data logger. Strain value obtained is converted into stress to get the respective stress value. Strain value obtained on strain gauge is 153.21 microstrain at 6000N load, which is equal to 31.5 MPa stress, which is very less than yield stress of bracket.

Fig.27. Strain Guage Testing

Fig.28. Strain Guage Location

Refer Figure.29, which shows Force Vs Strain graph obtained from strain gauge testing data. From graph we can see, as the load is increased gradually from 0 to 6000N, strain value also increases. Strain value obtained at 6000N is 157 microstrain, at which stress value is 31MPa, which is within yield limit.

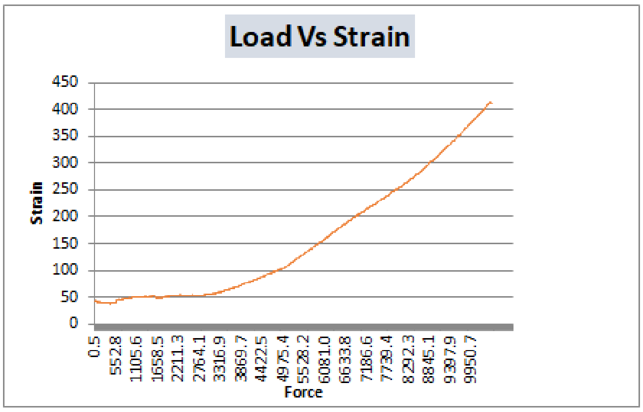

Fig.29. Load Vs. Strain Graph

To get the correlation between experimental and FEA results, experimental strain values are compared with FEA strain values at strain gauge mounting location, for load increasing from 0 to 6000N . At 6000N load, experimental strain value is 157 microstrain i.e. 31.5 MPa and FEA strain value is 180 microstrain i.e.36 MPa . Thus strain and stress values obtained experimentally and FEA are almost same and are within Yield stress limit. Refer Figure 30 showing Load vs. Strain graph Experimental and FEA data.

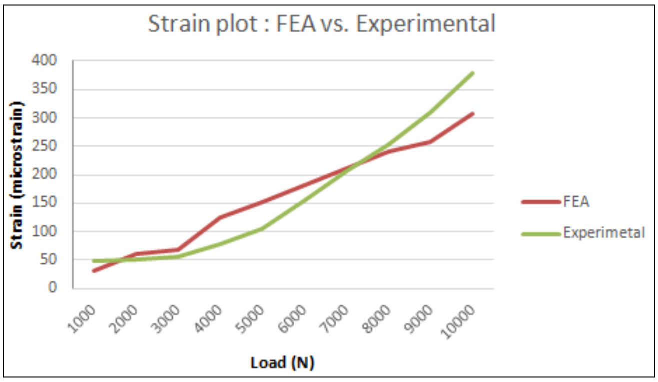

Fig.30. Load Vs. Strain Graph, Experimental and FEA

Thus, strain values obtained experimentally and in FEA at strain gauge location are same. Thus, experimental results are in line with FEA results. Hence, maximum stress value obtained will also be same as that of stress value obtained in FEA, which is 66.7 MPa. Hence, stress values of the optimized bracket are within the yield stress limit.

VI. RESULTS

Refer table 3, showing comparative results of original and optimized leaf spring bracket. We observe that FEA results for original and optimized bracket are almost same and weight of part has reduced from 1.8 Kg to 1.6 Kg, which is equivalent to 200 grams. Hence we obtained 12% mass reduction.

TABLE III FEA Results For Original and Optimized Bracket

Sr.No.	Parameter	Original bracket	Optimized bracket
1	Displacement	0.13 mm	0.125 mm
2	Stress	67.9 MPa	66.79 MPa
3	Fatigue life	INFINITE	INFINITE
4	FOS	2.018	2.02
5	Weight	1.8 Kg	1.6 Kg

From below table 4, we observe that experimental result match with the FEA results at strain gauge location. Hence, maximum stress value obtained experimentally is also same as FEA (ie. 66.7 MPa.), which is below yield stress value with FOS 5. Hence, optimized leaf spring bracket is robust.

TABLE IV Experimental Results vs. FEA Results

Model	FEA strain (microstrain) at strain gauge location	Experimental result (microstrain) at strain gauge location
Optimized bracket @ 6000 N	180.7 microstrain	153.216 microstrain

VII. CONCLUSION

1. Topology optimization technique can be effectively used for part optimization.
2. By using topology optimization method, weight reduction of 200 grams ie 12% has been achieved, without increasing component stress.
3. Maximum Equivalent Stress in original bracket is 67.9 MPa and maximum equivalent stress in optimized bracket is 66.79 MPa. Hence stresses in the optimized bracket does not increase, thus life of bracket is not affected due to optimization.
4. Minimum factor of safety for original and optimized bracket is 2 and maximum factor of safety is 15. Hence optimized bracket is robust.
5. Experimental validation is done by strain gauge testing. Strain value measured at location of strain gauge is 153.2 microstrain. FEA value at same location is 180.7 microstrain. Hence, experimental and FEA results are same.
6. Performance of optimized bracket remains same as original bracket. Hence topology optimization is good method of optimization.
7. Topology optimization is further more effective if additive manufacturing technique is used for making of final part as final part can be manufactured more similar to topology optimization output. So future scope is to study best manufacturing method that can be used for topology optimization part.

ACKNOWLEDGEMENT

The research support provided by the Sinhgad Academy of Engineering is greatly appreciated. Also, special thanks to the Yogeshwar steels and NDT testing laboratory, pune for providing machining and testing support respectively.

REFERENECES

[1] Jeong Woo Chang and Young Shin Lee, “Topology optimization of compressor bracket”, Journal of Mechanical Science and Technology, vol. 22 , pp. 1668~1676, April 2018.

[2] G. Chiandussi, I. Gaviglio, and A. Ibba, “Topology optimisation of an automotive component without final volume constraint specification”, Advances in Engineering Software, vol. 35, pp. 609–617, August 2004

[3] Po Wu, Qihua Ma, Yiping Luo and Chao Tao, “Topology optimization design of automotive engine bracket”, Energy and Power Engineering, vol.8, pp. 230-235, April 2016

[4] Amar Vhanshetti, Nitin S. Chava, “Design of integrated super bracket for heavy commercial vehicles”, International Journal of Advanced Engineering Research and Science (IJAERS), vol.2 , issue-1, ISSN: 2349-6495 , January 2015

[5] C.Radhakrishnan, Azhagendran.K, Mohanlal.K, Ponraj.P and Nivas.R, “Design and analysis of automotive shackle “,International Journal of Innovative Science, Engineering & Technology, vol. 2 issue 5, ISSN 2348 – 7968, May 2015

[6] Saverio Giulio Barbieria, Matteo Giacopinia, Valerio Mangerugaa and Sara Mantovani, “A design strategy based on topology optimization techniques for an additive manufactured high performance engine piston”, Procedia Manufacturing, vol. 11,pp. 641 – 649, June 2017

[7] Mahesh Dasgaonkar, Abhishek Hambar and P Baskar, “Design optimization and analysis of leaf spring using static load conditions”, International Journal for Research in Applied Science & Engineering Technology (IJRASET), vol. 4 Issue VII, ISSN: 2321-9653, July 2016

[8] Raymond Browell and Al Hancq, “Calculating and Displaying Fatigue Results”, (Book type)