Received Date: 27/05/2020 Acceptance Date: 12/06/2020 Publication Date: 30/06/2020

Topology Optimization of Mechanical Bracket using Finite Element Analysis

[1]Abhinav, [2]Arun Kumar B, [3]Adityakumar Anant Hegde

[1]Assistant Professor, [2,3] Undergraduate Student,
[1, 2, 3] Mechanical Engineering,
[1, 2, 3]Alliance University, Bangalore, India
Email – [1]abhinavtechno5@gamil.com, [2]arunaidu09817@gmail.com ,[3]adityakumarananthegde@gmail.com

Abstract: *The objective of the work is to optimize the mechanical bracket. Reduction in the total mass, reduction in von-mises stress, and reduction in the total deformation were prime motive to be achieved from the specially designed bracket. Optimization was carried out at the specific bearing load of 111.8 N. The geometry of the model, and the bracket optimization carried out using the Finite Element technique in an academic license, Ansys V19 software. Structural steel used for a bracket. Results showed that a forty percent reduction in mass significantly improved the stiffness and deformation of the optimized bracket. A decrease in von-mises stress from 86.969 to 46.134 MPa has obtained. A notable decrease in the total deformation from 0.003 to 0.002 mm also observed in an optimized bracket. The results obtained may be of a high technical significance and can recommend economical large-scale model manufacture.*

Key Words: *Optimization, Bracket, Numerical Analysis.*

1. INTRODUCTION:

Cutting, Sawing, and forming processes are the conventional material removal methods intended to accomplish weight reduction from the workpiece [1]. Fast, reliable, and lightweight product demands are the needs of today's customers. Due to advancements in software technology now, it is possible to achieve the demands above. Topology optimization using Finite element technique is one of the economic, time saving, and proven technique by which one can optimize different variables viz. increase or decrease in weight, minimize deformation, reduce the critical stress in a complex structure also, reported enormous cost saving to industries [2]. Till date the optimization techniques have applied to the development and optimization of various medical equipment's includes Prosthetic Runner Blade, ankle-foot prosthesis, etc.Aircraft and aerospace components includes: airframe structures, layout design of stiffener ribs, aircraft pylon, front fuselage, etc. and in automotive industries includes Engine Mounting Bracket, Chassis Design, Composite automotive driveshaft, etc. [3-10]. Above all, the optimization technique has been successfully applied to electrostatic precipitators, steering upright of a suspension system but not limited to [11-12]. In the present work, an effort has made to maximize the static stiffness by reducing the mass from the specially designed steel bracket manifested to minimize deformation and stress and develop an economic bracket.

2. GEOMETRICAL MODELLING AND OPTIMIZATION:

The 3-D Bracket model has been developed in Ansys Software. The thickness of the bracket has been considered 2 mm for the analysis. The bracket's critical dimensions shown in Fig. 1 (a). Three equal-sized holes have made in a bracket. Two holes have provided with cylindrical support, and a bearing load of 111.8 N applied to the remained hole. The selection of the loading condition has been set purely based on the assumption. The meshed model with the boundary conditions has shown in Fig.1(b). The SOLID186 tetrahedron element has been used to mesh the model and approximately 5870 elements generated in a bracket. Structural steel material has been considered for the bracket whose modulus of elasticity considered as E=2×105 MPa. The material properties have taken from Ansys material database. The design space and original design considered for topology optimization have shown in Fig.2.

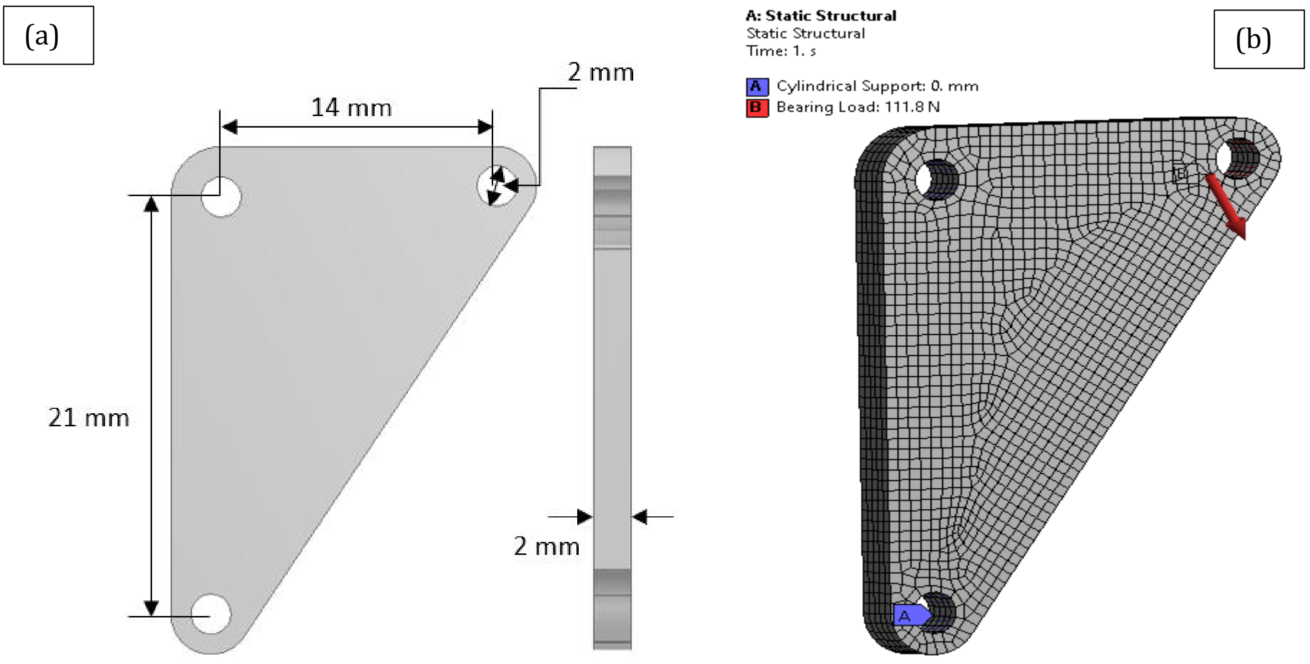

Figure 1 (a). Geometric model for topology optimization **(b)**. Meshed model

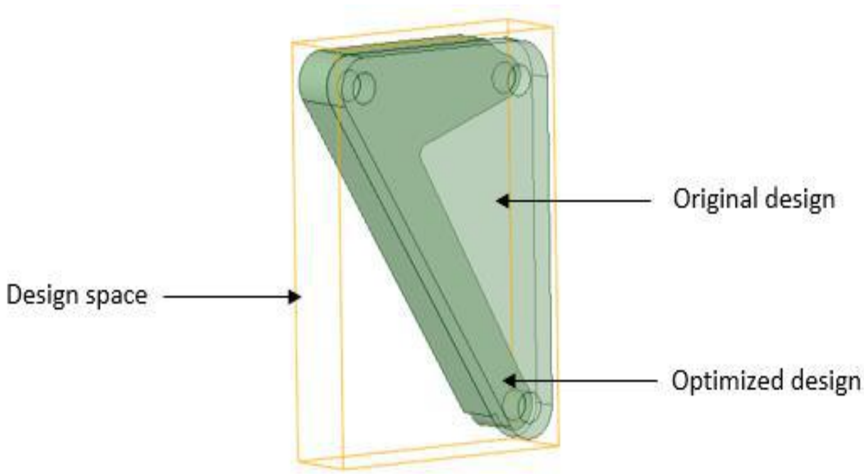

Figure 2. Design space developed for optimization

3. VALIDATION OF RESULTS:

The finite element optimization technique has found an approximate decrease in the magnitude of stress by 34%, a nearly 33% reduction in the total deformation in the optimized bracket. The comparative analysis between the original and optimized bracket shown in Fig.3. At the applied bearing load of 111.8 N, the bracket found to be structurally safe. The reason attributed to the maximum Von-Mises stress didn't exceed the yield limit of the material. The above observation is correct for both the original and optimized bracket. Fig.4 (a) shows the 40 % (red zone) of the total mass intended to remove from the bracket. The optimized bracket is shown in Fig 4 (b), which indicates the remaining 60 % of the total mass. The optimized bracket was found to have very rough contours and found it difficult to manufactured. Therefore, the bracket is further slightly reconstructed, which can ease the manufacturing and machining process. The final reconstructed optimized bracket is shown in Fig.5. The results showed a significant reduction in the maximum Von-Mises stress, i.e., from 86.969 to 46.134 MPa, refer to Fig 6 (a) & (b), respectively. The magnitude of maximum Von-Mises stress found to be maximum in the bearing load section in both cases. The von-Mises stress found to be very nominal in the entire bracket, excluding the load-bearing section, which has noticed in both the cases. The stress distribution found to be uniform in both cases. The von-Mises stress analysis of the optimized bracket clearly shows that there is still scope for a mass reduction refer to Fig.6 (b), especially near the blue coloured zone, where the magnitude of stress range varies between 0.02 to 5 MPa.

Further mass reduction can be possible, but the manufacturer suggested a trade-off with the bearing load condition. Similarly, the total deformation in an optimized bracket found to be reduced by 0.001 mm compared to the

INTERNATIONAL JOURNAL FOR INNOVATIVE RESEARCH IN MULTIDISCIPLINARY FIELD ISSN: 2455-0620 Volume - 6, Issue - 6, June – 2020
Monthly, Peer-Reviewed, Refereed, Indexed Journal with IC Value: 86.87 Impact Factor: 6.497
Received Date: 27/05/2020 Acceptance Date: 12/06/2020 Publication Date: 30/06/2020

original bracket refer to Fig.7 (a) & (b). The bracket's initial mass was calculated and found to be 4.3641 grams, and an optimized bracket mass found to be approximately 2.6191 gms.

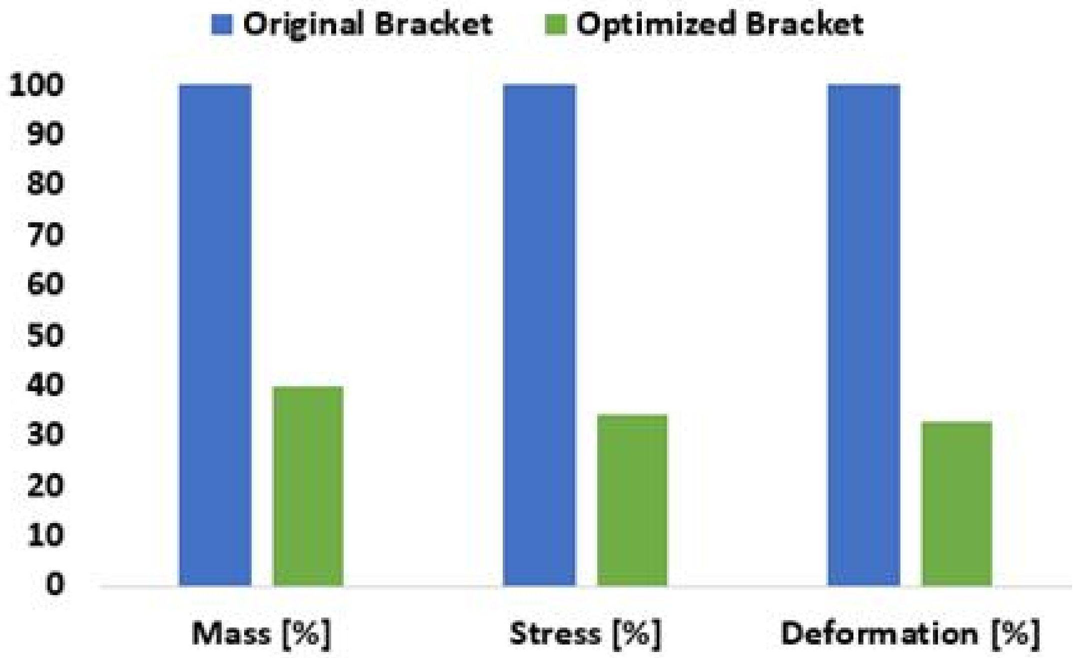

Figure 3. Comparison between original and optimized bracket

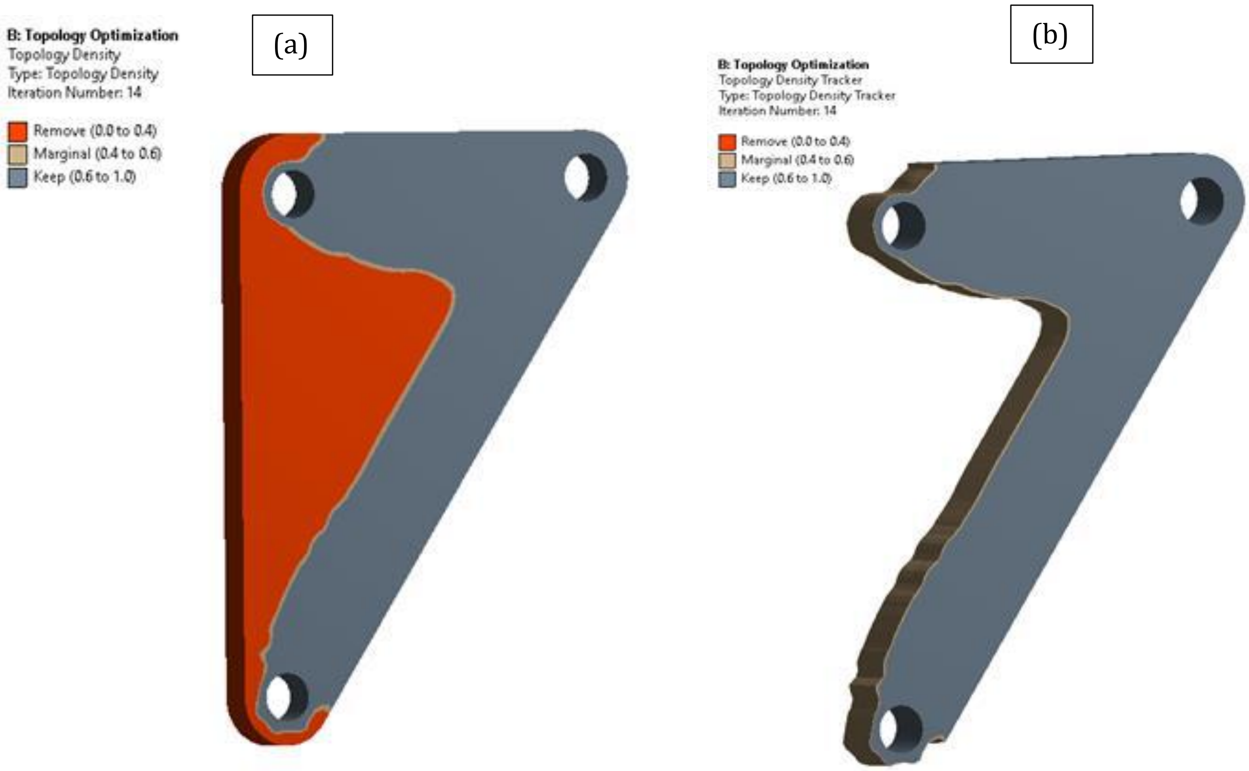

Figure 4 (a). Mass to be removed from the bracket **(b)**. Optimized bracket with rough contours

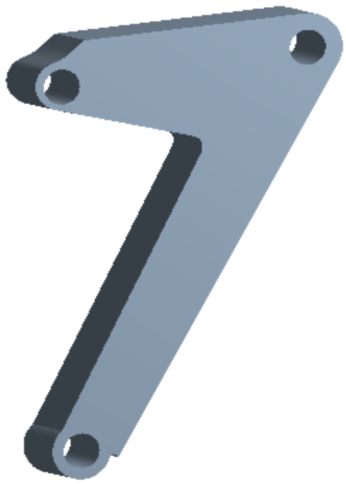

Figure 5. Reconstructed optimized bracket with smooth surfaces

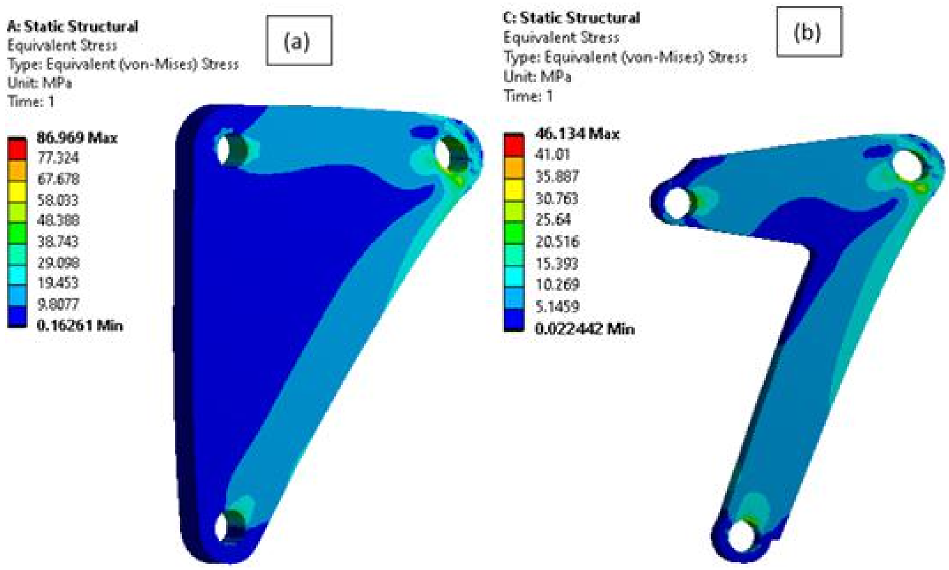

Figure 6. Von-Mises stress plot in an (a) original bracket and (b) optimized bracket

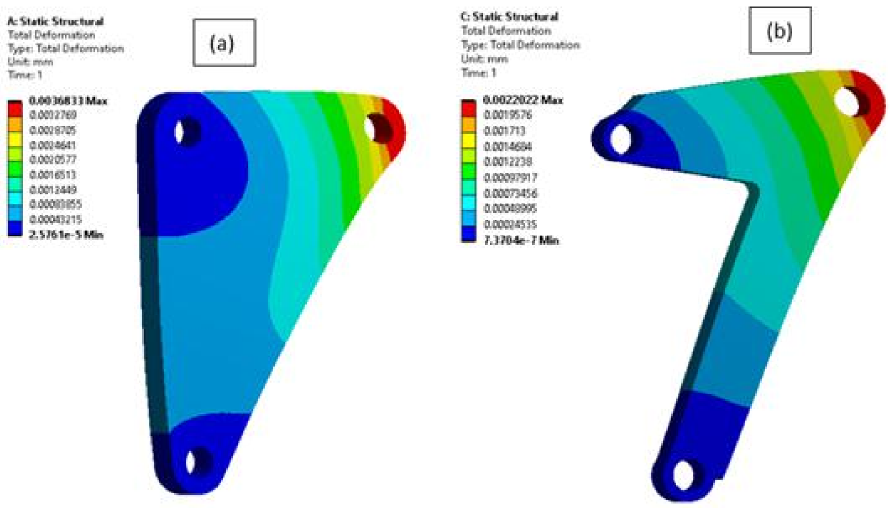

Figure 7. Total deformation plot in an (a) original bracket and (b) optimized bracket

4. CONCLUSION:

The objective of optimizing the bracket mass by 40 % made from the structural steel bracket has successfully achieved. Due to weight reduction in the bracket, a significant decrease in Von-mises stress and deformation was observed, and the reduction was reported to be nearly 33%.

4.1. Future work suggestion:

The present optimization technique can be extended to any size and can be tried with different and mix loading conditions that may result in high cost and material reduction.

4.2. Conflict of Interest: The authors confirm that this article content has no conflict of interest.

REFERENCES:

1. V. R. Naik, V. B. Magdum, A. P. Borkar (2012). Topology Optimization of Sheet Metal Brackets Using ANSYS. MIT International Journal of Mechanical Engineering, 2(2): 120-126
2. Satish M. Silaskar, Vilas B Shinde (2018). Weight Optimization of Valve for Cost Effectiveness: Using Value Analysis, 2nd International Conference on Materials Manufacturing and Design Engineering, Procedia Manufacturing 20: 329–337.
3. V.Upender, E. Srikanth, G. Karthik, Neeraj Kumar (2018). Design, modeling and optimization of a prosthetic runner blade, international journal of current engineering and scientific research,5(3): 2394-0697.
4. Dianbiao Dong, Wenjie Ge, Shumin Liu (2017). Design and optimization of a powered ankle-foot prosthesis using a geared five-bar spring mechanism, International Journal of Advanced Robotic Systems,14(3): 1–12.
5. Ji-Hong Zhu, Wei Hong Zhang, LiangXia (2016). Topology Optimization in Aircraft and Aerospace Structures Design, Arch Computat Methods Eng,23: 595–622. doi:10.1007/s11831-015-9151-2
6. ZhuJH, LiY, Zhang WH (2014). Topology optimization with shape preserving design, In: Proceedings of 5th international conference on computational methods, Cambridge, UK.
7. S. Kirthana, Mohammed Khaja Nizamuddin (2018). Finite Element Analysis and Topology Optimization of Engine Mounting Bracket, Materialstoday Proceedings, 5(9):19277-19283.
8. V. R. Naik, V. B. Magdum, A. P. Borkar (2019). Guide Bracket Optimization and Manufacturing Using Rapid Prototyping, International Journal of Innovative Technology and Exploring Engineering ,8 (11):161-165.
9. Nguyen Thanh Quang (2019). Finite Element Analysis in Automobile Chassis Design, Applied Mechanics and Materials, Trans Tech Publications, Switzerland,889: 461-468.
10. P. Satheesh Kumar eddy, Ch. Nagaraju (2017). Weight optimization and Finite Element Analysis of Composite automotive drive shaft for Maximum Stiffness, Materialstoday Proceedings,4(2):2390-2396.
11. Xuewen Wang, Bo Li, and Zhaojian Yang (2018). Finite Element Analysis and Lightweight Optimization Design on Main Frame Structure of Large Electrostatic Precipitator, Hindawi Advances in Materials Science and Engineering, Article ID 4959632, 11 pages. https://doi.org/10.1155/2018/4959632
12. Saksham Bhardwaj, Ashok B, Umashankar Lath, and Aman Agarwal (2018), Design and Optimization of Steering Upright to Reduce the Weight Using FEA, SAE Technical Paper 2018-28-0081. doi:10.4271/2018-28-0081