IJIRAE:: International Journal of Innovative Research in Advanced Engineering ISSN: 2349-2163
Volume 11, Issue 10, October 2024 https://www.ijirae.com/archives

Optimization in Modularity for Compressor Mounting Bracket: Enhancing Performance and Efficiency Through Weight Reduction

Dhende Rohit Kaluram*
Mechanical Engineering Dept,
DPCOE, Pune, India
rohitdhende544@gmail.com
Prof.Vikram Avhad
Mechanical Engineering Dept,
DPCOE, Pune, India
vikramavhad@dpcoepune.edu.in
Prof. Anantharama
Mechanical Engineering Dept,
DPCOE, Pune, India
baip1066@gmail.com

Publication History
Manuscript Reference No: IJIRAE/RS/Vol.11/Issue10/OCAE10081
Research Article | Open Access | Double-Blind Peer-Reviewed | Article ID: IJIRAE/RS/Vol.11/Issue10/OCAE10081
Received:28,September 2024|Revised:09,October 2024| Accepted:11,October 2024 | Published Online: 16, October 2024.
https://www.ijirae.com/volumes/Vol11/iss-10/02.OCAE10081.pdf
Article Citation: Dhende,Vikram,Anandtharama (2024), Optimization in Modularity for Compressor Mounting Bracket: Enhancing Performance and Efficiency Through Weight Reduction. IJIRAE:: International Journal of Innovative Research in Advanced Engineering, Volume 11, Issue 10 of 2024 pages 777-781
Doi:>https://doi.org/10.26562/ijirae.2024.v1110.02
BibTeX Key Dhende@2024Optimization

Abstract: In the automotive sector, vehicle weight plays a crucial role in determining both cost and fuel efficiency. To adhere to safety regulations while enhancing these factors, it is essential to design lighter components. For instance, air conditioners in cars are supported by brackets positioned under the bonnet. This project focuses on evaluating the design of these brackets and optimizing their weight by considering alternative materials. By reducing weight, not only are material costs lowered, but efficiency is also slightly improved. The research involves selecting appropriate materials based on the bracket's design criteria and addresses potential failure points through various analyses, such as deformation and Von Misses stress analysis. In this study, a compressor mounting bracket was designed and modeled using CAD software, followed by analysis in ANSYS. Through a design of experiments approach, the bracket was optimized, resulting in a 30% weight reduction.

Keywords: FEA, Mounting Bracket, Optimization, Structural analysis.

I. INTRODUCTION

In the pursuit of enhanced performance and efficiency in mechanical systems, the optimization of components plays a crucial role. One such component, the compressor mounting bracket, is often overlooked despite its significant impact on the overall functionality and reliability of compressor systems. Compressors are integral to various applications, including refrigeration, air conditioning, and industrial processes, where their efficiency directly affects energy consumption and operational costs. The mounting bracket serves not only as a support structure but also influences the vibration dynamics and load distribution of the compressor. An optimized bracket can reduce unnecessary weight, improve structural integrity, and enhance performance by minimizing vibrations and noise, which are critical for the longevity of the equipment and the comfort of its environment. This research aims to explore the weight optimization of compressor mounting brackets through advanced design techniques and material selection. By employing methods such as topology optimization, finite element analysis, and additive manufacturing, we can achieve a lightweight yet robust design that meets performance requirements. The findings of this study will contribute to the broader field of mechanical design and offer insights into sustainable practices by reducing material usage without compromising on functionality. Ultimately, this work seeks to pave the way for more efficient and environmentally friendly compressor systems in various industrial applications.

II. LITERATURE SURVEY

The following literature survey reviews the research work conducted on the stress concentration and optimization of cutouts in various plates and structures. These studies focus on understanding how different geometrical shapes of cutouts and loading conditions affect stress distribution, aiming to optimize design for improved mechanical performance.

K. Anusha Reddy (2017): In her study, "Relative Comparison of Geometrical Shapes for Cutouts," the author explores the effects of different geometrical cutout shapes on stress distribution in structural components. The research investigates various shapes such as circular, elliptical, and triangular cutouts and compares their influence on stress concentration factors. The study reveals that shape and orientation play a significant role in determining the extent of stress concentration around the cutouts, providing insights for selecting optimal shapes to minimize stress concentration and improve structural integrity.

Parveen K. Saini and Tarun Agarwal (2017): In their research titled "Stress Concentration around Countersunk Hole in Isotropic Plate under Transverse Loading," the authors examine the behavior of stress concentration in isotropic plates with counters unkholes under transverse loading conditions. They employ analytical and numerical methods to understand the impact of countersunk geometries on stress concentration factors. The study concludes that countersunk holes can significantly reduce the stress concentration compared to conventional circular holes, offering potential for weight and material savings in structural design.

Hardik Acharya (2017): The research "Stress Reduction Using Semi Elliptical Slots in Axially Loaded Plate Having Circular Hole" investigates the use of semi-elliptical slots as a method to reduce stress concentration in plates subjected to axial loads. By introducing these slots around a central circular hole, the study shows that the stress concentration factor can be effectively minimized, leading to enhanced load-bearing capacity and reduced risk of failure. This study emphasizes the importance of cut out shape modification in structural optimization.

Dharmendra S. Sharma (2017): In his paper "Stress Concentration around Circular/Elliptical/Triangular Cutouts in Infinite Composite Plate," the author conducts a comprehensive analysis of stress concentration in composite plates with various cutout shapes. The study, presented at the World Congress on Engineering, compares the effects of circular, elliptical, and triangular cutouts in an infinite composite plate, using numerical simulations to assess the stress distribution. The findings high light that elliptical cut outs generally result in lower stress concentration factors compared to circular and triangular ones, indicating that elliptical shapes could be optimal for applications involving composite materials.

III. OPTIMIZATION METHODOLOGY

Pre-Processing

Pre-processing involves preparing the input data required for subsequent analysis steps. In this project, ANSYS12 is used as the pre-processor. During this phase, the software processes the input information to create the necessary data files with user guidance. These files are then utilized in the next stages of analysis.

Examination/Solution

The solution stage in Finite Element Analysis (FEA) is automated. In this phase, software like ANSYS generates the element structures, calculates nodal values and derivatives, and stores the results in output files. These files are then used in the Post-Processing phase to evaluate and analyze the results through visual displays and tabulated information. The Post-Processor provides a detailed view of the simulation outcomes, allowing for comprehensive analysis of the structural behavior or response with graphical and numerical data.

Post-Processing

The results from the solution phase are presented numerically, including nodal values of field variables and derivatives, such as displacements and stresses in structural analysis. The Post-Processor interprets and presents this data graphically for verification and detailed analysis. It provides information in a user-defined format and automates the generation of graphical output. Tools like Result Viewer and Plot Result are used to visualize and analyze the data in this stage.

Key Assumptions in FEA

The solution phase outputs numerical data in the form of nodal values for field variables and their derivatives. For example, structural analysis results include nodal displacements and element stresses. The Post-Processor processes this information, converting it into graphical formats for verification and in-depth analysis. This visual representation gives detailed insights, with automated output generation based on user specifications. Result Viewer and Plot Result tools assist in this context.

Assumptions Related to Geometry

- Displacement values are assumed to be small, supporting the use of a linear analysis approach.
- Geometric simplifications outside the primary area of interest are considered acceptable as they do not affect results significantly.
- Only fillets within the critical area are considered in the solution.
- Local behavior at corners, joints, and intersections is prioritized, with no need for specialized modeling.
- External ornamental features that do not influences stiffness or performance are excluded from the model.
- Variations in mass due to excluded features are deemed negligible.
- Assumptions Related to Material Properties
- Material properties are assumed to remain within the line arrange, excluding on linear behavior.
- The possibility of component failure is acknowledged if stresses surpass the yield point or if displacements are excessive.
- Material properties are assumed to be unaffected by load ingrates.
- The component is considered free from surface flaws that could cause stress concentrations.

IV. OBJECTIVES

The objective of this dissertation is to conduct experiments on a composite component and validate the Results using the Finite Element Analysis (FEA) method.

- To determine the deflection in a composite-reinforced compressor mounting bracket under tensile load through experimental testing and validate these findings against FEA results, assessing the suitability of the new material for use.
- To measure the stress in a composite-reinforced compressor mounting bracket when subjected to tensile load through experimental testing and compare these results with FEA outcomes to verify the feasibility of utilizing this new material.

V. EXPERIMENTAL RESULTS AND DISCUSSIONS

The Finite Element Method is a numerical approach used to solve problems in continuum mechanics, providing a level of accuracy that meets engineering standards.

5.1 Post-processing of Structural Analysis

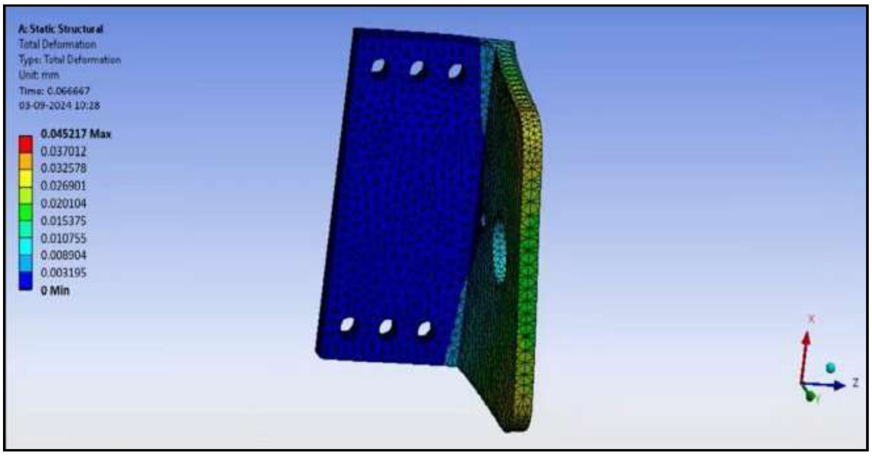

Figure 5.1: Total deformation contour plot for Compressor mounting bracket the maximum deformation shown by the Compressor mounting bracket design is 0.045217mm.

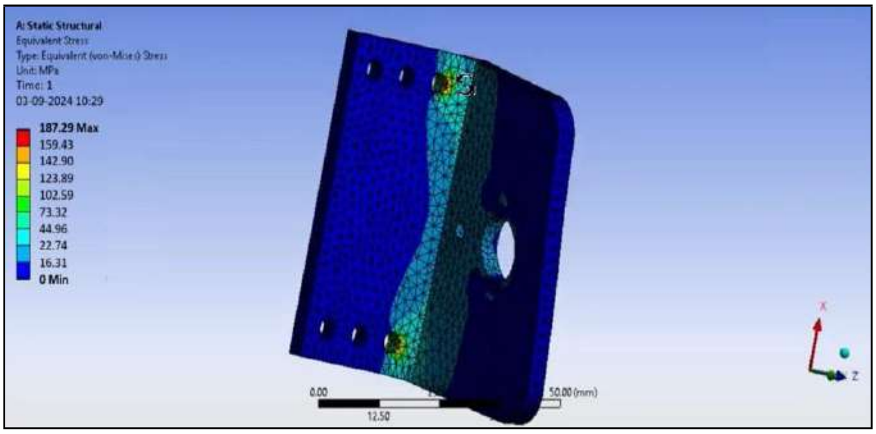

Figure 5.2: Von-Mises stress contour plot for Compressor mounting bracket the maximum equivalent stress observed in the Compressor mounting bracketis187.29MPa.

Testing of Compressor Mounting Bracket on UTM

Experimental testing is conducted on the compressor mounting bracket to examine the impact of composite material on its load-bearing capacity. The setup for this test, illustrated in Figure 6.1, utilizes a Universal Testing Machine with a capacity of 10tones.

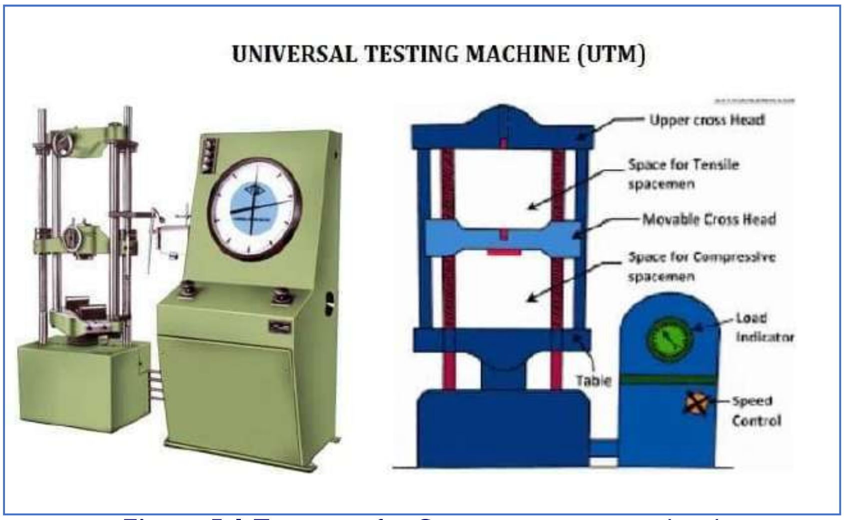

Figure 5.1 Test setup for Compressor mounting bracket

Table 5.1: Loading / deformation experimentation of Titanium Alloy (Ti-6Al2Sn-4Zr-6Mo) specimen sample at UTM

S.No	Load (N)	Deformation (mm)
1	1000	0.011
2	2000	0.022
3	3000	0.028
4	4000	0.035
5	5000	0.043

VI. CONCLUSION

- The simulation results were analyzed, revealing the following for the Titanium Alloy (Ti-6Al- 2Sn-4Zr-6Mo) specimen:
- Von Misses stress: 187 MPa Deformation: 0.045 mm both values are within the material's allowable Stress limits.
- The design is considered safe under tensile loading conditions.
- Strength was verified through both finite element analysis (FEA) and experimental testing.
- Stress values aligned with the design criteria for the Titanium Alloy (Ti-6Al-2Sn-4Zr-6Mo) specimen.
- Deformation values also met the design requirements for the composite material made from Titanium Alloy.
- The alloy is resistant to corrosion, eliminating: Friction noise the need for greasing or corrosion-resistant coatings.
- Deflection results from ANSYS FEM analysis were consistent with experimental findings.
- A comparison between experimental and FEA results for loading, deformation, and stress showed only a 1% to 1.60% discrepancy, indicating strong agreement.
- Compressor mounting bracket made of the Titanium Alloy (Ti-6Al-2Sn-4Zr-6Mo) composite material show eda discrepancy of only1% to 1.60%, indicating strong agreement.

Table 6.1. Experimental Validation FEA & Experimental Summary with % Error

Test	Experimental Results	FEA Results	% Error
Deformation (mm)	0.043	0.045217	1%
Stress (Mpa)	184	187.29	1.60%

REFERENCES

1. K.Anusha Reddy, “ Relative Comparison of Geometrical Shapes for Cut outs”, International Journal of Innovations in Engineering and Technology (IJIET), Volume5,Issue3,June2017,pp189-194.
2. Parveen K. Saini, Tarun Agarwal, “Stress Concentration around Countersunk Hole in Isotropic Plate under Transverse Loading”, World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, Vol.8, No:12, 2017, pp 1967-1973.
3. Hardik Acharya,Stress reduction using semiellipticals lots in axially loaded plate having circular hole, International Journal of Mechanical Engineering and Technology (IJMET), Volume5, Issue9,September(2017),pp.374-378.
4. Dharmendra S Sharma,“ Stress Concentration around Circular/Elliptical/Triangular Cut outs in Infinite Composite Plate”, Proceedings of the World Congresson Engineering, 2017,Vol.III,London,U.K.pp-303-312.
5. Mohammed Diany,“ Effects of the Position and the Inclination of the Hole in Thin Plate on the Stress Concentration Factor”, International Journal of Engineering Research & Technology (IJERT),Vol. 2 ,Issue 12,December– 2018.pp 8-12.
6. Lotfi Toubal,Moussa Karama,Bernard Lorrain,“Stress concentration in a circular hole in composite plate”, science direct, Composite Structures,68(2018),pp31–36.

7. C.K.Cheung, B.M.Liaw and F.Delale,B.B.Raju,“ Composite strips with a circular stress concentration under tension”, International conference,2018,U.S.Armyresearchcentre,Mishigun,pp193-199.
8. J.Rezaeepazh and, M.Jafari, “ Stress Analysis of Composite Plates with Non-circular Cut out”, Key Engineering Materials Vol,385(2018),pp365-368.
9. Moon Banerjee, “ stress concenstration in isotropic & orthotropic composite plates with center circular hole subjected to transverse static load”, International Journal of Mechanical Engineering and Technology (IJMET),Volume5,Issue 9,September(2019), pp.374-378.
10. Nitin Kumar Jain,“Analysis of Stress Concentration and Deflection in Isotropic and Orthotropic RectangularPlates with Central Circular Hole under Transverse Static Loading”,World Academy of Science, EngineeringandTechnology,Vol:32019,pp.12-22.
11. Patel Dharmin, Panchal Khushbu, Jadav Chetan, “ A Review on Stress Analysis of an Infinite Plate with Cut-outs”, International Journal of Scientific and Research Publications, Volume 2, Issue 11, November 2020 ,pp2250.
12. W.J.Padgett “A Multiplicative Damage Model for strength of Fibrous composite Materials ”IEFE transactions on reliability, VOL.47,No.1,2020 March.
13. Daniel David Samborsky, Montana state university, Bozeman.Montana.On“ Fatigue of e-glassfiber reinforced composite materials and sub structures”2020.
14. Kin-takLau,Li-minZhou,Xiao-mingTao“Control of natural frequencies of a clamped–clamped composite beam with embedded shape memory alloy wires” Composite Structures 58(2020)39–47 Elsevier.
15. Akirakuraishi,StephenW. Tsai,and Julie wang“Material Characterization of Glass,Carbon,andHybrid- Fiber SCRIMP Panels”Contractor 4 Report for Sandia national Laboratories,2020.
16. J.M.Corum and R.L.Battiste“ Eye Design Analysis of Single Leaf Spring In Automotive Vehicles Using CAE Tools”International Journal of Applied Engineering andTechnology Vol. 1,2020.
17. Liang M.Jones“Mechanics of Composite Materials ”Taylor and Francis ltd. second edition 2021.
18. Banuki,“ Mechanics of Composite Materials”.CRC press NewYork 2021.
19. Damodar,“ Metal Matrix Composite” Springer publication, NewYork,first edition2021.
20. Tita,“ Investigation of Fracture in Transparent Glass Fiber Reinforced Polymer Composites Using Photo elasticity ”ASME Journal of Engineering Materials and Technology January2021,Vol.126