International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 7 Issue 3, May-June 2023 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

A Review of Design and Material Optimization for Bearing Housing using Finite Element Method

Ashish Nain, Dr. Dinesh Kumar, Ganga Singh

Mechanical Department, Jan Nayak Ch. Devi Lal Memorial College of Engineering, Sirsa, Haryana, India

ABSTRACT

This research paper provides a review of the optimization of the mass of a bearing housing using the finite element method (FEM). In the competitive world of automobiles, having a lightweight yet strong component is crucial for fuel efficiency and cost savings. This is especially important when it comes to designing components that meet safety requirements. Failure in manufacturing, construction, servicing, raw materials, or operating conditions can lead to problems. One common issue is the continuous fatigue failure of the center bearing bracket, which can eventually cause the propeller shaft to fail and lead to transmission failure. To address this problem, computer-aided CAD design software is used to improve the bracket material without compromising its performance. Through the use of CATIA V5 software and finite element method (FEM) analysis with ANSYS software, the CAD model of the bearing bracket is thoroughly examined. The objective of this research is to find a technique that reduces the weight of the bearing bracket material using software while ensuring it remains strong and functions properly. By employing these tools and methods, we can achieve a lightweight yet reliable solution for automotive applications. Overall, this study focuses on using software to design a lighter bearing bracket without compromising its strength and functionality, aiming to improve fuel efficiency and reduce costs in the automotive industry.

How to cite this paper*:* Ashish Nain | Dr. Dinesh Kumar | Ganga Singh "A Review of Design and Material Optimization for Bearing Housing using Finite Element Method" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-7 | Issue-3, June 2023, pp.1198-1201, URL: www.ijtsrd.com/papers/ijtsrd58590.pdf

IJTSRD58590

KEYWORDS: *Bearing housing, CAD software, FEA analysis, ANSYS software*

INTRODUCTION

The Bracket: A Supporting Element for Overhanging Objects. A bearing housing is also known as bearing bracket. A bracket is an important component used to support overhanging objects. It comes in different sizes and shapes depending on its application, particularly in auto brackets. One crucial role of a rotating assembly is the transmission of torque in a propeller shaft. This torque is passed from the transmission gearbox to the axle differential gearbox through the propeller shaft. The propeller shaft constantly changes the relative angles between its axis and the transmission. It provides flexibility in length during torque transmission. The universal joint allows the shaft to work at different angles, and sliding joints enable the axle to expand and contract. Carrier or center loads support long propeller shafts to ensure smooth power supply to the axle. The center bearing contains rubber and bearing housings to support the ball bearing. The center bracket is mounted on the center of the chassis using nuts and bolts, positioning the entire mount [1].

Center Bearing Bracket: Supporting the Propeller Shaft Ball Bearing. The center bearing bracket is fixed to the chassis to support the ball bearing of the propeller shaft. To withstand the stressful working conditions, a rubber bed is placed between the ball bearing and the center bracket. This rubber bed helps reduce the shock loads on the center brackets. The highly elastic rubber bed is used to absorb sudden loads, and due to its elastic properties, the bracket returns to its original state. However, the rubber bed may not completely cushion the severity of abused loads, resulting in over stresses on the center joint bracket [7].

Finite Element Method (FEM): Solving Engineering Problems. The Finite Element Method (FEM) is a mathematical procedure used to solve engineering and technical problems. It helps address issues related to pressure fields, heat transfer, fluid flow, and mass exchange in various situations. This method is suitable for solving physical problems involving complex geometries, loadings, and material properties that cannot be solved using conventional methods. In FEM, the space where the analysis is conducted is divided into smaller bodies or units called Finite Elements.

Properties of each type of Finite Element are obtained, combined, and solved as a whole to find a solution. Problems are categorized into structural and non-structural issues based on their application. In Finite Element Analysis (FEA) or other numerical analyses, the development of structures relies solely on available calculations. For complex structures, simplifying assumptions are necessary to perform calculations, which can lead to conservative and valid designs. However, there may be uncertainties regarding whether the structure will be suitable for all design loads. In structural problems, displacements at each nodal point are obtained, and using these displacement solutions, stresses in each element are determined.

In non-structural problems, properties such as temperature or fluid behavior at each nodal point are obtained. Heat transfer and fluid flow components can be evaluated based on these nodal values. However, performing large calculations using FEM requires high-speed computational facilities with significant memory. The terms "Finite Element Method" and "Finite Element Analysis" are often used interchangeably, with FEA being more popular in industries and FEM being preferred at universities [9].

Literature Review: The world of Finite Element Analysis (FEA) is filled with a wealth of information and research. In this literature review, we will delve into significant advancements and developments in FEA implementation.

Dong, Yanhao (2022): The tension machine frame was created using Solidworks software and improved using ANSYS Workbench's optimization feature. The improved design scheme resulted in lower stress and weight reduction of 7.11%, satisfying the strength requirements. This will reduce design costs and bring significant economic benefits. [12]

Nangare, Vikram Ashok (2022): The current conveyor system had issues with sagging in the roller support. To address this, the roller, a crucial component of the system, was optimized. Geometric modeling, finite element analysis, and design optimization were conducted. ANSYS 14.5 and CATIA V5R20 were used for the analysis, and new rollers were manufactured and tested using UTM to validate the results. [13]

Vijayvergiya, Anurag (2021): The connecting rod of an internal combustion engine plays a crucial role in converting motion from the piston to the crankshaft. The study focused on analyzing the effects of force on different connecting rod shapes. Various models of connecting rods with different cross-sections (I section, H section, and rectangular section) were created. Static structural loads and buckling loads were analyzed using the finite element method and ANSYS software. [14]

Reddy et al. (2020) focused on the engine flywheel. They examined various speeds and performed analyses to achieve optimal results. A 2D drawing was created, and a parametric model of the flywheel was designed using CATIA, a 3D modeling software. The analysis software Ansys was used to verify the power of the flywheel. Additionally, the researchers compared the results obtained from analyzing the flywheel using two different materials: Cast Iron and Aluminum Alloy. [10]

Nitin et al. (2019) conducted a study on the Finite Element Analysis of an engine mount bracket. The objective was to evaluate its maximum deformation and equivalent stress. The engine bracket plays a crucial role in absorbing excessive vibrations from go-kart engines. However, it may fail under various forces. The researchers considered the load of the engine while designing the fixed bracket. Vibration and loading on the engine brackets have always been a concern, as they can lead to structural failure under high and excessive vibrations and pressures. This study highlights the need for in-depth research to understand the structural features and dynamic behavior of engine brackets, particularly in go-karts. The focus lies on design optimization and FEA analysis to determine the structural properties and material changes necessary for weight optimization. [11]

Adkine et al. (2016) conducted a literature review on the analysis of engine support brackets. Their work involved a thorough examination of existing studies on these brackets. They explored the design and analysis of engine support structures and investigated whether the current natural frequency of the engine brackets could be reduced. The study concluded that the Engine Support Frame plays a significant role in reducing noise, vibration, and hardness (NVH) in car engine systems, thereby increasing comfort. This reduction in NVH features is achieved through shape and mass optimization. [3]

In 2015, Kala et al. worked on the design and improvement of the V6 Engine Mount Bracket. Their project introduced FEA into engine bracket analysis. They used CAD software to model the engine brackets and performed Finite Element (FE) analysis on a typical bus engine. The researchers concluded that the V6 Engine Mounting Bracket made of Aluminum alloy had a natural frequency of 1181.5 Hz. The use of Computer-Aided Engineering (CAE) tools simplifies the design cycle, allowing for early problem detection and reducing the need for physical prototypes. This ultimately saves time and cost, thanks to simulation techniques. [8]

In conclusion, these literature reviews shed light on the advancements and applications of Finite Element Analysis in various aspects of engine design and optimization. Through the use of advanced software and simulation tools, engineers are able to achieve greater efficiency, cost savings, and improved performance in their designs.

Optimization Methodology

The optimization methodology in Finite Element Analysis (FEA) consists of three main steps. These steps are explained below:

1. Pre-Processing: In the pre-processing stage, the output data from previous steps is processed to be used as input for the next step. ANSYS 12, a pre-processor software, is used for this purpose. The input data is processed to generate data files automatically with the help of users. These data files will be utilized in the subsequent part of the analysis.
2. Analysis/Solution: The solution arrangement stage is fully automated. The FEA software creates the component structures, calculates nodal values and derivatives, and stores the output data in files. These files are then used by the next stage, known as the post-processor, to evaluate and analyze the results through graphical displays and tabulated listings.
3. Post Processing: The output from the solution stage is in numerical form and consists of nodal measurements of the field variable and its derivatives. For example, in structural analysis, the output includes nodal displacement and stress in the elements. The post-processor processes the output data and presents them in graphical form for result verification or analysis. The graphical output provides detailed information about the desired result data. The post-processing stage is automated and generates the graphical output in the format specified by the user. Result viewer and plot result tools are used for post-processing in this analysis.

Key Assumptions in FEA

There are several assumptions that affect the quality of the solution and must be considered in Finite Element Analysis. These assumptions are not universal, but they cover a wide range of applicable scenarios in the problem. It is important to note that not all the following assumptions apply to every situation, and only those assumptions relevant to the analysis are considered.

1. Assumptions Related to Geometry:

- Displacement values are small, allowing for a valid linear solution.
- Stress behavior outside the region of interest is not significant, so simplifications in those areas do not affect the results.
- Only internal fillets in the region of interest are included in the analysis.
- Local behavior at corners, joints, and intersections of geometries is of primary interest, so no special modeling of these areas is required.
- External decorative features are assumed to be insignificant for the stiffness and performance of the part and are excluded from the model.
- The variation in mass due to suppressed features is negligible.

2. Assumptions Related to Material Properties:

- Material properties remain in the linear region, and nonlinear behavior of the material is not considered. Exceeding the yield point or excessive displacement can lead to component failure.
- Material properties are not affected by the loading rate. The analysis assumes room temperature, unless specified otherwise.
- The effects of relative humidity or water absorption on the material are ignored.
- No compensation is made for the effect of chemicals, acids, wear, or other factors that may impact long-term structural integrity.

3. Assumptions Related to Boundary Conditions:

- Displacements are small, ensuring that the magnitude, direction, and distribution of the load remain constant throughout the deformation process.
- Frictional losses in the system are considered negligible.
- All connecting components are assumed to be rigid. The analyzed structure is considered a separate part from the rest of the system, so any reactions or inputs from adjacent features are disregarded.

The optimization methodology in FEA involves pre-processing, analysis/solution, and post-processing stages. Various assumptions related to geometry, material properties, and boundary conditions are made to simplify the analysis. These assumptions help in obtaining reliable and efficient results, but their applicability may vary depending on the specific analysis being conducted.

Results: Based on this research paper, it has been found that optimizing the design parameters and materials of the bearing bracket can significantly improve its performance. We can also conduct vibration analysis to identify any failure caused by vibrations in the bearing bracket. Additionally, dynamic and fatigue analysis can help determine the bracket's fatigue life. There are other mathematical methods available that can be used to optimize the bearing bracket.

Conclusion: This research paper focuses on the optimization of the mass of a bearing bracket using the Finite Element Method (FEM). The lightweight design of components is crucial for improving fuel efficiency and reducing costs in the automotive industry. The study utilizes computer-aided design (CAD) software and FEM analysis through ANSYS software to analyze the bearing bracket model. The research highlights the importance of reducing the mass of the bearing bracket without compromising its strength and functionality. The use of FEM allows for the evaluation of different design parameters and materials to achieve optimal results. The paper also discusses the application of FEA in analyzing the structural behavior, dynamic response, and fatigue life of the bearing bracket. Overall, the findings suggest that the optimization of the bearing bracket design parameters and material selection can lead to significant improvements in the performance and durability of the component. Further studies can explore additional mathematical methods and perform vibration and fatigue analysis to enhance the understanding of the bearing bracket's behavior and optimize its design further.

References:

[1] Amara Raja Goud, T.Nina "Design and Optimization of a Center Bearing Bracket Mount of a Propeller Shaft in BS-II Buses" IJMETMR, ISSN: 2348- 4845, Volume 4, Issue 2, feb.2017.

[2] Prasad Babu, Y. Vijaya Kumar, Dr. C. Udaya Kiran "Topology Optimization in Design of Engine Mounting Bracket" ICEME, ISBN: 978-93-82163-09-1, Volume 1, Feb.2014.

[3] A. S. Adline, Prof. G. P. Overikar, S. S. Surwase "Static And Modal Analysis Of Engine Supporting Bracket - A Literature Review" IJMTER, ISSN: 2349-9745, Volume 3, Issue 11, Nov.2016.

[4] David Roylance, "Finite Element Analysis".

[5] Jasvir Singh Dhillon, Priyanka Rao, V. P. Sawant "Design of Engine Mount Bracket for a FSAE Car Using Finite Element Analysis" IJERA, ISSN: 2248- 9622, Volume 4, Issue 9, Sep.2014.

[6] Koushik S. "Static and Vibration Analysis of Engine Mounting Bracket of TMX 20-2 using OptiStruct" Altair Technology Conference,2013.

[7] M. Jeevan Prasad, G. Guru Mahesh, D. Krishna Mohan Raju, "Failure Analysis of a Center Bearing Bracket Mount of a Propeller Shaft in BS-II Buses", IJCET, ISSN: 2277- 4106, Volume 5, Issue 2, April2015.

[8] P. Lakshmi Kala, V. Ratna Kiran "Mo deling and Analysis of V6 Engine Mount Bracket" IJIRSET, ISSN: 2319-8753, Volume 4, Issue 7, July2015.

[9] Robert D. Cook, "Basics of Finite Element Analysis".

[10] Reddy, B.Vishnu Vardhan, et al. "Design and optimization of petrol engine flywheel for variable speeds." IJMTARC, ISSN: 2320-1363, volume 8, issue 30, Apr- Jun,2020.

[11] Nitin, Jain Ankit, et al. "Design and Analysis of Engine Mount Bracket for Go- Kart." Proceedings of International Conference on Sustainable Computing in Science, Technology and Management (SUSCOM), Amity University Rajasthan, Jaipur-India.2019.

[12] Dong, Yanhao. "Optimization Design of Tension Machine Frame Based on Solidworks and ANSYS Workbench." Journal of Engineering Research and Reports 23.8 (2022): 11-17.

[13] Nangare, Vikram Ashok, and P. R. Sonawane. "Design, Analysis and Weight Optimization of Roller Conveyor System by using Glass Fiber Composite Material." Ijraset, Issue VII, Volume 10, 2321-9653. 2022.

[14] Vijayvergiya, Anurag, Emarti Kumari, and Shiv Lal. "Design and Shape Optimization of Connecting Rod End Bearing through ANSYS." IRJASH, Volume 3, Issue 11, Pages 235-242. *2021.*