International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 02 | Feb 2021 www.irjet.net p-ISSN: 2395-0072

A Review on Optimization of Mass of Bearing Bracket through Finite Element Method

JITENDER SINGH[1], SUMIT SINGH[2], SUMIT SINGLA[3]

[1]*dhaliwaljitender@gmail.com*

[2] *sumitsingh199@gmail.com Assistant Professor*

Department of Mechanical Engineering

Manav Institute of Technology and Management, Jevra Hisar,

Affiliated to Guru Jambheshwar University of Science and Technology, Hisar, Haryana, INDIA

[3] *sumitsingla598@gmail.com Researcher, Post Graduated in Mechanical Engineering*

Charkhi Dadri Haryana, INDIA

Abstract - A lightweight component with the requisite strength plays a vital role in the fuel efficiency and cost of a vehicle in the current developments in the competitive automotive world. This contributes to the design of a lightweight component for the safety requirements needed. Manufacturing, construction, servicing, raw materials, and operating conditions are some common causes of failure. Continuous fatigue failure of the center bearing bracket has been a problem that may lead to operational failure of the propeller shaft that eventually results in transmission failure. The computer-aided CAD design software is used to refine the bracket material and provide a satisfactory solution without changing output. Using CATIA V5 software and finite element method FEM analysis through ANSYS software, this work is analyzed by the CAD bearing bracket model. The aim of this paper is to describe the technique to reduce the mass of the material's bearing bracket using software without sacrificing its strength and functionality.

Key Words FEA, Bearing Bracket, Optimization

1. INTRODUCTION

The Bracket is a supporting element for overhanging object. Different sizes and shapes depending on the application in auto brackets. The rotating assembly of a propeller shaft plays a crucial role in the torque transmission system. The torque is passed through the propeller shaft from the gear box of transmission to the axle differential gearbox. The relative angles between the axis and the transmission change constantly via the propeller shaft. The propeller shaft provides length flexibility during torque transmission. The universal joint allows the shaft to work in different angles and the sliding joints that enable the axle to expand and contract. For the supply of smooth power to the axle, long propeller shafts are supported by carrier or centre loads. The centre bearing contains a rubber and bearing housings bearing ball bearing. The centre bracket is mounted on the centre of the chassis with nut and bolts positioning the entire mount[1].

Centre Bearing Bracket The Centre Bearing Bracket is fixed to the chassis to support the ball bearing of propeller shaft. Due to stressful working conditions the rubber bed is provided between the ball bearing and the center bracket which reduce the shocking loads on the center brackets. To minimize the load on the ball bearing and bracket, the highly elastic rubber bed is used to absorbed sudden load and due to elastic material the bracket again comes to its original state. The severity of abused loads are not cushioned by the rubber bed completely there by the over stresses are occurring on the centre joint bracket[7].

Finite Element Method FEM is a mathematical procedure for tackling designing and technical objects science problems. The technique is to tackle the issues of pressure field zone, heat move, liquid stream, and mass exchange in ordinary circumstances. This strategy can have the option to take care of physical issues including muddled geometrics, loadings and material properties which can't be settled by explanatory technique. In this strategy, the space wherein the examination to be completed is partitioned into littler bodies or unit called as Finite Elements.

The properties of each sort of Finite Element is acquired & gathered together and illuminated as entire to get arrangement. In view of utilization, the issues are characterized into auxiliary and non-basic issues. In Finite Element Analysis (or other numerical examination), advancement of structures must be founded available counts as it were. For complex structures, the rearranging presumptions required to make any counts conceivable can prompt a preservationist & substantial plan. An impressive factor of obliviousness can stay concerning whether the structure will be satisfactory for all plan loads. In auxiliary issues, dislodging at each nodal point is gotten. Utilizing these relocation arrangements, anxiety in every component are resolved.

Also, the non-auxiliary issues, a temperature or liquid property at each nodal point are gotten. For properties like warmth motion, liquid stream parts can assess from these nodal values. While enormous calculations are made, this technique requires rapid computational office with huge memory. Both are collectively same term i.e. Limited Element Method and Finite Element Analysis. Still the term FEA is progressively well known in ventures while FEM is popular at colleges [9].

1. LITERATURE REVIEW

There is a lot of Finite Element Analysis literature available. The literature review presented here looks at substantial changes to FEA implementation.

Reddy et al. (2020) presented a literature of engine flywheel for different speeds and analyses to get the best results. 2D drawing is drafted. The parametric modelof the flywheel designed to use the 3D modeling software CATIA. Active power on the flywheel also counts. The power of the flywheel is verified by using flywheel power in the analysis software Ansys. Analysis was performed with two Cast Iron and Aluminum Alloy materials to compare the results. CATIA[10].

Nitin et al.(2019) this paper deals with Finite Element Analysis (FEA) of an engine mount bracket that is designed to look at its maximum deformation and equivalent stress analysis. Excessive vibration of the go-kart engine is taken over by the engine bracket however it may fail due to various forces. The load of the engine is also considered during the fixed bracket design. Vibration and loading on the engine brackets has always been a concern that can often lead to structural failure when vibrating vibrations & pressures are high & excessive. An important study that requires in-depth research to understand the structural features and its dynamic behaviour. The main focus of Design and FEA on the engine brackets especially the go-kart where the analysis of the structure and the dynamic analysis of the mode will be determined by changing the physical properties and materials of the weight optimization [11].

Adkine et al. (2016) presented a literature review on a consistent and central analysis of the supporting engine brackets. In this work they reviewed the work that had been done on the brackets supporting the engine. Ongoing work included rigorous analysis and design of the engine supporting structure and investigating whether the current natural frequency of engine brackets was reduced to that of the brackets and concluded that the Engine

Support Frame played a significant role in reducing noise, vibration & hardness (NVH features) of the car engine system to increase comfort. It's by optimizing the shape and mass[3].

Kala et al. (2015) designed and updated the V6 Engine Mount Bracket. The project introduced FEA in engine-brackets. Modeling of bracket engines using CAD software. The FE analysis of a typical bus input engine will be performed and considered for natural frequency. They concluded that the Engine Mounting- Bracket V6 engine made of Al alloy had a natural frequency of 1181.5 Hz. The use of CAE tools leads to a simpler look & thus helps to detect problems at the beginning of the design cycle, reducing the num of physical prototypes leading to greater savings of time and cost and ultimately including simulation techniques [8]

Babu et al. (2014) Topology designed for the construction of brake engine brackets. In the work they described the design of the installation bracket using the use of Altair Optistruct, compared to the original construction of the mounted brackets, using the three most popular materials to produce a growing bracket such as Al alloy, Al Si carbide and gray steel, weight in the final structure is reduced by about 50 % without compromising its power & performance. There was enough energy for improved fatigue and performance. Mounting bracket is designed using a 3D modelingtoolegCATIAiV5-R-19 Software. The Altair Hyper-mesh tool was then used for processing & Altair Radios software to solve the structure of the engine bracket and concluded that an antimatter spacecraft would allow a person to go where no one had ever been in space. It is also possible to make a trip to Jupiter and even beyond heliopause, the point at which the sun's rays end [2].

Dhillon et al. (2014) designed and performed the FSAE Motor Engine Mount Bracket Analysis of FSAE. In this paper they discussed modeling, Finite Element Analysis, Modal analysis and great performance in FSAE car cable brackets. As brackets are often continuous vibrations with various stresses, fatigue strength & durability designed to confirm engine safety. They concluded that the extension of the ribs helped to reduce the maximum deviation by 30.5% in the case of severe loading. The high von-mises stress increased from 45.02MPa to 63.99MPa, but it helped us achieve a more satisfactory safety rating of 3.3[5].

Koushik et al. (2013) designed and performed the consistent and vibratory analysis of the TMX 20-2 Engine Mounting Bracket using Opti Struct. The purpose of paper was to perform static analysis and vibration of the mounts designed for the 2 ton class Mini engine index engine. The 2-ton class excavator made by Tata-Hitachi is the smallest in India and has been widely used in earth drilling, dozing, trenching, rock breaking, digging etc. The job was to look at the levels of engine degradation and stress and weight gain. Vibration in the form of multiple axial acceleration was recorded using a data logger while the machine was operating. From the above results, it was found that the difference in stress between physical examination and the simulation method was almost 18%. From the effects of field failure and the effects of Hyper Works, it has been observed that failure occurs when the ribs end in brackets. To reduce the level of stress at this stage, more ribs were given at the top of the brackets so that the total stiffness was increased[6].

2. OPTIMIZATION METHODOLOGY

The Finite Element analysis optimization methodology is performed in three primary strides as which are as following below in figure 1.

Pre-Processing Pre-Processing is a process that processes out the output data that is used as the next step (solution). Here ANSYS 12 is used as a Pre-processor. The input facts will be Pre-Processed for the output facts and Pre-Processor will create the data files automatically by the

help of users. These records files will be used by the subsequent part.

Examination/ Solution Arrangement stage is totally programmed. The FEA programming creates the component frameworks, registers nodal qualities and subsidiaries, and stores the outcome information in documents. These documents are additionally utilized by the ensuing stage with respect to the reason (Post-Processor) to survey and examine the outcomes through the realistic showcase and forbidden postings.

Post Processing The yield from the arrangement stage is in the numerical structure and comprises of nodal estimations of the field variable and its subordinates. For instance, in auxiliary examination, the yield is nodal dislodging and worry in the components. The Post-Processor forms the outcome information and showcases them in graphical structure to check or dissect the outcome. The graphical yield gives the itemized data about the necessary outcome information. The Post-Processor stage is programmed and produces the graphical yield in the structure determined by the client. Result watcher and plot result are utilized for Post-Processing in this issue[4].

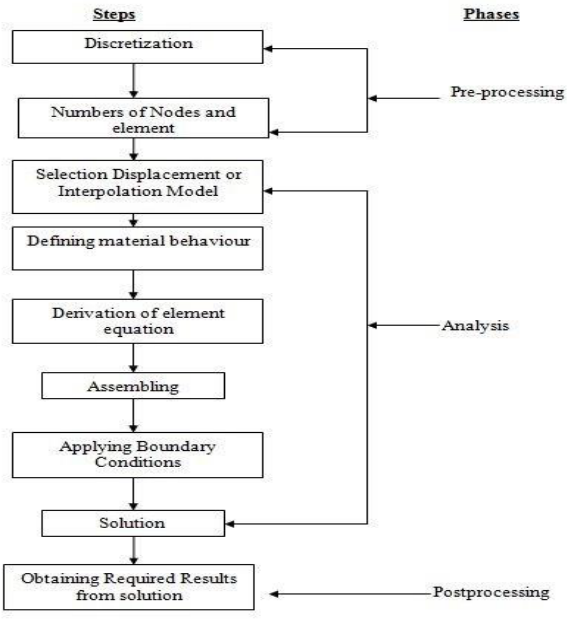

Figure 3.1 Procedure of FEM

Key Assumptions in FEA

There are following essential suspicions that influence the nature of the arrangement and must be considered for Finite Element Analysis. These suspicions are not farreaching, however spread a wide as sortment of circumstances pertinent of he issue. Also, in no way, shape or form, do all the accompanying suppositions apply to every one of the circumstances. In this way, it was guaranteed just those pre assumptions that apply to the investigation viable.

Assumptions Related to Geometry:

Displacement esteems will be little with the goal that a direct arrangement is substantial.

Stress conduct outside the region of intrigue isn't significant, so the geometric disentanglements in those zones won't influence the results.

Only inside filets in the region of intrigue will be remembered for the arrangement.

Local conduct at the corners, joints, and crossing point of geometries is of essential intrigue; in this manner no extraordinary displaying of these zones is required.

Decorative outer highlights will be accepted unimportant for the firmness and execution of the part and will be precluded from the model.

The variety in mass because of the stifled highlights is irrelevant.

Assumptions Related to Material Properties:

Material properties will stay in the direct locale and nonlinear conduct of the material property can't be acknowledged. For instance, it is comprehended that either the feelings of anxiety surpassing the yield point or inordinate dislodging will cause a segment disappointment.

Material properties are not influenced by the heap rate.

The part is free from surface flaws that can create pressure risers.

All re-enactments will accept room temperature, except

if generally indicated.

The impacts of relative mugginess or water ingestion on material utilized will be ignored.

No remuneration will be made on represent the impact of synthetic compounds, corrosives, wears or different elements that may affect the long haul basic respectability.

Assumptions Related to Boundary Conditions

Displacements will be little so the greatness, direction, and circulation of the heap stay steady all through the procedure of misshaping.

Frictional misfortune in the framework is viewed as irrelevant.

All interfacing segments will be expected unbending.

The bit of the structure being examined is accepted a different part from the remainder of the framework, in this way with the goal that any response or contribution from the contiguous highlights is dismissed[4].

3. RESULTS

On the basis of this research paper it is concluded that the design parameters of the bearing bracket with material optimization give sufficient improvement in the existing results.

Vibration analysis can also be performed to find out the failure condition due to vibration in the bearing bracket.

Dynamic and fatigue analysis can also be performed to find the fatigue life of the bearing bracket.

We can also use other mathematical methods to optimize the bearing bracket.

4. CONCLUSION

Vibration and fatigue of engine bracket can lead to structural failures when resulting frequencies & stresses exceeds permissible value. In some studies the features of the engine structure that supports the frame and its dynamic behaviour are also considered. Failure to get tired of the engine due to prolonged exposure shakes the whole body in the workplace may damage the car. According to basic concept of indirect vibration; if the natural frequency of the engine supporting frame is able to keep it well under the band of interest range where the design is safe and uncluttered with the suitability of the required application material.

5. ACKNOWLEDGEMENT

We (Jitender Singh and Sumit Singla) express my deep sense of gratitude to my esteemed guide Sumit Singh (Assistant Professor) Department of **Mechanical Engineering** Manav Institute of Technology and Management, Jevra Hisar affiliated to Guru Jambheshwar University of Science and Technology, Hisar, Haryana, INDIA for their erudite and prolific guidance and critical appraisal, constant supervision and valuable help during the study of this present research work.

6. 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] A. 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 "Modeling and Analysis of V6 Engine Mount Bracket" IJIRSET, ISSN: 2319-8753, Volume 4, Issue 7, July2015.

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

[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.