iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** I **Month of publication:** January 2018

DOI: http://doi.org/10.22214/ijraset.2018.1176

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor :6.887
Volume 6 Issue I, January 2018- Available at www.ijraset.com

Design and Development of Die Cutter Mounting Bracket

Amit Belaganvi[1], Rahul N. yerrawar[2]
[1] *Department of Mechanical Engineering, Savitribai Third University*

Abstract*: It is required that any machine component must be work properly during its service life time. But due to the loading conditions, there may be a chance of breakdown. It is required to design the proper Cutter mounting bracket. Sudden failure or break down may cause production idle time which further leads to loss of production rate and loss of money. In this work design and development of bracket is carried out.
Keywords: Modal Analysis, Mounting Bracket, Structural Analysis

I. INTRODUCTION

The important function of Cutter mounting bracket is to balance and hold the module at proper position with minimum deflection and maximum load carrying capacity. In this work design and weight optimization is done. The manufacturers have strong importance on the cost has the demand for the component, improve the material performance and to deliver these materials at low cost is the requirement. In this work 1020 Hot Rolled Steel is used for plate, and square tubing for stiffness which is easily available at customers end.

Analysis of the strength of bracket was made easier with the introduction of Computer Aided Engineering (CAE) software. The finite element method (FEM) is a very good tool available in CAE software.

II. ANALYSIS OF MOUNTING BRACKET

Modeling is done in inventor and analysis is carried out in Ansys. Cutter weight is 900 kg. And bracket should have deflection less than 0.5mm.

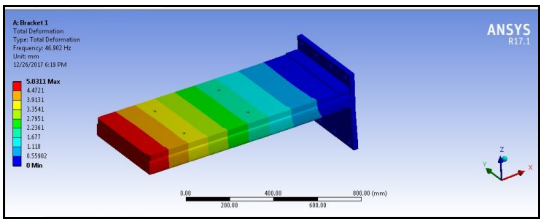


Fig. 1 First Natural Frequency

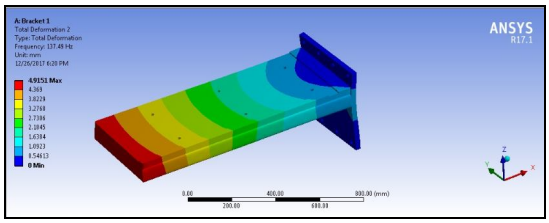


Fig. 2 Second Natural Frequency

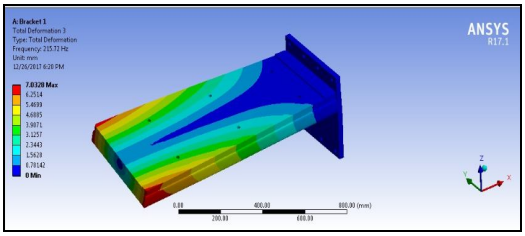


Fig. 3 Third Natural Frequency

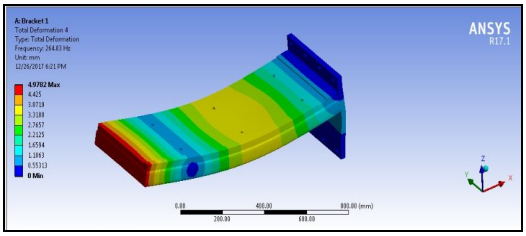


Fig. 3 Fourth Natural Frequency

Abobe figures shows modal analysis done in Ansys and output natural frequencies tabulated in below table 1.

Mode	Frequency Hz
1	46.902
2	137.49
3	215.72
4	264.83

TABLE INATURAL FREQUENCIES

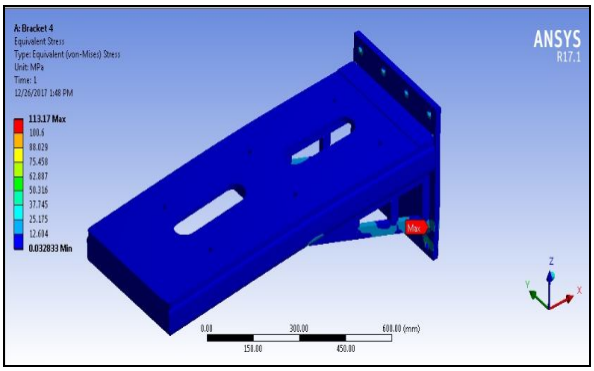


Fig. 4 Stress Analysis

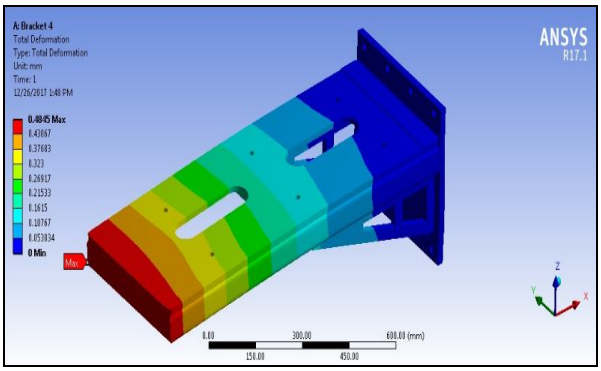


Fig. 5 Deflection Analysis

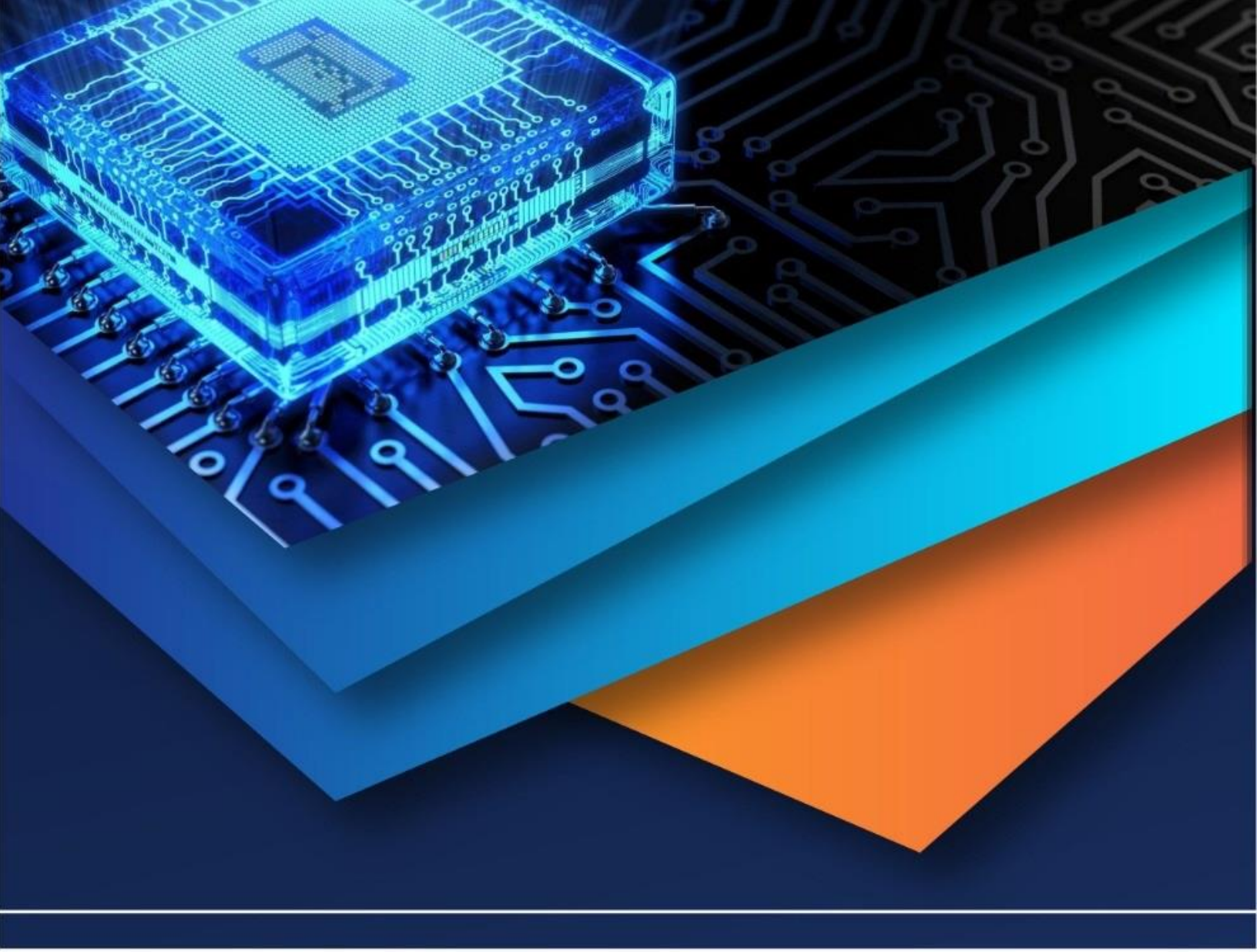
Above figure 4 and figure 5 shows stress and total deflection developed. For the loading of 900 kg stress developed is 113.17 Mpa and total deformation is 0.48 mm respectively.

III.CONCLUSIONS

An efficient design process, requires two parameters, the structure should fail at the design load and at the same time have minimum weight. Results it is concluded that designed bracket is safe which is having maximum stress of 113.17 Mpa and total deformation is 0.48 mm respectively.

REFERENCES

[1] Rahul N. Yerrawar, Vinod B. Tungikar And Shravan H. Gawande, Finite element analysis of dynamic damper for CV Joint, Scientific Research, doi:10.4236/epe.2012.44033, July 2012, PP. 241-247
[2] Ritesh Bhairappa, Dr.Santosh V Angadi and Prof. Anand S N3, Strength and stiffness analysis of an engine bracket, IJFTRIE, Volume 3, Issue 10, June-2016. pp 2637-2641
[3] Revanasidda Benur and Akshatha H T, Design evolution of a typical aircraft engine mount Bracket using FE based optimization technique, ISSN,Vol. 1, Issue. 3, June 2015. pp 7-11
[4] Sebastian C S, Shibin Baby and Anthony Tony, Design and optimization of engine mount bracket, IJAIEM, Volume 5, Issue 2, February 2016, pp 168-172.
[5] Gaurav Bhusari, Design and analysis of fixture bracket for engine assembly line, IJIRST, Volume 2, Issue 05, October 2015, pp 1-3
[6] Mr. Sagar N. Narute, Prof. Dr. Santosh G.Taji And Prof. Prabhuling G. Sarasambi, Design and Analysis of AC Mounting Bracket Using Composite Material, IJSART,Volume 3 Issue 9 –SEPTEMBER 2017, pp 21-29.
[7] Tushar P. Kamble And Rajratna A. Bhalerao, Optimization & Modal Analysis of Engine Mounting Bracket for Different Materials by Using Finite Element Analysis, IRJET, Volume 03 Issue 10 Oct-2016, pp 120-125
[8] P. Meghana , Y. Vijayakumar, Dr P. Ravinder Reddy, P.Seema Rani, Analysis of Cabin Mounting Bracket of Truck Using ANSYS, IJESI, Volume 5 Issue 11 November 2016 pp. 82-89
[9] Pooja N Sureja1, Ulkesh B Desai And Mitesh J Mungla,, Design, Optimization & Analysis of Antenna Mounting Bracket for QON System. IRJET,Volume: 04 Issue: 04, Apr -2017, pp 2397-



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY