iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** VI **Month of publication:** June 2018

DOI: http://doi.org/10.22214/ijraset.2018.6286

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 VI, June 2018- Available at www.ijraset.com

Static Structural Analyis of Engine Lifting Bracket

P. N. Godage[1], U.S. Chavan[2]
[1, 2]Department of Mechanical Engineering Vishwakarma Institute of technology, S.P. Pune University Pune, India

***Abstract**: A lifting bracket is a device for grabbing and lifting loads by means of a device such as crane or hoist. Engine block is required to be lifted from one place to another while they are being assembled and serviced in the field. The engine will be placed in to vehicle that will make it difficult for powertrain removal & installation using existing lifting bracket. So that there is a need for designing a proper lifting tool which will be able to conveniently lift the engine block and place it at the desired position without any accident and damage to the engine block. In the present study lifting tool assembly is designed and analyzed in such way that it may lift engine block from wide range of engine family with varying number of cylinders, dimensions and weight. The lifting tool assembly consists of plate lifting fixture, lifting ring, cap screws etc. Assembly details are mentioned in this report. A parametric model and assembly of Lifting tool is done in 3D modelling software CREO 2.0 and analysis is carried out in ANSYS Workbench 17.1.*

All the child components/parts of engine lifting tool assembly (expect rear and front bracket) are meeting the stress acceptance criteria in the analysis. Rear and Front bracket are not meeting the theoretical stress acceptance criteria however actual lab test report shows that the brackets have passed for higher load than considered for the analysis. Analysis is carried out to verify that tool design meets the ASME BTH-1 required safety margin.

Keywords: Lifting bracket; ANSYS; Static analysis; ASME BTH-1

I. INTRODUCTION

Engine is used in various applications including generators, marine, rail, automobile, industrial etc. In this paper we are going to discuss lifting of such type of engines by using specific lifting bracket. Bracket is used to lift engine. The main function of the bracket is to support the engine. As, there are many components in the engine which are heavy, there is need to design the bracket that can sustain the load properly and balance the engine. In the finite element analysis of engine mounting bracket by considering pretension effect and service load Sandeep Maski and Yadavalli Basavaraj[6] carried out the static and modal analysis. They selected the three different material for the analysis. The objective of their project was to select best material for the engine mounting bracket so that bracket can sustain high strength and vibrations. In the engineering optimization of engine mounting bracket Pramod Walunj[5] carried out the static analysis of the bracket for weight optimization. in this paper he used three different materials for bracket such as aluminum alloy, grey cast iron and magnesium alloy. Using the FEA software he carried out the analysis and validated it experimentally. For aluminum alloy material he found out that natural frequency and von-misses stresses are within the yield strength. For the aluminum bracket he reduced the thickness and check for the stresses. After weight optimization he again acquired the results within the yield strength. For this new aluminum alloy bracket he achieved the mass reduction as 0.43 kg as compared to previous. Present study focuses on the checking the stresses and deformation of the engine support bracket due to the bolt pretension and service load. For this project 12 liter engine is used having heavy duty truck application. For this application the analysis of the casting bracket is carried out considering the various load cases and results are verified from the material yield limit.

II. METHODOLOGY

Literature review: Technical papers, patent documents, etc. 2. Study and analysis of components of lifting tool & Bolt preload calculations. 3. Mechanical design of critical components to determine functional dimensions of the components to be used using various formulae and empirical relations. 4. 3-D modelling of lifting tool assembly using CAD software's Ex: PTC Creo 2.0, CATIA 5. Structural analysis of lifting tool assembly components using FEM software's Ex: ANSYS17.1, Abaqus. 6. Performance testing of lifting tool. 7. Comparison of stress by experimental & software basis.

III. MATHEMATICAL CALCULATIONS

Lifting load Calculations [3]

$$F = m*a$$
$$= 948.46(kg)*9.810(m/s^2)$$
$$F = 9304.5 \text{ N.}$$

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 VI, June 2018- Available at www.ijraset.com

For analysis we consider double load i.e. 2*9304.5 = 18609 N (According to ASME BTH-1 standard)
Free body diagram of Lifting tool

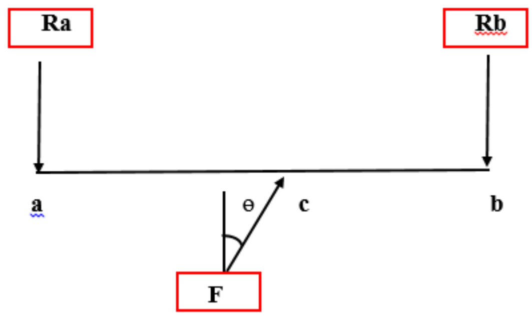


Fig. 1 Free Body Diagram

If we see the lifting tool, lifting load is applied through C.G. direction of engine in vertical direction. So that we get reaction forces at support & we get simply supported F.B.D.
Dimensions of above FBD given,
ac = 1.6620 inch
cb = 1.6620 inch
F = 18609 N
$ɵ = 3.1375^0$
Find below,
Ra, Rb, Ma, Mb, Mc
Ra = Reaction force at point a
Rb = Reaction force at point a
Ma = Moment at point a
Mb = Moment at point b
Mc = Moment at point c
By applying Equilibrium conditions,

a. Σ Fx = 0 --------------- 1)
b. Σ Fy = 0 --------------- 2)
c. Σ Ma = 0 ---------------- 3)

Using 3rd equation,
Σ Ma = 0,
+Fcos ɵ * 42.21 - Rb * 84.42 = 0
Rb = 9290.55 N
Using 2nd equation,
Σ Fy = 0,
+Ra +Rb - F*cos ɵ = 0
Ra = 9290.55 N
Ma = 0
Mb = 0
Mc = -Ra* (42.21)
Mc = -392154.11 Nmm

IV. MODEL FOR ANALYSIS

Modelling is done by using the creo 2.0 software.

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 VI, June 2018- Available at www.ijraset.com

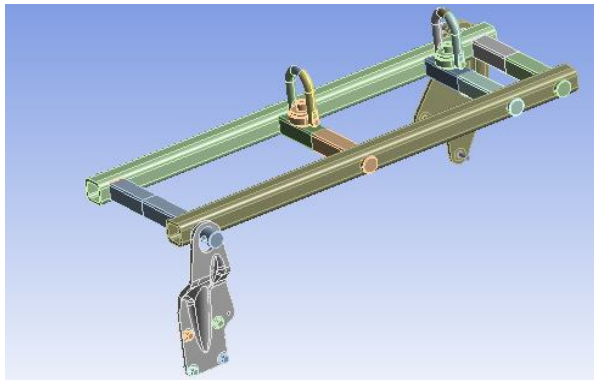


Fig. 2 bracket model

V. STATIC STRUCTURAL ANALYSIS

In the static analysis we can get the values of displacement, strain, stress and forces. For the static analysis the above bracket is used.

A. Geometry

Below image shows the C.G of engine, centre of eye bolt. As we know that force is applied in the direction of C.G. red dotted line showes the direction of force applied

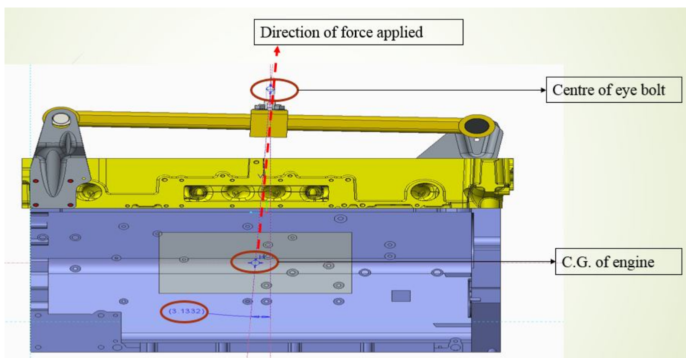


Fig. 3 Lifting load direction

VI. RESULTS & DISCUSSION

For all the load cases result is evaluated by using ANSYS18.2 software. Change in shape of body caused by the application of force (stress). Deformation is proportional to the stress applied within elastic limits of the material. It shows how tool get deform at load step 2 & 3.

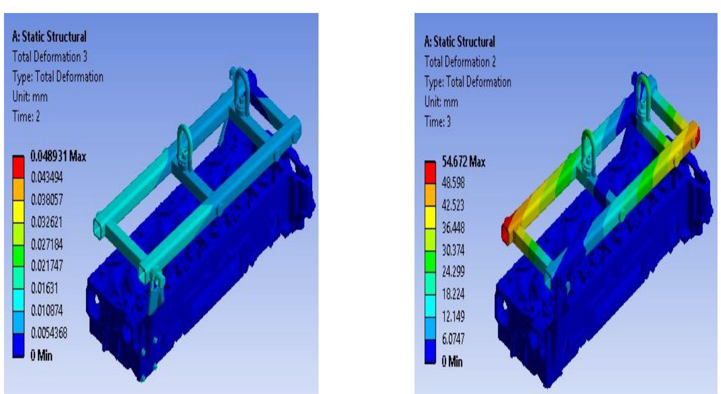


Fig. 4 Deformation plots

Front Bracket: - Von misses stress in lifting brackets are above the material yield strength of 345 MPa. So brackets does not meet acceptance criteria.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

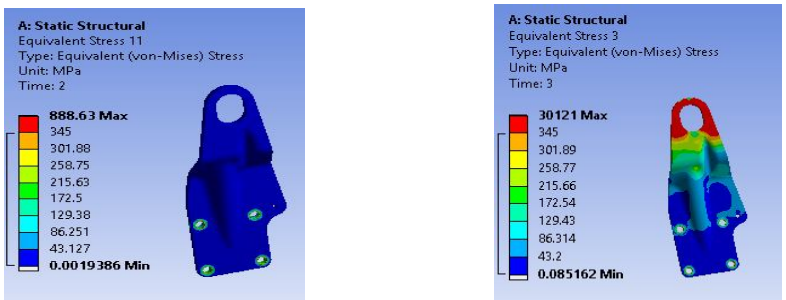


Fig. 5 Von misses stress plots of front bracket

Rear Bracket: - Von misses stress in lifting brackets are above the material yield strength of 345Mpa. So brackets does not meet acceptance criteria.

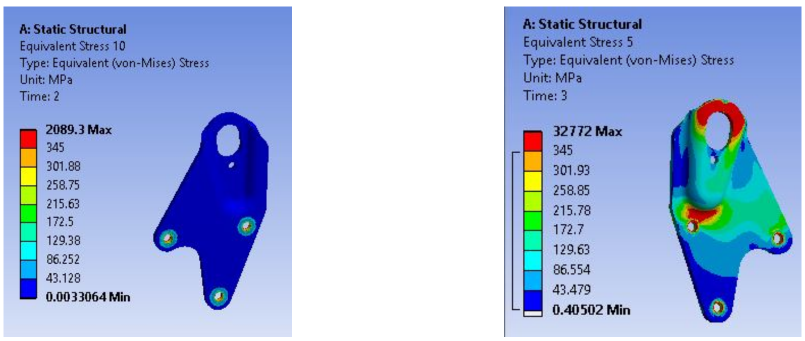


Fig. 6 Von misses stress plots of rear bracket

Plate Lifting Fixture:-Von misses stress in fixture is within the material yield strength of 655 MPa. High stress is at contact region which can be ignored.

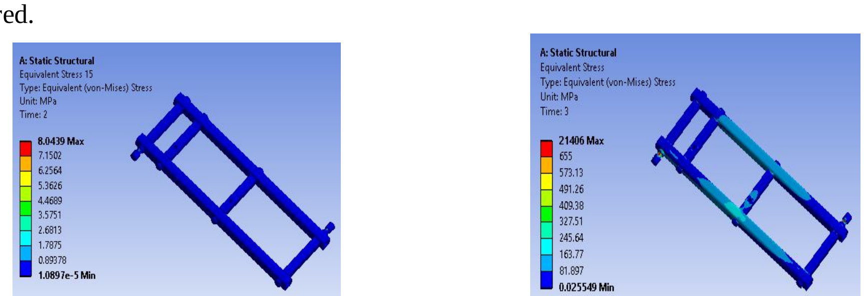


Fig. 7 Von misses stress plots of plate lifting fixture

Ring lift:- Von misses stress in ring, lifting is within the material yield strength of 805 MPa. High stress is localized which can be ignore.

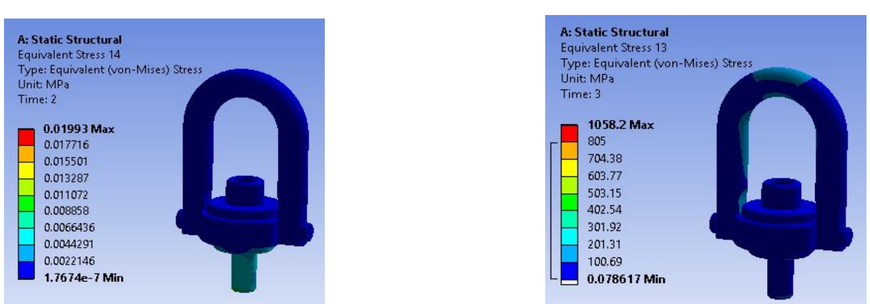


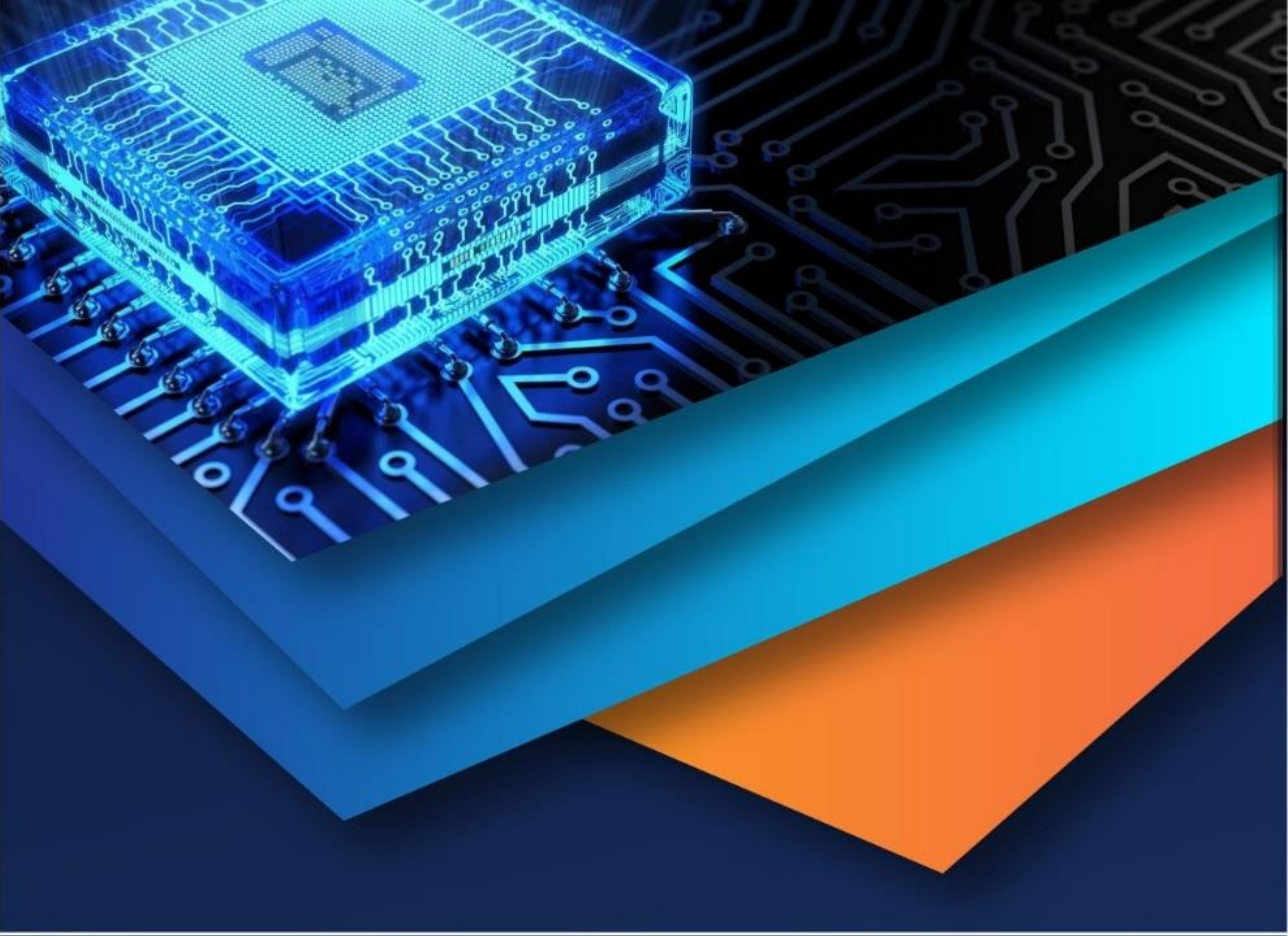
Fig. 8 Von misses stress plots of Ring Lift

VII. CONCLUSION

All other components/parts (expect rear and front bracket) of lifting tool are meeting the stress acceptance criteria in the analysis. Rear and Front bracket are not meeting the stress acceptance criteria however test report shows the brackets have passed for higher load than considered for the analysis.

REFERENCES

[1] Material database.
[2] Procedure for Performing FEA on Lifting Arrangements using ANSYS.
[3] FEP _ Lifting Bracket Load Calculations.
[4] Final Report for “Roush” Project, “Cummins 12L Turbo Diesel Engine for MOI/CG Evaluation”
[5] Walunj “ Static and vibrational analysis of engine mounting India- 2013.
[6] Sandeep Maski and Yadavalli Basavaraj “ Finite element analysis of engine mounting bracket by considering pretension effecr and service load” in International journal of research in engineering and technology vol. 04, Auguest 2015.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY