International Journal For Technological Research In Engineering
Volume 3, Issue 10, June-2016 *ISSN (Online): 2347 - 4718*

STRENGTH AND STIFFNESS ANALYSIS OF AN ENGINE BRACKET

Ritesh Bhairappa Dhamoji[1], Dr.Santosh V Angadi[2], Prof. Anand S N[3]
[1]PG Student, [2]Assistant Professor, [3]Associate Professor
School of Mechanical Engineering, REVA University, Bangalore.

Abstract: The objective of this project work is to study the strength and stiffness of the bracket utilized in the aircraft engine control systems. Improper stiffness design will lead to loss of pilot control or helicopter blade pitch angle variations and become catastrophic due to stall effects. The design process involves Finite element analysis to assess the strength and stiffness of the flight control systems and simulations to arrive at optimum stiffness and strength for safe flight controls for critical manoeuvres. This project work focusses on the utility of Finite element methods in design, analysis and optimization of a regular engine bracket using software's, Altair Hyper mesh and MSC Nastran.
Keywords: Engine bracket, natural frequency

I. INTRODUCTION

An engine mounting bracket is one of the most important part of a vehicle that reduces vibrations and harshness for the smooth ride of the vehicle. The important function of an engine mounting bracket is to balance the engine on the vehicle chassis for good balance control when the vehicle is in motion. In this work, an engine mount bracket of an aircraft is modelled using CATIA. In the current automobile technology, the need for light weight structural materials is growing as there is more focus on fuel intake and emission reduction. The magnitude of production volumes has historically positioned extreme requirements on the high-quality of the technique used in the manufacturing. 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. There are number of vibrations and noise equipment's that affect the body of a vehicle. Uneven loads and poorsuspension produce vibration and noise in a vehicle, which will result in the high frequency of an engine. The noise and vibration generated by the equipment's are transferred to the vehicle body.Stiffeners attached to the body perform a vital role for NVH performance development. To diminish and manage the vibrations, an engine mount bracket should be firm.

II. METHDOLOGY

In this work, Static structural and Model analysis were used to determine the characteristics of the engine mounting bracket and the Harmonic response analysis is done to check the response of structure in terms of acceleration.

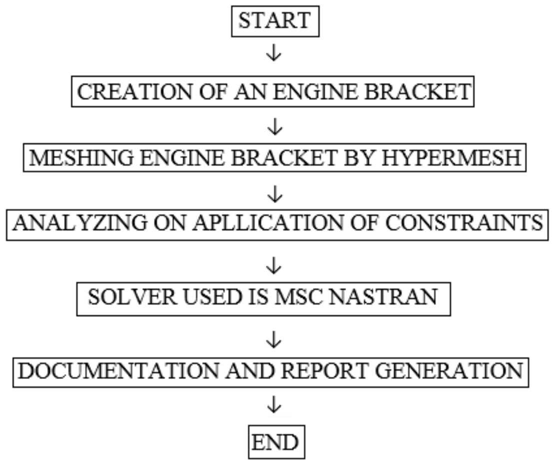


Fig.1 Methodology

A. Modal Analysis

Modal analysis determines the vibration traits of a shape factor in the form of natural frequencies. The natural frequencies are crucial inside the design of a shape for dynamic loading conditions. On this evaluation, most effective linear behaviour is valid. The evaluation of many engineering and automotive additives is performed with the help of finite element techniques.

B. Finite Element Analysis

The Finite Element Method is a numerical approximation method, in which the complex structure is divided into number of small parts that is pieces and those small elements are referred to as finite element. These small elements are connected to each other by means of small points called as nodes.

C. Hyper works

Hyper work is one of the most used software to solve finite element analysis which will provide solid meshing, surface meshing then the processors or solvers used in this work is MSC Nastran. Altair hyper works is having many equipment for the creation of models and to analyse for non-linear, thermal, linear, explicit dynamics and other types of finite element issues.

D. MSC Nastran

MSC Nastran is a multidisciplinary structural analysis utility used by engineers to perform dynamic, static, and thermal analysis across the linear and nonlinear domains.Engineers use MSC Nastran to make sure structural systems have the necessary strength, stiffness, and life to preclude failure.

NASTRAN is usually a solver for finite element analysis.

III. FINITE ELEMENT ANALYSIS OF AN ENGINE BRACKET

A. Model Description

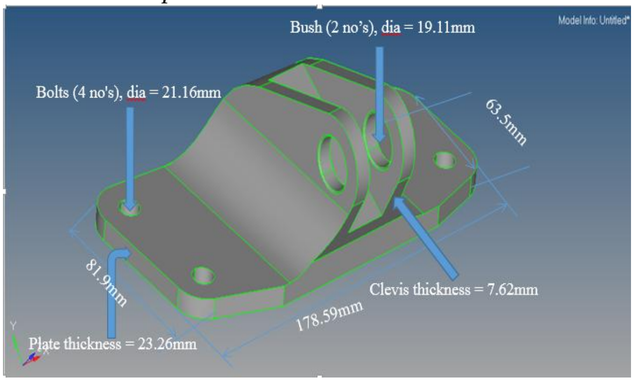


Fig. 3D model of engine bracket

B. Meshed Finite Element (FE) Model

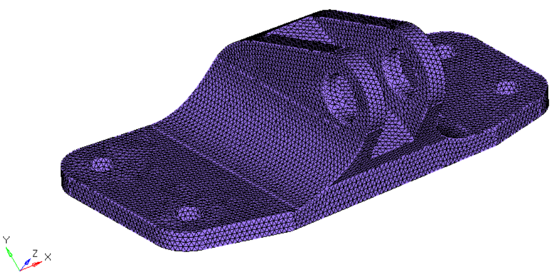


Fig. Meshed engine bracket

Model Summary for Tet:
Number of nodes=24003
Number of elements=106550
Number of RBE2 element=6
Average element size=2

C. Model Checks

The below-mentioned model checks are performed in order to ensure node to node connectivity. It is necessary that these model checks are eliminated and ensure that there is an equal distribution of a load on a model.

1. Free edge check.
2. T edge check.
3. Element quality check.
4. Material property check.
5. Property check.

1. Free edge check

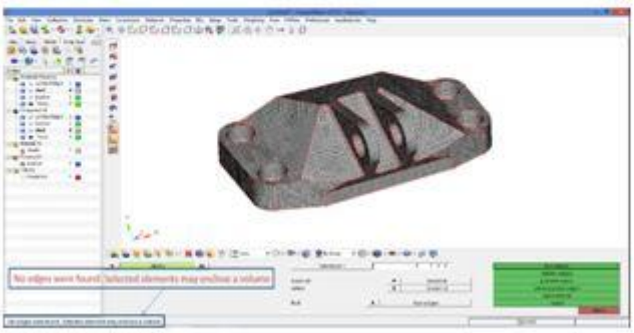


Fig. FE Model is verified for free edges as shown above

2. T edge check

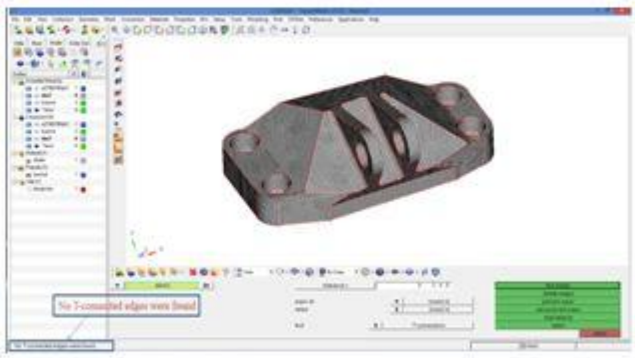


Fig.FE Model is verified for T edges

3. Element quality check

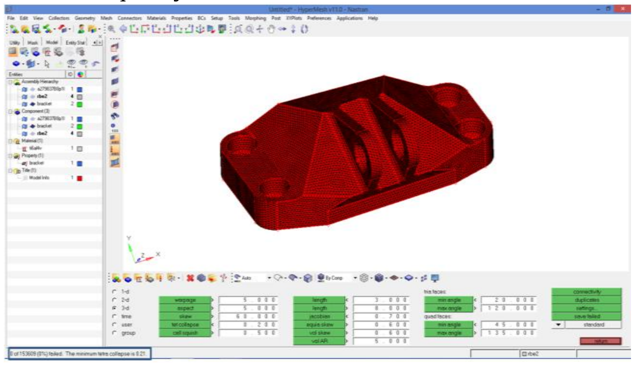


Fig.FE Model is verified for the Element quality checks. Minimum Tet collapse of 0.2 is maintained

4. Material property check

Fig. FE Model is verified for material properties

5. Property check

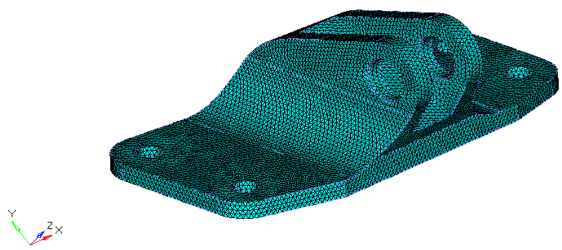


Fig.FE Model is verified for properties

IV. RESULT AND DISCUSSION

A. Tet4

Load case 1: Horizontal load

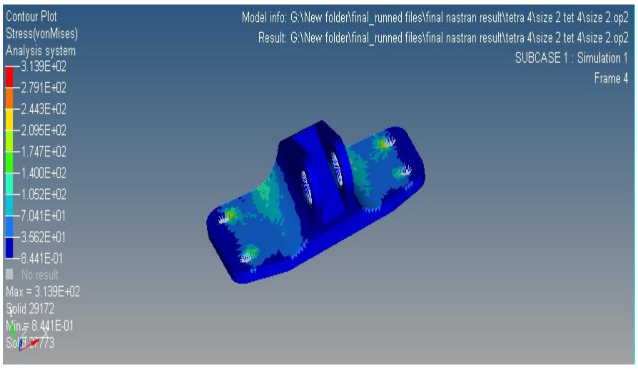


Fig. Stress Distribution

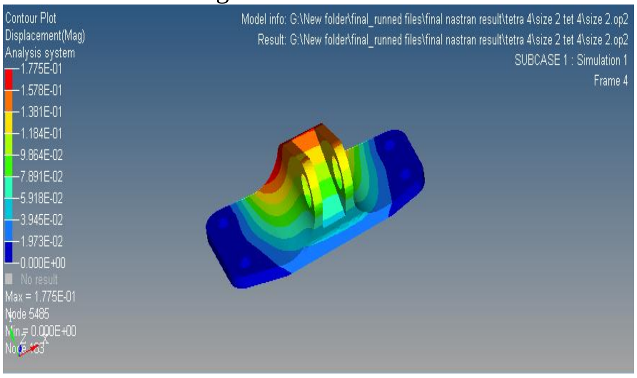


Fig. Displacement Variation

Component under the given loading condition observed the max vonmises stress of313 MPa, which is much lesser than yield strength of the material 903.21 MPa. So, Margin of safety (MS) =1.88. Component under the given loading condition observed maximum displacement of 0.177mm.

B. Tet4

Load case 2: vertical load

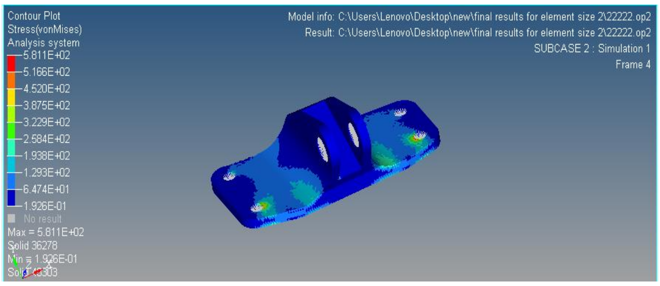


Fig. Stress Distribution

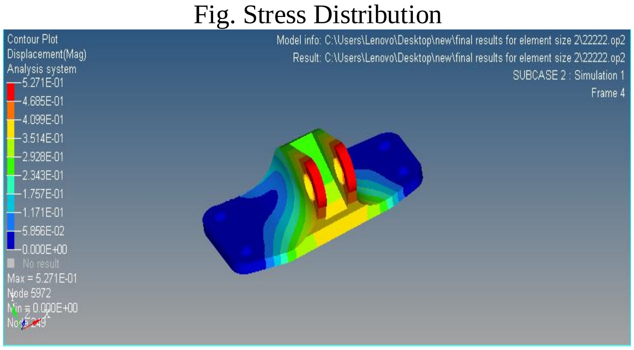


Fig. Displacement Variation

Component under the given loading condition observed the max vonmises stress of 581 MPa, which is much lesser than yield strength of the material 903.21 MPa. So, Margin of safety (MS) = 0.54. Component under the given loading condition observed maximum displacement of 0.52 mm.

C. Tet4

Load case 3: Inclined load

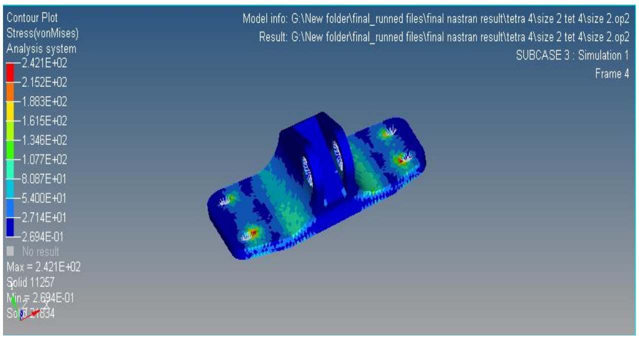


Fig. Stress Distribution

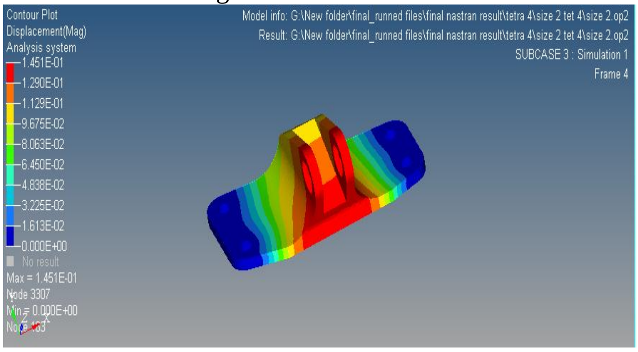


Fig. Displacement Variation

Component under the given loading condition observed the max vonmises stress of 242 MPa, which is much lesser than yield strength of the material 903.21 MPa. So, Margin of safety (MS) = 2.73.Component under the given loading condition observed maximum displacement of 0.1451 mm.

Convergence study for element size 2

Table. Comparison of results for tet4 element model of element size 2mm for material Titanium alloy (Ti6Al4V)

Load Case	TET4 Model		
	Disp.(mm)	Stress(MPa)	MOS
Horizontal Load	0.177	319	1.88
Vertical Load	0.527	581	0.54
Inclined Load	0.145	242	2.73
Combined Load	0.272	542	0.66

Convergence study for different materials of Titanium composition,
Material 1: titaniumalloy (Ti6Al4V),
Material 2: titanium alloy (8Al1MO1V),
Material 3: titanium alloy (Ti-6Al-6V-2Sn)
Displacement results for horizontal load**.**

	Material 1	Material 2	Material 3	Material 4
Load	Displacement	Displacement	Displacement	Displacement
8000	0.629	0.577	0.628	0.554
10000	0.786	0.722	0.785	0.693
11000	0.865	0.795	0.864	0.762
11500	0.865	0.795	0.864	0.762

Displacement results for vertical load.

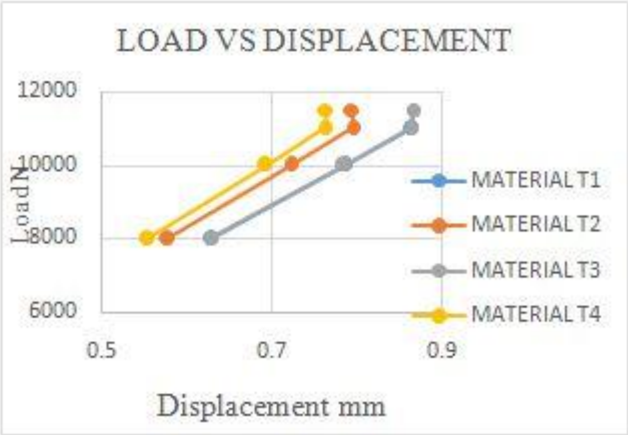
	Material 1	Material 2	Material 3	Material 4
Load	Displacement	Displacement	Displacement	Displacement
6000	0.527	0.484	0.527	0.465
7000	0.614	0.565	0.615	0.542
8000	0.704	0.647	0.702	0.620
8500	0.703	0.646	0.702	0.622

Displacement results for inclined load

	Material 1	Material 2	Material 3	Material 4
Load	Displacement	Displacement	Displacement	Displacement
9000	0.173	0.158	0.172	0.152
11000	0.178	0.194	0.211	0.186
12000	0.232	0.212	0.230	0.203
13000	0.439	0.403	0.437	0.385

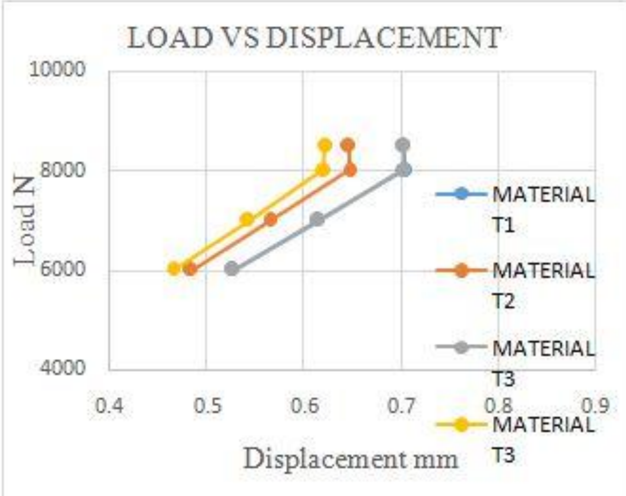
Displacement results for combined load.

	Material 1	Material 2	Material 3	Material 4
Load	Displacement	Displacement	Displacement	Displacement
9000	0.319	0.292	0.318	0.280
11000	0.415	0.370	0.403	0.356
12000	0.439	0.403	0.437	0.385
13000	0.451	0.413	0.449	0.397



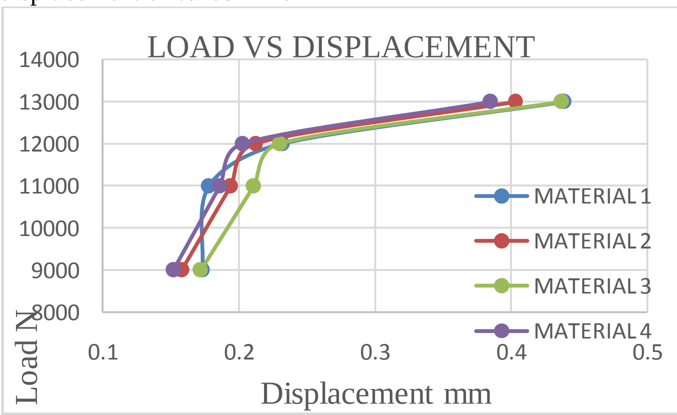
Horizontal load.

It can be noticed that material Titanium-6AL-6V-2.5Sn sustain maximum horizontal load of 11500 N with least displacement of 0.554 mm.



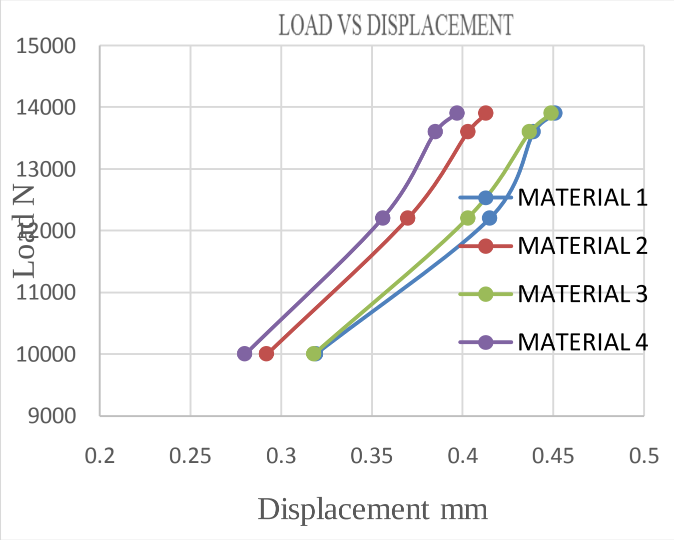
Vertical load.

It can be noticed that material Titanium-6AL-6V-2.5Sn sustains maximum vertical load of 8500 N with least displacement of 0.465 mm.



Inclined load.

It can be noticed that material Titanium-6AL-6V-2.5Sn sustains maximum inclined load of 13000 N with least displacement of 0.152 mm.



Combined load.

It can be noticed that material Titanium-6AL-6V-2.5Sn sustains maximum combined load of 13901 N with least displacement of 0.28 mm.

V. CONCLUSION

The engine bracket is one of the most important and critical components of an engine. Different analyses have been carried out considering different element size behaviour of the bracket. The study was done using Titanium alloy and it is being observed that the stresses and displacement were all within the permissible limit i.e. less than that of yield strength. From the study it is very evident that of all materials Titanium alloy had the minimal deviation from ideal behaviour, and it is safe to assume that Titanium alloy is the most suited alloy to make engine mounts. On comparison with different element size, element size 2 tends to provide optimum results in terms of both stress and displacement values. For all the four different load conditions, material Ti-5Al-2.5Sn has maximum load bearing capacity along with minimum displacement of an engine bracket, when compared to the remaining three materials.

REFERENCES

[1] Zhu J, Haghighi K ,Dynamic analysis of an engine chassis mount bracket using the finite element method; 1989, vol. 97 (6), pp. 1750-1758
[2] Pan, X., Zonni, D., Chai, G., Zhao, Y. et al., "Structural Optimization for Engine Mount Bracket," SAE Technical Paper 2007-01-2419, 2007, doi:10.4271/2007-01-2419.
[3] Umesh.S. Ghorpade, D.S. Chavan, Vinay Patil and Mahendra Gaikwad, “Finite element analysis and natural frequency optimization of engine bracket” (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-3, 2012.
[4] Gabriel-Petru Anton, Mihai Paval, Fabien Sorel, “Application on an updated Finite Element model of an engine in the Automotive Industry” SISOM 2011 and Session of the Commission of Acoustics, Bucharest 25-26 May.
[5] S.K. Loh a, W.M. China, Waleed F. Faris, “Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading”, Materials and Design 30 (2009) 3206–3216.
[6] Zhang Junhong, Han Jun “CAE process to simulate and optimise engine noise and vibration” Mechanical Systems and Signal Processing 20 (2006) 1400–1409.
[7] Mr. Pramod Walunje and Prof. V.K.Kurkute “Optimization of Engine Mounting Bracket Using FEA”, IJR, Vol.-2, ISSN-2250-1991, Issue- 12, Dec 2013, pp. 72-75
[8] P.D.Jadhav, Ramakrishna “Finite Element Analysis of Engine Mount Bracket” International journal of advancement in engineering technology, management & applied science vol.1 (sept. 2014).
[9] Sahil Naghate and Sandeep Patil, “Modal Analysis of Engine Mounting Bracket using FEA”, IJERA, Vol.-2, ISSN- 2248-9622, Issue-4, July-August 2012, pp. 1973-1979.

