Strength Enhancement of Expansion Tank Bracket in Excavators by FE Analysis.

Pradeep Chaudhari[1], Prof. K.H. Munde[2]

[1]PG Student, Mechanical Engineering Department, ABMSP's APCOER, SPPU, Pune
[2]Professor, Mechanical Engineering Department, ABMSP's APCOER, SPPU, Pune
Pune, Maharashtra, India

---***---

Abstract - *The Aim of paper is analyses the advanced design of expansion tank bracket in excavator's static loading condition. Excavators are known for heavy-duty earthmoving machines and utilized for excavation. In the age of globalization and mighty competition machines are used booming for the earth moving works like construction, mining, excavation, and forestry applications so huge attention is put on designing of the equipment's. Design engineers have to provide the improved strong design of machine parts which can work against unexpected forces and under inferior working condition. Thus, it is required for designers to provide a improved design of parts having more reliability, keeping design safe under all loading conditions.*

The expansion tank bracket is improved for better stiffness & strength. The current design had failure problem and hence its parameters were taken as basis for comparison with results of the improved designs. Based on finite element analysis outcome, optimum design is carried out for the bracket for strength. The improved design was judged for selected worst load cases of the existing design. The finite element analysis of improved models yielded displacement and stresses close to the existing design. The increase in displacement was not important and the improved design met the structural demand. The modified design is improved with one extra mounting with base to increase stability.

The present work having scope of finite element analysis as a way for improvement in strength due to improved design.

Key Words: FEA; Modal Analysis; Optimization; Mounting Bracket, Constriction machines.

1. INTRODUCTION

Present work deals with FEA analysis of Expansion Tank Bracket. It includes the modeling of the mounting brackets by changing the material & thickness of component. Materials selected are S275 and S355. Analysis includes Static and Modal Analysis of Expansion Tank Bracket using FEA tools. The study shows that this bracket will have a no failure compare to existing material S255 and withstand high stress.

A hydraulic shovel of a bucket type excavator is an earth moving machine comprising an upper rotatable chassis mounted on a drivable body with wheel or track and hydraulically powered mechanism consisting of boom, arm and bucket, mounted to the upper chassis.

The mechanism is actuated by the help of hydraulic cylinders. The machines are widely used for the digging, lifting and cleanup purpose. Trench digging in the application of placing pipes, digging applications in construction areas, rearranging face of the earth is some examples for the use of excavators.

1.1 Objectives

To design, development & analysis of expansion tank bracket for Excavator machine for improvement in strength and performance. Thus aching desired life of bracket

2. METHODOLOGY

Finite element analysis is computational tool meant for doing engineering investigation. FEM uses mesh generation method for converting a complex problem into smaller elements and a coded finite element algorithm software program. It is useful for components with complex loadings, material properties, and geometries where analytical solutions difficult to obtain. The field failure was observed on bracket thus need to optimize its design parameters.

Ansis software is used for finite element method for static analysis is used in the present study. By finding the weak link and the difference in strength between each node for overall structure, it can provide a theoretical basis and direction for the design and optimization of geometric entities.

Existing bracket is taken from Excavator for analysis, the material is steel 5000/0101 & thickness 6mm with 3 pos. mounting on cooler. The expansion tank weight 3.7 kg mounted on expansion tank bracket. The whole assembly subjected to engine vibration.

3. DESIGN STAGES

3.1 PRE-PROCESSING

i) CAD Modeling

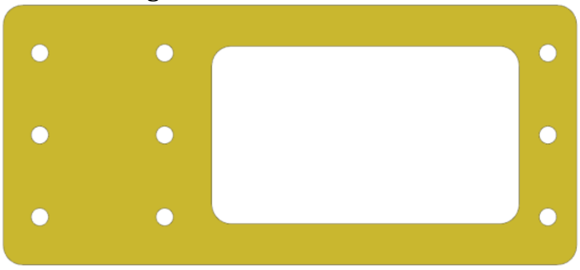


Fig-1: Original design

Material: 5000/0101; Thk: 6mm; 3 pos. mounting on cooler

Fig-2: Modified design

Material: 5000/0103; Thk: 8mm; 4 pos. mounting on cooler

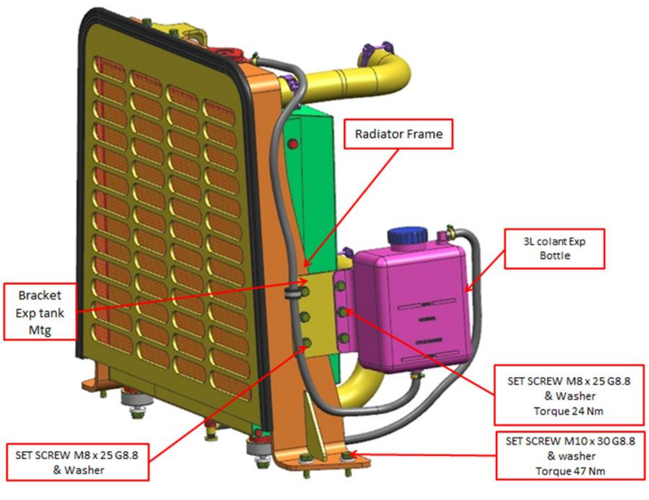


Fig-3: Test set up for Expansion Tank Bracket

A base model based on the expansion tank positioning on the support. The entire modeling is done using NX 8.5 Parametric. Material selection is important parameter as the geometry suggested a overhang bracket, became an consideration. To optimize strength a new material 5000/0103 & thickness 8mm with 4 position mounting on cooler is improved

ii) MESHING

It was decided to 3-D mesh the two component CAD models. Ansis software was used for meshing and Analysis. Tetra-mesh is done because the dimensions of the bracket are comparable to each other and also in-order to obtain accurate results.

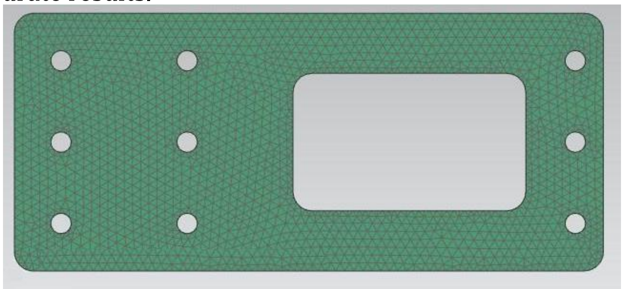


Fig-4: Original design Meshing

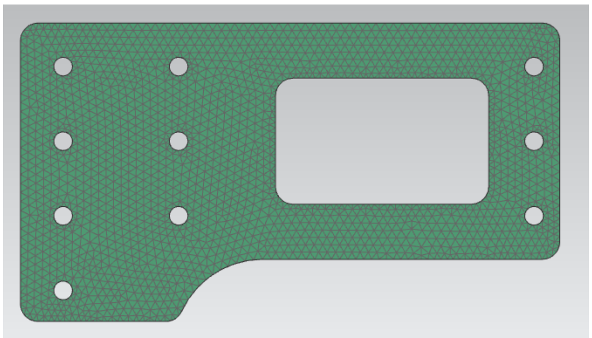


Fig-5: Modified design Meshing

iii) MATERIAL SELECTION

Component 1 was made of mild steel S275 as it was to be mounted on the structure. Young's modulus, E= 2.1 e05 MPa Poisson's ratio, u = 0.3 Density, rho = 7.9e-09 tones/mm3, Yield strength 275 N/mm2.

Component 2 made of mild steel S355, Yield strength 355 N/mm2, E= 2.1 e+05 MPa Poisson's ratio, u = 0.3 Density, rho = 7.9e-09 tones/mm^3

iv) STATIC & DYNAMIC FORCES

The expansion tank bracket is having the high vibration issue; the original design is to check under FEA and suggested modification to overcome this vibration issue. Lateral direction movement considered as critical loading which used before to overcome the fins failure. Here the Mass of Expansion tank with fluid is applied on the COG location. Refer below attached image for the scenario.

- Load case considered:

1. Lateral Loading with 37.15g
2. Modal Analysis

3.2 PRE-PROCESSING

i) Loads and Boundary Conditions

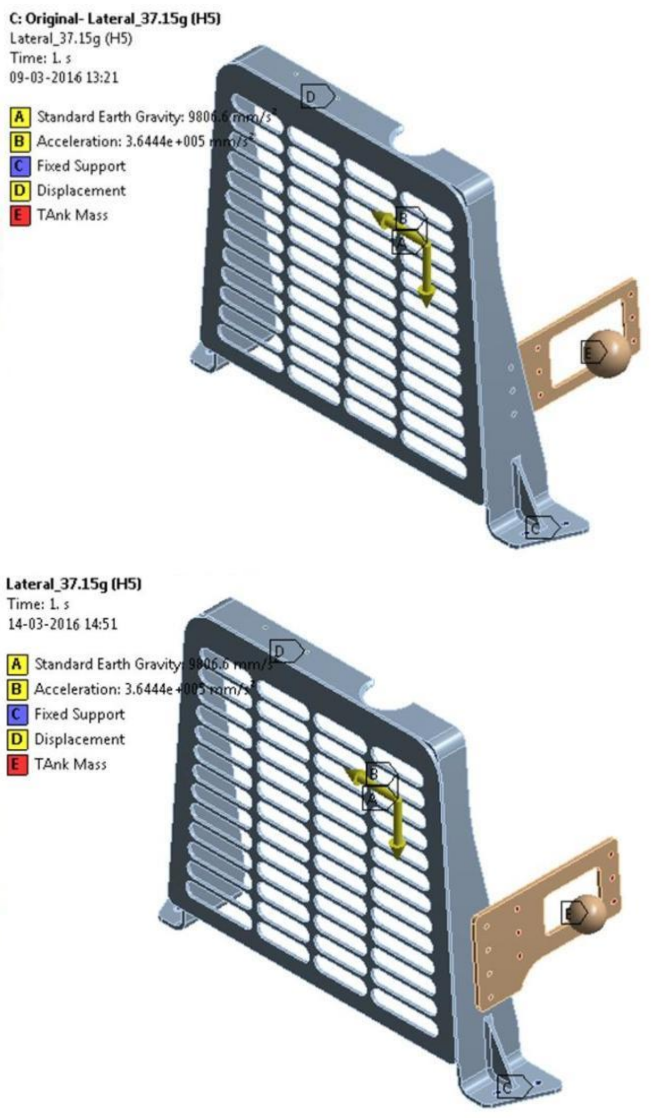


Fig-6 Loads and Boundary Conditions

After meshing boundary conditions are applied on meshed model From Fig 1,2; A is standard earth gravity which is 9806.6 mm/s2 applied, B is Acceleration 3.64 +005 mm/s2, C is fixed support, D is displacement, E is tank mass point where load of 3.7 kg is applied, so for above boundary condition result obtained as follows.

ii) VON MISES STRESS

Original design with 6 mm thickness bracket showing Von-Mises Stress beyond yield limit i.e. set to 250 MPa, which is major at center area f bracket. Increased thickness to 8mm shows improvement in Von-Mises Stress but need to upgrade the material for better strength.

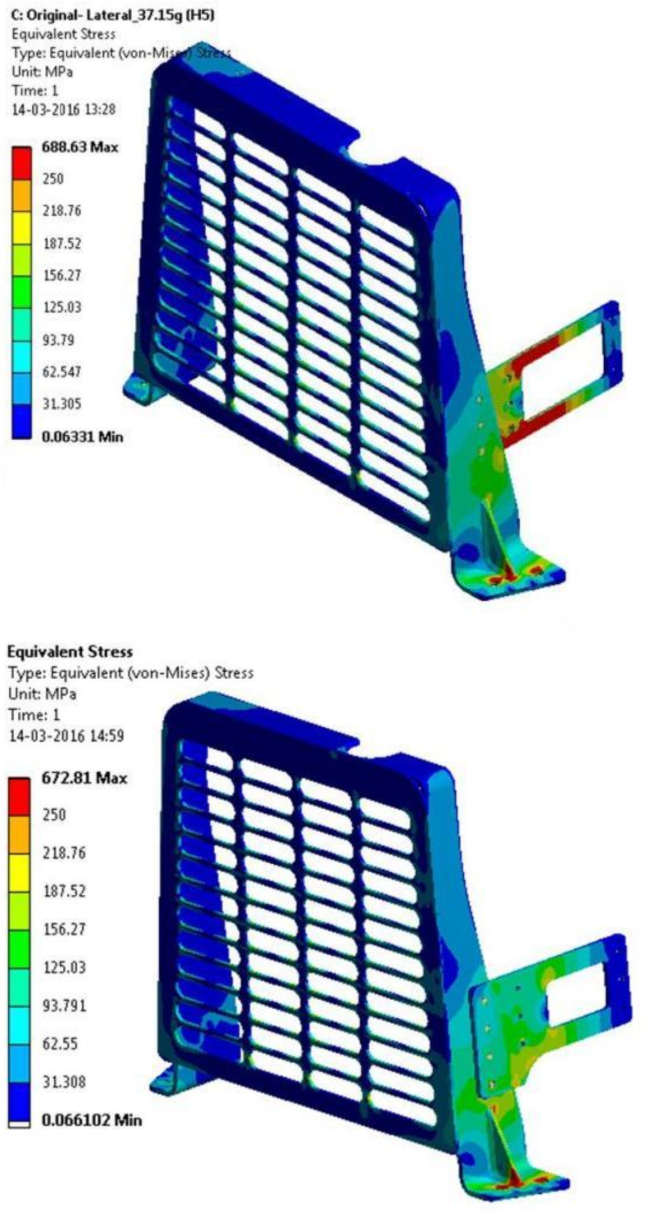


Fig-7 Von-Mises Stress Plot

iii) TOTAL DEFORMATION

The maximum value of total deformation is 22.99mm (original) & 14.207 mm (Modified) & is observed more at extreme end area of bracket.

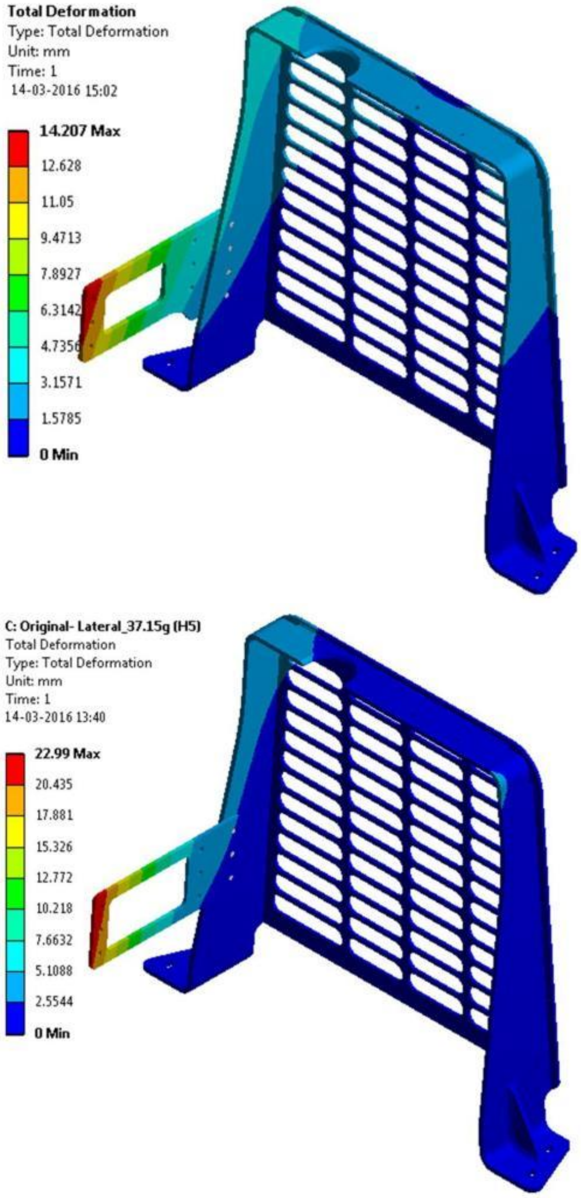


Fig-8 Deformation Plot

iv) MODAL ANALYSIS

From current analysis we can perform indepth dynamic analysis such as spectrum analysis, harmonic analysis or transient dynamic analysis. Modal analysis determines vibration characteristics of one particular component or structure in the form of mode shapes & natural frequencies. mode shapes & natural frequencies are vital in the design for dynamic loading conditions. In current analysis, linear behavior is considered, damping is not considered and applied loads are ignored in modal analysis. For performing pre stressed modal analysi, first a static structural analysis is required.

Finite element techniques are famus for the analysis of many automotive components & engineering. The design ultration depends on the product life cycle. It helps in deciding material & dimensions of the components. These analysis are very helpful the analysts or design engineers for making some design changes or approving during the post processing stage. FEA software used for analysis of an mounting bracket.

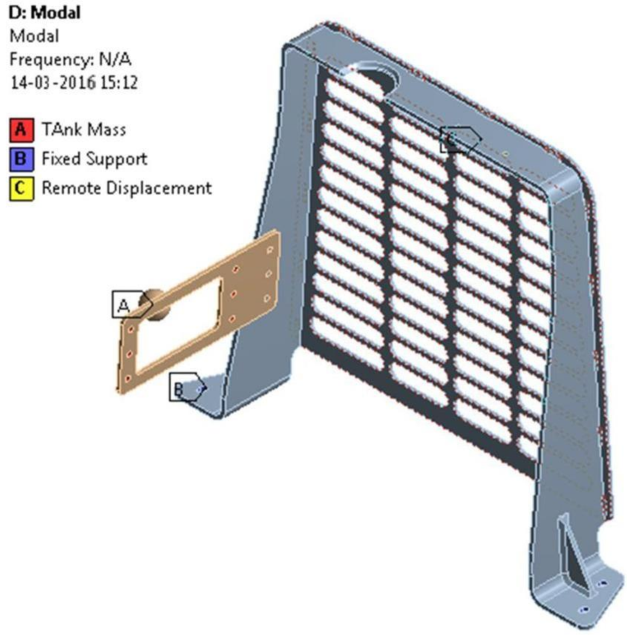
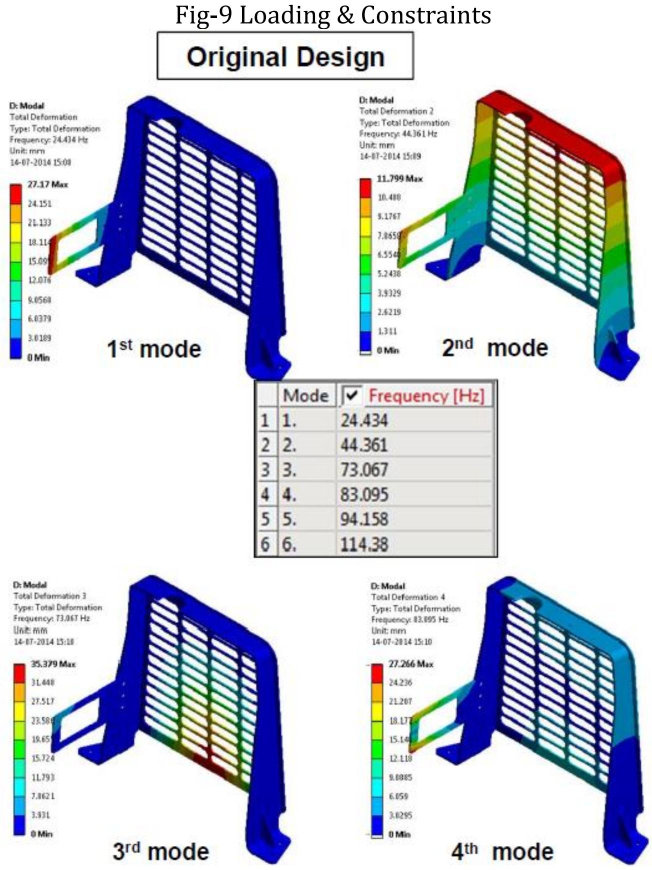


Fig-9 Loading & Constraints



	Mode	Frequency [Hz]
1	1.	24.434
2	2.	44.361
3	3.	73.067
4	4.	83.095
5	5.	94.158
6	6.	114.38

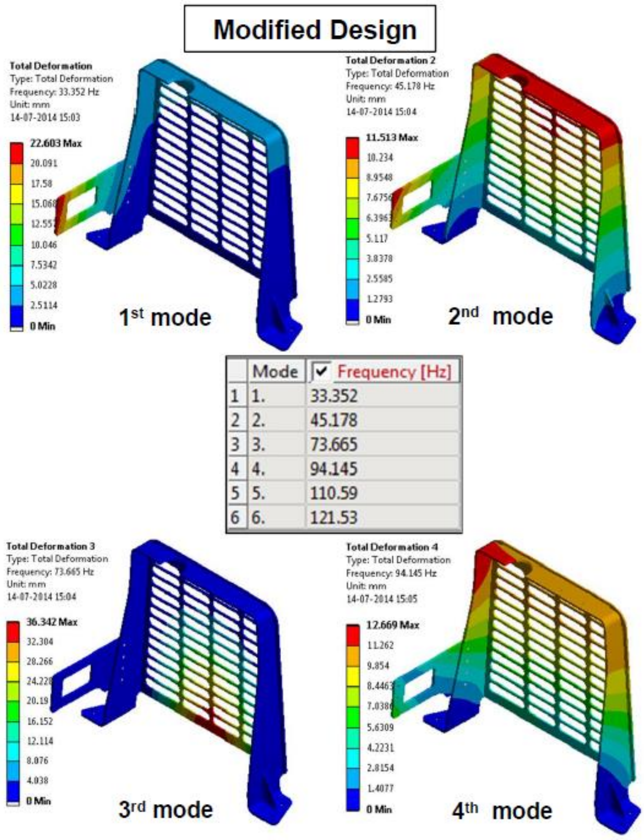


Fig- 9 FREQUENCIES & MODE SHAPES:

As seen in above table frequency mode shapes are within acceptable range for modified design.

4. Experimental setup:

Fig- 10 Assembly of Bracket on assembly line.

For the experimental method to analyses the bracket it was decided to actually test the bracket with actual loading condition on machine. The testing vehicle is assembled on JCB manufacturing plant, and is tested for its performance. It was decided to record the strain data with data logger. The full load case is tabulated in table. Since it was known that the weight of the Tank to be lifted is going to be 3.7 kg. The available load plates were of a specific load so (refer table of load plates) with a particular combination of selective plates, a load near to the load step was prepared. The load bucket was lifted by the overhead crane while the bracket was assembled to it.

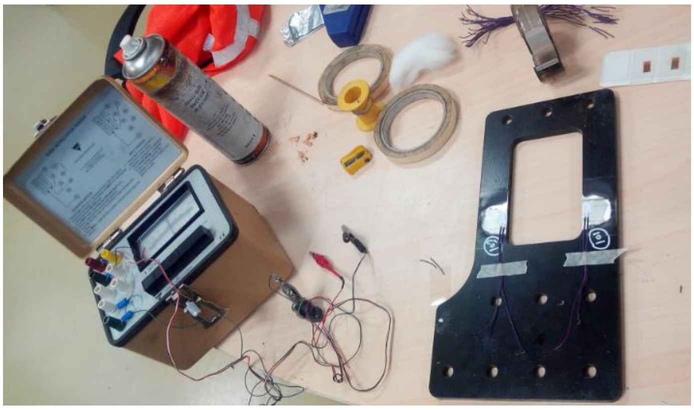


Fig- 11 Strain Gauge testing

After the static load cases, the dynamic load cases were taken. The dynamic load cases included strain measurement on a pot hole track and a wave track. The data logger was mounted on the liftall machine.

	INTENDED LOAD IN Kg	3.7	3.7	
	ACTUAL LOAD IN Kg	3.7	3.7	
	nCode FILE NAME	CFAG5	AFAG3	
Tension	CH1	1500	720	Strain values
Tension	CH2	1247	608.8	
		Original Design	Modified Design	

Fig- 12 All Strain readings

So, for 40 bar hydraulic pressure applied on piston head in cylinder body the clamping force is calculated from FEA is as follows.

Table-1: Result Table

	Maximum Displacement	Maximum Von-Mises stress	Experimental values stress
Original Design	23	325	315
Modified Design	14	160	152

5. RESULTS AND DISCUSSIONS

For original design maximum displacement is 23mm, Maximum Von-Mises stress by FEA is 325 MPa, FEA strain is 1547.6µE & experimentally stress is 315 MPa, experimentally strain is 1500 µE & the correlation is given by 96%. Fatigue life estimate for original design is 1 E6.

For Proposed design maximum displacement is 14mm, Maximum Von-Mises stress by FEA is 160 MPa, FEA strain is 761.9µE & experimentally stress is 152 MPa, experimentally strain is 720 µE & the correlation is given by 94%. Fatigue life estimate for original design is 1 E9.

6. CONCLUSIONS

In this project, the design requirement of the Bracket was studied and according to that two types of CAD had done in UG8.5. Verification of the design is carried out using ANSYS workbench. Forces and reaction forces are calculated at 3.7kg as actual weight of expansion tank. Which are validated FE analysis of the bracket and Experimental setup, improved design shows improvement in performance in terms of less vibration thus improved life of bracket.

Experimental testing is done with the use of Strain gauges for deformation measurements at different setup conditions.

So the validation results for total deformation by FEA and Experimental tests are nearly equals. Hence we conclude that results values of total deformations and Von-Mises stresses from FEA are true. Means the Bracket is accurately designed, analyzed and manufactured.

The design of connecting plate of the bracket

-Safe for the given loading conditions at 08 mm thickness

-Fails for the given loading conditions at 6mm thickness

The stress values are within the allowable stress limits 8 mm thickness.

ACKNOWLEDGEMENT

The satisfaction and exhilaration that accompany the successful completion of any task would be incomplete without the mention of the people whose constant guidance and encouragement aided in its completion.

The authors would like to express the voice of gratitude and respect to all who had directly or indirectly supported for carrying out this study and special thanks to my guide Prof.G.E.Kondhalkar, HOD, Mechanical department & Prof. K.H.Munde, ME Co-ordinator mechanical department and Prof. Dr.S.B.Thakare, Principal, Anantrao Pawar College of Engineering and Research, Parvati, Pune.

I would like to thanks Mr. Sachin kharat (Sr. Analyst FEA) who has been a source of inspiration. I am thankful to them for their support and encouragement.

REFERENCES

1. Sachin B. Bende, Nilesh P. Awate - DESIGN, MODELING AND ANALYSIS OF EXCAVATOR ARM. INTERNATIONAL JOURNAL OF DESIGN AND MANUFACTURING TECHNOLOGY (IMDMT) Volume 4, Issue 2, May – August (2013), pp. 14-20
2. Jasvir Singh Dhillon, Priyanka Rao, V.P. Sawant - DESIGN OF ENGINE MOUNT BRACKET FOR A FSAE CAR USING FINITE ELEMENT ANALYSIS Int. Journal of Engineering Research and Applications ISSN : 2248-9622, Vol. 4, Issue 9(Version 6), September 2014, pp.74-81
3. UMESH S. GHORPADE, D. S. CHAVAN, VINAAY PATIL & MAHENDRA GAIKWAD - FINITE ELEMENT ANALYSIS AND NATURAL FREQUENCY OPTIMIZATION OF ENGINE BRACKET
   Mechanical Engineering Dept., Rajarambapu Institute of Technology, Sakharale, Islampur, India. FEA (Finite Element Analysis) Dept., Vaftsy CAE, Pune, India.
   International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-3, 2012
4. R M Dhawale and S R Wagh- FINITE ELEMENT ANALYSIS OF COMPONENTS OF EXCAVATOR ARM—A REVIEW.
   International Journal of Scientific Research Engineering & Technology (IJSRET), ISSN 2278 – 0882, Volume 4, Issue 3, March 2015
5. Sahil Naghate, Sandeep Patil- MODAL ANALYSIS OF ENGINE MOUNTING BRACKET USING FEA.
   International Journal of Engineering Research and Applications, ISSN: 2248-9622, Vol. 2, Issue4, July-August 2012, pp.1973-1979
6. P.D. Jadhav 1, Ramakrishna, FINITE ELEMENT ANALYSIS OF ENGINE MOUNT BRACKET
   I.S. Jacobs and C.P. Bean, “Fine particles, thin films and exchange anisotropy,” in Magnetism, vol. III, G.T. Rado and H. Suhl, Eds. New York:Academic,1963,pp.271-350.