Design and Analysis of AC Mounting Bracket Using Composite Material

Mr. Sagar N. Narute[1], Prof. Dr. Santosh G.Taji[2], Prof. Prabhuling G. Sarasambi[3]
[1]Dept of Mechanical Engineering
[2]HOD, Dept of Mechanical Engineering
[3]Assistant Professor, Dept of Mechanical Engineering
[1, 3] SRCOE Pune
[2]Govt. Polytechnic Ahmednagar

Abstract- *Parameter like cost of vehicle and fuel efficiency mostly influenced by weights of the vehicle in the automotive industries as per the safety standard this is very important to design light weight component. The air conditioners used in cars are mounted on a bracket in the bonnet. This project intends to analyze the bracket and optimize the weight by keeping the same material of the bracket. Weight reduction will not only reduce the raw material cost, but also increase the efficiency, though very minute. The study of the topology optimization is done as per the requirement of the bracket design. This study also highlights the factors for the failure of the mounting bracket and the effect of the optimization by various analysis. In this project, we have designed an AC mounting bracket. The modelling of the bracket is done in modelling software and analyzed using ANSYS. The glass fiber bracket is designed using design of experiments and analyzed in ANSYS*

Keywords- AC mounting bracket, ANSYS, Design Of Experiments

I. INTRODUCTION

While designing the vehicle structure it is very tough job to obtain the higher stiffness and strength and also minimize the weight of the component. Compressor mounting bracket is the bracket used to mount the air conditioner compressor in the car. Mounting bracket goes under certain problems like design space issue, material used , weight of the bracket affecting the performance etc

A. Types of brackets

1. Engine Mounting Bracket of Car

Engine mounting bracket of the car is the bracket used to mount the engine from the back side. It is made of steel. The large face of the bracket is connected to the engine while the small end of the bracket is connected to the vehicle structure for taking load and vibrations. Due to less vibration rate and knocking rate of the engine its operational life is more. But if the engine is old or there are some other problems related with the vehicle structure, then there are large chances of failure of the engine mounting bracket. Crack in the bracket is the main failure due to high stresses generated in the bracket.

Fig.1: Engine mounting bracket of a car [8]

2. Aero plane engine’s continental engine mounting Bracket

Fig 2: Aero plane engine’s continental engine Mounting Bracket [8]

A mounting bracket is used as a base member having a flat upper surface and an elongated shoulder extending upward from the base surface. The mounting bracket consists of bracket member having an upper surface adapted to support a component and a flat lower bracket surface. The base is connected to the plane structure and the other part connected to the engine which takes most of the load. It is made up of aluminum casting.

3. AC compressor mounting bracket

The compressor plays a very important role in the automotive air conditioning system. The unbalanced forces produced from the engine and compressor causes the structure vibrations. The compressor is supported by the engine mounting to reduce the vibratory forces is called compressor mounting bracket.

II. LITERATURE WORK

Harshal Bankar and P. Baskar [2] In this study the researchers have said that simulation plays very important role in the Automotive industries for the higher levels of quality, better cost effectiveness and quick market response. In this paper, the use of dynamics analysis technique is used for the simulation of the compressor mounting bracket for various vibration loads. The standard testing conditions were used for the testing of the compressor mounting bracket. The results showed that resonance in the dynamic analysis is the major cause for the failure of the compressor mounting bracket, under static analysis, under the same **magnitude of load resonance cannot be predicted.** Thus, dynamic analysis gives best results for design validation of the compressor mounting bracket.

William Nadir and Kim Yong presented a paper "Structural Shape Optimization Considering Both Performance and Manufacturing Cost" which says a structural shape optimization method that considers not only structural performance but also manufacturing cost. Most structural optimizations only take into account structural performance metrics such as stress, mass, deformation, or natural frequency. However, it is often observed that structural performance improves at the expense of manufacturing cost. This work explores the tradeoff between mass and manufacturing cost with the application of the abrasive water jet (AWJ) manufacturing process. Structural performance, defined as maximum von Mises stress, is a constraint in this work. Work-in-progress results are presented for two structural design examples to demonstrate this tradeoff between mass and manufacturing cost while investigating shape optimization using non-uniform rational B-spines (NURBS). Additional work is still needed to complete this research project.

R. P. Kumar, Dr. K Rambabu [6] Studied the parameters like cost of vehicle and fuel efficiency are mostly influenced by the weight of the vehicle in the automotive industries. As per the safety standards this is very important to design the light weight component. This paper describes the study of the optimized design of the Air-Conditioner compressor mounting bracket. The study of the topology optimization is done as per the requirement of the bracket design. This study also highlights the factors for the failure of the mounting bracket and the effect of the optimization by various analysis.

M. Singh, D. Singh and J. S. Saini [3] Analysed the automotive air-conditioning industry aiming at higher levels of quality, cost effectiveness and a short time to market, the need for simulation is at an all time high. In this work, the use of dynamics analysis is proposed in the simulation of the automobile air conditioning condenser assembly for the vibration loads. The condenser assembly has been analyzed using the standard testing conditions. The results revealed that the components of condenser assembly may fail due to resonance in dynamic analysis. Thereafter, the condenser assembly was optimized, resulting in a 2 % reduction in mass.

Pushpendra Mahajan [5] In this the researcher have said that NVH is one of the major factors impacting quality for household appliances like refrigerators. In refrigerators, compressor is the main source for vibrations and noise. Compressor is attached to compressor mounting plate which is then attached to refrigerator body. Compressor being dynamic component also exerts harmonic exiting forces on the mounting plate. If compressor operating frequency matches with natural frequency of plate then resonance would occurred leading to excessive vibrations and noise. Hence the plate should have natural frequency beyond the operating range of compressor. Natural frequency and static state deflection of a compressor mounting plate are analyzed using FEA software, ANSYS. Further two methods of improving and optimizing the design to increase the natural frequency are illustrated and analyzed.

III. PROBLEM SPECIFICATION

Weight optimization of the components mounted on the automobile is one of the measure area of study in today's engineering studies. We need to design the compressor support bracket for Ashok Leyland 220 Bus, using different materials including conventional as well as composite materials. We need to find out optimum solution for this application's support bracket even considering material and manufacturing costs

IV. MATHEMATICAL CALCULATIONS

Compressor is installed on the bracket with the help of bolts, which allows us to change the width of the bracket which can be less or more than the width of the compressor. Bolts and base of the compressor may have shape which will help us bolting the compressor to the plate. This leads to shape of side closed L bracket, Dimension of the bracket will be 446 mm in length and 150 mm in width.

Following stress needs to be checked

- Shear Stress.
- Direct and Bending Stress

Weight on the bracket is 17.4kg.

Material for manufacturing of bracket is selected as steel.
Forces acting on the component installed on automobile are as follows 3g loading in all three directions independently acting on the component when applied on the compressor.
Area of the plate under compressor is 450mm by 150mm

For DOE of design calculations as sample calculations are considered. Area of plate under the compressor:-
$A=450\times150=6750MM^2$
acting load =17.4 kg×3×Gravetational acceleration
Load acting L=512.08 N
Stress direct =512.08÷67500
Stress direct=0.0079 Mpa
So direct stress is negligible.
Design thickness of plate according to shear limit.
Small area under the shear
$A_{sh\ ear=}$ W×t
Force =512.08 N
Shear stress =F/ A_{Shear}
Shear stress =512.08/(150×t)
Shear stress all=0.5×S_{yt} /FOS
Factor of safety selected as 2
Shear stress $_{all}$ =92.5
Hence
t=3.41/46.20
t=0.035MM
38 gauge sheet should be selected which is approx 0.152 mm thick. Also we must check thickness according to bending stress.

Bending forces acting on base plate are on the centre of compressor or acting downwards.

512 N downward causes couple with distance of 75mm frame support at cross-section B just near the support bending stress will be caused by 512 N as follows

M=F× (W/2)
M= 38.400
According to bending moment we know that
(M/I)-(σ/Y)-(F/R)
σ is maximum when Y = max
Where
Y = distance of stress from neutral axis
Y=t/2
$I=bd^3/12$
σ_{all} = M.Y /I
$=38400\times6\div(150\times t^2)$
σ_{all} = 370/ 2
t^2=8.30
t = 2.88 mm

According to these calculations different materials are selected and DOE is performed to select the best alternative for the design for the compressor support system links.

TABLE.1 VARIOUS DESIGN FOR MATERIALS AND WEIGHT

Mc(kg)	F(N)	L(mm)	W(mm)	Mt(N-mm)	FOS	Material	Yield Strength	E (Mpa)	Density (kg/m^3)	Allowable Stress	t
17.40	512.08	450.00	150.00	38406.15	2.00	MS (AISI 1018)	370.00	205	7.87E-06	185.00	2.88
17.40	512.08	450.00	150.00	38406.15	2.00	SS 304	205.00	193	8.00E-06	102.50	3.87
17.40	512.08	450.00	150.00	38406.15	2.00	304 L	170.00	193	8.00E-06	85.00	4.25
17.40	512.08	450.00	150.00	38406.15	2.00	304 H	205.00	193	8.00E-06	102.50	3.87
17.40	512.08	450.00	150.00	38406.15	2.00	SS321	205.00	193	8.03E-06	102.50	3.87
17.40	512.08	450.00	150.00	38406.15	2.00	SS 310	205.00	200	8.00E-06	102.50	3.87
17.40	512.08	450.00	150.00	38406.15	2.00	SS 316 H	205.00	193	8.00E-06	102.50	3.87
17.40	512.08	450.00	150.00	38406.15	2.00	SS 420	345.00	200	7.80E-06	172.50	2.98
17.40	512.08	450.00	150.00	38406.15	2.00	SS 309	310.00	200	8.00E-06	155.00	3.15
17.40	512.08	450.00	150.00	38406.15	2.00	AL 6063	90.00	60	2.70E-06	45.00	5.84
17.40	512.08	450.00	150.00	38406.15	2.00	AL 2014	414.00	72.3	2.80E-06	207.00	2.72
17.40	512.08	450.00	150.00	38406.15	2.00	AL 6351	150.00	75	2.70E-06	75.00	4.53
17.40	512.08	450.00	150.00	38406.15	2.00	GFRP	200.00	35	1.7E-06	100.00	3.92

TABLE.2COST OF VARIOUS MATERIALS

Material	Weight (kg)	per Kg cost	Material cost
MS (AISI 1018)	4.08	60.00	244.93
SS 304	5.57	175.00	975.59
304 L	6.12	175.00	1071.33
304 H	5.57	175.00	975.59
SS321	5.59	205.00	1146.70
SS 310	5.57	310.00	1728.19
SS 316 H	5.57	300.00	1672.45
SS 420	4.19	94.00	393.85
SS 309	4.53	290.00	1314.70
AL 6063	2.84	250.00	709.91
AL 2014	1.37	400.00	549.21
AL 6351	2.20	270.00	593.88
GFRP	1.2	200	240

From material list table 1 we selected MS (AISI 1018) as design material. This is used to manufacture the steel design. Calculations are repeated for GFRP material. And values for cost and weight are calculated. Best selected steel has weight of 4.1 kg and costs around 245 Rs for material, as shown in the graph. When designed using GFRP material. Allowable stress for material is100 MPa, and cost per kg for GFRP material is 200 Rs. Values for the GFRP are put in the calculations and thickness for the bracket is calculated. Where ρ – Density of GFRP material which is 1.70 E-6 kg/mm^3 so design of GFRP with 4 mm is selected as the weight reduction alternative which only costs 240 Rs, which is similar as material cost of the steel which provides around 70 % weight reduction

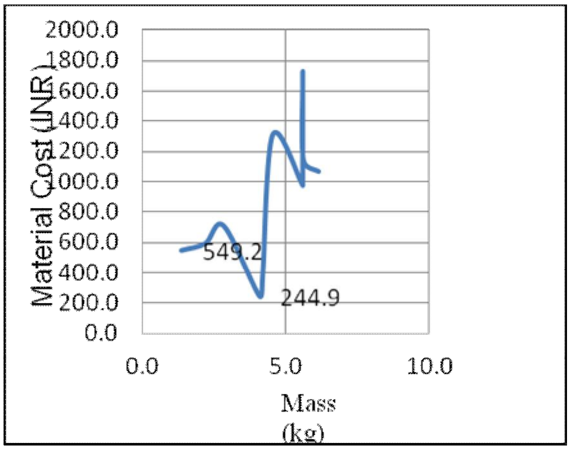

Fig.3. Weight of design Vs Material cost for bracket

The above graph shows relation between material cost and weight of the material. On X axis, the weight of the material is plotted. On Y axis, the material cost is plotted. The above graph is plotted with help of table no 2.

V. FINITE ELEMENT ANALYSIS

Finite Element Analysis is simulation method to solve the real world physics problems using numerical analysis techniques and finite element shapes. In this current work we focus on static structural analysis of the steel bracket for compressor support of the Ashok Leyland Bus and we also run the analysis on the bracket with different materials that is using run analysis and design. Compressor support bracket model is created using CAD software and analyzed in ANYSY software

A. *Steel Bracket*

Steel bracket of AISI 1018 material as selected from the design chapter will be used

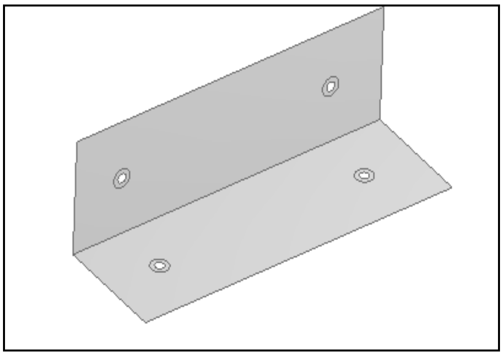

Fig.4. Geometry of compressor support bracket

Meshing is performed on the bracket using shell elements of 2D with element type Shell 181. Rectangular mesh elements are used to create the grid. Image of the meshing module for the same is shown Fig.4 Total of 0.13 million elements and approximately same number of nodes are used to create the meshing.

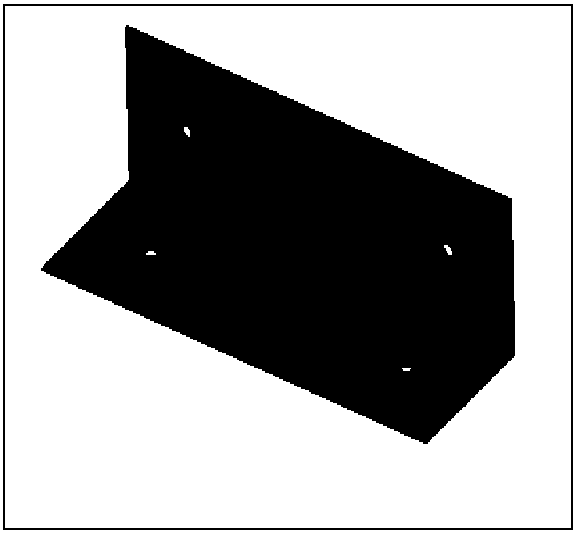

Fig.5. Meshing Compressor mounting bracket

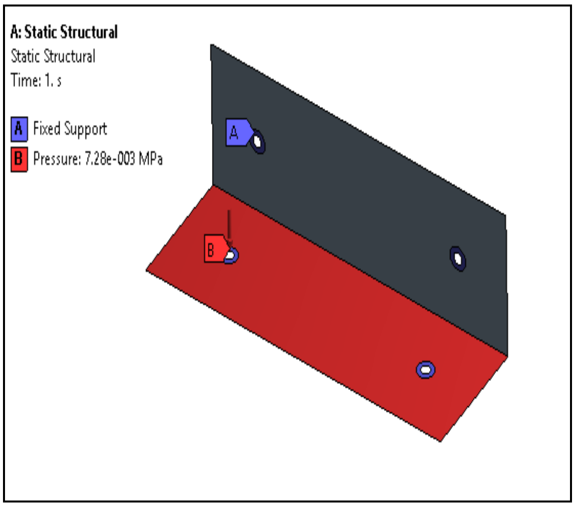

Fig.6. Boundary loading condition

Boundary condition and loading for the static analysis are shown in the Fig 5. It shows that total of 0.00728 MPa pressure is applied on the surface to simulate 512 N load due to 3 gram acceleration of the compressor assembly towards the base. Also All four bolt holes are applied with fixed boundary conditions to simulate its bolting connection with the chassis. Total mass of the bracket is around 4 kg.

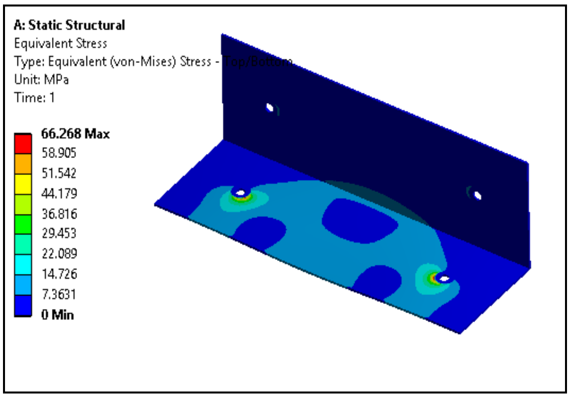

Fig.7. von Mises stress of Compressor mounting Bracket

Von Mises stress plot shows that maximum stress at the steel bracket of compressor due to 3 gram loading is 66.27 MPa which is well within the acceptance criteria of 185 MPa . So it shows that steel design is safe according to static FEA results.

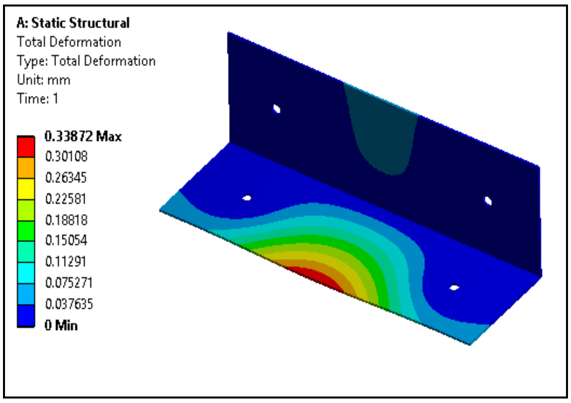

Fig.8. Total deformation of Compressor mounting Bracket

Total deformation of the steel bracket in fig.7 the maximum deformation in the bracket observed as 0.34 mm which is negligible considering only 1/3 of the load is applied continuously on the bracket in actual conditions

B. GFRP design bracket

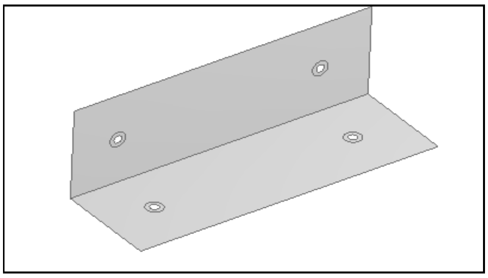

Fig.9. Geometry of GFRP compressor support bracket

Surface is extracted from the model to create compressor support bracket using GFRP material. This will be assigned with good meshing size and thickness in terms of ply material properties in Advance Composite Preprocessor of ANSYS.

Fabric 2,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=90.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=90.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=90.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25

Fig.10. Ply model for ANSYS ACP

In the plies model as shown in the fig.9 Fabric 1 is of uni directional Glass fiber properties and fabric 2 is made of Epoxy Resin used to bind Glass fiber cloth together. Alternate cloth layer of glass fiber is given the angle of 90 degrees which means it is the woven cloth with 90 degree inclination of fibers in between. Polar material properties created due to lay up formation are shown in the fig.10.Total 4 mm thickness for GFRP material is created using this layup technique. Total weight of the component using GFRP material is 1.2 kg.

Fabric 2,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=90.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=90.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25
Fabric 1,a=90.0, t=0.25
Fabric 1,a=0.0, t=0.25
Fabric 2,a=0.0, t=0.25

Fig.11. Material Properties and plies of GFRP

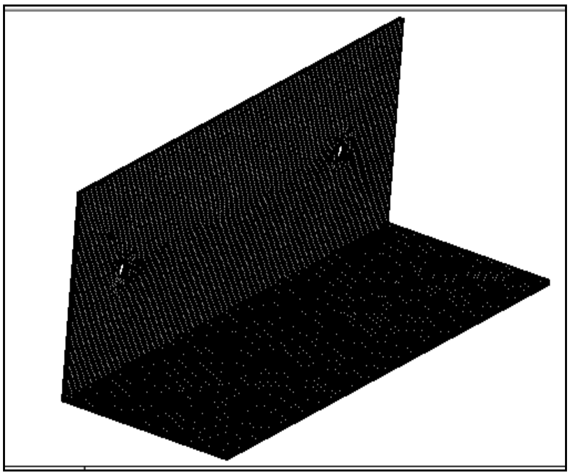

Fig.12. Meshed model

Fig.12 shows the meshed model for GFRP compressor support bracket. Nodes are used in meshing is 5 lakh and elements are around 5 lakh 33 thousand elements are used. Solid model created in preprocessing module for ACP ANSY is imported in the static analysis module of the ANSYS. Following boundary conditions are applied on the model.

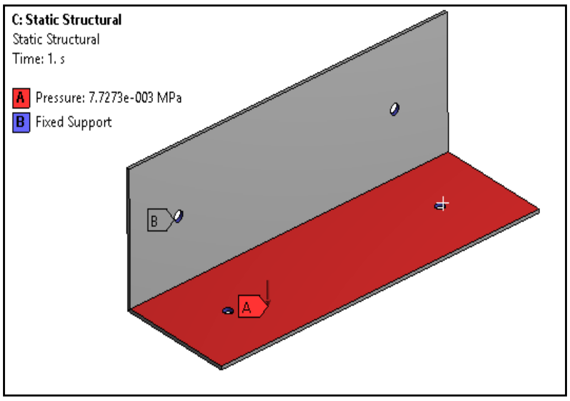

Fig.13.Boundary Conditions on the GFRP bracket

Fig.13 shows that boundary conditions on the compression mounting bracket. Horizontal face is applied with the pressure of 0.007723 MPa to simulate the 512 N force applied due to 3 g loading of the weight of compressor assembly. Loading calculations are done in the design chapter. Load is assumed to be distributed all along the surface of the bracket. Fixed degrees of freedom are applied to all four bolding locations where bracket is bolted to the chassis. From fig.13 show that the results for the analysis ran on the GFRP bracket using these boundary conditions.

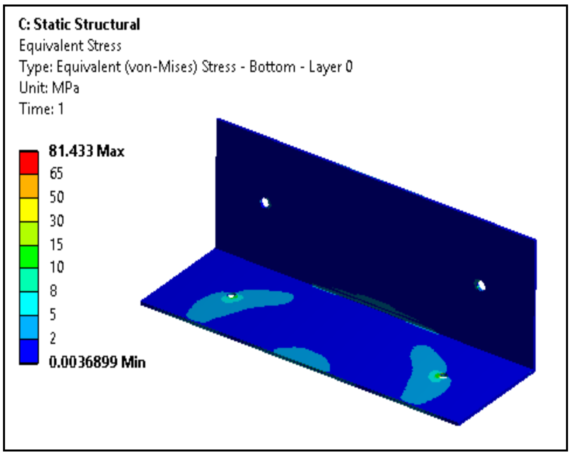

Fig.14. von Mises stress plot GFRP design

Maximum von Mises stress observed in GFRP design is 81.43 MPa which is point stress. Stress is within the acceptance criteria of 100 MPa used in design.

Maximum deformation plot for GFRP design shows us as 3.46 mm. this is acceptable deformation which is within the acceptable limit at 3gram loading. In terms of gravitational load 1.5 mm. GFRP being less stiff material that steel the results are as per expectation.

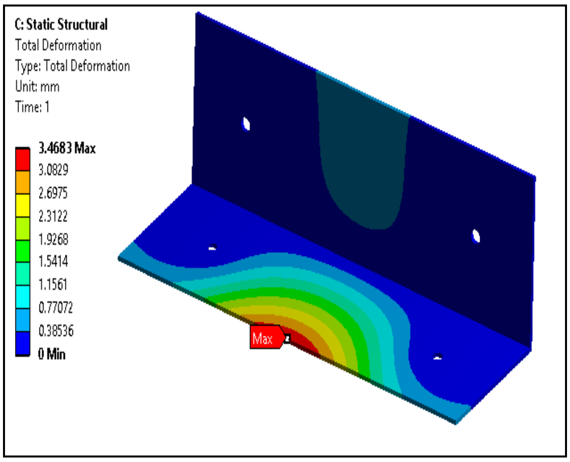

Fig.15. Maximum deformation plot for GFRP design

VI. EXPERIMENTAL VALIDATION

For experimental validation of GFRP material bracket is to be manufactured for the design dimensions. Manufacturing technique used for the GFRP bracket fabrication is hand layup technique where, Glass fiber cloth is cut using scissor in the required shape of the component. Mould is created using steel or wood for the required shape of the component. Resin and hardener are mixed in the ratio as per guidelines to create epoxy adhesive. Release agent is applied to the mould before starting the fabrication. After that adhesive layers are applied on the top and bottom of each glass fiber cloth. Once the required thickness is achieved, mould is closed with the second part of it. Pressure is applied on the mould using bolting or by applying external load. Mould is kept under that pressure during curing time. Once the curing period is over material is removed from the mould to find ready our shape of the GFRP component. Holes and other small features in the components are machined later. Fig 15&16 are of the fabrication process for compressor support bracket

.

Fig.16. Test setup for GFRP material bracket

Fig.17 Compression loading of GFRP bracket

VII. .RESULTS AND DISCUSSION

TABLE.3 FEA RESULTS SUMMARY

Iteration	FEA Max Stress(MPa)	FEA max deformation (mm)	Weight (grams)	% material Remove d
Steel	66.27	0.3387	4008	0
GFRP	81.43	3.47	1200	70.06%

TABLE.4 EXPERIMENTAL RESUT FOR STEEL

	FEA for Steel	Experimental for Steel	% Error
Deformation (mm)	0.3387	0.37	9.24%

TABLE.5.EXPERIMENTAL RESULT FOR GFRP

	FEA for GFRP	Experimental for GFRP	% Error
Deformation (mm)	3.47	3.5	0.8%

FEA results table 3 shows that us the comparison of FEA results between steel and GFRP material designs of bracket. Following observations are made from the analysis results table .

FEA result summary table 3 shows that the maximum stress observed in the steel bracket is 66.27 MPa which is well within the acceptance criteria for the steel design. GFRP design FEA that the maximum stress observed in the bracket is 81.43 MPa which is within the acceptance criteria of the GFRP material. Total deformation observed in the steel and GFRP designs in FEA are 0.34 mm and 3.47 mm respectively. Even though deformations observed in GFRP material design are quite high they are well within the acceptance criteria for the regular loading of the bracket so results for FEA are acceptable limit. Weight of the steel bracket was observed to be 4.1 kg in while GFRP bracket design for the same application weighs only about 1.2 kg. Overall 2800 grams weight is reduced. From the table number 4 it is observe that the deformation is FEA for steel was 0.338mm while in experiment it was observe 0.35mm percentage error is 9.24% For GFRP bracket FEA results shows deformation 3.47 mm and experimental testing it was 3.50 mm which very closed to FEA results percentage error was 0.8%.

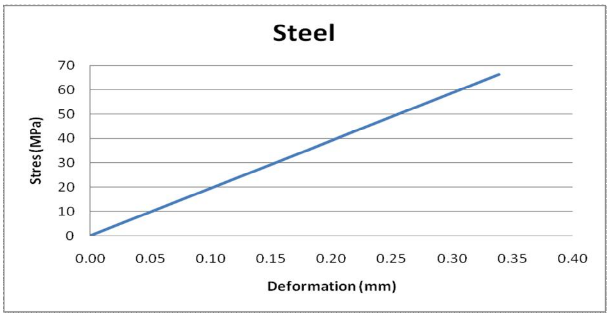

Fig.18. Stress vs Deformation for Steel bracket of FEA result

Graph of steel material AC mounting bracket stress vs deformation in FEA shows that maximum stress of 66.27 MPa is observed at maximum deformation of 0.3387 mm.

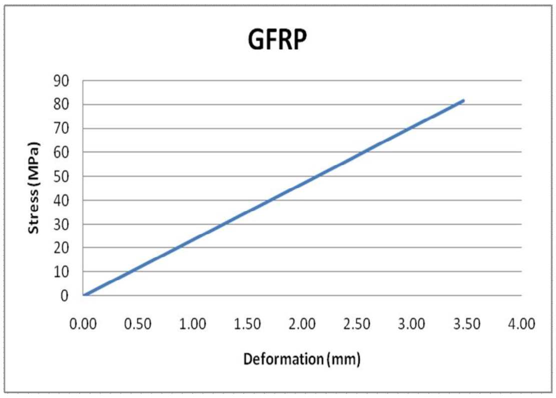

Fig.19. Stress vs. Deformation for GFRP bracket of FEA result

Graph of stress vs. deformation for GFRP material shows that the maximum stress of 81.43 MPa is observed in the FEA of GFRP bracket at maximum deformation of 3.47 mm.

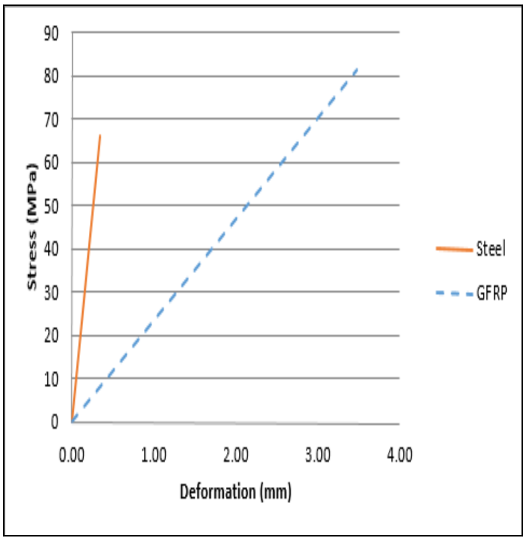

Fig.20. Stress vs.Deformation combined

In comparison it can be clearly seen that steel bracket deforms much lower than the GFRP bracket for same loading conditions but stress and deformation both are within the acceptance criteria

VII. CONCLUSION

From work done in the design, FEA and testing chapters of the project it can be concluded that best suitable steel for the manufacturing of the compressor support bracket with least material price is AISI 1018 MS for which weight of the bracket design is 4.1 kg. Best alternative material to save weight of the steel bracket is GFRP material to replace the steel as bracket design material. Using GFRP when designed for the same application bracket weighs 1.2 kg which is 70 % less weight than the steel design.

With same material cost and may be slightly increased manufacturing cost we can achieve 70 % weight reduction in the conventional steel design of the bracket which is calculated in the design chapter and verified by performing FEA on the bracket geometry.

REFERENCES

[1] .S. Adkine, V. S. Kathavate, G. P. Overikar S. N. Doijode, “Static Behaviour of Engine Mounting Bracket”, International Advanced Research Journal in Science, Engineering and Technology Vol. 2, Issue 4, April 2015

[2] Harshal Bankar, P. Baskar, “Dynamic Analysis of Air Conditioner Compressor Mounting Bracket”, International Engineering Research Journal (IERJ) Special Issue 2 Page 4825-4829, 2015, ISSN 2395-1621.

[3] M. Singh, D. Singh, J. S. Saini, “Dynamic Analysis of Condenser Assembly of Automobile Air Conditioning System Using CAE Tools”, Springer, J. Inst. Eng. Ser. C (April–June 2013) 94(2):135–145.

[4] Mallikarjun B Patil, Prof. S.S Kelkar, “Finite Element Analysis and Natural Frequency Optimization of Engine Mounting Bracket”, International Engineering Research Journal Page No 981-989

[5] R.P. Kumar, Dr. K Rambabu, “Study of Design and Analysis of Air Conditioner Compressor Mounting Bracket”, International Journal of Science and Research (IJSR) ISSN (Online): 2319- 7064

[6] Umesh S. Ghorpade, D. S. Chavan, Vinaay Patil & Mahendra Gaikwad, “Finite Element Analysis And Natural Frequency Optimization Of Engine Bracket”, International Journal of Mechanical and Industrial Engineering (IJMIE) ISSN No. 2231 –6477, Vol-2, Iss-3, 2012.

[7] Pushpendra Mahajan, Prof. Abhijit L. Dandavate, “Analysis and Optimization of Compressor Mounting Plate of Refrigerator using FEA”. Website: www.ijetae.com (ISSN 2250-2459, ISO 9001:2008 Certified Journal, Volume 5, Issue 5, May 2015).

[8] Vyankatesh D. Pawade, Pushkaraj D. Sonawane, “Study of Design and Analysis of Air Conditioner Compressor Mounting Bracket”. International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064 Index Copernicus Value (2013): 6.14 | Impact Factor (2013): 4.438.