Design & Impact Analysis of Engine Mount Bracket byVarying Weld Bead Size

Mr. Muzammil Inamdar, Prof (Dr) D Y Dhande

Department of Mechanical Engineering, AISSMS College of Engineering, Pune - 411001, India

ABSTRACT

Fatigue damage assessment is very important in design process of the component to determine their durability under service loading conditions. In service, the great majority of structures and components are subjected to stress of variable amplitude loading. The purpose of this project is to analyses statistically strain & stress concentration factors from road loading and compare their effect on fatigue damage of the engine mount bracket. Meanwhile, damage of the engine mount bracket will be evaluated using finite element commercial software. From the analysis, we will observe that the fatigue damage increment with the respective statistical values of the boundary condition & also by varying throat thickness of the weald bead of the bracket to reduce other machining operations.

Keywords: static structural analysis, FEA testing, ANSYS workbench, Vibrational investigation, & topology optimization.

1.0 INTRODUCTION

The need for light weight structural materials in automotive applications is increasing as the pressure for improvement in emissions and fuel economy increases. Most effective way of increasing automobile mileage while decreasing emissions is to reduce vehicle weight. The incorporation of aluminum and magnesium alloys into automotive structures has steadily increased to meet all these requirements. At present the average use of aluminum per vehicle is of 87 kilograms as per the study.

The strategy of increasing use of light alloy content in vehicle has proven to be a successful method of achieving fuel efficiency and environmental concerns. The strong emphasis on the cost has demanded the component manufacturers to improve the performance of their materials and to find the methods to deliver these materials at reduced cost.

Requirement of Engine Mounting Bracket

The mounts can cause a variety of problems for your vehicle.

1. **Engine Vibration:** The first symptom to note is an excessive amount of engine vibration. Because motor mounts are meant to keep an engine secure, bad mounts will lead to an insecure engine that will bounce about.
   At times, there may be a sound emitting from the engine hinting of vibration, however, the more common symptom is a felt vibration on the passenger's side.
   If you do not often have visitors sitting on the passenger's side that can tell you something is wrong, place your hand on the passenger's seat from time to time to check for excessive vibration.
1. 2. Misalignment: Securing a motor not only fastens an engine, but aligns it, meaning that the engine's height is ensured to be equal on all sides.
   If the motor mounts are in fact bad, the engine will sag and droop to one side. Again, there may be noises emitting from the engine that do not sound quite right.
   Inspect the engine to see if it is tilted. If so, your mounts aren't doing their job.
   3. Engine Damage: The third symptom is an extreme case. If motor mounts break off completely from an engine, and are not just loose or cracked, an engine can shift from one side to another, bouncing about.
   This presents a safety hazard if you are driving your vehicle in high speeds; the engine may shift and bounce about so violently that various parts will fly off.

Fig 1.1 bracket with weald

In this automotive era the need for light weight structural materials is increasing as there is a more focus on fuel consumption reduction and improvement in decreasing the emission.

The magnitude of production volumes has traditionally placed severe requirements on the robustness of process used in the manufacturing.

The manufacturers have strong importance on the cost has the demand for the component to improve the material performance and to deliver these materials at low cost is the requirement.

In automobile sector the extremely competitive automotive business needs manufactures to pay a lot of attention to travelling comfort. Resonant vibration is from unbalanced masses exist within the engine body; this is causing the designers to direct their attention to the event of top-quality engine mounting brackets so as to confirm that there is improvement in riding comfort.

The demand for higher playacting engine mount brackets should not be offset by arise within the production prices and/or development cycle time.

In diesel engine, the engine mounting bracket is the major problem as there is unthrottled condition and higher compression ratio and even there are more speed irregularities at low speed and low load when compared to gasoline engines. So due to this there are more vibration excitation.

By this vibration engine mount bracket may fail, so by optimizing the shape and thickness of engine mount bracket we can improve the performance at initial design stages.

By some studies it is observed that brackets saved 38% of mass. Structural optimization is an important tool for an optimum design; comparison in terms of weight and component performance structural optimization techniques is effective tool to produce higher quality products at lower cost.

1.1. Problem Statement:

One of the most common signs of a failing motor mount is what we call "impact noises" that you will hear coming from the engine bay. You may hear significant clunking, banging or rattling, and that means the engine could be loose at the point of one or more of the motor mounts.

As we all know that how important, have transportation has become in today's life. In each & every home at least, we find more than two automobiles. So, in the same concern poor people were not eligible to buy automobile so they travel by bus, auto, trains & other source of transport. Most of the Indian revenue is by exporting goods to foreign country's/other parts of India, hence need a transportation vehicle, these vehicles are heavy vehicle hence safety factors play an important role in manufacturing it Because of load carrying. Whenever a vehicle fails disaster takes place. In order to provide safety to automobiles testing & checking of each component in vehicle will be an important task. So, our project scopes over analysis of engine mounting bracket for safety consideration of the vehicle. The objective of this work is to newly design an existing component and compare out best material and most suitable design assembly with the constraints of maximum shear stress, equivalent stress and deflection of the bracket under boundary load condition. The problem to be dealt with for this work is that, the mounting bracket thickness should be optimized for its minimum weight and also be taken care that stress should be below yield stresses. In India with 24 hours of running vehicles with uneven and worst road conditions due to which there are always possibilities of being failure/fracture in the mounting bracket. Therefore, brackets with high strength cross section are needed to minimize the failures including factor of safety in design. Hence our project helps in identifying the strength of an engine mounting bracket under varying boundary condition and material sustainability, also vibrational model analysis for bad road conditions.

1.2. Objective:

1. To check the fatigue factors of mount bracket for engine carrying its load while moving condition and also to check its life, damage, Safety factors and biaxiality induction of nominal stress value.
2. To optimize the bracket using topology companion density based in order to reduce the usage of material in manufacturing it.
3. To make sure that the bracket sustains all the natural force evacuated while in vehicle moving condition.
4. To isolate the vibrations produced during damping on irregular road terrain.
5. To vary the weald bead size at the joints to sustain cyclic loading condition.

In this project work, I'm going to design the mounting bracket in such way that it should withstand for road load conditions. Both thickness, material, topology & vibrational (model analysis) are studied for the design. As the work deals with the design of mounting bracket, the Existing Design is analyzed and the results will be taken as the reference for the present work. The main problem is that, the mounting bracket thickness should be optimized for its minimum weight and also be taken care that stress should be below yield stresses. The main aim of this work is to create 3d model of the existing bracket as per 2d measurements & ANSYS is performed on this model to observe stresses developed. To reduce the failure testing is mandatory. In order to reduce the testing cost software testing using Analysis of system is efficient. In this project I'm going to design a 3d model using CATIA v5 software, by making a market survey for materials & its strength contribution in order to sustain the structural, vibrational, fatigue & redesign topology boundary conditions applied from all the direction.

After creating CAD model I'll convert the file to ANSYS format & perform static structural analysis & evaluate the following factors.

- Total deformation/Directional deformation.
- Von-misses stress.
- Strain.
- Fatigue factors like,
- Life, safety factor, damage & Biaxiality indication.

Vibrational model analysis will be performed after structural to know the behavior of nature against the applied boundary condition.

If in case the designed model fails to achieve the applied boundaries, then redesign topology is followed up to enhance the model from its weight & cost reduction purpose.

1.3. Scope:

Automobile sector is one of the largest branches of mechanical engineering industry. It consumes a lot of fuel while transporting goods and people from one place to other by road. Reducing automobile weight for better economy is the big challenge industry faces right now. Our Project work is focused on design and weight optimization of HCV truck's front Cabin mounting bracket. Study will be focused on finding alternative design or material for mounting bracket of the automobile.

So main scope for the project is:

- To obtain more strength from the bracket after optimization by means of design.
- To maintain the cost and better safety design.
- To obtain design stability with superior material performance.

LITERATURE REVIEWS

[1] Saket Kanunje, Kashinath H. Munde

One of the main causes of vibration that produce by a car is engine and transmission. Leaf spring mounting bracket are used to hold the leaf spring firmly with chassis. From last so many years Automobile industry continuously growing with different idea & technology for change in existing parts of vehicles. The most effective way of increasing automobiles mileage while decreasing emission is to reduce vehicle weight. Existing bracket has scope of mass optimization in current design .finite element analysis of bracket will be done using hyper mesh and ansys. Optistruct software will be used for topology optimization. Experimental stress of bracket will be done using strain gauge and applying corresponding loading through UTM. Validation for strain vector from FEA, Experimental results& Fatigue life Analysis.

[2] Naga Sai, Krishna Rohit, R Seetharaman

Mounting Tray is a interface between the unit and the aircraft structure (i.e. Avionics bay/racking system). It is usually made of thin aluminum alloy sheet, reinforced with stainless steel brackets. The structure is assembled with riveted joints in two stages i.e. sub assembly and final assembly. The main purpose of the mounting tray is to install the LRU in Aircraft, reduce and control the vibration and shocks that are transmitted to the Avionics equipment. It is designed with or without isolators, depending on vibration and shock levels. The sufficiency of the fixity of mounting tray will also be studied from vibration point of view in the following analysis. In this project the Design and Strength of vibration isolation of isolator mount avionics mounting tray has been investigated. Much effort has been focused on determining the mode shapes, Sway Space and damping produced with respect to the vibration mount positions. Determine no of isolator optimum location for isolators for a given load for Sine on Random vibrations inputs. Determined the sway space required and von-mises stresses at load points and desired locations to find out the factor of safety

[3] P. Meghana , Y. Vijayakumar, Dr P. Ravinder Reddy, P.Seema Rani

In an automobile industry while designing the components, the most critical aspect considered is the compactness and the weight of the component. The mounting brackets are meant for supporting the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passenger's weight. In the initial stage the bracket is designed according to the specifications of the mountings without considering any other factors. Analysis is performed for Existing and New modified designs. The design structure is optimized for its topology and topography. In the present work an attempt has been made to produce optimized design of a mounting bracket. The modeling is carried out in CATIA and meshing with quality is ensured through Hyper Mesh. The analysis is carried out using ANSYS by the objective function as shape and topology and the weighting function as weight, and constraints are on deflection and stresses induced.

[4] F.C. Campbell

DURING THE INDUSTRIAL REVOLUTION of the 18th and 19th centuries, the basic materials of construction were sparse. There was wood, stone, brick and mortar, and iron and steel. However, the irons and steels of that day were vastly inferior to the ones used today. In 1709, Abraham Darby established a coke-fired blast furnace to produce cast iron, and by the end of the 18th century, cast iron began to replace wrought iron, because it was cheaper. The ensuing availability of inexpensive iron was one of the factors leading to the Industrial Revolution. In the late 1850s, Henry Bessemer invented a new steelmaking process, involving blowing air through molten pig iron, to produce mild steel. This made steel much more economical. With the advent of the 20th century, improved lightweight materials such as aluminum, magnesium, beryllium, titanium, titanium aluminides, engineering plastics, structural ceramics, and composites with polymer, metal, and ceramic matrices began to appear.

[5] P. Meghana Y. Vijayakumar

In an automobile industry while designing the components, the most critical aspect considered is the compactness and the weight of the component. The mounting brackets are meant for supporting the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passenger's weight. In the initial stage the bracket is designed according to the specifications of the mountings without considering any other factors. Analysis is performed for Existing and

New modified designs. The design structure is optimized for its topology and topography. In the present work an attempt has been made to produce optimized design of a mounting bracket. The modeling is carried out in CATIA and meshing with quality is ensured through Hyper Mesh. The analysis is carried out using ANSYS by the objective function as shape and topology and the weighting function as weight, and constraints are on deflection and stresses induced.

[6] Mr. Suraj N. Mahadik

Foot mounting bracket is the bracket used to mount the seating system in the vehicle. These brackets support to the seating system and hold the seat rigidly in place and also prevent it from shifting around. Keeping this in mind the current paper discusses static analysis of foot mounting bracket by using Hypermesh and Abaqus. Static analysis of foot mounting bracket was done for checking design of existing and modified bracket. The results were analyzed for stresses and displacements. From the design and analysis, the stress and displacement of modified bracket were 111.2 MPa and 4.928 mm which is less than existing design so the design is safe. After experimentation, results were validated with FEA results.

[7] P. Meghana, Y. Vijayakumar

In an automobile industry while designing the components, the most critical aspect considered is the compactness and the weight of the component. The mounting brackets are meant for supporting the structural component and electronic components such as batteries, seats, cabin, chassis, rear body and also it should support the external load such as passenger's weight. In the initial stage the bracket is designed according to the specifications of the mountings without considering any other factors. Analysis is performed for Existing and New modified designs. The design structure is optimized for its topology and topography. In the present work an attempt has been made to produce optimized design of a mounting bracket. The modeling is carried out in CATIA and meshing with quality is ensured through Hyper Mesh. The analysis is carried out using ANSYS by the objective function as shape and topology and the weighting function as weight, and constraints are on deflection and stresses induced.

[8] Ms.Suvarna M Shirsath

Increasing competition and innovation in automobile sector tends to modify the existing products or replace old products by new and advanced material products. A mounting system of vehicle is also an area where these innovations are carried out regularly. More efforts are taken in order to increase the comfort of user. Appropriate balance of comfort riding qualities and economy in manufacturing of mounting bracket becomes an obvious necessity. Mounting brackets are widely used as suspension system & & cabin mounting in automotive vehicles. In this project, mounting bracket design optimization will be performed by changing from conventional steel to composite material. Composite materials are highly used in several different fields like aerospace structure, marine, automobile, etc. Due to high strength they are widely used in the low weight applications and also as an alternate for metals to reduce the material cost. FEA analysis in Ansys will be carrying out by using ACP toll where actual composites layer modeling will be used to create fiber ply. Conventional steel, glass fiber material will be studied and among best will be recommended for future products. To validate FEA results, Compression testing of mounting bracket will be done on UTM machine. Static deflection of mounting bracket and will be compared with FEA results for validation. The developed program for mounting brackets may be useful for fast and reliable optimization of various parameters of mounting brackets in automotive vehicles. Consequently, there will be tremendous saving in material in mounting brackets manufacturing industries as a result of optimization

[9] Vijay Kalantre

Today automobile sector is one of the largest growing technological fields and which is continuously striving for weight reduction of vehicles as the today's major need of fuel economy and emission reduction demands it. To reduce weight engineers have either to search for better and better materials or to do the optimization. Out of various structural optimization techniques like size, shape & topology optimization, application of Topology Optimization (TO) in automotive structure design is overviewed in this paper and using Evolutionary Structural Optimization (ESO) method driver cabin mounting bracket of a heavy commercial vehicle is optimized here. With the objective of mass reduction and compliance minimization topology optimization is performed using Ansys tool. Various topologies were studied and compared for static structural, fatigue safety factor and finding out minimum natural frequency i.e. modal analysis and an optimized topology was obtained with predefined level of compromise in constraint parameter like strength, minimum natural frequency and fatigue safety factor. Ansys results of static structural and modal analysis will be validated experimentally. 8.164 % of mass reduction is obtained with little compromise in constraint parameter. Fatigue results can be experimentally validated using component level testing or onsite testing as future scope. Also this technique can be used for other automotive structural components to reduce the overall weight of the vehicle.

[10] Ankit Vyas

Today automobile sector is one of the largest growing technological fields and which is continuously striving for weight reduction of vehicles as the today's major need of fuel economy and emission reduction demands it. To reduce weight engineers have either to search for better and better materials or to do the optimization. Out of various structural optimization techniques like size, shape & topology optimization, application of Topology Optimization (TO) in automotive structure design is overviewed in this project and using Evolutionary Structural Optimization (ESO) method driver cabin mounting bracket of a heavy commercial vehicle is optimized here. With the objective of mass reduction and compliance minimization topology optimization is performed using CATIA tool.

RESEARCH METHODOLOGY

My study

Most of the bracket have been studied for their stress concentration factors and fatigue analysis for its span life under vibrational behavior. But in this project a engine mount with weald application is taken to study the fatigue behavior under impact loading in structural and vibrational condition by varying bracket weald size.

Fig 1.2 Buick 231 & 3.8L V6 Weld-In Engine Mount Kit (#713003)

After measuring the dimensions, we have created the 2D drawing of the silencer. It iscreated in the AutoCAD.

PROPOSED WORK

1st step Material Selection for working

Structural steel ASTM A36 grade

Physical Properties	Metric	Imperial
Density	7.85 g/cm^3	0.284 lb/in^3
Mechanical Properties	Metric	Imperial
Tensile Strength, Ultimate	400 - 550 MPa	58000 - 79800 psi
Tensile Strength, Yield	250 MPa	36300 psi
Elongation at Break (in 200 mm)	20.0 %	20.0 %
Elongation at Break (in 50 mm)	23.0 %	23.0 %
Modulus of Elasticity	200 GPa	29000 ksi
Bulk Modulus (typical for steel)	140 GPa	20300 ksi
Poissons Ratio	0.260	0.260
Shear Modulus	79.3 GPa	11500 ksi

Table1.1 material properties of ASTM A36 grade

It's Less Expensive and More Efficient. Choosing structural steel for your next project will help reduce its overall expense and enhance its value.

- It's Stronger.
- It's Sustainable.
- It's Modifiable.
- It's Predictable.
- It Looks Great.

Magnesium

Property	Minimum Value (S.I.)	Maximum Value (S.I.)	Units (S.I.)	Minimum Value (Imp.)	Maximum Value (Imp.)	Units (Imp.)
Density	1.73	1.75	Mg/m^3	108	109.249	lb/ft^3
Compressive Strength	65	100	MPa	9.42745	14.5038	ksi
Poisson's Ratio	0.28	0.295	NULL	0.28	0.295	NULL
Shear Modulus	16	18	GPa	2.3206	2.61068	10^6 psi
Tensile Strength	175	235	MPa	25.3816	34.0839	ksi
Young's Modulus	44	45.5	GPa	6.38166	6.59921	10^6 psi
Glass Temperature			K			°F

Fig 1.3 properties of mag

Aluminum 3003 grade

Properties	Metric	Imperial
Density	2.73 g/cm^3	0.0939 lb/in^3
Melting point	644°C	1190°F
Properties	Metric	Imperial
Tensile strength	130 MPa	18855 psi
Yield strength	125 MPa	18130 psi
Shear strength	83 MPa	12039 psi
Fatigue strength	55 MPa	7977 psi
Elastic modulus	70-80 GPa	10153-11603 ksi
Poisson's ratio	0.33	0.33
Elongation	10%	10%
Hardness	35	35

Table 1.2 properties of Alu 3003

Analytical approach

Ford 3.8 V6-90

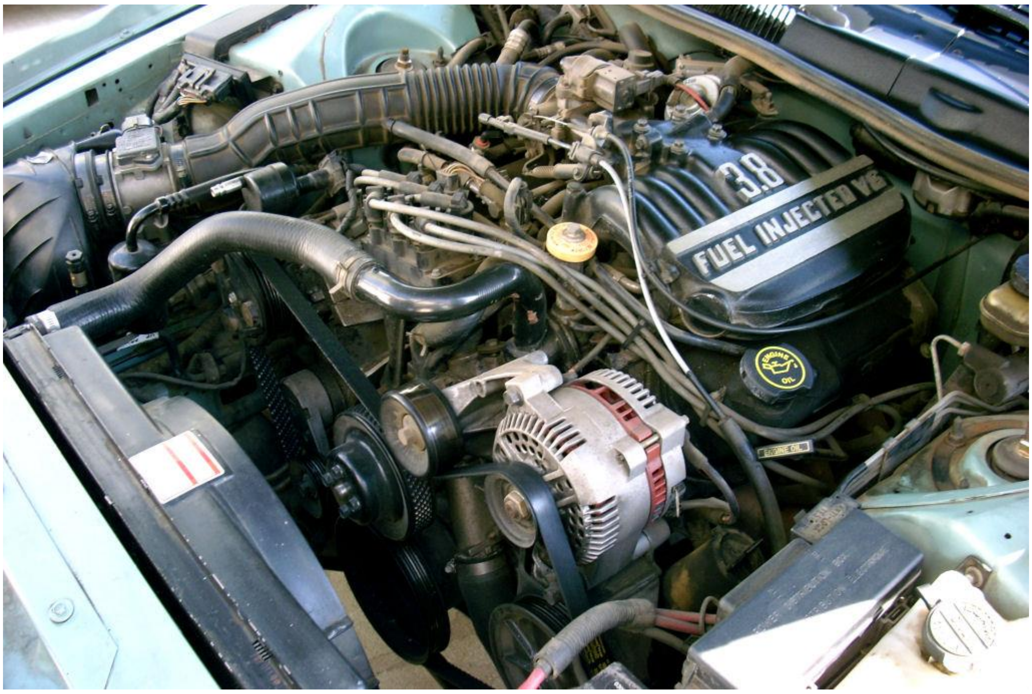

Figure Ford 3.8 V6-90 engine

The average weight of a 4-cylinder modern aluminum car engine is 200 to 350 pounds (90 to 160 kg), quite much lighter than a v6 or v8 engine. If it's a cast-iron block you can expect it to be slightly heavier than an aluminum engine.

Specification of the engine

Displacement 3,797 cc (231.7 cu in)

3,886 cc (237.1 cu in)

4,195 cc (256.0 cu in)

Cylinder bore 96.8 mm (3.81 in)

Piston stroke 86 mm (3.39 in)

88 mm (3.46 in)

95 mm (3.74 in)

Ford 3.8 V6-90 351 pound = 159.211 kg = 160 kg

\

Load Calculations:

Weight of the engine = 160 kg = 160*9.81 = 1569.5 N

No of bracket for engine = 2

Load on one bracket = 1570/2 = 785N

Designer factor of safety = 1.25

Final load to apply on the bracket = Fos*load on one bracket

= 1.25*785 = 981.25 N

= 1000N

Figure engine Mounting Bracket

Design - Catia

Procedure for 3D Development of model.

1. Take tracing of 2D drawing of any car engine mount bracket model with standard dimensions available, and download.
2. Open CATIA software, select sketch tracer from shape designing, select the downloaded 2D drawing and extract all the views on required plane, using create an immersive sketch.

3. Now the importation part is over, after importation trace the sketch using free style section using desired plane. (tracing involves creating of spline on a 2D drawing).
4. Create nodes and join nodes using curvature.
5. Extract the area to surface,
6. Add material
7. Offset thick surface to the required value.
8. Convert to IGS or STP file for ANSYS import.

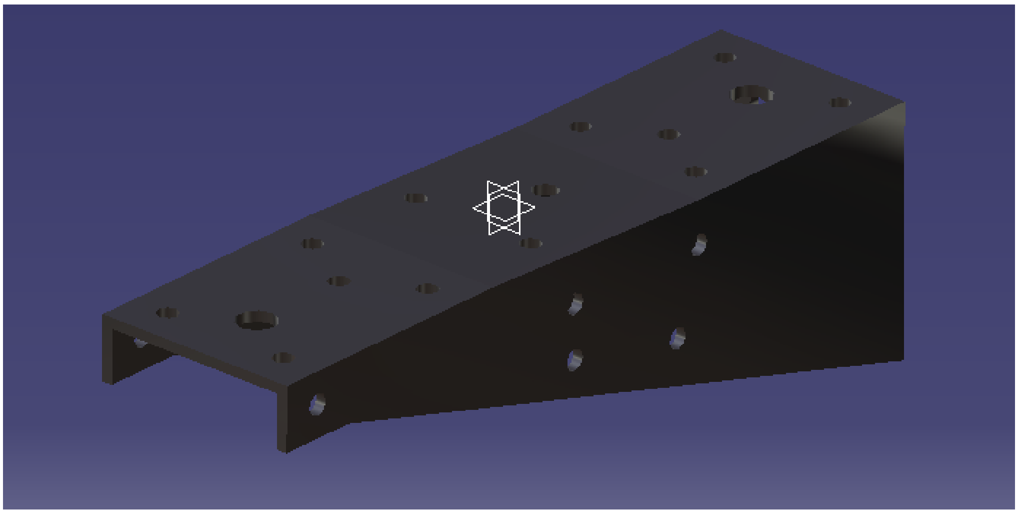

Figure Catia v5 engine Mount Bracket

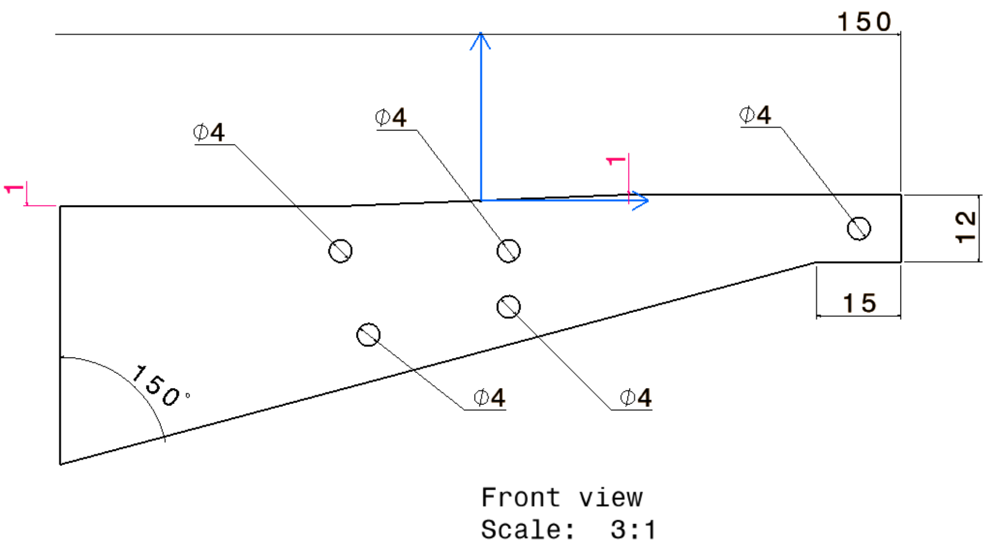

Figure Front view of bracket

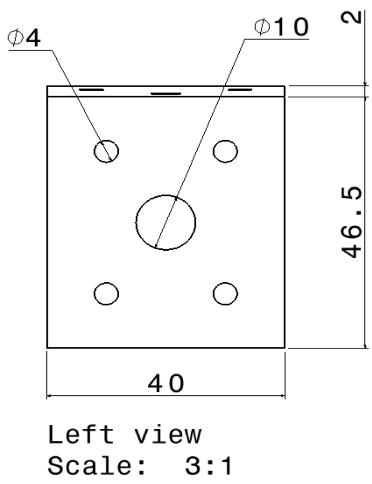

Figure Side view of bracket

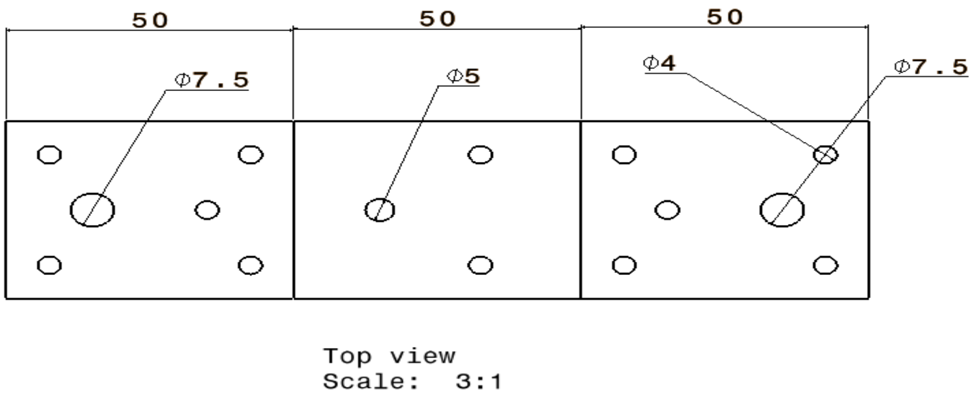

Figure Top view of bracket

Static Structural Analysis

The Basics of ANSYS Mechanical

Ansys Mechanical creates an integrated platform that uses finite element analysis (FEA) for structural analysis. Mechanical is a dynamic environment that has a complete range of analysis tools, from preparing geometry for analysis to connecting additional physics for even greater fidelity.

Learning Path

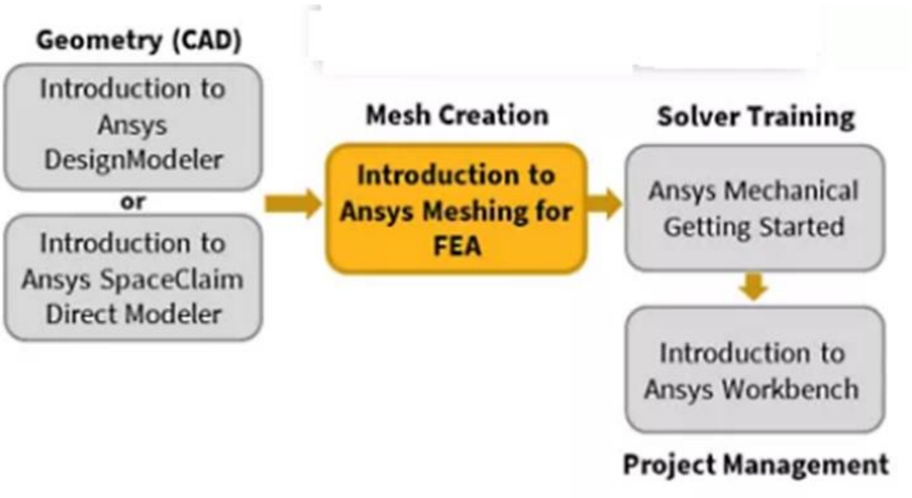

Figure 1. Learning path definition

Methodology for Static structural analysis

In this chapter, performing linear static structural analyses in Simulation will be covered:

A. Geometry and Elements
B. Assemblies and Contact Types
C. Analysis Settings
D. Environment, including Loads and Supports
E. Solving Models
F. Results and Postprocessing

Linear static structural analysis matrix equation

For a linear static structural analysis, the displacements {x} are solved for in the matrix equation below:

$$[K]\{x\} = \{F\}$$

Assumptions:

[K] is constant

- Linear elastic material behavior is assumed
- Small deflection theory is used
- Some nonlinear boundary conditions may be included
- {F} is statically applied

- No time-varying forces are considered
- No inertial effects (mass, damping) are included

Step Involved Static structural analysis

3D Modelling

- In structural analyses, all types of bodies supported by Simulation may be used.
- For surface bodies, thickness must be supplied in the "Details" view of the "Geometry" branch.
- A Point Mass can be added to a model (Geometry branch) to simulate parts of the structure not explicitly modeled: A point mass is associated with surface(s) only. The location can be defined by either:
- (x, y, z) coordinates in any user-defined Coordinate System.
- Selecting vertices/edges/surfaces to define location. Point mass is affected by "Acceleration," "Standard Earth Gravity," and "Rotational Velocity". No other loads affect a point mass. The mass is 'connected' to selected surfaces assuming no stiffness between them. No rotational inertial terms are present.

Material Selection

- Young's Modulus and Poisson's Ratio are required for linear static structural analyses:
  - Material input is handled in the "Engineering Data" application.
  - Mass density is required if any inertial loads are present.
  - Thermal expansion coefficient is required if a uniform temperature load is applied.
  - Thermal conductivity is NOT required for uniform temperature conditions.
  - Stress Limits are needed if a Stress Tool result is present.
  - Fatigue Properties are needed if Fatigue Tool result is present.
  - Requires Fatigue Module add-on license.

Analysis settings

- The "Analysis Settings" details provide general control over the solution process:

- Step Controls:
– Manual and auto time stepping controls.
– Specify the number of steps in an analysis and an end “time” for each step.
- “Time” is a tracking mechanism in static analyses (discussed later).
- Solver Controls:
- Two solvers available (default program chosen):
- Direct solver (Sparse solver in ANSYS).
- Iterative solver (PCG solver in ANSYS).

CONCLUSION

Until now I’ve surveyed lots of research paper form one of the papers, I have generated this idea of evaluating fatigue solution for mounting brackets. Basics like problem identification, objective & methodology have been concluded and also to system of work to prepare it for simulation in ANSYS has been decided as presented in the methodology. And study of materials from researcher gap has already been done. In next procedure I'm going to design a 3D model using CATIA v5 from the actual measurement of components to the required 4-wheeler. After designing, Simulation Using FEM technique is evaluated to enhance the component life. Finally, comparison will be done among the materials Best design is selected for its further requirement.

Plan of action

Sr. No.	Activity	AY 2021-22											
		July	Aug	Sept	Oct	Nov	Dec	Jan	Feb	Mar	Apr	May	June
1	Problem Identification												
2	Objective definition												
3	Research study												
4	Material selection process												
5	Parameter consideration												
6	Design development												
7	Structural analysis												
9	varying weald bead size study												
10	comparing the results												
11	Finally thesis writing												

REFERENCES

[1] Zhang Junhong, Han Jun "CAE process to simulate and optimise engine noise and vibration" Mechanical Systems and Signal Processing 20 (2006) 1400–1409.

[2] 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.

[3] Senthilnathan Subbiah, O.P. Singh, "Effect of muffler mounting bracket designs on durability", Engineering Failure Analysis 18 (2011) 1094–1107.

[4] Youngwoo Choia, Dohyun Jungb, Kyoungchun Hamc, "A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles", Procedia Engineering 10 (2011) 851–856.

[5] S.K. Loh a, W.M. Chin a, Waleed F. Faris, "Fatigueanalysis of Package Terminal Air Conditioner motor bracket under dynamic loading", Materials and Design 30 (2009) 3206–3216.

[6] S. Irving *, F.Ferguson-Smith, X.Z. Hu, Y. Liu, "Comparative fatigue assessment of soft toe and nested bracket welded aluminium structures", Engineering Failure Analysis 12 (2005) 679–690.

[7] Mehmet,OsmanH.Mete,UmitKocabicak,MuratOzso y,"Stamping process design using FEA in conjunction with orthogonal regression", Finite Elements in Analysis and Design 46 (2010) 992– 1000.

[8] Hong Suk Chang, "A Study on the Analysis Method for Optimizing Mounting Brackets," SAE International, 2006-01-1480.