Design, Analysis and Natural Frequency Optimization of Engine Mount Bracket

Tapan Shinde[1], Yash Sheth[2], Maanas Sindkar[3], Atharva Shinde[4], Sharon Silas[5]

[1,2,3,4,5]*Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, India*

Abstract - *Engine mounting systems in 4-wheelers are very important when vehicle performance is into consideration. Since the engine experiences a lot of vibration and noise, the transmitted vibration is also experienced by the engine mount bracket. The paper is a study of the engine mount bracket with respect to the noise, vibration and similar parameters. The designing and analysis of the bracket is done by ANSYS and SolidWorks software to attain various requirements as well as design considerations. Noise, Vibration and Harshness (NVH) are the principal parameters that causes discomfort from the engine. Vibrations can be reduced by maintaining the stresses under the determined conditions of safety which can be calculated by analysis and selection of the bracket material. The paper performs analysis on two engine bracket models under consideration. Each model is analyzed for four materials viz. Al Alloy, Mg Alloy, Gray CI and MS 1018. The results obtained are then used for comparative study.*

Key Words: Engine bracket, optimization, vibration, frequency, NVH, ANSYS

1. INTRODUCTION

Engine is an important component of a vehicle. Due to the vibration caused by various factors, the engine mount is supported by brackets. The engine mount bracket is the part that holds the engine where the engine and the bracket are bolted together by three or four mounts. Engine mounts are generally made up of rubber material so as to avoid direct metal to metal contact between the engine and the car body and to better absorb the vibrations from the engine. Hence it keeps the high frequency vibrations and noise arising from the engine away from the passengers. Engine mount brackets can deteriorate with time and due to stress. The elasticity of the rubber casing could be lost over time which develops cracks and eventually cause fluid to leak out. The failure of an engine mount can lead to serious engine damage and also greatly affects the vehicle performance. The failure can also lead to excessive vibrations and reduces the absorption ability. Some effects of the vibrations are high impact noises and engine movement.

The important vibration considerations in an engine are combustion force, bearing reaction forces like mass forces, damper function and flywheel whirling, piston side forces, belt movement, gear drive or gear train forces inside transmission and drive train reaction forces. According to the basic non- linear vibration theory, improvement in the vibration control can be achieved by determining the natural frequency of an engine bracket system. In order to obtain low transmissibility, the natural frequency of the mounting system, must be below the engine disturbance frequency of the engine idle speed to avoid excitation of the system resonance during normal driving conditions. If the brackets have a resonance frequency near the engine working frequency, it may result in high amplitude vibrations and may damage the chassis and bracket. If harmonic response values of the bracket are more than acceptable values, it results in loud noise. Both the cases of vibration and noise reduce the life of bracket, causes discomfort to the driver as well as passengers. Hence, modal analysis and harmonic analysis of the bracket is necessary. The vibration and noise can be reduces by shifting the suitable dimensions and material of the bracket. But the static and dynamic forces on bracket can increase the deflection and stresses. The paper considers standard parameters studied through available literature, considers two model designs available [6] [7] for use by rebuilding them in SolidWorks and finally compares the analysis on different materials. Various parameters listed ahead are considered the modal, harmonic and harmonic acoustics analyses are done using ANSYS while the design was rebuilt in SolidWorks.

2. LITERATURE REVIEW

In a paper titled “Finite element analysis and natural frequency Optimization of engine bracket (2013)” they have carried out NVH analysis and FEA analysis for stress and vibrational parameters comparison between real time and obtained value. They concluded that Gray Cast Iron is essentially a brittle material of the bracket. In terms of analysis, Al alloy and Mg alloy are showing almost same value of natural frequency and indicate that any one of them would be a better choice than Gray Cast Iron. However, in terms of FEA there is a caveat, which being that in Modal FEA, the effect of Damping is not considered. In Practical terms, Mg alloy exhibits better damping characteristics than Al alloy. Hence as far as the recommendation goes, Mg alloy will be preferred. In another paper titled “Finite Element Analysis and Natural Frequency Optimization of Engine Mounting Bracket”, they have optimised Manufacturing & Weight. Finite Element (FE) analysis of a typical engine bracket of a car and natural frequency is also determined. From results of finite element analysis it is observed that both steel and carbon glass fiber materials have stress values less than their respective permissible yield stress values. So the design is safe. From analysis results and comparison of properties of all the analysis, it is found that glass fibre is the material which is having the least density; also it is easily available. The natural frequencies at each mode found after modal analysis of the suggested model are more than existing model; hence the bracket is safe under the vibrations. In "Stamping process design using FEA in conjunction with orthogonal regression", Finite Elements in Analysis and Design (2010)" paper, a process design technique is presented for the formability assessment of sheet metal stamping parts and feasibility analysis of process conditions of which comparisons with the experimental data indicated the suitability of the proposed approach in spring back predictions. One of the papers included objectives like to analyse and optimize the Engine Mounting Bracket for different mesh size case and optimize by using different material grade and to determine the optimal and critical point having highest stress and optimized in terms of reducing weight and reducing stresses. After modelling in CATIA and completion of analysis they have obtained up to some extend for the elemental mesh size 4mm the corresponding stress slightly varies and it starts to change the behaviour after the mesh size 4mm from this point onwards as the mesh size goes on increasing the stress will decrease. Lastly, in a paper where modal analysis of an Engine Mounting bracket is done, they have optimized the weight and manufacturing, FE analysis and determining natural frequency of the car. Evaluation of modified engine mounting bracket was done using static structural analysis and modal analysis. It was found that, for the modified design deformation was 0.4950 mm with equivalent von-Mises stress 164.87 MPa which was very less than initial design with 1.14 mm deformation and equivalent von-Mises stress 189.11 MPa. Further natural frequency of modified design was found to be 257.83 Hz which was well within the range below self-excitation frequency and less than the natural frequency (268.59 Hz) of initial design. It was found that aluminium bracket limit its use for the said application due to greater deformation and less stiffness. Magnesium bracket can be the option to ERW-1 steel for the Engine supporting bracket application but it cannot be deployed as it is highly susceptible to corrosion. From the results obtained for initial design and modified design, it can be concluded that ERW-1 material best suit the requirement of the desired application and can be deployed with some safety standards.

3. METHODOLOGY

3.1 Modeling

Initially, from the literature review we decided to consider the available models so as the study is application based. This design taken was rebuilt in SolidWorks, followed by static structural analysis, Modal analysis, NVH analysis on ANSYS and comparison of the materials. The validation was from the literature surveys and that by comparing to the permissible range where the ANSYS analysis followed the permissible parameters with the specific material used. Hence, comparison of the models in the survey enabled us to get common parameters.

Firstly theoretical study of the Engine mounting bracket was done. This model was taken from an open-source site.[6] The first engine mounting bracket used is from PEUGOT 206 GTI 180 Engine Mount Bracket car which is a common running car used in the UK. The car is a front-wheel drive front-engine small hatchback with 3 doors & 5 seats. With 175 BHP, the naturally aspirated 2 Liter 16V Inline 4 petrol engine accelerates this 206 GTI 180 RC to 62 mph in 7.4 seconds and on to a maximum speed of 137 mph. Having a kerb weight of 1125 kg, it achieves 32.8 mpg on average and can travel 360 miles before requiring a refill of its 50 liter capacity fuel tank. It comes with a 5 speed manual gearbox. Luggage space is 245 liters, but can be expanded to 1130 liters for larger loads, with a maximum payload capacity of 435 kg. Some specifications of the car are given are as follows:

Performance data:

Top Speed – 137mph

Fuel economy – 32.8mpg

Fuel Range – 360 miles

Power to Weight Ratio – 156 BHP/Tonne

Petrol Engine Specs:

Front engine, Transversal, 50L

Displacement – 2L

Peak Power – 175 BHP @ 7000rpm

Peak Torque – 202NM @ 4750rpm

Other dimensions:

Compression ratio – 11:1

Bore × Stroke – 85mm × 88mm

Power/Cylinder – 44BHP

Unitary Capacity – 499cc

BMEP – 12.7 bar

Drivetrain and Chassis Specs:

Gearbox – 5 Gears

Tires – 205/40R17 Z

Front Suspension – MacPherson Strut

Front Brakes – Vented Disc

Rear Suspension – Trailing Arm

Rear Brakes – Solid Disc

Figure - 1[8] - PEUGEOT 206 GTI 180

Figure - 2: PEUGEOT 206 GTI 180 Engine Mount Bracket

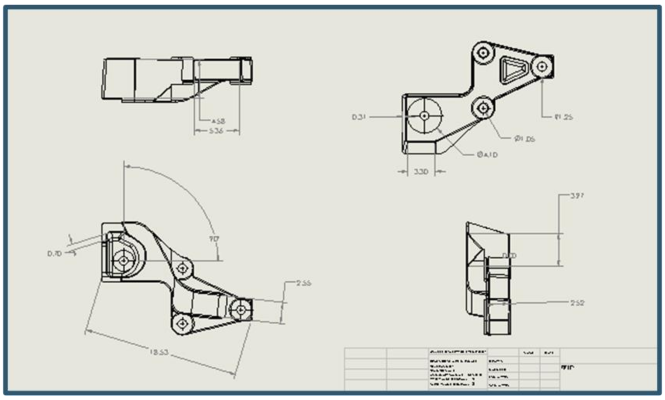

Figure- 3: PEUGEOT 206 GTI 180 Engine Mount Bracket Dimensions Sheet

The second engine bracket was taken from and was developed from the drawing and model found on an open source site [7]

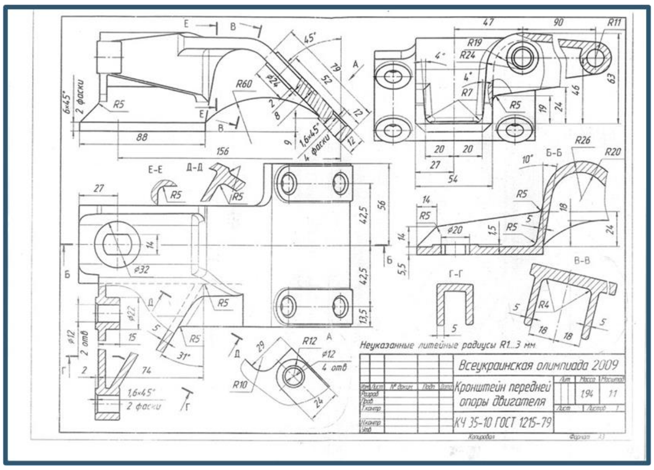

Figure - 4[7]: Dimensions Sheet Engine Mount Bracket

SolidWorks models were made on SolidWorks 2018. All the parts have used their respective profiles which is specified by the conventional shapes and owing to the dimensional changes made. In SolidWorks with the dimensional parameters of the tools, they are modeled by using the SolidWorks software. Key points are created along the profile in the working plane. The points are joined by drawing curves to obtain a smooth contour. The contour (2D model) is then converted into area and then volume (3D model). This was generated by extrusion. Multiple volumes of different parts are generated similarly. These volumes are then combined into single volume if the part assembly is to be done.

3.1.1 Model 1- PEUGEOT 206 GTI 180 Engine Mount Bracket

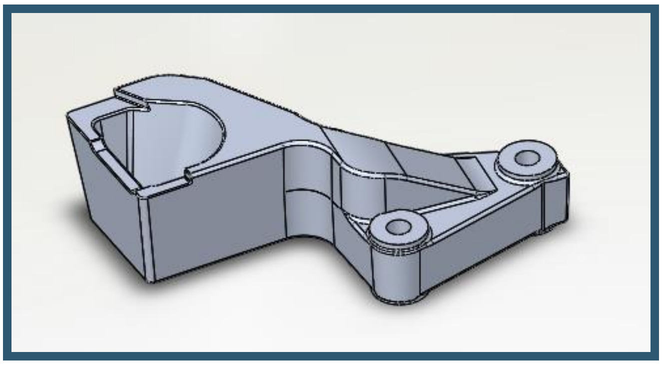

Figure - 5: Isometric View of Model 1

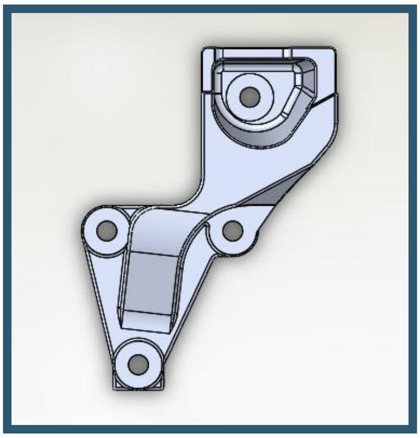

Figure - 6: Top View of Model 1

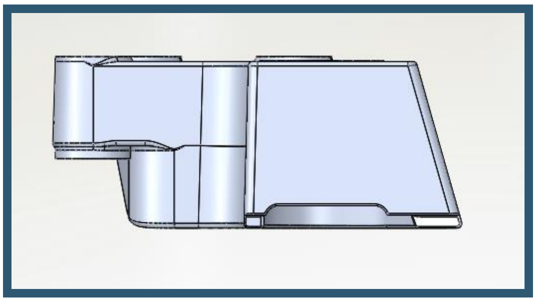

Figure - 7: Side View of Model 1

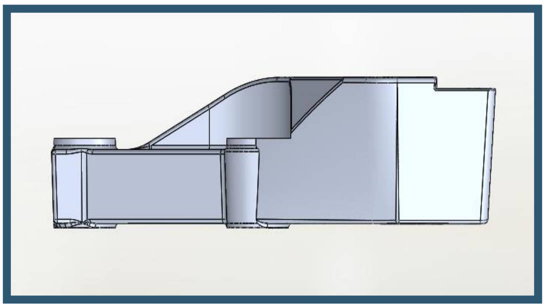

Figure - 8: *Side View of Model 1*

3.1.2 Model 2

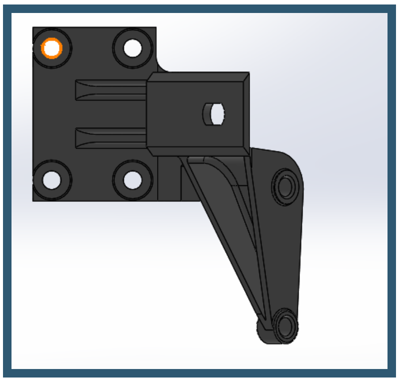

Figure - 9: Bottom View of Model 2

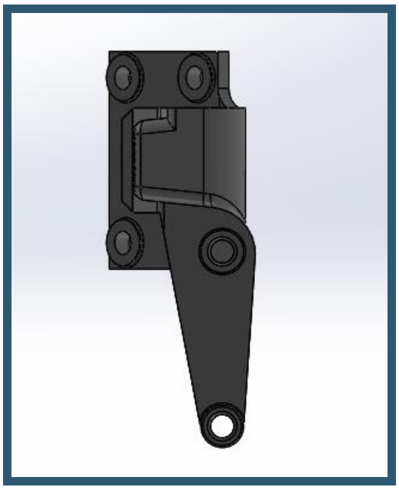

Figure - 10: Side View of Model 2

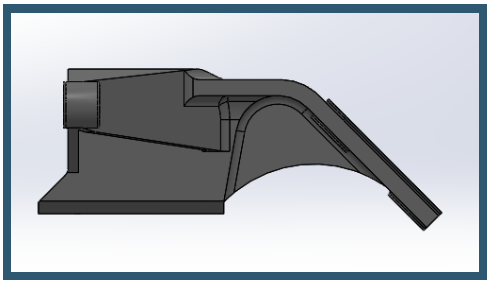

Figure - 11: Side View of Model 2

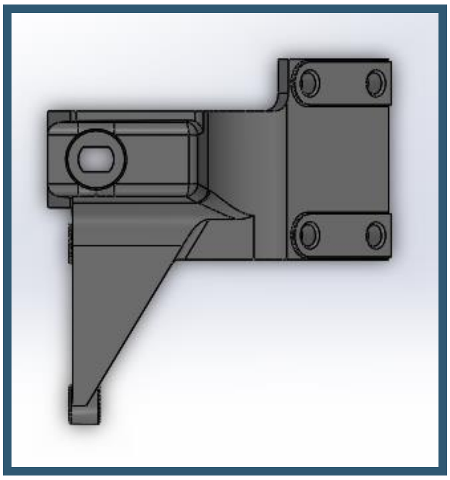

Figure - 12: Top View of Model 2

4. Analysis and Results

4.1 Analysis

4.1.1 Static Structural Analysis

The Analysis is done using ANSYS 2020: The software uses the Finite Element Method (FEM). FEM is a numerical technique for analysing engineering designs. FEM is accepted as the standard analysis method due to its generality and suitability for computer implementation. FEM divides the model into many small pieces of simple shapes called elements effectively replacing a complex problem by many simple problems that need to be solved simultaneously.

Static Structural Analysis was done for the necessary parts which contribute to the major strength and can be considered as structural parts of the system. This uses Finite Element Analysis by creating proper meshing. We have evaluated the total deformation and stress concentration in the given parts. The static structural analysis works as- When loads are applied to a body, the body deforms and the effect of loads is transmitted throughout the body. The external loads induce internal forces and reactions to render the body into a state of equilibrium. Linear Static analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads.

Static Structural analysis is used to determine the stress concentration and total deformation. The finite element analysis for steady state thermal analysis and static structural analysis of all the parts is carried out by using ANSYS 2020 software.

Table - 1: Details obtained During Analysis of Model 1

NAME OF THE PROPERTY (Model 1)	DETAILS
Physical Preference	Mechanical
Size Function	Adaptive
Relevance Center	Fine
Nodes	45157
Elements	28339
Smoothing	Fine
Refinement	3
Element Size	2 mm
Convergence	2%

Table - 2: Details obtained during Analysis of Model 2

NAME OF THE PROPERTY (Model 2)	DETAILS
Physical Preference	Mechanical
Size Function	Adaptive
Relevance Center	Fine
Nodes	23962
Elements	14231
Smoothing	Fine
Refinement	3
Element Size	2 mm
Convergence	2%

4.1.1.1 Al alloy

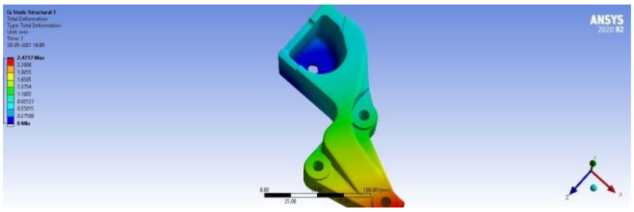

Figure - 13: Total deformation of Model 1

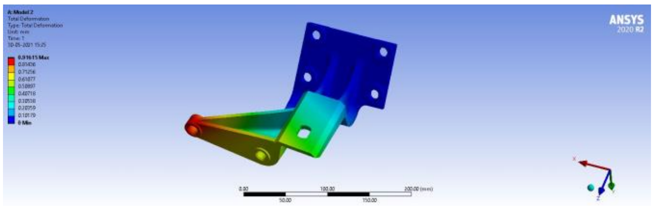

Figure - 14: Total Deformation of Model 2

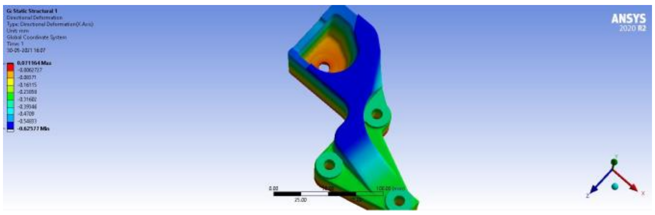

Figure - 15: Directional Deformation of Model 1

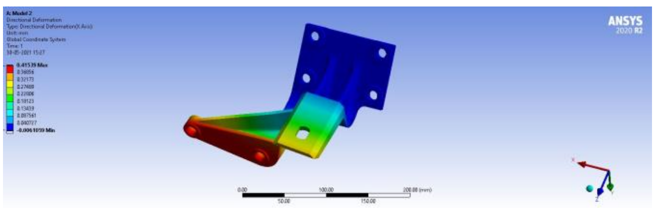

Figure- 16: Directional Deformation of Model 2

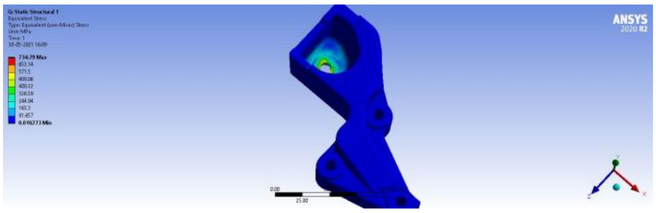

Figure - 17: Equivalent Stress for Model 1

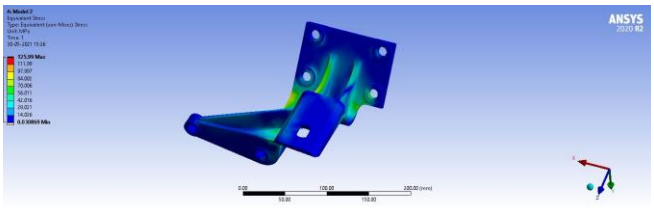

Figure - 18: Equivalent Stress for Model 2

4.1.1.2 Mg alloy

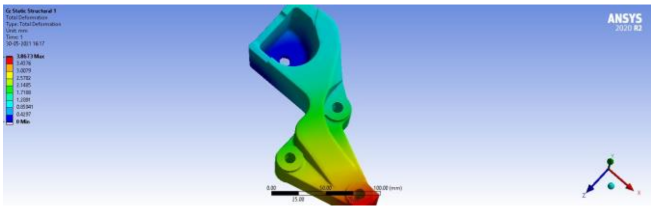

Figure - 19: Total deformation of Model 1

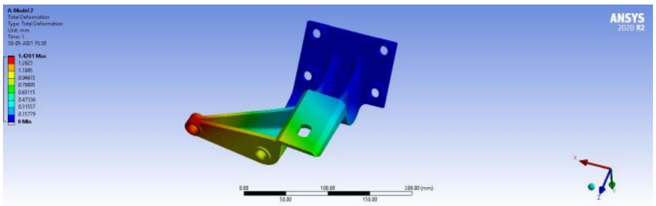

Figure - 20: Total Deformation of Model 2

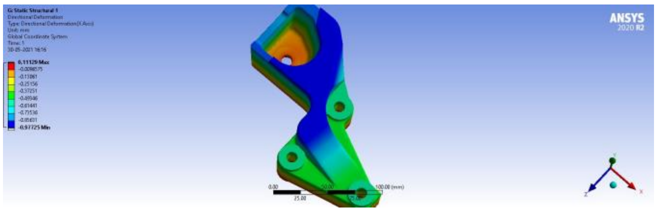

Figure - 21: Directional Deformation of Model 1

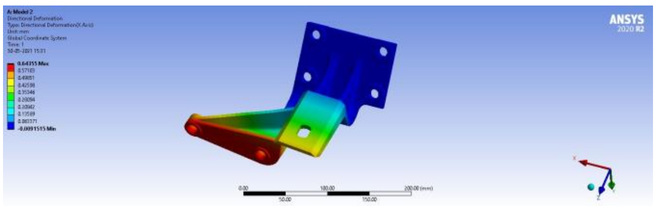

Figure - 22: Directional Deformation of Model 2

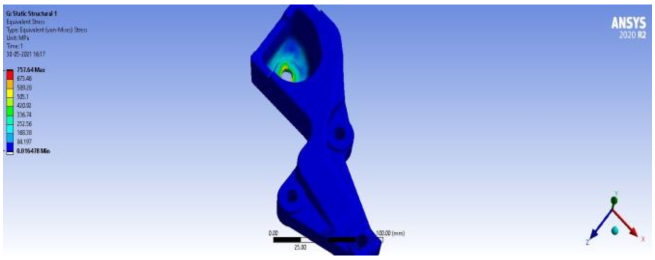

Figure - 23: Equivalent Stress for Model 1

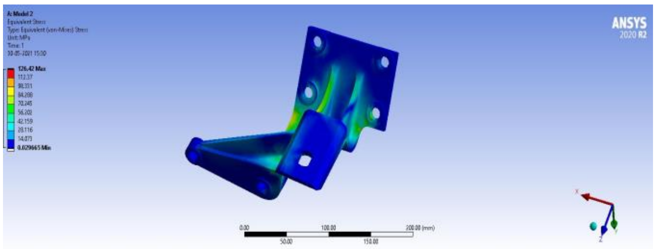

Figure - 24: Equivalent Stress for Model 2

4.1.1.3 Gray CI

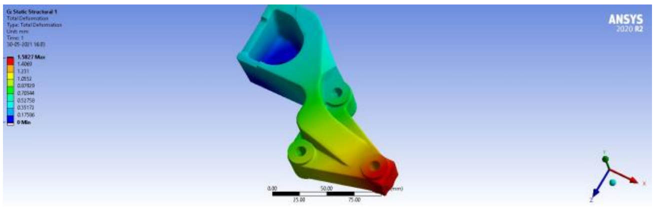

Figure - 25: Total deformation of Model 1

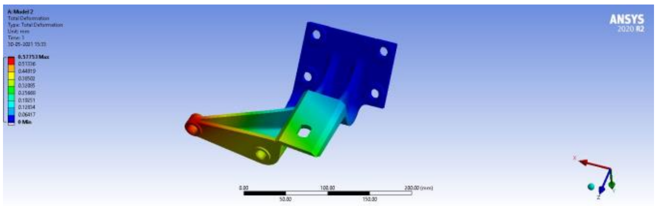

Figure - 26: Total Deformation of Model 2

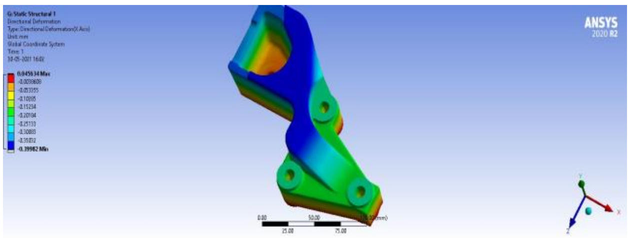

Figure - 27: Directional Deformation of Model 1

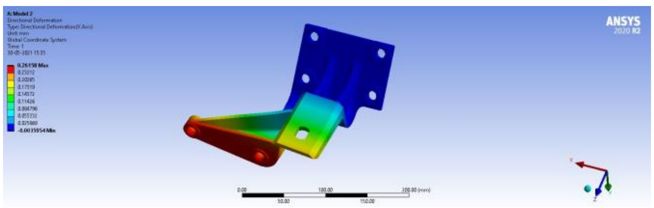

Figure - 28: Directional Deformation of Model 2

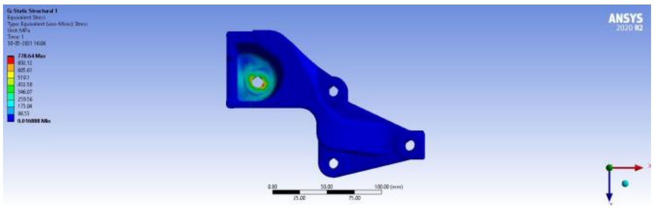

Figure - 29: Equivalent Stress for Model 1

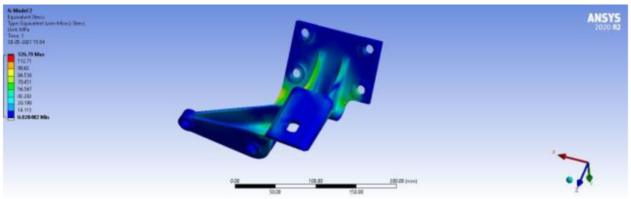

Figure - 30: Equivalent Stress for Model 2

4.1.1.4 MS 1018

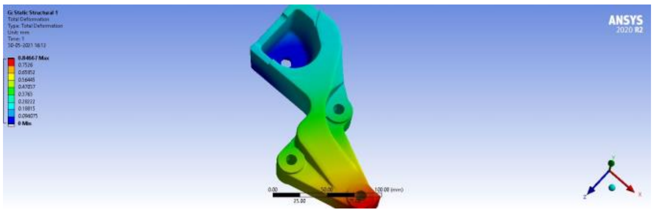

Figure - 31: Total deformation of Model 1

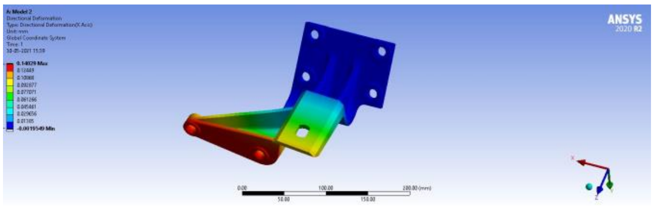

Figure - 32: Total Deformation of Model 2

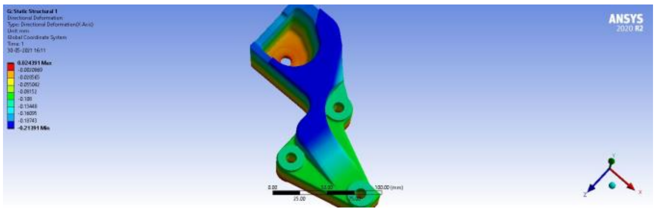

Figure - 33: Directional Deformation of Model 1

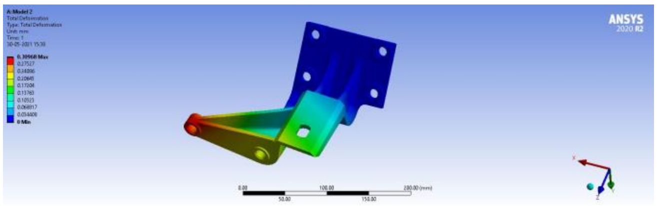

Figure - 34: Directional Deformation of Model 2

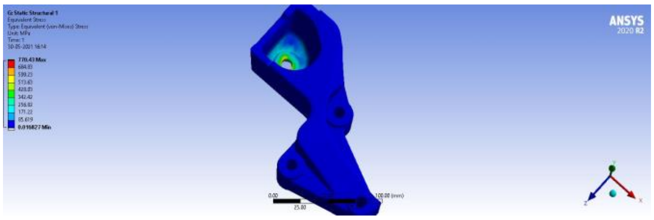

Figure - 35: Equivalent Stress for Model 1

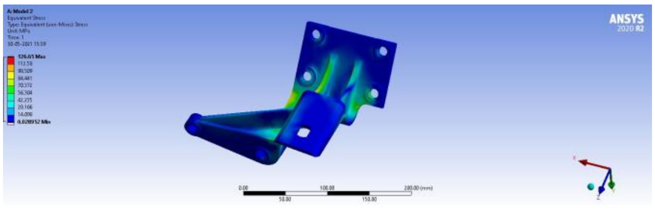

Figure 36: Equivalent Stress for Model 2

4.1.2 NVH Analysis

We have done the NVH Analysis on ANSYS 2020 R2. Noise, vibration, and harshness (NVH), also known as noise and vibration, is the study and modification of the noise and vibration characteristics of vehicles, particularly cars and trucks. Our complete NVH testing and NVH analysis solutions are ideal for optimizing noise and vibration of the vehicles, such as reduction, design, and quality assurance of the interior and exterior noise. We have done the Harmonic acoustics analysis and Modal analysis which is the study of the dynamic properties of systems in the frequency domain. The ANSYS harmonic acoustics toolbox utilizes far-field solutions for evaluating the sound pressure level on an infinite domain. After designating the outer bounds of the air domain as a radiation boundary, the far-field is set as a sphere of radius 1000mm centred at the origin of the geometry. The results of the sound pressure level throughout the far-field domain are plotted against the phi and theta angles of the sphere. Additionally, a far-field sound pressure level microphone is placed at a distance of 1000mm from the centre of the sphere, which records the sound pressure level at the given distance. The microphone tool is useful for evaluating the sound pressure level at a specific point.

4.1.2.1 Al alloy

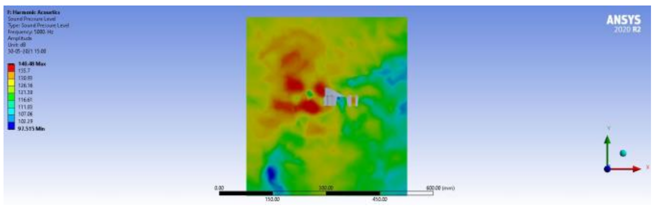

Figure - 37: Sound Pressure Level of Model 1

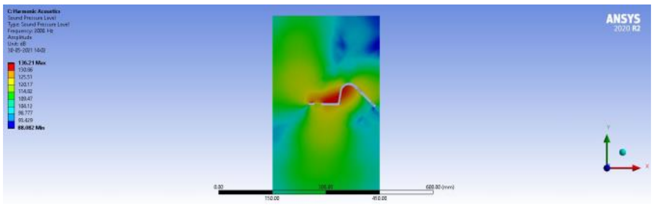

Figure - 38: Sound Pressure Level of Model 2

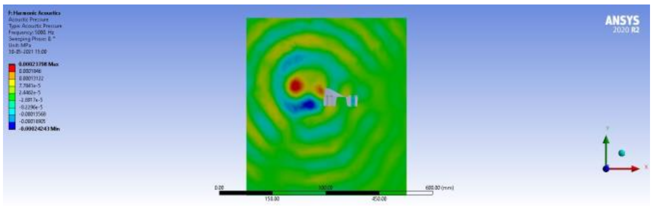

Figure - 39: Acoustic pressure of Model 1

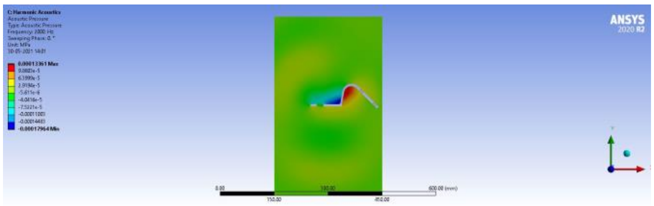

Figure - 40: Acoustic pressure of Model 2

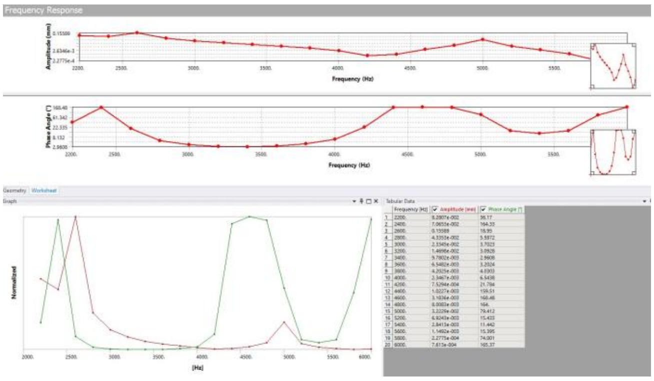

Figure - 41: Frequency Distribution of Model 1

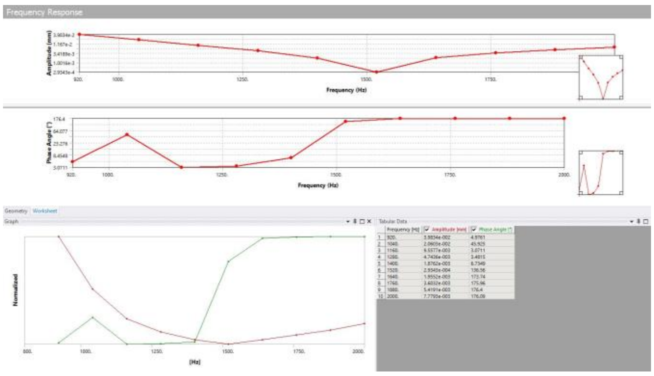

Figure - 42: Frequency Distribution of Model 2

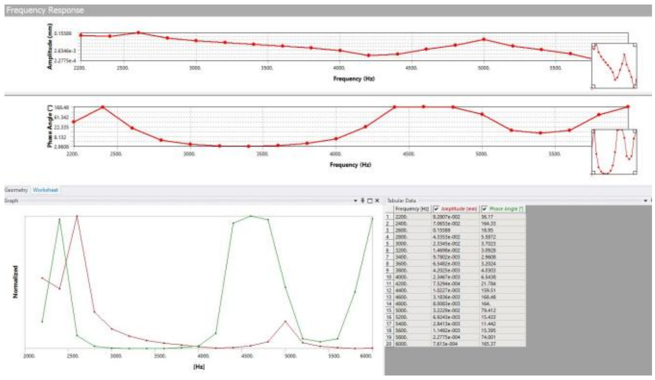

Figure - 43: Frequency Response of Model 1

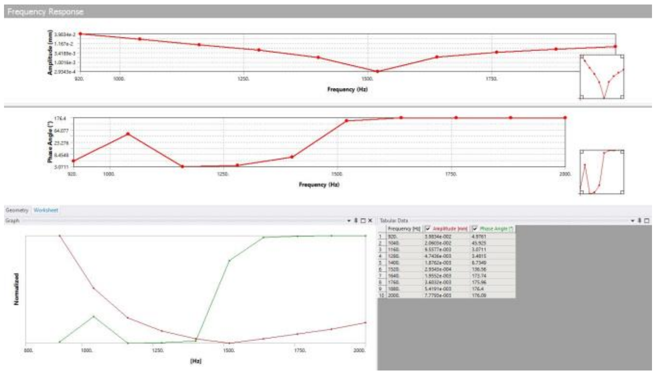

Figure - 44: Frequency Response of Model 2

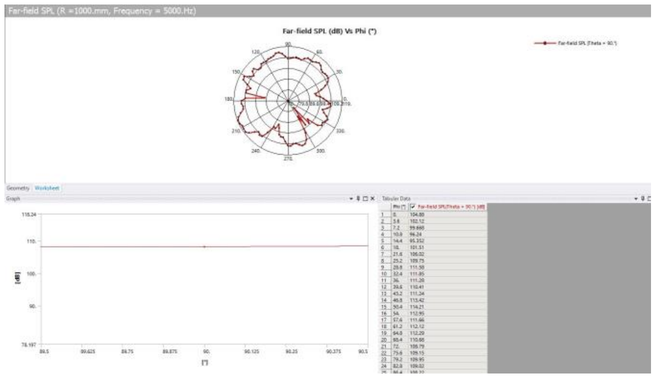

Figure - 45: Far Field SPL vs. Phi for Model 1

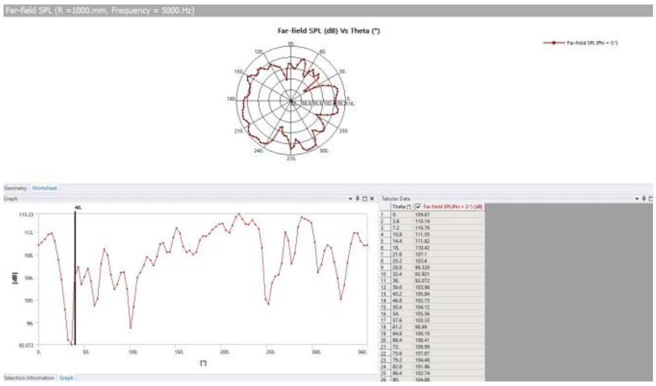

Figure - 46: Far Field SPL vs. Theta for Model 1

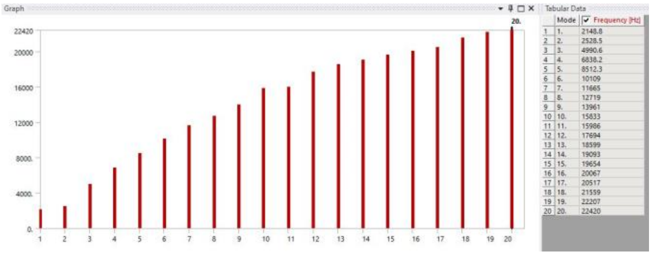

Figure - 47: Mode vs. Frequency for Model 1

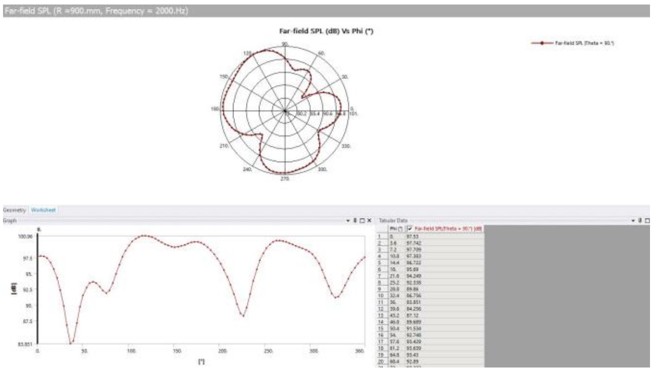

Figure - 48: Far Field SPL vs. Phi for Model 2

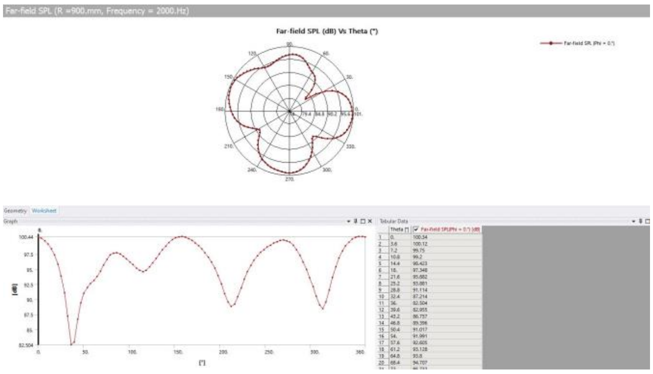

Figure -49: Far Field SPL vs. Theta for Model 2

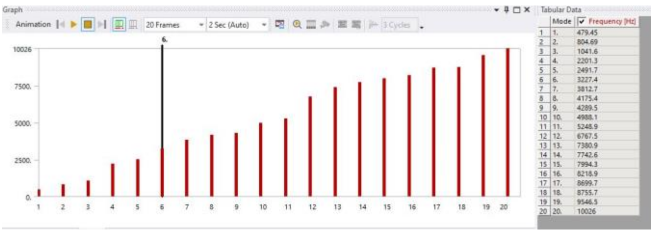

Figure - 50: Mode vs. Frequency for Model 2

4.1.2.2 Mg alloy

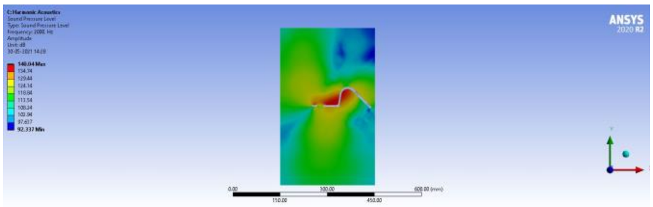

Figure - 51: Sound Pressure Level of Model 1

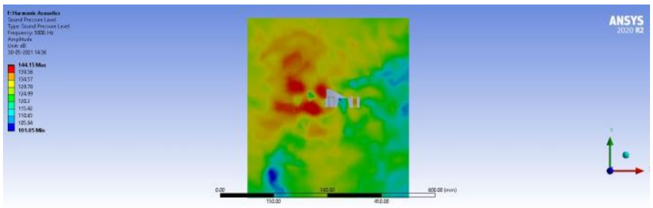

Figure - 52: Sound Pressure Level of Model 2

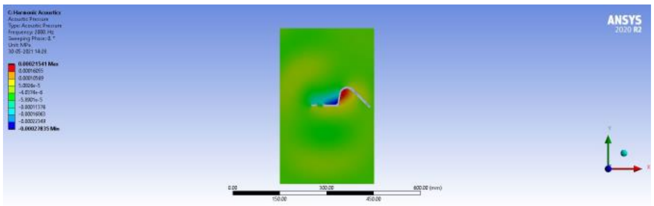

Figure - 53: Acoustic Pressure of Model 1

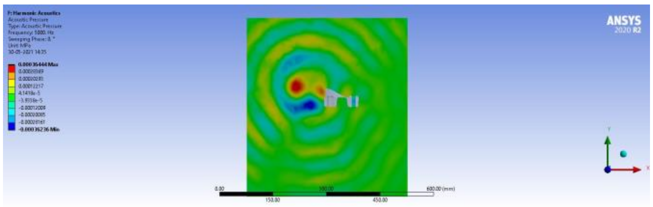

Figure - 54: Acoustic Pressure of Model 2

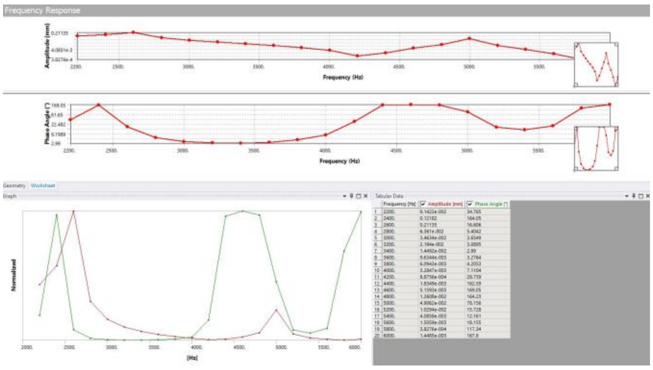

Figure - 55: Frequency Distribution of Model 1

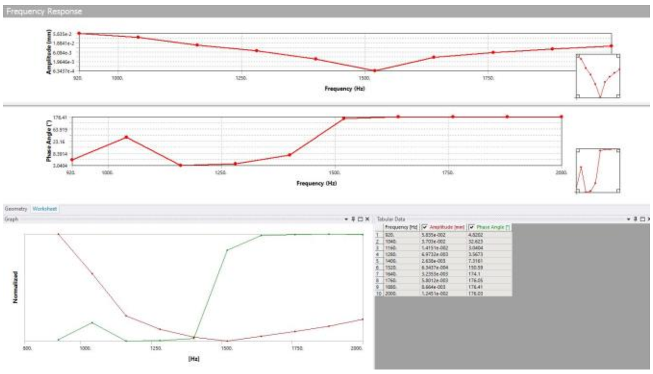

Figure - 56: Frequency Distribution of Model 2

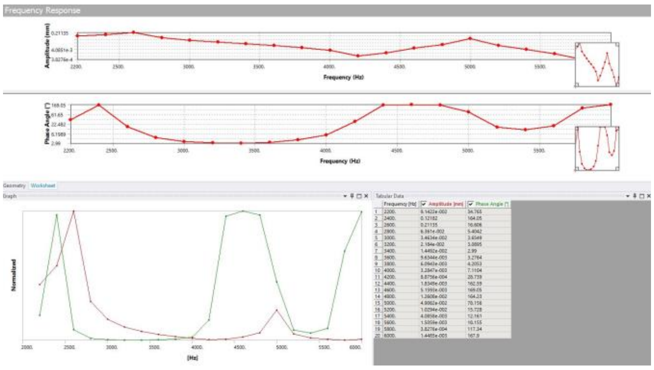

Figure - 57: Frequency Response of Model 1

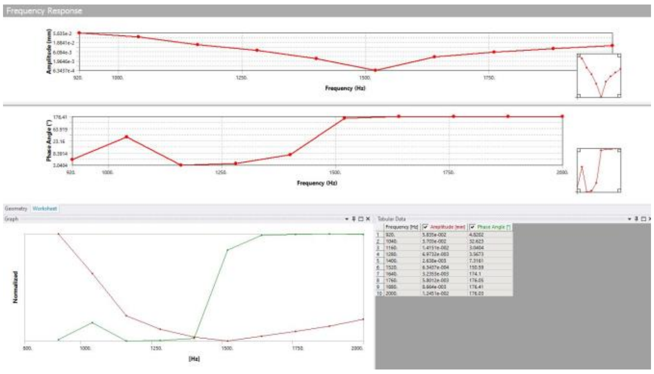

Figure - 58: Frequency Response of Model 2

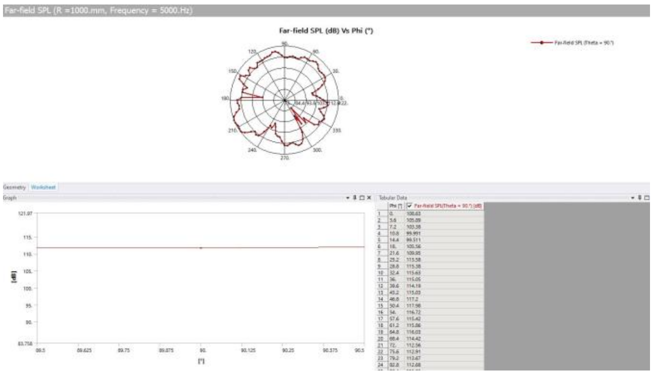

Figure - 59: Far Field SPL vs. Phi for Model 1

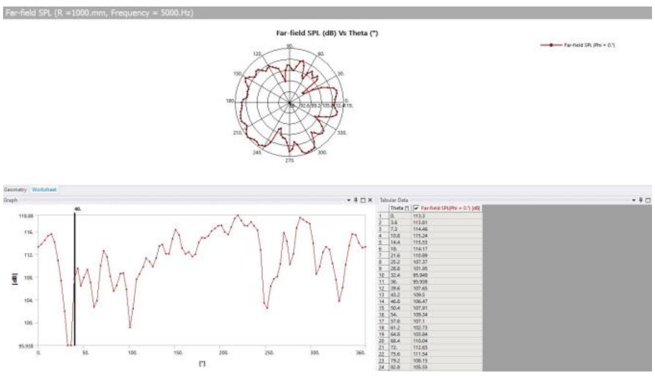

Figure - 60: Far Field SPL vs. Theta for Model 1

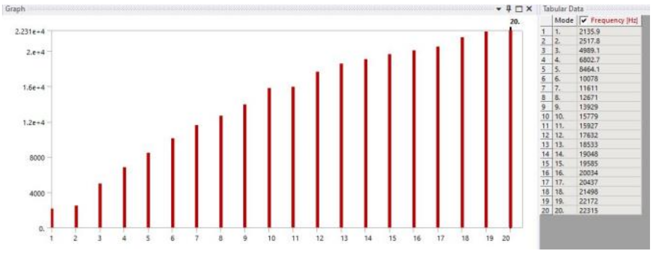

Figure - 61: Mode vs. Frequency for Model 1

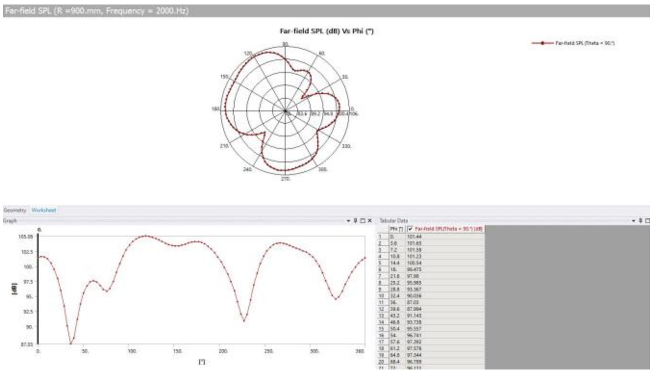

Figure - 62: Far Field SPL vs. Phi for Model 2

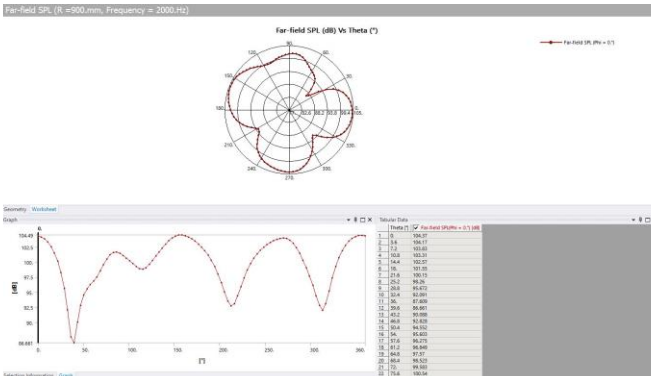

Figure - 63: Far Field SPL vs. Theta for Model 2

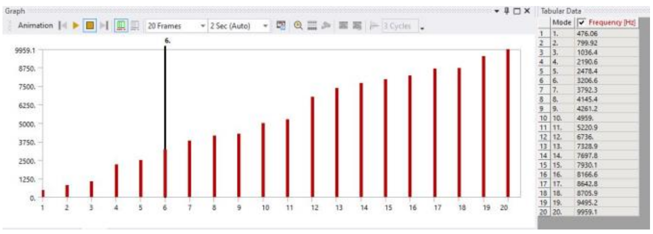

Figure - 64: Mode vs. Frequency for Model 2

4.1.2.3 MS1018

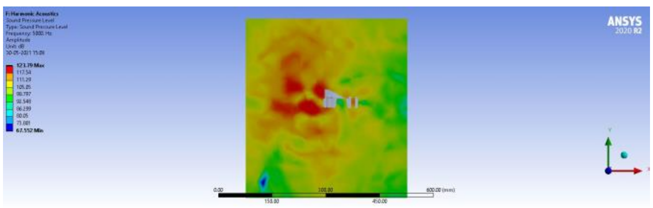

Figure - 65: SPL of Model 1

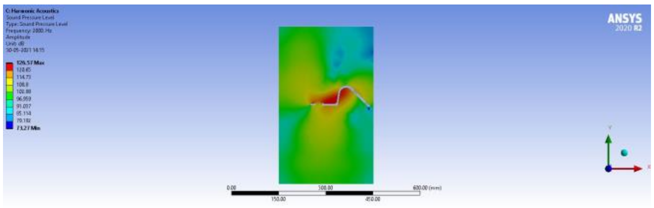

Figure- 66: SPL of Model 2

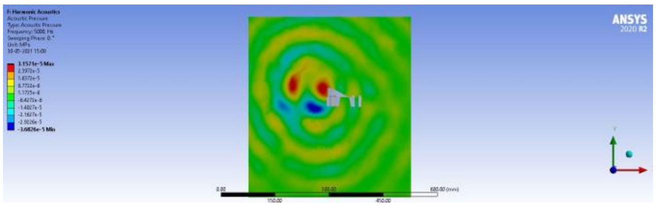

Figure - 67: Acoustic Pressure of Model 1

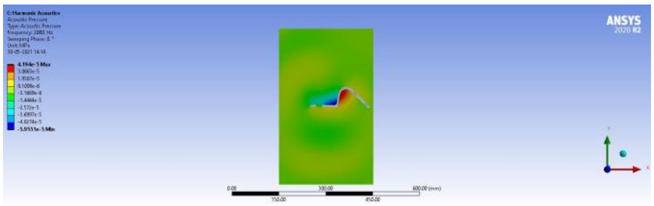

Figure - 68: Acoustic Pressure of Model 2

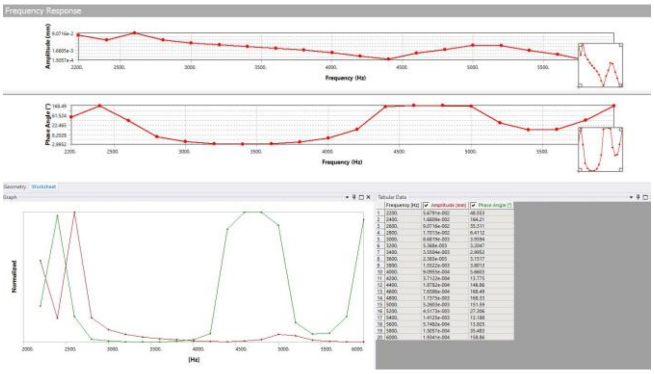

Figure - 69: Frequency Distribution of Model 1

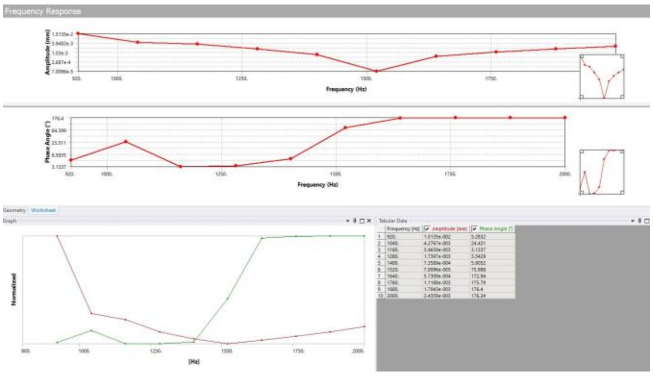

Figure- 70: Frequency Distribution of Model 2

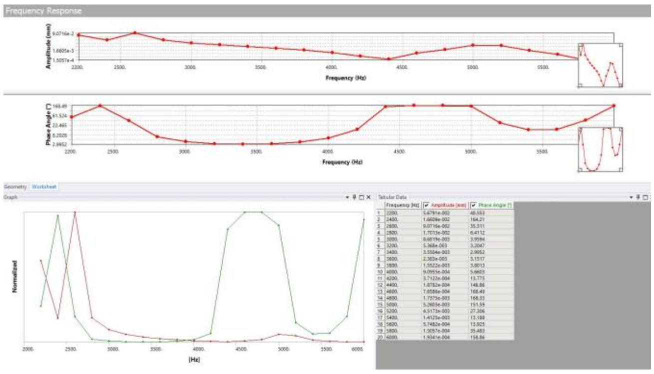

Figure - 71: Frequency Response of Model 1

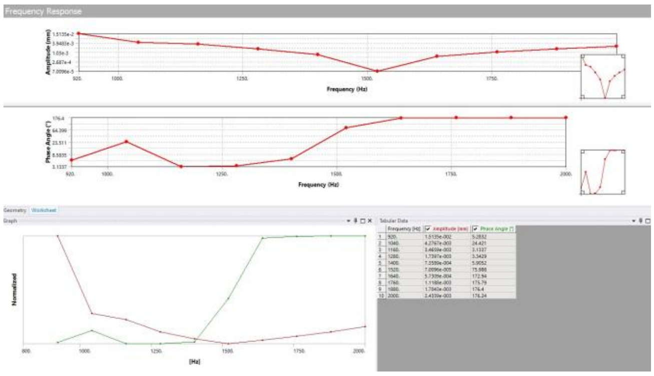

Figure - 72: Frequency Response of Model 2

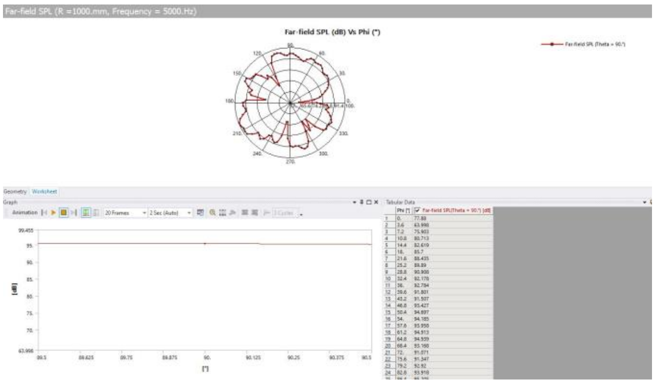

Figure - 73: Far Field SPL vs. Phi for Model 1

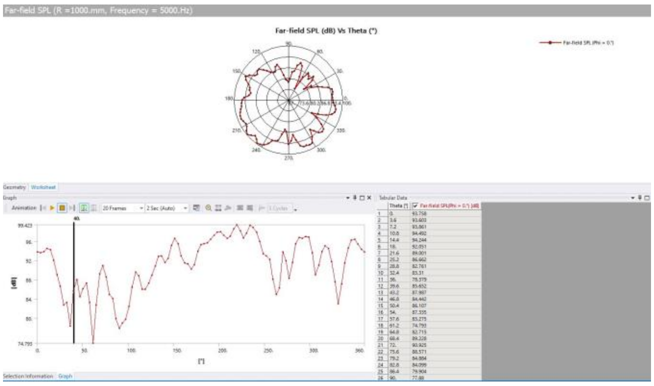

Figure - 74: Far Field SPL vs. Theta for Model 1

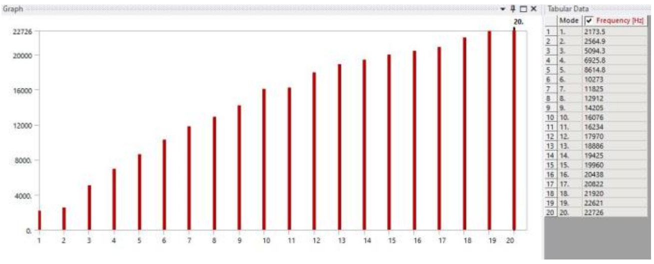

Figure - 75: Mode vs. Frequency for Model 1

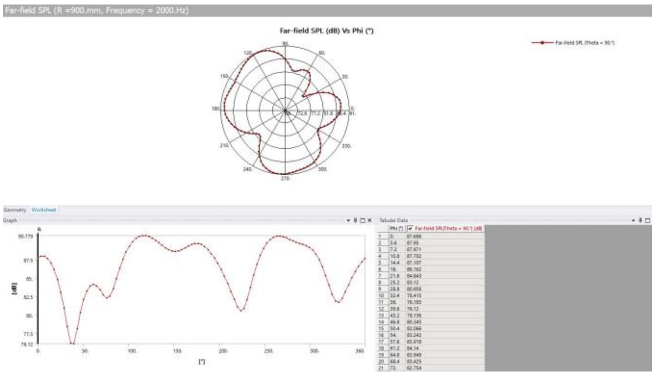

Figure - 76: Far Field SPL vs. Phi for Model 2

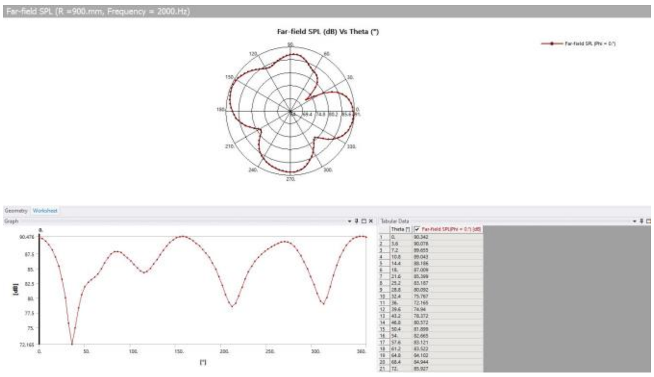

Figure - 77: Far Field SPL vs. Theta for Model 2

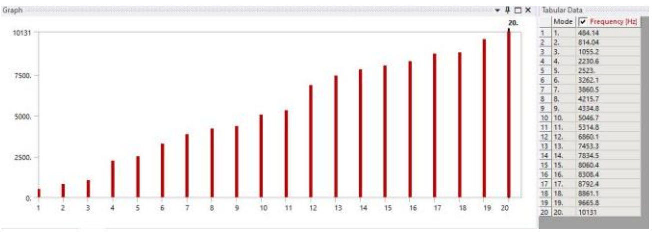

Figure - 78: Mode vs. Frequency for Model 2

4.1.2.4 Gray CI

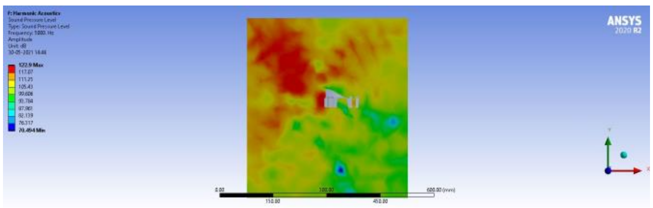

Figure - 79: SPL of Model 1

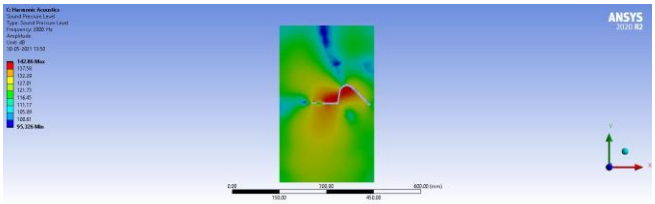

Figure - 80: SPL of Model 2

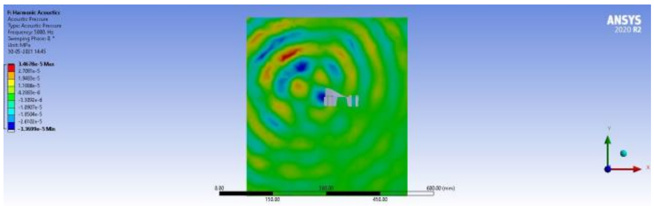

Figure - 81: Acoustic Pressure of Model 1

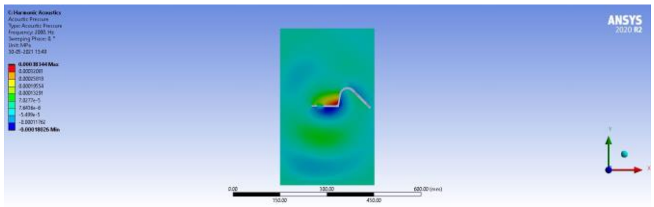

Figure - 82: Acoustic Pressure of Model 2

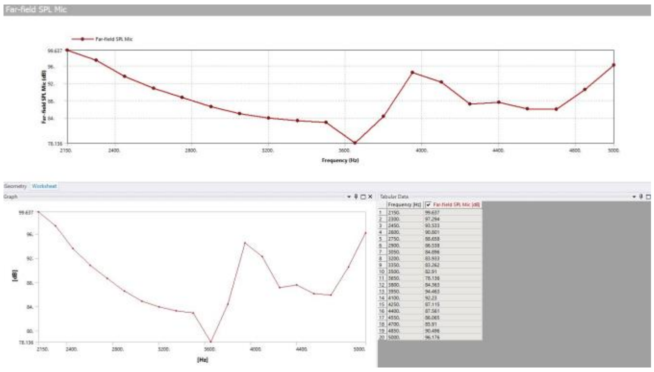

Figure - 83: Frequency Distribution of Model 1

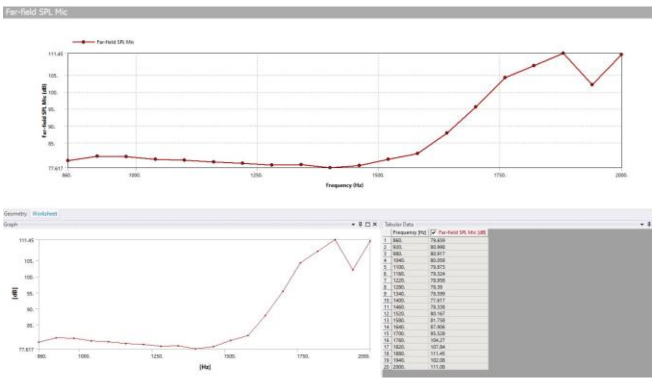

Figure - 84: Frequency Distribution of Model 2

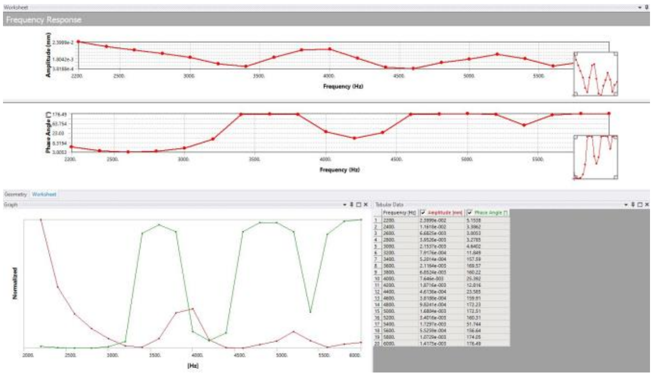

Figure - 85: Frequency Response of Model 1

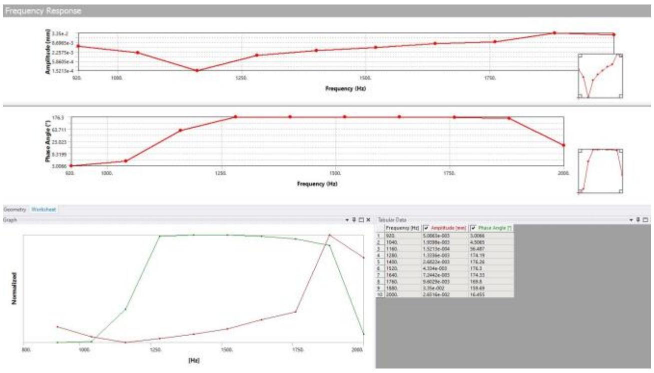

Figure - 86: Frequency Response of Model 2

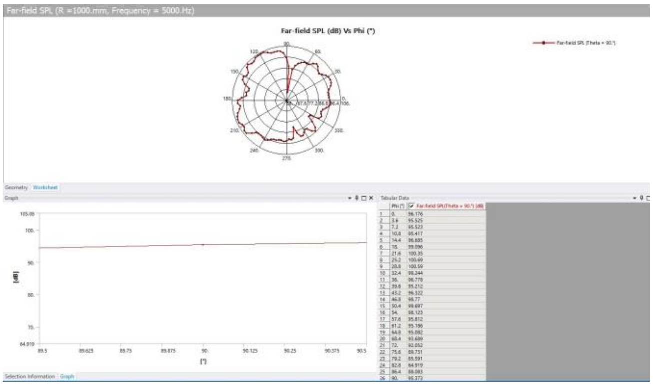

Figure - 87: Far Field SPL vs. Phi for Model 1

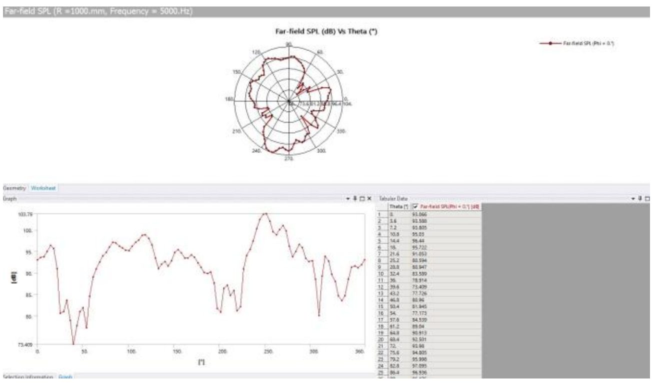

Figure - 88: Far Field SPL vs. Theta for Model 1

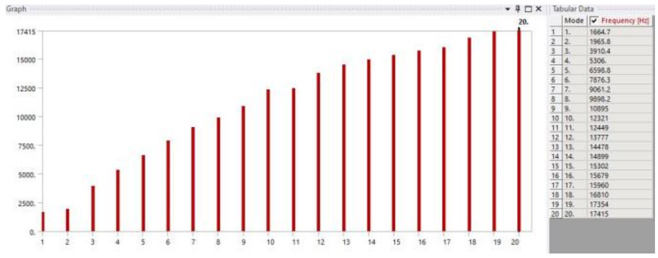

Figure - 89: Mode vs. Frequency for Model 1

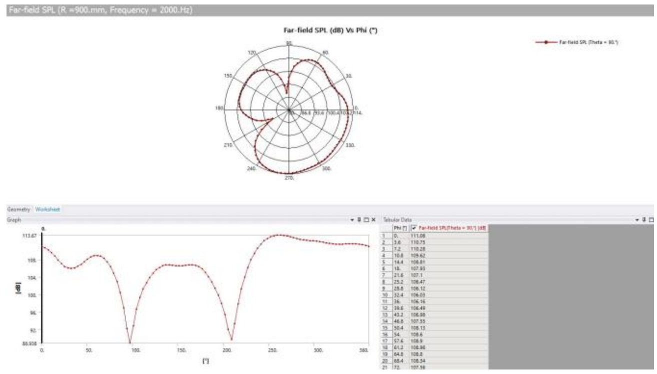

Figure - 90: Far Field SPL vs. Phi for Model 2

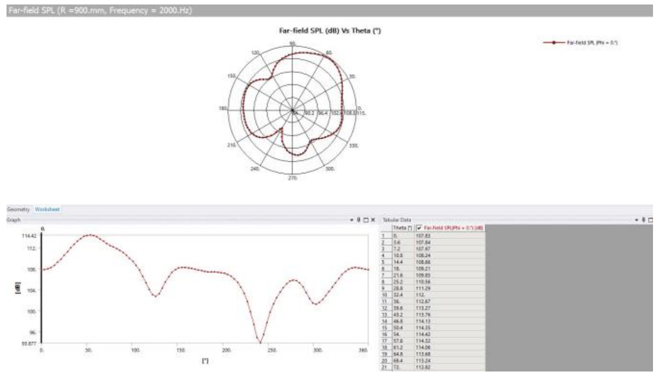

Figure - 91: Far Field SPL vs. Theta for Model 2

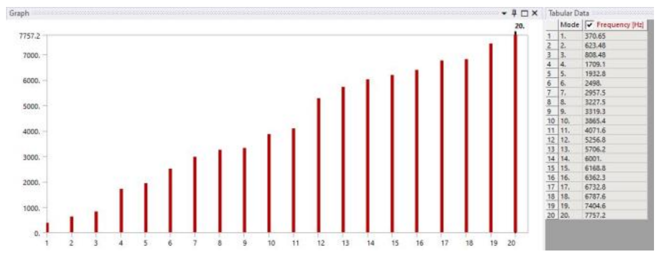

Figure- 90: Mode vs. Frequency for Model 2

5. FATIGUE ANALYSIS

While many parts may work well initially, they often fail in service due to fatigue failure caused by repeated cyclic loading. Characterizing the capability of a material to survive the many cycles a component may experience during its lifetime is the aim of fatigue analysis. In a general sense, Fatigue Analysis has three main methods, Strain Life, Stress Life, and Fracture Mechanics; the first two being available within the ANSYS Fatigue Module.

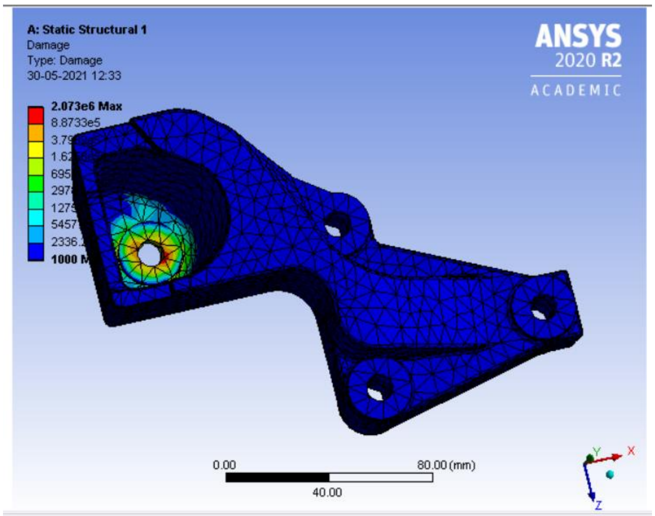

Figure - 91: Fatigue Analysis - Damage

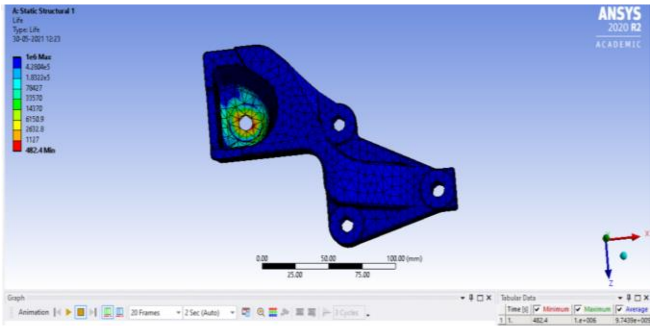

Figure - 92: Fatigue Analysis - Life

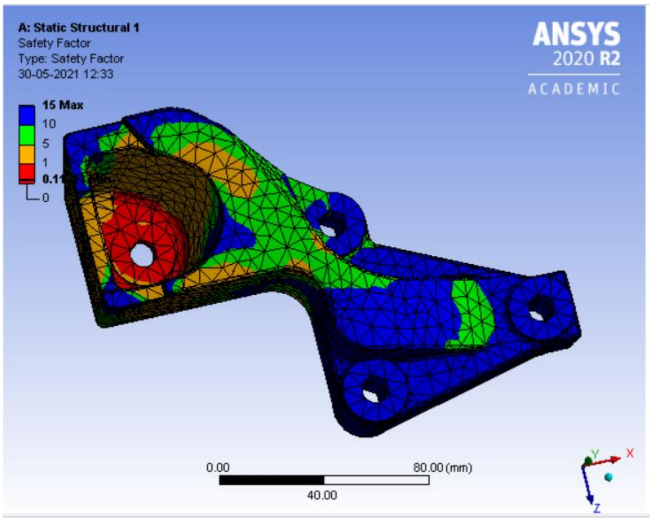

Figure - 93: Fatigue Analysis - Safety Factor

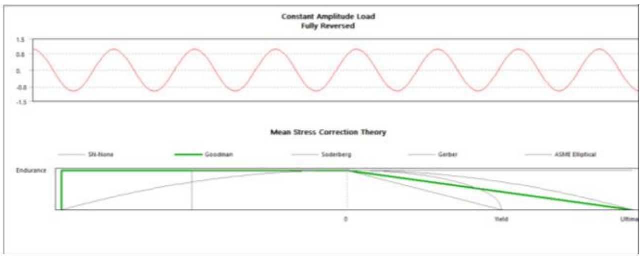

Figure - 94: Fatigue Analysis

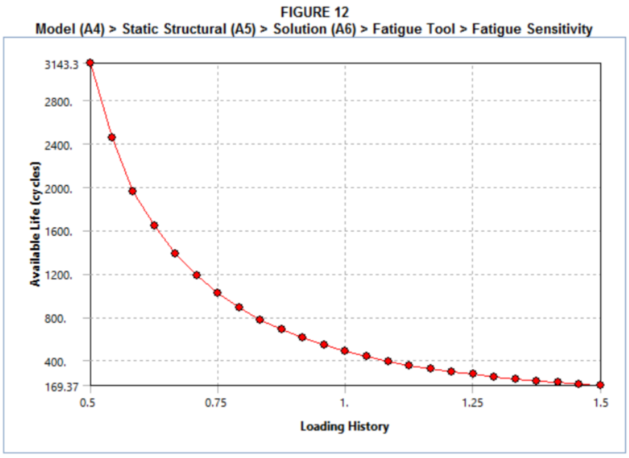

Figure- 95: Fatigue Sensitivity

Object Name	*Fatigue Sensitivity*
State	Solved
Scope	
Geometry	All Bodies
Definition	
Sensitivity For	Life
Suppressed	No
Options	
Lower Variation	50. %
Upper Variation	150. %
Number of Fill Points	25
Chart Viewing Style	Linear

Figure - 96: Table of Fatigue Sensitivity

6. RESULT TABLES

6.1. Static Structural Analyses and NVH Analyses

Table -3: Total Deformation of Model 1

Material	Total Deformation (Model 1) (in mm)	
Al Alloy	0	2.4757
Mg Alloy	0	3.8673
Gray CI	0	1.5827

MS1018	0	0.84667

Table -4: Total Deformation of Model 2

Material	Total Deformation (Model 2) (in mm)	
Al Alloy	0	0.91615
Mg Alloy	0	1.4201
Gray CI	0	0.57753
MS1018	0	0.30968

Table -5: Equivalent Stress of Model 1

Material	Equivalent Stress (Model 1) (in MPa)	
Al Alloy	0.016273	734.79
Mg Alloy	0.016478	757.64
Gray CI	0.016888	778.64
MS1018	0.016827	770.43

Table -6: Equivalent Stress of Model 2

Material	Equivalent Stress (Model 2) (in MPa)	
Al Alloy	0.030869	125.950
Mg Alloy	0.029665	126.420
Gray CI	0.028482	126.790
MS1018	0.028952	126.650

Table -7: Directional Deformation of Model 1

Material	Directional Deformation (Model 1) (in mm)	
Al Alloy	-0.62577	0.071164
Mg Alloy	-0.97725	0.11129
Gray CI	-0.39932	0.045634
MS1018	-0.21391	0.024391

Table -8: Directional Deformation of Model 2

Material	Directional Deformation (Model 2) (in mm)	
Al Alloy	-0.0061059	0.41539
Mg Alloy	-0.0091515	0.643552
Gray CI	-0.0035954	0.26158
MS1018	-0.0019549	0.14029

Table -9: SPL of Model 1

Material	SPL (Model 1) in Decibels	
Al Alloy	97.515	140.48
Mg Alloy	101.05	144.15
Gray CI	70.494	122.90
MS1018	67.552	123.79

Table -10: Acoustic Pressure of Model 1

Material	Acoustic Pressure (Model 1) in MPa	
Al Alloy	-0.000242423	0.00023798
Mg Alloy	-0.00036236	0.00036444
Gray CI	-3.3699e-5	3.4678e-5
MS1018	-3.6826e-5	3.1571e-5

Table -11: SPL of Model 2

Material	SPL (Model 2) in Decibels	
Al Alloy	88.082	136.21
Mg Alloy	92.337	140.04
Gray CI	95.326	142.86
MS1018	73.27	126.57

Table -12: Acoustic Pressure of Model 2

Material	Acoustic Pressure (Model 2) in MPa	
Al Alloy	-0.00017964	0.00013361
Mg Alloy	-0.00027835	0.00021541
Gray CI	-0.00018026	0.00038344
MS1018	-5.9551e-5	4.194e-5

7. CONCLUSION

The parameters of the SolidWorks Models were validated from PEUGEOT 206 GTI engine mount bracket and other data found [6] [7]. In terms of Structural strength, Gray CI would be a suitable material to use. After the NVH analysis, it can be concluded that Gray CI material is the most optimum material. For all the materials used the parameters are within the permissible range implying these can be implemented as a substitute for the current material. The parameters to be considered can be varied based on vehicle to vehicle requirement.

8. ACKNOWLEDGMENT

We would like to thank Prof. (Dr.) S. S. Shinde for giving us the objective of this project and guiding us throughout the duration of the project. We appreciate the support by Prof. (Dr.) Mangesh Chaudhari, Head of Department of Mechanical Engineering, to approve our project in our curriculum. We express our sincere gratitude towards the management of Vishwakarma Institute of Technology, Pune for providing us the right means to successfully complete the project.

REFERENCES

[1] Finite Element Analysis and Natural Frequency Optimization of Engine Mounting Bracket by Mallikarjun B Patil, Prof. S. S. Kelkar, Mechaical Engineering, JSPM Jayawantrao Sawant College of Engineering, Hadapsar, Pune, India..

[2] Finite Element Analysis and Natural Frequency Optimization of Engine Bracket by Umesh G. Ghorpade, D. S. Chavan, Vinay patil and Mahendra Gaikwad, International Journal of mechanical and Industrial Engineering.

[3] Modal Analysis of Engine Supporting Bracket using Finite Element Analysis by A. S. Adkine, prof. G.P. Overikar, Prof. S. S. Surwase International Journal of Advaced Engineering Research and Science (IJAERS)

[4] Finite Element Analysis of Engine Mounting Bracket by Rahul S. Kadolkar and Anand C. Mattikalli, International Research journal of Engineering and Technology (IRJET).

[5] Engine Mounts and its considerations by Atul Adhau and Prof. V. Kumar, International Research journal of Engineering and Technology (IRJET)

[6] PEUGEOT 206 GTI 180 TOP ENGINE MOUNT DRIVER SIDE UPPER ARM (https://grabcad.com/library/engine-mount-3)

[7] Front Engine Mount Bracket from the GrabCad Library (https://grabcad.com/library/front-engine-mount-bracket-1)

[8] https://www.encycarpedia.com/peugeot/04-206-gti-180-rc-hatch