Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 510-522

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Vibration Characteristics of Engine Support Bracket Using Finite Element Analysis

M Akmalhakim[1], S. Kanna Subramaniyan[1]*

[1]Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Batu Pahat, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.057
Received 02 Aug. 2021; Accepted 27 Nov. 2021; Available online 25 December 2021

Abstract: Engine support bracket play a vital role of reducing the noise, vibration, and harshness. Engine support bracket is important part of the Engine mount that have function to absorbing the vibration of the car engine and hinder this vibration from going into car passenger. Vibration is producing the natural frequency of the moving part and the speed of the car that produce by car engine. The higher speed of car will provide more vibration. In order to reduce the natural frequency of the engine support bracket, the Finite Element Analysis (FEA) has been done using the Abaqus software. The objective of this research is to investigate about the natural frequency of the car engine support bracket. Also, another objective is to modify the engine support bracket by doing design optimization and comparing the design frequency with the previous study. Based on these two objectives, the model of engine support bracket has been modified from the previous study model using the design optimization method by SolidWork 2019 software. The new part has been adding in the engine support bracket to make the differences between previous study models. The dimension of the addition part is rectangular par that have length 200 mm, width 20 mm and the thickness 10 mm. For the natural frequency and its deformation, Abaqus 2017 Software was using for the simulation test. The research has provided the natural frequency and deformation for the engine support bracket model for two materials which is Magnesium Alloy and ERW-1 material. ERW-1 material is the material that commonly use in the automation industries. This simulation produce the frequency result and it also has reducing the frequency for the ERW-1 material from the previous study. The simulation has shown the ERW-1 material is suitable material for the modified model of the engine support bracket. The vibration of the engine support bracket has been reduced because the natural frequency was reduced. To conclude, this simulation is likely to succeed and can be carried forward to further simulation for better improvement.

Keywords: FEA, Vibration, Natural Frequency, Abaqus, ERW-1.

**Corresponding author: skanna@uthm.edu.my*

1. Introduction

Engine mounts are block of rubber mounted on steel brackets that are bolted to protect the engine and reduce the vibration and noise in the car that is sensed and perceived. If the rubber wears out or breaks and if the steel brackets snap, more vibration, and noise will be provided by the car engine. If the mounts are damaged, the engine still can be used, and the car can also travel a few distance more but it will be affected to the people inside the car because the vibration of the engine will be transferred to car body. The assembly of the engine support bracket includes support members connected to the main frame vehicle body. A strong mounting system distinguishes the vibration producing engine from the vehicle body and suppresses the effect on the vehicle driver of rough ground surface inputs.

Noise, vibration, and harshness are critical aspects of cars and the reduction in these features will increase the engine performance. Due to reduce the vibration of the car engine, the engine support bracket is very important. For the finite element analysis (FEA), the finite element model of the engine support bracket is created using ABAQUS software to find their vibration characteristic and do the simulation and try new material for engine support bracket to reduce vibration and noise of the car engine. To find the vibration characteristic need to find two measurable quantities. The quantities are the amplitude or intensity and the frequency of the vibrating objects. Also, in this research we also find the vibration response for engine support bracket. In general, the vibration response is a unique concept in that it can be both a desirable and undesirable form of energy. It can be created by simple motion and then move on its own until return to its natural, resting state. The resonance condition or mechanical resonance is the tendency of a mechanical system to respond at amplitude when the frequency of its oscillation matches the system's natural frequency of vibration, or can we call it resonance frequency than it does at other frequency.

2. Literature Review

2.1 Materials

The materials that suitable to be used to be test on the engine support bracket is gray cast iron (GCI), Aluminium Alloy (Al), Magnesium Alloy (Mg) and Electrical resistance welded (ERW) steel tube. Electrical resistance welded (ERW) steel tube is commonly used in application of the automobile because it is low cost, suitable weight, and high strength. (A.S. Adkine P. P., 2017) The lowest frequency of these material was produced by Gray cast iron when comparing to the initial engine support bracket. (A.S. Adkine, 2017)

Gray Cast Iron (GCI): These are mixing of alloy that containing more than 2% of carbon, up to 3% of silicon and less than 1% of alloy element which is chromium, copper, magnesium, molybdenum, nickel, phosphorous, silicon, titanium, vanadium, and sulphur. This material is used to make heavy machine. (A.S. Adkine, 2017)

Aluminium Alloy (Al): Soft silvery white metal with low tensile strength. This material is good electric conductor and anti-corrosion. To improve this strength of this material, need to add some material to be fix on this such as silicon, zinc, and more other material. (A.S. Adkine, 2017)

Magnesium Alloy (Mg): lightest of all metals that used as basic for constructional alloys. It is used to mix with another alloy for improving that material. This material good because has high specific strength, suitable for casting and welding process. (A.S. Adkine, 2017). The mechanical properties of this material are shown in the table above. (Sahil Naghate, 2012):

Table 1: Mechanical properties of the Mg alloy material. (Sahil Naghate, 2012)

Magnesium Alloy (Mg)	
Young's modulus (N/m^2)	4.5 e+10 (N/m^2)
Poisson's ratio	0.35
Density (Kg/m^3)	1800 (Kg/m^3)

Yield strength (N/m^2)	1.93 e+8 (N/m^2)

ERW-1: the standard material that be use in the automobile's product. The mechanical properties of this material are shown in the table above (A.S. Adkine P. P., 2017):

Table 2: Mechanical properties of the ERW-1 material (A.S. Adkine P. P., 2017)

IS 3074 ERW-1	
Young's modulus (N/m^2)	2.1 e+10 (N/m^2)
Poisson's ratio	0.29
Density (Kg/m^3)	7800 (Kg/m^3)
Yield strength (N/m^2)	2.4 e+8 (N/m^2)

2.2 Natural frequency

Frequency is the important part in order to reduce the vibration of the vehicle engine. High frequency will produce the high vibration because these two components were linked. For the engine support bracket, frequency of the bracket should be more than engine frequency to avoid the resonance that may lead to failure and will give some damage to the vehicle in some cases. (Tushar P. Kamble, 2016). Table 2.3 above shown the example of the natural frequency from the initial design of the engine support bracket using ANSYS software. (A.S. Adkine P. P., 2017).

Table 3: Frequency of the Engine support bracket with ERW-1 Material (A.S. Adkine P. P., 2017)

Mode	Frequency (Hz)
1	145.86
2	164.62
3	257.83
4	489.68
5	607.07
6	697.86

3. Methodology

3.1 General Procedure of FEA

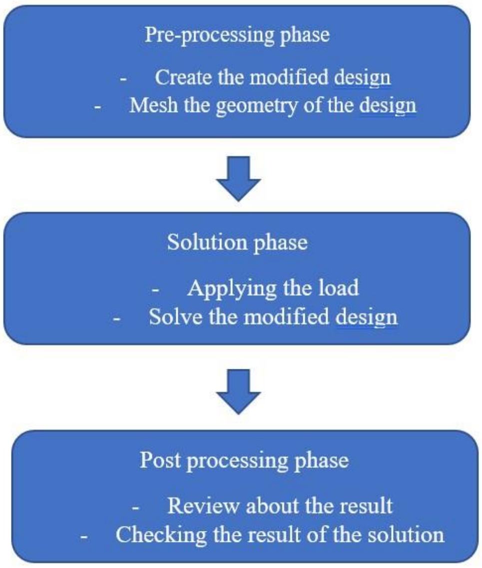


Figure 1: General Process of FEA

From the figure 1 above, on the pre-processing phase, this study will be created the modified design and mash the geometry of the design until get the required design. For the solution phase, the load will be applied on the structure to get the boundary condition of the modified design. The solution of this will get the natural frequency, and static structural of the modified design. For the post processing phase, the result of the frequency will be analysed, and the frequency will be compared with the initial design. The flow chart of the FEA method is shown on figure 2.

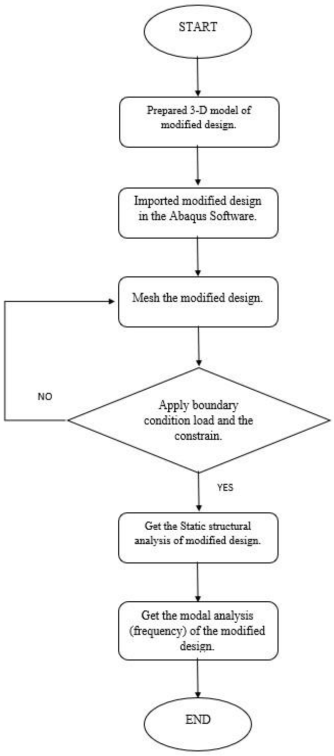


Figure 2: Flow chart of the FEA analysis of this study

3.2 Mesh

Mesh was the process of separating the entire object into several elements so that when load was applied, the load could be divided evenly. To get the better solution, successful mesh is important in the simulation method. Figure 3.10 shown the mesh of the model of the engine support bracket.

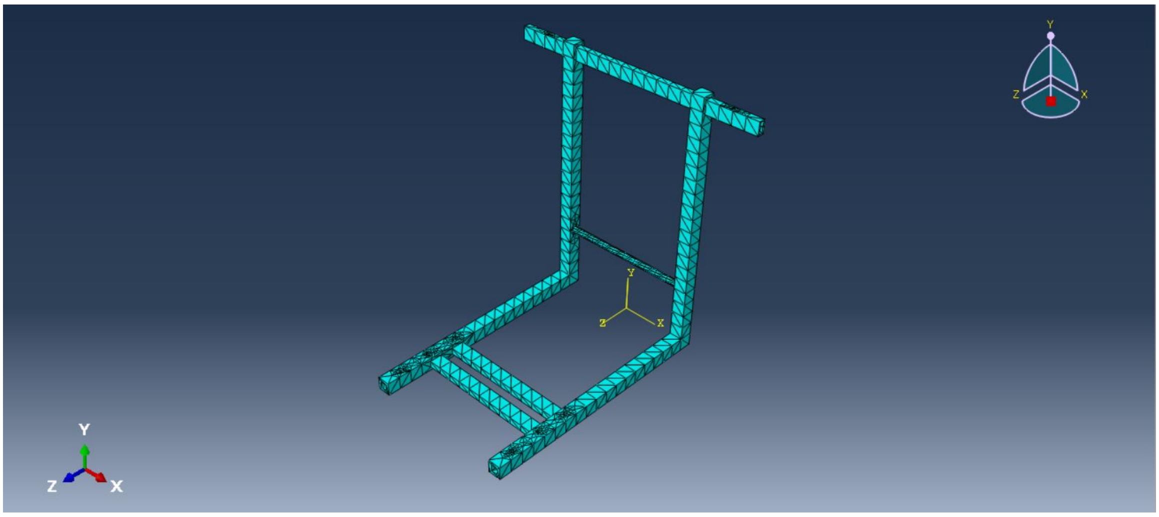


Figure Error! No text of specified style in document.**: Model mesh**

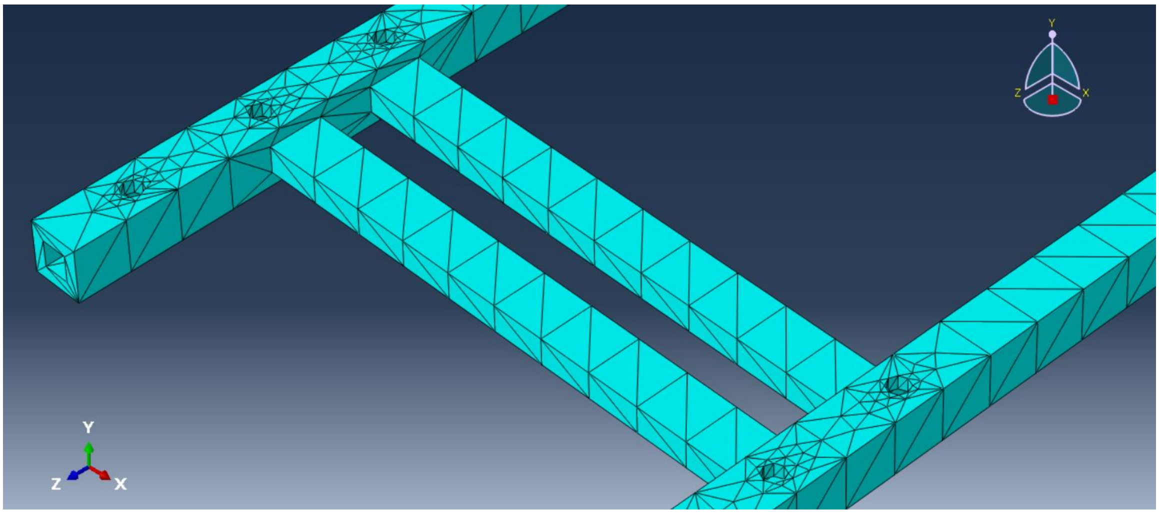


Figure 4: Mesh Model for Modified Part

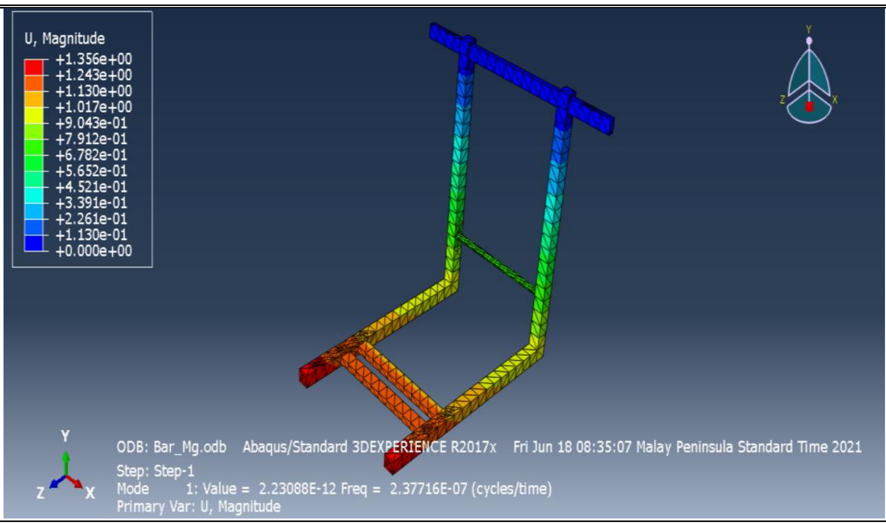
All the elements of this model are 5239. The percentage of the error for this mesh is 0 (0 %) but for the analysis warning is 901 (17.1979 %). This percentage for the analysis warning is quite higher but this data is the lowest percentage comparing with all the data that produce after the refine mesh process.

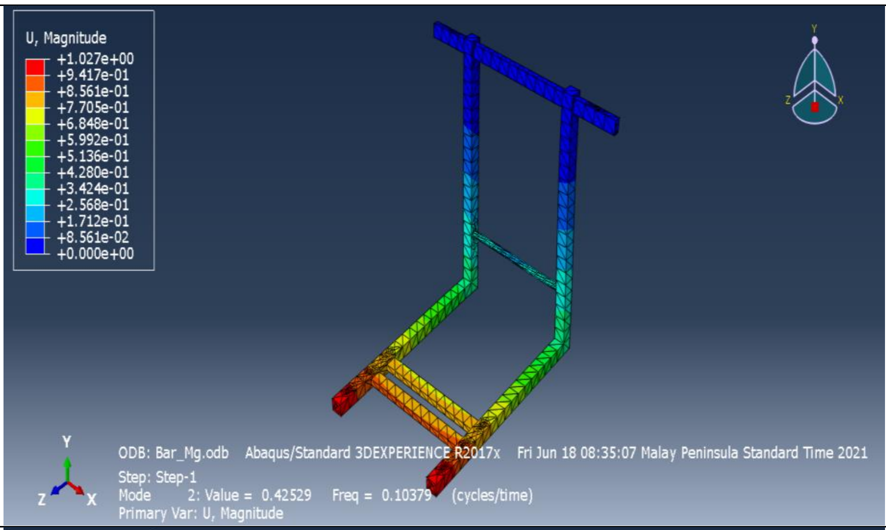
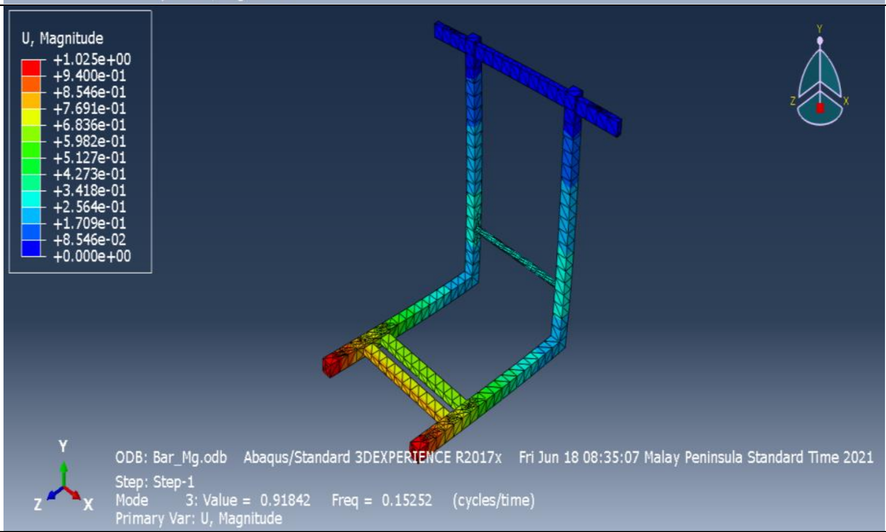
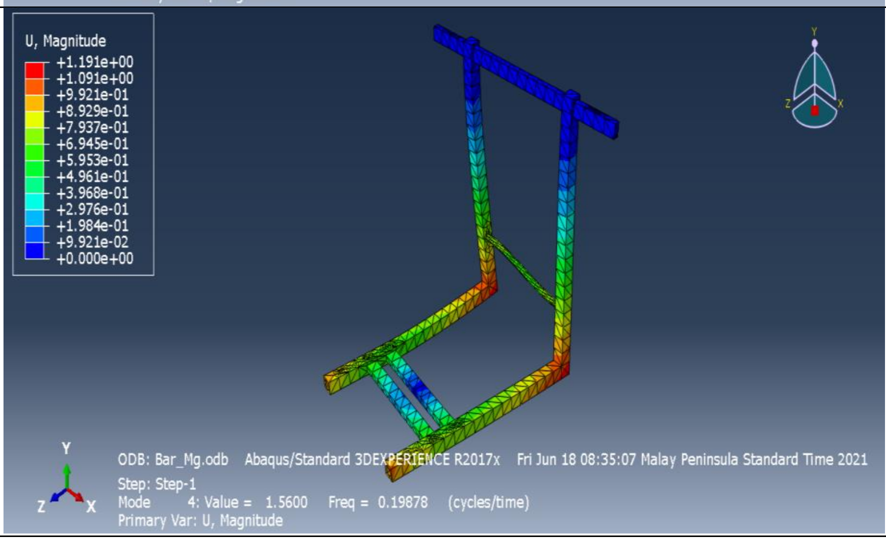
4. Results and Discussion

4.1 Natural Frequency

The Natural frequency and its deformation are shown on the table 4 and table 5. The Natural frequency results is provided by Abaqus software by doing simulation test on Frequency Step. Two different materials are being used. First material is Magnesium alloy on table 4 and second material is ERW-1 material on table 5.

Table Error! No text of specified style in document.**: Frequency results for Magnesium Alloy**

Mode	Image	Frequency (Hz)
1		2.37716e-7

2		0.10379
3		0.15252
4		0.19878

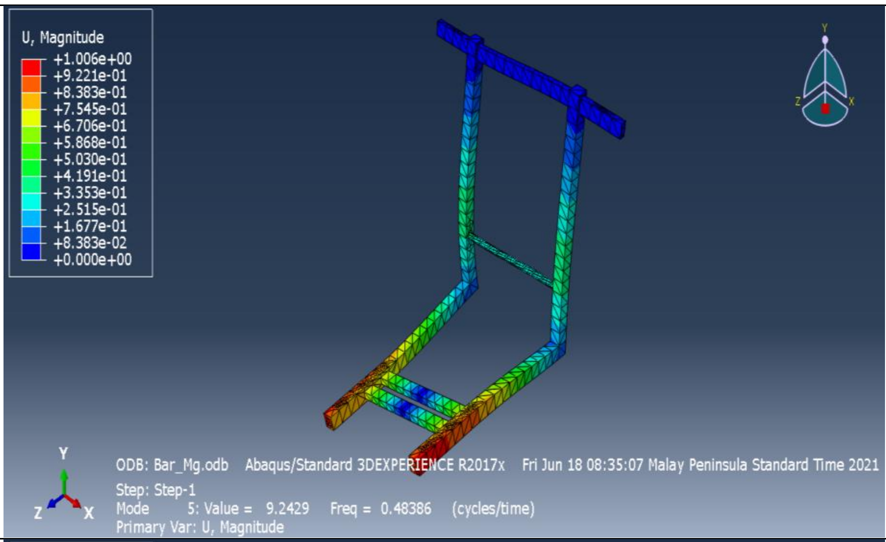
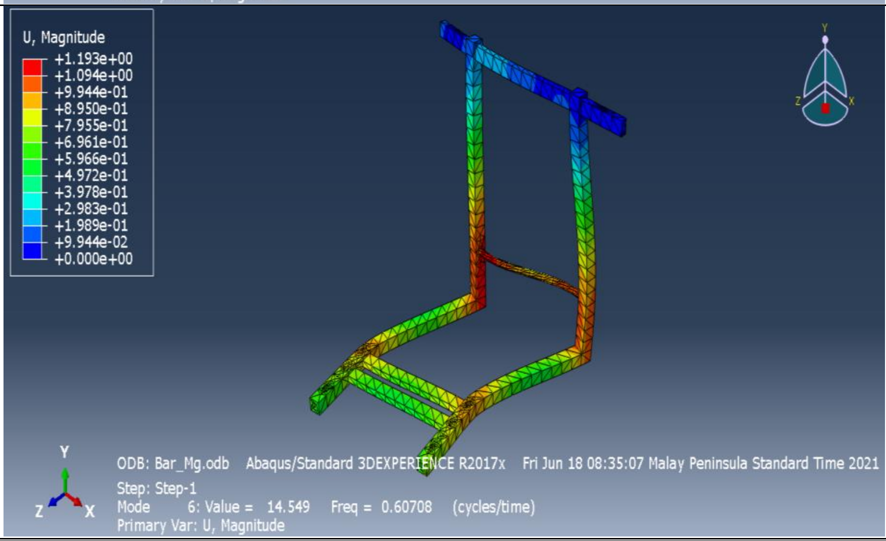
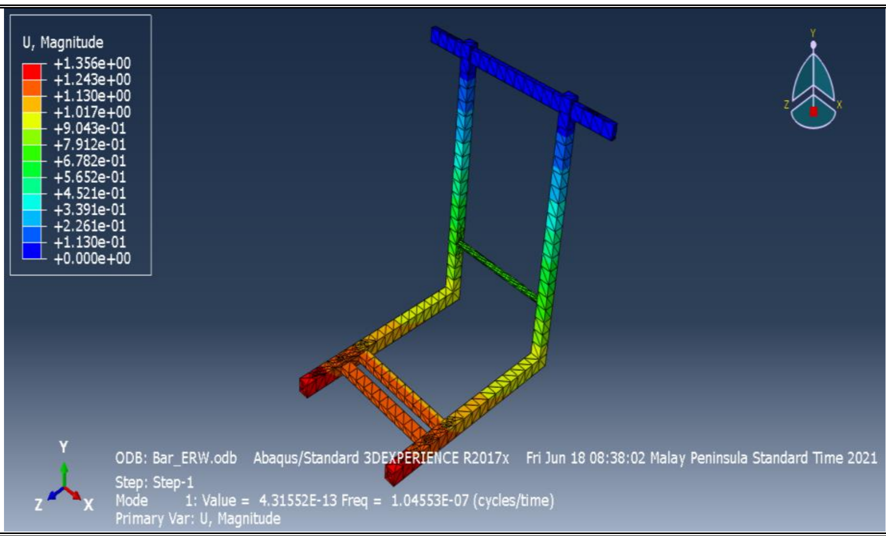
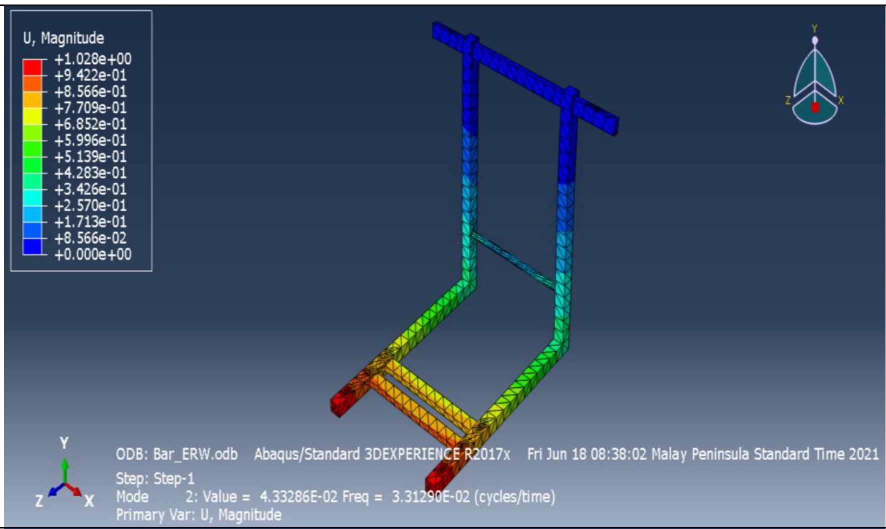
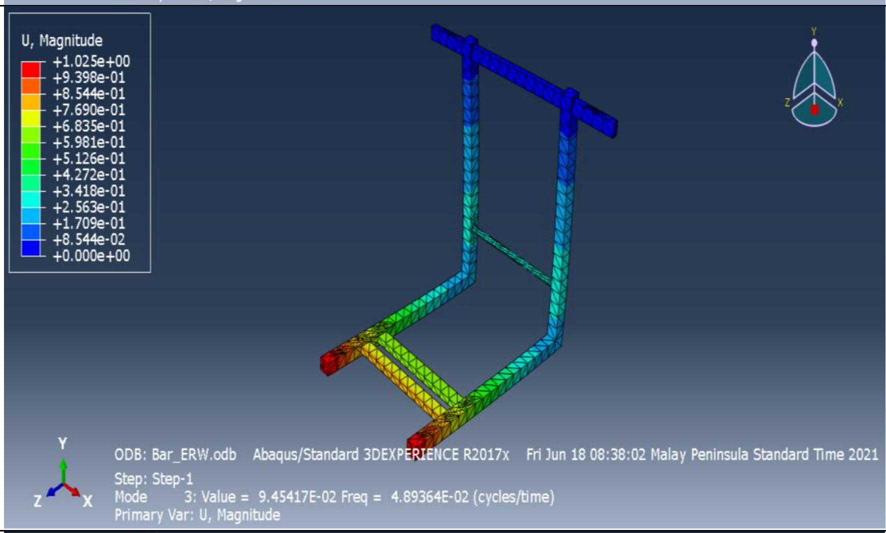
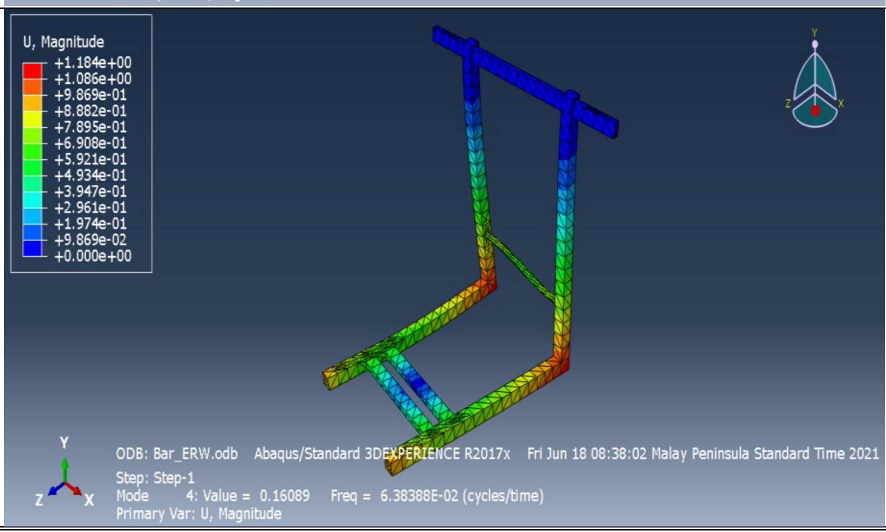
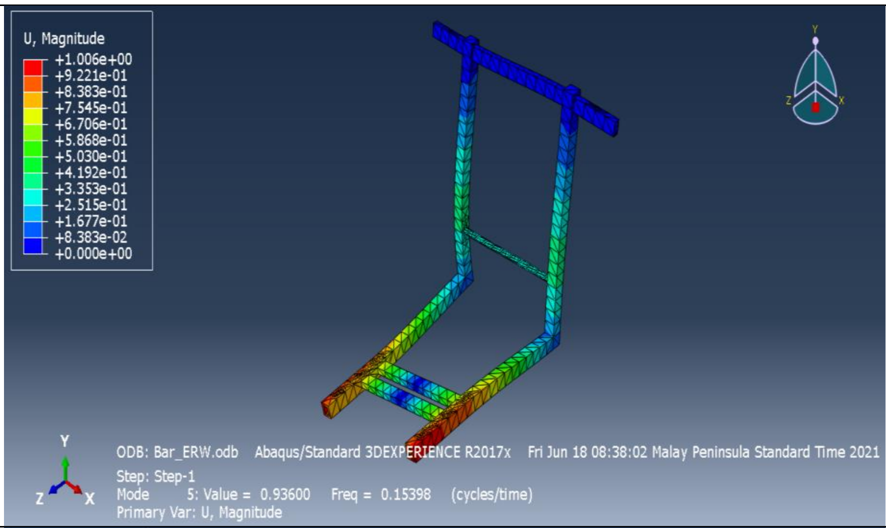
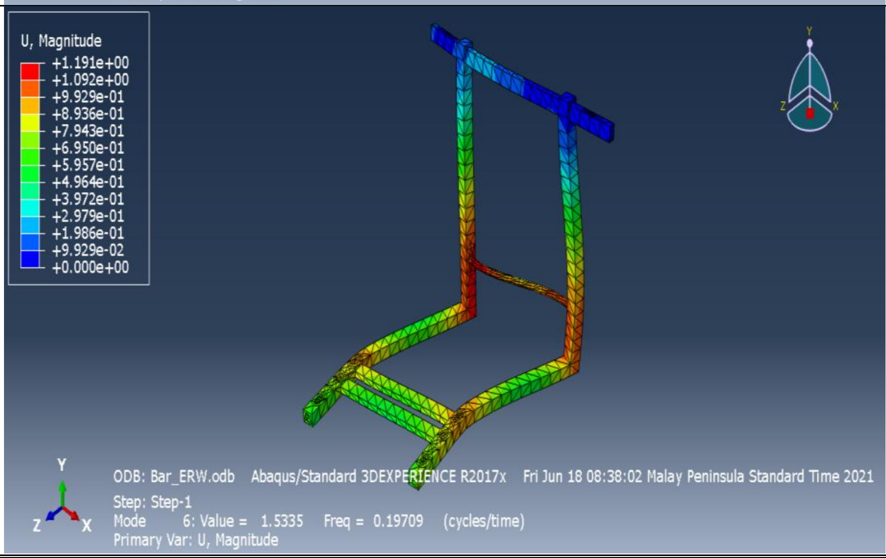
5		0.48306
6		0.60708

Table 5: Frequency results for ERW-1

Mode	Image	Frequency (Hz)
1		1.04553e-7

2		3.31290e-2
3		4.89364e-2
4		6.38388e-2

5		0.15398
6		0.19709

4.2 Graph

The graphs of the Natural frequency are shown on figure 5 and figure 6, This graph shown two different materials. On figure 5 shown material Magnesium alloy and on figure 6 shown ERW-1 material.

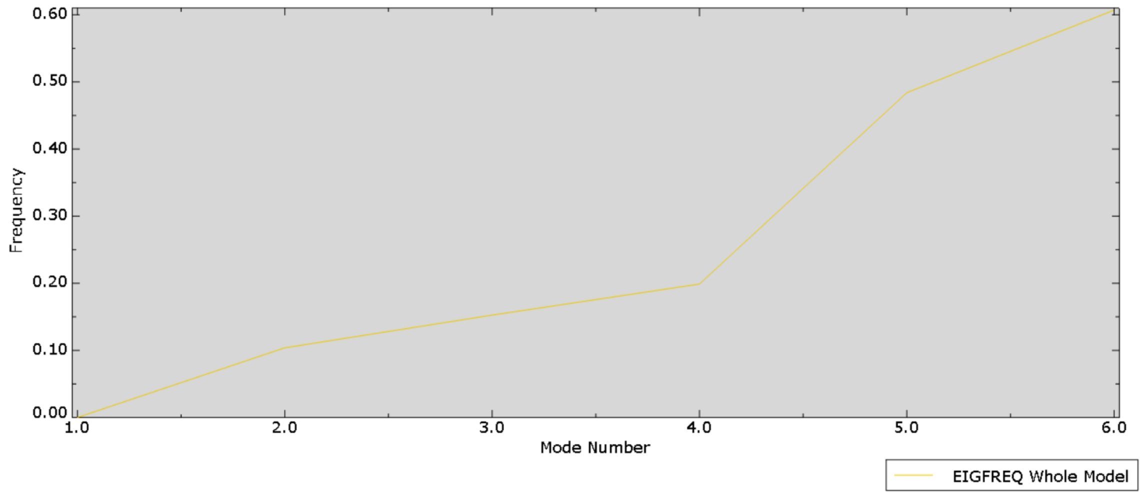


Figure 5: Frequency graph for Magnesium Alloy

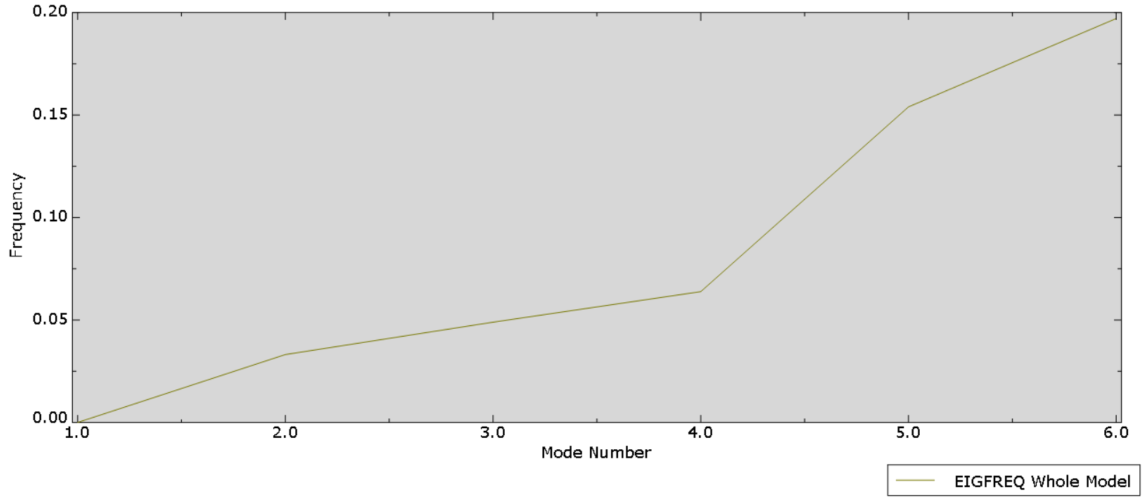


Figure 6: Frequency graph for ERW-1

4.3 Comparison data

Table 6: Comparison data between previous study

Mode	Previous Study design		Modified design	
	Mg Alloy (Hz)	ERW-1 (Hz)	Mg Alloy (Hz)	ERW-1 (Hz)
1	140.06	145.86	2.37716e-7	1.04553e-7
2	158.32	164.62	0.10379	3.31290e-2
3	248.62	257.83	0.15252	4.89364e-2
4	471.68	489.68	0.19878	6.38388e-2
5	585.14	607.07	0.48306	0.15398
6	672.85	697.86	0.60708	0.10709
Frequency comparison	ERW-1 > Mg Alloy		Mg Alloy > ERW-1	

To discuss it briefly the data on the table 6, previous study design has produced the frequency starting from 100 Hz to 700 Hz. This data has shown the higher frequency comparing to the data that have been produce from modified design. The frequency that produces the frequency between 0 Hz to 0.6 Hz. This is because some different between these two simulations. For the previous study simulation, they were adding some load on the model which is 820 N load to be consider as the engine load while for the modified design simulation, the weight of the engine load has been neglected. In order to get the natural frequency, loading is not important in this simulation part. Abaqus Software simulation also not require inserting the input for the loading. Also, another different between these two simulations is using the different simulation software. For the previous study design was using ANSYS software that require to inserting the load value but for the modified design was using Abaqus software that do not need the load as their requirement to get the Natural Frequency. for the frequency comparison for these two results on table 4.3, previous study data has shown that the Magnesium alloy has the lower natural frequency comparing with the ERW-1 material. For modified design data has shown that the ERW-1 material has the lower frequency. This result has shown the successful data that can reduce the natural frequency for ERW-1 material from being higher than Magnesium alloy to being lower than Magnesium alloy. This data also proves that the suitable material for modified design is ERW-1 material which can produce lower natural frequency for engine support bracket.

5. Conclusion

Throughout this analysis, the approach of the Finite Element Analysis (FEA) is used to evaluate the value of the Natural frequency for mode 1 until mode 6 and its graph. The simulation research was carried out using steady procedure to run the simulation. As in the case of design optimization, a modified design has been set and been created in the SolidWork software. However, the correct meshing need to be taken to arching the objective. The mashing quality has been chosen that has the lower Analysis warning which is 17.1979 %. Also, boundary condition also needs to be chosen in the right place for get better natural frequency and deformation. The boundary condition has been chosen at the screw place and be setup as pinned that will make all the axis is zero. The result obtained from the simulation test was be compared with the previous study. From this comparison, the suitable natural frequency for the previous study data is produce by Magnesium alloy but for this research study, The suitable natural frequency is produced by ERW-1 material. This material shown the stable increment for natural frequency. This research has archived their two objectives. The first objective is the first objective was to investigate about the natural frequency of the car engine support bracket and last objective is modify the engine support bracket by doing design optimization and comparing the design frequency with the previous study. These two objectives are being done successfully with simulation test using Abaqus Software. However, this simulation analysis had its limitation because it demanded very high specification of the computer. Therefore, due to this problem, the time limitation of the duration of the project would take a lot of time to run simulation for one material. These studies are categorized into two case studies. All case studies are approved base on the key objectives of the research.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] A.S. Adkine, P. a. (2016). STATIC AND MODAL ANALYSIS OF ENGINE SUPPORTING BRACKET - A LITERATURE REVIEW. *International Journal of Modern Trends in Engineering and Research (IJMTER)*, 76-83.

[2] A.S. Adkine, P. P. (2017). An Experimental Investigation and Modal Analysis of an Engine Supporting Bracket. *International Journal of Advanced Engineering Research and Science (IJAERS)*, 152-159.

[3] A.S. Adkine, P. S. (2017). Modal Analysis of Engine Supporting Bracket using Finite Element Analysis. *International Journal of Advanced Engineering Research and Science (IJAERS)*, 55-63.

[4] A.S.Adkine, V. G. (2015). Static Behaviour of Engine Mounting Bracket. *International Advanced Research Journal in Science, Engineering and Technology*, 68-73.

[5]. Sahil Naghate, S. P. (2012). Modal Analysis of Engine Mounting Bracket Using FEA. *International Journal of Engineering Research and Applications*, 1973-1979.

Masahiko Kondo, Y. H. (2019). ANTI - VIBRATION DEVICE FOR VEHICLE. *United States Patent*, 1-20.

[6] Monali Deshmukh, P. K. (2014). Analysis and Optimization of Engine Mounting Bracket. *International Journal of Scientific Engineering and Research (IJSER)*, 2347-3878.

[7] Mungi Prasad, S. T. (2020). Experimental investigation and material characterization of rubber engine mount for automobile application. *Materials Today: Proceedings*, 4133-4141.

[8] Murat Mayda, N. G. (2017). The Effect of Use of Air Conditioner on Engine Vibration in Gasoline Car Engines. *International Journal of Energy Applications and Technologies*, 182-185.

[9] N C Nutu, C. P. (2018). An experimental approach on fuelling a passenger car diesel engine with LPG. *The 8th International Conference on Advanced Concepts in Mechanical Engineering*, 1-8.

[10] Panda, K. C. (2016). Dealing with Noise and Vibration in Automotive Industry. *Procedia Engineering 144*, 1167-1174.

[11] Priyanka S. Dahalea, P. D. (2018). Assessment of Boundary Conditions for FEA of Mounting Bracket Using Co-relation with Experimental Results. *Materials Today: Proceedings 5*, 13290-13300.

[12] Rajendra Kumar Prajapati, D. S. (2020). A LITERATURE REVIEW ON FEA ANALYSIS OF ENGINE SUPPORTING BRACKET. *INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT*, 15-20.

[13] Ritesh Bhairappa Dhamoji, D. V. (2016). STRENGTH AND STIFFNESS ANALYSIS OF AN ENGINE BRACKET. *International Journal For Technological Research In Engineering*, 2637-2641.

[14] S. Kirthana, M. K. (2018). Finite Element Analysis and Topology Optimization of Engine Mounting Bracket. *Materials Today: Proceedings 5*, 19277-19283.

[15] S. Santhosh, V. V. (2020). Experimental investigation and comparative analysis of rubber enginemount vibration and noise characteristics. *Materials Today: Proceedings*, 638-642.