Santhoshkumar et al. 2018. *Int. J. Vehicle Structures & Systems*, 10(6), 415-416
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.10.6.08

International Journal of
Vehicle Structures & Systems
Available online at www.maftree.org/eja

Finite Element Modelling and Analysis of Novel Engine Mount: Technical Note

D. Santhoshkumar, V. Jayakumar[a], M. Ravi Teja Reddy and A. Muniappan

Dept. of Mech. Engg., Saveetha School of Engg., Saveetha Institute of Medical and Technical Scis., Tamil Nadu, India
[a]Corresponding Author, Email: jkmails2k2@yahoo.com

ABSTRACT:

Vibrations are caused in an engine due to the continuous motion of reciprocating and rotating parts. Engine mounts are used as the vibration isolators from the engine to frame. The absorbing capacity of the mounts should be large enough to withstand the vibrational force for longer periods. Engine mount's strength depends upon the material and the type of the design. The modelling and assembly of the engine mount is done using solidworks and the analysis of the mount is done using Ansys software. Modal analysis is done on the engine mount at working frequency. With the results obtained, factor of safety is calculated and it is compared with the existing mount. The proposed engine mount model results in encouraging factor of safety value.

KEYWORDS:

Vibrations; Engine mount; Modal analysis; Finite element analysis

CITATION:

D. Santhoshkumar, V. Jayakumar, M.R.T. Reddy and A. Muniappan. 2018. Finite Element Modelling and Analysis of Novel Engine Mount: Technical Note, *Int. J. Vehicle Structures & Systems*, 10(6), 415-416. doi:10.4273/ijvss.10.6.08.

1. Introduction

In automobiles, the continuous motion of reciprocating and rotating parts is responsible for generation of vibrations. Now-a-days due to increase in the capacity and decrease in engine weight, the frequency of emitted vibrations are very high. Generally, engine mounts are made of natural rubber and metal, and used for connecting a car frame to its engine. The metal portion acts as connector between engine and its frame to the engine on one side and to the frame on the other. The flexibility is provided by the rubber mount between them, so that the vibrations caused in the engine does not transmit to frame. Engine mounts used by newer cars are slightly different but the purpose is same which is isolating vibrations from engine to frame. The number of engine mounts used may vary from vehicle to vehicle. If engine mounts are not used, vibrations get transmitted to the frame. This affects the comfort of the passenger and functioning of many components of engine.

Yu et al [1] developed a design to meet the pre-requisites of the engine mount using parameter improvement technique. Frequency response functions are developed using finite element analysis (FEA) with which design sensitivities analysis of vibro-acoustic system is done by Lee et al [2]. Ramachandran et al [3] developed a mathematical model using Minitab software based up on the deformations obtained from FEA and the data modelling using I-DEAS [3]. Two reasons for vibrations are determined namely firing pulse which is caused due to fuel explosion and also due to the reciprocating and rotating parts [4]. By simulating decoupled, switch mechanism a new hydraulic mount is developed using lumped non-linear model [5]. A new model is introduced which can reduce the vibration, fatigue strength, displacement and to increase durability [6]. Based on indirect measurement of actuators' inductance, a new method is developed which accounts for the wear and assembly tolerances of semi active engine mount [7]. The following sections details a description of the novel design for an automotive engine mount, modelling and modal analysis followed by results and discussions.

2. CAD model and materials selection

The CAD model of the novel engine mount mainly comprises of two circular bodies, which is done purposely to have better surface area. These circular bodies are connected by arc like structures so that vibrational force can be transmitted smoothly throughout the body of the mount. In the middle of mount there is a straight structure supporting the two circular bodies which serves the purpose of taking impact forces at high frequencies. It is made up of natural rubber due to its high impact strength compared with fluorocarbon. The holes are included for fitting the engine mount in the clamp of the engine frame. The isometric and top views of the part model of the engine mount are shown in Fig. 1 and Fig. 2 respectively. The dimensions are taken with respect to the existing mounts for a two-cylinder tractor engine as reference. Material used for the mount is fluorocarbon because of its high mechanical properties like high shear strength and fatigue strength. Rubber is used at the centre of the engine mount due to its high impact strength so that it can take sudden shock and distribute the mount.

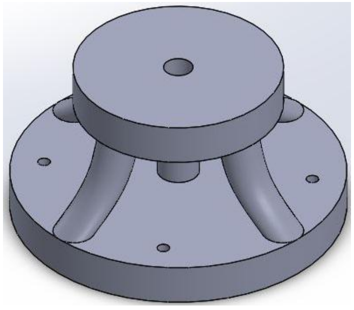

Fig. 1: Model of the engine mount

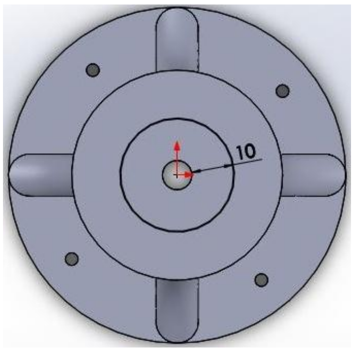

Fig. 2: Top view of the mount

3. Modal analysis using FEA

The part modelling is done in Solidworks. The model was saved in the IGES format and then imported into ANSYS for FEA. Fixed support boundary condition is applied on all the holes at the bottom part of the mount. Fine meshing is done for better analysis of the mount. After performing the modal analysis, result is taken out at particular frequency. The stress caused in the mount at the working frequency is shown in Fig. 3. Since the impact is at the centre, the maximum stress is also observed at the centre. Adding rubber at the centre has become an advantage as it distributes the whole force throughout the mount

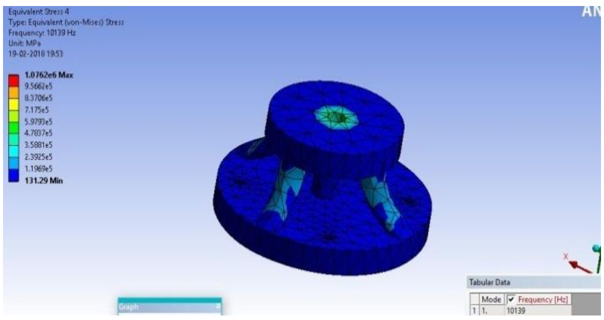

Fig. 3: Working stress in the mount

The deformation in the mount is shown in Fig. 4. At working frequency, the mount tends to rotate. As the shear force of fluorocarbon is very high, the rotating motion would not affect the model of the mount. The working stress of the mount is calculated at the working rpm of the engine. In general, the efficiency of the model is calculated by using factor of safety which defines the safety margins of the design at particular working conditions. Table 1 presents the results of modal analysis. At 1000 rpm the mean torque and speed of the engine mount is maximum. So, the frequency at 1000 rpm is considered as working frequency which is found to be 10.28 kHz. When modal analysis is done on the engine mount, working stress corresponding to the working frequency is taken. The stress at working conditions is mainly concentrated on the circumference of the engine mount. The ultimate stress is calculated at the point where the material tends to break. The ultimate stress of fluorocarbon is found to be 49.26 MPa.

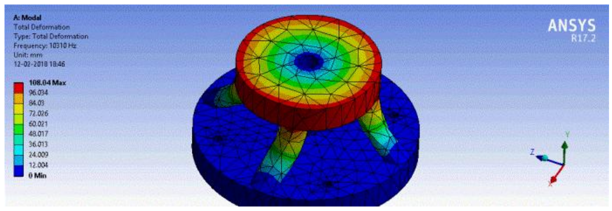

Fig. 4: Total deformation in the mount

Table 1: Results of modal analysis

rpm	Frequency (kHz)	Working stress (MPa)	Ultimate stress (MPa)	Factor of safety
1000	10.28	24.375	49.26	2.105

4. Conclusions

From the presented work, it can be observed that the designed model of the novel engine mount is very safe. This means that the model have the capacity of withstanding the vibrations produced for longer time.

REFERENCES:

[1] Y. Yu, S.M. Peelamedu, N.G.N. Rao and V. Dukkipati. 2001. A literature review of automotive engine mounting systems, *Mechanism and Machine Theory*, 36, 123-142. https://doi.org/10.1016/S0094-114X(00)00023-9.

[2] D.H. Lee, W. Schwan and C.M. Kim. 2002. Design sensitivity analysis and optimization of an engine mount system using an FRF-based sub structuring method, *J. Sound and Vibration*, 255(2), 383-397. https://doi.org/ 10.1006/jsvi.2001.4160.

[3] T. Ramachandran, K.P. Padmanaban and P. Neesamani. 2012. Modelling and analysis of IC engine rubber mounts using finite element method and RSM, *Proc. Engg.*, 3(1), 63-73.

[4] A. Adhau and V. Kumar. 2013. Design and sensitivity analysis of paramount engine system, *Int. J. Engg. Research and Tech.*, 2(11), 764-769. https://doi.org/10. 1115/1.2930238.

[5] P. Priyanka, P.M. Pawaskar, S. Patil and B. Gawas. 2015. Engine mounts optimization based on parametric approach, *Int. J. Sci. Research and Management*, 3(4), 2652-2655.

[6] T.P. Kamble and R.A. Bhalerao. 2016. Optimization and modal analysis of engine mounting bracket for different materials by using finite element analysis, *Int. Research J. Engg and Tech.*, 3(10), 120-125.

[7] V. Velmurugan, T.S.A. Suryakumari, A. Pandiyan and V. Jayakumar. 2016. Modelling and analysis of semi-active hydraulic engine mount using finite element analysis, *ARPN J. Engg and Applied Scis.*, 11(16), 10097-10102.