IJREAT International Journal of Research in Engineering & Advanced Technology, Volume 2, Issue 1, Feb-Mar, 2014
ISSN: 2320 - 8791
www.ijreat.org

Design and Analysis Of Mechanical Evaluation System For Vibration Absorbers

G.NagaBabu[1], D.Devsingh[2]

[1,2] **Department of mechanical engineering,**
CMR Institute Of Technology, Medchal, Hyderabad, India

Abstract

Automobile vehicles have to pass over rough roads during their course of use. Because of road roughness vibration will be induced which will reach passenger through tyres and vehicle chassis. If this vibration crosses the tolerable limits of human being (Passenger) then he will experience discomfort. To avoid this situation vibration absorber will be installed between tyre and chassis. Vibration absorber is supposed to reduce the vibration coming from the road before it reaches the passenger. Functional effectiveness of this vibration absorber needs to be ascertained before it is installed in the vehicle. For this an evaluation system is required over which vibration absorber has to be placed and this system should give vibration from bottom side of the vibration absorber. If the vibration absorber reduces the vibration to 10% of that comes from the road, then vibration absorber can be called effective. This project aims at design of such evaluation system for testing the effectiveness of vibration absorber. This evaluation system will make use of rotating unbalance for generating vibration. Initially configuration of the intended system will be framed in which all the necessary subsystems will be identified. Then all these subsystems will be designed and their dimensions will be obtained. All these dimensional models will be transformed to solid modeling using SOLIDWORKS software. Then structural analysis of the intended system will be carried out using Finite Element Method (FEM) and the design will be checked for its adequacy. Design will be finalized based on the outcome of analysis results.

Keywords: *Design, Simulation, Evaluation system, Finite Element Analysis.*

1. Introduction

Studies have been conducted to investigate the cause behind these accidents. These studies revealed that the psychological stability of riders of the vehicle which is essential to take right decision quickly without delay to avoid accident is getting influenced by the vibration that is passing on to rider. It was also observed that this vibration is getting transmitted to the rider mainly from three different parts of the vehicle which are pedals (Clutch, brake, etc.), seat and the steering. Simulation techniques presently in use will vibrate the entire vehicle and see whether all parts of the vehicle are intact or not. An unique way of testing will be possible using the design that will be brought out as an outcome of this project. The intended system will provide a simulator which will consists of seat, pedals and steering. In total it simulates the automobile vehicle. Three mechanical exciter systems will be positioned beneath seat, pedals and steering. These mechanical exciter systems will be useful for exciting each of which. Design of such mechanical exciter system is taken up in this project. To begin with a preliminary design configuration is arrived at. Then after identifying the critical elements of the configuration, each individual component is designed in detail based on machine design aspects. Once after deriving the dimensional models, all components are modeled in 3D CAD software and integrated to get model of the assembly. This configuration is subjected to kinematic simulation studies using kinematic simulation software and ensured proper functioning of the designed configuration (Generation of vibration). Then the design is fine tuned to achieve the desired specification with additional margin. Then structural analysis is carried out against the functional loads in order to assess the deign adequacy.

2.Structural design of mechanical evaluation system

Based on the design philosophy a configuration is identified. All the subsystems of the configuration are to be designed taking functional loads into account. While designing the subsystems various mechanical design aspects are considered. The outcome of the structural design would be dimensional models of all subsystems of the intended system. The total design process is concluded with mention of available factor of safety for the design

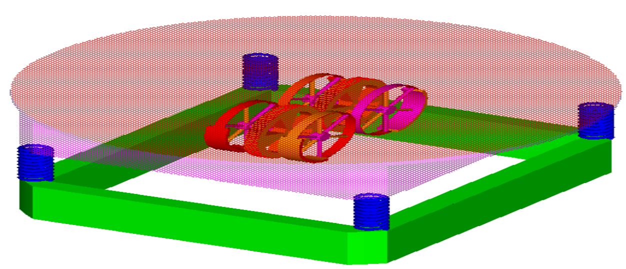

Fig 1 Solid model of the assembly

3.Finite Element Analysis

Geometry of the mechanical evaluation system for shock absorbers is built up in 3-D CAD software. The mechanism basically consists of two sets of eccentric rotors. As both of them are dynamically identical, one set of eccentric rotor comprises of shaft, two discs (Disc –1 & Disc – 2) and gear – 1 are considered for the analysis

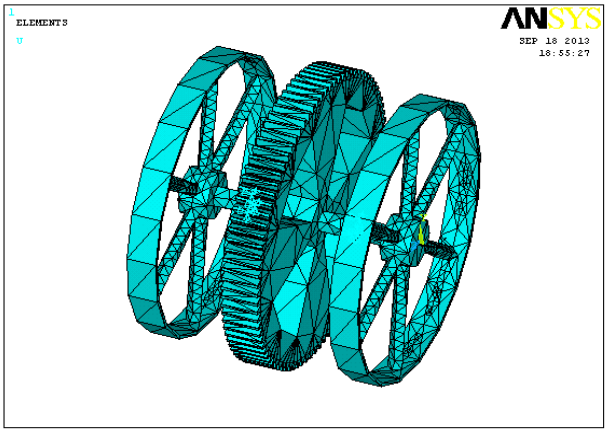

Fig 2 FE model of the mechanical evaluation system for shock absorbers

The FE model of the mechanical evaluation system for shock absorbers is constrained at two locations where bearings are provided on the shaft, The FE model is then solved for Von Misses stress and displacement using ANSYS software. Maximum stress plot is shown in Figure 3 in which maximum stress location is visible in red color.

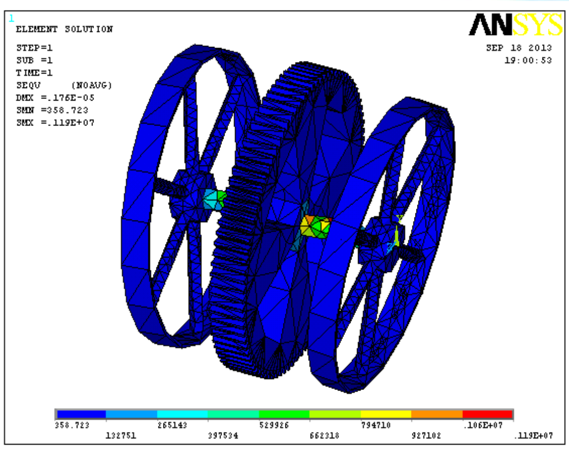

Fig 3 Stress plot

Modal analysis is the study of the dynamic properties of structures under vibration excitation. Modal analysis is the field of measuring and analyzing the dynamic response of structures and or fluids when excited by an input. Examples would include measuring the vibration of a car's body when it is attached to an electromagnetic shaker, or the noise pattern in a room when excited by a loudspeaker.

In structural engineering, modal analysis uses a structure's overall mass and stiffness to find the various periods that it will naturally resonate at. These periods of vibration are very important to note in earthquake engineering, as it is imperative that a building's natural frequency does not match the frequency of expected earthquakes in the region in which the building is to be constructed. If a structure's natural frequency matches an earthquake's frequency, the structure could continue to resonate and experience structural damage.

Although modal analysis is usually carried out by computers, it is possible to hand-calculate the period of vibration of any high-rise building by idealizing it as a fixed-ended cantilever with lumped masses.

A modal analysis calculates the undamped natural modes of a system. These modes are given in decreasing order of period and are numbered starting from 1.

The analysis calculates the natural modes of the discretised model, not those of the real continuous system. However the discretised modes are close to the continuous ones and for a mode number the accuracy improves as more and more elements are used to model the system. For any given level of discretisation the accuracy is better for the lower modes and progressively worsens as you go to higher and higher modes. The highest numbered modes are unlikely to be realistic since they are oscillations whose wavelengths are of the same order as the segment length.

Same FE model used for static analysis is extended for modal analysis. The list of frequencies are given below.

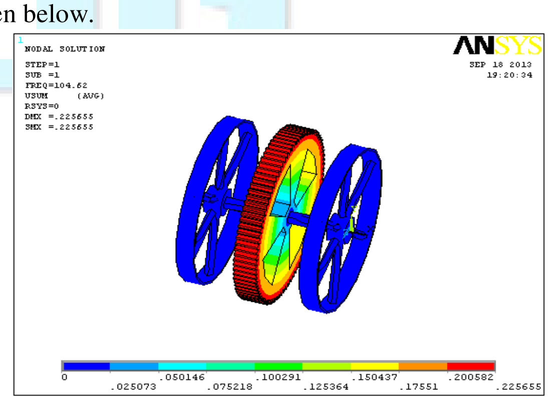

Fig4 Mode shape

4 Results

Results of structural analysis carried out using Finite Element Method (FEM).

STATIC ANALYSIS		Allowable stress=140 MPa	
		Maximum stress	**FOS**
1.	Self weight	1.2 MPa	> 10
2.	Functional loads	18 MPa	7.7
DYNAMIC ANALYSIS		Critical frequency = 10 Hz	
1.	First natural frequency	104.6 Hz	

5. Conclusions

Design of a mechanical evaluation system for shock absorbers is done which can simulate vibration on shock absorbers of automobile vehicles.

Static analysis – Self weight

Maximum Von Misses stress is observed to be 1.2 MPa.
Available factor of safety is observed to be (>10) by comparing the maximum stress with that of allowable stress (Yield) of steel material i.e. 140 MPa.
As the available factor of safety (>10) is more than minimum desired factor of safety (1.5) the design is safe.
Maximum displacement is observed to be 0.00176 mm.

Static analysis – Functional load

Maximum Von Misses stress is observed to be 18 MPa.
Available factor of safety is observed to be 7.7 by comparing the maximum stress with that of allowable stress (Yield) of steel material i.e. 140 MPa.
As the available factor of safety (7.7) is more than minimum desired factor of safety (1.5) the design is safe.
Maximum displacement is observed to be 0.0268 mm.

Modal analysis

Frequency of the intended system corresponding to first bending mode is found to be 104.6 Hz.
This first natural frequency is much higher than the critical frequency associated with functioning i.e. 10 Hz.

6.References

1. M.S.M.Sani,et.al.,, "Study on Dynamic Characteristics of Automotive Shock Absorber System", Malaysian Science and Technology Congress, MSTC08, 16~17Dec, KLCC, Malaysia, 2008.
2. Yan Cui, "Testing and Modeling of Nonlinear Properties of Shock Absorbers for Vehicle Dynamics Studies", Proceedings of the World Congress on Engineering and Computer Science 2010 Vol II WCECS 2010, October 20-22, 2010, San Francisco, USA.
3. Shaohua Li, "Dynamical Test and Modeling for Hydraulic Shock Absorber on Heavy Vehicle under Harmonic and Random Loadings", Research Journal of Applied Sciences, Engineering and Technology 4(13): 1903-1910, 2012.
4. Krzysztof Parczewski, "Exploration Of The Shock-Absorber Damage Influence On The Steerability And Stability Of The Car Motion" Journal of KONES Powertrain and Transport, Vol. 18, No. 3 2011.
5. I. Voicea, "Dynamic Testing Of Shock Absorbing Systems Using A Specialized Stand", UGC 631.2 technical note, 2010.
6. McClintock and Argon, Mechanical behavior of materials, Addsion-Wesley, Reading, MA, 1966.
7. S. L. Hoyt, Metals and alloys data book, Reinhold, New York, 1943.
8. C. Lipson and R. Junival, Handbook of stress and strength, Macmilan, New York, 1963.
9. F. R. Shanley, Strength of materials, McGraw-Hill, New York, 1957.
10. Tirupathi R. Chandrupatla and Ashok D. Belegundu,
11. Introduction to Finite elements in engineering.
12. ANSYS user manual