International Journal of Engineering Research
Volume No.3 Issue No: Special 1, pp: 183-186
ISSN:2319-6890)(online),2347-5013(print)
22nd March 2014

Structural Design of Attitude Simulator for Performance Evaluation of Automobile Systems

Dr.G. Janardhana Raju[1] G.Guruvaiah Naidu [1] K.Suresh Kumar[1]
[1]Department of Mechanical Engineering , *Nalla Narasimha Reddy Educational Society Group of Institutions, Ghatkesar, Hyderabad.*
Email:gjraju_06@rediffmail.com; mignonsuresh@gmail.com

Abstract— **Performance of automobile transmission systems is evaluated at different attitudes of the engine in order to simulate dynamic acceleration inputs. Dynamic test rigs presently being used for this purpose are of very large in physical size and not readily adaptable for use in the confines of conventional engine test systems. More over these systems makes use of hydraulic power, which is not well adapted to an engine test system environment because of lack of centralized hydraulic power in these systems and potential problems with the use of high-pressure hydraulics in a high temperature environment where mechanical damage to hydraulic power is possible. To get rid of the problems with conventional test rigs, design of power train attitude simulator is taken up in this project. This simulator can be comfortably used for testing engines, transmissions, gearboxes and etc. The intended system includes inner and outer frames that are rotated independently by gear reduced electric motors. The end result of this project would be a simulator, which provides both static and dynamic testing of automobile transmission systems during their operation in varying attitudes.**

Keywords— Automobile Transmission Systems, Hydraulics and Simulator

I. Introduction

Automobile transmission systems are to be evaluated for their expected performance at different attitudes of the engine in order to simulate dynamic acceleration inputs. Dynamic test rigs presently being used for this purpose are of very large in physical size and not readily adaptable for use in the confines of conventional engine test systems. To get rid of the problems with conventional test rigs, design of attitude simulator is taken up in this project. This simulator can be comfortably used for testing engines, transmissions, gear boxes, etc. The intended system includes inner and outer frames that are rotated independently by stepper motors through specified angle. The outer frame will be supported by two support stands, which will be secured to a base or a test bed. The proposed design makes use of stepper motors and gimbaled frame layout enables the size of the simulator to be minimized to make it possible for its use in conventional engine test systems. In addition to this the use of electric power avoids the problems of possible hydraulic leaks during engine operation. More over this sophisticated simulator enables controllability by a computer which can be programmed to simulate a series of dynamic acceleration inputs, such as would be encountered during on road or race track vehicle maneuvers.

PRINCIPLE OF OPERATION

In this design essential element is the actuation source in two numbers for providing

angular motion to the engine in two mutually perpendicular directions for meeting which stepper motor has been chosen. Stepper motor is chosen in contrast with normal motor because normal motor can provide continuous rotation. But what exactly needed in the proposed design is step wise angular motion for a specified angle and specified step size and specified breaking time. More appropriately saying it should be like ceiling fan regulator (Which rotates in intervals in a specified band of angle and stops and starts as and when required).

Another objective behind choosing stepper motor is that requirement of high breaking torque which means motor should provide motion and it should hold also in a specified angular position by applying a break which can be electrical. Stepper motors having high breaking torque is well suitable for this application because engine will be mounted on top of the simulator platform. Having a look at the engine dimensions one can clearly make out its height is more compared to other two dimensions. Hence the simulator platform has to suffer with high C.G. (C.G. of the engine is much away from C.G. of the mounting platform) problem imposed by the engine. Heavy weight of the engine (100 Kg) along with high C.G. causes an overturning moment to act on the entire system which tends to topple the whole simulator structure. To counter this effect stepper motor is chosen as it can inhibit the simulator not to topple. Outline sketch of the proposed design is shown in figure 1. Other than the drive source, simulator will have two supporting platforms one of which is inner platform and the other one is outer platform. Engine will be directly mounted on the inner platform. Motor 1 will drive the inner platform along with engine mounted on it for one direction motion and motor 2 will drive the outer platform along with inner platform and the engine in orthogonal direction to that of first one.

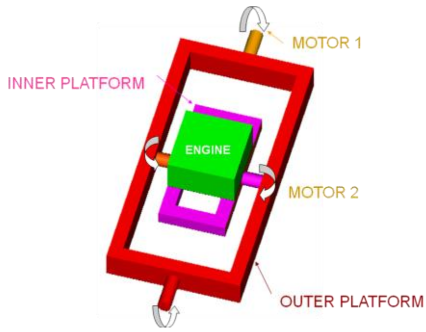


Fig No. 1 Outline sketch of the proposed design

The objective of providing two platforms is that when engine is subjected to inner most angular motion other part of the simulator should be stand still. But when engine has to experience other angular motion inner platform along with the engine should experience same motion without any disturbance. To accomplish this complex task this project is being proposed with two platforms aided by two stepper motors.

Table No.1.1 Specifications of the proposed design

S No.	Parameter	Specification
1.	Weight of the engine	100 Kg
2.	Dimensions of the engine	400 mm x 320 mm x 433 mm
3.	Number of mounting holes for the engine	6
4.	Angular rotation on either side	60^0 (1.047 rad/sec)

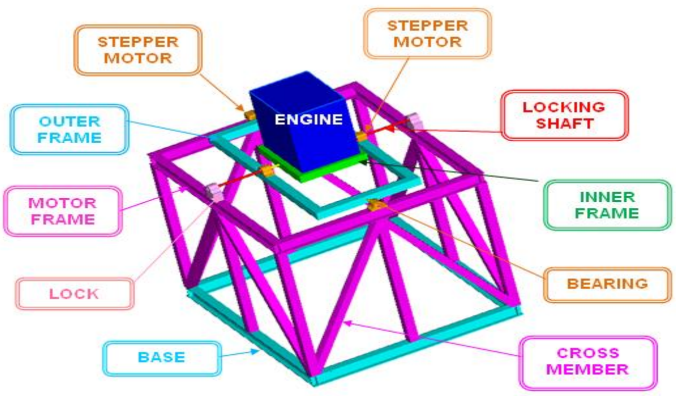


Fig No. 2 Assembly of the intended design of simulator

The primary requirement of linking the motion of two platforms for one angular motion and delinking the motion of two platforms for angular motion orthogonal to first one has been met with the help of bearings and shafts. Finally a supporting structure is required to position the simulator on the test floor and also for housing the motor and external bearings. In addition to this supporting structure has to provide sufficient rigidity for the simulator as the nature of loads that will be encountered will be dynamic. The detailed specifications of proposed design are given in Table 1.1.

Structural Design Of Attitude_Simulator

The objective of the project is to evolve the design of a test platform which simulates the attitude of an automobile vehicle during which performance evaluation can be done. To begin with a basic configuration has been evolved as discussed, to meet this requirement while identifying its subsystems. Design inputs as per Table 1.1 along with design constraints have been compiled. Then all the subsystems have been designed having an aim of arriving at dimensional configuration for each subsystem. The design of attitude simulator was done by CATIA V4 software.

DESIGN CONSTRAINTS

The size of the simulator should not exceed 1.5 m x 1.5 m x 1.5 m.

Minimum desired factor of safety for the design should be 2.

First natural frequency of the system should be $\geq$ 2 x operating frequency (i.e. 0.17 Hz).

Subsystems of The Simulator

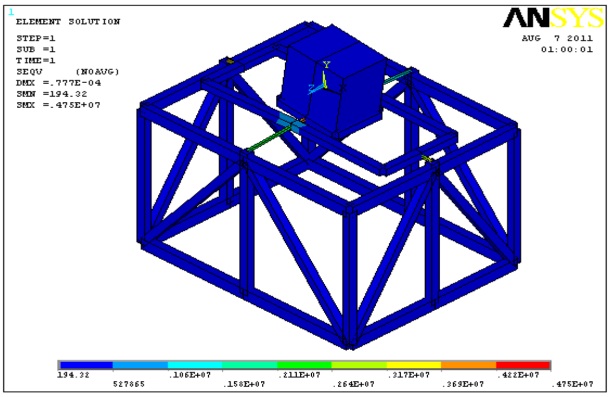
The following subsystems are identified which as a whole will constitute the simulator.

- Inner frame
- Outer frame
- Stepper motor
- Inner frame shaft
- Motor frame
- Locking shaft
- Outer frame shaft
- Bearing
- Simulator base

With the dimensional models solid modeling is done and assembled as shown in fig no.2. The solid modeling assembly is carried out with the help of the sub system components, where all the critical subsystems of the simulator are designed based on stress. The design adequacy has been ascertained by calculating the maximum stress and comparing with allowable stress for each subsystem. Maximum stress calculated theoretically is compared with that of obtained from FEM. The results obtained using analytical method is compared with that of FE method in order to assess the confidence associated with that of analytical method.

III. Results and Tables

Structural Analysis Of Attitude Simulator Using Finite Element Method (FEM)



Structural analysis of attitude simulator is carried out using Finite Element Method (FEM) against the functional loads in order to assess the design adequacy. To begin with structural analysis is carried out against angular disturbance that will be experienced by the intended system during its course of application. Then the analysis is extended for modal analysis followed by harmonic analysis. Finite element analysis gives out the maximum stress developed in all the subsystems. Based on the analysis results the available factor of safety estimated during design will be validated. The outcome of the analysis would be useful in ascertaining that the design is safe. For carrying out structural analysis the all the subsystems of the intended system are considered. The basic approach followed in finite element analysis is represented in form of flow chart shown in figure no.3.

Table No.1.2 Material Properties

S.No	Material property	Value
1.	Young's modulus (E)	2.1 e 11 Pa
2.	Density (ρ)	7850 Kg/m^3
3.	Poisson's ratio (μ)	0.25

The following structural analyses are carried out.

- Static analysis – Against the functional load of 1.047 rad/sec.
- Modal analysis (Free vibration analysis).
- Harmonic analysis.

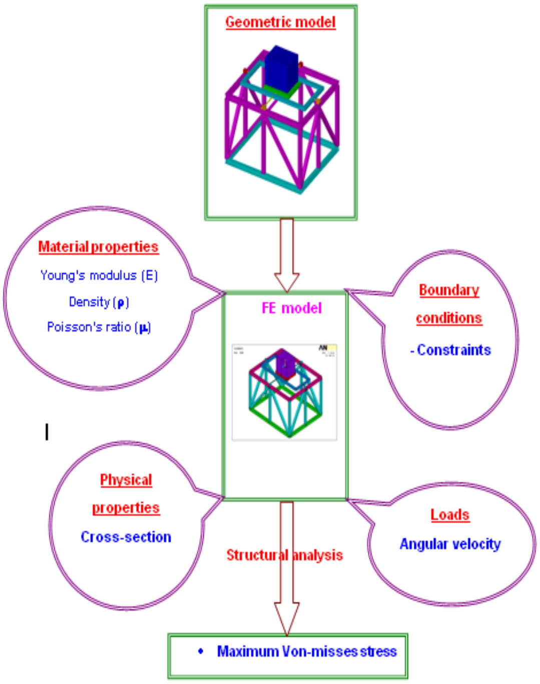


Fig No. 3 Flow Chart of Finite Element Analysis

Results And Discussion

Static analysis is carried out against the functional load of 1.047 rad/sec and the FE model is solved for maximum Von Misses stress in ANSYS software. Maximum stress plot for the entire system is shown in Figure 4.

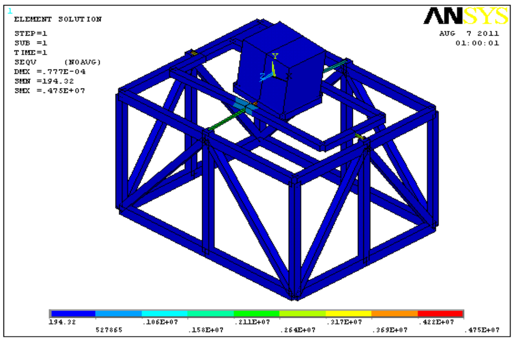


Fig No.4. Maximum stress plot for the entire system

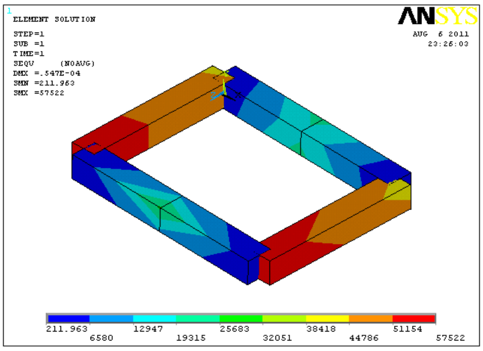


Fig No.5. Maximum stress plot of inner frame

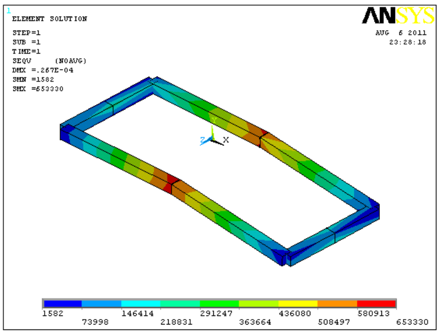


Fig No.6. Maximum stress plot of outer frame

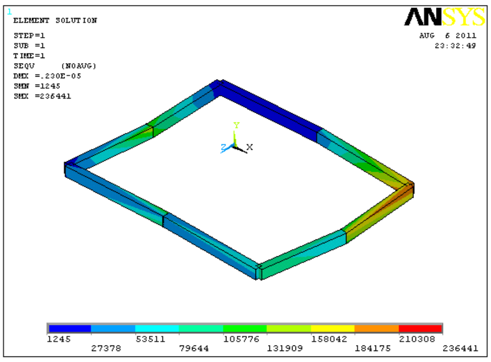


Fig No.7. Maximum stress plot of motor frame

International Journal of Engineering Research
Volume No.3 Issue No: Special 1, pp: 183-186
ISSN:2319-6890)(online),2347-5013(print)
22nd March 2014

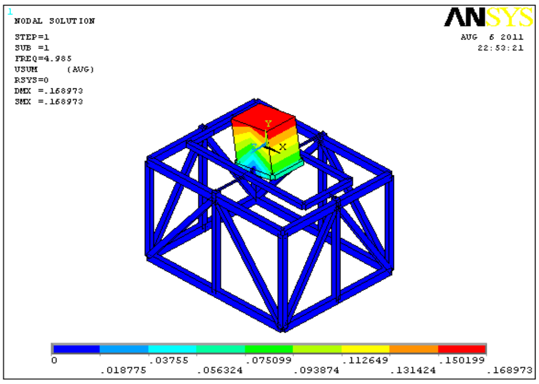


Fig No. 8 First mode shape of the system

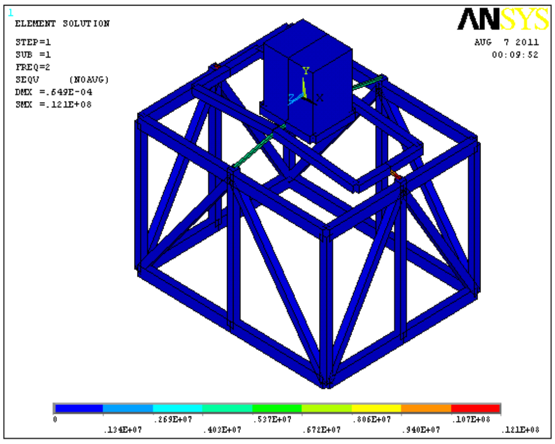


Fig No.9 Dynamic bending stress plot

The objective of harmonic analysis is to assess the dynamic performance of the system at its first natural frequency. Maximum dynamic bending stress is observed to be 12 MPa and this maximum stress is concentrated at outer frame shaft. Available factor of safety is observed to be > 10 against the allowable stress (Yield stress) for steel (200 MPa).

IV. Conclusion

- An attitude simulator is designed which will be useful to simulate various attitudes of an automobile engine during its course of application and there by enables to check the performance of various internal subsystems like lubrication circuit, etc.
- Uniqueness of this project is implementation of stepper motor in the design which provides smooth control of angular orientations for the engine.
- All the subsystems are designed using theoretical method and the design calculations are validated using FEM. The degree of closeness between the results obtained using theoretical method and FEM indicated the confidence associated with the design.
- The design has been done for a engine weight of 100 Kg in the present project.
- Outcome of modal analysis conveys a message that there is no interference between first natural frequency of the structure and operating frequency.

References

i. G. C. Viot, Mechanics of structures, Janes journal, 1992.
ii. George B. Bauer, Selection criteria for standard structures, AIAA journal, 1997.
iii. M. J. Wictor,et al., Stress analysis of structural elements, ASME journal, 1995.
iv. L. Lewis, Design of load supporting platform for containers, AIAA journal, 1997.
v. Tirupathi R. Chandrupatla and Ashok D. Belegundu, Introduction to Finite elements in engineering.
vi. Cook, R. D., Finite element modeling for stress analysis, New York: Wiley, 1995.
vii. Bathe, K. J., Finite element procedures in engineering analysis, Englewood Cliffs, NJ: Prentice Hall, 1981.
viii. O.C. Zienkiewicz and R.L. Taylor, The Finite Element Method, McGraw-Hill.
ix. ANSYS user manual