International Conference on Ideas, Impact and Innovation in Mechanical Engineering (ICIIME 2017) ISSN: 2321-8169
Volume: 5 Issue: 6 577 – 581

Analysis of Diesel Generator Control Panel using Finite Element Method

Amar R. Kadam [1], Dr.Sanjay B. Zope[2]

[1]Student, *Sahyadri Valley College of Engineering & Technology, Rajuri, Pune,* kadamamar167@gmail.com
[2] Dr.Sanjay B. Zope, Guide, *Sahyadri Valley College of Engineering & Technology, Rajuri, Pune*

ABSTRACT

Now a days Diesel generator set have become basic need of the modern world for constant electric supply or for power back up solutions. Generally a DG set runs between 1500 rpm to 3500 rpm to maintain a constant electrical frequency of 50 Hz. Many vibration issues are related with the DG set. The vibration level becomes severe when the engine runs at highest speed. Control Panel is one of the key elements of DG set. The control panel covers some of the intricate components and electrical circuits which is hard to replace frequently. The main objective of this paper is to discuss the design of a control panel for a generator and optimize its vibration to avoid resonance.
***Keywords**: Control panel, DG Set, Resonance.*

I. INTRODUCTION

Control Panel is vital component of a DG set. The control panel should be able to sustain the vibrations generated by the Generator. The control panel box should be designed to avoid the resonance .The present thesis focuses on the increase in natural frequency of the control panel box beyond the frequency of engine vibrations. There are many methods to do this. In this paper we are dealing with FEM.
The analysis approach is different based on whether the control panel is rigid or relatively flexible. For the given maximum speed of 3500 rpm, the frequency of vibration is maximum Hence 60 Hz is designated as threshold frequency. If the lowest natural frequency of the system is more than the threshold frequency, the system is considered as rigid. When the lowest natural frequency of system falls below the threshold frequency; the system is considered as flexible. For flexible system a dynamic analysis approach need to be adopted.
The model was created using SOLID WORKS. Then the model was imported to ANSYS 16.0 and then it was analysed.

II. PROBLEM STATEMENT

A Diesel generator set includes diesel engine, alternator and control panel mounted on a common base frame. The control panel is less rigid in comparison with generator and alternator. With higher engine speed mechanical vibrations are also higher, due to its speed dependency. The objective of the study is to design the control panel to meet required norms without resonance.

III. METHODOLOGY

The 3D model of the control panel assembly is created using Solid Work. A Finite Element Model was built with-as element.
Finite element analysis was carried in the following steps:
1. Static (dead weight) analysis of Control Panel assembly.
2. Modal analysis for first six natural frequencies.
3. Response spectrum analysis.
4. Harmonic Analysis.

TABLE I

MATERIAL PROPERTIES

Material Specification	Structural Steel
Density	7850 kg m^{-3}
Young's Modulus	2.0 x 10^{11}
Poisson's Ratio	0.3
Yield Strength	2.5 x 10^{08}
Specific Heat	434 kg^{-1} C^{-1}

1. Static Structural Analysis

The static structural analysis is used to determine the effect of static load on a structure. Static Structural Analysis is carried out for all structures used to carry the loads such as vehicles, machinery etc. Static Structural Analysis is incorporated to compute a structures deformation, internal stresses, support reaction, acceleration and stability. The boundary conditions are

1) Top end of the control panel is fixed.
2) The acceleration is equal to acceleration due to gravity.
3) The force on control panel through centroid in downward direction is 10 N.

The results obtained from Static Structural Analysis are tabulated below

TABLE 2:
RESULTS OF STATIC STRUCTURAL ANALYSIS

Entity	Value
Maximum Total Deformation	3.4125 x 10^{-5} m
Maximum Equivalent Stress	3.2735 x 10^{6} Pa
Maximum Equivalent Strain	1.7603 x 10^{-5} m/m

2. Modal Analysis

Modal Analysis includes study of dynamic properties of a structure under vibration. The Dynamic response of a structure which is under vibration is measured and analysed by Modal Analysis. In modal analysis pre-stressed static structural environment is considered. Modal analysis was carried out to determine the natural frequencies and mode shapes of a structure.

The following bar chart indicates the frequency at each calculated mode.

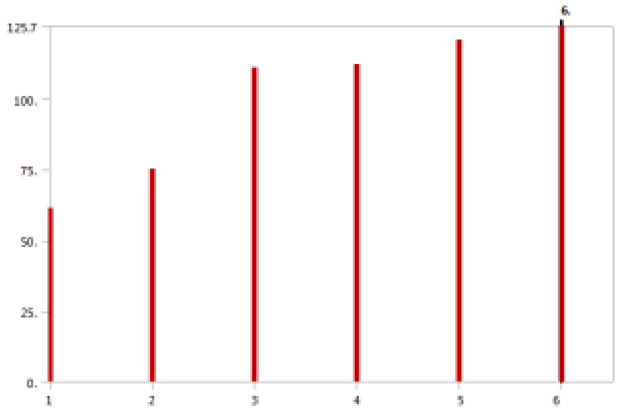


Fig 1: Frequency vs Mode

ISSN: 2321-8169
577 – 581

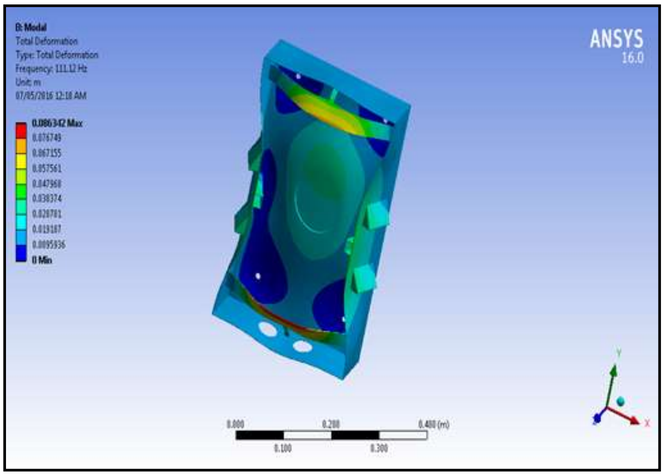


Fig 2: Total Deformation

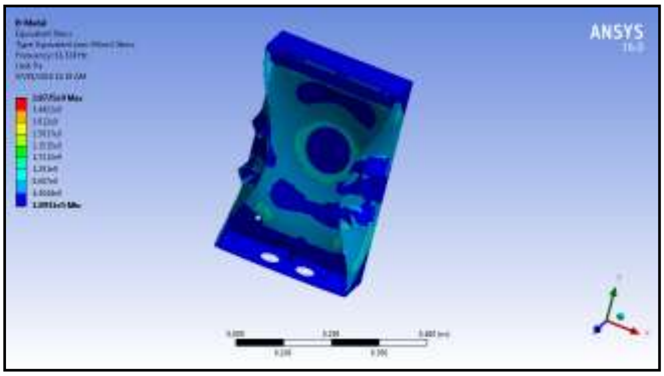


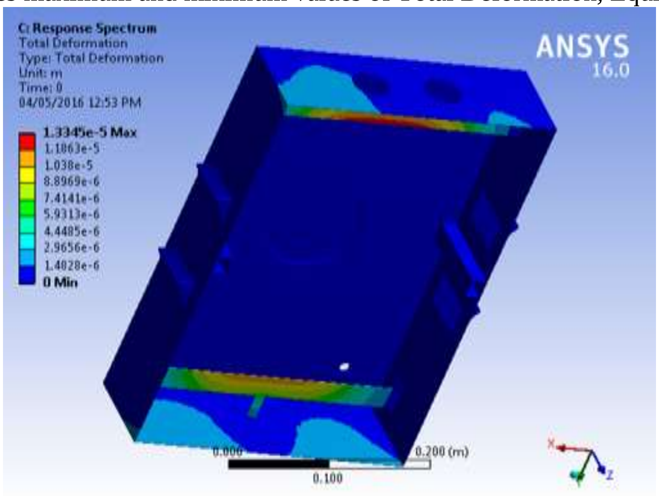
Fig 3: Equivalent Stress

In modal Analysis maximum deformation observed is 8.6342 x 10^{-2} m, whereas the maximum equivalent stress is 3.8725 x 10^{9} Pa.

3. Response spectrum Analysis

Response spectrum analysis is simply a plot of the peak or steady state response (displacement, velocity or Acceleration) of a series of oscillators that are forced into motion by same base vibration.

The response spectrum analysis gives maximum and minimum values of Total Deformation, Equivalent Stress and Elastic Strain.



ISSN: 2321-8169
577 – 581

Fig 4: Total Deformation (Maximum Total Deformation 1.3345 x 10^{-5} m)

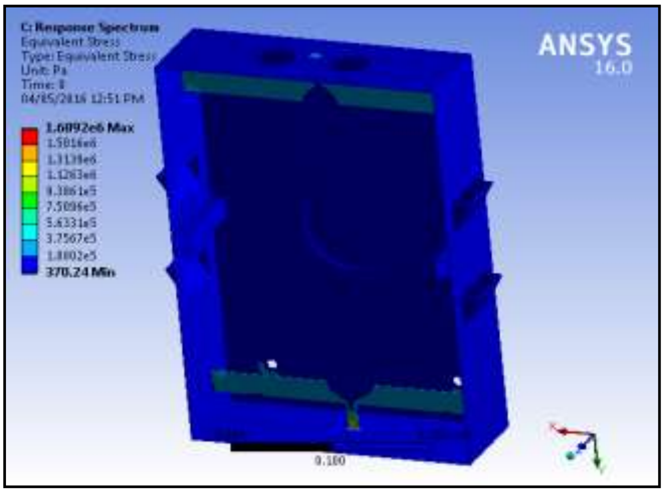


Fig 5: Equivalent Stress (Maximum Equivalent Stress is 1.6892 x 10^{6} Pa.)

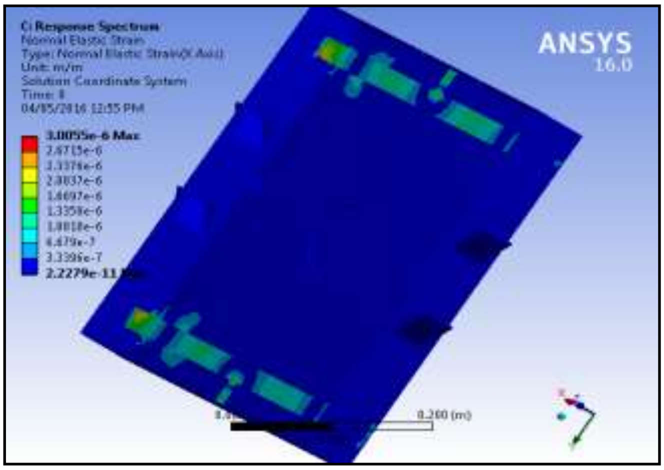


Fig 6: Normal Elastic Strain (The Maximum Normal Elastic Strain is 3.0055 x 10^{-6} m/m.)

4. Harmonic Analysis

Harmonic Analysis is concerned with the representation of functions or signals as the superposition of basic waves and the study of generalization of the notions of Fourier series and Fourier transform.

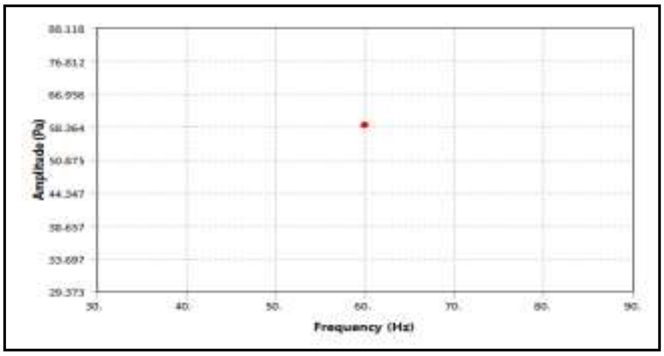


Fig 7: Stress vs Frequency Plot

For a frequency of 60 Hz, Maximum Normal Stress is 58.746 Pa.

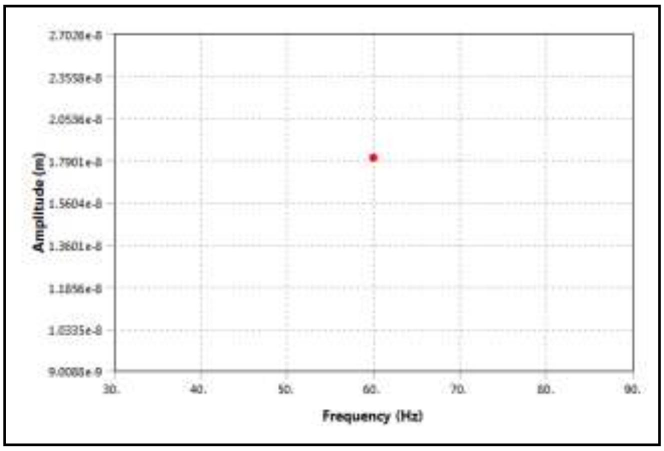


Fig 8: Amplitude vs Frequency Plot

For a frequency of 60 Hz, Maximum Deformation is 1.8018×10^{-8} m.

IV. RESULTS AND DISCUSSIONS

The results obtained are tabulated for first six modes of vibration for the analysis of Control Panel box.

TABLE 3
MODES AND FREQUENCIES

Mode	Frequency [Hz]
1.	61.324
2.	75.329
3.	111.12
4.	112.4
5.	120.57
6.	125.7

VII. CONCLUSION

The minimum natural frequency obtained from finite element analysis of control panel box is found to be greater than the system frequency of vibration due to external excitation. This avoids the resonance of control panel which leads to the satisfactory design of the control panel box.

VIII. FUTURE WORK

Validation of the results obtained by FEM Method with experimental results obtained on FFT Analyser.

REFERENCES

[1] S. Dubey (2015, June), Mechanical Vibration Analysis of HVAC System and its Optimization techniques *Krishi Sanskriti Publication*. [Online].*2*, 72-82. Available: http://www.krishisanskriti.org/AREEE.html
[2] K.Satyavati, N.Kumar (2013, November) Finite Element Analysis of Control Panel Assembly for Earth Quakes. *International Journal of Emerging Technology and Advanced Engineering. 2(11)*
[3] P.K Jain (2013, June), Noise Reduction of 3000 rpm Diesel Genset through Design Optimization of Canopy. *Global Research Analysis. 2(6)*
[4] A. Kadam, P. Hujare (2014, June.). Optimization Segmented Constrained Layer Damping. *IJEAT.3(5)*.
[5] A.Montazeri, J.Poshtan, A.Y.Komi (2008, July.). The use of Particle Swarm to optimize the Control System in a PZT Laminated Plate. *IOP Publishing*. [Online], Available: stacks.iop.org/SMS/17/045027