Seismic Analysis of Electronic Cabinet using ANSYS

M. Waqas Anjum[1], Basit Khalid[2] and Abdul Rehman[3]

Abstract

Structural analysis using Finite Element Method gives a preliminary assessment of dynamic behavior of mechanical structures in order to establish the guidelines for designing the experiment. The main objective of this study was to develop a methodology of seismic analysis of electronic cabinets using ANSYS. Generally, two approaches i.e. transient analysis and response spectrum analysis are adopted for the seismic analysis in ANSYS. Since, the given loading was in time domain, therefore, transient structural analysis was preferred for the detailed dynamic investigation of electronic cabinet. The cabinet was modeled with Beam188 and its modal analysis was performed rendering the fundamental natural frequency of 8.94Hz, sufficiently close to the experimentally measured frequency of 8.33Hz. Subsequently, Transient analyses were performed for horizontal components of ground motion that resulted in peak displacement of 37mm and 28mm at the top of the cabinet which were in good agreement with experimentally measured maximum displacement of 40mm and 30mm at the same location, respectively. In addition, peak accelerations obtained from transient analyses were 0.6g and 0.82g while the experimentally measured peak accelerations were 0.8g and 0.89g respectively. It can be concluded that transient analyses in ANSYS produced equivalently reliable assessment of dynamic behavior of electronic cabinets.

Key words: Transient analysis, Modal analysis, Structural analysis, Seismic analysis.

Introduction

Locally, one of the departments of Pakistan Atomic Energy Commission (PAEC) was to install various electronic cabinets at some nuclear facility situated in seismically active zone. It is a common practice to conform the functionality and integrity of structures experiencing strong ground motions before their field installation. Therefore, seismic analysis of these electronic cabinets becomes necessary. Seismic analysis is a subset of structural analysis that is used for the calculation of the response of a structure to earthquakes. It gives complete picture of the behavior of structure subjected to seismic event. It is part of the process of structural design, earthquake engineering or structural assessment and retrofit in regions where earthquakes are frequently prevalent. Earthquake Engineering Centre (EEC) of Department of Civil Engineering, UET Peshawar was contacted for experimental testing and seismic qualification. A series of shake table tests of these cabinets were performed at EEC for various time histories and the performance of these cabinets were found satisfactory under the given conditions Ali M. (2011).

It is a common practice that numerical simulations are performed before experimental testing of prototype structures in order to establish the guidelines of designing the experiment. This also serves to save time and cost involved in experimentation. In this work, the availability of experimental data enables us to validate the simulated results when compared with experimental measurements in order to establish an equivalently reliable approach for experimental testing in future.

Seismic Analysis in ANSYS Finite Element Model

The finite element model of electronic cabinet was developed using beam element (BEAM188) as shown in Figure 1. BEAM188 has six or seven degrees of freedom at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. A seventh degree of freedom (warping magnitude) is optional. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications. The beam element is a one-dimensional line element in space. The cross-section details are provided separately by a section that is associated with the beam element having unique reference identity Ansys (2007). All members in numerical model are assigned ASTM A36 steel having the following properties. Modulus of Elasticity, E= 200GPa Poisson Ratio, v= 0.3 Density, ρ= 7850 kg/m3

[1,2 & 3]Instrumentation Control and Computer Complex (ICCC), PAEC, Islamabad

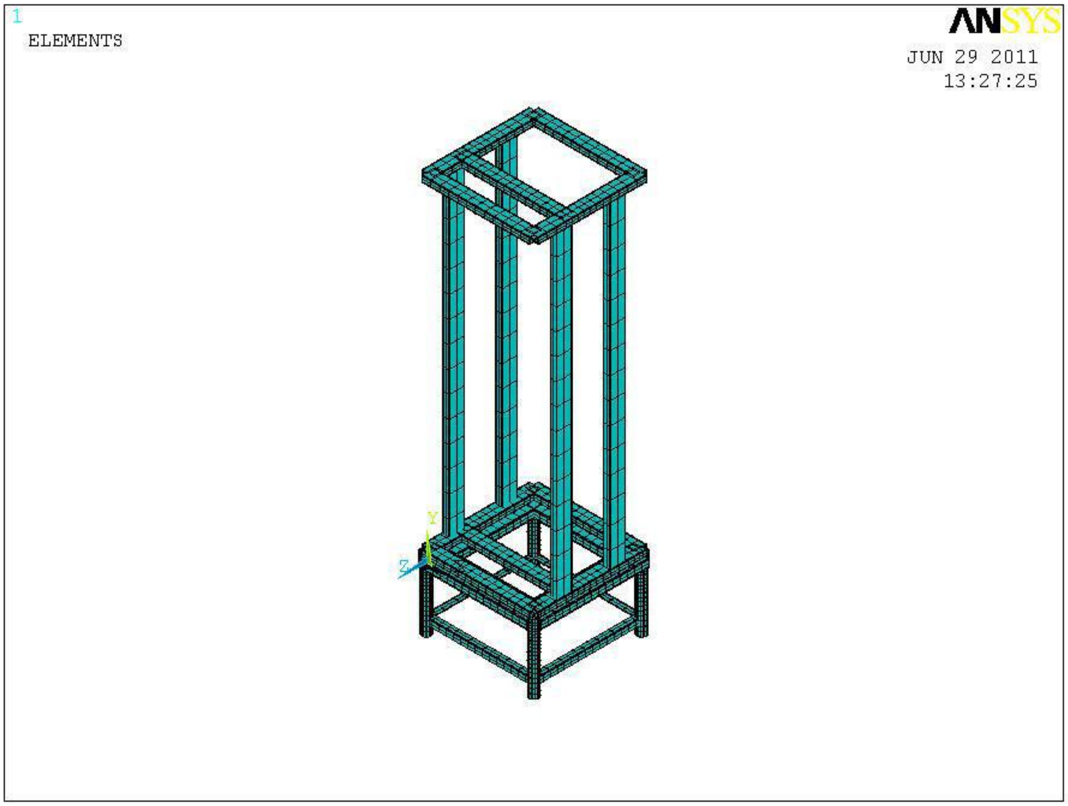

Figure 1: FE model of cabinet

The model was constrained in all degrees of freedom at its base. These boundary conditions were used to simulate the experimental situation where base frame was fixed in the concrete pad.

Modal Analysis

Modal analysis, also known as free vibration analysis, is used to determine the natural frequencies and mode shapes of a structure. Modal analysis of electronic cabinet was performed to determine the vibration characteristics, natural frequencies and mode shapes, of cabinet. Block 3 Lanczos method was used to extract the eigenvalue (natural frequency) and eigenvectors (mode shape) of the cabinet. First 10 modes of vibration of electronic cabinet were extracted and their corresponding natural frequencies are given in Table 1 below. Table 1 Natural frequency.

Table 1: Natural frequencies V/S Mode No.

Mode No.	Natural frequency (Hz)
1	8.94
2	17.09
3	24.19
4	51.44
5	57.57
6	58.46
7	59.63
8	82.86
9	88.27
10	93.75

Transient Analysis

Generally, two approaches are adopted in ANSYS for seismic analysis. They are the transient analysis and response spectrum analysis. Former approach gives detailed response of the dynamic behavior of the structure while the latter approach only gives the peak response of structure Ansys (2007). As, the seismic loading was also in time domain, therefore, transient analysis was preferred for detailed investigation of dynamic response of cabinets.

Displacement time histories recorded at concrete pad during shake table test were used as input seismic load for the transient analysis. Two displacement time histories; 1H (horizontal in direction-1 i.e. parallel to the door of cabinet) and 2H (horizontal in direction-2 i.e. orthogonal to direction-1) were used as input load at the base with a constant modal damping of 5% as specified by EEC Ali M. (2011). Both displacement time histories are given in Figure 2 and Figure 3 respectively.

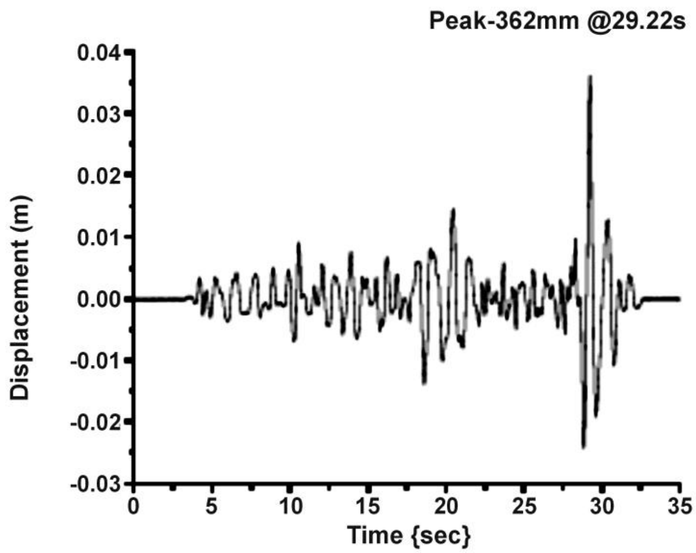

Figure 2: 1H displacement load

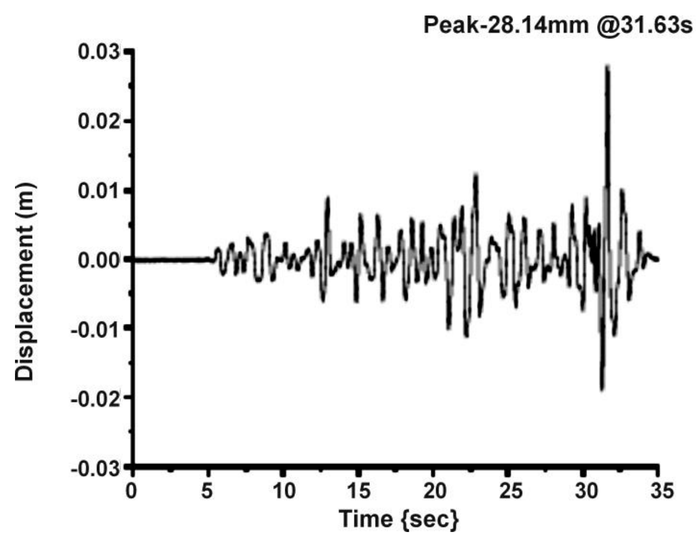

Figure 3: 2H displacement load

Results and Discussion

The fundamental frequency of the cabinet obtained from modal analysis was 8.94Hz, see Table.1, sufficiently close to the experimentally obtained frequency of 8.33Hz Ali M. (2011).

1H Loading

The maximum displacement of the cabinet was found to be 37.02mm @29.22sec at the top corresponding to the location exactly where the displacement transducer was actually installed and is shown in Figure 4. The peak acceleration was 0.59g at accelerometer location and is shown in Figure 5.

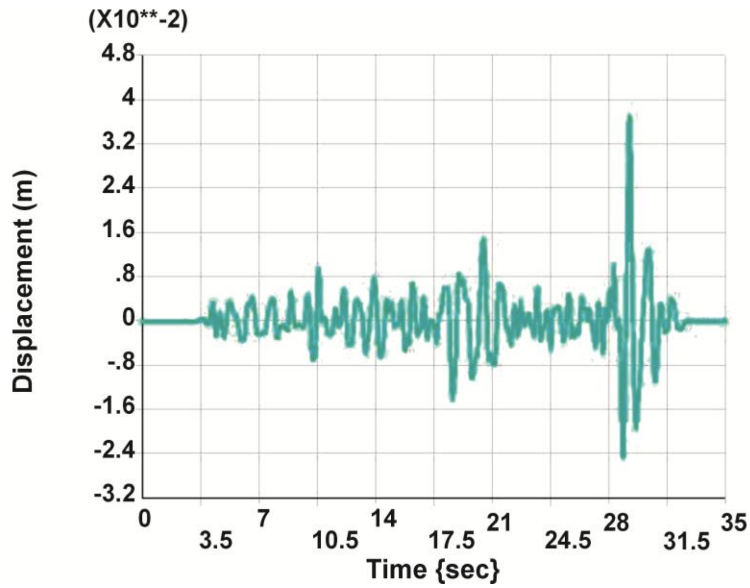

Figure 4: Displacement response corresponding to 1H

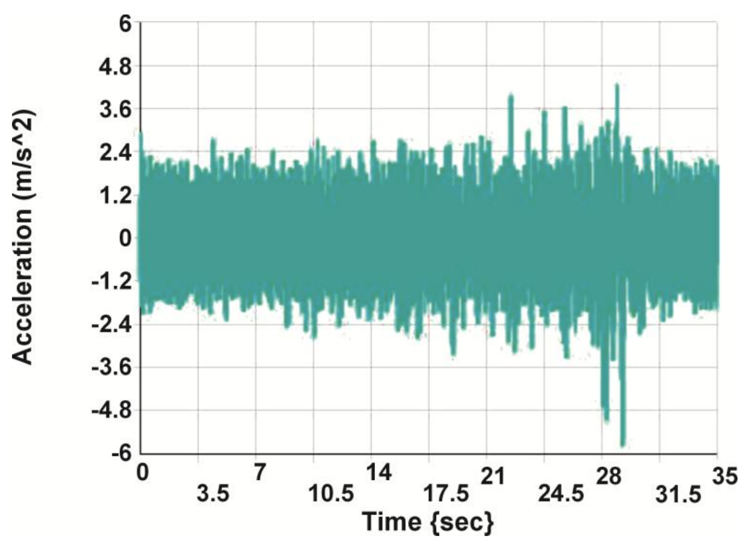

Figure 5: Acceleration response corresponding to 1H

Figure 4 shows the maximum displacement of 37.02mm@29.22sec obtained from transient analysis at the top of the cabinet (i.e. location of displacement transducer). The maximum displacement obtained experimentally was 40mm@29.22 sec, shown in Figure 6. Both the numerical and experimental displacement values were in good agreement with each other. However, peak acceleration obtained from transient analysis was 0.6g shown in Figure 5 while the experimentally measured peak acceleration was 0.8g and is shown in Figure 7.

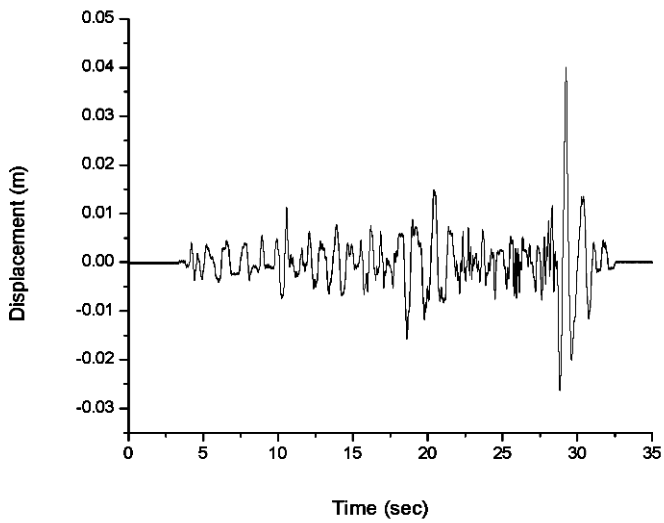

Figure 6: Displacement response (experimental)

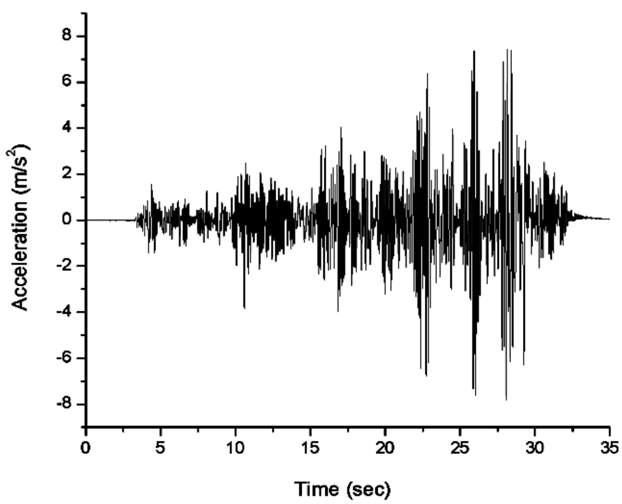

Figure 7: Acceleration response (experimental)

For 2H loading the maximum displacement of the cabinet was found to be 28.33mm @31.63sec at the top corresponding to the location exactly where the displacement transducer was actually installed and is shown in Figure 8. The peak acceleration was 0.82g at accelerometer location and is shown in Figure 9.

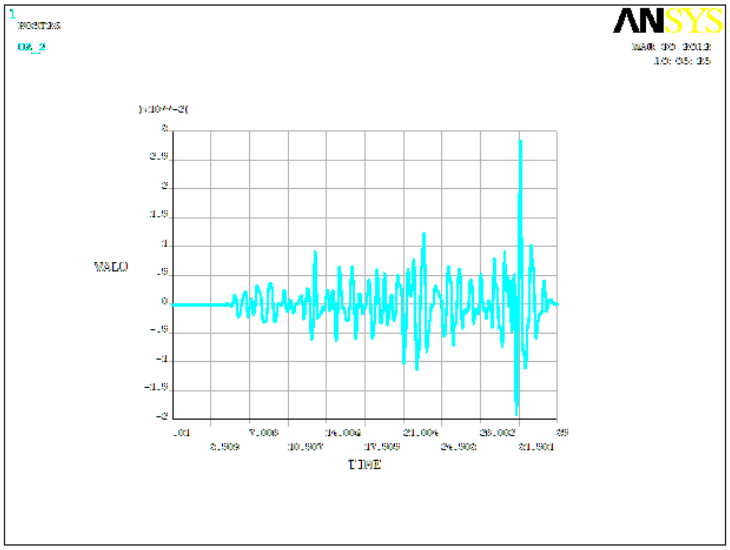

Figure 8: Displacement response corresponding to 2H

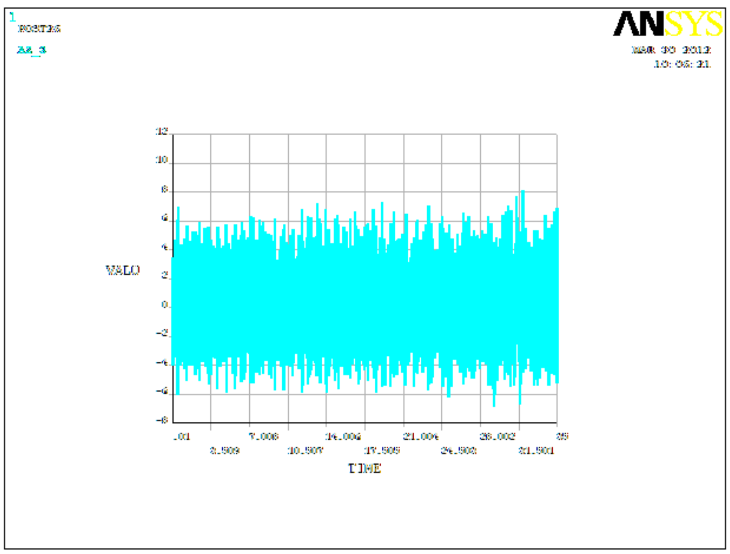

Figure 9: Acceleration response corresponding to 2H

Figure 8 shows the maximum displacement of 28.33mm @31.63sec obtained from transient analysis at the top of the cabinet (i.e. location of displacement transducer). The maximum displacement obtained experimentally was 30.44mm@31.63 sec, shown in Figure 10. Both the numerical and experimental displacement values were in good agreement with each other. However, peak acceleration obtained from transient analysis was 0.82g shown in Figure 9 while the experimentally measured peak acceleration was 0.89g and is shown in Figure 11.

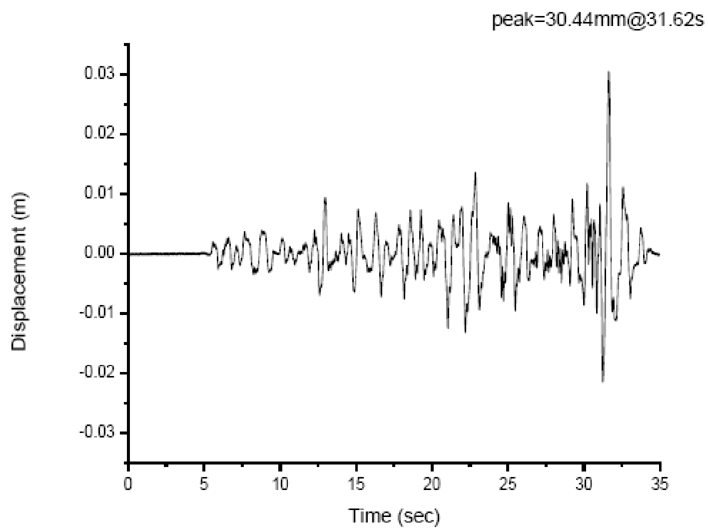

Figure 10: Displacement response (experimental)

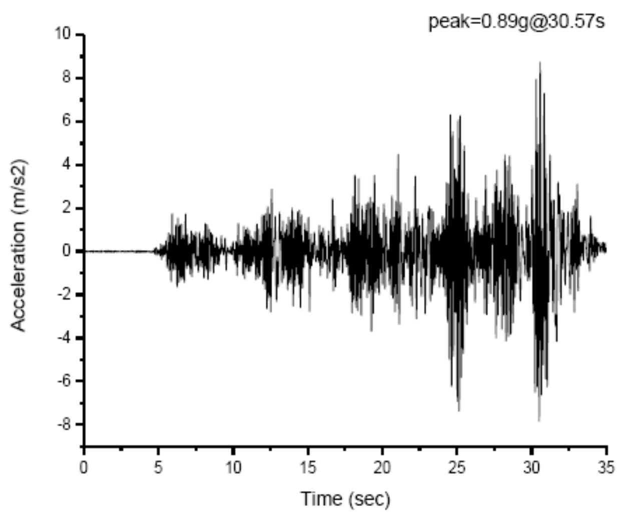

Figure 11: Acceleration response (experimental)

Conclusion

It can be concluded that seismic analysis performed in ANSYS produced equivalently reliable assessment of the dynamic behavior of electronic cabinets. Although, ANSYS simulations cannot exclude the experimentations but greatly reduce the design of experiments where time and cost do not allow experimentations.

Reference

[1] Ali M., (2011), "Seismic Qualification of Electronic Cabinets for Pakistan Atomic Energy Commission", Earthquake Engineering Centre, UET Peshawar, Final Report.

[2] ANSYS, ANSYS (2007), Structural Analysis Guide, ANSYS, Inc.