International Journal of Scientific & Engineering Research, Volume 7, Issue 10, October-2016
ISSN 2229-5518

Finite element modeling and its validation using experimental modal analysis

Hasan Zakir Jafri, Aas Mohammad

Abstract— The determination of vibration characteristics of a mechanical structure is very essential in order to predict the dynamic behavior of a mechanical system. Finite element analysis (FEA) is a well known numerical technique for determining the dynamic characterstics of any mechanical structure. However, the results obtained by FEA are often error prone due to modelling errors or incorrect boundary conditions. In the present work, the experimental modal analysis of a rectangular plate with known geometrc holes is conducted to determine the vibration characteristics. This plate was excited by impact hammer and the response was recorded using accelerometers. The frequency response functions were obtained by conducting Fast Fourier Transform using a 2 channel FFT analyzer. a Modal software was then used to extract the modal properties of the mechanical system. These results in form of vibration characteristics of the system were then compared by conducting the finite element analysis of the plate using ANSYS. The results were then compared and reported.

Index Terms— Finite Element Analysis, Experimental modal analysis, natural frequency, Frequency response function, plate elements, ANSYS.

1 Introduction

Structures have made important advancements during the last some decades with the advancement of their aesthetic appeal, long span capability, quick and efficient construction, and their ability to make full and efficient use of material. Dynamic design aims at obtaining desired dynamic characteristics in machines and structures, which may include shifting of natural frequencies, mode shapes which are required and vibratory response. The conventional dynamic design is basically hit and trial method in which we try to achieve desired dynamic characteristics by producing several prototypes. Computer modeling, usually finite element analysis (FEA) has been extensively used with in the structural industry to aid in the design process of new product. FE modeling has been shown to be capable of predict the required mechanical properties of a structure that will give the desired performance characteristics.in this paper we have developed computer models to study the effect of natural frequencies and compare FE data with the experimental results to validate the model.

Many researchers have studied FE modelling and its validation such as [1] which studied the drilling machine. In the first, modal tests have been carried out on a drilling machine using instrumented impact input hammer. Modal identification has been done using global method of modal identification. For analytical FE modeling of the drilling machine, a computer program has been developed. The result obtained using finite element modeling, have been correlated with the experimental program which has been developed. In the second study, modal testing has been carried out using random noise generator and modal exciter. [2] has taken 20 KW horizontal axis wind turbine blade as an example, the program for optimization of dynamic design of the blade's aerodynamic contour has been solved using MATLAB tool, based on the design process of Wilson method. The way of combining Solid works and ANSYS was adopted to establish the blade model so as to describe actual shape and layer structure of the composite blade precisely. The dynamic performance of the blade was checked by modal analysis, providing a reference for structure design and other analysis. Comprehensive review on the research of modal identification method can be found in [3], [4]. Depending on the domain where identification is carried out, experimental modal analysis techniques can be categorized in to the time- and frequency domain method as one of the frequently used time domain techniques, the complex exponential method seeks to calculate the modal parameters by analyzing the impulse response functions which are the inverse Fourier transforms of the measured FRF data. Based on similar concept, Ibrahim developed a time domain method [5], [6] to estimate the modal parameters by curve fitting measured free decay time response of a system. Following the time series approach in system engineering, identification algorithms [7] have also been developed to estimate the system modal parameters using the measured free and response time signals. The main objective of the work done in this project is to make a Finite Element model and compare it with the experimental observations obtained from the analysis of the set-up which is similar in dimensions and shape as in Finite Element analysis. Further, the errors in the result obtained from the FE analysis are observed in order to develop a strategy for FE model validation in structural dynamics which includes provision of a balanced set of algorithms for all the procedures in the process of FE model validation.

FE modeling has been shown to be a useful tool in optimizing performance characteristics [8] it is also a useful technique that can be used to reduce the time and financial cost of designing and manufacturing a new structure. Before a new design of a structure can be manufactured on the production line, the engineers must first be satisfied that the end product will have

- *Hasan Zakir Jafri is currently pursuing Ph. D degree program in Mechnaical engineering department from Jamia Millia Islamia, University, India, PH-+917838050905, E-mail: hasan.jafri@rediffmail.com*
- *Prof.Aas Mohammad is professor in Mechnaical engineering department in Jamia Millia Islamia, University, India, E-mail: am200647@rediffmail.com*

International Journal of Scientific & Engineering Research, Volume 7, Issue 10, October-2016
ISSN 2229-5518

the desired mechanical properties. Before the advent of computer modeling, the traditional method was to manufacture prototypes, which were then subjected to mechanical tests to measure the structure's properties [10]. This process would have to be repeated until a prototype satisfied all the test requirements. An accurate FE model can be used to predict the mechanical properties of the manufactured structure without the need for numerous prototypes. To ensure that an FE model is accurate it needs to be validated. Modal analysis can be used as a method to dynamically validate the FE model by comparing its first few modes of vibration with experimental modal analysis data from the matching manufactured part [9], [11].

2 FINITE ELEMENT MODELING

The following section deals with the FE modeling of a rectangular plate. First a model of rectangular pate having one circular and a triangular hole. the plate 3-D model is created in ANSYS with web structure established in ANSYS. material properties were defined as isotropic. The material properties were defined by the Material Models menu. The corresponding real constants were distributed to every part of the plate and appropriate grid size was set. All the areas were meshed by MASHTOOL in the way of Free Mesh.

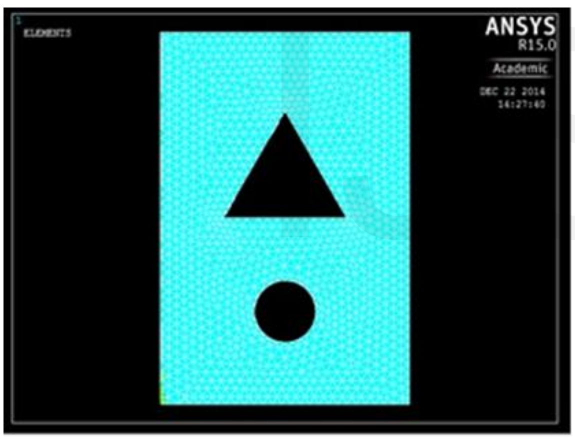

Fig. 1. FEM Plate model

The plate was modeled using solid185 tetrahedral elements. The modal superposition method was used to compute the harmonic response of the FEM model. The difference between the experimental data and the FEM data is the result of shifting of the resonance frequencies. The largest shift (about 5 percent) occurs at the fourth resonance frequency. Thus, the FEM model is stiffer than the real system.

There are four modes in the frequency range 100 to 500 Hz. All the mode shapes from the FEM model share the same trend and shape with their100 200 300 400 500 counterparts from the experiment.

2.1 Dynamic finite element analysis of the plate

Dynamic finite element analysis of the plate mainly refers to the vibration modal analysis using the finite element theory. Modal analysis is used to identify natural frequencies, especially low-order frequencies and vibration modes of the plate. From the modal we can learn in which frequency range the plate will be more sensitive to vibrate. Plate should be designed to avoid the resonance region with the tower and other components in order to prevent some destruction of related components. In this paper, the finite model of the plate has been established in ANSYS by importing the plate model created previously.

2.2 Modal analysis of the plate

There are many ways for ANSYS modal analysis, of which the Block Lanczos method is most widely used because of its powerful features. Moreover, it is frequently applied with model of solid units or shell units. The vibration modes of the first six orders were extracted with the frequency range of 0~1500Hz. The connections of plate with the frame could be regarded as free, so all DOFs are free as modal analysis does not require applying loads. At last, after solving with the solver, the vibration modes of all the orders and the result of frequencies could be observed in the post-processor.

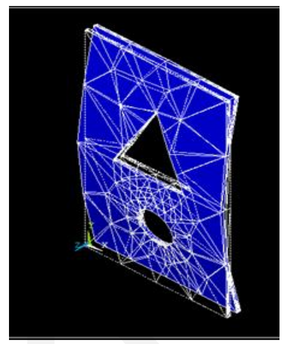

Fig 2(a) First mode

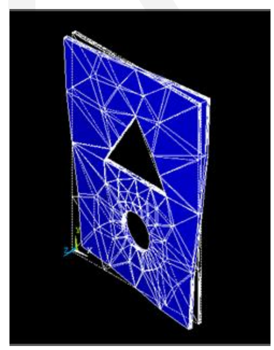

Fig 2(b) second mode

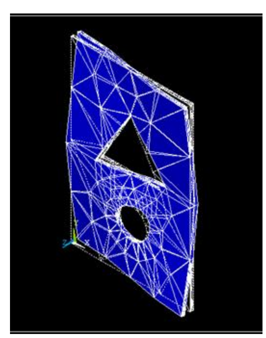

Fig 2(c) third mode

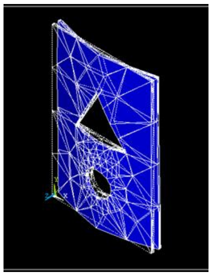

Fig 2(d) fourth mode

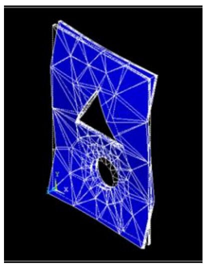

Fig 2(e) fifth mode

Table 1 FE results from ANSYS

Mode	I	II	III	IV	V
Frequency (Hz)	210	230	375	418	629

3 EXPERIMENTAL SETUP

The specimen for analysis is prepared from the stock of commercial mild steel rectangular plate. The dimensions of the plates are randomly selected as preferably available in the stock. Further, in order to fabricate the plates, two irregularities are induced in plate one is circular and other is triangular, the dimensions of both holes are taken randomly. The photographs of the involved specimen to be analyzed are shown in the following fig 4.
The dimension of rectangular plate is: (Length x width x thickness) = (300mm x 200mm x 5mm),
Diameter of circular hole = 50mm, Dimension of equilateral triangle = 100mm each side, Weight of plate = 2100gms, Poison ratio (υ) = 0.3

The experimental setup used in this paper consists of a specimen that is in the form of a rectangular plate consisting of a circular and a triangular hole. The modal testing of this MS plate specimen has been done by supporting on the soft springs to achieve free-free conditions [4]. The various equipment involved in the apparatus required to perform the experiment are:

1. Accelerometer (Model 352C68 SN 66757) PCB Piezoelectronics, USA
2. Signal conditioner (2Nos.) Model 480E09 (ICP Sensor Signal Conditioner) PCB Piezoelectronics, USA
3. Digital Signal Analyzer (Signal Calc ACE)
4. Impact hammer PCB Piezoelectronics, USA

The setup is arranged in order to perform the experiment has a rectangular MS plate with circular and triangular hole as mentioned above. The plate is marked with six points which represents as nodes named from 1 to 6. Further, an accelerometer is attached at the node 1 with help of wax [4]. Now the impact hammer is made to strike the plate at the remaining nodes from 2 to 6 to measure the transfer mobility. Due to these excitations at various nodes vibrations are produced in the plate which are sensed by the accelerometer placed at node 1 and the effect at the respective node is transferred to the signal conditioner which conditions the signal or converts these signals into measurable form. The digital signals so obtained from the signal conditioner are provided to the digital signal analyzer which analyzes these signals and helps the software to plot the suitable FRF curve. The procedure is again repeated for various nodes one by one and their respective FRF curves are obtained. The photographic view of the experimental setup is shown below in fig 3:

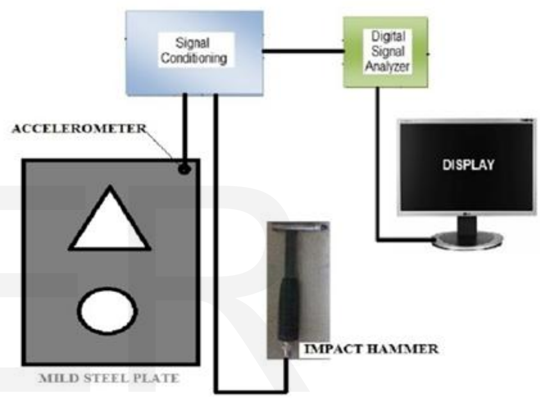

Fig 3 Experimental setup

4 MODAL TESTING AND IDENTIFICATION

In the two studies mentioned earlier, different techniques have been used, first by using commertially available software ANSYS and second using Experimental Modal Analysis. In the first study ANSYS is used to obtain natural frequency and mode shapes and in second, impact hammer is used to excite the plate at various points as shown in Fig 3. Response is taken at a fixed point with the help of an accelerometer. Response in the form of FRFs is recorded in the FFT analyzer. In the present study, the plate is excited at five locations and therefore, five FRFs are obtained. These FRFs are recorded in the form of inertance.

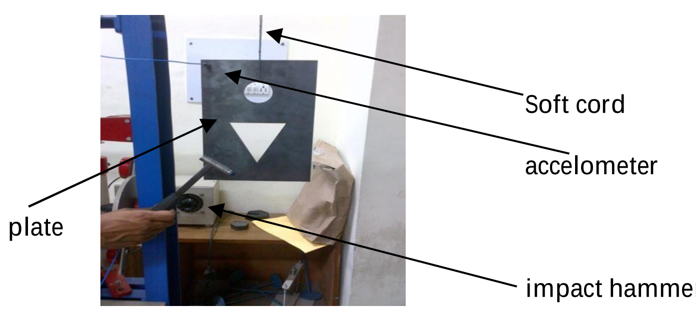

Fig 4 Actual experimental setup

4.1 Frequency Response Function

The Frequency Response Function (FRF) is a fundamental measurement that isolates the inherent dynamic properties of a mechanical structure. Experimental modal parameters (frequency, damping, and mode shape) are also obtained from a set of FRF measurements. The FRF describes the input-output relationship between two points on a structure as a function of frequency, since both force and motion are vector quantities, they have directions associated with them. Therefore, an FRF is actually defined between a single input DOF (point & direction), and a single output DOF. When the specimen (in this case a rectangular plate) is excited by an external source which is an impact hammer, vibrations are produced within the system. These vibrations are sensed by accelerometer in the form of acceleration. The displacement, velocity, acceleration or other factors which may vary with respect to time are defined for a particular mode shape with the help of plot between the amplitude and frequency. This plot for a particular mode of vibration thus formed is known as frequency curve. This graph thus plotted represents the nature of plate at various frequency levels.

After arranging the experimental set up, the quantities for the abscissa and the ordinate must be specified. Further, the quantities with their units in which the output is to be determined are fed into the software. So, as mentioned we assumed log of acceleration in m/s^2 on Y-axis and log of frequency in hertz on X-axis. As, the system is externally excited the curve is automatically plotted on the basis of frequency response function applicable for the system. Thus the peak value of the amplitude for respective value of frequency is obtained.

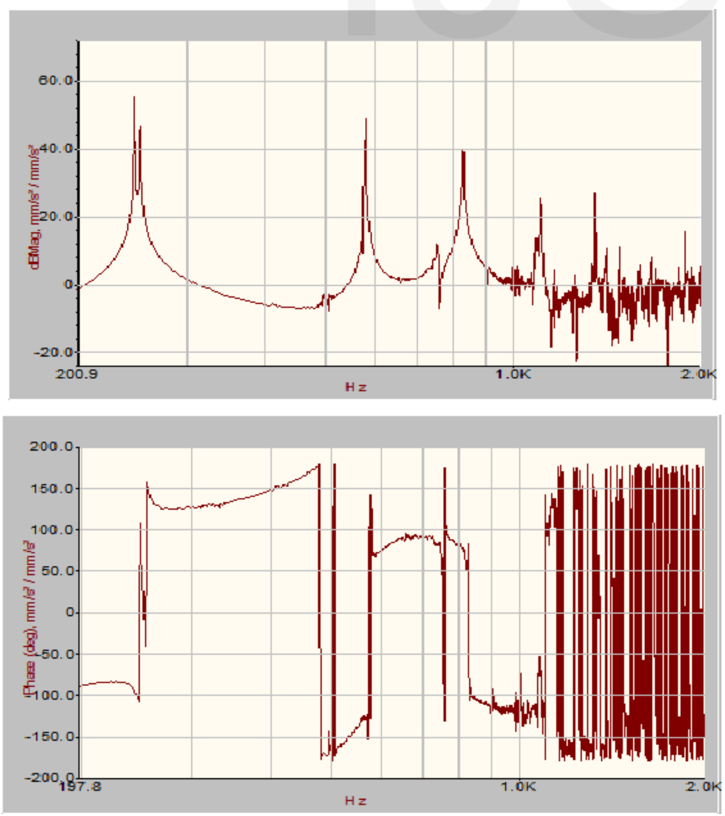

Figure 1: Frequency response function

5 COMPARISON OF ANALYTICAL AND EXPERIMENTAL RESULTS

The first stage of any reconciliation exercise is to determine how closely the experimental and analytical models correspond. If we are unable to obtain a satisfactory degree of correlation between the initial analytical FE model and the test data, then it is extremely unlikely that any form of model updating will succeed. Thus, a successful correlation is crucial for the success of model updating. Table 2 gives the comparison between experimental and analytical natural frequencies, for the first two modes, obtained from both the studies. If there are differences between analytical FE model predictions and experimental results, which is due to assumptions and approximations involved in the FE model the FE models need to be updated. However, the differences between the corresponding results of both studies are minor. Apart from natural frequency comparison (as given in Table 2), another method of model correlation is mode shape comparison. To compare the mode shapes, we plot the deformed shapes of the structure for a particular mode, using experimental as well as analytical model. These mode shapes are plotted side-by-side for quick comparison. This is a graphical approach to model correlation.

S. N.	Modes	FEM	EMA	% error
1	I	210	232	-0.014
2	II	230	245	-0.065
3	III	375	380	-0.013
4	IV	418	420	-0.004
5	V	629	630	-0.001

Table 2 Comparison of experimental and FE natural frequencies

4 CONCLUSIONS

The finite element analysis of a rectangular MS plate has been carried out using ANSYS and an experimental test has been performed on the same using an experimental setup the results obtained from both the results have been compared and both the results shows a good agreement with each other. However, the the post analysis on frequency response function (FRF) obtained from the experiment have not carried out (which is not required in this case due to close corelation). Thus FE results have been verified using experimental results.

ACKNOWLEDGMENT

The author wishes to thank Dr. S.M Muzakkir (Assosiate Professor, Jamia Millia Islamia, New Delhi). In carrying out experimental work, his valuable guidance and support.

REFERENCES

[1] R. S. Bais, A.K. Gupta, B.C. Nakra, T.K. Kundra, Studies in dynamic design of drilling machine using updated finite element models, Mechanism and Machine Theory 39 (2004) 1307–1320

[2] Fangfang Songa, Yihua Nia, Zhiqiang Tanb, Optimization Design, Modeling and Dynamic Analysis for Composite Wind Turbine Blade, Elsevier Ltd. (2011)

[3] D.J. wins, Modal Testing: Theory and Practice, Research Studies Press, London, England, 1984.

[4] N. Maia, Modal analysis identification techniques, Philosophical Transactions of the Royal Society 539 (1778) (2001) 29–40.

[5] S. Ibrahim, E. Mikucik, A time-domain modal vibration test technique, Shock and Vibration Bulletin 43 (4) (1973).

[6] S. Ibrahim, E. Mikucik, The experimental determination of vibration parameters from time responses, Shock and Vibration Bulletin 51 (3) (1976).

[7] S.M. Pandit, S.M. Wu, Time Series and System Analysis with Applications, Wiley, 1983.

[8] K.J. Bathe, E.J. Wilson, Numerical Methods in Finite Element Analysis, Prentice-Hall, Englewood Cliffs, NJ, 1982.

[9] R.M. Lin, Identification of modal parameters of unmeasured modes using multiple FRF modal analysis method, Elsevier Ltd. (2010)

[10] T.K. Kundra, Structural dynamic modification via model, Sadhana 25 (2000) 261-276.

[11] S.V. Modak, Studies in Finite Element Model Updating and Application to Dynamic Design, Ph.D. Thesis, Department of Mechanical Engineering, IIT Delhi, 2001.