ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Vol 9, Issue 10, October 2022

Comparative Study of the Accuracy of the Finite Element Method and the Experimental Method on A 3D Printed Model

[1] Mohammed Kashama Guzunza

[1] Istanbul Gelisim University, Dept. of Civil Engineering, İstanbul/ Turkey.
Corresponding Author Email: [1] kashmedguzunza@gmail.com

***Abstract*— One of the engineering challenges is understanding natural phenomena, analysing systems, and using results found as needed based on scientifically approved processes. In this study, two methods are analysed, the finite element method and the experimental method to know which method gives the precise results compared to the other. The case study considered in this work is a 3D printed structure that was modelled as a two-story shear building system with an irregular torsion. The finite element method is computed as a numerical model that is developed by using SAP2000. The experimental method is computed by collecting modal parameters data at the laboratory by ambient vibration and white noise test, then modelled by ARTeMIS pro, based on numerical techniques for the identification of systems in the time domain. The results obtained from the numerical and experimental models are compared then found the most accurate method.**

***Index Terms*— irregular torsion, modal analysis, numerical model, sensitivity analysis, System Identification.**

I. INTRODUCTION

One of the engineering challenges is understanding natural phenomena, analyzing systems, and using results found as needed based on scientifically approved processes.

The finite element (FE) method is used in several fields to solve and analyze complex problems that other empirical methods could not solve. Although it is difficult to fully represent the model by the finite elements due, for example, to the complex behavior of geometries, boundary conditions, and properties of materials. The results obtained enable reliable decisions to be made [1].

This work deals with 3D printed structures which are needed to know about materials that should have different chemical or physical properties.

This work was carried out of white noise and ambient vibration tests that were performed to obtain the natural frequencies and mode shapes of the frame structures. ARTEMIS pro Software was used for predicting the Dynamic characteristics of the structure which were estimated by the EFDD and SSI technique, and these parameters are used to compare the FE model of the structure modelled in SAP2000.

II. DESCRIPTION OF THE STRUCTURE

The framework structure was performed to behave as a model that will work with irregular torsional. After conception, the model was sent to SAP2000 software to verify if the sought features were found. After verification and discussion, it was decided to make a study with a 3D printed model structure. The model was designed by SOLIDWORKS Software, see Figure 1. The 3D printed model structure was done with ABS (Acrylonitrile butadiene styrene) filled with 50% materials. ABS is one of the most versatile plastic materials available for 3D printing today.

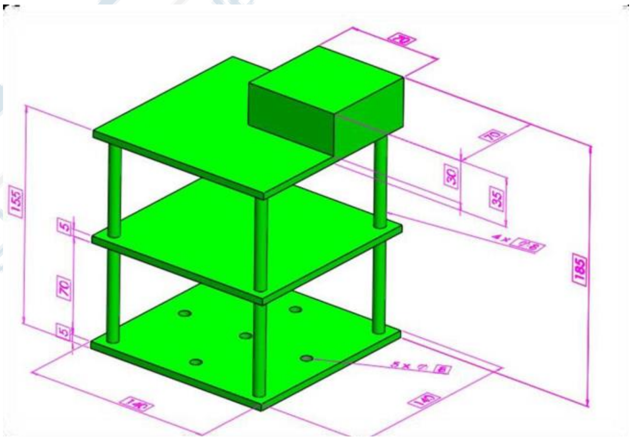


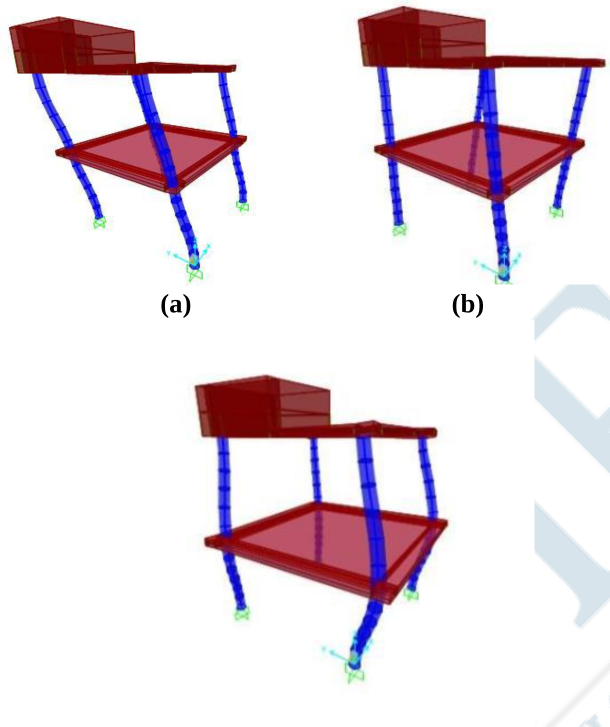
Figure. 1. The design modelled in SOLIDWORKS

The 3D printed structure is modelled as a two-story shear building system with a small shell on top as shown in Figure 2. The structure composite of 2 squared slabs of 14x14 cm, 0.5 cm of thickness supported by 4 circular columns 0.8 cm in diameter. Then a small shell above the upper story, it has 3.5 cm of thickness, a squared of 7x7 cm. the height under the floor of every story is 7 cm then the total high of the frame is 18.5 cm. the structure is bolted with the same plate mention above (14x14x0.5) cm.

III. FINITE ELEMENT METHOD

The finite element (FE) name comes from these small entities. It is based on assumptions such as linearity of parameters, variation according to polynomial functions, or considering certain parameters as constant. Its importance in engineering studies provides that most natural phenomena are complex [2].

The FE model of the structure was computed in SAP2000. The numerical modal properties were obtained by using the modal analysis of the model in SAP2000. In this work, only the results of the 2nd, 3rd, and 5th modes, those that correspond to the modes extracted from the laboratory measurements, are used and shown in Figure 3. Natural frequencies of 25.73 Hz, 34.94 Hz, and 99.76 Hz were obtained for the 2nd, 3th, and 5th modes, respectively



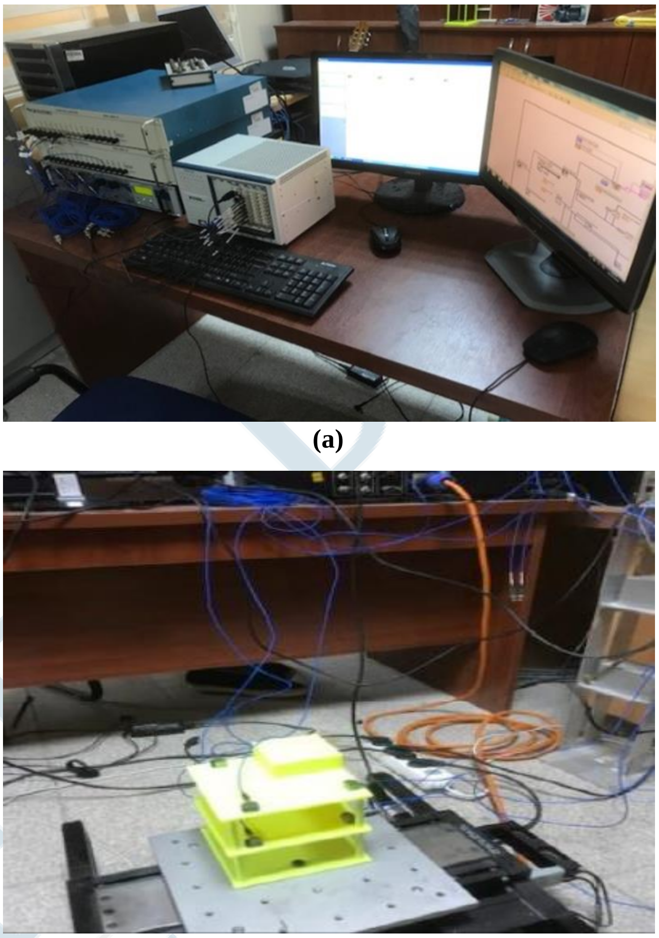
(a) (b) (c)

Figure 2. (a) 2nd, (b) 3rd, and (c) 5th modes of the numerical model

IV. EXPERIMENTAL METHOD (LABORATORY WORK)

In this study, the PCB Piezotronics set (See in Figure 3(a)) was used to collect the data from the structure to transfer to a computer that uses LabVIEW as software for performing the experimental model. For each Test, approximately 10 minutes of long data with a sampling rate of 500 Hz were collected. After collecting data, filtering and detrending of the dynamic data were performed in MATLAB [3]. After, modal parameter estimation was conducted by using modal analysis software Artemis [4].

The application of these data was used for identifying the modal parameters of the structure and for studying the dynamic features of the specimen.



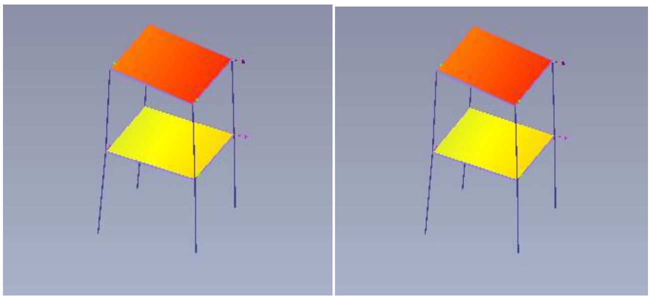
(a)

(b)

Figure 3. (a) PCB Piezotronics set, (b) structure with accelerometers

In this case, the ambient vibration test cannot be able to give a good frequency for the torsional modal. Then the frequencies used for performing the FE model to match the experimental came from the White noise vibration test.

Only 3 modes in the experimental method are considered according to the modes found in SAP2000. Therefore, the experimentally obtained natural vibration frequencies for the 1st, 2nd, and 3rd modes are 24.902 Hz, 44.537 Hz, and 114.46 Hz, respectively. (See Figure 4)

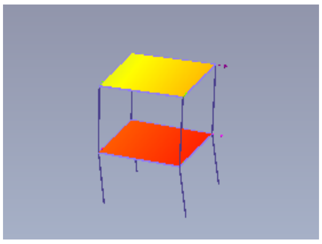


(a) (b)

ISSN (Online) 2456-1290

International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)

Vol 9, Issue 10, October 2022



(c)

Figure 4. (a) 1st mode from EFDD and (b) 2nd and (c) 3rd modes from SSI.

V. COMPARISON OF FINITE ELEMENT AND EXPERIMENTAL METHOD

The main purpose is to compare the numerical model performed by SAP2000 software with the experimental model performed by Artemis software to verify these two tools and to ensure that which results are more accurate.

A. Finite Element Model Updating (FEMU)

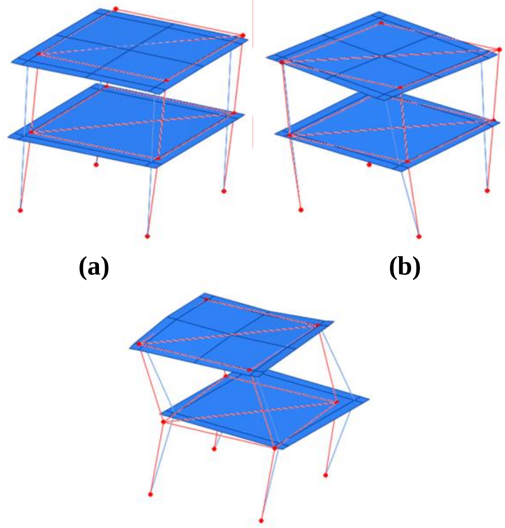
Before making a comparison between the experimental and analytical models, the models must be well matched and adapted to provide a reference model.

The FEMU method is an updated technique that produces more accurate results because the studies are based on measured data compared with those found in the mathematical analysis.

This study was done by FEMtools model updating software and was used to improve the match between the dynamic properties of FE and test data to provide an updated model [5].

- **Mode shapes pairs and comparison**

It has been assumed that models are perfectly adapted as shown in Figure 5. This matching helps to understand the differences between these two models.



(a) **(b)** **(c)**

Figure 5. (Red is FE model and bleu is the Experimental model)

(a) Mode shape pairs of 1st mode of FE analysis and 2nd mode of Experimental Analysis
(b) Mode shape pairs of 2nd mode of FE analysis and 3rd mode of Experimental Analysis
(c) Mode shape pairs of 3rd mode of FE analysis and 5th mode of Experimental Analysis

Table 1. Comparison of the modal parameters obtained by Numerical (FE) and experimental method

MODE #	**FEA**		**EMA**		**Diff. (%)**	**MAC (%)**
	Mode	Freq. [Hz]	Mode	Freq. [Hz]		
1	2	25.73	1	24.90	3.28	64.6
2	3	34.94	2	44.53	-24.13	49.5
3	5	99.76	4	114.26	-13.55	38.8

- **Updating**

For understanding the discrepancy between the dynamic properties of FE and the experimental model, it must start by doing an update and finding the reference model, in the end, to make a comparison of those models.

Table 2. Table of Comparison estimated modal parameters between analytical and experimental results

MODE #	**FEA (Numerical modal)**		**EMA (Experimental modal) Freq. [Hz]**	**Diff. (%) Freq Before updating**	**Diff. (%) Freq After updating**
	Before updating Freq. [Hz]	After Updating Freq. [Hz]			
1-2	25.73	24.95	24.90	3.28	0.20
2-3	finding	44.16	44.53	-24.13	-0.83
3-5,	99.74	A 115.87	114.74	-13.55	1.41

VI. CONCLUSION

The purpose of this work was to identify the modal parameters of this structure and to create a reference model by the Finite element model updating method for making comparaison.

After comparison, the results found in the experimental analysis are more accurate than those found in the finite element model.

VII. ACKNOWLEDGMENTS

Dokuz Eylül University civil engineering laboratory corps for support and assistance during this work.

REFERENCES

[1] Girardi, M., Padovani, C., Pellegrini, D., Porcelli, M., & Robol, L. (2020). Finite element model updating for structural applications. Journal of Computational and Applied Mathematics, 370, 112675. J. U. Duncombe, "Infrared navigation—Part I: An assessment of feasibility," IEEE Trans. Electron Devices, vol. ED-11, pp. 34-39, Jan. 1959.

[2] Jonsson, F., & Johnson, D. (2007). Finite element model updating of the new Svinesund Bridge. Master's Thesis, Chalmers University of Technology, Göteborg, Sweden.

[3] MATLAB (2017). The MathWorks Inc. Natick, Massachusetts, United States.

[4] ARTeMIS Extractor Pro (2016). Structural vibration solutions. Aalborg, Denmark.

[5] FEMtools (2017). Dynamic design solutions. Leuven, Belgium.