ISLAMIC UNIVERSITY OF TECHNOLOGY

ORGANIZATION OF ISLAMIC COOPERATION

Vibration Analysis of Multi-Story Structures

B.Sc Engineering (Mechanical) Thesis

Authors

S.M. Nazrul Anam (160011028)

Abid Md. Fahim (160011038)

Supervised By

Dr. Md. Zahid Hossain
Professor
Department of Mechanical and Production Engineering
Islamic University of Technology (IUT)

DEPARTMENT OF MECHANICAL AND PRODUCTION ENGINEERING
ISLAMIC UNIVERSITY OF TECHNOLOGY (IUT)

March, 2021

ACKNOWLEDGEMENT

The authors express gratefulness to the Almighty Allah (SWT) for his blessings, which enabled to complete this thesis successfully.

The authors express gratitude to their supervisor Professor, Dr. Md. Zahid Hossain, Department of Mechanical and Production Engineering (MPE), Islamic University of Technology (IUT) for his continuous guidance, helpful suggestions and supervision at all stages of this thesis work.

And the authors are indebted to their family members for providing the financial and mental support in perusing the Bachelor’s degree in Mechanical Engineering.

Contents

Abstract:

To determine the dynamic characteristics and properties of an object, Modal analysis, and Harmonic response analysis both are quite essential tools. Natural frequency is a significant dynamic parameter as every structure in the world vibrates with a high amount of amplitude at its natural frequency. For improving a structure's strength, reliability, and durability at its design stage or pre-production stage at variable operating conditions we need to know and determine the modal parameters of an object. Those modal parameters are natural frequency, mode shape, and damping characteristics. By determining those parameters, we will be able to know the actual structural and dynamic properties of an object. In this paper, we work with a two-story and three-story structure by giving different force conditions and boundary conditions. The paper clarifies the dynamic and structural behavior of the two-story and three-story structures by performing Modal analysis and harmonic response analysis with the assist of FEM (Finite Element Method). The analysis is completely based on the simulation process. No experimental analysis has not conducted due to some circumstances. By determining as well as observing the dynamic characteristics of the two stored and three stored structures we have been able to relate and predict the properties of those structure's behavior during an earthquake consequence. We have been able to find that out by giving a base structure at the downward side of those structures as well as giving some horizontal forces to that base structure. Along with all of these analyses, we have shown a generic comparison among the structures at different operating conditions by giving different kinds of masses at different locations of the structures.

Keywords: Dynamic characteristics, Natural frequency,Amplitude,Modal parameters

Nomenclature:

A	Amplitude
ω	Angular frequency
t	Time

Introduction:

A machine consists of different kinds of parts or components of different shapes and sizes. Each component of a machine has different functions to perform a certain activity. Every constituent of a machine has a defined design requirement to prevent any kind of structural failures. So, it is quite essential to determine an object's dynamic and structural characteristics at various kinds of operating conditions. To develop an object's structural strength and condition it is quite essential to know as well as determine the dynamic behavior of the structure at different kinds of loading and unloading operations. Those essential statistics, as well as data, can be achieved by performing the Modal analysis which is considered as a quite significant tool to find any object's fundamental modal parameters such as natural frequency, mode shapes, stiffness, and dynamic characteristics. By knowing as well as observing those parameters a structure's performance, stability and durability can be determined quite precisely.[1]

Finite Element Method (FEM)

For conducting Modal analysis at first FEM (Finite Element Method) is an essential methodology. It is an essential method that has been widely applied for differential equations numerically and mathematical modeling. The FEM can be defined as a specific numerical method in order to determining as well as analyzing PDE(Partial Differential Equation)Some boundary condition value-related problems can be solved efficiently by using FEM.[2]

In order to solving a problem, the FEM system actually branches a substantial system into plathour amount of microscopic systems with uncomplicated parts that are known as finite elements. For performing FEM first of all meshing of the defined object should be done.For getting a precise result meshing should be done perfectly. That meshing of an object should be as fine as possible. Meshing is a basically nothing but a domain in numerically satisfied which has a demarcated number of points. It will create discretization in the space dimensions.[3]

By conducting the Finite Element Method during the formulation of a boundary value problem at last we generally get a set of equations which can be numerical as well as algebraic. The method basically identifies any unknown functions in case of domain analysis.The simple equations have been modeled into finite elements. These definable elements are congregated into a more substainal a system of equations.That substainal system of equations models as well as defines the problem. Differential processes can be obtained from the calculus of differentiations. Corresponding error function can be determined precisely by using these differential processes.[4]

It is quite significant for determining the displacements, stresses, and strains in a structure under various kinds of loads.FEM is considered as a key tool for calculating and sorting those properties of a structure. It basically shows how a component proceeds to certain conditions and parameters stress,strain,load, force and pressure.[5]

Modal Analysis

In the case of structural analysis, Modal analysis is the most fundamental and essential tool. By performing modal analysis we will be able to determine and observe an object's dynamic, static as well as structural behavior and properties. The result from the modal analysis will provide us a certain idea about a structure's performance under various operating conditions such as different loads, forces, pressures, and stresses. In order to develop a structure's reliability, durability as well as strength we must conduct the Modal analysis. The modal analysis provides us different kinds of fundamental modal parameters such as resonant frequency, damping properties as well as mode shapes. By observing those parameters we will be able to determine as well as improve a structure's build quality.[6]

The quasi-static analysis is considered as one of the most basic type of analysis. In case of quasi-static analysis, the acceleration is quite ignorable because the rate of of applying load here is quite steady.The acceleration can be considered zero in that case.But the influences of acceleration can not be ignored in case of Dynamic analysis where change of acceleration is significant.The quasi-static analysis and dynamic analysis both have one thing in common which can be considered as both types of them provide a one-to-one relationship between a particular input to its system response.Suppose a force is being applied to an object so it can be counted as input.And the deformation of the structure due to certain force can be considered as the response of system.

The modal analysis actually provides the numerical limitations of a system's response.From the figure given below we can observe that for a certain amount of frequency the amplitude of an object gets maximum.That kind of frequencies are defined as natural frequencies.The main and fundamental objective for performing modal analysis is to find those certain frequencies.

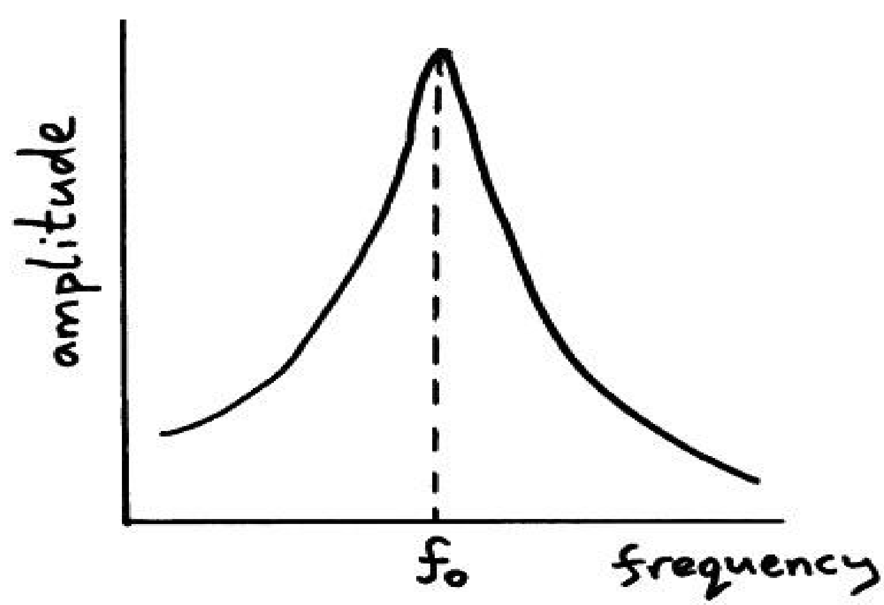

Fig 1: Amplitude of system response

When an object usually vibrates naturally it must be in its resonant frequency condition. It is also the frequency where the object will provide a minimum loss of energy while transfering energy from one form to other kind of for.The role of Modal analysis is to determine those resonant frequencies at which the structure's amplitude can increase to the infinite level.[7]

A structure should be designed to ignore resonant vibration. The structure should be vibrated at a specified frequency. So structural engineers will know the consequences of that structure at different operating conditions. In case of finding that modal analysis is an essential tool or technique.[8]

Deformation of a designed mechanical structure could affect the stiffness of that structure. So according to the change of stiffness, the modal parameters of that structural change. The magnitude and direction of the change totally depend on the quantity and location of the deformation. Each of the vibration modes will be affected by a single deformation. Some of these deformations will have a significant effect on certain modes and some of these deformations will have less effect on some modes. By observing all of those effects on different modes, damage to the structure can be captured quite easily.[9]

The modal analysis provides all essential information and foundations for optimizing the mechanical structure. It helps engineers by providing all modal parameters of a structure to determine the complete dynamic description of an object. A structure is known as a free structure in vibration analysis if there is no force acting on it. The intrinsic characteristics of a free structure are expressed by the modes of vibration.[10]

To find any mechanical structure's stability and develop stability Modal analysis is a fundamental tool. Modal analysis can be conducted in both damped and undamped conditions. The stability of a structure can be ensured by ignoring or decreasing the excitation effects of the vibrational frequencies. [11]

Modes are intrinsic properties of a mechanical structure. Modes are actually influenced by the resonant frequencies of the structure.If some fundamental material properties such as density, stiffness, and damping characteristics of a structure affect the modes.[12]

Harmonic Response Analysis

Harmonic response analysis is considered as one of the fundamental analysis in case of structural analysis.It is basically an additive dynamic analysis.That additive dynamic analysis is usually used to determine the response of a system when the system is being excited at particular frequencies.It is also referred as Frequency Response Analysis as it determines the response or excitation of a system in certain resonant frequencies.[13]

To perform the Harmonic analysis, Modal analysis is the most fundamental prerequisite. The modal analysis provides us the basic modal parameters which actually contain the input frequencies. These input frequencies are part and parcel of performing the Harmonic response analysis. These inputs are quite essential for fixing the maximum and minimum range to perform the Harmonic analysis.[14]

So, after all, Harmonic Response Analysis is a type of restart analysis that generally uses modal superposition to calculate its results. By performing a Harmonic response analysis, we will be able to check at which modal resonant frequency the mechanical structure gives the highest amount of amplitude at various loading or force conditions. During Harmonic analysis, the load or force which is being applied is a steady-state sinusoidal force or load at a specified frequency. The structure will show various kinds of excitations at different natural frequencies.[15]

While a machine has been doing steady-state operations some sort of stress formed. Harmonic response analysis is generally used to determine the desired stresses. In the case of vibration analysis Harmonic response analysis leads us to determine the different kinds of strains, total deformations in different directions also.[16]

A Frequency Response (FR) or Harmonic Response analysis is a type of simplification where we assume the input load is being applied at a single and specified frequency and the response only occurs at the same frequency as the duration of the load-application is quite adequate so that the response also only occurs at the initial frequency as before.In case of this type of loading, in order to saving a plathour amount of time and effort Frequency Response analysis should be performed accurately and precisely.This is the most efficient way to determine the dynamic properties of an object.[17]

Assumptions should be considered as some sort of simplifications.The simplification does not Influence the result's accuracy if and only if the assumptions are made perfectly and precisely.The assumptions we have considered.should be understood thoroughly.By understanding these assumptions, we can precisely determine if a frequency response analysis is accurate or not.[18]

If a signal is being applied which is a actually sinusoidal signal is to a Linear Time-Invariant (LTI) system as an amount of input.,a steady state output will be determined.That steady state output is a a sinusoidal signal as well. So here both of the input and output signals are sinusoidal signals.Both signals have the same frequency.But these input and output signals have quite different amplitudes and phase angles

Let the input signal −

$r(t)=A\sin(\omega_{\circ}t)r(t)=A\sin(\omega_{\circ}t)$ $r(t)=A\sin(\omega_{\circ}t)$

The transfer function for open loop would be −

$G(s)=G(j\omega)G(s)=G(j\omega)$

- The amplitude of the output sinusoidal signal is determined by multiplying the amplitude of the input sinusoidal signal and the magnitude of $G(j\omega)$ at $\omega=\omega_{\circ}$
- The phase of the output sinusoidal signal is obtained by adding the phase of the input sinusoidal signal and the phase of $G(j\omega)$ at $\omega=\omega_{\circ}$[19]

Models

There are four kinds of models introduced here, the first one being a simple two-story unloaded structure. The rest two are just variants of masses applied upon the unloaded structure. The final one is a three-story structure. The dimension of the plates in every model is 30cm×15cm×0.5cm. The dimension of the columns in every model is also the same, that being 2.5cm×0.5cm. The height is 39.25cm considering the middle plate thickness. The total height of the two-story structures is 80cm. The plates and columns are considered as separated bodies and joined together by basic extrusion rules of SOLIDWORKS®. Also, one of the most important things done here is a base plate large enough to negate the fixed support boundary condition is added in every model to make it easy to apply external excitation forces.

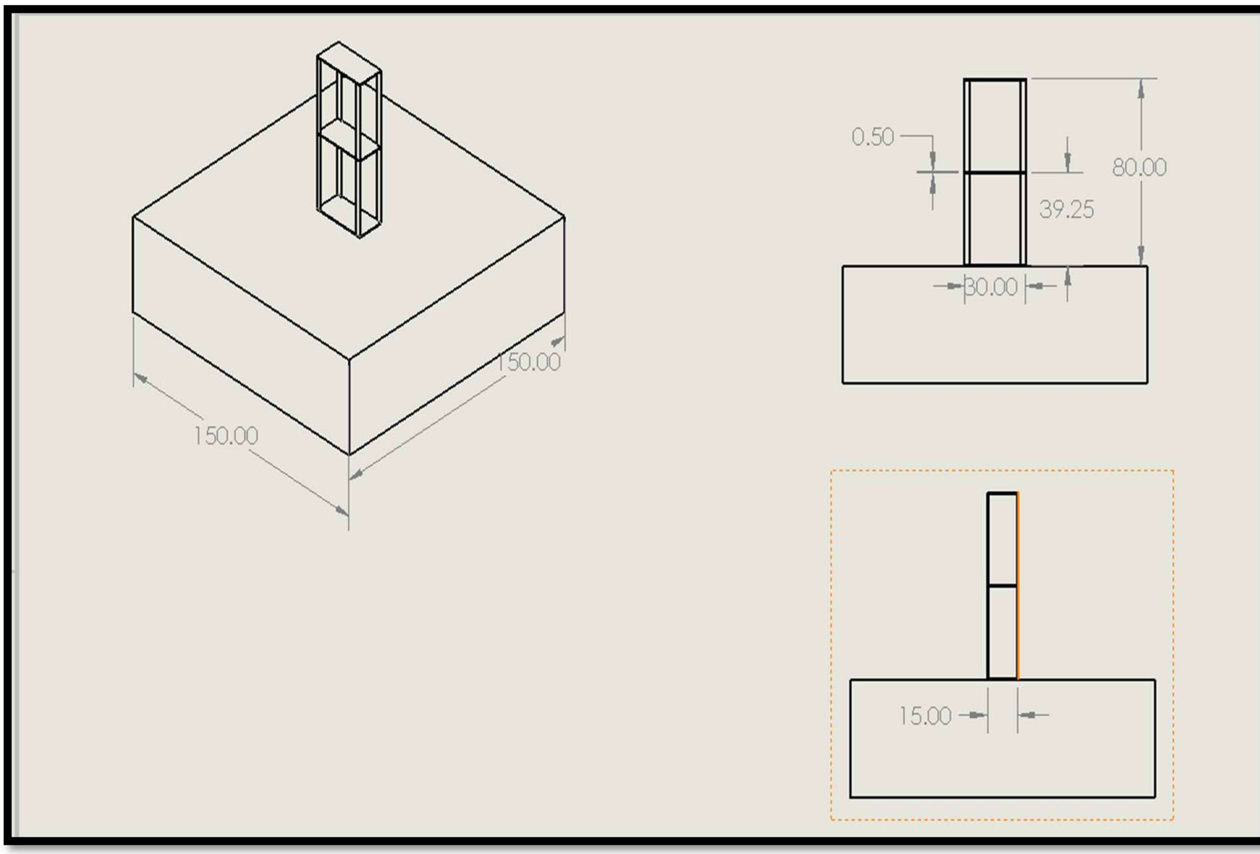

Fig 2: Unloaded structure with base plate.

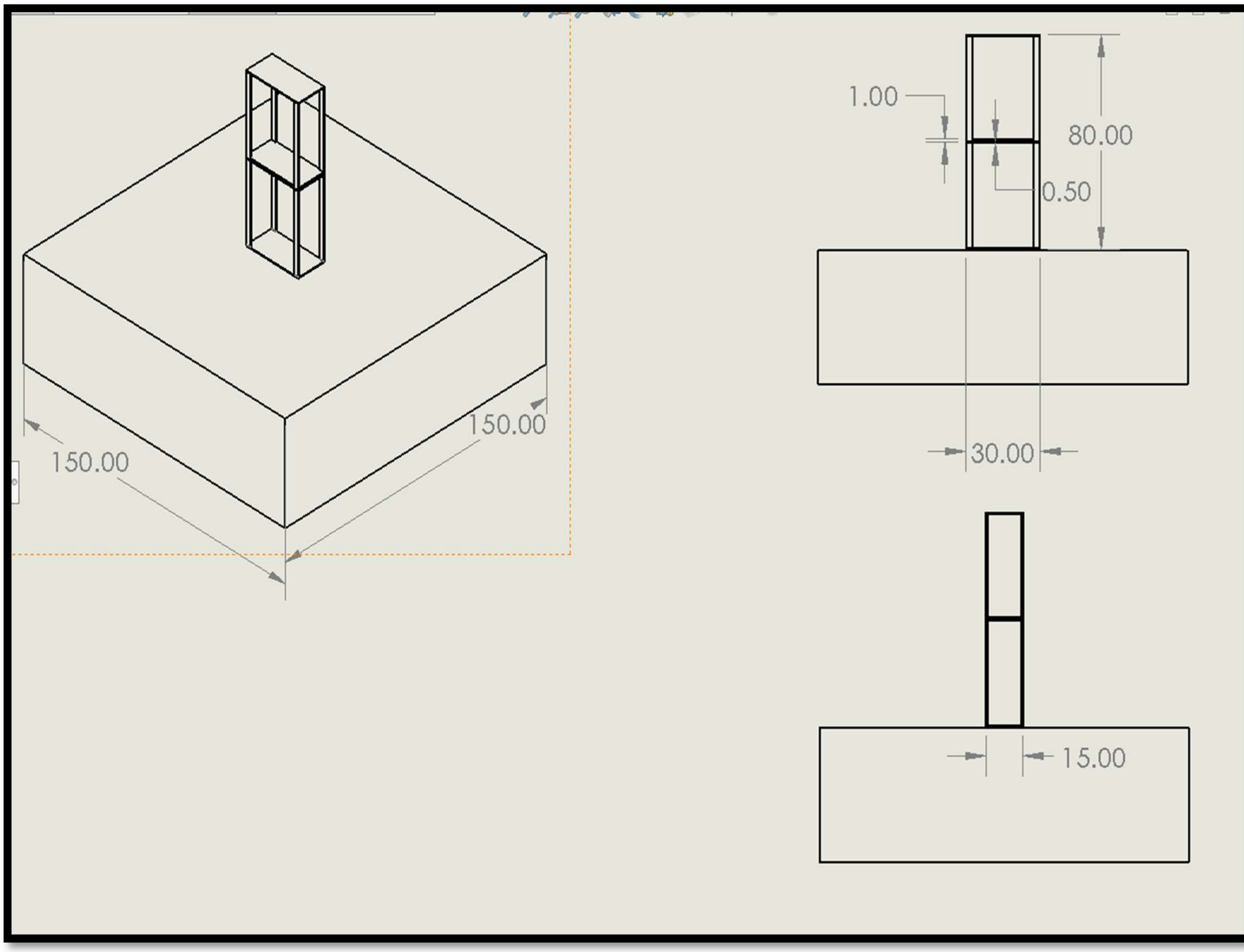

Fig 3: Structure with a distributed mass on the first floor.

A distributed mass on the first floor is added in this model. The mass has a thickness of 1 cm and it is distributed throughout the plate. The other dimensions are the same as the unloaded two-story structure.

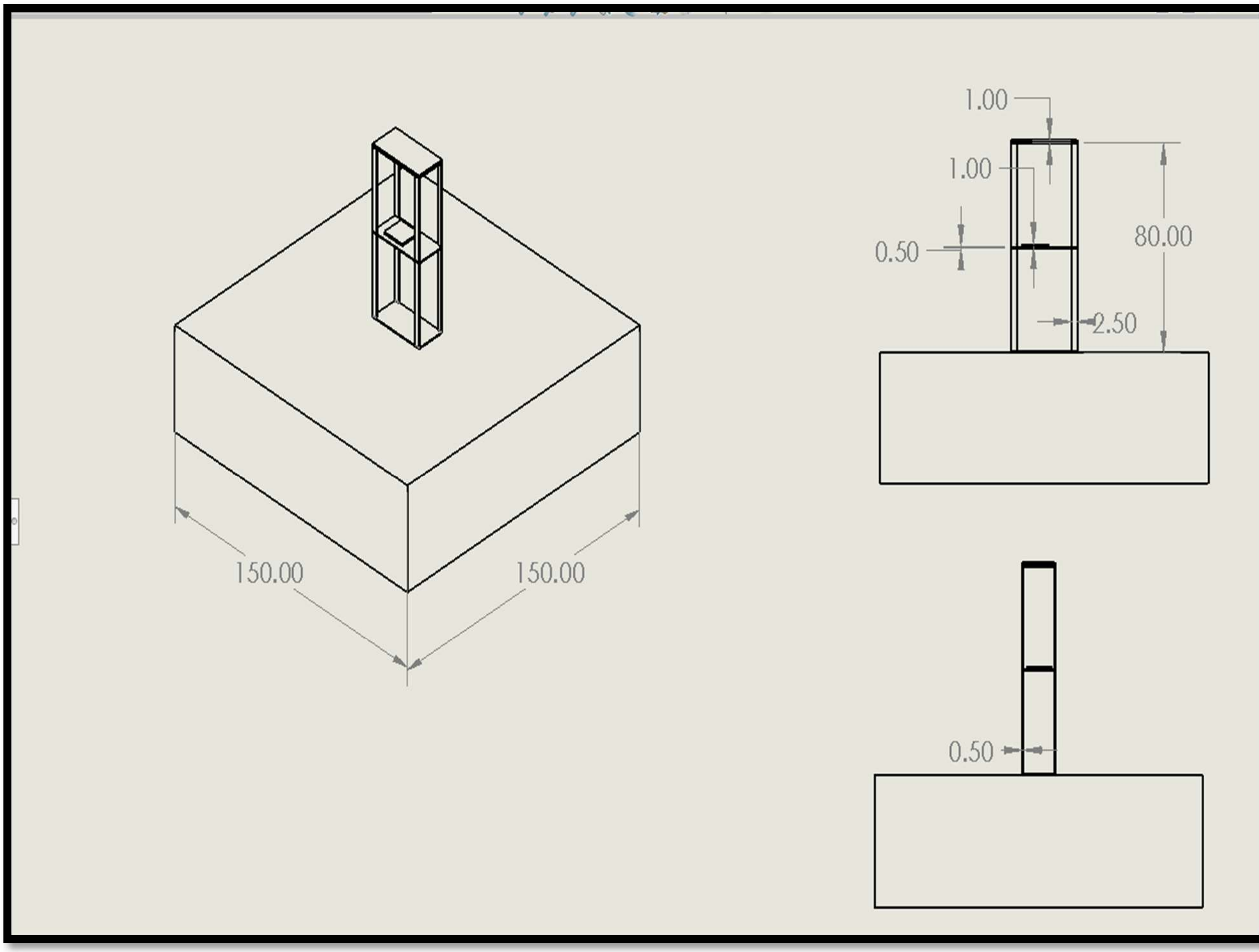

Fig 4: Structure with a distributed mass on the second floor.

A distributed mass has been added on the second floor here which has a thickness of 1 cm. Also, a simple mass is put on the first floor which increases the total mass of the structure, making this model the heaviest among the three two-story structures.

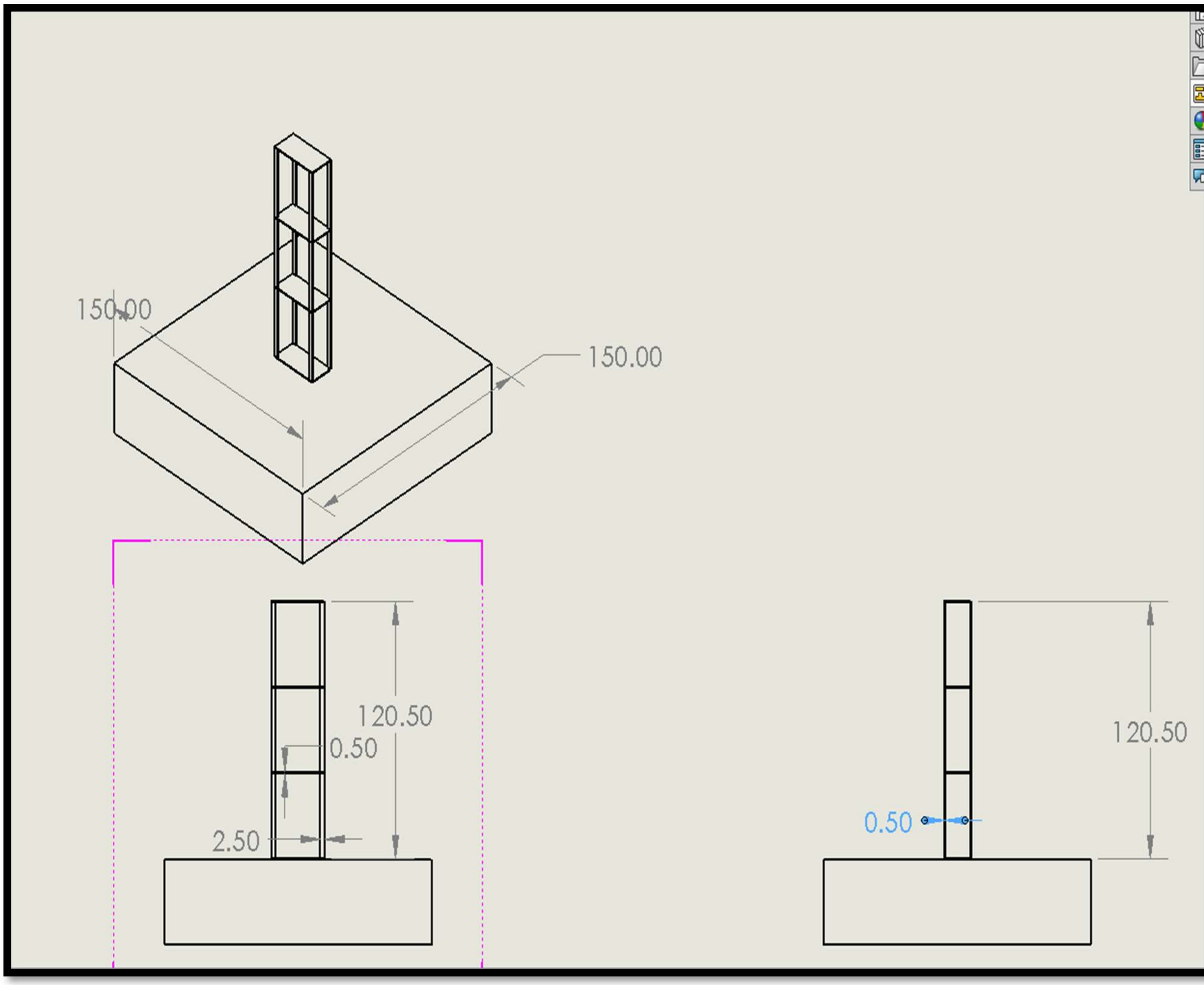

Fig 5: Unloaded three-story structure.

This structure has the same plate and column dimensions. For adding an extra floor, the total height of the structure becomes 120.50 cm.

Engineering Data

The structure is made of mild steel having the following properties:

Young's Modulus	**210 GPA**
Density	7800 kg/m³
Poisson's ratio	0.295

The corresponding data which is automatically generated in Ansys® is as follows

Properties of structural steel

Property	Value	Unit
Density	7800	Kg m^3
Coefficient of thermal expansion	1.2E-05	C^-1
Young's Modulus	2.1E+11	Pa
Poison's Ratio	.295	None
Bulk Modulus	1.707E+11	Pa
Shear Modulus	8.108E+10	Pa

Methodology:

Meshing

The element type selected here is SOLID 186, which is the default element type in ANSYS® Mechanical while working with a solid volume. The huge base plate has very less or no significance on the structure's analysis. So, to reduce the analysis time the base plate element edge length size is selected 40 cm. The most important part is the structure and the meshing needs to be very accurate and fine there. Element edge length is selected 0.5 cm for the structure.

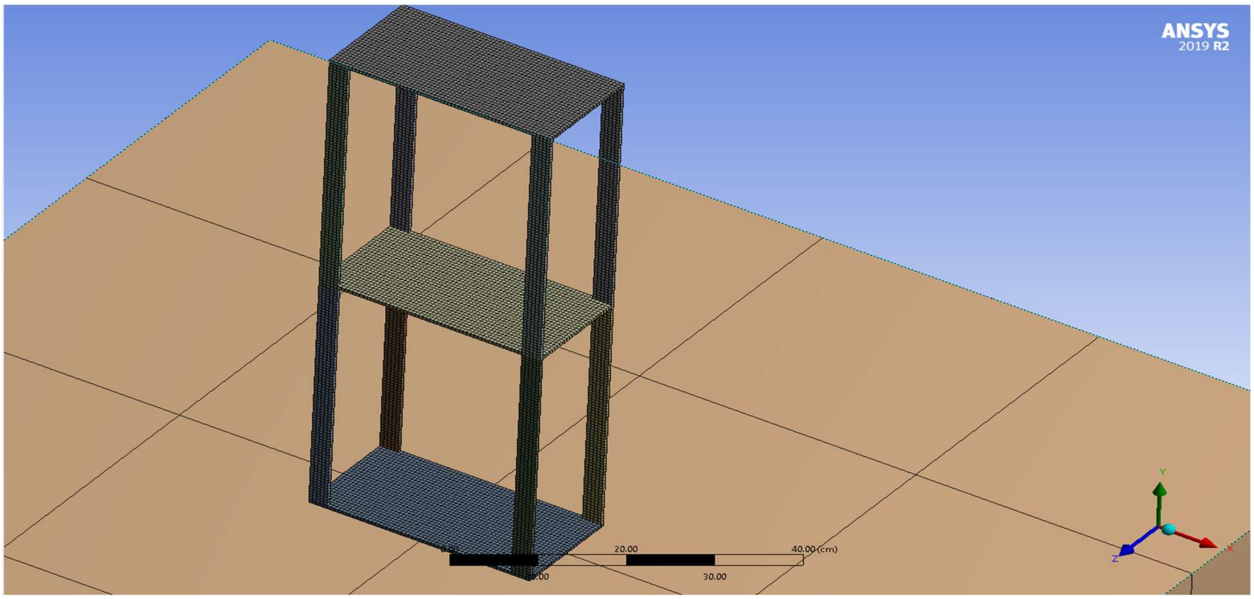

Fig 6: Meshing of the unloaded structure.

Modal Analysis

To find out the natural frequencies modal analysis has been conducted using default Ansys® analysis settings. As the large base plate is present, no fixed support is provided additionally. The structures' natural frequencies are found out as follows:

Unloaded		Distributed mass on the first floor		Distributed mass on the second floor		Three-story	
Mode	**Frequency**	**Mode**	**Frequency**	**Mode**	**Frequency**	**Mode**	**Frequency**
1	11.35	1	10.4	1	8.006	1	7.6763
2	26.919	2	26.715	2	22.419	2	16.145
3	27.277	3	32.927	3	24.026	3	16.947
4	32.152	4	41.354	4	26.087	4	22.872
5	90.589	5	78.021	5	78.058	5	35.786
6	119.63	6	78.059	6	92.826	6	56.193

Table 1: Natural Frequencies of the structures

Mode Shapes

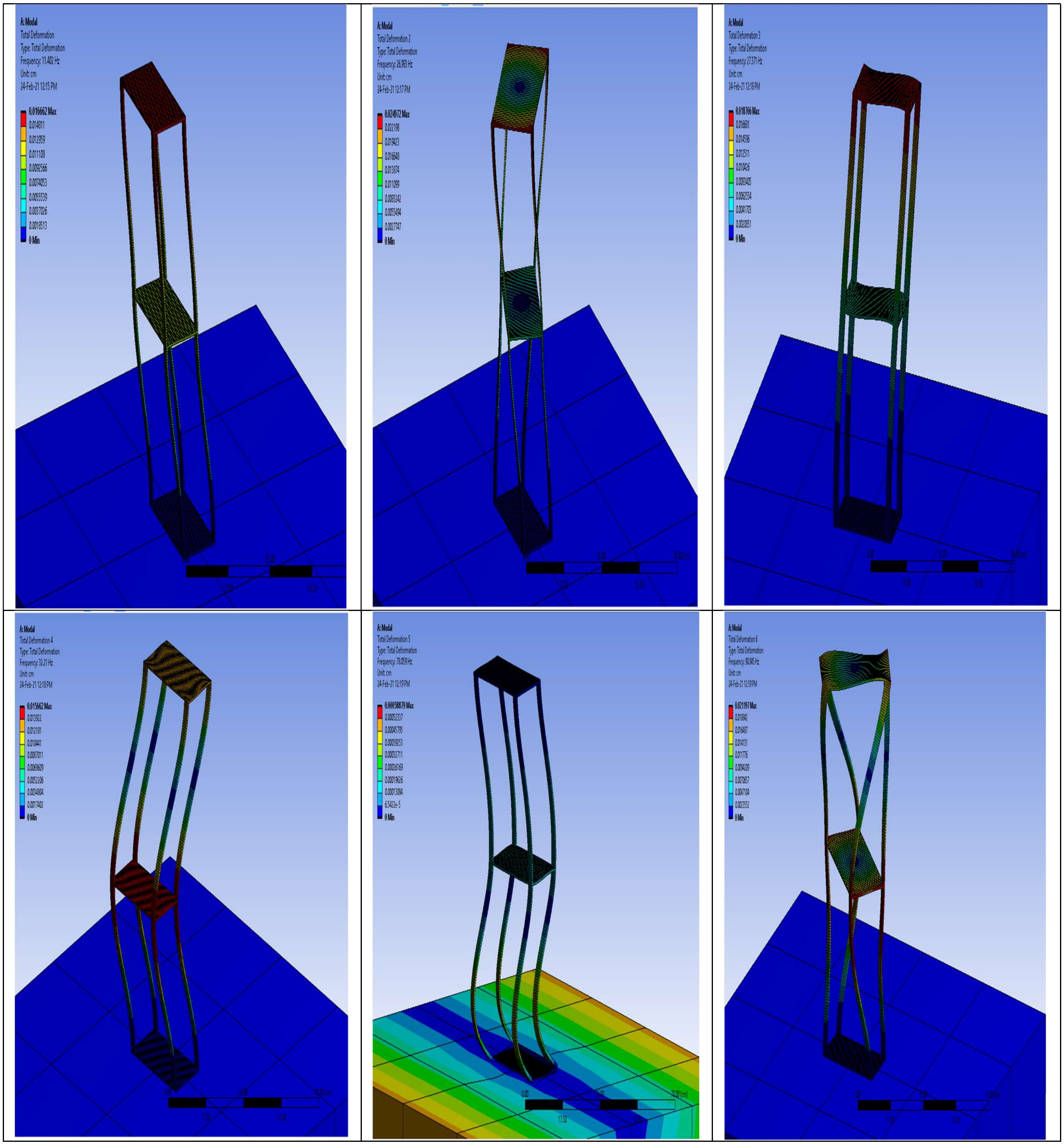

Fig 7: Mode shapes of the unloaded structure.

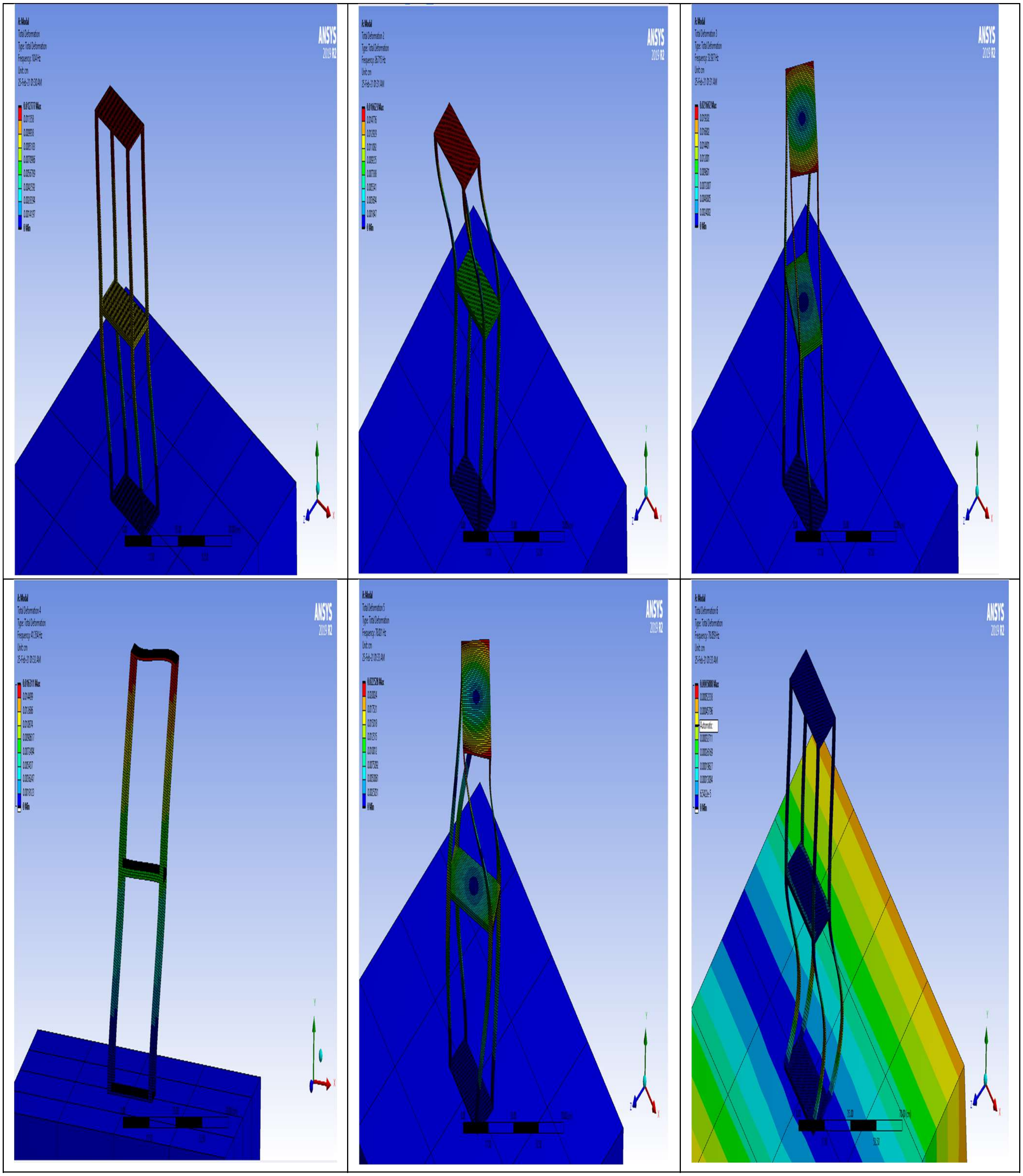

Fig 8: Mode shapes of the structure having distributed mass on the first floor.

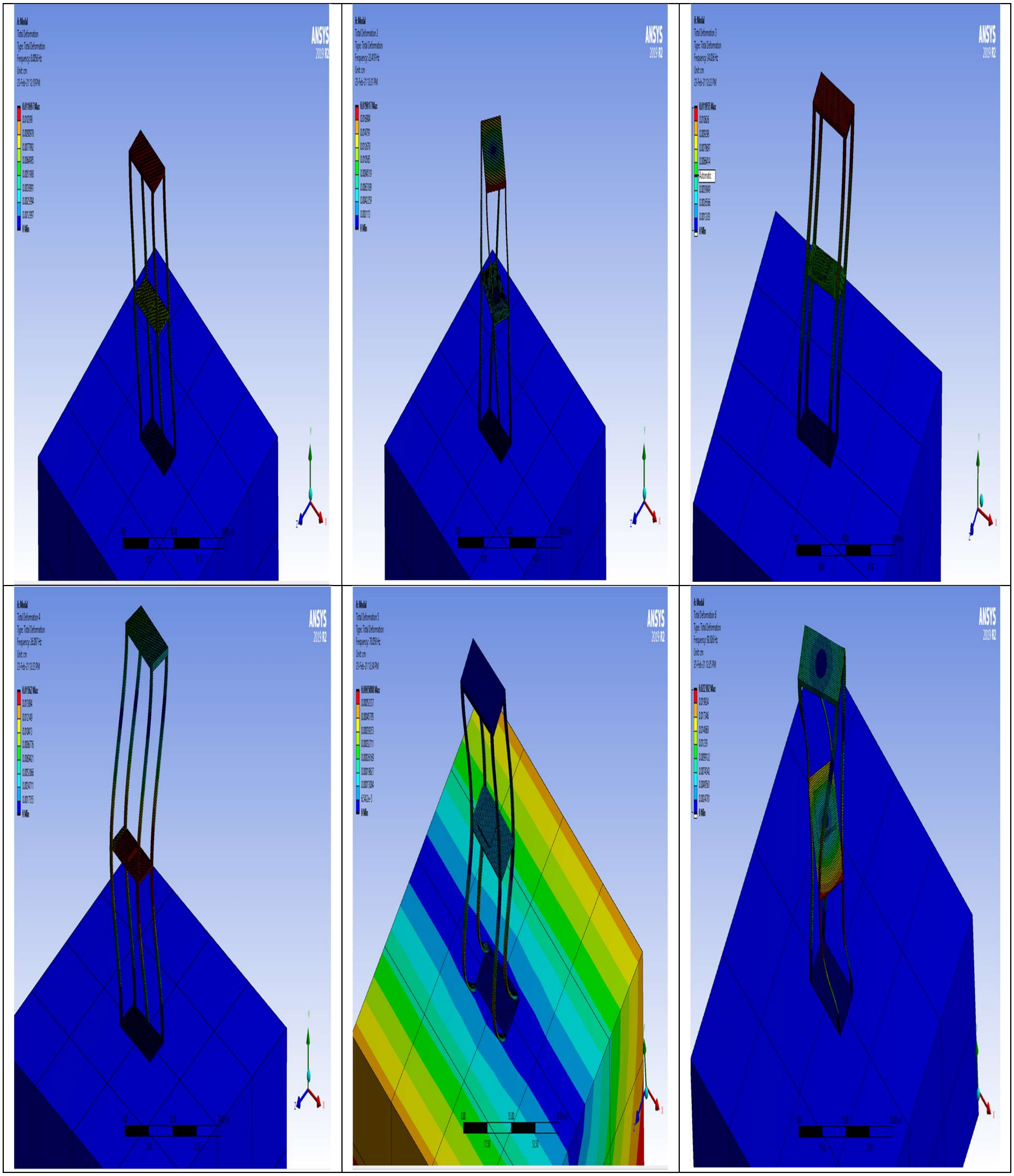

Fig 9: Mode shapes of the structure having distributed mass on the second floor and a simple mass on the first floor.

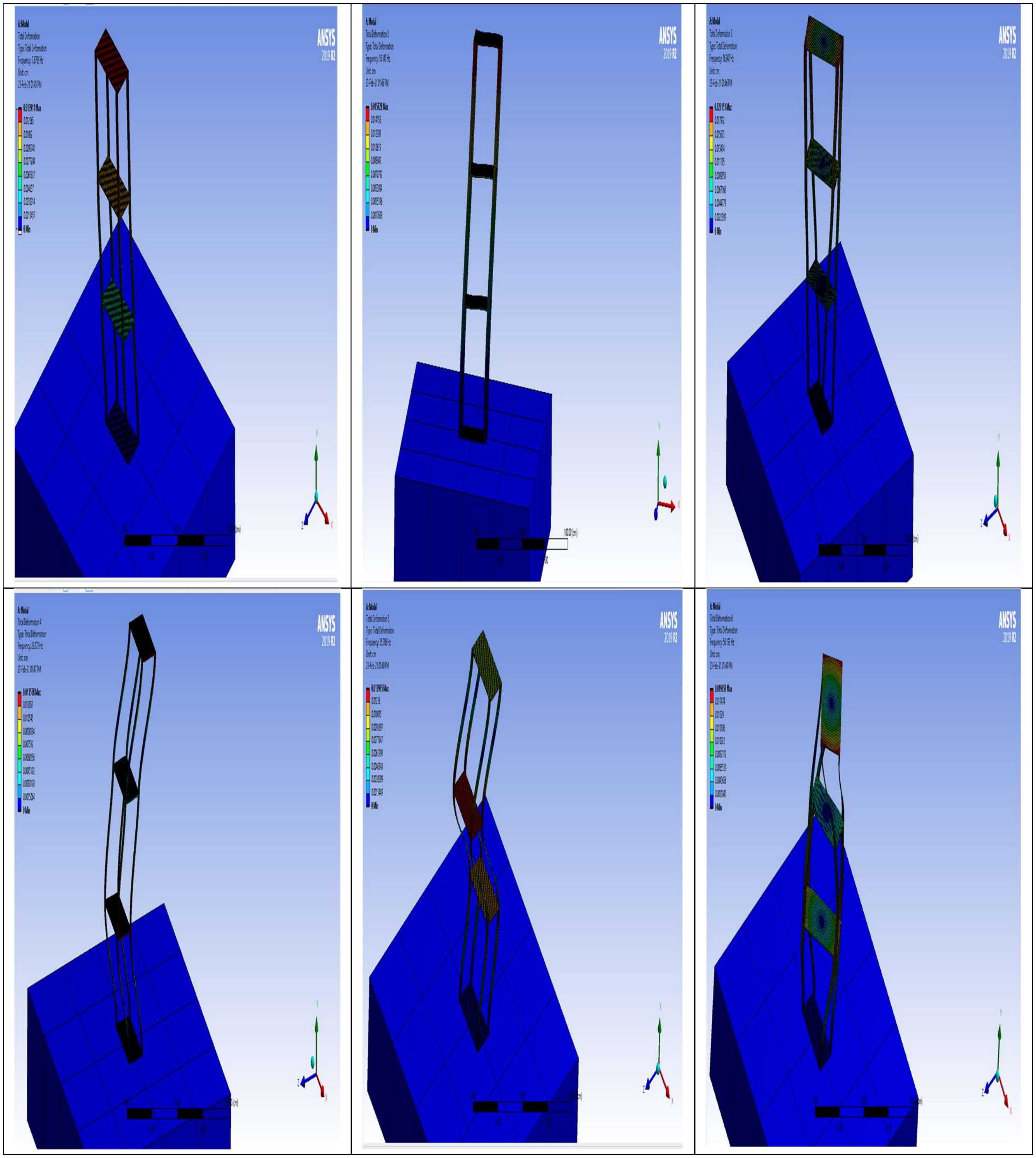

Fig 10: Mode shapes of the three-story unloaded structure.

Harmonic Analysis

After conducting modal analysis, two response points are selected in each structure and a harmonic response curve is generated that gives the maximum deflection of the structure vibrating in its natural frequencies. Here the models are designed on a very small scale as they were intended to go under experimental analysis. That's why the deflections found from the harmonic response curves are also very small. If the models are upscaled they can give an insight into real-life structural analysis scenarios. In this paper, response curves are generated for two arbitrary points in the 4 models.

At first external excitation is applied upon the big base plate in the Z-axis direction. The force has an amplitude of 10000N with a phase angle of 0º.

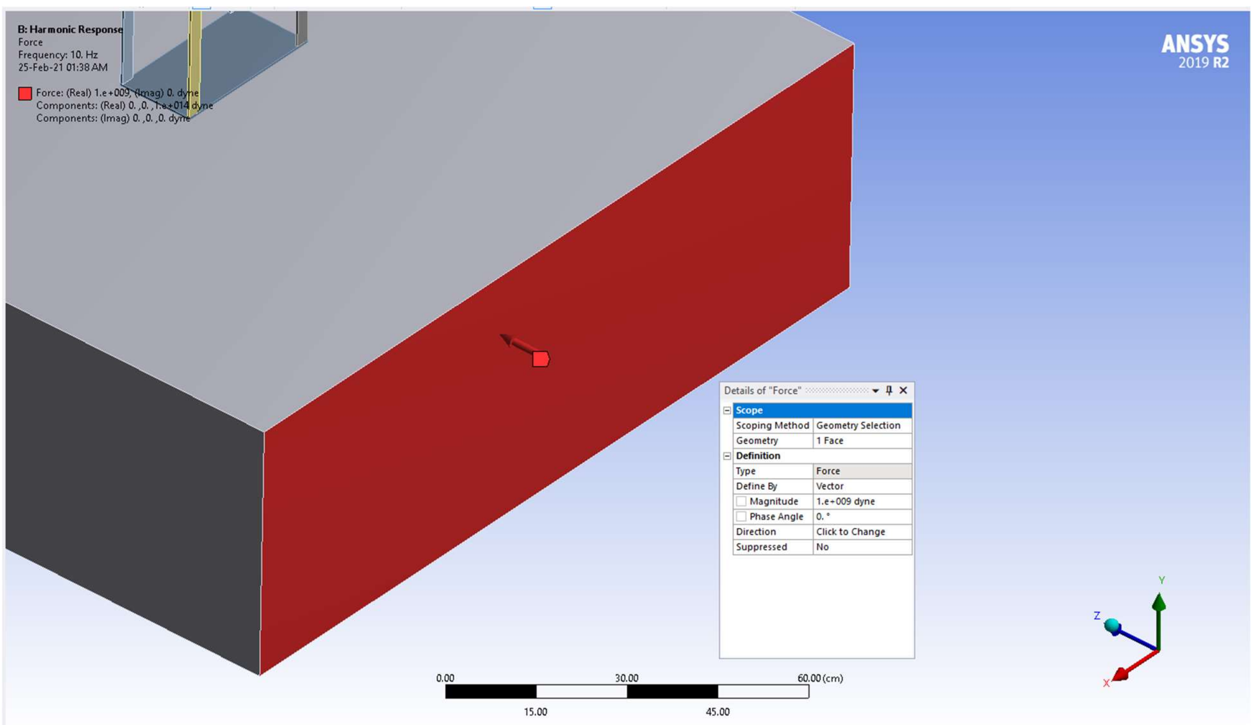

Fig 11: External excitation application.

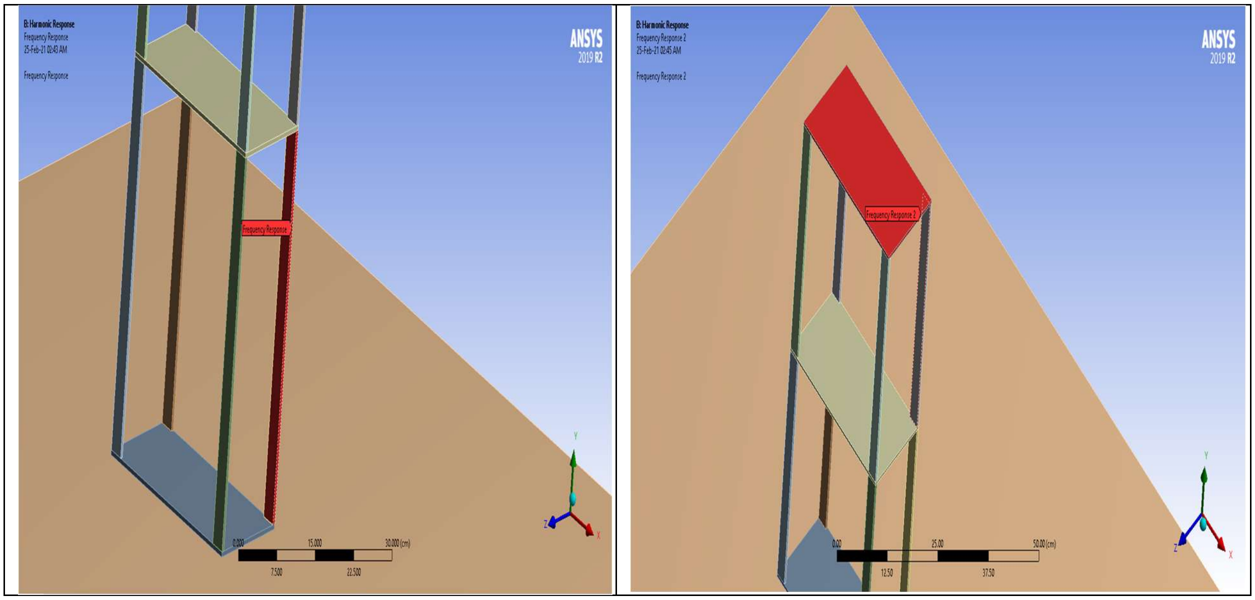

Fig 12: Frequency response points for unloaded structure.

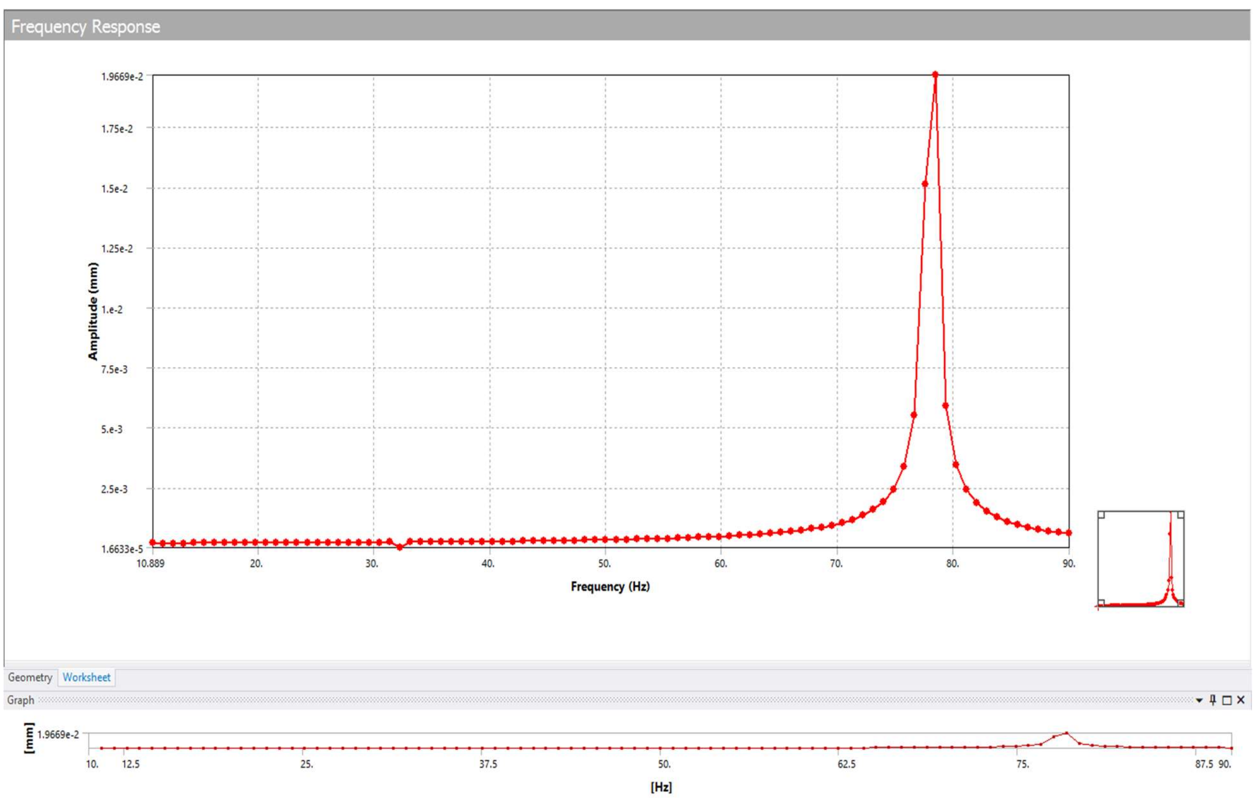

Fig 13: Frequency response curve for point 1 (Unloaded).

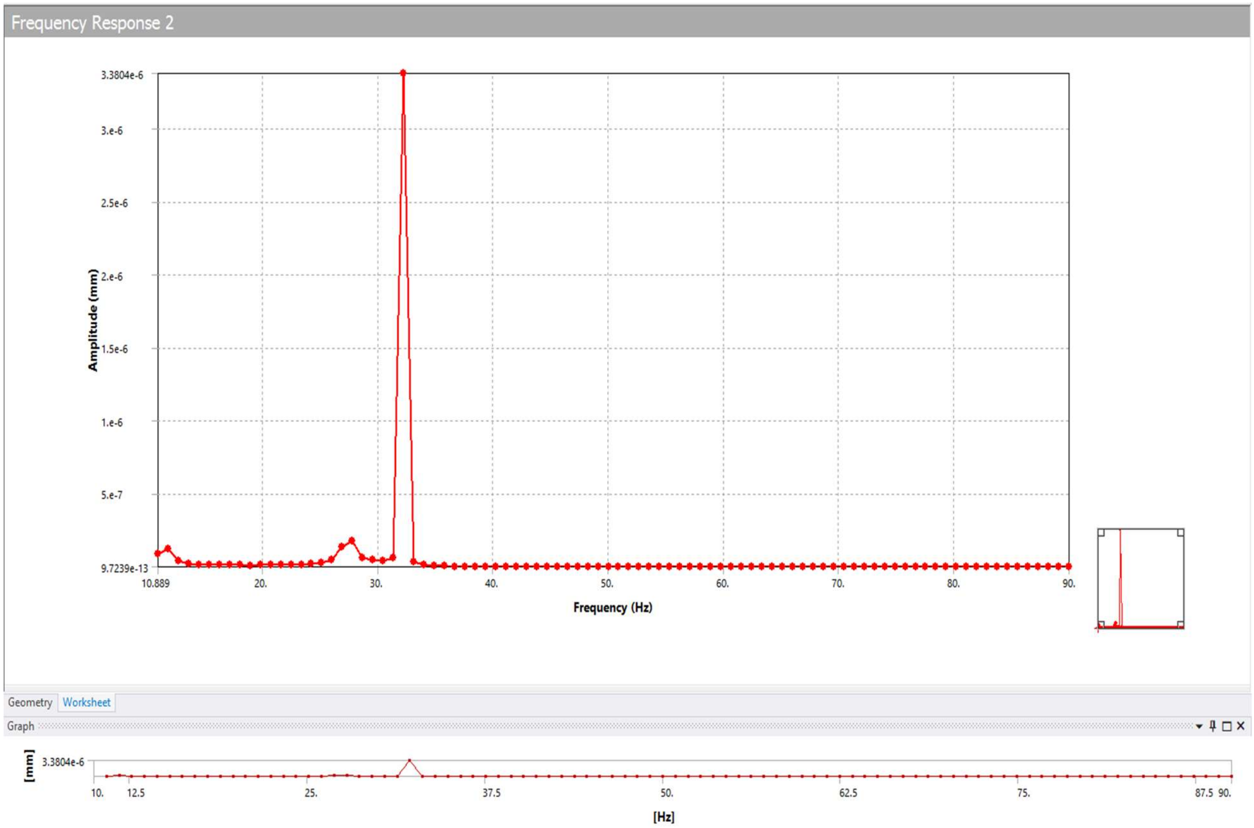

Fig 14: Frequency response curve for point 2 (Unloaded).

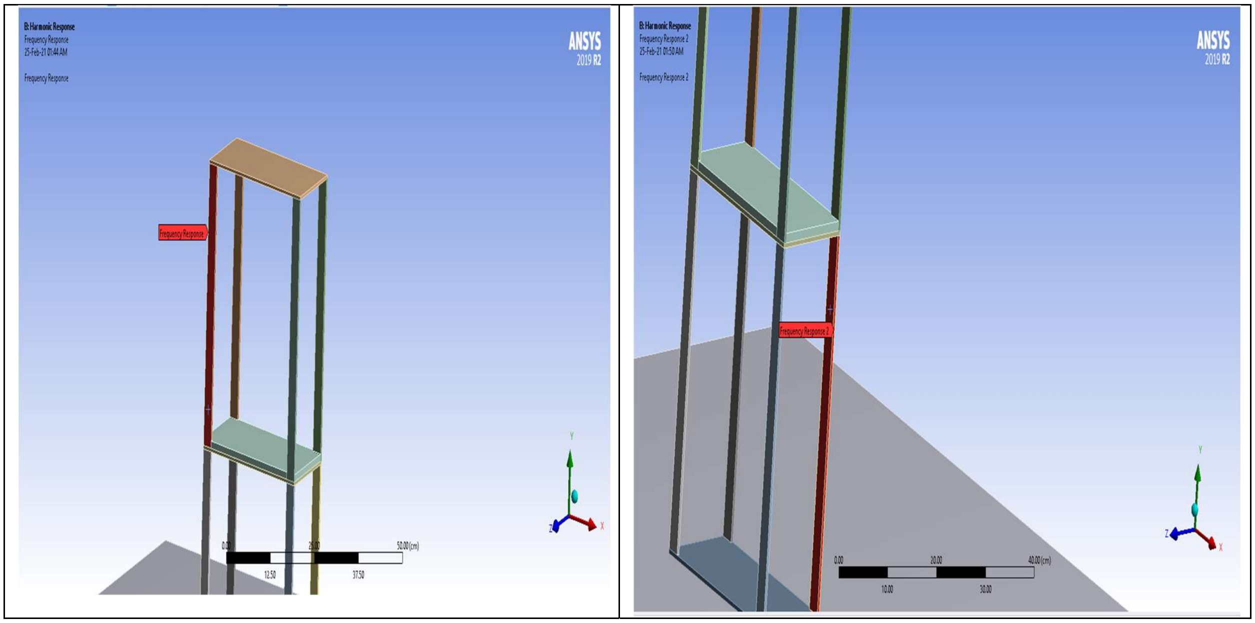

Fig 15: Frequency response points for the structure having distributed mass on the first floor.

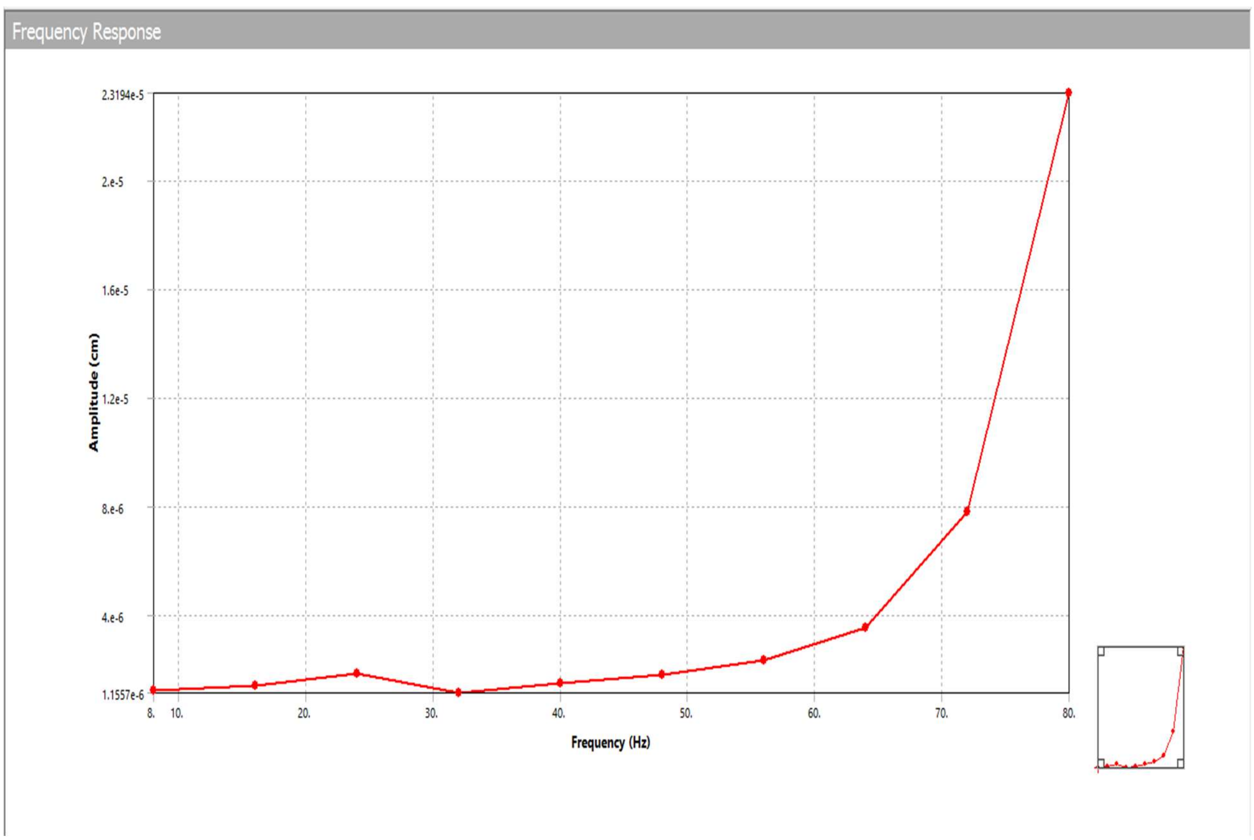

Fig 16: Harmonic response curve for point 1 (distributed mass on the first floor).

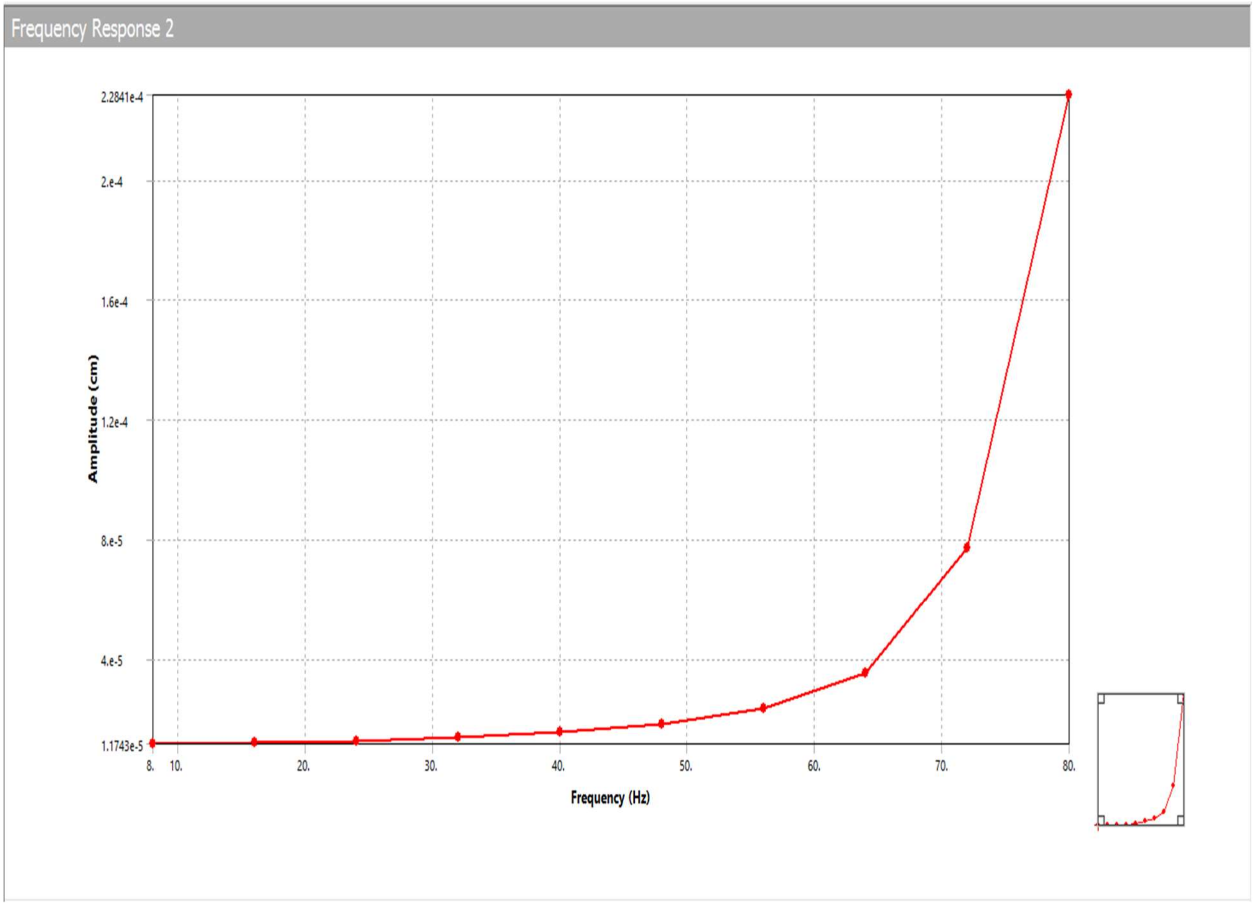

Fig 17: Frequency response curve for point 2 (distributed mass on the first floor).

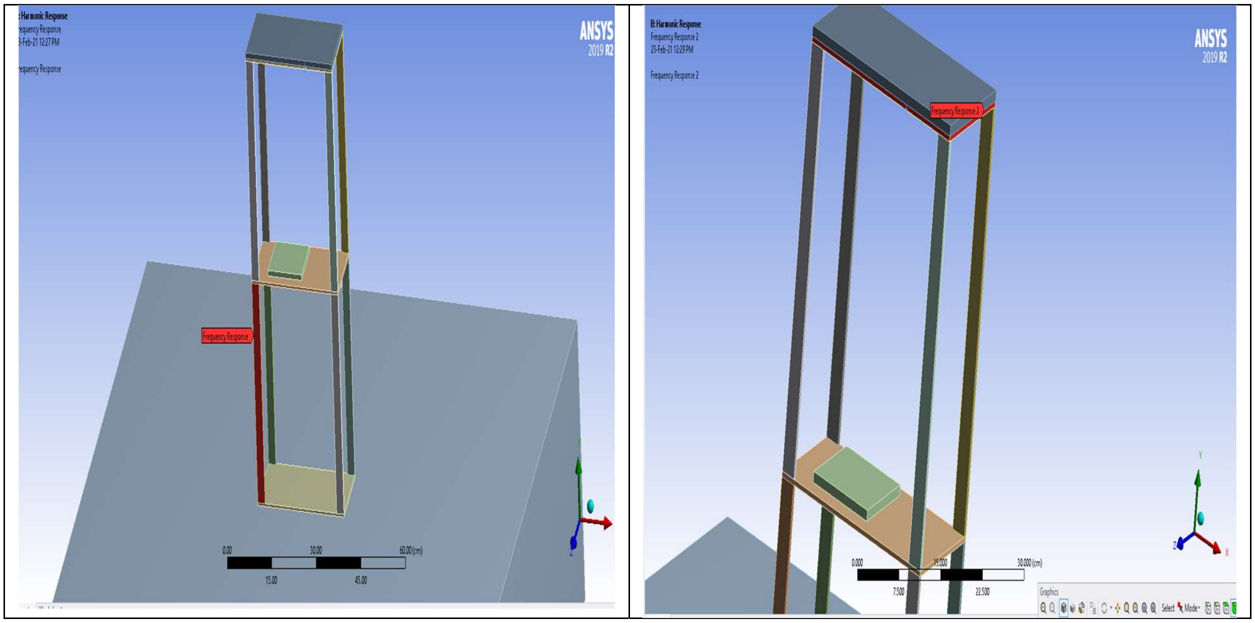

Fig 18: Frequency response point for the structure having distributed mass on the second floor and a simple mass on the first floor.

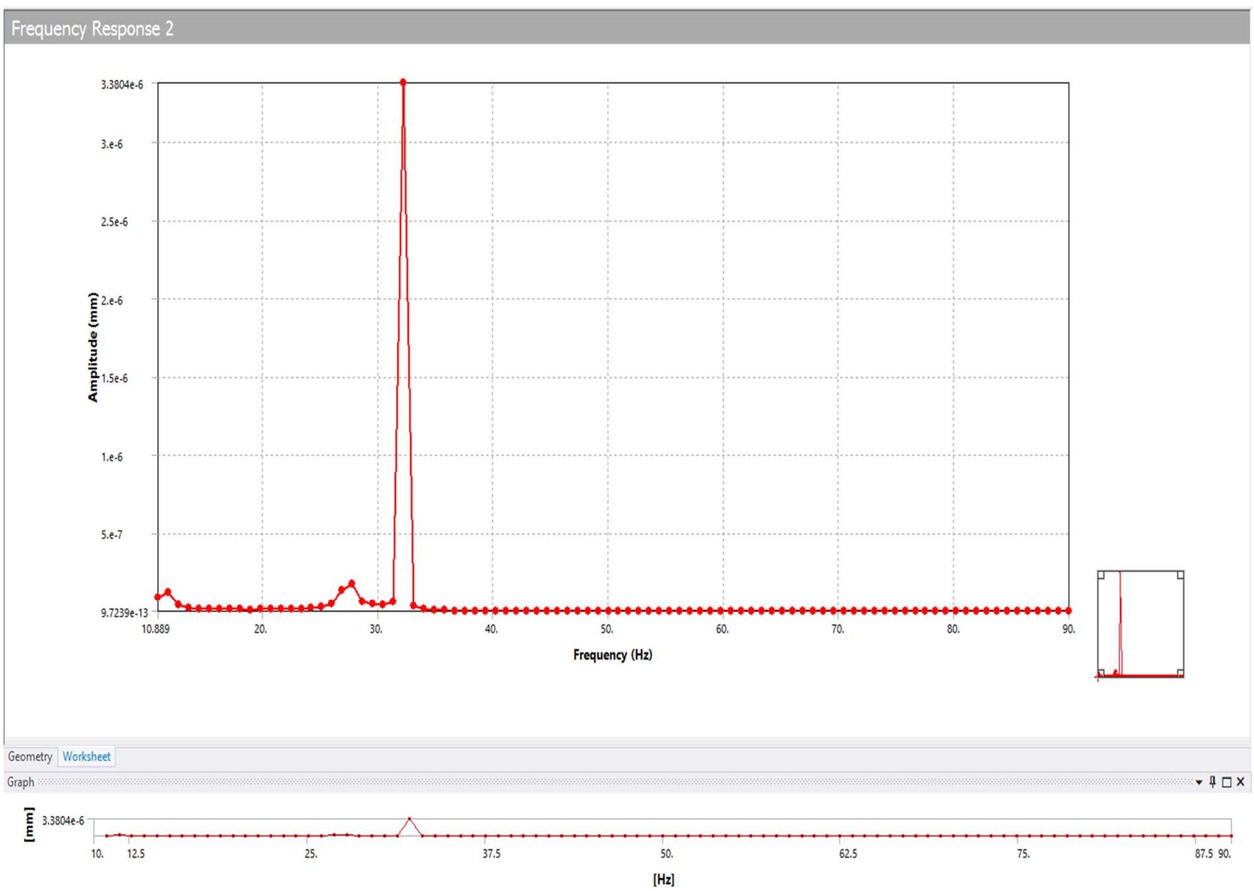

Fig 19: Frequency response curve for point 1 (distributed mass on the second floor)

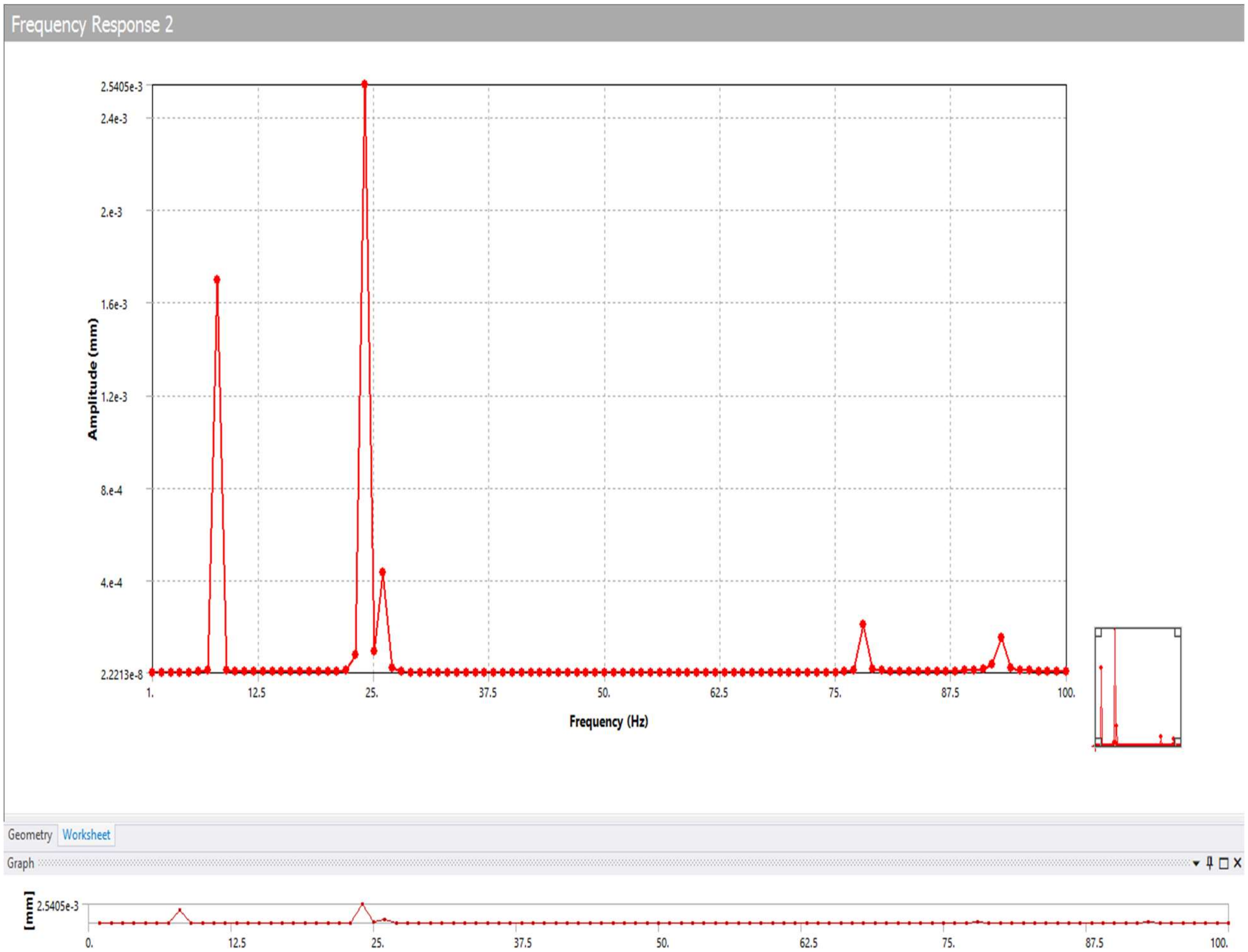

Fig 20: Frequency response for point 2 (distributed mass on the second floor).

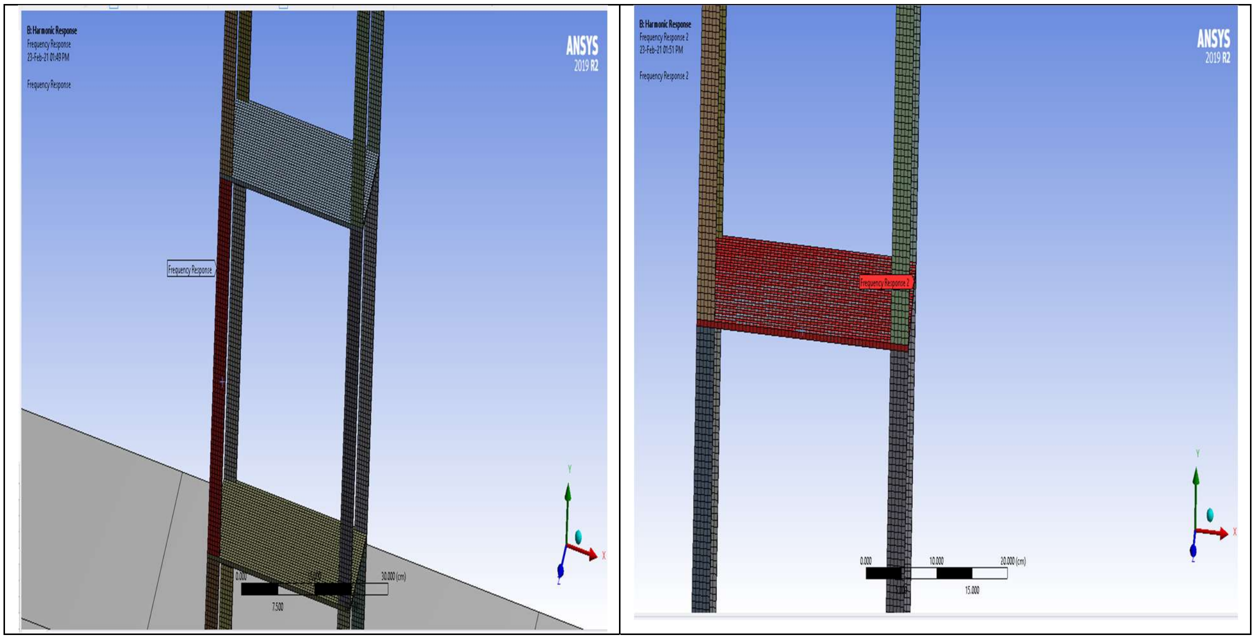

Fig 21: Frequency response point for the three-story unloaded structure.

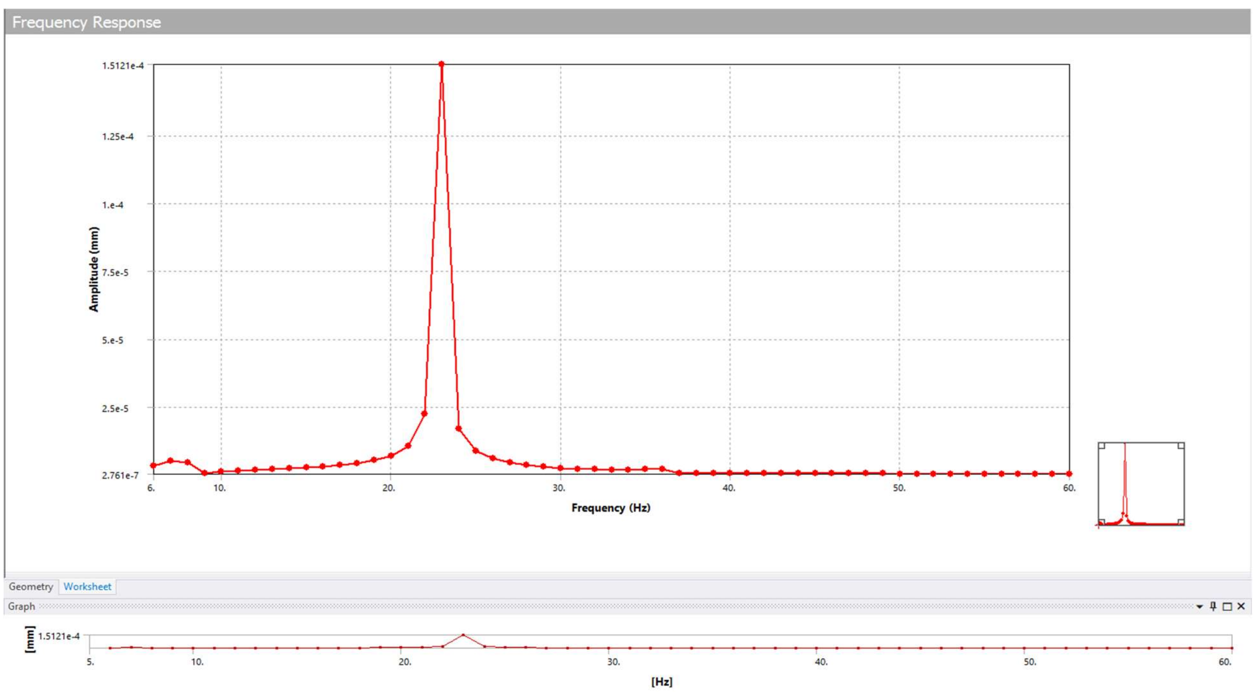

Fig 22: Frequency response curve for point 1 (three-story unloaded).

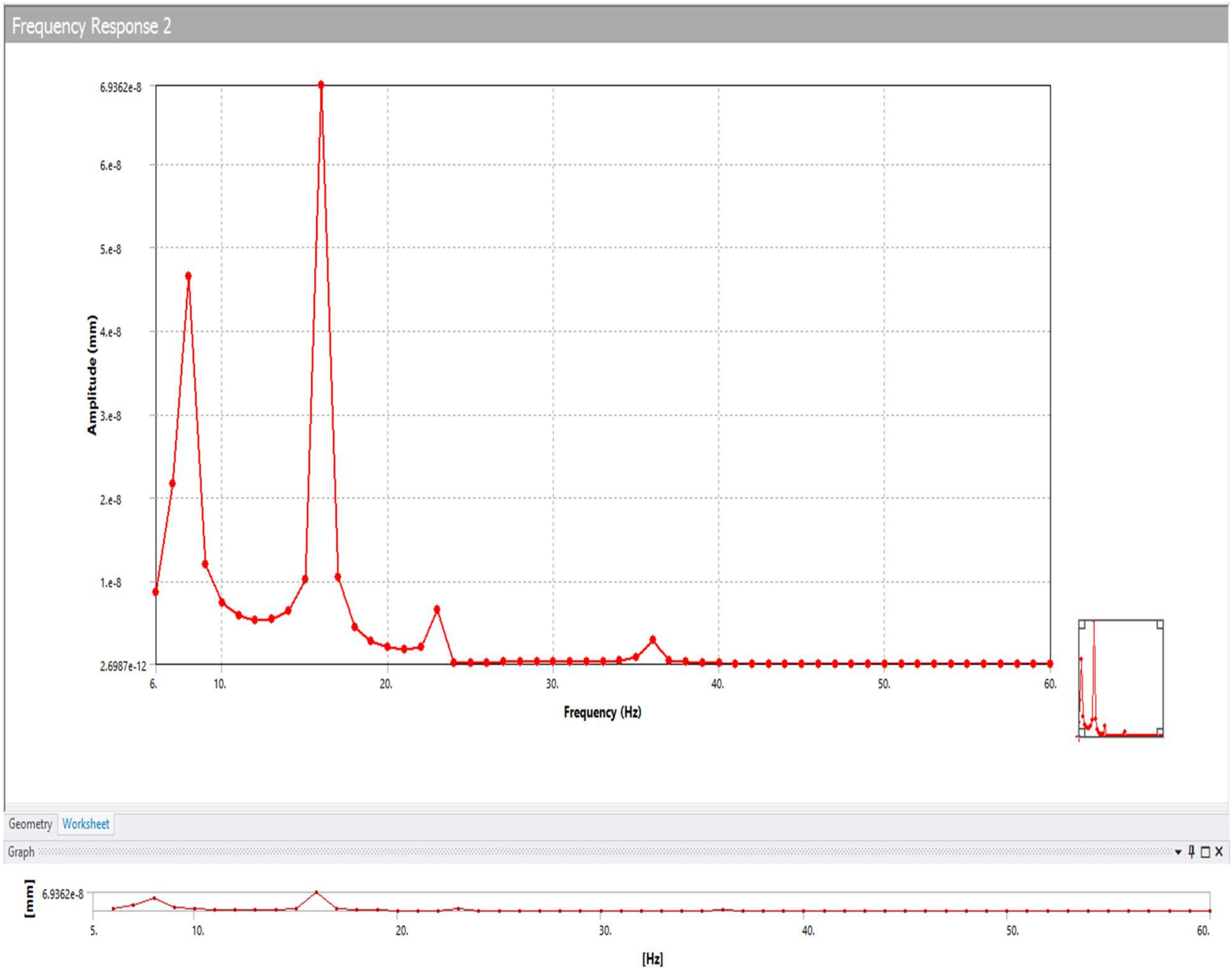

Fig 23: Frequency response curve for point 2 (three-story unloaded).

Observations:

Observing the mode shapes of the models, it is visible that among the 6 mode shapes 4 of them show bending and 2 of them show twisting motion. Though there is no fixed mode number for bending and twisting motions, it can be seen that the first mode for every model shows the same bending deflection towards the Z-axis direction creating just one node. It has maximum impact on the topmost floors. So, when a frequency response point is selected on the top floors, the response curves show a deflection in the first mode's frequency.

Among the four bending deflections, three of them are towards the Z-axis direction. One exceptional bending deflection can be found, having the plates bend towards the Y-axis direction and the structure bend towards the X-axis direction. That is the Mode 3 of unloaded and "distributed mass on the top floor" model, Mode 4 of "distributed mass on the first floor" model, and Mode 2 of the three-story unloaded model.

Another interesting observation is the Mode 5 of the unloaded and "distributed mass on the top floor" model and Mode 6 of the "distributed mass on the first floor" model. From the shape image, it is visible that the big base plate is absorbing some impact and the lower columns are under great deflection. The frequency response point selected in the lower columns gives a value of this deflection in the frequency response curve. This mode generates 3 nodes.

The most interesting finding from the theoretical analysis is that as the structure mass is increased, the natural frequencies for modes decrease though giving away the same mode shapes. This is more visible in the graph below:

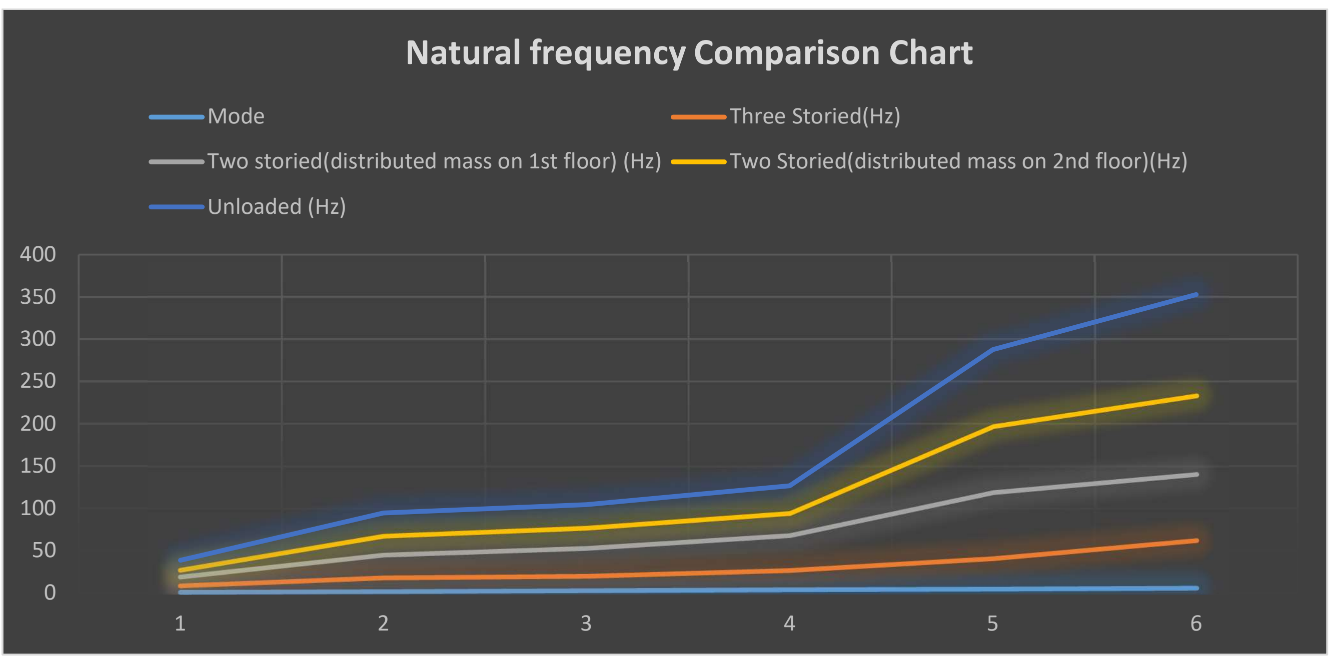

Fig 24: Natural frequency comparison chart

The unloaded two-story structure has the lowest mass thus has bigger natural frequencies. The three-story unloaded structure has the highest mass thus has the smallest natural frequencies.

The harmonic response curve (**Fig 16 & 17**) for the "distributed mass on the first floor" model is quite different from the rest of the harmonic response curves. The reason behind this might be the selection of the frequency response points, in this case, point 1 is the top left column, and point 2 is the bottom right column. To find out if the response curve behaves differently in the case of another response point other than the columns, a separate harmonic analysis is done. The result is shown below:

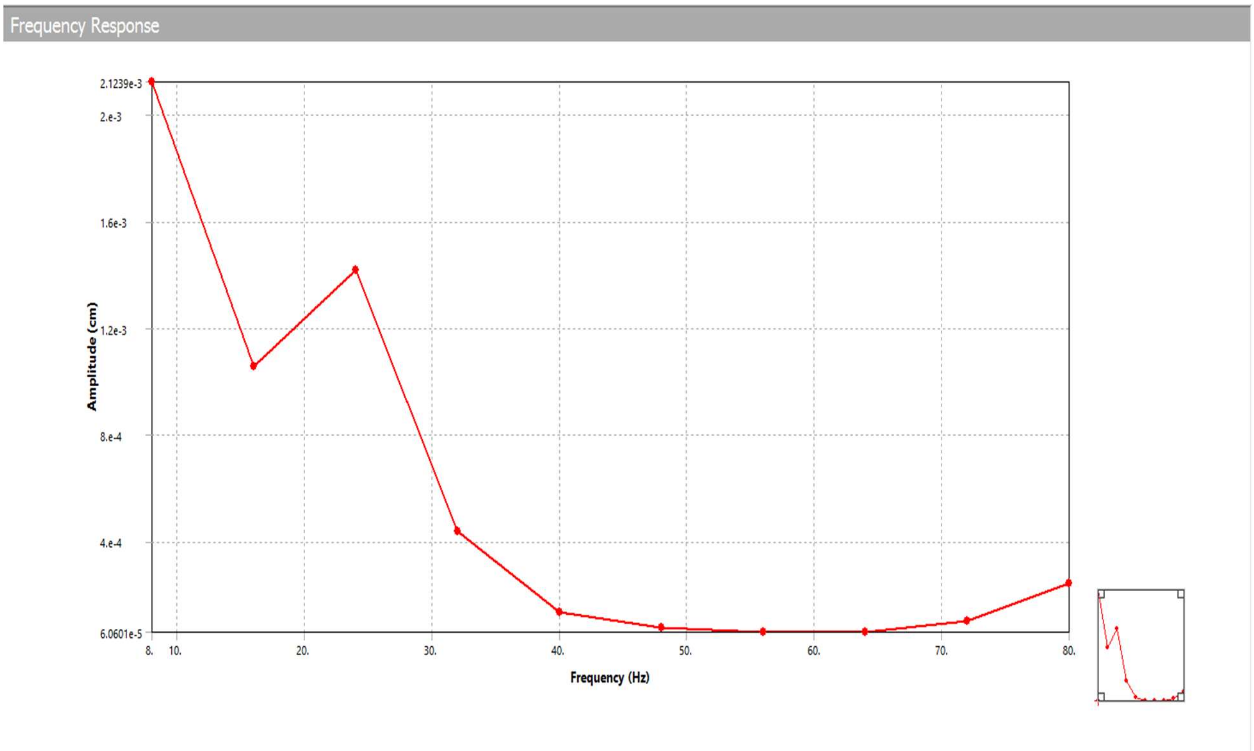

Fig 25: Frequency response curve for the top floor acting as a response point

It gives the maximum deflection right away in the first mode, which is totally different from the other response curves (**Fig 16 & 17**) generated for this model.

External excitation is an important factor in the case of harmonic analysis. The magnitude of the force in these analyses was 10,000N. If this magnitude is increased, it may fetch greater deflections from the structure. These analyses are done on a very small scale as the primary target was to conduct an experimental analysis.

Conclusion:

- Natural frequencies of any structure can be found by conducting Modal Analysis and it's load limit can be tested
- Only bending deflections are focused here and the mode peaks are found according to them
- The highest deflection at any point in the structure can be found by conducting Harmonic Analysis
- Further work can be done upon this project by taking into consideration the torsional motions of the structure also
- Earthquake-related studies can also be conducted. The inclusion of the big base plate and not a simple fixed support is for this particular reason.

References:

[1] M. L. Chandravanshi and A. K. Mukhopadhyay, "Modal analysis of structural vibration," *ASME Int. Mech. Eng. Congr. Expo. Proc.*, vol. 14, no. November 2013, 2013, doi: 10.1115/IMECE2013-62533.

[2] K. A. Ramsey, "Experimental modal analysis, structural modifications and FEM analysis on a desktop computer," *S V Sound Vib.*, vol. 17, no. 2, pp. 19–27, 1983.

[3] M. Narwariya, A. Choudhury, and A. K. Sharma, "Harmonic analysis of moderately thick symmetric cross-ply laminated composite plate using FEM," *Adv. Comput. Des.*, vol. 3, no. 2, pp. 113–132, 2018, doi: 10.12989/acd.2018.3.2.113.

[4] H. Ning, W. Zhigang, J. Chengyu, and Z. Bing, "Finite element method analysis and control stratagem for machining deformation of thin-walled components," vol. 139, pp. 332–336, 2003, doi: 10.1016/S0924-0136(03)00550-8.

[5] M. Petyt, "Introduction to finite element vibration analysis, second edition," *Introd. to Finite Elem. Vib. Anal. Second Ed.*, vol. 9780521191, pp. 1–500, 2010, doi: 10.1017/CBO9780511761195.

[6] Y. Yu, S. Zhang, H. Li, X. Wang, and Y. Tang, "Modal and Harmonic Response Analysis of Key Components of Ditch Device Based on ANSYS," *Procedia Eng.*, vol. 174, pp. 956–964, 2017, doi: 10.1016/j.proeng.2017.01.247.

[7] A. S. Elliott and M. H. Richardson, "Virtual experimental modal analysis (VEMA)," *Proc. Int. Modal Anal. Conf. - IMAC*, vol. 2, pp. 884–890, 1998.

[8] C. M. Ramesha, K. G. Abhijith, A. Singh, A. Raj, and C. S. Naik, "Modal Analysis and Harmonic Response Analysis of a Crankshaft," *Int. J. Emerg. Technol. Adv. Eng.*, vol. 5, no. 6, pp. 323–327, 2015.

[9] B. G. Hearn and R. B. Testa, "M o d a l analysis for damage detection in structures," vol. 117, no. 10, pp. 3042–3063, 1992.

[10] H. Lv and J. Zhao, "The experimental modal analysis of motorcycle structure," *ICEOE 2011 - 2011 Int. Conf. Electron. Optoelectron. Proc.*, vol. 2, no. Iceoe, pp. 275–278, 2011, doi: 10.1109/ICEOE.2011.6013233.

[11] H. Van Der Auweraer, "Structural dynamics modeling using modal analysis: Applications, trends and challenges," *Conf. Rec. - IEEE Instrum. Meas. Technol. Conf.*, vol. 3, pp. 1502–1509, 2001, doi: 10.1109/imtc.2001.929456.

[12] S. Orhan, "Analysis of free and forced vibration of a cracked cantilever beam," *NDT E Int.*, vol. 40, no. 6, pp. 443–450, 2007, doi: 10.1016/j.ndteint.2007.01.010.

[13] W. Tangsopa and J. Thongsri, "Development of an industrial ultrasonic cleaning tank based on harmonic response analysis," *Ultrasonics*, vol. 91, no. July 2018, pp. 68–76, 2019, doi: 10.1016/j.ultras.2018.07.013.

[14] L. Bertini, P. Neri, C. Santus, and A. Guglielmo, "One exciter per sector test bench for bladed wheels harmonic response analysis," *Proc. ASME Turbo Expo*, vol. 7B-2017, no. June, 2017, doi: 10.1115/GT2017-63628.

[15] D. Laxalde, F. Thouverez, and J. P. Lombard, "Forced response analysis of integrally bladed disks with friction ring dampers," *J. Vib. Acoust. Trans. ASME*, vol. 132, no. 1, pp. 0110131–0110139, 2010, doi: 10.1115/1.4000763.

[16] Y. Ishida and T. Inoue, "Detection of a rotor crack using a harmonic excitation and nonlinear vibration analysis," *J. Vib. Acoust. Trans. ASME*, vol. 128, no. 6, pp. 741–749, 2006, doi: 10.1115/1.2346693.

[17] C. Villa, J. J. Sinou, and F. Thouverez, "Stability and vibration analysis of a complex flexible rotor bearing system," *Commun. Nonlinear Sci. Numer. Simul.*, vol. 13, no. 4, pp. 804–821, 2008, doi: 10.1016/j.cnsns.2006.06.012.

[18] M. A. Al-Shudeifat, E. A. Butcher, and C. R. Stern, "General harmonic balance solution of a cracked rotor-bearing-disk system for harmonic and sub-harmonic analysis: Analytical and experimental approach," *Int. J. Eng. Sci.*, vol. 48, no. 10, pp. 921–935, 2010, doi: 10.1016/j.ijengsci.2010.05.012.

[19] E. Prestini, "Introduction to harmonic analysis," *Appl. Numer. Harmon. Anal.*, no. 9781489979872, pp. 25–52, 2016, doi: 10.1007/978-1-4899-7989-6_2.