Engineering and Technology Journal 41 (11) (2023) 1390 -1402

Engineering and Technology Journal
Journal homepage: https://etj.uotechnology.edu.iq

A Static and Dynamic Analysis of A High-Speed Turbo Machine Foundation

Ameer A. Ahmed, Mohammed Y. Fattah, Makki K. Mohsen
Civil Engineering Dept., University of Technology-Iraq, Alsina'a street, 10066 Baghdad, Iraq.
*Corresponding author Email :Ameer.A.Ahmed@uotechnology.edu.iq.

HIGHLIGHTS

- Ansys software was utilized to verify and compute the natural frequencies of a frame foundation
- The effect of machine's mass on the frame's calculated natural frequencies is studied.
- The effect of the machine's mass elevation on dynamic response and calculated natural frequencies is investigated

ARTICLE INFO

Handling editor: Mahmoud S.Al-Khafaji

Keywords:
Modal analysis
Natural frequency
Frame foundation
Ansys workbench
Harmonic analysis

ABSTRACT

The foundation plays a significant role in safe and efficient turbo machinery operation. Turbomachinery generates harmonic load on the foundation due to its high-speed rotating motion, which causes vibration in the machinery foundation. Any increase in machine vibration reduces the machine's performance. In studying engineering problems, using numerical programs helps get a well-designed foundation. Before conducting a parametric study, the program used shall be validated and/or verified by conducting a series of analyses to ensure that the program output represents the reference studied cases well. In many cases, the real problem is simplified by assuming that the machine's loads are applied at the upper slab of the frame foundation. The present work included the effect of the machine's mass elevation on the response of a high-speed turbo machine's frame foundation. The mass elevations of (12.6, 13.0, 13.3, 13.6, 13.9, and 14.2) m are selected for the dynamic analyses. A verification process is carried out to calculate the three-dimensional frame foundation's natural frequencies and mode shapes using Ansys software. The results show that the machine's masses must be included and applied at its specific elevation to reflect the true dynamic vertical response of the system as far as the mass height to top slab elevation ratio exceeds 5%. The results show a good agreement in calculating static and modal analysis with the study case. When neglecting the machine mass, the difference between the calculated natural frequencies for any mode shape is less than 10%.

1. Introduction

Turbo-generators are power generation machinery used in the power plants. It is the most vital and expensive equipment of a power plant complex and is generally placed inside a powerhouse. The turbo-generator foundation consists of a turbine, generator, and auxiliaries mounted on a reinforced cement concrete (RCC) table-top structure consisting of the top deck, columns, and bottom raft. Considering the difficult natural parameters, enormity of the machines, and risk involved in public outcry, the analysis and design of turbo-generator foundations remain one of the most challenging tasks in civil engineering. A key ingredient to the successful foundation design for a turbo-generator is the careful engineering analysis of the foundation response to the dynamic loads from the anticipated operation of the machine [1].

Before studying any geotechnical engineering problem or conducting a parametric study using the finite element method FEM, the program used shall be validated and/or verified by conducting a series of analyses to ensure that the specified program output well represents the reference studied case. These reference cases are sometimes the results of field prototype tests, or they may represent laboratory tests on scaled models representing the prototype structure.

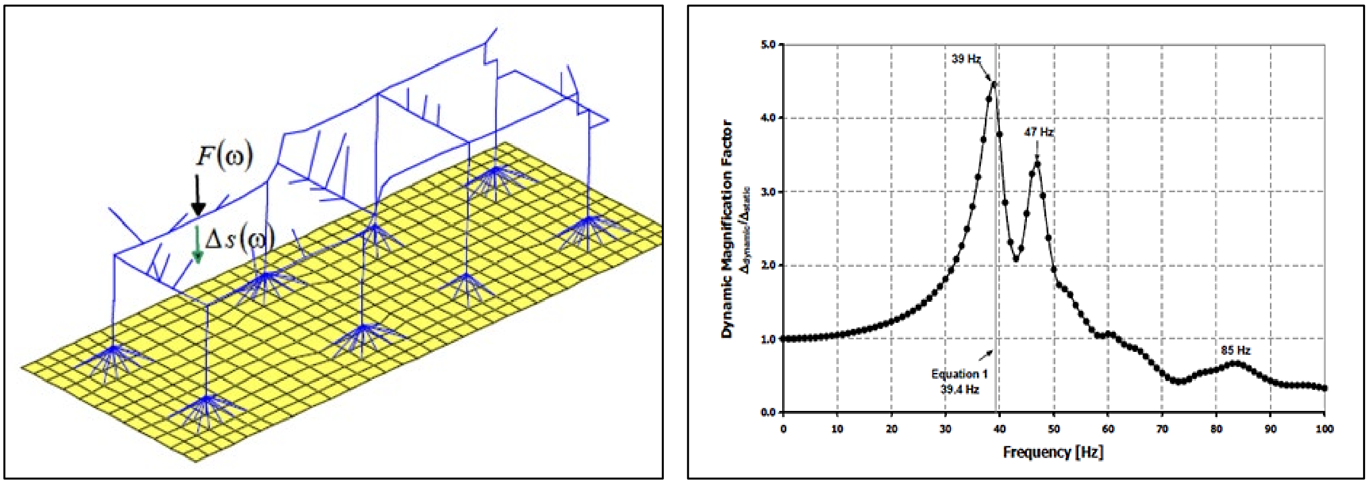
Based on the operation speed, prototype machines can be classified as follows:

1) Very low-speed engines have a rate of less than 100 rpm.
2) Low-speed machines have a speed of 100-1500 rpm.
3) Medium-speed machines have a speed of 1500-3000 rpm.
4) High-speed machines have a speed of 3000 rpm and above [2].

http://doi.org/10.30684/etj.2023.142820.1547
Received 30 August 2023; Received in revised form 30 September 2023; Accepted 05 October 2023; Available online 13 November 2023
2412-0758/University of Technology-Iraq, Baghdad, Iraq

When dealing with laboratory (1-g) or centrifugal (n-g) tests, the required machine's speed must be scaled up by multiplying the speed of the prototype machine by the appropriate scaled factor to fully simulate the behavior of the prototype machine and eventually extrapolate the laboratory model results to prototype model.

Moreschi and Farzam [3] discussed identifying vibration properties for individual structural members of tabletop-type turbine-generator foundations, as shown in Figure 1 (a and b). A procedure for accurately determining local natural frequencies using the harmonic analysis technique is outlined. Case studies are presented which involve applications of the proposed approach to the design of large steam-turbine-generator foundations.



a b

Figure 1: a) FE model of the steam turbine generator and b) frequency response curve [3]

Bhatia [4] described the design aids/methodologies for foundation design. Various issues related to mathematical modeling and interpretations of results are discussed at length. Details of designing a vibration isolation system for heavy-duty machines were also discussed. Influences of dynamic characteristics of foundation elements, viz., beams, columns, pedestals, etc., on the machine's response, along with some case studies, are also presented. The paper also touches upon the effects of earthquakes on machines and their foundations. Using commercially available finite element packages for analysis and design of the foundation is strongly recommended but with caution. His paper highlights the need for better interaction between foundation designers and machine manufacturers to ensure improved machine performance.

Fattah et al. [5,6] studied experimentally the response and behavior of machine foundations resting on dry and saturated sand. A physical model was manufactured to simulate steady-state harmonic load at different operating frequencies to investigate the response of soil and footing to steady-state dynamic vertical loading. The footing parameters are related to the size of the rectangular footing and the embedment depth. Two sizes of rectangular steel model footing were tested at the surface and 50mm depth below the model surface. The investigated parameters of the soil condition include dry and saturated sand for two relative densities of 30% and 80%. The response of the footing was elaborated by measuring the displacement amplitude by the vibration meter. It was concluded that the maximum displacement amplitude response of the foundation resting on dry sand models is more than that on the saturated sand by about 5.0–10%. The maximum displacement amplitude of the footing is reduced to half when the footing size is doubled for dry and saturated sand. The foundation's final settlement (St) increases with increasing the amplitude of dynamic force, operating frequency, and degree of saturation.

Abbas et al. [7] used the three-dimensional-finite element analysis modeling program Abacus (v 6.13) to investigate the dynamic response of the block-type turbine-generator foundation of Al-Mansurya power plant station in Iraq under the vertical harmonic excitation; layered-soil with linear elastic model considering and ignoring the soil-structure interaction effect were concerned in the study. The soil structure interaction has been analyzed directly (i.e., the soil and structure are analyzed together in one step). Free vibration analysis was also performed to find the natural frequencies and the corresponding mode shapes in addition to force vibration analysis. They concluded that the soil structure interaction must be considered when analyzing such sensitive structures due to its significant effect on the overall response.

Tripathy and Desai [8] developed a computational model for the turbo machinery table-top foundation in SAP 2000 software to analyze the effect of Kathmandu earthquake (2015) in five different soil conditions, i.e., very hard generic rock, generic rock, generic soil, NEHRP C class, and NEHRP D class. The results revealed that turbo machinery foundations with barrettes can be used in seismic areas as barrettes safeguard the foundation by absorbing/reducing the seismic load due to high specific surface and side resistance. However, for poor soil conditions like NEHRP D (clay soil), it was found that barrettes alone are insufficient to limit the vibrations induced by dynamic loading due to the rotating motion of the types of machinery or seismic loading. Geosynthetic applications and barrettes considerably reduce the vibrations at the top deck for poor soil conditions. Soil has been modeled as links with 6 degrees of freedom (DOF) and assigned to each mesh of the raft and each node of barrettes. Machine components are modeled as rigid links placed on the top deck of the foundation. The rotating speed of the machine is considered as 3995rpm in the study. The unbalanced forces are due to the weight of the motor and the rotational motion. Sine functions are added at the top deck to model the machine's harmonic, dynamic loads transmitted through rigid links. Similarly, earthquake time history is applied at the bottom of the raft for carrying out seismic excitation and vibration analysis.

Bhattacharya [9] discussed through a practical example the step-by-step procedure adopted in designing a table-top foundation supported on piles for a steam turbine generator with an operating speed of 60 Hz. The finite element model of the table-top foundation is generated in ANSYS. Analysis results are used to perform the static design checks. Dynamic analysis was performed to check for the foundation's resonance and allowable amplitude limits as specified by the machine vendor. For dynamic analysis, machine mass is modeled at each sole plate location by adjusting the mass density of the embedded volumes at each sole plate location. Foundations are analyzed for unbalanced loads of the machine at the bearing locations. These loads either act in-phase or out-of-phase with each other, and effects are estimated for each condition.

Rajkumar et al. [10] studied the influence of soil-structure interaction (SSI) on a torsionally coupled turbo-generator (TG) machine foundation, as shown in Figure 2, under earthquake ground motions by using a three-dimensional finite element model. The beneficial effects of base isolators in the TG foundation under earthquake ground motions are also studied properly, considering the effects of SSI. Two superstructure eccentricity ratios are considered to represent the torsional coupling. Soft soil properties are considered to study the effects of SSI. They concluded that the effects of torsional coupling alter the natural frequencies, which could lead to unsafe design if ignored. The deck accelerations and displacements are increased with an increase in superstructure eccentricity. On the other hand, the deck accelerations and displacements are greatly reduced with the help of base isolators, thus confirming the beneficial use of base isolators in machine foundations to protect sensitive equipment from strong earthquake ground motions. However, the effects of SSI reduce the natural frequencies of the TG foundation resting on soft soil conditions and activate the higher mode participation, amplifying the response.

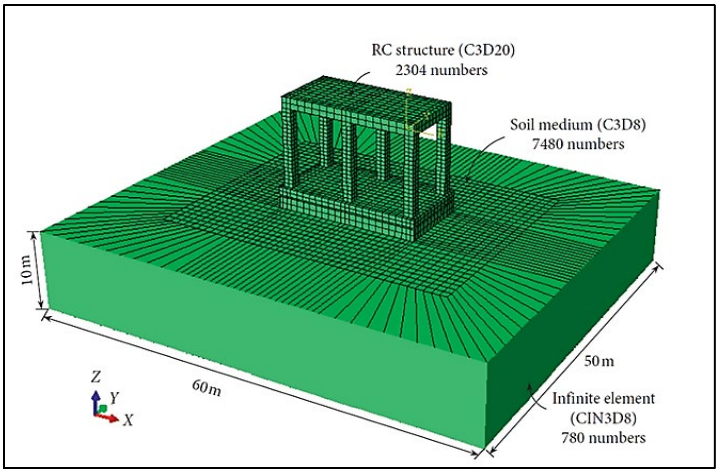


Figure 2: Three-dimensional FE model of framed type machine foundation resting on soil medium [10]

Experiments on machine foundations (circular and rectangular) resting on clay soil with different degrees of saturation (100% and 60%) were carried out by Abdulrasool et al. [11] to find the displacement amplitude of the foundation under different operating frequencies. In addition, the paper describes the vertical stress and displacements inside the soil distributed at three points under the foundation (0.5 B, B, and 2B, B is the foundation width). The experiment results showed that the effect of increasing the degree of saturation is to reduce the amplitude displacement of the foundation to about 61%.

The process of comparing the results of the prototype or model behavior with the results generated by the conceptual model of the FEM is called validation. On the other hand, when we compare the results of a specific program results with another program's results for the same problem on a prototype or laboratory model, and whether the two programs produce consistent results, the process is called verification. This process focuses on the implementation and agreement between different computational models, irrespective of the real-world data [12].

The goal of this study is to determine the effect of applying the machine's masses at different elevations above the surface of the top slab to clarify the effect of the machine's masses on the responses of frame foundation in x,y, and z directions, to reach this goal, a verification process is carried out to determine how well the numerical program conceptual model predicted results would simulate the prototype or model measurement or behavior.

2. Frame Foundation Problem

The turbo-generator forms the heart of a power plant. It is the most vital and expensive equipment of a power plant complex and is generally housed inside a turbo-generator building. A turbo-generator consists of a turbine, generator, and other auxiliaries like condensers, pipelines carrying superheated steam, etc. Turbo-generator falls under high-speed rotary-type machines, and its capacity varies from 2 MW to 2000 MW. The turbo-generator foundation and its auxiliaries are mounted on a tabletop foundation. The foundation can be either made of steel or RCC. A RCC table top type foundation is commonly adopted. The top deck, column, and bottom raft constitute the turbo-generator foundation. Sometimes, the turbo-generator foundation is mounted on a vibration isolator.

The present section focuses on the verification process of Ansys Workbench software with a reference study case representing a frame foundation subjected to dynamic loading produced by a high-speed rotary machine. The case study was published in the well-known book entitled "Foundations for Industrial Machines Handbook for Practicing Engineers" by Bhatia [2].

2.1 Case Study Description

The case study consists of a concrete frame foundation for supporting a high-speed rotating machine type, as shown in Figure 3. The prototype frame structure consists of four columns of (1.0×1.0) m and two columns of (1.1×1.0) m, the table top concrete slab of dimensions (8.0×13.8×1.8) m, which contains four bearing locations for the turbine and generator, and finally a lower raft foundation of dimensions (8.0×13.5×2.0) m.

Figure 4 shows the load-bearing locations of the rotating machines, including the turbine and generator. Table 1 shows the value of the total static and the dynamic loads exerted by the machine and its rotating masses on the specified bearing points on the upper table-top of the frame foundation.

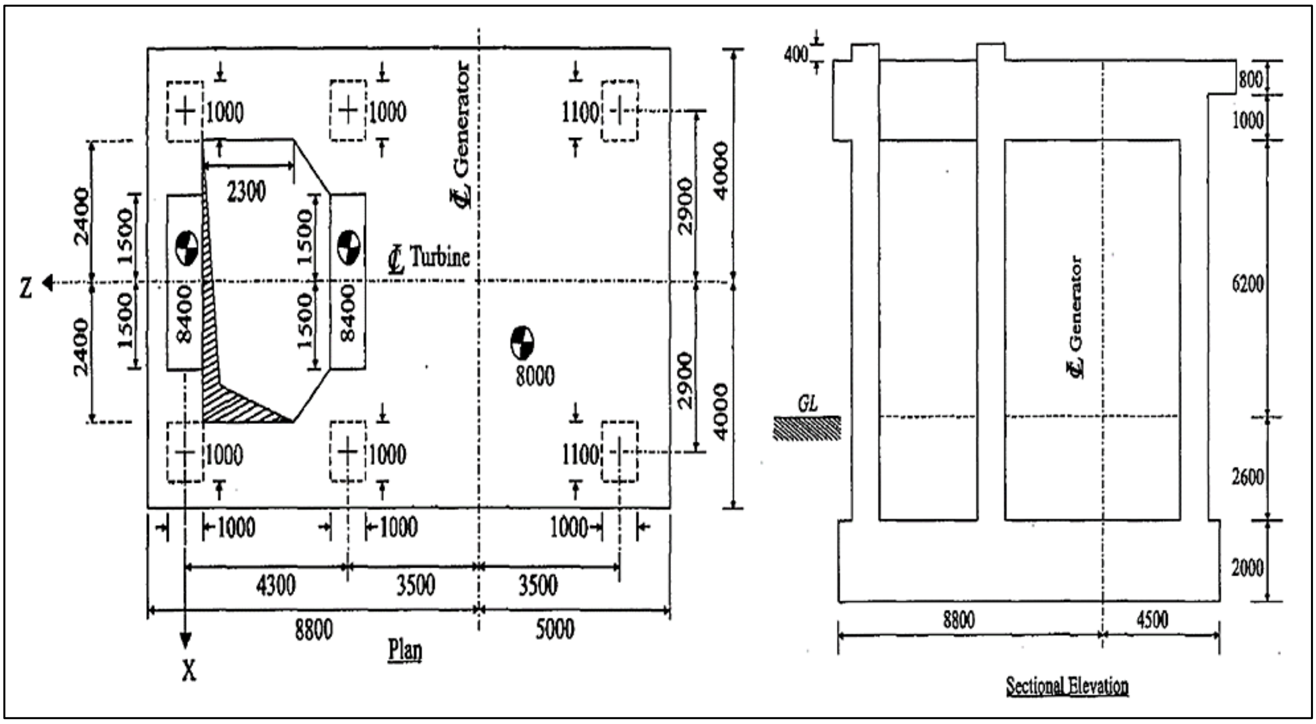


Figure 3: Details of frame foundation [2]

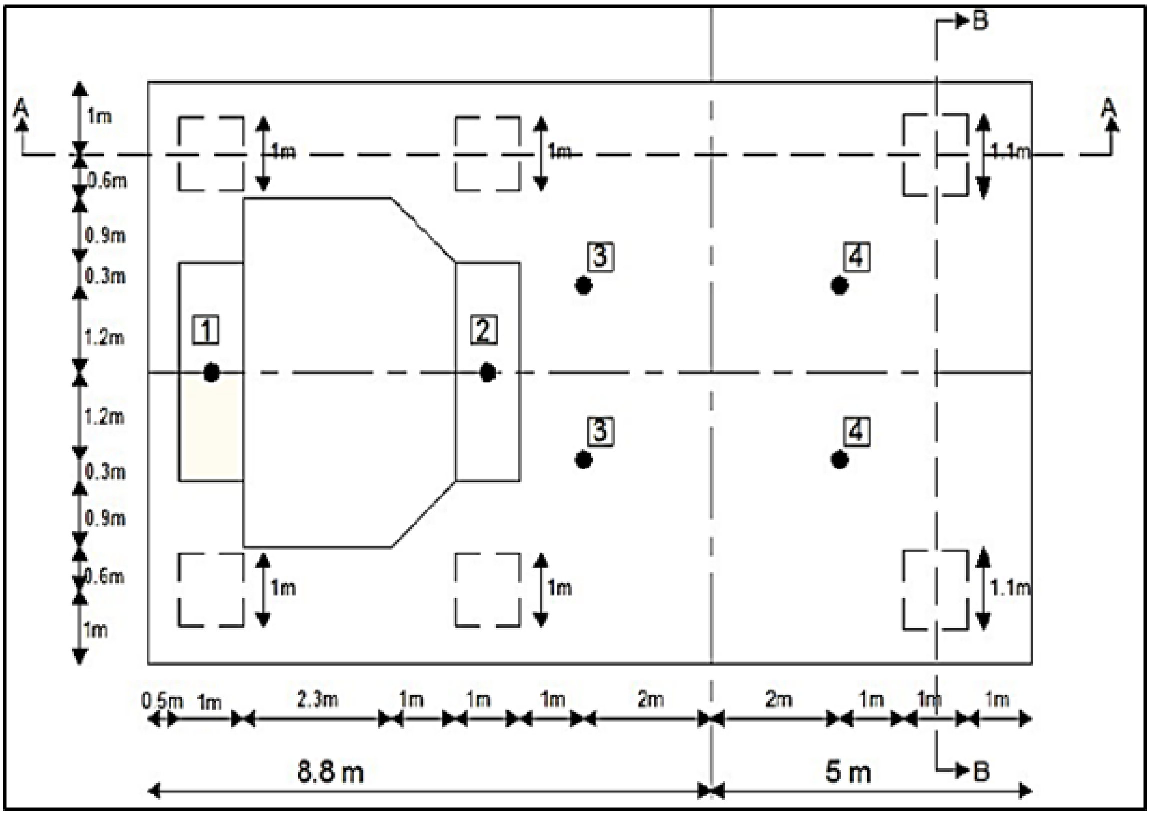


Figure 4: Bearing locations and table-top dimensions of frame foundation [2]

Table 1: Machine loads and unbalanced forces at top deck [2]

Bearing Points	1	2	3	4	Total
Total Machine Weight (kN)	400	360	200	200	1160
Rotor Weight (kN)	25	35	70	70	200
Unbalance Force					
Lateral/Vertical (kN)	5	7	15	15	42
Longitudinal (kN)	2	3	6	6	17
Blade Loss Force (kN)	3	11	-	-	14

2.2 Program Data

The simulation of the problem is started by constructing the frame foundation. The structure was initially constructed using Ansys Space claim with specified dimensions for the prototype model. The ANSYS 21 Workbench is used for the 3-D static and dynamic analysis of the selected problem. The frame foundation consists of eight bodies, the table-top, the raft, and six columns, after assigning the concrete material properties as shown in Table 2. The 3-D frame foundation structure, with six degrees of freedom "U_X, U_Y, U_Z, ROT_X, ROT_Y, and ROT_Z," three translational and three rotational degrees of freedom, is meshed using two types of elements, element SOLID186, for raft and columns bodies of the model, meshed with (500)mm element size to increase the accuracy of the output and to obtain a regular and uniform shape for all elements.

In contrast, due to the non-uniform shape, the table-top was meshed with SOLID187 tetrahedron element type, which is suitable for these irregular bodies. Figure 5 shows the elements used in the analyses. The final meshed model consists of 32175 nodes and 14715 elements distributed, as shown in Figure 6 for the prototype frame foundation. SOLID186 is a higher-order 3-D, 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node, translations in the nodal x, y, and z directions, as shown in Figure 5. SOLID187 element is a high-order 3-D, 10-node element. The element has a quadratic displacement behavior and is well suited to modeling irregular meshes. The element is defined by ten nodes with three degrees of freedom at each node: translations in the directions of the nodal x, y, and z. Figure 6 presents the finite element mesh for the frame foundation.

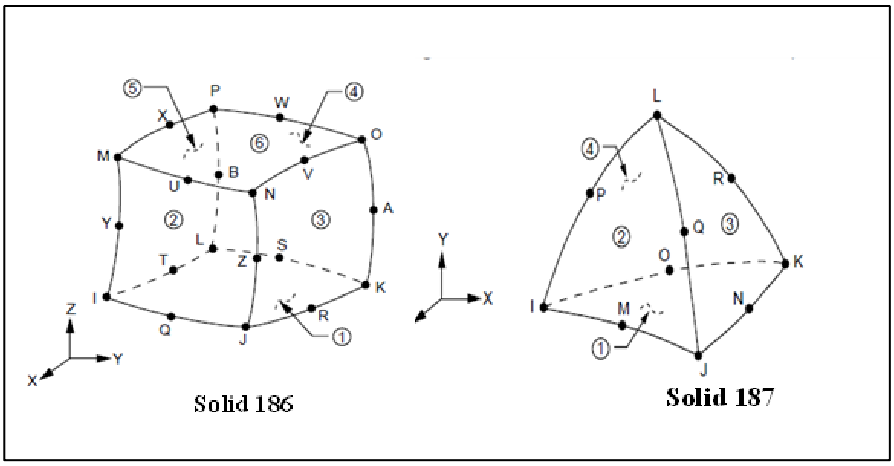


Figure 5: Solid 186 and solid 187 element types used in simulation

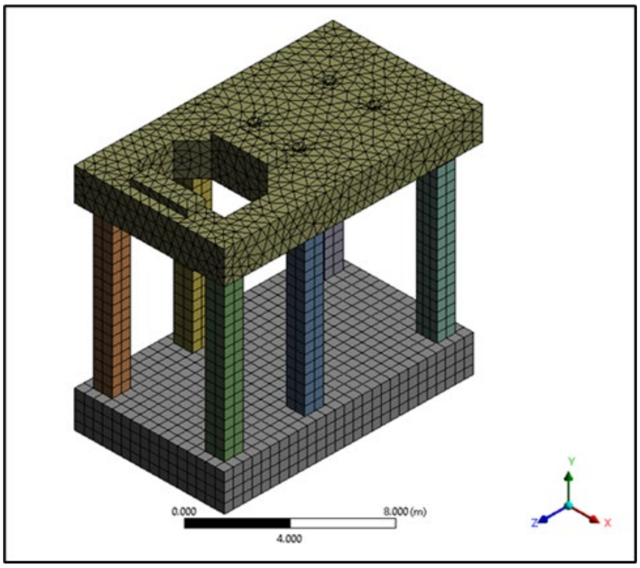


Figure 6: Finite elements meshing for concrete frame foundation

Table 2: Physical properties of concrete material [2]

Material Properties	Value
Mass Density of Concrete (kg/m^3)	2500
Young's Modulus E (kN/m^2)	30000000
Poisson's Ratio μ	0.15
Shear Modulus G (kN/m^2)	13000000

The turbomachine masses were applied according to Table 1 at specified bearing locations using the remote mass points technique in Ansys with rigid behavior for these mass points to simulate the behavior of the rigid machine in comparison with the less rigid concrete material of the frame foundation. It is worth mentioning that this new technique is more representing and less complex than the previous traditional modeling adopted by Bhatia and other authors for modeling the machines' masses by using rigid links to transfer machine mass to the frame foundation. The total machine masses were applied through these four mass points at the elevation of the four bearing locations, and their influence was transferred to the frame foundation through these bearing areas.

2.3 Static Analysis

The static analysis is performed on the structure under the model's specified loading and boundary conditions. The boundary condition was applied to the structure by assigning a fixed boundary to the raft's bottom area, constraining both displacement and rotations in three directions to zero ($Ux = U_Y = Uz = ROT_X = ROT_Y = ROT_Z = 0$). The static analysis results will then be used as a prestress analysis for the modal analysis.

2.4 Dynamic Analysis

A complete dynamic analysis of a machine foundation system is normally performed in two stages:

1. Frequency analysis: Calculate the natural frequencies and mode shapes of a machine-foundation system and compare them with the frequency of the applied harmonic force produced by the Turbo machine.
2. Forced response analysis: Calculating the machine foundation system amplitudes caused by the applied dynamic forces.

These two stages involve a solution of the equation of motion of spring mass damper theory:

$$M\{\ddot{y}\} + C\dot{y} + Ky = F(t) \quad (1)$$

Where: M is the mass, $\ddot{y}$ is the acceleration, C is the damping coefficient, $\dot{y}$ is velocity, K is stiffness, y is displacement, and F(t) is the time-varying external machine force.

Mass is calculated from the properties of the foundation and machine, damping is determined from the properties of the foundation, soil properties, and foundation configuration, stiffness is determined from the properties of the foundation, soil properties, and foundation configuration, and F(t) is the operating machine force.

In general, a six DOF model is needed to represent the dynamic performance of a rigid machine-foundation system properly. In this situation, a rigid foundation will contain three translation modes (along the vertical and two lateral directions) and three rotation modes (about the vertical and two lateral directions). Solving six coupled equations of motion is typically handled by computer software. For a flexible type of foundation like a frame foundation, there will be thousands of degrees of freedom, and a multi-degree of freedom (MDOF) model should be used for the general equation of motion as expressed in the following equation [13]:

$$[M]\{\ddot{y}\} + [C]\{\dot{y}\} + [K]\{y\} = \{F(t)\} \quad (2)$$

where:[ ] represents a square matrix, and { } represents a column matrix. The size of the matrixes is equal to the number of unknown degrees of freedom.

The natural frequencies of the foundation are calculated by modifying equation 2 for free vibration by suppressing the damping and the external force, i.e. C = F(t) = 0

$$[M]\{\ddot{y}\} + [K]\{y\} = 0 \quad (3)$$

Substituting for acceleration and amplitude proper functions and rearranging yield:

$$[K\text{-}M\omega_n^2]\ \{\phi\} = 0 \quad (4)$$

for non-trivial solution $\phi \neq 0$

ϕ is the relative amplitude or mode shape

$$[K\text{-}M\omega_n^2] = 0 \quad (5)$$

For a low degree of freedom, a direct solution can be manually obtained, but for a large degree of freedom, it becomes inefficient, and computer software is used to solve these equations for natural frequencies and mode shapes. The modal analysis is used to determine the vibration characteristics of a structure while it is being designed, i.e., finding natural frequencies and mode shapes of the structure. It can be a starting point for another dynamic analysis, such as a harmonic analysis.

Modal analysis in ANSYS is a linear analysis. Any nonlinearities, such as plasticity, are ignored even if they are defined. Modal analysis is a free vibration analysis. No load is applied during the analysis. The only "loads" that affect the solution are displacement constraints. In modal analysis, both Young's modulus and density are required. For performing a prestressed modal analysis, we need to prestress the structure by performing a static analysis before performing the modal analysis. For a prestress modal analysis, the boundary condition is the same as that used for the static analysis. No need to redefine it in modal analysis. When considering the structure's prestress, it is generally known that tension forces will increase the calculated natural frequencies of the structure. In contrast, compression forces will decrease the calculated natural frequencies of the structure.

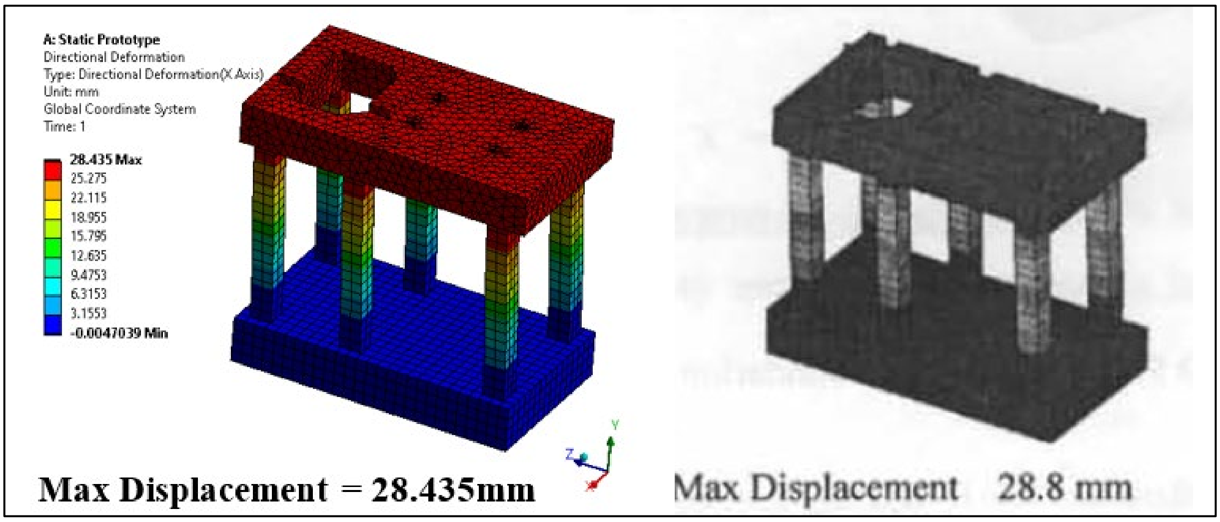
3. Results and Discussion

3.1 Mode Shapes

The static analysis is used as a prestress for the modal analysis and to check the model. After applying the boundary condition, the linear static solution is carried out. Results of applying a gravity force in the X, Y and Z direction are performed on the structure with machine masses applied at bearing elevation. Results are compared with those published by Bhatia [2], as shown in Figure 7 (a and b) below for the transverse direction 1g-X gravity force. It is observed that the frame foundation of Bhatia is somewhat different from the original one presented in Figures 3 and 4 as there are three openings, which may affect the dynamic results marginally, in addition to modifying and simplifying the original structure by neglecting the effect of bearing points volumes on the turbine side to minimize the complexity of the problem as stated by Bhatia. In contrast, the present work is done for the original structure without neglecting the bearing points volumes. A good agreement is achieved between the present work and Bhatia's work [2].

The general shape is similar for the two figures. It is observed that neglecting the machine mass will produce a lower value for a maximum lateral displacement of 24.839 mm. In contrast, the presence of machine mass, which is the standard case, will produce a maximum displacement of 28.435 mm, which agrees well with the results of Bhatia [2] of 28.8 mm. The gravity load is applied on the opposite side of the Y-Direction, and the static analysis results are used as a prestress for the dynamic analyses.

The first stage in any dynamic analysis is to perform modal analyses to determine the natural frequencies of the prototype concrete frame foundation. These frequencies are essential in the design process of these foundation types to compare them with the rotating machine operating frequencies to ensure that the machine and foundation natural frequencies do not match to avoid the system's resonance, thereby affecting the safety of the highly cost rotating machine. Modal analysis was performed for the model with a maximum number of modes to extract 600, which was reached with the range of frequencies between (0-1000) Hz. The modal analysis is performed for two cases: the first is carried out without considering the machine weight, and the second is carried out by adding the machine weight at bearing locations.



a. Present work b. Bhatia (2008) work

Figure 7: Transverse displacement for 1g-X direction force

The results calculated from the finite element modal analysis using ANSYS workbench 21 are presented in Table 3 and Figure 8 for the first 20 modes. The ANSYS 21 finite element results show a very good agreement with the results Bhatia [2] obtained for the two cases, indicating the accuracy of the modeling process in this study using ANSYS workbench software.

It is observed that the fourth mode of vibration, which corresponds to the first vertical mode of vibration, has two different values of (26.358 and 23.859) Hz in comparison with (26.48) Hz tabulated by Bhatia [2]. This suggests that the results of the frame foundation without considering the frame weight are more convergent, but since the standard codes and procedures for calculating the natural frequencies stated that the weight of the machine must be included during calculations and analysis and in addition to that Bhatia didn't state whatever the results tabulated including the machine weight or not, and in addition to that several authors calculate the first vertical frequency in the order of (23-24) Hz by using several programs like SAP2000 and other programs. It is clear that including the machine weight will decrease the natural frequencies since the natural frequency is reversely proportional to the mass. This may suggest that the results tabulated by Bhatia 2008 did not include the machine weight during analysis. Still, it is observed that the difference in values between the two calculated cases is marginal, with a maximum difference in value of (2.942) Hz for mode 5 of vibration, corresponding to an accuracy of 91.33%, which is acceptable. It is worth mentioning that the details of Bhatia's frame foundation details somewhat differ from the frame foundation mentioned by him in his book, which is adopted in this paper, and it seems that the original frame is subjected to many changes during design operation and this may lead to some accuracy issues. Figures (9–11) compare the first four vibration modes for the current study for the cases when the machine mass is excluded and included and Bhatia's work [2].

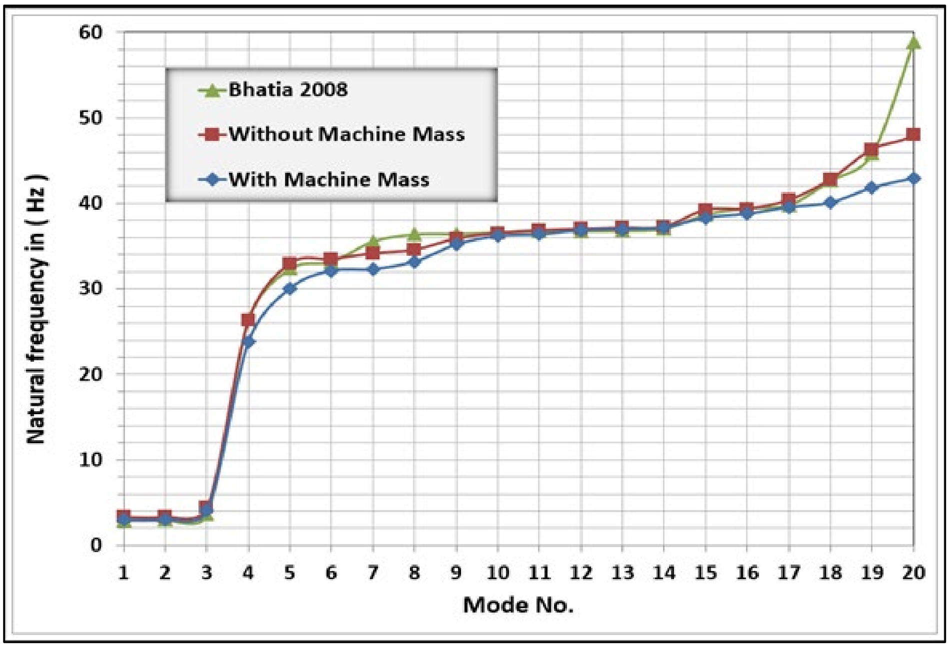


Figure 8: Natural frequency vs mode number for the reference frame foundation

Table 3: Modal analysis results for natural frequencies with and without machine weight

Machine weight	Mode No.	Freq. (Hz) Bhatia	Freq. (Hz) Ansys	Mode No.	Freq. (Hz) Bhatia	Freq. (Hz) Ansys
with	1	2.95	2.9755	11	36.75	36.362
without			3.320			36.846
with	2	3.02	3.0458	12	36.78	36.849
without			3.362			37.014
with	3	3.67	4.0814	13	36.8	36.974
without			4.468			37.161
with	4	26.48	23.859	14	37.05	37.19
without			26.358			37.275
with	5	32.36	30.057	15	38.7	38.299
without			32.999			39.238
with	6	33.2	32.113	16	39.43	38.764
without			33.5			39.367
with	7	35.57	32.304	17	39.83	39.518
without			34.42			40.385
with	8	36.4	33.188	18	42.67	40.067
without			34.583			42.796
with	9	36.45	35.184	19	45.81	41.799
without			35.884			46.32
with	10	36.62	36.169	20	58.88	42.918
without			36.537			48.031

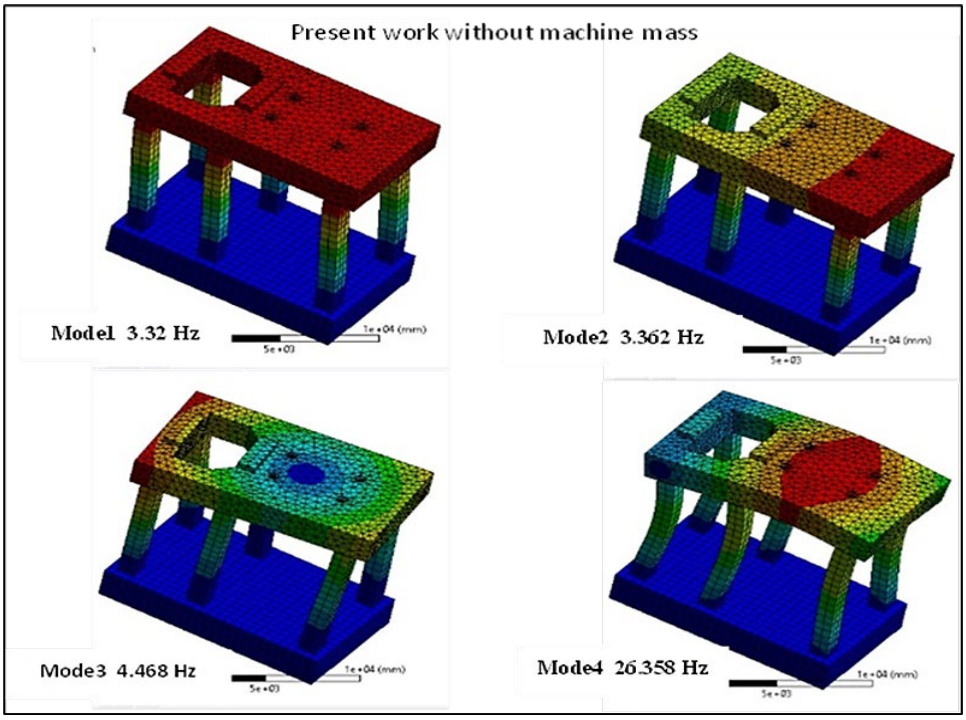


Figure 9: First four mode shapes of vibration for Ansys with machine mass excluded

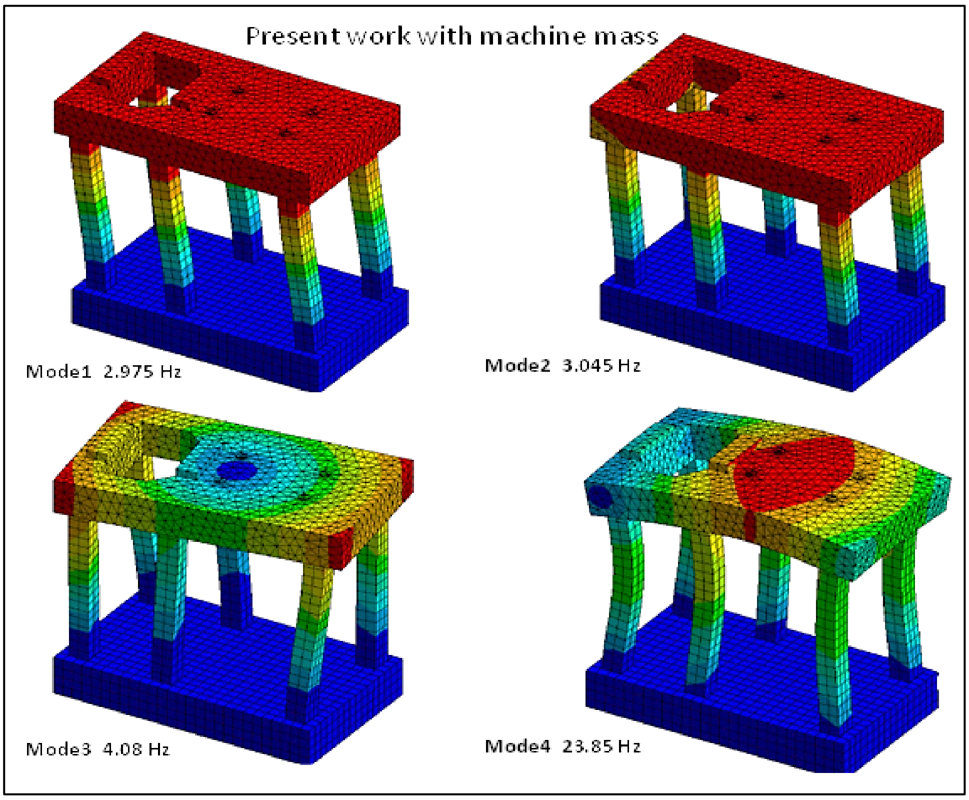


Figure 10: First four mode shapes of vibration for Ansys with machine mass included

By comparing the results, it is clear that the second mode shape of vibration when the machine mass is excluded differs from that published by Bhatia, even though the frequency value is similar in magnitude. Still, the mode shape is not similar. On the other hand, the shape of the second mode when machine mass is included is identical to Bhatia's results but with a lower frequency value of 23.859 Hz. This suggests that the results when machine mass was included are more accurate. It is worth mentioning that many authors tabulated a value of 23.958 Hz [14] and 24.77 Hz [15] for the vertical mode of vibration using SAP2000 software by applying machine mass by rigid links.

Comparing the results when machine mass is included with Bhatia's work, The first and second modes are the translational modes in the Y and X directions. The two modes are identical in shape and the frequency of each mode. The first mode frequency is (2.975) Hz versus (2.95) Hz for Bhatia. The second mode frequency is (3.045) Hz versus (3.02) Hz. The third mode is rotational vibration mode about the vertical y-axis, with a frequency of (4.08) Hz versus (3.67) Hz. The fourth mode is the vertical bending mode of vibration. It is obvious that the same trend is achieved, but the corresponding frequency for vertical is somewhat different, as mentioned earlier in this study.

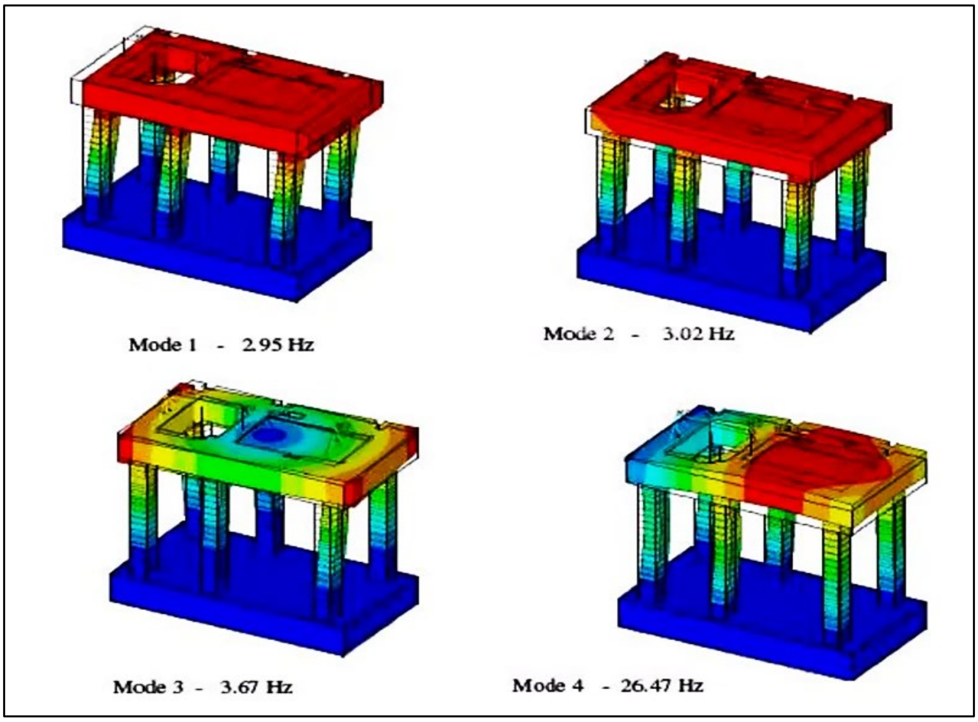


Figure 11: First four mode shapes of vibration [2]

3.2 Effect of Machine's Masses Elevation

Since the mass location is crucial in design criteria, and in many cases, the designers assume the machine masses are distributed on the tabletop slab surface, seeking the simplicity of the problem and neglecting the effect of masses elevation on the results. It is, therefore, necessary to study the effect of the machine's masses elevation on the natural frequencies during free vibration and the responses to harmonic loading of the machine's frame foundation.

The concrete frame foundation shown in Figure 6 is used for FEM analysis with the same boundary conditions. It is assumed that the elevation of the machine mass for the four bearing locations was fixed at the same elevation for each case. The lower raft surface is assumed as the datum, and the mass elevation of (12.6, 13.0, 13.3, 13.6, 13.9, and 14.2) m are selected for the dynamic analyses to clarify the mass effect on the response for the x, y, and z direction displacement. Figure 12 shows the free vibration analysis results for the selected elevations of masses at corresponding bearing locations. The effect seems similar and marginal for elevations up to 13.6 m, after which the natural frequency at each mode decreases with the increase of mass elevation, especially for modes 4 to 10.

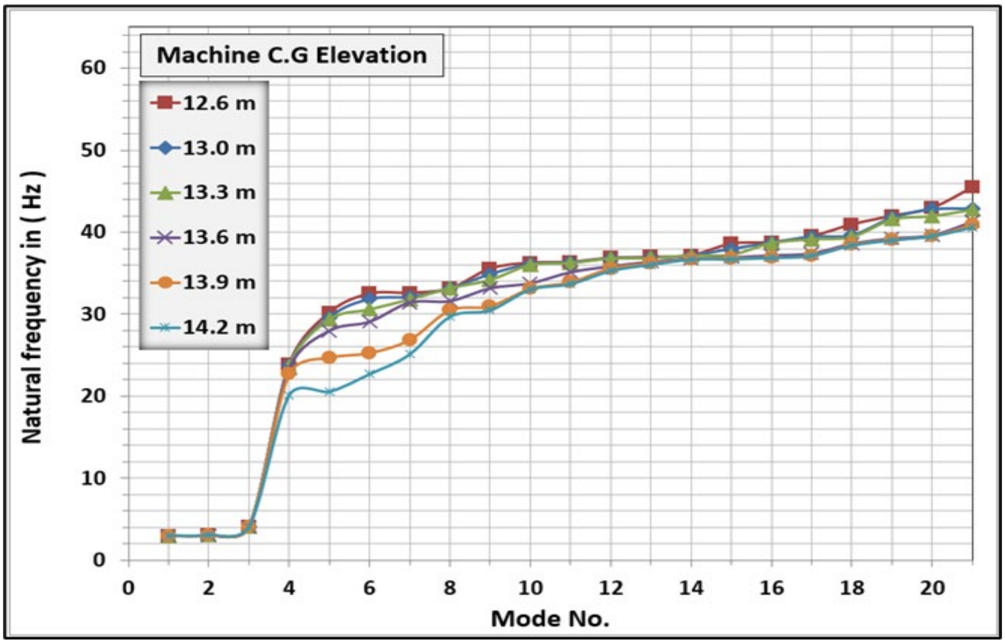


Figure 12: Natural frequency vs mode number for different machine's mass elevation

Figures (13–15) show the steady state harmonic loading displacement response results with different elevations of the machine's masses center of gravity calculated from the datum as indicated in Figure in the three principal directions, x, y, and z direction. Changing the elevation of the machine's center of gravity does not affect the lateral response of the frame foundation in the x and z direction. At the same time, the behavior is different for the vertical displacement response in the y direction. Up to an elevation of 13.3 m, the response is similar in shape and magnitude; this corresponds to a ratio of 5.55% representing the machine's center of gravity height above the tabletop surface to the elevation of the top slab surface. Still, with further increase in the machine's masses height, the response shape is changed to a wavy shape with double or multi peaks but with lower values of displacement response. The maximum response value is nearly fixed to 40 micrometers for elevations up to 13.3 m, with a single peak at 23.75 Hz frequency.

In contrast, the response for the 13.6 m is lower at the resonance frequency of 23.5 Hz, and noticing that another peak has started to appear at 28.2 Hz. The response for the 13.9m elevation shows a lower peak value of about 25 micrometers at a frequency of 22.7 Hz, after which the response has fluctuated with different peak values. The last elevation of 14.2 m shows a wavy shape, but this time, the first peak response is lower than the second peak, which is similar in maximum value to the 13.9 m elevation. They both gave a maximum value of about 25 micrometers but at different frequencies.

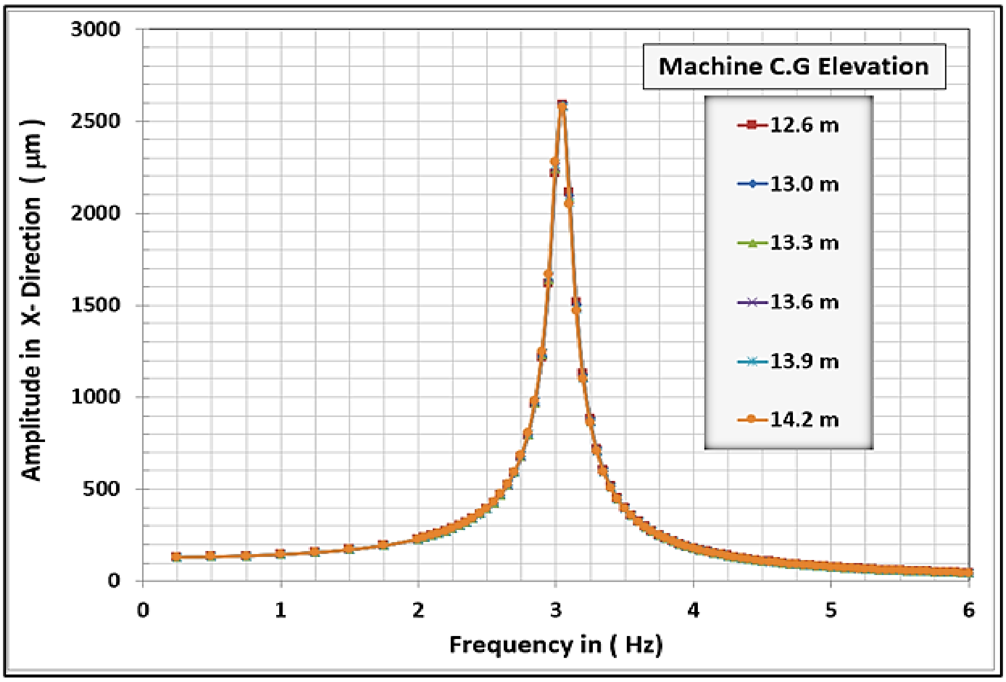


Figure 13: Amplitude vs frequency in X-direction for different machine's mass elevation

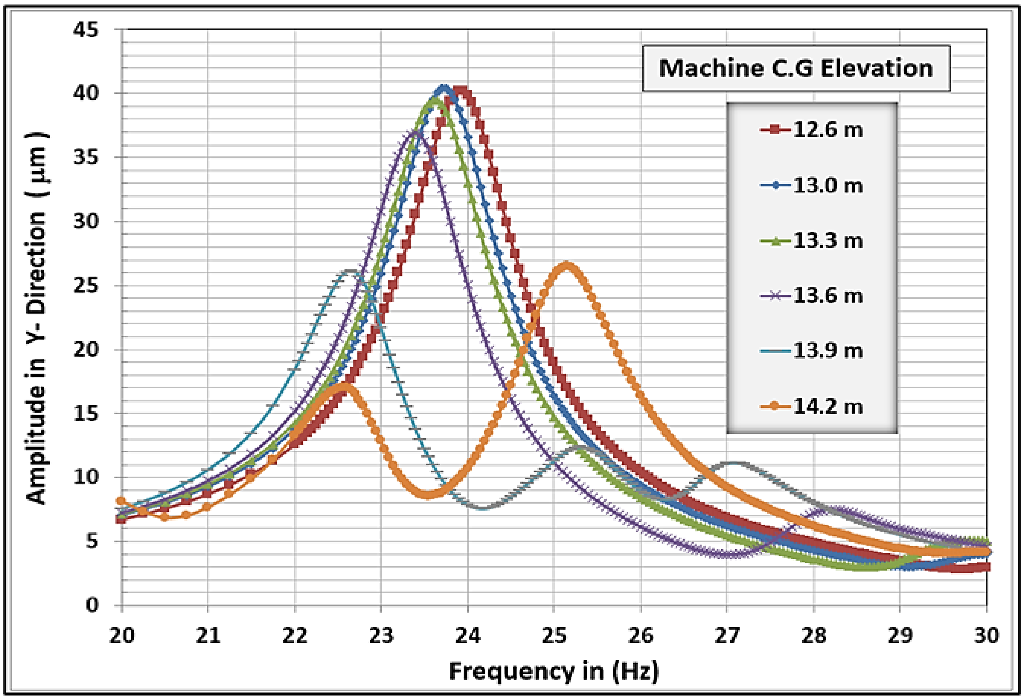


Figure 14: Amplitude vs frequency in Y-direction for different machine's mass elevation

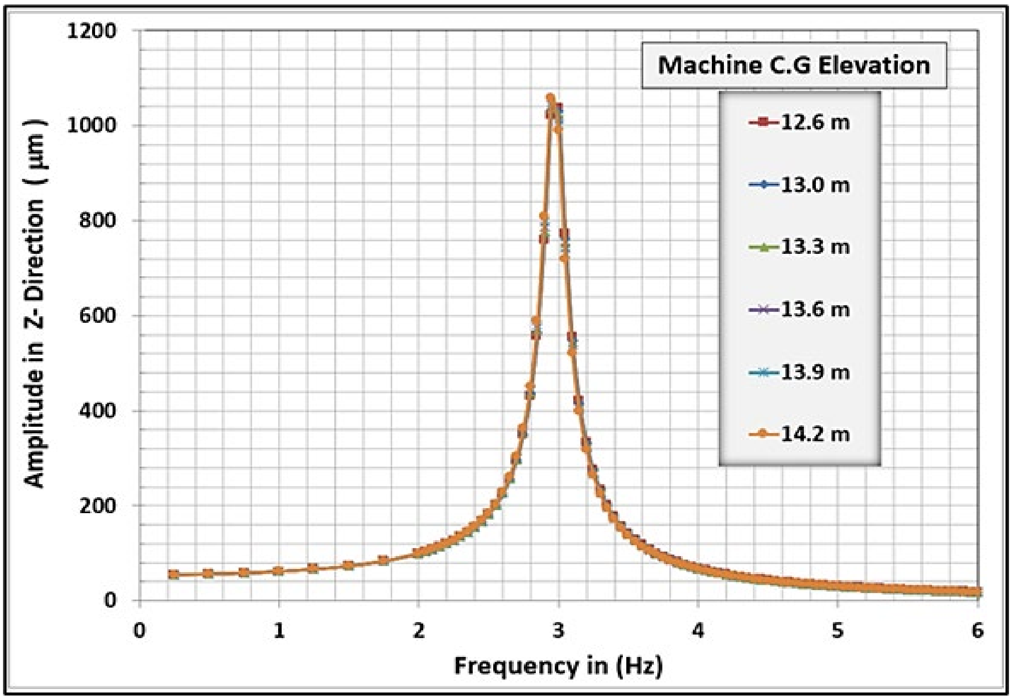


Figure 15: Amplitude vs frequency in Z-direction for different machine's mass elevation

4. Conclusion

From the results of finite element analysis for the specified frame foundation and material properties and boundary conditions, the following conclusions are restricted to these conditions:

1) Eliminating machine mass during modal analysis will reduce the value of the calculated natural frequencies.
2) When neglecting the machine mass, the difference between the calculated natural frequencies for any mode shape is less than 10%.
3) The mass of the machine must be included in the analysis to obtain more realistic behavior and frequency values.
4) During dynamic analysis of the frame foundation, the machine's masses must be applied at specified locations and elevations to reflect the true vertical response of the system unless the load / top slab elevation ratio exceeds 5%.
5) The machine's masses elevations do not affect the lateral response of frame foundation.

Author contributions

Conceptualization, A. Ahmed, M. Fattah and M. Mohsen; methodology, A. Ahmed, M. Fattah and M. Mohsen; formal analysis, A. Ahmed, M. Fattah and M. Mohsen; resources, A. Ahmed; data curation, A. Ahmed; writing—original draft preparation, A. Ahmed, M. Fattah and M. Mohsen; writing—review and editing, A. Ahmed, M. Fattah and M. Mohsen; supervision, M. Fattah and M. Mohsen. All authors have read and agreed to the published version of the manuscript.

Funding

This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Data availability statement

The data that support the findings of this study are available on request from the corresponding author.

Conflicts of interest

The authors declare that there is no conflict of interest.

References

[1] P. Jayarajan and K. M. Kouzer, Dynamic Analysis of Turbo-generator Machine Foundations, J. Civ. Eng. Environ. Technol., 1 (2014) 30-35.

[2] Bhatia, K. G., Foundations for Industrial Machines—A Handbook for Practising Engineers, First Edit. New Delhi: D-CAD Publishers, 2008.

[3] L. M. Moreschi and F. Farzam, Determination of local structural vibration properties for the design of machine foundations, Structures Congress 2005: Metropolis and Beyond, 2005, 1–10.https://doi.org/10.1061/40753(171)114

[4] K. G. Bhatia, Foundations for industrial machines and earthquake effects, ISET J. Earthq. Technol., 45 (2008) 1–2.

[5] M. Y. Fattah, M. J. Al-Mosawi, and A. F. I. Al-Ameri, Dynamic response of saturated soil-foundation system acted upon by vibration, J. Earthq. Eng., 21 (2017) 1158–1188.http://dx.doi.org/10.1080/13632469.2016.1210060

[6] M. Y. Fattah, M. J. Al-Mosawi, and A. F. I. Al-Americ, Stresses and pore water pressure induced by machine foundation on saturated sand, Ocean Eng., 146 (2017) 268–281.http://dx.doi.org/10.1016/j.oceaneng.2017.09.055

[7] A. L. Abbas, J. M. Abbas, and S. I. Jawad, Analysis of Gas Turbine Foundation under Harmonic Loading with and Without Consideration of Soil–Structure Interaction Effect, Int. J. Appl. Eng. Res., 12 (2017) 8063–8070.

[8] S. Tripathy and A. K. Desai, Analysis of seismically induced vibrations in turbo machinery foundation for different soil conditions: case study, J. Vibroeng., 19 (2017) 4356–4364.https://doi.org/10.21595/jve.2017.17436

[9] S. Bhattacharya, Analysis and Design of Table-top Foundation for Turbine Generators, Recent Advances in Structural Engineering, 1, 2019, 3–17.https://doi.org/10.1007/978-981-13-0362-3_1

[10] K. Rajkumar, R. Ayothiraman, and V. A. Matsagar, Effects of Soil-Structure Interaction on Torsionally Coupled Base Isolated Machine Foundation under Earthquake Load, Shock Vib., 2021 (2021) 1–18.https://doi.org/10.1155/2021/6686646

[11] A. S. Abdulrasool, M. Y. Fattah, and N. M. Salim, Displacements and stresses induced by vibrations of machine foundation on clay soil of different degrees of saturation, Case Stud. Constr. Mater., 17 (2022) e01327.http://dx.doi.org/10.1016/j.cscm.2022.e01327

[12] C. J. Roy and W. L. Oberkampf, A comprehensive framework for verification, validation, and uncertainty quantification in scientific computing, Comput. Methods Appl. Mech. Eng., 200 (2011) 2131–2144.https://doi.org/10.1016/j.cma.2011.03.016

[13] ACI 351.3 R-18:, Report on Foundations for Dynamic Equipment, [Online]. Available: www.concrete.com.

[14] A. Mohamed, Y. Alzeni, and A. Ghalab, Modelling of Turbo Generator Foundation using FEM to Study the Isolation Effect on the System, Int. Res. J. Innov. Eng. Technol., 3 (2019) 31.

[15] S. Tripathy and A. K. Desai, Seismic Analysis of Turbo machinery frame foundation using Barrettes, S. V. National Institute of Technology, 2017.