

Analysis of Pile Behavior due to Damped Vibration by Finite Element Method (FEM)

Rasel M. Chowdhury*, Faqrul, A.K., Riyad, A.S.M. and Momeen-Ul-Islam

Abstract— Pile foundation is analyzed for damped vibration by finite element method. Laboratory experiments for the similar study were not held previously because, it is not practically possible to penetrate a vibrator in the large depth; collection of soil sample in the laboratory largely disobey the need for undisturbed soil; preparation of models cannot completely describe the infinite soil boundary conditions; and the presence of faults in the large depths has varying hydrostatic and air pressure, and also have the temperature variations, leaving huge difficulties in model construction. Different physical or mechanical properties that may have significant consequences are analyzed to obtain the displacement at the top most point of the pile. Graphical representations of those variables are prepared to take further steps to make efficient design for safer structures. To carry out the analysis, a 3D pile soil model is built using comprehensive computer software ANSYS 10 © 2005 SAS IP, Inc package. The model is firstly verified with previous studies for horizontal loading, and further modal and spectrum analysis are done for different soil conditions or other variables to obtain graphical relationships of the pile top displacement. Many variables such as ratio of modulus of elasticity's of pile to soil, equivalent spring constant of soil, vertical loads, frequencies of vibration are studied to obtain the displacement of pile top. It is found that, Displacement at the top of the pile is increased up to 10% for the increment of the frequency of 5 Hz, where other parameters remain the same. 50% increase of ratio modulus of elasticity's of pile to soil will result in 70 to 80% decrement of deflection at pile top and, concentric vertical loads reduce the pile top displacement due to vibration up to 20 to 60% when the pile is designed for service load equal to 80% of the pile capacity.

Index Terms— Pile foundation, damped vibration, Finite Element, Continuum, Modeling, Meshing, Spectrum, ANSYS.

1 INTRODUCTION

Finite element analysis is a method for numerical solution of a differential equation. Most engineering problems are expressed in terms of certain governing equation and boundary conditions. The basic idea behind the finite element method is to replace a continuous function by means of piecewise polynomials. Such an approximation is called piecewise polynomial approximation. In finite element method, the domain of integration is subdivided into a number of smaller pieces or regions- this piece is called element and over each of these elements the continuous function is approximated by a suitable piecewise polynomial [1].

Continuum: It is the region over which the governing equation prevails. In finite element method the continuum is subdivided into a finite number of sub-regions often called finite element mesh. This sub-region is called element, and over each element the variation formulation of the given differential equations constructed using simple interpolation function for approximation. The finer mesh the more approximation of

the result to exact solution.

Discretization or Meshing: The process of division of the continuum into a finite number of elements is called discretization or meshing.

Element: The continuous function is divided into finite piece called elements.

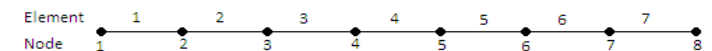
Node: The end point of each element is called nodes.

Interpolation Functions: For each element the continuous function is approximated by discrete model which is composed of interpolation polynomials.

Shape Function: It is the coefficient of interpolation polynomial. The shape function is usually denoted by N. The shape function is written for each node of the element and has a magnitude 1 at the node and 0 for all other nodes in that element.

Stiffness Matrix: The assemblage of shape function for all the finite elements in matrix form is called global stiffness matrix.

Connectivity: First each element and its nodes is recognized by local identification and then they are connected globally by using connectivity array.



So the i^{th} element has node i and $i+1$.



Damping: Damping is evaluated for each mode and is defined as:

$$\xi'_i = \frac{\beta \omega_i}{2} + \xi_c + \frac{\sum_{j=1}^{N_m} \beta_j^m E_j^s}{\sum_{j=1}^{N_m} E_j^s} + \xi_i^m$$

- Rasel M. Chowdhury, Post-graduate Student; Dept. of Civil Engineering, Khulna University of Engineering & Twchnology, Khulna-9203, Bangladesh, PH-+8801713173437. E-mail: civilizedrasel073@yahoo.com
- Faqrul, A.K., Undergraduate Student; Dept. of Civil Engineering, Khulna University of Engineering & Twchnology, Khulna-9203, Bangladesh, PH-+8801683292501. E-mail: kuet.sajib25@gmail.com
- Riyad, A.S.M., Undergraduate Student; Dept. of Civil Engineering, Khulna University of Engineering & Twchnology, Khulna-9203, Bangladesh, PH-+8801825281736. E-mail: riyadtoahid@yahoo.com
- Momeen-Ul-Islam, Undergraduate Student; Dept. of Civil Engineering, Khulna University of Engineering & Twchnology, Khulna-9203, Bangladesh, PH-+8801678641195. E-mail: momeen.civil2k8@gmail.com

Where,

- β = beta damping
 - ω_i = undamped natural circular frequency of the i th mode
 - ξ_c = damping ratio
 - N_m = number of materials
 - $\{\phi_i\}$ = displacement vector for mode i
 - $[K_j]$ = stiffness matrix of part of structure of material j
 - ξ_i^m = modal damping ratio of mode i
 - β_i^m = damping constant stiffness matrix multiplier for material j
- $$E_j^S = \frac{1}{2} \{\phi_i\}^T [K_j] \{\phi_i\} = \text{strain energy}$$

Note that the material dependent damping contribution is computed in the modal expansion phase, so that this damping contribution must be included there.

Pile Foundation: Piles are structural members of timber, concrete, and/or steel those are used to transmit surface loads to lower levels in the soil mass. This transfer may be by vertical loads to lower levels in the soil mass. This transfer may be by vertical distribution of the load along the pile shaft or a direct application of load to a lower stratum through the pile point. A vertical distribution of the load is made using a friction pile and a direct load application is by a point, or end-bearing, pile [2].

Pile Subjected to Lateral Loads: Any structure is inevitably subjected to a certain amount of horizontal force due to wind earth pressure, earthquake, etc., or traction force from automobiles or trains. It is unsafe to assume that frictional resists between the bottom of the pile cap and the soil because in this type of foundation the vertical load is transmitted through the piles to the lower stratum, not to the soil immediately below the pile cap. In extreme cases, the soil may even settle away from the pile caps and leaving a small space in between. Unless the structure is supported laterally by other means, the pile should be designed to resist such lateral loads [3]

2 ELEMENT MODELING AND ANALYTICAL STUDY

2.1 ANSYS Finite Element Modeling

To create the finite element model in ANSYS there are multiple tasks that have to be completed for the model to run properly. Models can be created using command prompt line input or graphical user interface (GUI). For the simulation of the model, the GUI method was utilized to create the model. The section describes the different tasks and entries into used to create the FE calibration model.

2.2 Element Types

The element types for this model are shown in Table 1.

TABLE 1
 ELEMENTS USED IN THIS STUDY

Pile	SOLID45
Soil	SOLID45
Spring Damper	COMBIN14

2.2.1 SOLID45

Two different SOLID45 elements were used for pile and soil. SOLID45 element is an eight nodes hexahedral element with three translational degrees of freedoms at each node- translations in the x, y, and z directions. SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. A reduced integration option with hourglass control is available. The geometry, node locations, and the coordinate system for this element are shown in Fig. 1. The element is defined by eight nodes and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. The element coordinate system orientation is as described in Coordinate Systems.

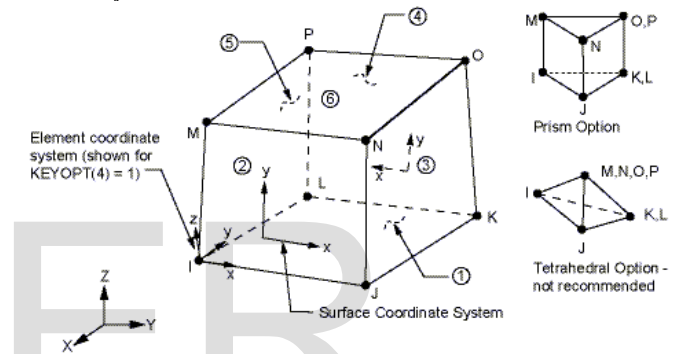


Fig. 1. SOLID45 geometry

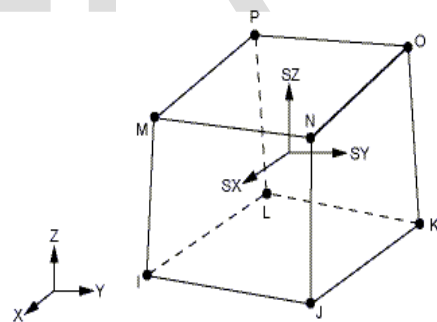


Fig. 2. SOLID45 stress output

Element loads are described in Node and Element Loads. Pressures may be input as surface loads on the element faces as shown by the circled numbers on Fig. 2. Positive pressures act into the element. Temperatures and fluencies may be input as element body loads at the nodes.

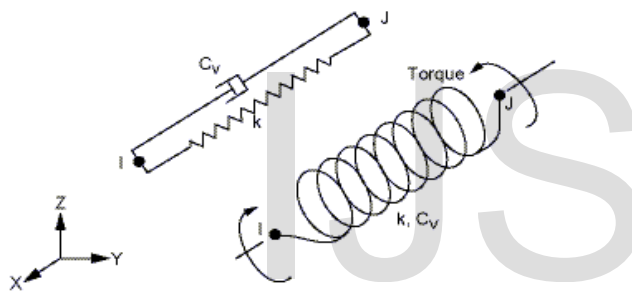
SOLID45 Assumptions and Restrictions

- Zero volume elements are not allowed.
- Elements may be numbered either as shown in Fig. 3. "SOLID45 Geometry" or may have the planes IJKL and MNOP interchanged.

- The element may not be twisted such that the element has two separate volumes. This occurs most frequently when the elements are not numbered properly.
- All elements must have eight nodes.
 - ✓ A prism-shaped element may be formed by defining duplicate K and L and duplicate O and P node numbers (see Triangle, Prism and Tetrahedral Elements).
 - ✓ A tetrahedron shape is also available. The extra shapes are automatically deleted for tetrahedron elements.

2.2.2 COMBIN14

It may be called as a spring damper. COMBIN14 as shown in Fig. 4 has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered. The torsional spring-damper option is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y, and z axes. No bending or axial loads are considered.



2-D elements must lie in a z = constant plane

Fig. 3. COMBIN14 geometry

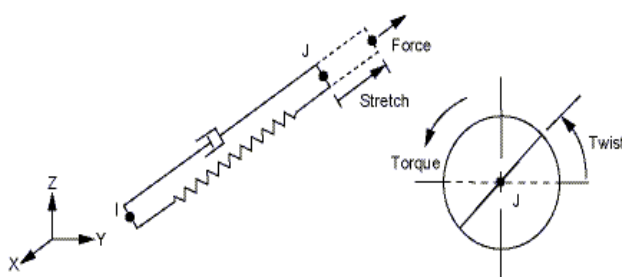


Fig. 4. COMBIN14 stress output

The spring-damper element has no mass. Masses can be added by using the appropriate mass element. The spring or the damping capability may be removed from the element. The element is defined by two nodes, a spring constant (k) and damping coefficients $(c_v)_1$ and $(c_v)_2$. The damping capability is not used for static or undamped modal analyses. The longitudinal spring constant should have units of Force/Length, the damping coefficient units are Force*Time/Length. The torsional spring constant and damping coefficient have units of Force*Length/Radian and Force*Length*Time/Radian, respec-

tively. For a 2-D axisymmetric analysis, these values should be on a full 360° basis.

COMBIN14 Assumptions and Restrictions:

- The length of the spring-damper element must not be zero, i.e., nodes I and J should not be coincident, since the node locations determine the spring orientation.
- The longitudinal spring element stiffness acts only along its length. The torsion spring element stiffness acts only about its length, as in a torsion bar.
- The element allows only a uniform stress in the spring.
- In a thermal analysis, the temperature or pressure degree of freedom acts in a manner analogous to the displacement.
- Some option supports stress stiffening or large deflection. Also, there is also some options which is used with large deflection, the coordinates will not be updated.
- The spring or the damping capability may be deleted from the element by setting K or C_v equal to zero, respectively.
- If C_{v2} is not zero, the element is nonlinear and requires an iterative solution

2.3 Real Constants

There is no real constant set for Solid45 element. The real constant set for COMBI14 requires two values.

- **Spring constant:** The spring constant is the study variable of this study. So the values of different equivalent spring constant for soil are given to obtain desired output. It ranges from 1500 kN/m to 2000 kN/m.
- **Damping coefficient:** Damping coefficient may be defined as the amount of force exerted by a damping medium per unit velocity of the simple harmonic damping system. There is a large range of damping coefficient of soil. Damping coefficient of soil depends on the soil properties.

2.4 Material Properties

As stated earlier, pile and soil are both defined with SOLID45 element in the model. The properties of SOLID45 elements are listed below:

- Liner isotropic properties:
 - ✓ EX= modulus of elasticity
 - ✓ PRXY= Poisson's ratio [for soil= 0.35, for pile= 0.2]
- Moduli of elasticity's of pile and soil are the parts of design parameters
Density:
- ✓ Mass densities of the pile= 2450 kg/m³
 - ✓ Mass density of soil= 1280 kg/m³

2.5 Modeling

Modeling is the most important part in the simulation through GUI method. The features of the pile-soil finite model are shown in the Fig. 5 below:

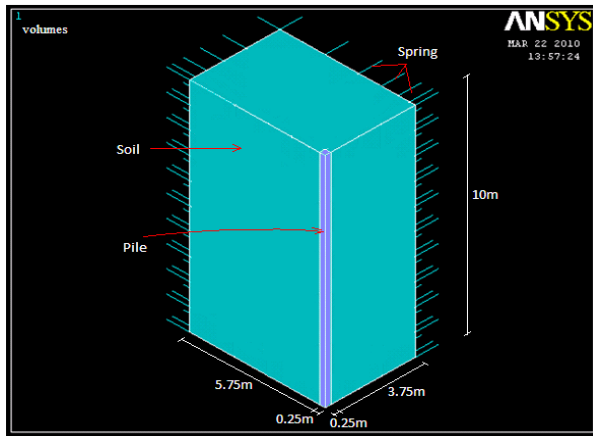


Fig. 5. Pile-soil model geometry

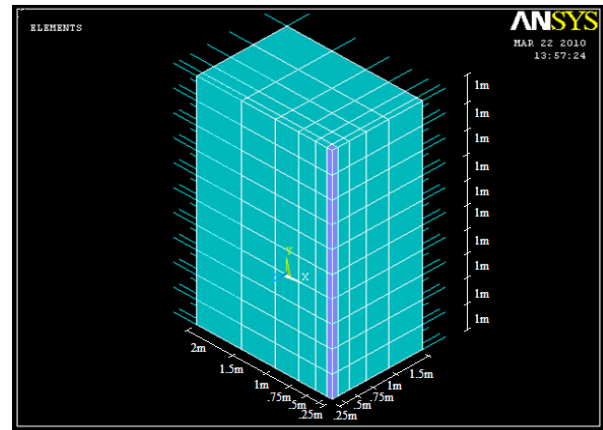


Fig. 1. Mesh configuration for the model

For the use of symmetric conditions, only one fourth of the model is prepared in this case. The volume is dimensioned for 6m, 4m and 10m at x, z & y directions respectively. The volume is symmetric about xy and zy planes. To make the l/d ratio of the pile, the x & z dimensions of the pile is given as 0.25m. Remaining of the volume is for soil element. Soil volume is little bit complex so it may be prepared by 3 different volumes as given in Table 2 below.

TABLE 1
 COORDINATES FOR PILE AND SOIL VOLUME

Portions	Volume number	x-ordinate		y-ordinate		z-ordinate	
		0	5.75	0	10	0	3.75
soil	1	0	5.75	0	10	0	3.75
	2	5.75	6	0	10	0	3.75
	3	0	5.75	0	10	3.75	4
pile	4	5.75	6	0	10	3.75	4

Then the next step is to provide spring elements at the two faces of the volume. Spring element, COMBIN14 provides a certain advantages because of its spring constant as a real constant. The unit of spring constant is force per unit length so, the length of the provided spring doesn't create any effect on this analysis. Offset nodes are made at the two boundary faces are provided, and a line is created between the boundary node & its offset node. This line is further defined as the COMBIN14 element.

2.6 Meshing

Meshing is the important part of this model. Size control of the mesh element is also necessary because, huge numbers of elements are prepared with many nodes, the matrix required to solve the nodal solution will be large, and its solution may fail for some circumstances.

Variable mesh sizes of the volume are required for this purpose. The results desired for this study depends on the pile behavior, the accuracy required may be obtained by providing smaller mesh at the vicinity of the pile as shown in the Fig. 6.

2.7 Numbering Control

The command merge items merge separate entities that have the same location. These items will then be merged into single entities. Caution must be taken when merging entities in a model that has already been meshed because the order in which merging occurs is significant. Merging key points before nodes can result in some of nodes become "orphaned", that is, the nodes lose their association with the solid model. The orphaned nodes can cause certain operations (such as boundary condition transfers, surface load transfers, and so on) to fail. Care must be taken to always merge in the order that the entities appeared. All precautions were taken to ensure that everything was merged in the proper order. Also, the lowest number was retained during merging.

2.8 Boundary Conditions

The pile is assumed to be bearing on the bed rock, all the bearing nodes are taken as fixed, i.e., all nodes along the base in fixed in all three directions. For this purpose, the plane at the bottom of the model is given three degrees of freedom, namely UX, UY & UZ. Values of these DOFs are given as constant value, and the constant value is 0.

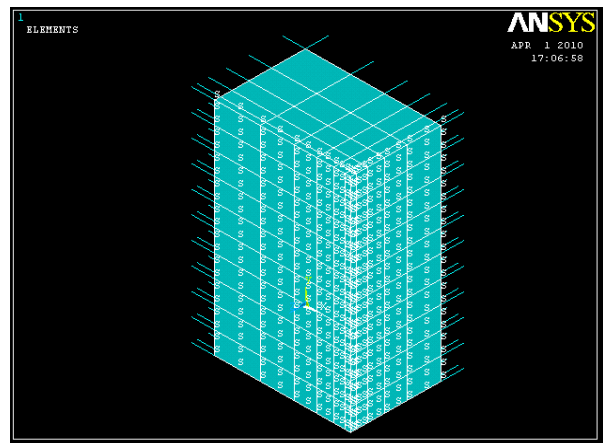


Fig. 7. (a) Provided boundary conditions (fixed at bearing stratum) at the bottom of the model

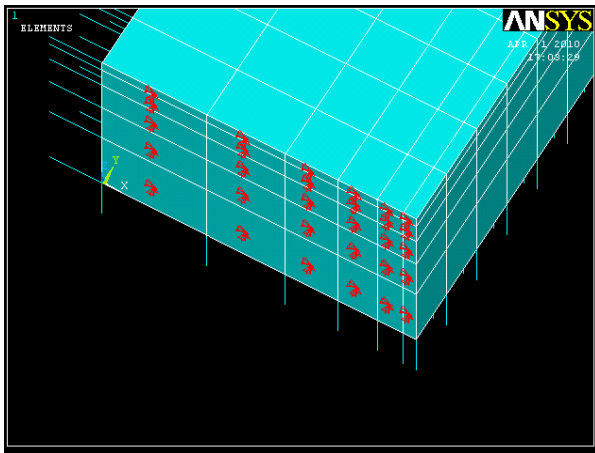


Fig. 7. (b) Symmetry boundary conditions at the two inner faces (denoted by the letters S)

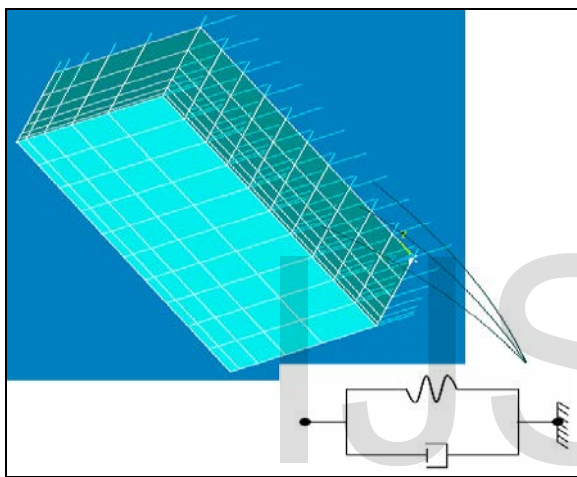


Fig. 7. (c) Equivalent soil boundary condition

The symmetry boundary conditions are then set. The model being used is symmetric about two planes. The symmetric boundary conditions are applied at the planes xz and yz , where the planes cut the pile section to make a quarter models. Providing symmetry boundary conditions enables one to reduce the node numbers, element numbers, help in forming of simple element matrices, & make the solution swift. Soil boundary conditions are then provided. No initial displacement is provided because the spring elements transfer stiffness of the soil elements and hence the infinite soil boundary region is described in this way shown in Fig. 7.

2.9 Model Verification

The model has been verified with reported literature for static loading (Fig. 8). The verification has been discussed in the following sections.

Static loading: The model is verified by loading in the x direction at top and the horizontal deflection of the pile head. Horizontal deflections at the pile head are computed for different amplitudes of applied load for the elastic case.

The results are compared with those presented in Bently and El Naggar (2000), Poulos and Davis (1980), Maheshwari et al. (2005) and Rajib Sarkar (2008). The comparison is shown in Fig. 9 below.

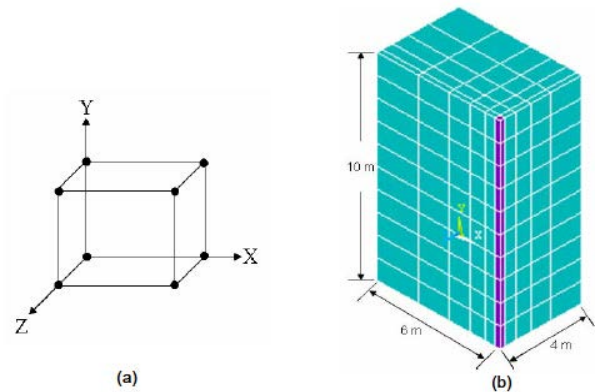


Fig. 8. Three dimensional finite element models used by Rajib Sarkar: (a) Eight node block element for soil and pile, (b) Finite element quarter model.

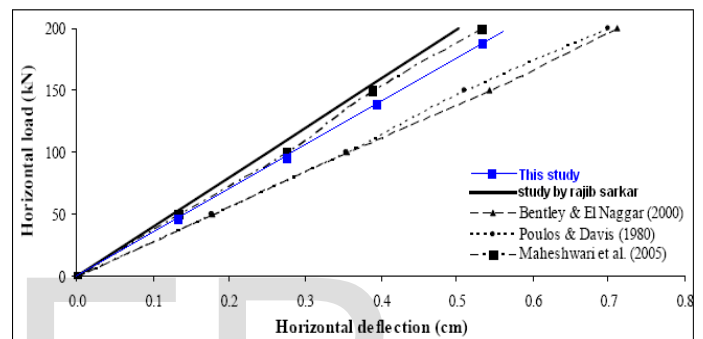


Fig. 9. Response of a single pile fixed at base for static loading

2.10 Analysis Type: Static Analysis

The finite element model for this analysis is a pile model under horizontal loading. For the purpose of this model, the static analysis type is utilized. The restart command is utilized to restart an analysis after the initial run or load step has been completed. The use of restart option will be detailed in the analysis portion for the discussion.

The Sol'n Controls command dictates the use of a linear or non linear solution for the finite element model. Typical commands utilized in a non linear static analysis are shown in Table 3 below.

TABLE 2
 COMMANDS USED TO CONTROL NONLINEAR ANALYSIS

Analysis Options	Small Displacement
Calculate prestress effects	No
Time at end of load step	8400
Automatic time stepping	On
Number of substeps	100
Max no. of substeps	200
Min no. of substeps	1
Write Items to results file	All solution items
Frequency	Write every substep

In the particular case considered in this portion of this study, the analysis is small displacement and static. The time at the end of load step refers to the ending load per load step. Table

below shows the first load step taken. The substeps are set to indicate load increments used for this analysis. The commands used to control the solver and outputs are shown in another Table 4.

**TABLE 3
COMMANDS USED TO CONTROL OUTPUT**

Equation solver	Sparse direct
Number of restart files	1
Frequency	Write every substep

All these values are set to ANSYS (SAS 2005) defaults. The commands used for the nonlinear algorithm and convergence criteria are shown in table below. All values for the nonlinear algorithm are set to defaults shown in Table 5.

**TABLE 4
NONLINEAR ALGORITHM AND CONVERGENCE CRITERIA
PARAMETER**

Line search	Off	
DOF solution predictor	Prog chosen	
Maximum number of iteration	100	
Cutback control	Cutback according to predicted number of iteration	
Equivalent plastic strain	0.15	
Explicit creep ratio	0.1	
Implicit creep ratio	0	
Incremental displacement	10000000	
Points per cycle	13	
Set convergence criteria		
Label	F	U
Ref. value	Calculated	Calculated
Tolerance	0.005	.05
Norm	L2	L2
Min. ref.	Not applicable	Not applicable

The values for the convergence criteria are set to defaults except for the tolerances. The tolerances for force and displacement are set as 5 times the default values. Table below shows the commands used for the advanced nonlinear settings. The program behavior upon non-convergence for this analysis was set such that the program will but not exit. The rest of the commands were set to defaults shown in Table 6.

**TABLE 5
ADVANCED NONLINEAR CONTROL SETTINGS USED**

Program behavior upon non convergence	Terminate but do not exit
Nodal DOF sol'n	0
Cumulative itr.	0
Elapsed time	0
CPU time	0

3 ANALYSIS PROCESS FOR THE FINITE ELEMENT MODEL

The FE analysis of the model was set up to examine three different behaviors: the displacement at the pile head, multidirectional strain components and multidirectional stress components. The Newton-Raphson method of analysis was used to compute the nonlinear response.

The applications of the loads are done incrementally as required by Newton-Raphson procedure. After each load increment was applied, the restart option was used to go to the next step after convergence.

3.1 Analysis Type

After the verification of the model, it is ready for the seismic vibration analysis. Form the command of analysis type, we need to accomplish spectrum analysis, but for the purpose of modal calculation, direction of excitations etc. modal analysis is chosen first.

3.1.1 Modal Analysis

Modal analysis is the prerequisite for the spectrum analysis. Spectrum analysis requires the mode combination and the element matrices, so modal analysis is required shown in Table 7.

**TABLE 6
MODAL ANALYSIS OPTION PROCEDURES**

Mode Extraction Method	Reduced
No. of mode to extract	0
Expand mode shape	Yes
No. of mode to expand	1
Calculate element results?	Yes
Use lumped mass approx?	No
Incl prestress effect?	No
Memory save	No
Options for reduced modal analysis	
Frequency range	Select the default
No. of modes to print	3

Master degree of freedom: The master degree of freedom (master DOF's) are to be set. A user defined master degree of freedom to be selected. As, all of the elements will be subjected for seismic excitations, all of the nodes are taken for 1st degree

of freedom. Another option, named additional degree of freedom, may be selected when a degree of freedom is not selected as the master degree of freedom.

The model is then solved for modal analysis (Fig. 10).

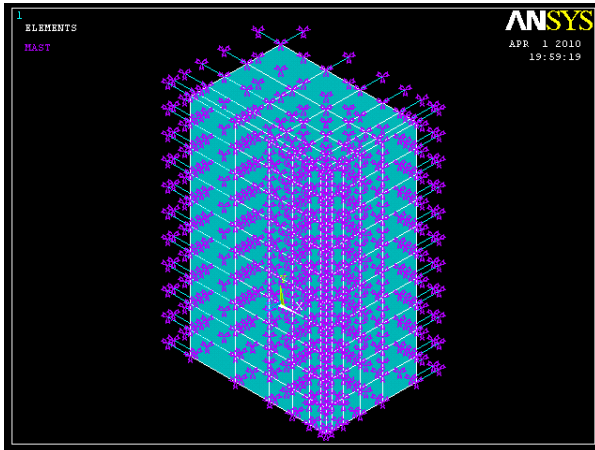


Fig. 10. Model with master degree of freedom in all directions.

3.1.1.1 Spectrum Analysis 1

After the modal analysis, the model is to be solved for certain predominant frequency. New analysis is to be chosen as spectrum analysis. In the load step option, one can give the excitations from a single point. The settings required are given in the following Table 8.

TABLE 7
 SETTINGS FOR SINGLE POINT RESPONSE SPECTRA

Type of response spectr	Seismic displac
Scale factor	1
Excitation direction SEDX, SEDY, SEDZ	1, 1, 1 respectively
Other options should remain as default	

Different frequencies are given in the frequency table. The excitation frequency is another variable of this study. So, frequencies of different magnitude are to be put at each analysis. The frequency table can accept 20 frequency values with different magnitude. The conditions for the frequency input are: the input frequency value must not be equal to zero; and the values of frequencies must be of ascending order. If a small frequency value is put after a greater value, the frequency table will end before the smaller value.

Value of damping ratio is to be put. As the model prepared is a damping system itself, the damping of the frequency table may be neglected at this stage. So, 0(zero) is to be put for damping ratio. The spectrum values for the different frequencies are 0.44.

The system is then solved for spectrum analysis for the first time.

3.1.1.2 Spectrum Analysis 2

Another spectrum analysis is required for mode combination. The modes are to be combined and hence, a new spectrum

analysis is required. Mode combination methods for the single point response spectra are given in the Table 9 below.

TABLE 8
 METHOD OF MODE COMBINATION

Mode combination method	SRSS
Significant threshold	0.15
Type of output	Displacement

Note: **SRSS method:** Specifies the square root of sum of squares mode combination method. The system is then solved for spectrum analysis for the 2nd time.

3.2 Postprocessing

ANSYS 10.0 gives many facilities to find out the desired results at any point of interest in the model. Corresponding sub-step of the result from result-list is to be read by pick. Both graphical and listing result system is a unique feature for the feature. Graphical results representation system is of two types: contour plot & vector plot. Contour plot gives certain advantages to understand and utilize results with different color stripes having different value ranges for each color. Vector plot gives arrow diagram with the direction of changes of values with different color.

3.3 Graphical Reprerentation of the Results

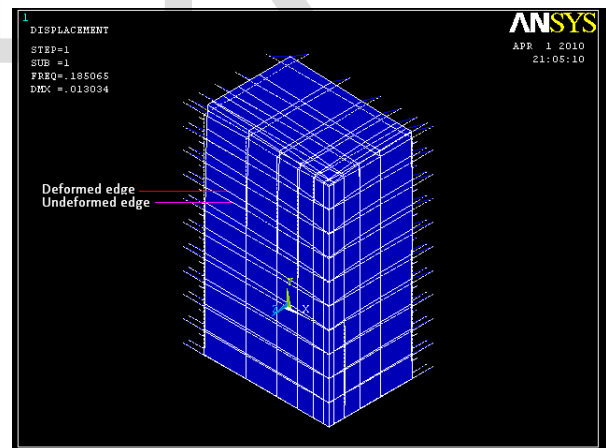


Fig. 11. Maximum deformed shape of the model

3.4 Nodal Solution

3.4.1 Degree of freedom solution

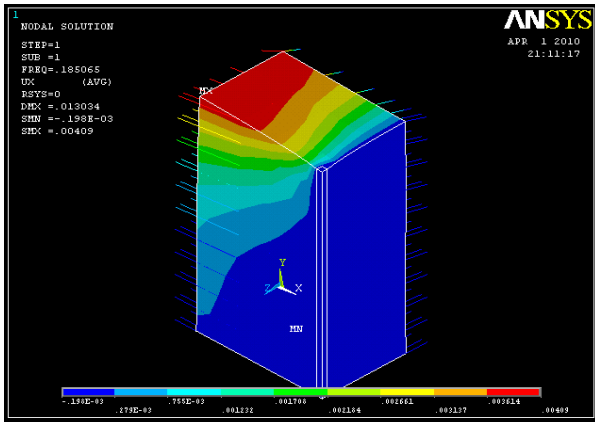


Fig. 2. Degree of freedom solution- x-component displacement

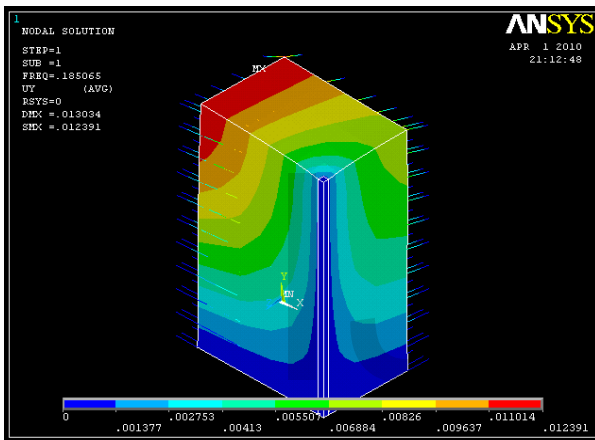


Fig. 13. Degree of freedom solution- y-component displacement

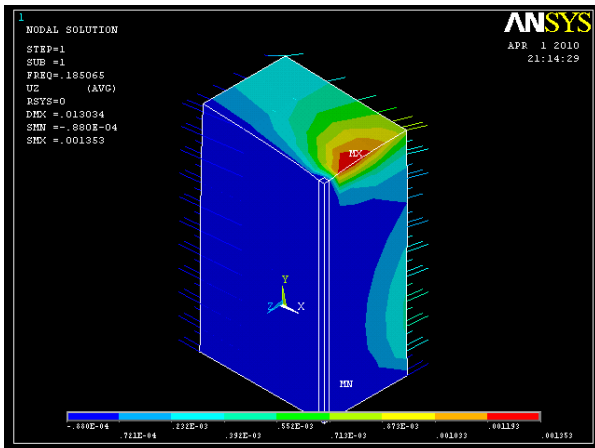


Fig. 14. Degree of freedom solution- z-component displacement

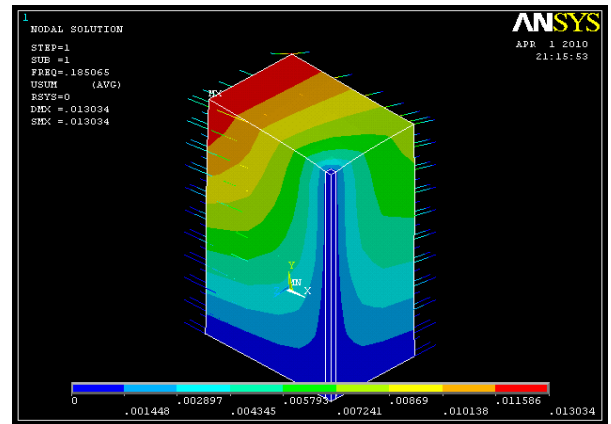


Fig. 3. Degree of freedom solution- vector sum of all displacement

3.4.2 Total strain (elastic, plastic, creep and thermal)

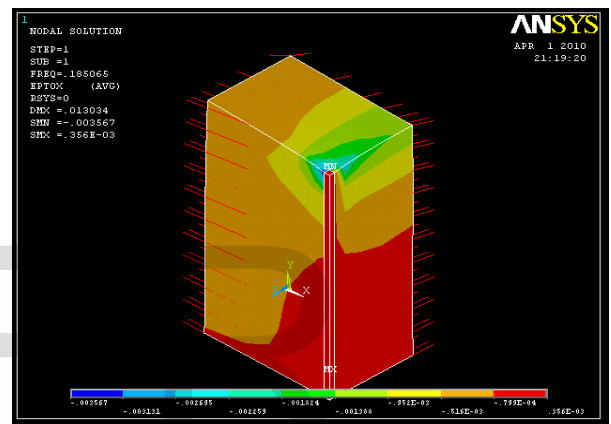


Fig. 16. Total strain plot- x-direction

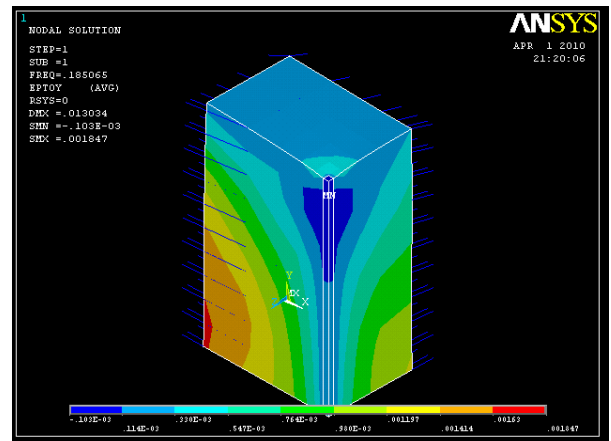


Fig. 17. total strain plot- y-direction

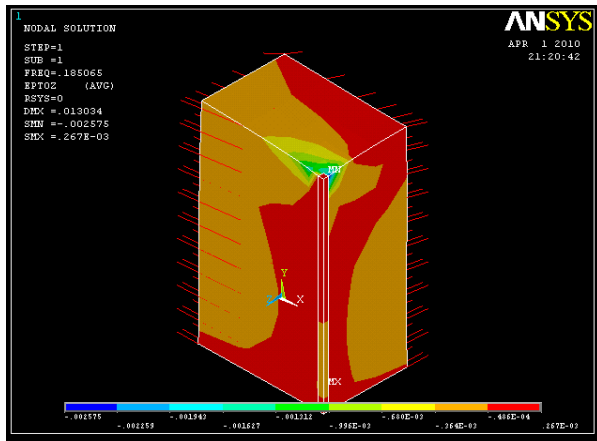


Fig. 18. Total strain plot- z-direction

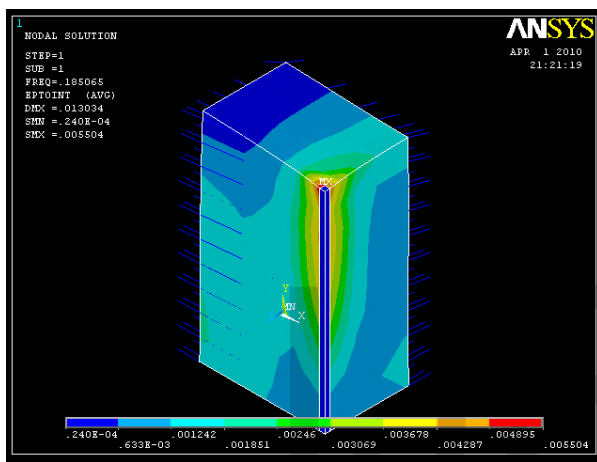


Fig. 19. Total strain intensity plot

4 RESULTS AND DISCUSSIONS

The model prepared for this study is verified for horizontal static loading at the pile top. A similar model was prepared by Rajib Sarkar, research scholar, University of Roorki, India. For the dynamic analysis, following assumptions were taken,

$$Q_p/Q_s = 1.92$$

$$v_p/v_s = 0.60$$

Q_p = mass density of pile

Q_s = mass density of soil

v_p = Poisson's ratio of pile

v_s = Poisson's ratio of soil.

Mass density of pile is taken as 2450kg/m³. So, mass density of soil is taken as 1280 kg/m³.

Poisson's ratio of soil varies from 0.3 to 0.4, with lower values for firm soil and higher values for soft soil. A value of Poisson's ratio of soil is taken as 0.35 for this study. So, the Poisson's ratio of pile is 0.2.

E_p/E_s is one of the study variables. It ranges from 500 to 1000, a dimensionless number.

E_p = modulus of elasticity of pile material

E_s = modulus of elasticity of soil

The value of soil modulus of elasticity ranges from 7000000 kN/m² to 35000 kN/m². The value of E_p is calculated from the desired ratio.

f_n = natural frequency of vibration, in this study, frequency range 5 Hz to 30 Hz is taken. Frequencies below 5 Hz don't create much effect, so ignored. Equivalent spring constant of soil (k) ranges from 1500 kN/m to 2000 kN/m.

4.1 Variation of Top Displacement with E_p/E_s

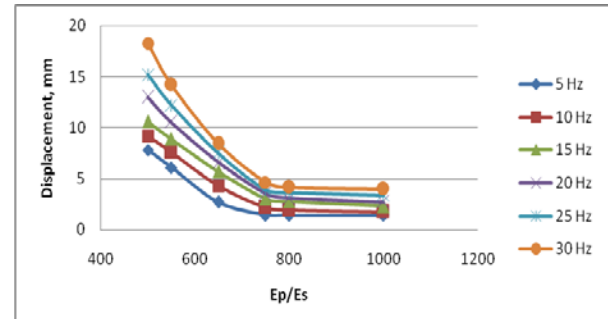


Fig. 20. Top displacement vs. E_p/E_s plot with varying frequencies. No vertical load. $E_s = 7000000$ kN/m². $k = 1500$ kN/m

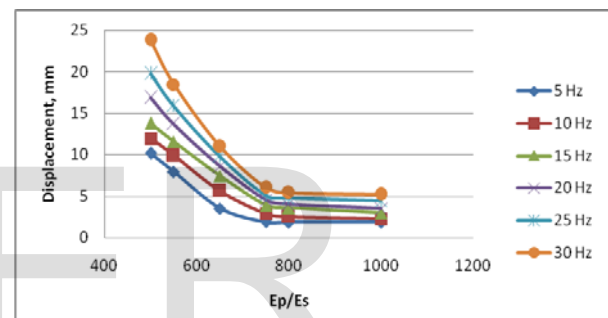


Fig. 21. Top displacement vs. E_p/E_s plot with varying frequencies. No vertical load. $E_s = 1000000$ kN/m². $k = 1500$ kN/m

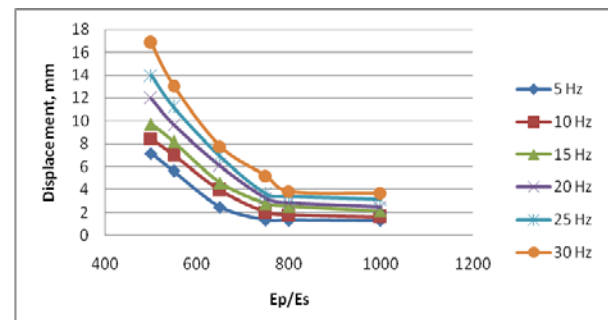


Fig. 22. Top displacement vs. E_p/E_s plot with varying frequencies. No vertical load. $E_s = 7000000$ kN/m². $k = 1750$ kN/m

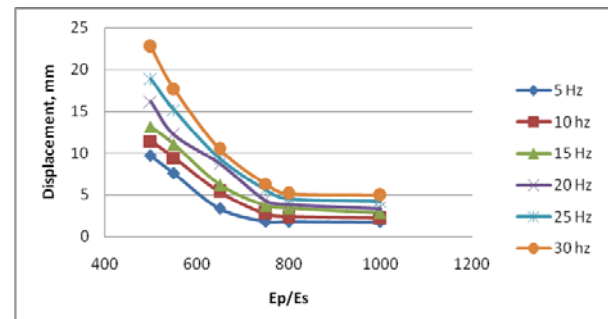


Fig. 23. Top displacement vs. E_p/E_s plot with varying frequencies. No vertical load. $E_s = 1000000$ kN/m². $k = 1750$ kN/m

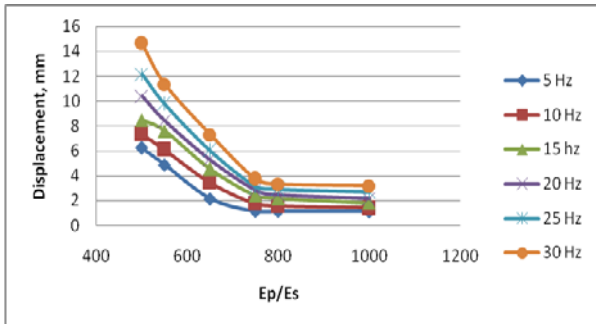


Fig. 24. Top displacement vs. E_p/E_s plot with varying frequencies. No vertical load. $E_s= 7000000 \text{ kN/m}^2$. $k= 2000 \text{ kN/m}$

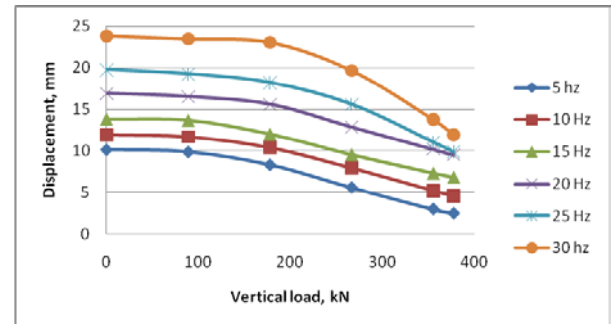


Fig. 28. Top displacement vs. Vertical load plot with varying frequencies. $E_p/E_s= 500$. $E_s= 1000000 \text{ kN/m}^2$. $k= 1500 \text{ kN/m}$

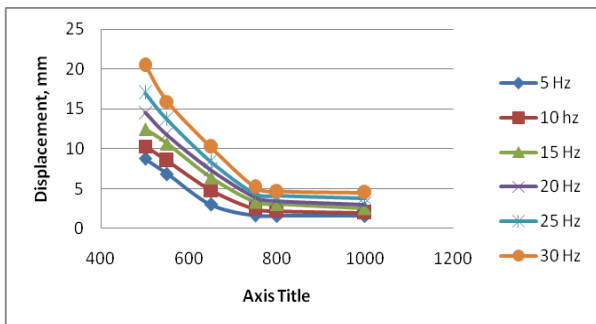


Fig. 25. Top displacement vs. E_p/E_s plot with varying frequencies. No vertical load. $E_s= 1000000 \text{ kN/m}^2$. $k= 2000 \text{ kN/m}$

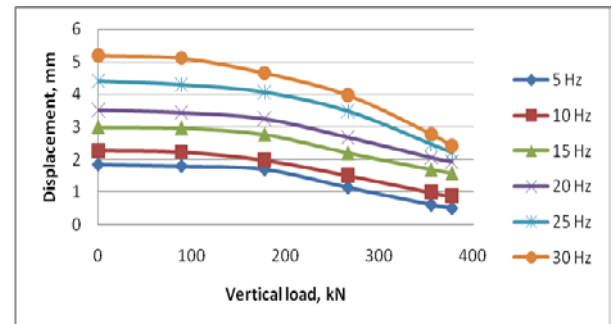


Fig. 29. Top displacement vs. Vertical load plot with varying frequencies. $E_p/E_s= 1000$. $E_s= 1000000 \text{ kN/m}^2$. $k= 1500 \text{ kN/m}$

4.2 Variation of Top Displacement with Concentric Vertical Load

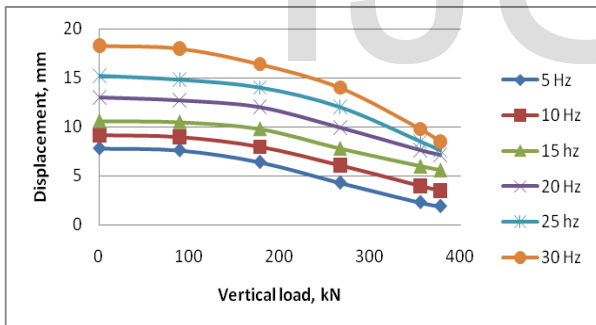


Fig. 26. Top displacement vs. Vertical load plot with varying frequencies. $E_p/E_s= 500$. $E_s= 7000000 \text{ kN/m}^2$. $k= 1500 \text{ kN/m}$

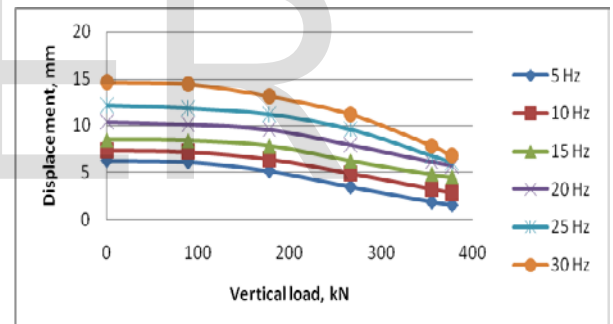


Fig. 30. Top displacement vs. Vertical load plot with varying frequencies. $E_p/E_s= 500$. $E_s= 7000000 \text{ kN/m}^2$. $k= 2000 \text{ kN/m}$

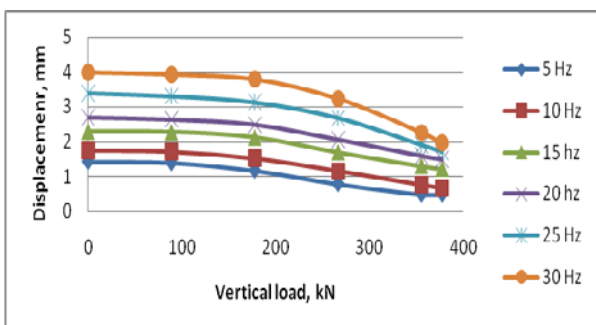


Fig. 27. Top displacement vs. Vertical load plot with varying frequencies. $E_p/E_s= 1000$. $E_s= 7000000 \text{ kN/m}^2$. $k= 1500 \text{ kN/m}$

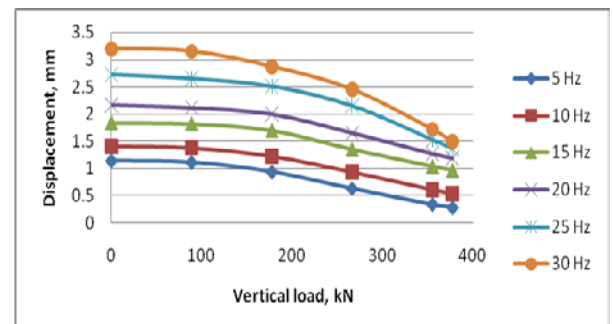


Fig. 31. Top displacement vs. Vertical load plot with varying frequencies. $E_p/E_s= 1000$. $E_s= 7000000 \text{ kN/m}^2$. $k= 2000 \text{ kN/m}$

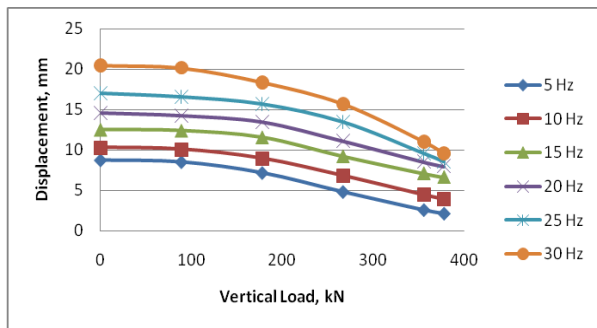


Fig. 32. Top displacement vs. Vertical load plot with varying frequencies.
 $E_p/E_s = 500$. $E_s = 1000000 \text{ kN/m}^2$. $k = 2000 \text{ kN/m}$

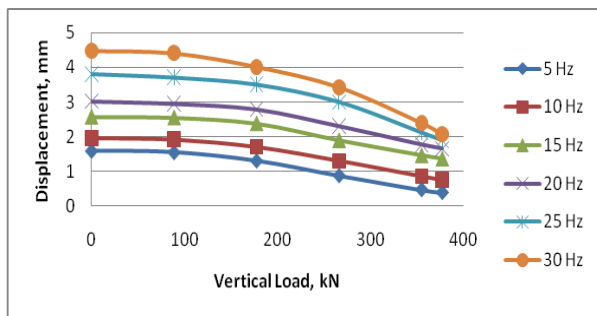


Fig. 33. Top displacement vs. Vertical load plot with varying frequencies.
 $E_p/E_s = 1000$. $E_s = 1000000 \text{ kN/m}^2$. $k = 2000 \text{ kN/m}$

4.3 Comments on the Graphical Representations

Linear interpolation between the suitable ranges is applicable for the values of the variables for which the displacement are not plotted. Suitable factor of safety must be considered to overcome the risk of any accidental case. The soil mass was assumed to be homogeneous and isotropic throughout the analysis but it is a rare case. Values of E_p/E_s and k are not constant in any practical pile-soil system. Better solution may be obtained if the variables are computed for the soil in the vicinity of the pile.

4.4 Results Analysis

- Displacement at the top of the pile is increased up to 10% for the increment of frequency of 5 Hz, where other parameters such as modulus of elasticity of soil, soil spring constants remain the same, and vertical concentric loads are absent.
- While the modulus of elasticity of soil remains constant, the pile may be strengthened to increase the E_p/E_s value. 50% increase of E_p/E_s ratio will result in 70 to 80% decrease of deflection at pile top, lower percentage of deflection decrement occur at lower frequencies, while larger percentage of decrement occurs at higher frequencies.
- Increase of E_p/E_s value may not be efficient, for example, 100% increase of E_p/E_s ratio will results 75 to 85% decrease of deflection at top of pile.
- From pile top displacement vs. E_p/E_s plot it is observed that, the slopes of the curves are steep at the lower values of E_p/E_s . As the E_p/E_s value increases the slope decreases.

- No significant displacement variations are found if the E_p/E_s value increase after a value of 750. So it would be worthwhile to strengthen the pile to obtain an E_p/E_s value greater than 750 when the earthquake protection is the only purpose.
- Concentric vertical loads reduce the pile top displacement due to vibration up to 20 to 60% when the pile is designed for service load equal to 80% of the pile capacity. Higher displacement reduction is obtained when the E_p/E_s value is small.

5 CONCLUSION

The major purpose of this study was to prescribe some probable solution to counteract the detrimental effects of earthquake vibrations. The study is expected to generate reasonable solutions of focused problem defined under some parametric conditions. Some of the variables are chosen which may be improved for better solutions. To be familiar with the graphical representations, one has to examine for the probable vibration frequencies that may occur during the design period of the structures. Then he has to investigate the allowable pile deflection that may not cause the serious threat to the proposed structures while vibrating with investigated frequency. The graphical representations would help to chose the counter measures, such as increase of vertical load, soil improvement to increase its spring constant or Young's modulus of elasticity, modification of pile dimension, or provision of high steel contribution to increase the modulus of elasticity of pile. Cost concern to adopt suitable technique should be another variable. In recent, most of the tall structures are constructed for residential purposes. Only a few of the high rise structures are constructed for commercial or industrial purposes. Residential buildings are not equipped with too much valuable materials, so, earthquake proof design for residential buildings is done to save people from death, and it is the intangible benefit of this system, leaving no opportunity to evaluate the benefit-cost analysis. When the buildings are equipped with valuable machineries, economic analysis must be carried out to investigate the proper solution.

- 50% increase of E_p/E_s ratio will result in 70 to 80% decrease of deflection at pile top, lower percentage of deflection decrement occur at lower frequencies, while larger percentage of decrement occurs at higher frequencies.
- Concentric vertical loads reduce the pile top displacement due to vibration up to 20 to 60% when the pile is designed for service load equal to 80% of the pile capacity.
- Displacement of the pile depends on the moduli of elasticity of pile, soil. Frequency, vertical loads, viscous damping properties of soil.
- Provision of higher steel area would reduce the pile top displacement.
- Vertical loads provide stiffness and so give the inertia effect of the structure.

- Equivalent spring constant of soil materials is an important variable. Viscous damping is useful for mitigation of vibrations.

REFERENCES

- [1] ANSYS 2005, *Documentation for ANSYS, Version 10.0*
- [2] Bowels, J. E., (1997), *Foundation Analysis and Design*, 5th ed., McGraw Hill, pp. 929-963.
- [3] Kuhlemeyer R.L., (1979). Vertical vibration of piles, Journal of geotechnical engineering division, ASCE, 105(GT2), 273-287.
- [4] Nilson, A. H., Darwin, D., Dolan, C. W., (2003), *Design of concrete structures*, 13th ed., McGraw Hill, pp. 700-723.
- [5] ANSYS workbench tutorial.
- [6] Sarkar, R., Maheshwari, B. K., (2008), *Three dimensional seismic analysis of pile groups*, 12th international conference of IACMAG, Goa, India.

IJSER