ISSN 2395-1621

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

#1L.M.Sardesai, #2G.A.Kadam

1sardesaileena@rediffmail.com.com
2ganeshkadam07@gmail.com

#1Mechanical Department, Savitribai Phule Pune University
SKN- Sinhgad Institute of Technology and Science, Pune, India
#2Mechanical Department
SKN-Sinhgad Institute of Technology and Science,Pune,India

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, 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.

Keywords— **Pile foundation, damped vibration, Finite Element, Continuum, Modeling, Meshing, Spectrum, ANSYS**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. 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 sub-divided 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 exact solution.

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

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

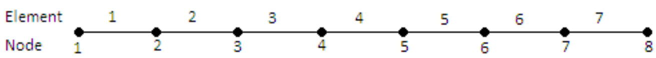
Node: The end point of each element is called nodes.

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

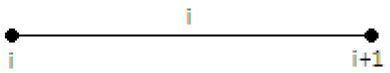
Shape Function: It is the coefficient of interpolation polynomial. The shape function is usually denoted by N. The shape function is written for each node of the element and has a magnitude 1 at the node and 0 for all other nodes in that element.

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 1+i.



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

Where,

β = beta damping

ω_i = undamped natural circular frequency of the ith mode

ξ_c = damping ratio

N_m = number of materials

$\{\varphi_i\}$ = displacement vector for mode i

$[K_j]$ = stiffness matrix of part of structure of material j

$$E_j^s = \frac{1}{2}\{\Phi_i\}^T[K_j]\{\Phi_i\} =$$

Strain energy

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].

II. ELEMENTMODELING AND ANALYTICAL STUDY

2.1 ANSYS Finite Element Modelling

To create the finite element model in ANSYS there are multi-ple 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 is,

Pile –SOLID45

Soil –SOLID 45

Spring damper-COMBIN14

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 sys-tem. 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 soil=0.35,for pile=0.3]

Moduli of elasticity's of pile and soil are the parts of design parameters

Density:

- Mass densities of the pile= 2450 kg/m^3
- Mass density of soil= 1280 kg/m^3

2.5 Modelling

Modelling is the most important part in the simulation through GUI method. The features of the pile-soil finite model are shown in the Fig. 1 below:

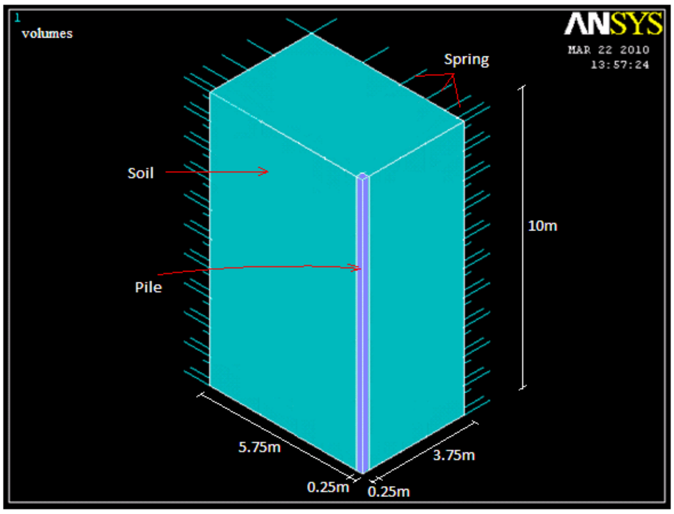


Fig. 1. Pile-soil model geometry

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 1 below

portions	Volume number	X coordinate	Y Coordinate	Z coordinate
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 behaviour, the accuracy required may be obtained by providing smaller mesh at the vicinity of the pile as shown in the Fig 2

.

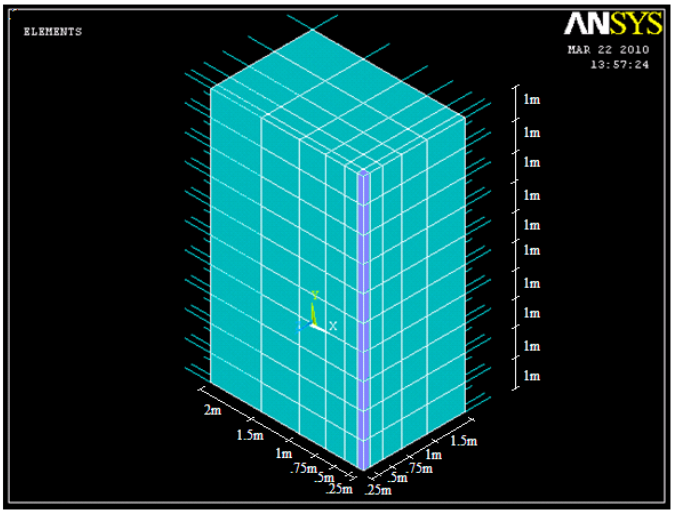
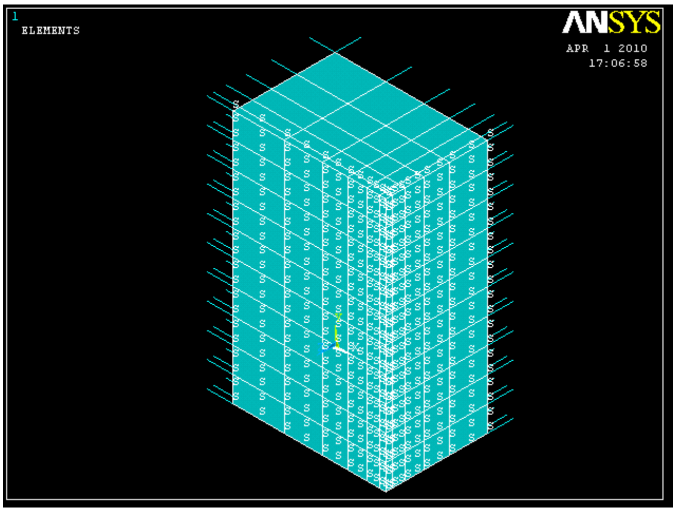


Fig 2.Meshing of model

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.



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 in fig.3

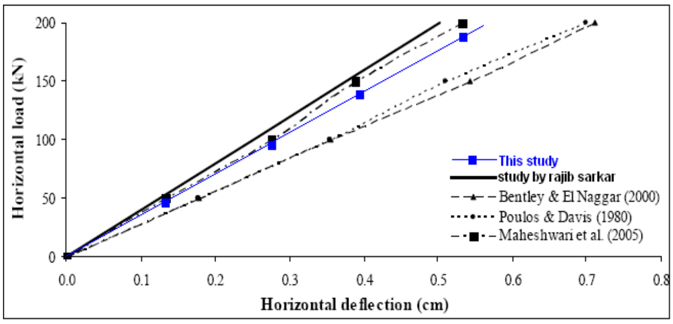


Fig. 3. Response of a single pile fixed at base for static loading

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.

III. ANALYSIS PROCESS FOR THE FINITE ELEMENT MODEL

The FE analysis of the model was set up to examine three different behaviours: 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 re-quired 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.

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 freedom

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

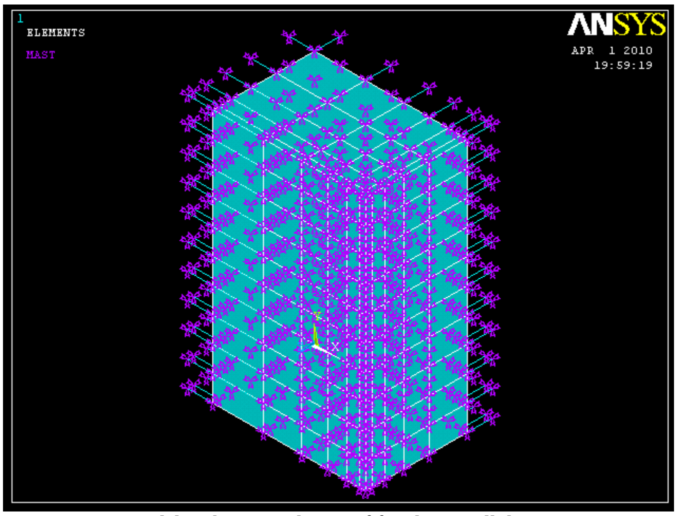


Fig 4.model with master degree of freedom in all direction

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 colour stripes having different value ranges for each colour. Vector plot gives arrow diagram with the direction of changes of values with different colour.

3.3 Graphical Reprentation of the Results

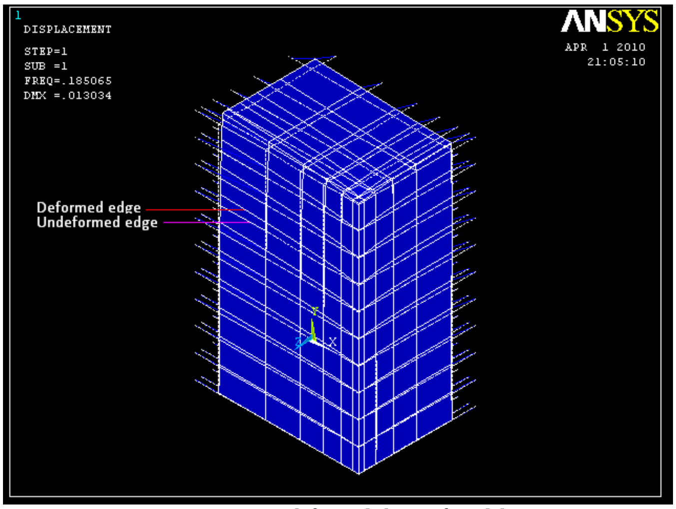


Fig 5.maximum deformed shape of model

3.4 Nodal Solution

3.4.1 Degree of freedom solution

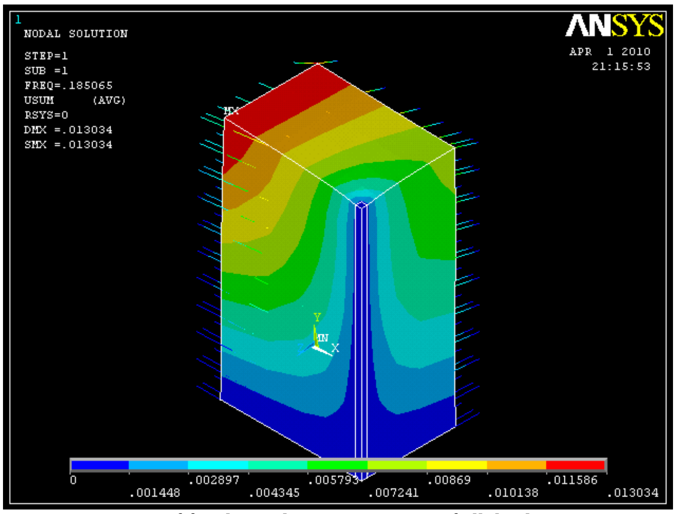


Fig5.Degree of freedom solution-vector sum of all displacement

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

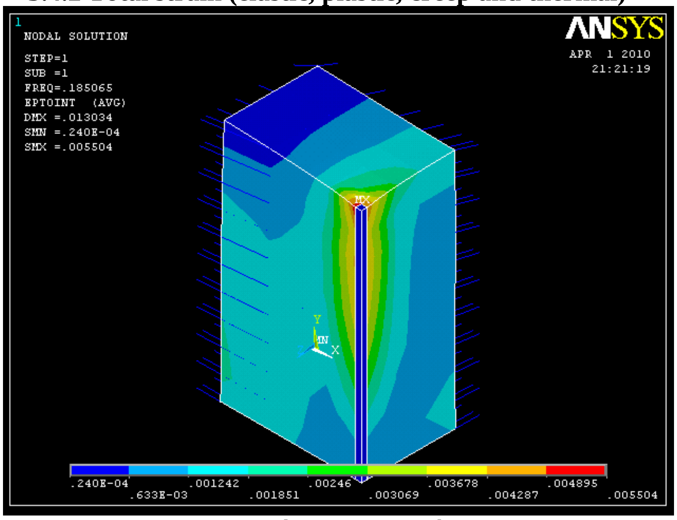


Fig 6.Total strain intensity plot

IV. 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 Ep/Es value. 50% increase of Ep/Es 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 oc-curs at higher frequencies.
- Increase of Ep/Es value may not be efficient, for ex-ample, 100% increase of Ep/Es ratio will results 75 to 85% decrease of deflection at top of pile.
- From pile top displacement vs. Ep/Es plot it is ob-served that, the slopes of the curves are steep at the lower values of Ep/Es. As the Ep/Es value increases the slope decreases.
- No significant displacement variations are found if the Ep/Es value increase after a value of 750. So it would be worthwhile to strengthen the pile to obtain an Ep/Es 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 Ep/Es value is small.

V. CONCLUSIONS

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 Ep/Es 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

REFRENCES

[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.

