



**A THEORETICAL STUDY ON THE PERFORMANCE  
OF PILED FOUNDATION AS A METHOD OF  
ASEISMIC DESIGN**



*P- 2238*

**PROJECT REPORT**



**Submitted by**

**C.Jayakumar**

**Register no: 71206413007**

in partial fulfillment of the requirements

for the Award of the degree

of

**MASTER OF ENGINEERING**

in

**STRUCTURAL ENGINEERING**

**KUMARAGURU COLLEGE OF TECHNOLOGY**

**COIMBATORE – 641 006**

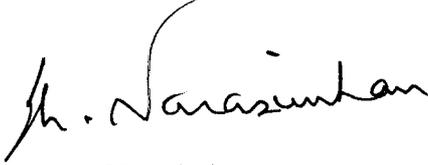
Affiliated to

**ANNA UNIVERSITY, CHENNAI - 600 025.**

July 2008

## BONAFIDE CERTIFICATE

Certified that this project report titled “**A THEORETICAL STUDY ON THE PERFORMANCE OF PILED FOUNDATION AS A METHOD OF ASEISMIC DESIGN**” is the bonafide work of **C.JAYAKUMAR, Reg No:71206413007**, who carried out the research work under my supervision. Certified further, that to the best of my knowledge the work reported herein does not form part of any other project report or dissertation on the basis of which a degree or award was conferred on an earlier occasion on this or any other candidate.



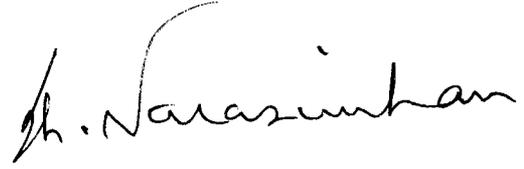
Dr.S.L.Narasimhan

Head of the Department

Department of Civil Engineering

Kumaraguru College of Technology

Coimbatore-641 006



Dr.S.L.Narasimhan

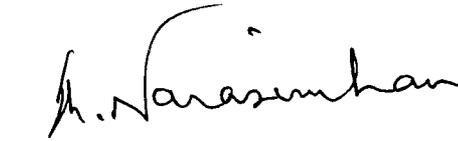
Thesis Supervisor

Department of Civil Engineering

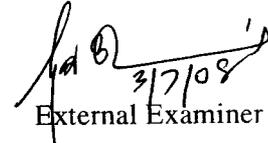
Kumaraguru College of Technology

Coimbatore-641 006

Submitted for the project Viva voce examination held on... 03.07.08 ... at  
Kumaraguru College of Technology, Coimbatore-641 006.



Internal Examiner



External Examiner

## ACKNOWLEDGEMENT

I have immense pleasure and profound gratitude in expressing heartfelt thanks to my guide and **Dr.S.L.Narasimhan**, professor and Head, Department of civil engineering for his valuable guidance, suggestions and great encouragement throughout the project work. His involvement and guidance were invaluable and helped me complete it effectively and efficiently.

I take this opportunity to thank **Dr. Joseph V.Thanikal**, Principal, Kumaraguru College of technology, Coimbatore for having provided the facilities for carrying out the project.

I express my sincere gratitude to **Dr.D.L.Venkatesh Babu**, Professor and STC in charge, for his valuable suggestions throughout the project work.

I express my sincere gratitude to all the **Faculty of civil Engineering** for their timely suggestions and help rendered in connection with this thesis work.

I express my sincere thanks to **Mr.Ragunathan**, CAD Lab Technician for his timely help in connection with this thesis work

I am grateful to all my **Friends** for their timely help and support throughout my project.

Finally I express my very special thanks to my **Family** members for their patience and moral support without whom nothing would have been successful in this project.

## Abstract

Traditionally in Earthquake Engineering, it is usually assumed that Soil-Structure-interaction is beneficial during an Earthquake. Because of this, it has become a common practice to avoid the complication of accounting for Soil-Structure-interaction by simply ignoring its effects. This avoidance is thought to lead to improved safety margins while simplifying the analysis. Soil-structure-interaction induces damping, which may be of high magnitude, ignoring which the analysis may be too conservative. It can also cause increased displacement in the overall structure, due to liquefaction of soil at times. Due to this possibility, it is necessary to take Soil-Structure-interaction into consideration during analysis, for more accurate results. Literature in such analysis is very limited and more work is required to be done. Hence the objective of this investigation is to perform an analytical study of the dynamic response of the super structure-pile-soil system under seismic loads.

For this study two types of analytical models are created. The models are performed with and with out having piles. The most widely used model to perform the analysis of piles under seismic loads consists of modeling the pile as a series of beam elements and representing the soil as a group of unconnected, concentrated springs perpendicular to the pile along with dashpot to model dampening. The above simplified lumped model was created using ANSYS software and superstructure has been modeled as beam column elements. In the present approach the pile-soil system is taken into account through three frequency independent elements: a spring with stiffness  $K_a$ , a mass with value  $M_a$ , and a dashpot with coefficient  $C_a$ . The spring mass dashpot coefficients  $K_a$ ,  $M_a$  &  $C_a$  that represent the soil has been obtained from previous literature.

Both the models are analyzed by transient analysis, and the response of the super structures is measured. Finally both the results are compared to each other. Comparison of the results from the analytical study of the superstructure with and without considering pile and soil, gives an idea about the effect of ignoring soil-pile-structure interaction.

## TABLE OF CONTENTS

DESCRIPTION	Page No
Acknowledgement	
Abstract	
Table of contents	
List of figures	
List of tables	
 Chapter 1	
<b>INTRODUCTION</b>	
1.1.Justification	1
1.2 Objectives and Scope	2
1.3 General Procedures and Methods	2
1.4. Review of literature	
1.4.1. General	3
 Chapter 2	
<b>INTRODUCTION TO FINITE ELEMENT METHOD</b>	
2.1.Introduction	10
2.2.Historical Background	11
	11
2.3.What is Finite Element Analysis?	
	12
2.4. How Does Finite Element Analysis Work?	
2.5.Types of Engineering Analysis	13
2.6.Stages involved in FEM	14
2.6.1.Pre-Processor	14
2.6.2.Solution	15
2.6.3.post-processor	15
2.7.Pre-requisite for Finite Element Methods:	16
2.8.Basics of solid mechanics	16

## 2.9.Features available in ANSYS

2.9.1. Introduction	20
2.9.2.List of industries using ANSYS	20
2.9.3.A Typical Analysis	21
2.9.4.Sub modeling	22
2.9.5.ANSYS Parametric Design Language (APDL)	22
2.9.6.Design optimization	22
2.9.7.Sub structuring	23
2.9.8.Element birth and death	23

## Chapter 3

### **FINITE ELEMENT ANALYSIS**

3.1. Introduction	24
3.2.Numerical modeling	24
3.2.1.Frame configuration	24
3.2.2Superstructure alone	24
3.2.3. Superstructure-soil-pile system	26
3.2.3. (a) Horizontal stiffness and damping co efficient for single pile	26
3.2.4. Deformed shapes of the models	29
3.2.4 (i) Super structure without soil-pile system	29
3.2.4 (ii) Super structure with soil-pile system	30
3.2.5. Finite element meshing	31
3.2.6. Boundary conditions	31
3.2.7. Material properties	33
3.2.8. Seismic loading	33
3.3. Results	
3.3.1. General	34
3.3.2. Lateral Displacements	34

3.4. Stresses at the elements	38
3.4.1. Stresses at all the elements in superstructure without pile system	38
	39
3.4.2. Minimum and maximum stress value in superstructure without pile system	
	39
3.4.3. The stresses at the roof levels	
3.4.4. Stress at all the elements in the superstructure with pile system	40
3.4.5. Minimum and maximum stress value	41
3.4.6. The stress at the roof levels	42
3.5.conclusion and future work	
3.5.1.General	42
3.5.2.Conclusions	43
3.5.3.Scope for further work	44

## LIST OF FIGURES

<b>TABLE</b>	<b>TITLE</b>	<b>PAGE NO</b>
3.1	Super structure elements	25
3.2	Super structure-soil-pile system elements	28
<b>DISPLACED POSITIONS</b>		
3.3	Super without soil pile system	29
3.4	Super with soil pile system	30
<b>BOUNDARY CONDITIONS</b>		
3.5	Super without soil pile system	31
3.6	Super with soil pile system	32
<b>RESULTS</b>		
3.7	Initial displacement pattern curve	35
3.8	Displacement pattern of super structure without soil-pile system	36
3.9	Displacement pattern of super structure with soil-pile system	37

## LIST OF TABLES

TABLE	TITLE	PAGE NO
3.1	Stresses at all the elements in superstructure without pile system	39
3.2	Minimum and maximum stress value in superstructure without pile system	39
3.3	Stresses at the roof levels in superstructure without pile system	40
3.4	Stresses at all the elements in superstructure with Pile system	41
3.5	Minimum and maximum stress value in superstructure with pile system	41
3.6	Stresses at the roof levels in superstructure with pile system	42
3.7	Comparison of results	43

# **Chapter1**

## **Introduction**

This chapter presents the justification of the developed research project, and the objectives and scope of the work, together with a brief discussion of the methodology adopted to develop the project.

### **1.1 Justification**

Foundation systems composed by isolated piles or pile groups are extensively used to support different types of structures placed over soft soil layers, where shallow Foundations are not appropriate because they do not provide the required capacity, or may experience too large settlements.

These pile foundations have to be designed to support lateral loads due to earthquakes, wind, and vehicle impact loads, among others. The most widely used model to perform the analysis of piles under lateral loads consists in modeling the pile as a series of beam elements, and considering the pile-soil interaction by representing the soil as a group of unconnected, concentrated springs perpendicular to the pile (Discrete Winkler Model). In order to consider soil non-linear behavior, the springs can have a varying stiffness given through a non-linear load-deflection relationship that depends on the type of soil and type of pile, known as p-y curves.

To adequately address pile response under earthquake actions, or to analyze heavy Vibratory machine foundations, it is often required to perform a dynamic analysis of the pile for transverse (lateral) vibrations. For a dynamic analysis it is critical to have an adequate representation of the system stiffness (force-deflection relationships), and adequate representation of the system mass involved in the vibration phenomena. Up to now, the Winkler model that is generally used only takes into account the mass of the pile, and does not consider the possible soil mass contribution to the inertia Characteristics of the system.

The literature review (presented in Chapter 2) shows that the soil stiffness and damping properties are included in a dynamic analysis through lumped springs and dashpots, but a lumped mass to represent soil inertia is not included. The objective of this investigation is to perform an analytical and numerical study of the dynamic response of the pile-soil system under lateral loads in order to assess the importance of the soil mass in the system response

## **1.2 Objectives and Scope**

The main objective of this investigation is to assess the importance of the soil mass in the soil-pile-super structure system response under dynamic lateral loads, through lumped masses consistent with the discrete Winkler model. This would lead to a better representation of the dynamic soil-pile interaction.

After performing the analytical studies necessary to develop a procedure for the inclusion of the soil mass into the system, a series of worksheets and computer programs were developed in order to perform the dynamic analysis of the system response.

After performing the verification and validation of the model, and the evaluation of some case studies, conclusions regarding the significance (or not) of including soil inertial properties in the analysis are presented, the accuracy of the proposed model is discussed, and recommendations for possible future work are developed.

In order to limit the scope of the project, the analysis focuses on a single pile located in a homogeneous soil deposit, evaluated as a semi-infinite half space. The pile has a straight axis, circular cross section, and is placed vertically. The soil was assumed not saturated, so pore pressure effects (including liquefaction) were not an issue.

## **1.2 General Procedures and Methods**

**Following is a list of the primary tasks conducted during this research:**

- Develop a thorough study and literature review of relevant topics. The summary of the findings are presented in Chapter 2.

- Perform a numerical study of the influence of the soil mass on the system response by performing a transient analysis of 2D and 3D Finite Element Models of a case study of a pile-soil-superstructure. The study is performed without considering the pile system, and including the pile system.
- The main objective is to compare the response of the super structure with and without the soil- pile system.

## **1.4. REVIEW OF LITERATURE**

### **1.4.1. General**

Several researchers have studied the performance of the piled foundation under seismic loading over last two decades. Literature in the field of soil-pile-superstructure and the response of the superstructure are scanty. A brief discussion on the available literature in the fields relevant to the present work is presented in the following few paragraphs.

#### **Analytical Study on Dynamic Characteristic of Piled Raft Foundation (Kenji Iwamoto, Hajime Hagiwara, and Kojiro Takesue)**

Piled Raft Foundation is categorized as a type of combined raft and pile foundation. Efficient design of foundation with a high degree of freedom is possible because the combination of raft and pile resists the loads of the superstructure. On the other hand, it is necessary to predict the performance in advance because soil-structure-interaction becomes so complicated. The objective of this paper is to clarify the dynamic characteristics of piled raft foundation using the newly proposed method which is based on Thin Layer Element Method (TLM). Dynamic impedance, foundation input motion, and earthquake response of piled raft foundation is investigated. Parameters such as soil conditions and arrangement of pile were adopted for the analytical study. From the results, the main conclusions obtained were as follows. For foundation of the same shape, when the number of piles is increased, the rotational dynamic impedance of piled raft

foundation becomes larger. On the other hand, the influence of the number of piles on the horizontal dynamic impedance was small. Even if the distance between piles is increased, the pile stress at the pile head in piled raft foundation depends on the location of each pile, although in pile foundation under same conditions differences in pile stress at the pile head are small.

### **Soil-pile-superstructure interaction in liquefiable sand**

**( Boulanger r, R W; Wilson, D W; Kutter, B L; Abghari, A)**

Soil-pile-superstructure interaction in liquefiable sand is evaluated using dynamic centrifuge model tests and pseudo static p-y analyses. Select recordings from a recent centrifuge test are presented to illustrate typical behavior with and without liquefaction in an upper sand layer. Pseudo static p-y analyses of single-pile systems in two recent centrifuge model tests show that the apparent reduction in p-y resistance due to liquefaction was strongly affected by changes in the relative density of the sand and drainage conditions.

### **Analysis of pile soil interaction (yasser a. khodair , sophia hassiotis )**

A three-dimensional, nonlinear finite element (FE) model is developed and used to study stresses on piles and pile-soil interaction in the Scotch Road, integral abutment bridge, located in Trenton, New Jersey. The FE model consists of soil continuum elements, with material non-linearities, for the piles and soil respectively. The handling of soil-structure interaction in the analysis and design of integral abutment bridges has always been problematic. The objective of this paper is to study the mechanism of pile-soil interaction and the effects of thermal expansions/contractions on the lateral deformations of the pile-soil system embedded in galvanized steel sleeves filled with sand. The bridge substructure was fully instrumented using strain gages. The displacements induced by temperature changes were applied to the piles and the FE results were compared using experimental data and finite difference solutions. The FE model provided reliable results.

### **Three Dimensional Finite Element Analyses of Laterally Loaded Piles**

**(Varghese, Sabu P)**

Behavior of a single pile and two-pile groups in clay, subjected to horizontal load at the pile head has been investigated through parametric studies. Pile and soil are modeled using three-dimensional finite element techniques treating pile as linear elastic and soil as linear-elastic-perfectly-plastic material. Soil is also assumed to be incapable of sustaining any tensile stresses. Von Mises criterion has been used to define plastic yielding in soil. Separation occurring at the interface of soil and pile under the lateral load has been accounted for by incorporating interface surface elements in the modeling. Two different configurations, viz. parallel and series have been considered for the two-pile group analysis. Individual piles in a group are assumed to be rigidly connected together at the pile head. Parametric studies have been performed to examine the effects of soil-pile separation, pile length, soil shear strength and pile modulus on the single pile capacity and bending moment distribution in the piles. Two different pile head fixity conditions, viz. free and fixed, have also been examined. Effects of pile length, pile spacing, pile configuration and soil shear strength have been investigated on the capacities of individual piles of two-pile groups. Development pattern of tension and plastic yield zones around the piles has also been identified. It has been observed that soil-pile separation causes reduction in pile lateral capacity and increase in bending moments developed in the pile. Capacity of individual piles of a two-pile group has been compared with that of a single pile to identify the reduction in capacity due to group effect. It has also been observed that the individual piles in two-piles in series configuration carry the applied lateral load unequally and the pattern of load sharing has been identified with respect to pile length and pile spacing. Unique relationships are found to be existing between the maximum bending moments developed in the piles, applied pile head load, pile penetration length and soil shear strength for all the cases of single pile and two-pile groups investigated. Progressive development of tension and plastic yield zones around single pile and two-pile group has been identified for various pile lengths, pile head conditions, pile configurations and pile spacing.

### **Advance simulation tools soil structure interaction (Mary Grondin)**

Traditionally, in earthquake engineering, it has been common to assume that soilstructure interaction (SSI) is beneficial during an earthquake. Because of this it has become common practice to avoid the complication of accounting for SSI by simply ignoring its effects. This avoidance is thought to lead to improved safety margins while simplifying the analysis. New trends in earthquake engineering include analyzing the displacement that a structure undergoes during an earthquake, and considering the structural as well as nonstructural damage that this causes. Even though soil-structure interaction induces dampening, it can also cause increased displacement in the overall structure. Due to this possibly detrimental effect, it is necessary to take soil-structure interaction into consideration during analysis. The problem is that this interaction is a very complex phenomenon, dealing with nonlinear and frequency-dependent behavior. Although methods for analyzing SSI in a linear environment have been developed, a method that includes the nonlinear behavior of the soil and foundation is not available.

The objective of this project is to develop advanced response analysis methods that can estimate the response of different structures subject to earthquake induced ground motion, and include linear as well as nonlinear behaviors.

### **3D finite element analysis on pile-soil interaction of passive pile group (Daichao Sheng, K. Dieter Eigenbrod and Peter Wriggers )**

This paper presents some observations on stress and displacement characteristics during the installation and loading of pushed-in piles. A commercial finite element code with the capability of simulating large-strain frictional contact between two or more solid bodies is used to simulate the pile installation and pile loading. The soil is treated as a modified Cam clay material, whereas the pile is treated as a rigid body. The computed total resistance and shaft resistance during pile installation are first compared with measured values from centrifuge tests, which indicates that the total resistance is well predicted by the finite element model, but not the shaft resistance. The difference between the computed shaft resistances and the measured values is mainly due to the cone effects introduced in the finite element model. The computed stress paths indicate

that both the mean and deviator stresses first increase when the pile cone is above or at the level of the observation point in the soil, and then decreases once the pile cone has moved below the observation point. When the soil is represented by the modified Cam clay model, a thin layer of soil of one pile radius immediately around the pile, extending from the ground surface to a distance of one pile radius above the pile cone, is under elasto-plastic expansion. Just outside this expansion (softening) zone, a compression zone of a 'U' form is observed. The characteristics of the stress paths and volumetric behavior are not significantly affected by the initial OCR of the soil. The volumetric behavior is however strongly affected by the constitutive model used for the soil. The so-called h/R effect is also well captured by the finite element model.

### **Simplified Finite Element Modeling of Nonlinear Dynamic Pile-Soil Interaction (A. Anandarajah and J. Zhang)**

Using a fully-coupled finite element analysis method, the earthquake behavior of a single pile subjected to an earthquake base shaking in a centrifuge is analysed. The fully-coupled formulation is based on the mixture theory, and uses solid and fluid displacements and pore water pressure as variables. The resulting matrix equations are symmetric. An effective stress based elastoplastic constitutive model is used to represent the stress-strain behavior of the sand. The constitutive model is based on associated flow rule, and hence yields a symmetric stiffness matrix. The pile-soil interaction problem is analysed using a two-dimensional model of the problem, and it is shown that the results from the fully-coupled procedure compare reasonably well with centrifuge data.

### **Soil-Pile-Superstructure Interaction in Liquefying Sand and Soft Clay (D. W. Wilson)**

The behavior of pile foundations under earthquake loading is an important factor affecting the performance of many essential structures. Analysis and design procedures have been developed for evaluating pile behavior under earthquake loading. The

application of these procedures to cases involving soft or liquefied ground is uncertain, however, due to both a lack of physical data against which they can be evaluated, and the continued lack of understanding of the mechanisms involved in soil-pile-structure interaction. Resolving these uncertainties is an important step in current earthquake hazard remediation.

This dissertation describes the results of a study on the dynamic response of pile foundations in liquefying sand and soft clay during strong shaking. The research consisted of: (1) a series of dynamic centrifuge tests of pile supported structures; (2) a critical study of modeling techniques and limitations; (3) back-calculation of p-y behavior; and (4) comparison of pseudo-static analyses to the dynamic centrifuge model tests.

These dynamic model tests were among the first performed using the new shaking table on the 9 m radius centrifuge at UC Davis. The results of the modeling study Presented herein will benefit other current and future projects utilizing the large centrifuge.

Back-calculation of dynamic p-y curves for liquefying sand was needed because the dynamic interaction cannot necessarily be extrapolated from static tests. This dissertation presents the first experimentally determined dynamic p-y curves in liquefying sand of which the author is aware. The p-y resistances showed characteristics that are consistent with the undrained behavior of liquefying sand, including the effects of relative density, dilation, cyclic degradation, and displacement history.

It is expected that dynamic numerical models will need at a minimum to account for undrained loading conditions to capture behaviors such as those observed in these tests. Alternatively, simplified pseudo-static analyses using reduction factors on p-y resistance can also yield reasonable design criteria provided the factors are applied with an appreciation for the time varying properties of soils during seismic loading, and special care is taken where the soil may dilate. Sensitivity studies should be performed to help determine the critical loading conditions when using simplified methods.

**Static analysis of soil/pile interaction in layered soil by BEM/BEM coupling  
(Velnario S.Almeida)**

In this article we propose to use the boundary element method (BEM) to analyze soil-foundation interactions. The soil structure is modeled as several dissimilar strata placed one on top of the other, composing a sandwich-like profile. Any of these layers may contain components of the foundations. Each region occupied by a soil layer or by a foundation component is handled as a 3D isotropic, elastic and homogeneous domain, and is analyzed by BEM. As a consequence of the positioning of the various sub regions in this model, the technique of successive rigidity can be applied directly, resulting in a considerable reduction in the volume of data being stored and manipulated throughout the analysis.

**Pile-Soil Interaction Determined by Laterally Loaded Pile Groups (Robbie Barton)**

It was once said that it is wise to build your house on a rock. However, what if the closest rock that is big enough is 30 feet under the soil? What happens when a structure much larger than a house needs to be built? It was these questions that guided engineers towards the concept of pile design. Piles are long, firm, column-like members that are embedded in the soil to provide axial as well as lateral support of structures such as buildings, piers, locks, and bridges. Often, piles are installed near each other to create groups to optimize the support of the structure. Both a single pile and groups of piles rely significantly upon the conditions of the surrounding soil. This study aims to take a closer look at the interaction of piles and soil to determine the optimal pile group design.

## **Seismic analysis of infinite pile groups in liquefiable soil**

**(Assaf Klar, Sam Frydman, Rafael Baker)**

Numerical analysis of an infinite pile group in a liquefiable soil was considered in order to investigate the influence of pile spacing on excess pore pressure distribution and liquefaction potential. It was found that optimal pile spacing exists resulting in minimal excess pore pressure. It was also found that certain pile group configurations might reduce liquefaction potential, compared to free field conditions. It was observed that for closely spaced piles and low frequency of loading, pile spacing has little influence on the response of the superstructure.

equations, to provide the displacement of every node in the model. Once the displacement field is determined, the strains and hence the stresses can be derived using the strain-displacement and stress-strain relations respectively.

## 2.2 Historical Background

As is often the case with original developments, it is rather difficult to quote an exact "date of invention", but the roots of the finite element method can be traced back to three separate research groups: **Applied Mathematicians -R.Courant; Physicists - J.L.Synge; and Engineers-J.H.Argyris and S.Kelsy.** Although the principle published already, finite element method obtained its real impetus from the development of engineers. The original contributions appeared in the papers by J.H.Argyris and S.Kelsy; R.W.Clough coined M.J.Turner, R.W.Clough, H.C.Martin, and L.J.Topp. The name "finite element". Important early contributions were those of J.H.Argyris and O.Zienkiewicz and Y.K.Cheung. R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems, first developed finite Element Analysis (FEA) in 1943. Shortly thereafter, a paper published in 1956 by M. J.Turner,R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day super computers are now able to produce accurate results for all kinds of parameters.

## 2.3 What is Finite Element Analysis?

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing



product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis that are used in industry: 2-D modeling, and 3-D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms (functions), which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

#### **2.4 How Does Finite Element Analysis Work?**

FEA uses a complex system of points called nodes, which make a grid called a mesh. This mesh is programmed to contain the material and structural properties, which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. Regions, which will receive large amounts of stress usually, have a higher node density than those, which experience little or no stress. Points of interest may consist of: fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The mesh acts like a spider web in that from each node, there extends a mesh element to each of the adjacent nodes. This web of vectors is what carries the material properties to the object, creating many elements (Theory).

A wide range of objective functions (variables within the system) are available for minimization or maximization: Mass, volume, temperature, Strain energy, stress strain, Force, displacement, velocity, acceleration, Synthetic (User defined) There are multiple loading conditions which may be applied to a system. Point, pressure, thermal, gravity, and centrifugal static loads, Thermal loads from solution of heat transfer analysis, Enforced displacements, Heat flux and convection, Point, pressure and gravity dynamic

loads. Each FEA program may come with an element library, or one is constructed over time.

Some sample elements are: Rod elements, Beam elements, Plate/Shell/Composite elements, Shear panel, Solid elements, Spring elements, Mass elements, Rigid elements & Viscous damping elements. Many FEA programs also are equipped with the capability to use multiple materials within the structure such as: Isotropic - identical throughout, orthotropic - identical at 90 degrees, General anisotropic - different throughout.

## **2.5 Types of Engineering Analysis**

Structural analysis consists of linear and non-linear models. Linear models use simple parameters and assume that the material is not plastically deformed. Non-linear models consist of stressing the material past its elastic capabilities (into the plastic range). The stresses in the material then vary with the amount of deformation. Vibrational analysis is used to test a material against random vibrations, shock, and impact. Each of these incidences may act on the natural vibrational frequency of the material, which, in turn, may cause resonance and subsequent failure.

Fatigue analysis helps designers to predict the life of a material or structure by showing the effects of cyclic loading on the specimen. Such analysis can show the areas where crack propagation is most likely to occur. Failure due to fatigue may also show the damage tolerance of the material. Heat Transfer analysis models the conductivity or thermal fluid dynamics of the material or structure. This may consist of a steady state or transient transfer. Steady-state transfer refers to constant thermal properties in the material that yield linear heat diffusion.

The finite element method is applicable to a wide range of boundary value problems in engineering. In a boundary value problem, a solution is sought in the region of the body, while on the boundaries (or edges) of the region the values of the dependent variables (or their derivatives) are prescribed. The three major categories boundary value problem: equilibrium or steady state problem, Eigen value problem, and propagation or transient problems.

Since the majority of the applications of the method are in the realm of solid mechanics (including structural, soil, and rock mechanics), the descriptions in this volume are presented primarily in terms of this field of study. Problems in these fields usually tackled by one of three approaches: the displacement method, the equilibrium method, or the mixed method. Displacements are assumed as primary unknown quantities in the displacement method, stresses are assumed as primary unknown quantities in the equilibrium method; and some displacements and stresses are assumed as primary unknown quantities in the mixed method.

## **2.6 Stages involved in FEM**

### **2.6.1 Pre-Processor**

The pre-processor stage involves the following sections:

Specifying the title, that is the name of the problem. Setting the preferences, this is the type of filtering to be used, e.g. structural, fluid, thermal or electromagnetic. Defining the element type, this may be 2d or 3d in the structural element types, and there are many types then to choose from as mentioned in the introductory paragraph to the FEM method. This is possibly the most crucial part of an analysis if a highly accurate set of results is required.

Defining the material properties, i.e. the Young's modulus, Poisson's ratio, the density, and if applicable, the coefficients of expansion, friction, thermal conductivity, damping effect, specific heat etc.

Creating the model in appropriate dimensions. This is where the actual model is drawn in 2D or 3D space in the appropriate units.

Defining the mesh density. This may be done by manually defining the number of elements along the lines of the model, thus customizing the number of elements. In complex cases, specifying the element edge length may generate the mesh density, and hence the mesher meshes the model automatically on the command using the edge length specified.

### **2.6.2 Solution**

Here, the loading and boundary conditions are applied to the model. The boundary conditions are the second most critical stage of the analysis (element type is first). The boundary conditions usually are in the form of zero displacements on a structural model in either or all directions ( $x$ ,  $y$ ,  $z$ ). These zero displacements can be placed on nodes, key points, areas or on lines. The one on lines can be in the form of symmetric or anti-symmetric type boundary conditions, one allowing in plane rotations and out of plane translations, the other allowing in plane translations and out of plane rotations for a given line.

The loading may be in the form of a point load, a pressure or a displacement, again the values should be in the same units as the model drawn and the material properties given in the pre-processor section of the analysis. The solution of the problem is done automatically by executing the appropriate command. The package then proceeds to form the element-stiffness matrix for the problem, followed by solving for the matrix and then updating the displacement value for each node within the component or continuum.

### **2.6.3 Post-processor**

Here the results of the analysis can be read. They can be in the form of a table, a contour plot, deformed shape of the component or the mode shapes and natural frequencies if frequency analysis is involved.

From experience of the package, contour plots are found to be the most effective way of viewing the results for structural type problems, as tabular form only gives a reference in node ID number form which can be difficult to locate on the model. Also for the contour plots with a 3D model the plot on the other side of the model may be viewed simply by using pan, zoom, & rotate facilities. The contour plots can be completed for the displacements, stress or strains in the  $x$ ,  $y$  or  $z$  directions. The principal stresses and strains may also be plotted, or if required the yield stresses and strains according to the main theories of failure (von mises, St.Venant, Tresca etc.). Other information such as the

strain energy, plastic strain and creep strain may be obtained, which may otherwise not be available due to the complexity of the geometry, loading and/or boundary conditions.

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs, which would accrue if each sample was actually built and tested.

## 2.7 Pre-requisite for Finite Element Methods:

For the understanding of any subject, one needs to be familiar with the basics and pre-requisites of the particular subject, to understand and master the subject. Finite element methods has indeed needs the pre-requisite subjects like:

1. Fundamentals of Matrix Algebra.
2. Fundamentals of Solid Mechanics (for structural analysis).
3. Numerical methods.
4. Computer fundamentals.
5. Computer programming skills (for development).

## 2.8 Basics of Solid Mechanics

### Stress

The state of stress in an elemental volume of a loaded body is defined in terms of six components of stress, expressed in a vector form as:

$$\{\sigma\}^T = [\sigma_x \sigma_y \sigma_z \tau_{xy} \tau_{yx} \tau_{zx}]$$

Where  $\sigma_x$ ,  $\sigma_y$  and  $\sigma_z$  are the normal components of stress, and  $\tau_{xy}$ ,  $\tau_{yx}$ , and  $\tau_{zx}$  are the components of shear stress.

### Strain

The state of strain at a point can be divided into six strain components given by the following strain vector.

$$\{\epsilon\}^T = [\epsilon_x \epsilon_y \epsilon_z \gamma_{xy} \gamma_{yz} \gamma_{zx}]$$

Where  $\epsilon_x$ ,  $\epsilon_y$  and  $\epsilon_z$  are the normal components of strain, and  $\gamma_{xy}$ ,  $\gamma_{yz}$  and  $\gamma_{zx}$  are the components of shear strain.

### Strain-Displacement Equations

The relations between the components of strain and the displacement components  $u$ ,  $v$ , and  $w$  at a point are

$$\begin{aligned}\epsilon_x &= \frac{\partial u}{\partial x} + \frac{1}{2} \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right], \epsilon_y = \frac{\partial v}{\partial y} + \frac{1}{2} \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] \\ \epsilon_z &= \frac{\partial w}{\partial z} + \frac{1}{2} \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right], \gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} + \frac{\partial u \partial u}{\partial x \partial y} + \frac{\partial v \partial v}{\partial x \partial y} + \frac{\partial w \partial w}{\partial x \partial y} \\ \gamma_{yz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} + \frac{\partial u \partial u}{\partial y \partial z} + \frac{\partial v \partial v}{\partial y \partial z} + \frac{\partial w \partial w}{\partial y \partial z}, \gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} + \frac{\partial u \partial u}{\partial z \partial x} + \frac{\partial v \partial v}{\partial z \partial x} + \frac{\partial w \partial w}{\partial z \partial x}\end{aligned}$$

Equations are one version of the strain-displacement equations, in which the strain components are expressed in terms of only the first (linear) and the second order changes in the displacement components while the higher terms are neglected. The expressions for the strain components can be further simplified by retaining only the first order or linear terms and neglecting the second order terms, that is

$$\epsilon_x = \frac{\partial u}{\partial x}, \quad \epsilon_y = \frac{\partial v}{\partial y}, \quad \epsilon_z = \frac{\partial w}{\partial z}, \quad \gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}, \quad \gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}, \quad \gamma_{zx} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}$$

The simplest specialization of the generalized Hooke's law is the case in which the material is assumed to be linear, isotropic, and elastic. An isotropic material is one that has point symmetry; that is, every plane in the body is a plane of symmetry of material behavior.

It can be shown that only two independent elastic constants are necessary to represent the behavior in the case of such symmetry. Hence the strain equation in terms of  $E$  and  $\nu$  becomes

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} 1/E & -\nu/E & -\nu/E & 0 & 0 & 0 \\ & 1/E & -\nu/E & 0 & 0 & 0 \\ & & 1/E & 0 & 0 & 0 \\ & & & \frac{2(1+\nu)}{E} & 0 & 0 \\ & & & & \frac{2(1+\nu)}{E} & 0 \\ & & & & & \frac{2(1+\nu)}{E} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}$$

*symm*

Or in terms of stress components, the equation becomes

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ & 1-\nu & \nu & 0 & 0 & 0 \\ & & 1-\nu & 0 & 0 & 0 \\ & & & \frac{1-2\nu}{2} & 0 & 0 \\ & & & & \frac{1-2\nu}{2} & 0 \\ & & & & & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$$

*symm*

### Plane Strain

Problems involving a long body whose geometry and loading do not vary significantly in the longitudinal direction are referred to as plane strain problems. Some examples of this configurations are a loaded semi-infinite half-space such as footing on a soil mass; a long cylinder such as a tunnel, culvert, or buried pipe, a laterally loaded retaining wall, and a long earth dam. In these problems the dependent variables can be assumed to be functions of only the  $x$  and  $y$  co-ordinates, provided we consider a cross section some distance away from the ends. If we further assume that  $w$ , the displacement component in the  $z$  direction, is zero at every cross section, the strain components  $\varepsilon_z$ ,  $\gamma_{yz}$  and  $\gamma_{zx}$  will vanish and the remaining non-zero components will be

$$\varepsilon_x = \frac{\partial u}{\partial x}, \quad \varepsilon_y = \frac{\partial v}{\partial y} \quad \text{and} \quad \gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$$

Moreover, from the vanishing of  $\varepsilon_z$ , the stress  $\sigma_z$ , can be expressed in terms of  $\sigma_x$  and  $\sigma_y$  as

$$\sigma_z = \nu(\sigma_x + \sigma_y) \quad \text{and} \quad \sigma_x, \sigma_y \quad \text{and} \quad \tau_{xy} \quad \text{are thus the only dependent stress variables.}$$

The constitutive law for elastic isotropic material, the equation reduces to,

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}.$$

In plane strain problems we usually consider a slice of unit thickness.

### Plane Stress

In contrast to the plane strain condition, very small dimensions in the z-direction characterize the plane stress condition. A thin plate loaded in its plane is the well-known example of the plane stress approximation. We consider the case where no loadings are applied on the surfaces of the plate. Then the stress components  $\tau_{yz}$  and  $\tau_{zx}$  vanish on the surfaces, and  $\sigma_z$  is zero throughout the thickness.

The nonzero components  $\sigma_x, \sigma_y$  and  $\tau_{xy}$  may be averaged over the thickness and assumed to be independent of z. The state of stress characterized by the above description is referred to as generalized plane stress. The strain components  $\gamma_{yz}$  and  $\gamma_{zx}$  vanish on the surfaces, while the component  $\epsilon_z$  is given by:

$$\epsilon_z = -\frac{\nu}{1-\nu}(\epsilon_x + \epsilon_y)$$

The strain-displacement equations will be the same as in the plane strain case, the constitutive relation becomes

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}$$

### Axisymmetric problems

Many engineering problems involve solids of revolution (Axisymmetric solids) subjected to axially symmetric loading. Examples of this situation are a circular cylinder loaded by uniform internal or external pressure or other axially symmetric loading, and a semi-infinite half space loaded by a circular area, for example a circular footing on a soil mass.

It is convenient to express these problems in terms of the cylindrical co-ordinates. Because of symmetry, the stress components are independent of angular ( $\theta$ ) co-ordinate; hence, all derivatives with respect to  $\theta$  vanish and the components  $\nu$ ,  $\gamma_{r\theta}$ ,  $\gamma_{\theta z}$ ,  $\tau_{r\theta}$ , and  $\tau_{\theta z}$  are zero. The nonzero stress components are  $\sigma_r$ ,  $\sigma_\theta$ ,  $\sigma_z$ , and  $\tau_{rz}$ .

The strain-displacement relations for the nonzero strain becomes

$$\epsilon_r = \frac{\partial u}{\partial r}, \epsilon_\theta = \frac{u}{r}, \epsilon_z = \frac{\partial w}{\partial z}, \gamma_{rz} = \frac{\partial u}{\partial z} + \frac{\partial w}{\partial r}$$

The constitutive relation is

$$\begin{Bmatrix} \sigma_r \\ \sigma_z \\ \sigma_\theta \\ \tau_{rz} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ & 1-\nu & \nu & 0 \\ & & 1-\nu & 0 \\ & \text{symm} & & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \epsilon_r \\ \epsilon_z \\ \epsilon_\theta \\ \gamma_{rz} \end{Bmatrix}$$

## 2.9. FEATURES AVAILABLE IN ANSYS

### 2.9.1. Introduction

The ANSYS program is a general purpose computer program for finite element analysis and design. General-purpose refers to the fact that the program can be used in all disciplines of engineering-structural, mechanical, electrical, electromagnetic, electronic, thermal, fluid, and biomedical. One can use the program to find out how a given design works under operating conditions. One can also use the ANSYS program to calculate the proper design for given operating conditions.

### 2.9.2. List of industries using ANSYS

Following are the industries using ANSYS.

- Automobiles
- Aerospace
- Bridges & Buildings

- Machinery
- Electronics & Appliances
- Sporting goods
- Heavy equipment & Machinery
- MEMS - Micro-Electromechanical systems

### **2.9.3. A Typical Analysis**

The ANSYS program has many finite element analysis capabilities ranging from a simple, linear static analysis to a complex, nonlinear transient dynamic analysis. Each analysis involves the following.

- Build the model
- Apply loads and obtain the solution
- Review the result

### **Structural analysis**

Many types of structural analysis are available in the ANSYS program. The primary unknowns (nodal degree of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements.

The following types of structural analyses are possible:

- Static Analysis
- Modal Analysis
- Harmonic analysis
- Transient dynamic analysis
- Spectrum analysis
- Buckling analysis

In addition to the above analysis types, several special-purpose features are available, such as fracture mechanics, composites and fatigue.

## **Thermal analysis**

Thermal analyses are used to calculate the temperature distribution and related thermal quantities in an object. The ANSYS program uses a heat balance equation obtained from the principle of conservation of energy as the basis for thermal analysis. The ANSYS program handles all three primary modes of heat transfer-conduction, convection, and radiation.

## **Magnetic field analysis**

Magnetic analyses are used to calculate the magnetic field in devices such as power generators, transformers, video display devices and so forth. The ANSYS program uses Maxwell's equations as the basis for magnetic field analysis. The three types of magnetic analyses are possible:

- Static magnetic analysis
- Harmonic magnetic analysis
- Transient magnetic analysis

## **Electric field analysis**

Electric field analyses are used to calculate the electric field in conductive or capacitive systems. The ANSYS program uses Laplace equation as the basis for static electric field analysis.

## **Fluid analysis**

A "fluid analysis" in the ANSYS program may mean any of the following capabilities:

- Fluid flow with heat transfer
- Acoustics
- Contained fluid

## **Chapter 3**

### **FINITE ELEMENT ANALYSIS**

#### **3.1. Introduction**

By using ANSYS, the 2D FEM Modeling of the building frame has been developed. The building frame has 3 bays and 3 floors. Each floor has 3m height and each bay has 3m width. For this study, two types building frames are developed. First one has super structure alone and the second one has super structure-soil- and pile system. The depth of the pile foundation is taken as 6m.

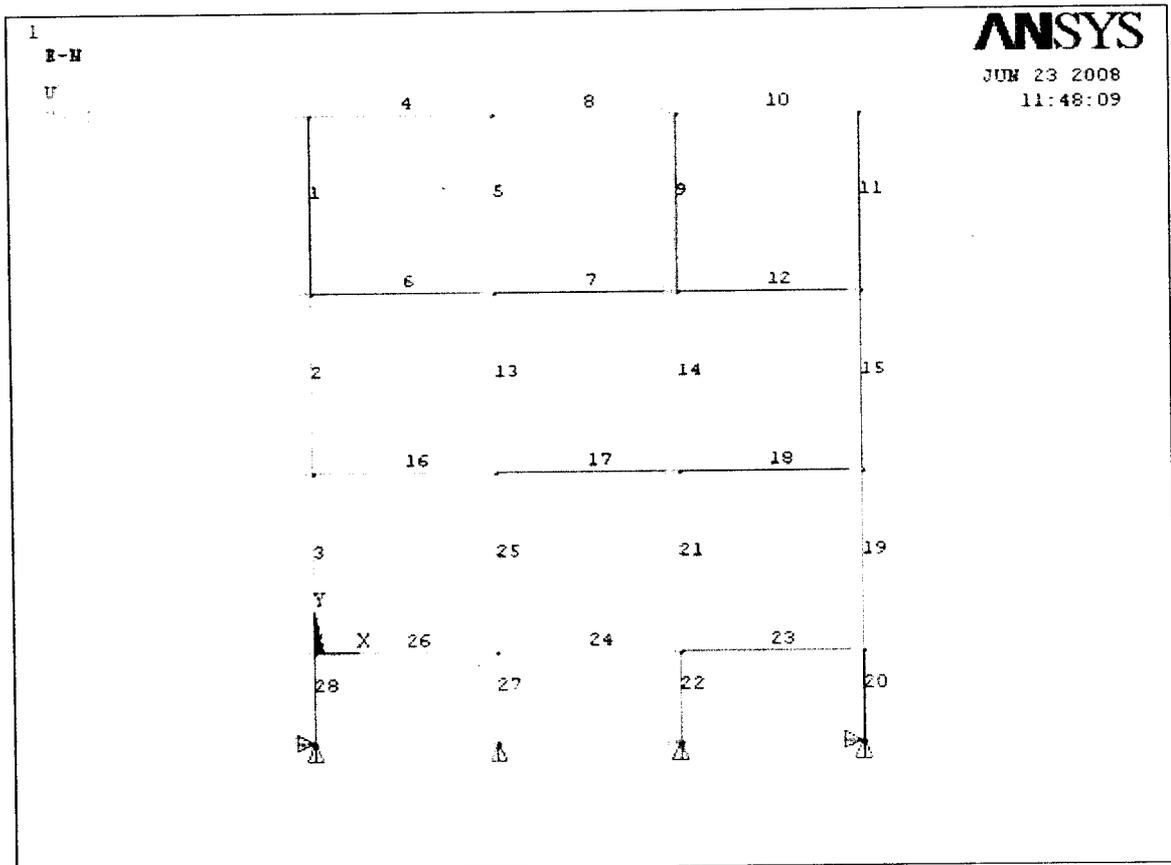
#### **3.2. Numerical modeling**

##### **3.2.1. Frame configuration**

No of bays	-	3
No of piles in a plane frame	-	7
Bay width	-	3m
No of storeys	-	3
Storey height	-	3m
Size of beam element	-	0.23 x 0.23m
Size of pile	-	0.23m dia.
Depth of pile	-	6m

##### **3.2.2. Superstructure alone**

Superstructure has been modeled as beam column elements and the piles have been modeled as beam elements. The soil has been included by means of their parameters namely stiffness and damping.



**Fig 3.1.Super structure elements**

In The above fig. the superstructure alone has been modeled as beam elements and the sizes of the beams are taken as 230mmx230mm.in modeling, the beam, columns are represented as a single line but all their properties like young's modulus, density, poisson's ratio etc... are included. Bottom of the frame is assumed that it is well fixed by means of raft foundation. For this 2D analysis, it is assumed that the building resting on rollers. Seismic time dependent displacements are applied at the bottom end of the frame. The displacement variation with respect to time is assumed to be linearly increasing up to maximum amplitude and then linearly decreasing. The amplitude and frequency are varied and analyzed.

### 3.2.3. Superstructure-soil-pile system

For this 2D modeling, vertical and horizontal line elements are introduced at the bottom of the super structure and the elements are considered as soil element by assigning the young's modulus value and all other soil properties to the line element. The pile elements are sub divided in to 3 nodes. The stiffness and damping properties of the pile has been calculated by using the NOVAK approach.

#### 3.2.3. (a) Horizontal stiffness and damping co efficient for single pile

**NOVAK solution:**

$$\text{Horizontal stiffness, } K_{U1} = \left( \frac{E_p \cdot I_p}{r^3} \right) f_{u1} \left[ \because \frac{l}{r} \geq 25 \right]$$

$I_p$  = Moment of inertia of pile C.S.

$$= \frac{\pi r^4}{4} = \frac{\pi (0.15)^4}{4} = 0.000397 \text{ m}^4$$

$f_{u1}$  = Stiffness factor in horizontal direction  
= f (l/r,  $V_s$ ,  $E_p/G_s$ )

$\mu_s$  = Poisson's ratio as 0.4

$E_p$  = Young's Modulus of pile Material  
=  $25 \times 10^6$  kN/m<sup>2</sup>

$G_s$  = Shear Modulus

$$= 17420 \text{ kN/m}^2 \text{ [obtained from wave propagation Test]}$$

$$\mu_s = 0.4$$

$$\frac{E_p}{G_s} = \frac{25 \times 10^6}{17420} = 1435$$

$$\frac{l}{r} = \frac{6}{0.15} = 40$$

∴  $f_{U1}$  = from Table 7 [Fixed head piles]

$$= 0.020 \left[ \text{Based on } \frac{E_p}{G_s} \right]$$

$$K_{U1} = \frac{E_p I_p}{r^3} \cdot f_{U1} \text{ For } \frac{\ell}{r} \geq 25$$

$$\therefore K_{U1} = \frac{25 \times 10^6 \times 0.000397}{(0.15)^3} \times 0.020$$

**Horizontal stiffness,  $K_{U1} = 58903.7$  kN/m**

**Horizontal damping for single pile:**

Damping in horizontal Mode for single pile:

$$C_{u1} = \left[ \frac{E_p I_p}{r^2 V_s} \right] \cdot f_{u2}$$

Where  $f_{u2}$  = Damping factor [from Table no:7]

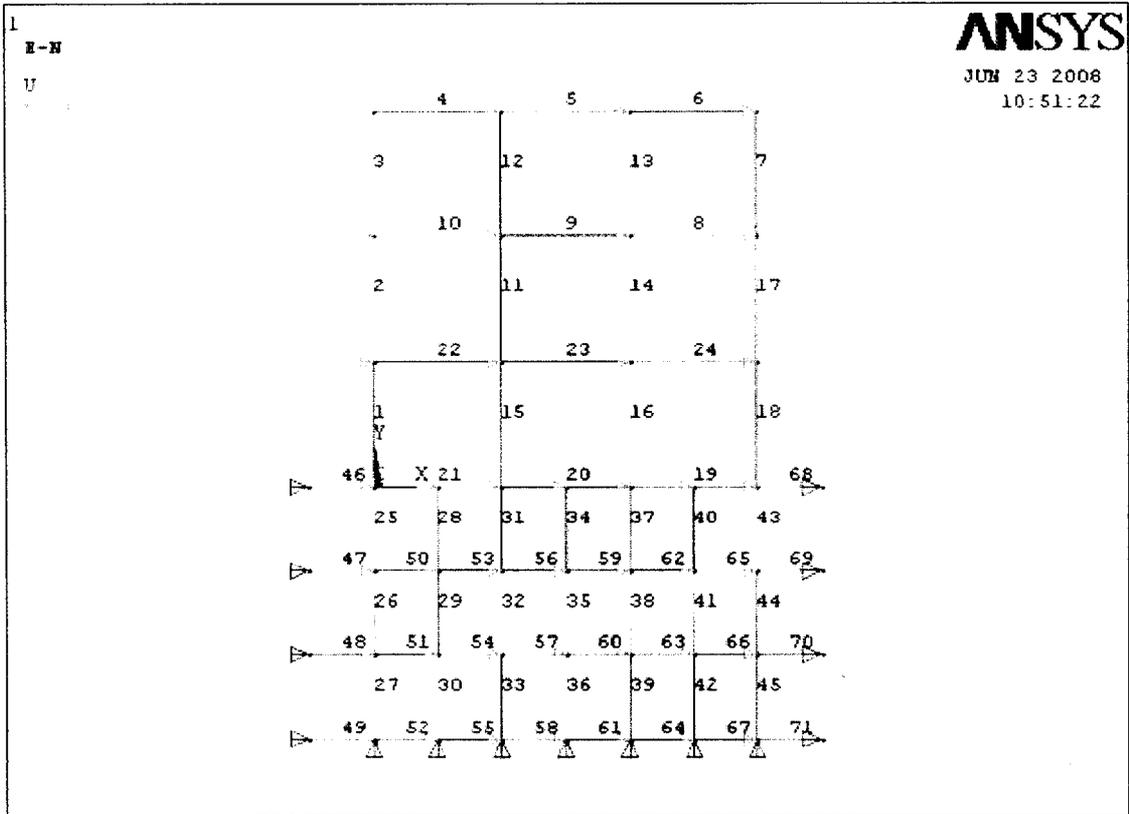
$$f_{u2} = 0.0490 \text{ for } l/r \geq 25$$

$$\mu_s = 0.4$$

$$\frac{E_p}{G_s} = 1435$$

$$C_{U1} = \frac{25 \times 10^6 \times 0.000397}{(0.15)^2 \times 95} \times 0.0490$$

**Horizontal damping,  $C_{u1} = 227.86$  kN-s/m**

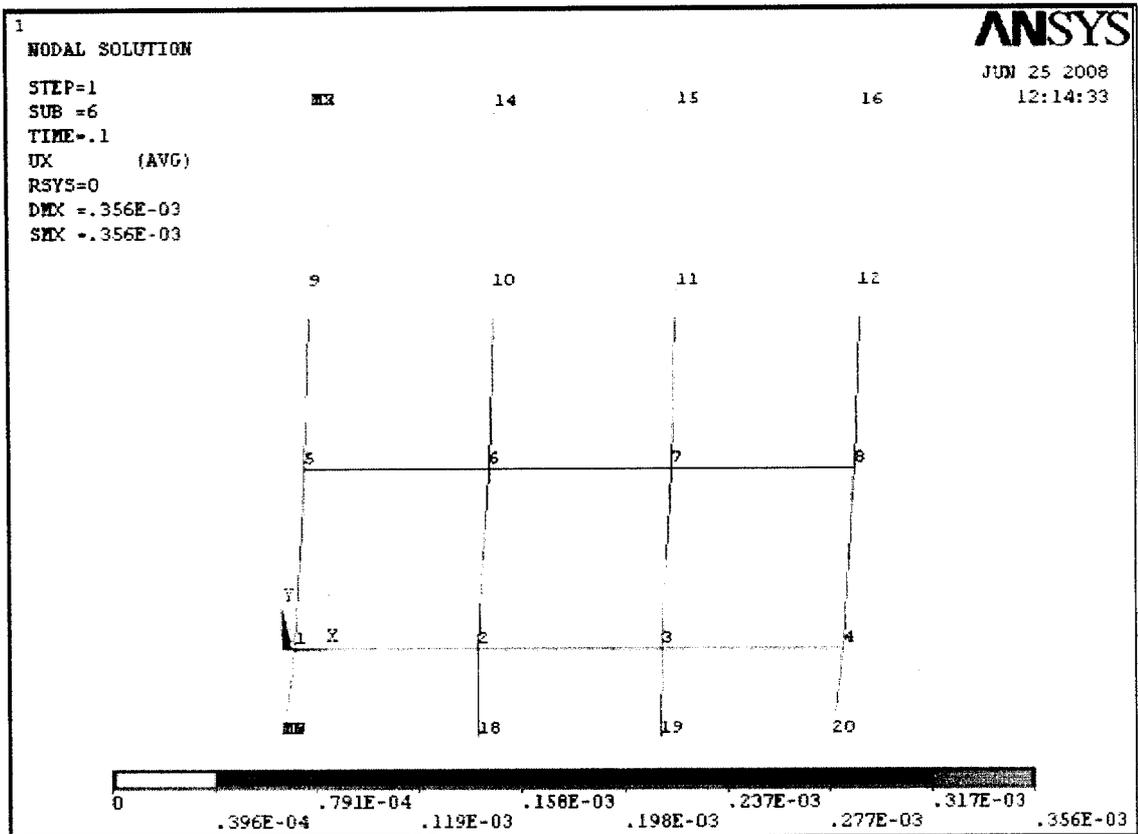


**Fig 3.2.super structure-soil-pile system elements**

The stiffness and damping is applied at the pile nodes. Seismic displacement is applied at the bottom of the end pile.

### 3.2.4. Deformed shapes of the models

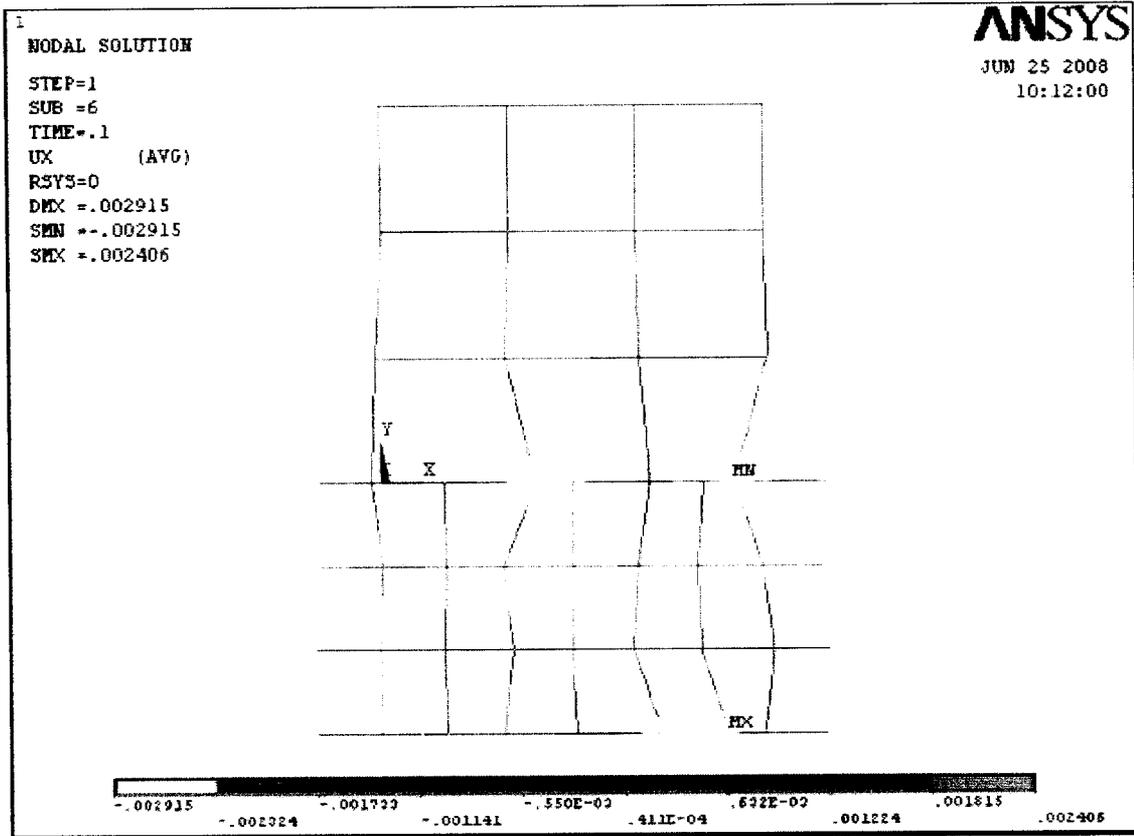
#### 3.2.4 (i) Super structure without soil-pile system



**Fig 3.3.super structure without soil-pile system**

The above fig.3.3. Shows the deformation of the superstructure for the displacement 0.025m at time 0.5 sec.

### 3.2.4 (ii) Super structure with soil-pile system



**Fig 3.4.super structure with soil-pile system**

The above fig.3.5. Shows the deformation of the superstructure-soil-and pile system for the displacement 0.025m at the 0.5 sec.

### 3.2.5. Finite element meshing

After the modeling work completed the finite element mesh is used to carry the simulation includes 20 nodes, 28 elements in superstructure and 48 nodes, 71 elements in superstructure-soil-and pile system. Transient analysis is performed at the bottom level of both the models.

### 3.2.6. Boundary conditions

In the superstructure alone, the bottom nodes of the beam elements are assumed as resting on roller support. The far end of the element is assumed as resting on a hinged support.

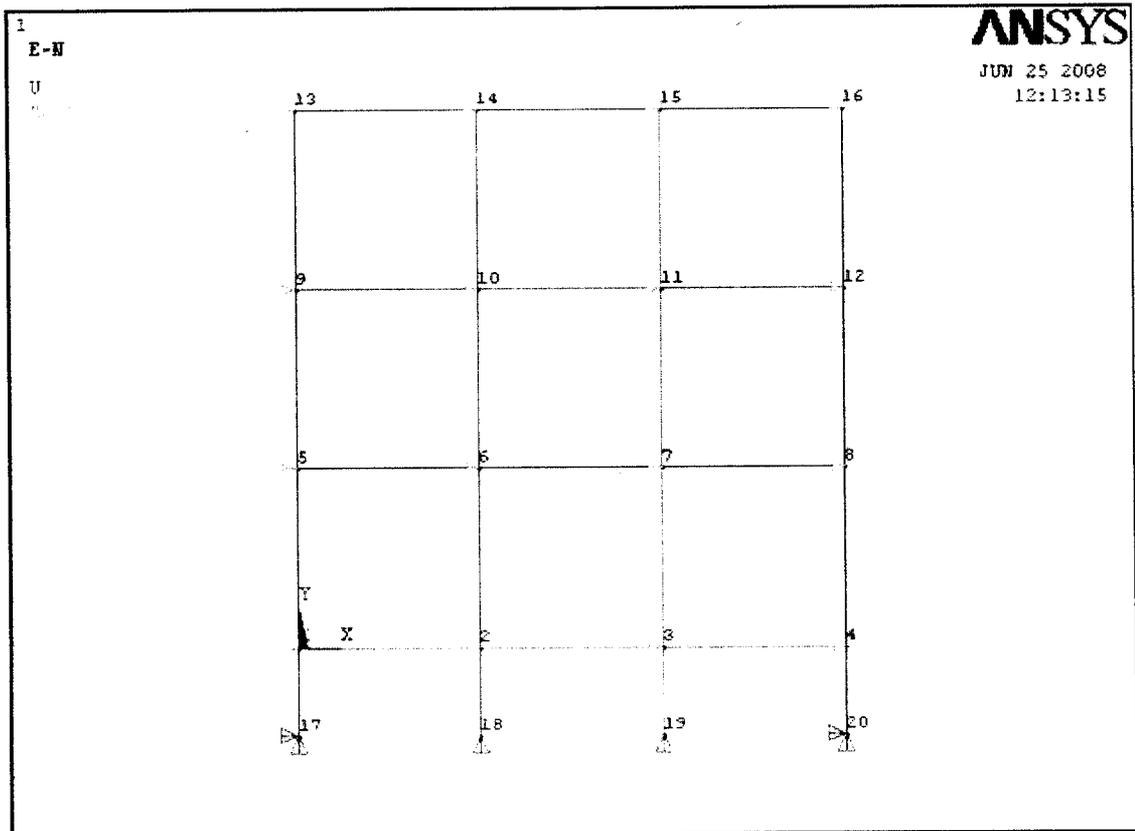
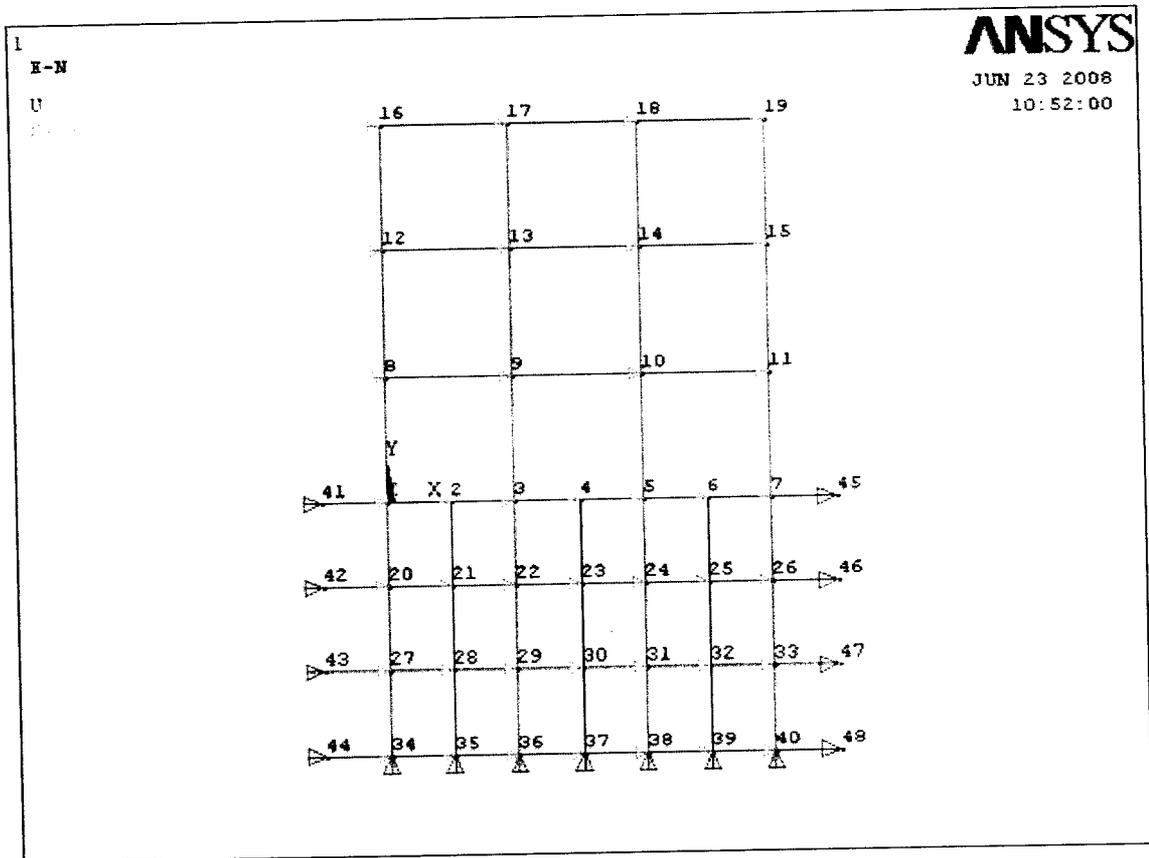


Fig 3.5. Boundary conditions for super structure with out soil-pile system

For the superstructure without pile –soil system also assumed that the bottom nodes all resting on the roller support. And the horizontal dampers are arrested in X-direction of motion because of the soil infinity.



**Fig 3.6. Boundary conditions for super structure with soil-pile system**

### **3.2.7. Material properties**

The soil is assumed it is homogenous and the properties taken for the analysis are young's modulus, ( $E_s$ ) of  $2.23 \times 10^7 \text{ N/m}^2$ , Poisson's ratio of 0.3, and density of  $19000 \text{ N/m}^3$ . The Horizontal stiffness and damping coefficient have been calculated from NOVAK'S approach. The Horizontal stiffness of the single pile foundation is  $5.890370 \times 10^7 \text{ N/m}$  and the damping force is  $2.27 \times 10^5 \text{ N-s/m}$ . The concrete properties taken are young's modulus, ( $E_c$ )  $2.23 \times 10^{10} \text{ N/m}^2$ , Poisson's ratio of 0.15 and density of  $25000 \text{ N/m}^3$ . The density of the pile element is  $2.44 \times 10^6 \text{ N/m}^3$ . The density of the horizontal beam element is  $4.361 \times 10^6 \text{ N/m}^3$ .

### **3.2.8. Seismic loading**

The amplitude is set as 0.025m and the duration is taken as 0.1 sec and time interval is 0.001 sec. The time is divided by 16. The seismic displacement is given for the every  $1/16^{\text{th}}$  time. This displacement with respect to time is applied at the bottom of the structure for the analysis of with out pile and bottom of the pile for the analysis of with pile foundation. The transient analysis is carried out.

### **3.3. Results**

#### **3.3.1. General**

The analysis is carried out using ANSYS finite element program. The models are subjected to the same amplitude. The results are plotted from the numerical modeling are the roof level lateral displacement of each floor, stress occurring at the level. The results are discussed in the following sections.

#### **3.3.2. Lateral Displacements**

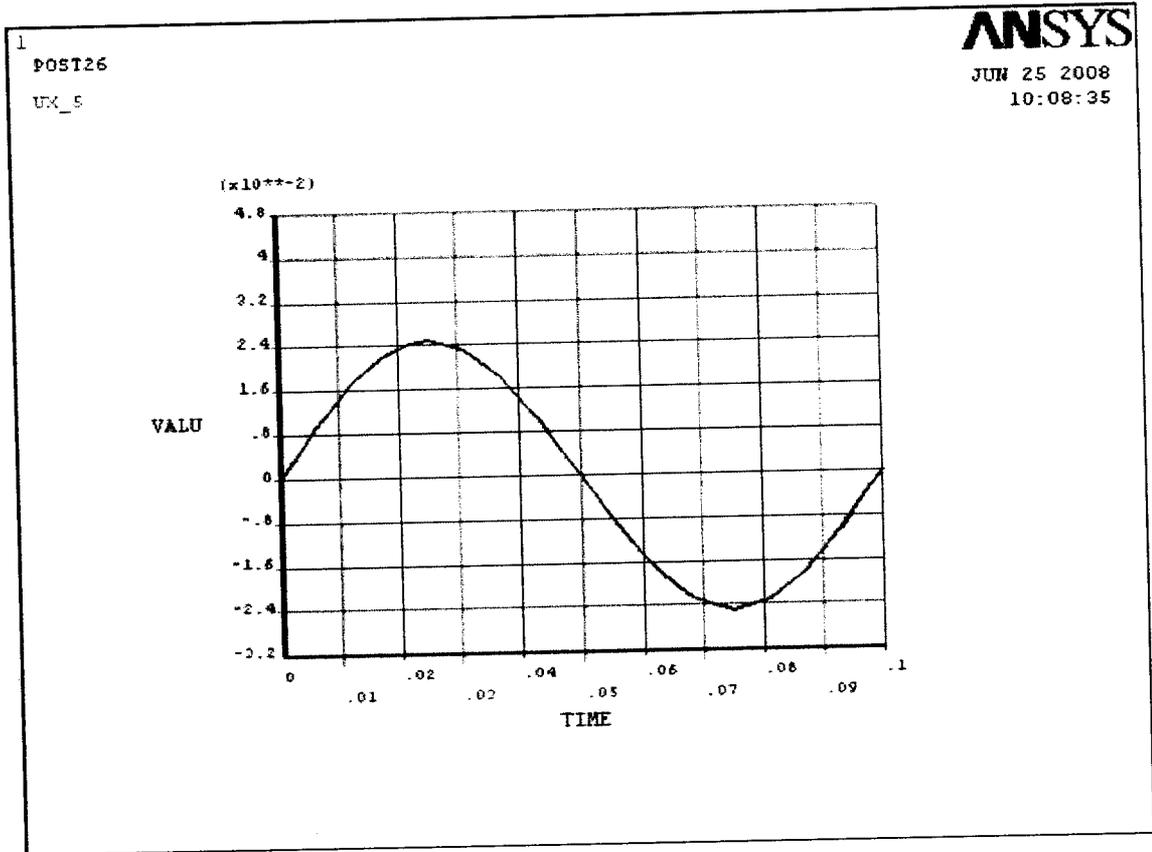
The displacement patterns for the superstructure are plotted in three locations one at top floor, mid floor and another at ground floor roof levels. The displacement patterns are plotted with respect to the time.

Fig. 3.7.shows the displacement pattern for the initial displacements with respect to the time for this 2D analysis.

Fig. 3.8.shows the displacement pattern of the roof level at each floor for the superstructure without considering the soil-pile effect in 2D analysis for the displacements with respect to the time applied at the level of bottom of the structure.

Fig.3.9.shows the displacement pattern of the roof level at each floor for the superstructure with considering the pile and soil effect in 2D analysis for the for the displacements with respect to the time applied at the level of bottom of the pile.

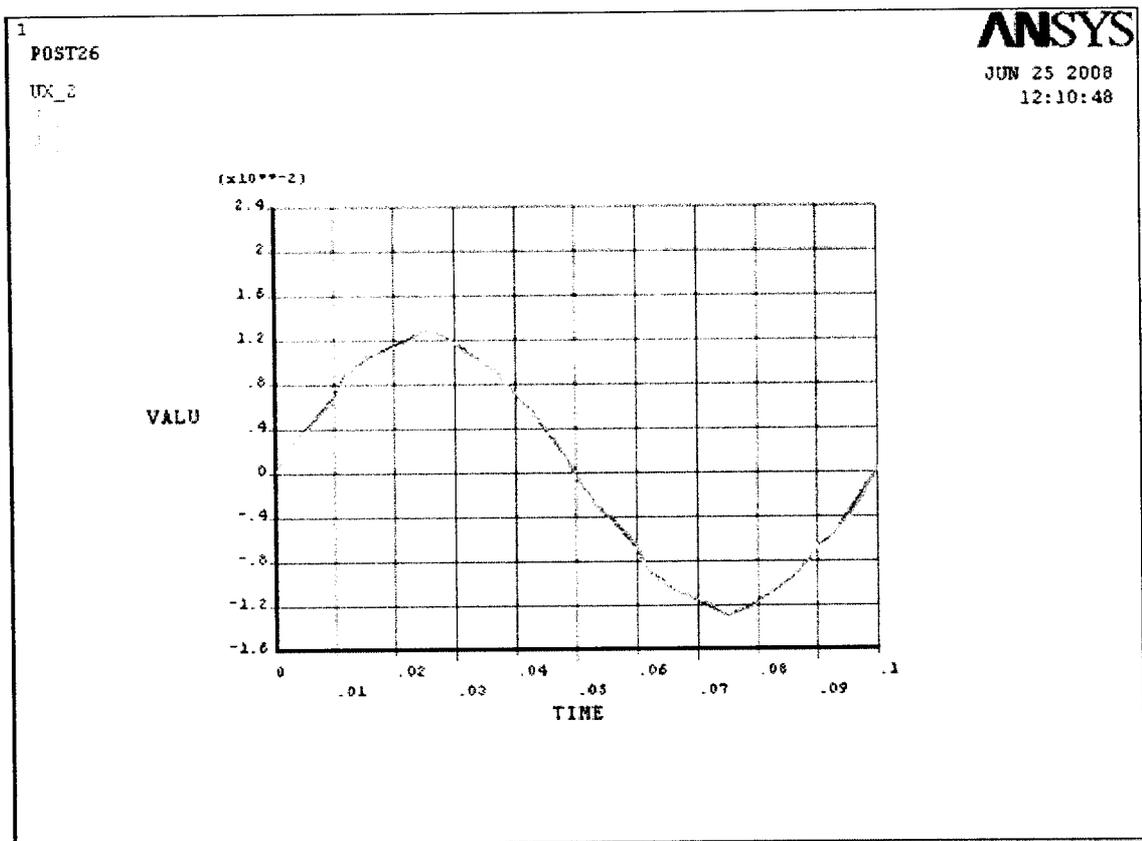
## Displacement pattern for the initial displacement



**Fig 3.7.Initial displacement pattern curve**

The above fig. Shows the initial ground displacement for the given amplitude 0.025m for 10cycles per sec.

### Displacement pattern of superstructure with out pile-soil system

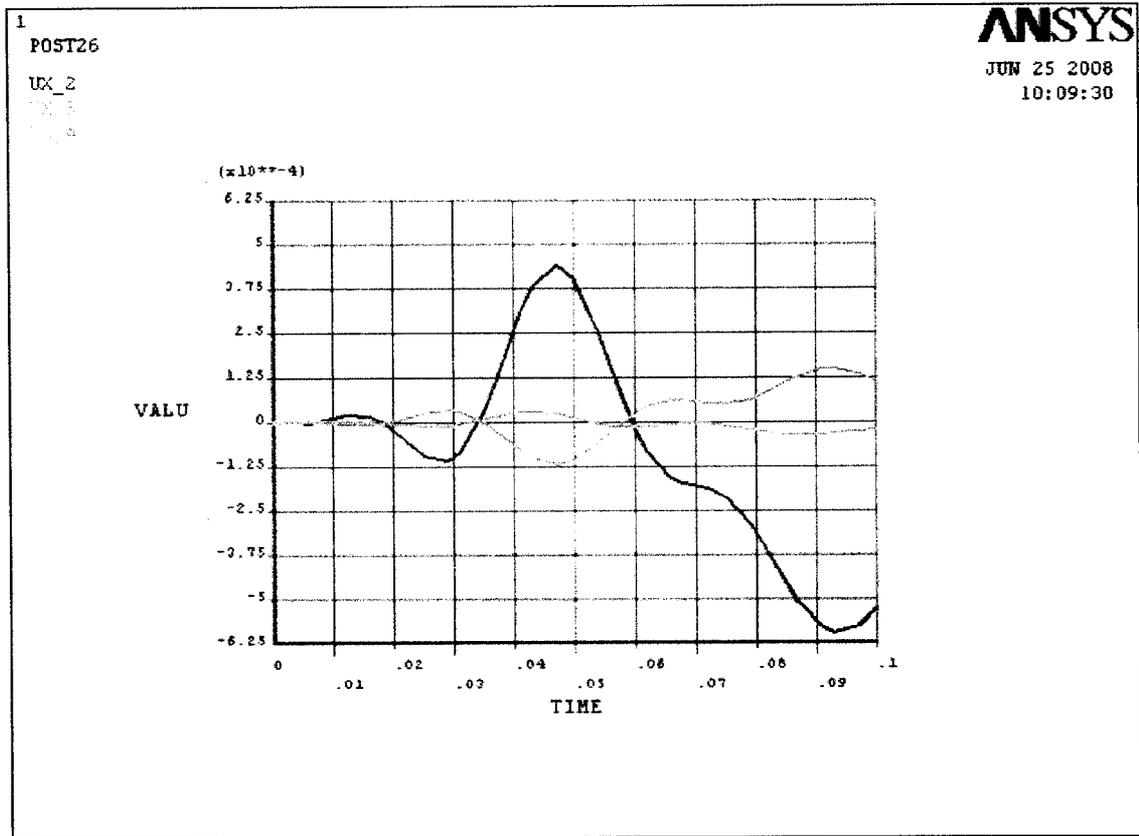


**Fig 3.8. displacement pattern of super structure without soil-pile system**

The above fig shows the displacement positions at the each roof level of the superstructure with out considering the soil-pile system for the given amplitude with respect to the time. This graph indicates the deformation of the superstructure for ten cycles per sec.

The maximum displacement occurring at top floor goes to  $1.3 \times 10^{-2}$  m at 0.025 sec.

### Displacement pattern of superstructure with pile-soil system



**Fig 3.9. displacement pattern of super structure with soil-pile system**

The above fig shows the displacement positions at the each roof level of the superstructure with considering the soil-pile system for the given amplitude with respect to the time. This graph indicates the deformation of the superstructure for 10 cycles per sec.

The maximum displacement occurring at top floor goes to  $5.75 \times 10^{-4}$  m at 0.045 sec.

### 3.4. Stresses at the elements

Stresses in X-direction are at the superstructure are measured in three locations one at top floor, mid floor and another at ground floor roof levels.

#### 3.4.1. Stresses at all the elements in superstructure without soil-pile system

<b>ELEMENTS</b>	<b>LS1 N/m2</b>	<b>LS4 N/m2</b>
1	53897	53897
2	-82817	-82817
3	-21499	-21499
4	-74004	-74004
5	-4205.0	-4205.0
6	-3406.2	-3406.2
7	-68632	-68632
8	4818.8	4818.8
9	26407	26407
10	29220	29220
11	0.28245E+06	0.28245E+06
12	-0.21625E+06	-0.21625E+06
13	24978	24978
14	-5527.1	-5527.1
16	1078.4	1078.4
17	1503.2	1503.2
18	619.23	619.23
19	28619	28619
20	-0.12009E+06	-0.12009E+06

21	36450	36450
22	24741	24741
23	40837	40837
24	-0.24776E+06	-0.24776E+06
25	-0.17719E+06	-0.17719E+06
26	-0.20872E+06	-0.20872E+06
27	0.71614E+06	0.71614E+06
28	0.74245E+06	0.74245E+06
29	0.84730E+06	0.84730E+06

**Table 3.1. Stresses at all the elements in superstructure without soil pile system**

**3.4.2. Minimum and maximum stress value in superstructure without pile system**

	<b>ELEMENT</b>	<b>VALUE</b>
Minimum value	23	-0.24776E+06N/m <sup>2</sup>
Maximum values	29	0.84730E+06 N/m <sup>2</sup>

**Table 3.2. Minimum and maximum stress value in superstructure without soil pile system**

**3.4.3. The stress at the roof levels**

<b>ELEMENTS</b>	<b>LS1 N/m<sup>2</sup></b>	<b>LS4 N/m<sup>2</sup></b>
1	53897	53897
2	-82817	-82817
3	-21499	-21499
4	-7400.4	-7400.4
5	-4205.0	-4205.0
6	-3406.2	-3406.2
7	-6863.2	-6863.2

8	4818.8	4818.8
9	26407	26407

**Table 3.3. Stresses at the roof levels in superstructure without soil pile system**

**Stress at the top roof = 53897 N/m<sup>2</sup> at element 1**

**3.4.4. Stress at all the elements in the superstructure with pile system**

<b>ELEMENTS</b>	<b>LS1 N/m2</b>	<b>LS4 N/m2</b>
1	3711.9	3711.9
2	3694.5	3694.5
3	-3077.0	-3077.0
4	3288.2	3288.2
5	1796.0	1796.0
6	-2944.8	-2944.8
7	3348.9	3348.9
8	-1268.4	-1268.4
9	5387.3	5387.3
10	-2925..2	-2925..2
11	7193.5	7193.5
12	-21348.	-21348.
13	0.14707E+04	0.14707E+04
14	54845.	54845.
15	-8633.8	-8633.8
16	-0.47165E+04	-0.47165E+04
17	-0.90339E+04	-0.90339E+04
18	0.47969E+04	0.47969E+04
19	0.15205E+04	0.15205E+04

20	0.43258E+05	0.43258E+05
21	-0.11430E+03	-0.11430E+03
22	0.24010E+05	0.24010E+05
23	-0.94031E+03	-0.94031E+03
24	-0.29896E+04	-0.29896E+04
25	9007.2	9007.2
26	9007.2	9007.2
27	9007.2	9007.2
28	0.0000	0.0000
29	0.0000	0.0000
30	0.0000	0.0000
31	3956.3	3956.3
32	3956.3	3956.3
33	3956.3	3956.3
34	0.0000	0.0000
35	0.0000	0.0000
36	0.0000	0.0000
37	-16023.	-16023.
38	-16023.	-16023.
39	-16023.	-16023.
40	0.0000	0.0000

**Table 3.4. Stress at all the elements in the superstructure with pile system**

**3.4.5. Minimum and maximum stress value**

	<b>ELEM</b>	<b>VALUE</b>
Minimum values	16	-0.417583E+04
Maximum values	22	0.24540E+05

**Table 3.5. Minimum and maximum stress value**

### 3.5.2. CONCLUSIONS

Based on the detailed discussions made in the previous chapters the following salient conclusions can be derived.

#### 3.5.2. (i) Comparison of results

Roof level	Stress at roof levels in N/m <sup>2</sup>		Maximum displacement at the roof levels in m	
	Without piles	With piles	Without piles at 0.025 sec	With piles at 0.025 sec
Ground	53897	3711.9	$1.32 \times 10^{-2}$	$0.75 \times 10^{-5}$
First	-74004	3288.2	$1.97 \times 10^{-2}$	$1.62 \times 10^{-4}$
Second	-68632	3348.9	$1.84 \times 10^{-2}$	$5.62 \times 10^{-4}$

**Table 3.7. Comparison of results**

- Inclusion of soil pile system with the superstructure in general decreases the lateral deformation of the superstructure under the seismic base motion.
- Which also reduces the horizontal stress at each element
- The ratio of the displacement for the superstructure without and with pile-soil system is 25:1.
- The ratio of the stresses for the superstructure without and with pile-soil system is 14:1.
- Provision of piles may reduce the effect of the seismic ground motion on superstructure to a large extent.
- NOVAK approach used in the present analysis to be a very simple way of performing seismic analysis in the case of structures founded on pile foundation. If the soil and pile are to be discretized it will increase the computation by very large extent.

### **3.5.3. SCOPE FOR FURTHER WORK**

- The effect of soil and pile on superstructure under seismic load in the present study is based on the Novak's approach. Transient 3D analysis may be carried out for the superstructure with and without pile soil system by discretizing the soil and beam elements as solid members in finite element method. This may be compared with Novak's approach.
- This study is performed by only using transient analysis. Other type of analysis with different type of frequencies and amplitudes may be carried out.
- The possibility of provision of piles as an alternate method aseismic design may be explored.
- The effect of pile geometry on the effectiveness of provision of pile in mitigating Earthquake effect on superstructure may be studied.

## REFERENCES

- Advance simulation tools soil structure interaction - MARY GRONDIN
- Horizontal stiffness and damping Coefficients for single piles -Novak
- Interactive behaviour of soil-pile-superstructure system in transient state to liquefaction by means of large shake table tests - SHINTARO YAO, KOICHI KOBAYASHI
- Seismic analysis of infinite pile groups in liquefiable soil - ASSAF KLAR, SAM FRYDMAN, RAFFAEL BAKER
- Analysis of pile soil interaction – YASSER A.KHODAIR, SOPHIA HASIOTIS
- Structural dynamics – Anil Chopra

# MAHENDRA ENGINEERING COLLEGE

Mahendhirapuram, Villasapattanam (W) Vadugapalayam (P.O.),  
Tiruchchirappalli, Tamil Nadu - 637 503



An ISO 9001:2000 Certified Institution

Department of Civil Engineering

## NATIONAL CONFERENCE ON RECENT TRENDS IN CIVIL ENGINEERING

# NCRTE '08

## CERTIFICATE

This is to certify that Mr./ Ms. C. JAYAKUMAR of \_\_\_\_\_

KUMARAGURU COLLEGE OF TECHNOLOGY presented paper entitled STUDY ON THE PERFORMANCE OF PILED FOUNDATION WHEN SUBJECTED TO EQU NCRTE '08 held at Mahendra Engineering College

on 24 March 2008.

Convenor

N. N. S. M.  
Chairman