NUMERICAL ANALYSIS OF A PILE SUBJECTED TO LATERAL LOADS

Mohammed Younus Ahmed
Graduate Student, Earthquake Engineering Research Center, IIIT Hyderabad, Gachibowli, Hyderabad–32, India.
E-mail: younus.mohammedpg08@students.iiit.ac.in

Satyam D. Neelima
Assistant Professor, Earthquake Engineering Research Center, IIIT Hyderabad, Gachibowli, Hyderabad–32, India.
E-mail: neelima.satyam@iiit.ac.in

ABSTRACT: In the case of foundations of bridges, transmission towers, offshore structures and for other type of huge structures, piles are also subjected to lateral loads. This lateral load resistance of pile foundations is critically important in the design of structures under dynamic loading. Load carrying capacity and load deformation behavior of a single pile and group of piles subjected to lateral load is obtained using nonlinear finite element method of analysis. According to Poulos & Davis (1980) the maximum deflection of the pile is the major criterion in its design and made the initial attempts to study the lateral behavior of piles included two-dimensional finite element models in the horizontal plane. Several investigations have attempted to study the behavior of pile under lateral load using 3D finite element analysis. In this paper a detailed study on the piles subjected to lateral loads are investigated.

1. INTRODUCTION

Pile foundations are used to support the heavy structure and can act in the dual role of carrying the applied load to deeper, strong layers, and also of reinforcing the soil. In the case of foundations of bridges, transmission towers, offshore structures and other type of huge structures, piles are also subjected to lateral loads. This lateral load resistance of pile foundations is critically important in the design of structures under loading from earthquakes, soil movement, waves etc. According to Poulos & Davis (1980) the maximum deflection of the pile is the major criterion on the design.

Three-dimensional linear and nonlinear finite element analyses are gradually becoming commonplace for geotechnical applications. With the Available computational power continues to allow for such more realistic representation of the actual involved geometric scenarios. In this paper, nonlinear three dimensional finite element analysis are carried out to investigate the behavior of single pile subjected to laterally applied load.

2. NUMERICAL MODEL

Finite element analyses were performed using the nonlinear three dimensional analysis code. According to finite element method a continuum was divided into a number of (volume) elements. Each element consists of number of nodes. Each node has a number of degrees of freedom that correspond to discrete values of unknowns in the boundary value problem to be solved.

2.1 Mesh Generation

In order to perform the finite element calculations, the geometry have be divided into elements. A composition of finite elements is called a finite element mesh. The basic soil elements of a three dimensional finite element mesh are represented by 20 noded brick elements. The dimensions of the element size are user defined given by the user. A model mesh is shown in Figure 1. According to (Karthigeyan et al. 2007), the soil mass dimension depends on the pile diameter and length. The width of soil mass is taken as 40B, in which, B is the pile diameter or pile width. The soil mass effect on the pile response is diminishing for the width more than 40B. The height of soil mass is L+20B, in which, L is the length of pile.

Fig. 1: Finite Element Model of Soil (6 m × 6 m × 16 m) with a Square Pile
Numerical Analysis of a Pile Subjected to Lateral Loads

2.2 Linear Elastic Model of Pile

This model represents Hooke's law of isotropic linear elasticity used for modeling the stress-strain relationship of the pile material. The model involves two elastic stiffness parameters, namely Young's modulus, \( E \), and Poisson's ratio, \( \nu \) as shown in Figure 2.

2.3 Mohr-Coulomb Soil Model

This elastic-plastic model is based on soil parameters that are known in most practical situations. The Mohr-Coulomb model is used to compute realistic bearing capacities and collapse loads of footings, as well as other applications in which the failure behavior of the soil plays a dominant role. The model involves two main parameters, namely the cohesion intercept, \( c \) and the friction angle, \( \phi \). In addition to these parameters namely Young's modulus, \( E \), Poisson's ratio, \( \nu \), and the militancy angle, \( \psi \) need to calculate the complete \( \sigma-\epsilon \) behavior. Mohr Coulomb's failure surface criteria shown in Figure 3. According to Johnson et al. (2006), the failure envelope only depend on the principal stresses (\( \sigma_1, \sigma_3 \)), and is independent of the intermediate principle stress (\( \sigma_2 \)). When mapped in three-dimensional stress space, Mohr–Coulomb criteria resolved into an irregular hexagonal pyramid. This pyramid forms the failure/yield envelope, which is turn governs how soil will behave. The material behaves elastically if the stress point lies within the failure envelope. However, if the stress reaches the yield surface the material will undergo a degree of the plastic deformation. In the Mohr–Coulomb model used herein, it is assumed that until the soil has linear elastic relationship until failure.

The usual definition of the equation of Mohr-Coulomb surface is given below (Smith & Griffith, 1982).

\[
F = \frac{\sigma_1' + \sigma_3'}{2} \sin \phi - \frac{\sigma_1' - \sigma_3'}{2} - c \cos \phi
\]

3. INTERFACE ELEMENTS

Interfaces are modeled with 16-node interface elements. Interface elements consist of eight pairs of nodes, compatible with the 8-noded quadrilateral side of a soil element. Along degenerated soil elements, interface elements are composed of six node pairs, compatible with the triangular side of the degenerated soil element. Each interface has a 'virtual thickness' assigned to it which is an imaginary dimension used to obtain the stiffness properties of the interface. The virtual thickness is defined as the virtual thickness factor times the average element size. The average element size is determined by the global coarseness setting for the 2D mesh generation. The default value of the virtual thickness factor that is used in this study is 0.1. The stiffness matrix for quadrilateral interface elements is obtained by means of Gaussian integration using 3 × 3 integration points. The position of these integration points (or stress points) is chosen such that the numerical integration is exact for linear stress distributions. The 8-node quadrilateral elements provide a second-order interpolation of displacements. Quadrilateral elements have two local coordinates (\( \xi \) and \( \eta \)) (Fig. 4).

4. FINITE ELEMENT ANALYSIS

The pile and soil are discretized into a number of 20 noded isoparametric continuum elements. The interface between the soil and pile is modeled using 16 noded isoparametric interface elements, having aero dimension.
4.1 Continuum Element

Relationship between strains and nodal displacements expressed as

$$\{\varepsilon\}_e = [B] \{\delta\}_e$$  \hspace{1cm} (2)

Where $\{\varepsilon\}_e$ is the strain vector, $\{\delta\}_e$ is a vector consisting of nodal displacements, and $[B]$ represents the strain-displacement transformation matrix.

The stress strain relation is given by,

$$\{\sigma\}_e = \{D\} \{\varepsilon\}_e$$  \hspace{1cm} (3)

Where $\{\sigma\}_e$ is the stress vector, and $[D]$ is the constitutive relation matrix.

The stiffness matrix $[k]_e$, of an element is given as,

$$[k]_e = \int [B]^T[D][B] \, dv$$  \hspace{1cm} (4)

4.2 Interface Element

The relative displacement (strains) between the surfaces of soil and structure induce stresses in the interface element and these relative displacements are given by,

$$\{\varepsilon\}_i = [B]_i \{\delta\}_i$$  \hspace{1cm} (5)

Where $[B]_i$ represents the strain-displacement transformation matrix.

The element stiffness matrix is obtained by usual expression,

$$[k]_i = \int [B]^T_i[D_i][B]_i \, dv$$  \hspace{1cm} (6)

Where $[D]_i$ is the constitutive relation matrix for the interface.

4.3 Boundary Conditions

In this simulation the boundary conditions were

(i) Base of the ground domain was fixed in longitudinal (x), transverse(y), and vertical (z) directions

(ii) Left, right, back and front were fixed in x, y directions (the lateral directions) and free in the z direction.

4.4 Computer Simulation Details

Element size for all simulation was 50 cm $\times$ 50 cm $\times$ 100 cm. Details of simulation are given below. The program was implemented on Intel Core2Duo 3.0 GHz processor. Mesh size was taken to be 6m $\times$ 6m $\times$ 16 m. Number of 20 nodded elements in the mesh was 1728 and number of nodes associated with the mesh was 8281. Flowchart for FEM code is given in Figure 5. Properties of the four different soils used in the study are given in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Sand I</th>
<th>Sand II</th>
<th>Clay I</th>
<th>Clay II</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's Modulus (kN/m$^2$)</td>
<td>14000</td>
<td>13000</td>
<td>8500</td>
<td>10000</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
<td>0.3</td>
<td>0.3</td>
<td>0.35</td>
</tr>
<tr>
<td>Cohesion (c) (kN/m$^2$)</td>
<td>1</td>
<td>1</td>
<td>10</td>
<td>5</td>
</tr>
<tr>
<td>Friction Angle $\phi$</td>
<td>45</td>
<td>31</td>
<td>21</td>
<td>25</td>
</tr>
</tbody>
</table>

Table 1: Properties of Four Types of Solid Considered in the Study

The Modulus of elasticity of pile is $2 \times 10^9$ kN/m$^2$ and Poisson ratio 0.15. Three cases were taken for Finite Element Analysis, each having two layered soil and details of it are given below in Table 2.
Numerical Analysis of a Pile Subjected to Lateral Loads

Table 2: Three Cases of Soil Strata Considered in the FEM Analysis

<table>
<thead>
<tr>
<th>Cases</th>
<th>Layer 1 (7m)</th>
<th>Layer 2 (9m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Sand II</td>
<td>Sand II</td>
</tr>
<tr>
<td>2.</td>
<td>Clay I</td>
<td>Clay II</td>
</tr>
<tr>
<td>3.</td>
<td>Clay II</td>
<td>Sand I</td>
</tr>
</tbody>
</table>

5. RESULTS AND DISCUSSIONS

Three slenderness ratios were taken to analyze i.e., \( L/d = 8, 12 \), and 16. By keeping the cross-section of the pile constant to 1m and varying the length of pile to 8m, 12 and 16m. Lateral deflections were obtained along the longitudinal direction for the three cases considered are shown in Figure 6. A lateral load of 40 kN was applied at the ground surface. The results from the finite element method code were compared and checked against Opensees PL (2006) software. From this research work it is observed that the slender ratio of the pile increase there is a corresponding decrease in the deflection of the pile which is evident from the lateral deflection of the pile. The deflection from the finite element code is well comparable with the Opensees PL software.

6. CONCLUSIONS

The upper part of pile is the most critical part of pile in case of laterally loaded pile (Poulos & Davis, 1980) because of its greater deflection and its ability to carry higher lateral loads than the lower parts. It is important to study the lateral soil pressure along pile depth in order to understand which part of the pile carry large soil pressure that may cause pile foundation collapse.

From this research work on Numerical simulation of a single pile subjected to lateral loads, it is concluded that the lateral soil pressure under lateral loads changed with pile depth depending on the load intensity. It is also concluded that, water table depth influences the response of the lateral piles. That is dry soil condition gives higher lateral resistance than fully saturated soils. Here, the results are well compared with Opensees PL software and concluded that the slenderness ratio of the pile increases with the decrease in the deflection of the pile.

REFERENCES


