CHAPTER III.

A FINITE ELEMENT METHOD FOR PREDICTING THE CAPACITY OF AXIALLY LOADED SWAY PILES

Introduction

According to the problem statement in chapter I, a straight pile situated in a soft soil stratum with the pile tip embedded in an underlying stiff soil stratum is subjected to horizontal soil movements resulting in development of bending moments and deflections in the pile. At this step, an initial straight pile is transformed to a sway pile with initial eccentricity. The sway pile is then subjected to vertical pressure representing the design axial load of a straight pile under concentric loading, thus resulting in development of additional bending moments or curvatures in the sway pile. Therefore, for comprehensive study of the behaviour of pile for this problem, the method of analysis to be proposed herein must take account of the responses of the pile subjected to a number of external causes mentioned above.

This chapter describes the method employing the finite element for large deformation elastic static analysis and the computation of pile curvatures developed in the previous chapter. The computational procedures are then reviewed and, finally, the development of computer program is described and commented upon.

Method of Analysis

Finite element idealization of the problem can be done as follows:

In the state of plane strain, the finite element representing the geometric cross section of a pile and soil and their parameters is shown in Figure 3.1. It was found that for problems in soil mechanics the 8-node isoparametric finite element representation proved to be superior over other types of elements in its family (19).

For a reasonable representation of the path of deformation in this problem, the method of analysis which starts from a known initial

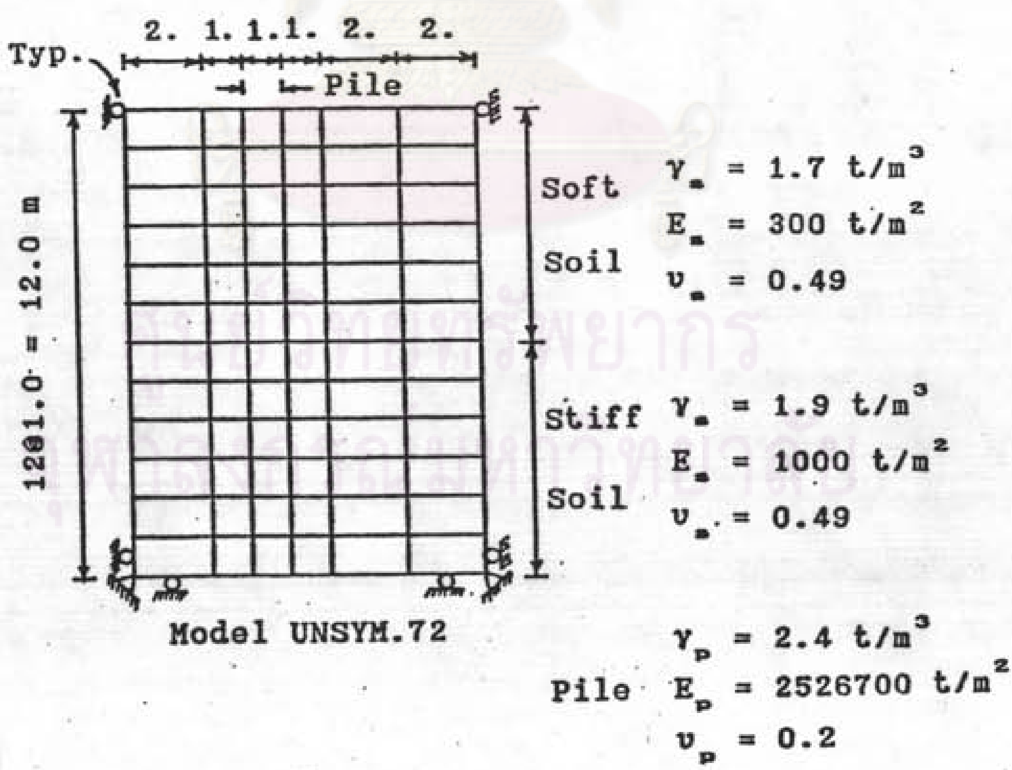

FIGURE 3.1 Idealized Finite Element Model of a Pile and Soil and Associated Parameters

condition and then simulates the external loading is performed continuously by the following procedures:

(a) Run No.1 (volume load)

In this initial step of analysis, the idealized finite element model representing the installed pile and soil continuum is analyzed for their weight densities. This accounts for the initial stress in the pile and soil continuum prior to the application of horizontal soil mevements.

(b) Run No.2 (specified displacements)

To represent the application of horizontal soil movements, a simple distribution of horizontal soil movement in the vicinity of the pile suggested by Poulos (16) as shown in Figure 3.2 is applied in terms of a set of specified displacements on the boundaries in the idealized finite element model. In this figure, z is the depth below

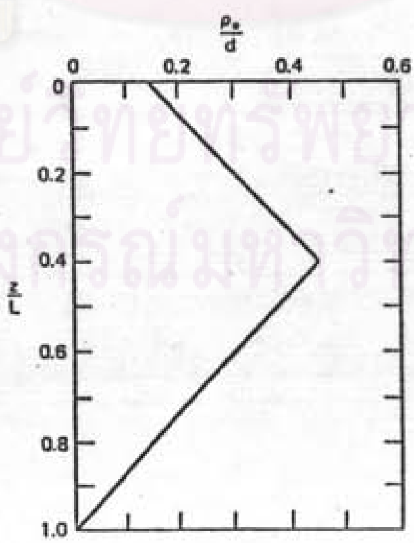

FIGURE 3.2 Standard Soil Movement Profile for Theoretical Solutions Suggested by Poulos (16)

ground surface; L is the depth of soil movement distribution; ρ_e is the magnitude of soil displacements; and d is the width or diameter of the pile. Such set of specified displacements is applied incrementally to analyze for the movement and curvature distributions along the pile. This accounts for the sway of the pile and gives the initial pile eccentricity prior to application of vertical pressure on the pile head.

(c) Run No.3 (pressure load)

In this step of analysis, the head of the sway pile is restrained from horizontal movement to represent the stage of superstructure construction. Then the vertical pressure on the pile head representing the design axial load of a straight pile under concentric loading is appiled incrementally to analyze for the additional movement and curvature distributions along the sway pile. This accounts for the capacity of axially loaded sway piles.

The above analyses are based on the total stress concept in which suitable soil parameters derived from laboratory and field tests are employed.

Computational Procedure

The essential features of the proposed finite element formulation can be described as follows:

In the global analysis, the path of deformation of a nonlinear elastic body (element) will be divided into a number of equilibrium states corresponding to the discrete load points $0, \Delta P, 2\Delta P, \ldots,$ where ΔP is an increment in loading. This formulation must take account of the following significant effects:

(a) At the end of each load increment, it is necessary that the second Piola-Kirchhoff stress corresponding to the equilibrium configuration at load P but measured in the configuration at load $P-\Delta P$ be transformed to the Cauchy stress in the configuration at load P. This is called the state determination.

(b) Before proceeding to the next step of analysis, it is necessary that the corresponding equilibrium configurations of the body be updated prior to application of such loading to the structural systems.

The computational procedures for the proposed method are summarized as follows:

(a) For each step of analysis

1. Update the state of stress, i.e., the state of the second Piola-Kirchhoff stress in the prior step of analysis is transformed into the state of Cauchy stress.

2. Update the nodal coordinates resulting from changes in geometry.

(b) For each load increment in each step of analysis

3. Update the state of stress, i.e., the state of the second Piola-Kirchhoff stress in the prior load increment is transformed into the state of Cauchy stress.

(c) For each iteration within the load increment

4. Compute the stiffness matrices, K_L and K_G.

5. Compute consistent nodal load vector in equilibrium with the state of stresses in configuration of the prior cycle of iteration.

6. Calculate consistent nodal load vector from the evaluation of the body forces and surface tractions.

7. Assemble master stiffness matrix and force vector.

8. Apply boundary conditions.

9. Solve for incremental responses. That is incremental displacements, q, incremental strains, E, incremental stresses, S, respectively.

10. Update the state of motion. That is

$$^{(p,r)}q = {}^{(p,r-1)}q + {}^{(r)}q$$

in which $^{(p,r)}q$ is the incremental displacement vector at r^{th} iteration of p^{th} load step; $^{(p,r-1)}q$ is the incremental displacement vector at $r\text{-}1^{th}$ iteration of p^{th} load step; and $^{(r)}q$ is the incremental displacement vector calculated at r^{th} iteration.

11. Convergence check. If ratio of the norm of increment $^{(r)}q$ to the norm of the current $^{(p,r)}q$, i.e., $\| ^{(r)}q \| / \| ^{(p,r)}q \| \leq$ Tolerance, repeat step 3 for the next load increment; otherwise proceed to steps 4 to 11.

12. Repeat step 1 for the next step of analysis until the final step of analysis is reached.

The flow sequence of the computation is shown in Figure 3.3.

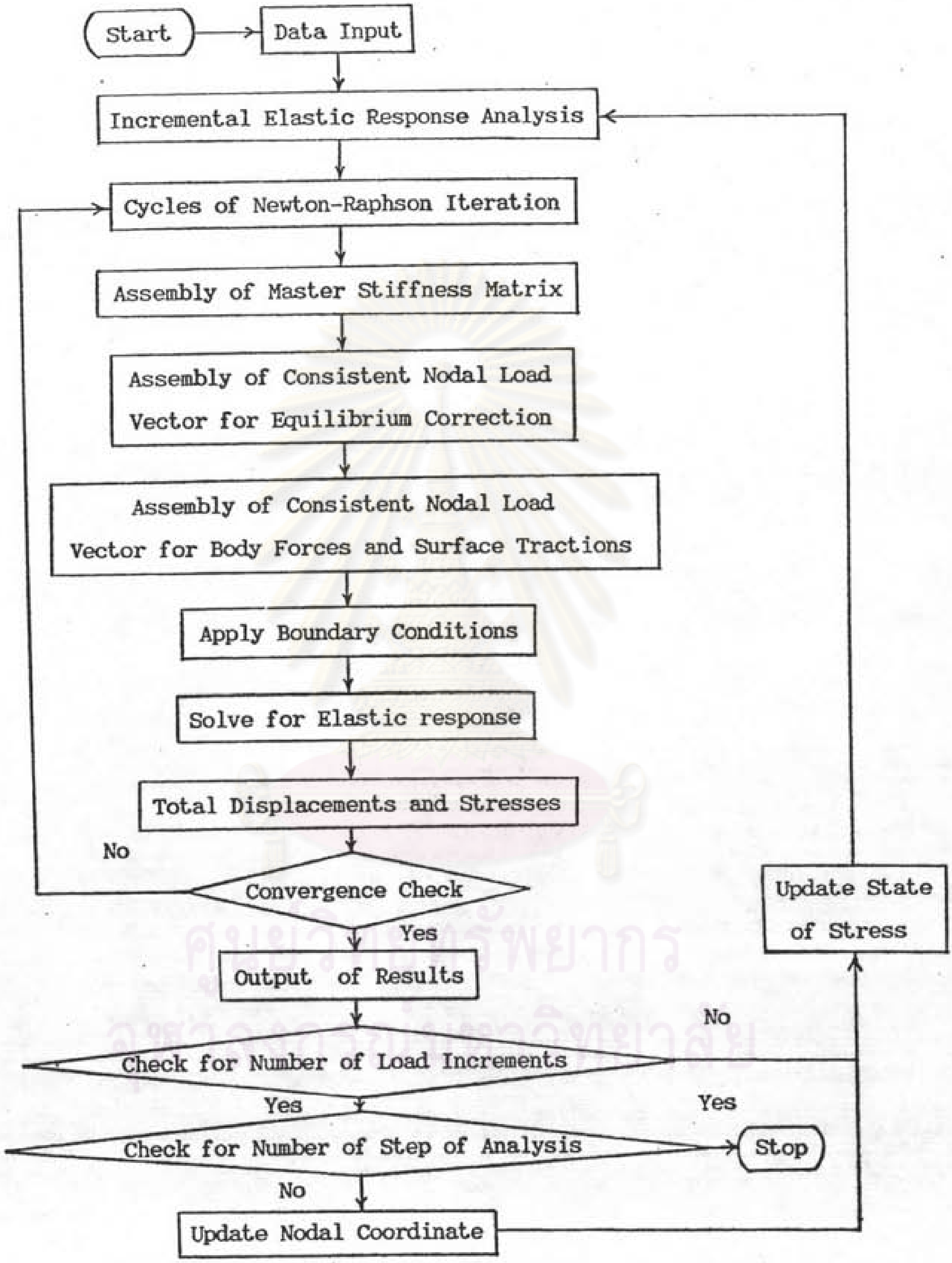

FIGURE 3.3 Flow Chart for the Computational Procedure of the Nonlinear Elastic Static Finite Element Analysis.

Computer Program Development

The finite element computer program developed herein with the main objective to predict the capacity of axially loaded sway piles is primarily a modification to general purpose nonlinear elastic static axisymmetric finite element computer program developed by Dr. Karoon Chandrangsu of Chulalongkorn University. The special features of the proposed program are as follows:

i. Nonlinear elastic static analysis of the structures subjected to a number of external causes.

ii. An external cause of specified displacements on the boundaries in the model.

iii. The options for changing in boundary conditions and material properties during the analysis.

iv. The option for limiting shear stress in the element.

v. Computation of pile curvatures.