NUMERICAL ANALYSIS OF PILE LOADING TEST
A CASE STUDY IN DUNKIRK, FRANCE

CHI HUNG LE[1], SUNGRYUL KIM[2]
[1] PhD student, Dong-A University, 840 Hadan2-dong, Saha-gu, Busan
[2] Asst. Professor, Dong-A University, 840 Hadan2-dong, Saha-gu, Busan

ABSTRACT: The Static Loading test (SLT) has been used for long time, it is an effective and economical method for load test in deep foundations. SLT often uses to determine the contribution of ultimate bearing capacity of pile which is very important for obtaining of pile length, pile size and number of piles to a project. This paper presents a 3D axisymmetric numerical analysis by using Finite Element method developed to simulate a full scale axial pile load test which reported by Chow (1997) naming the Imperial College Pile (ICP) in Dunkirk, France. The Mohr-Coulomb plastic model was used to present soil material, and linear elastic model was used for pile material.

Keyword*:* Static Loading test, Pile, Soil, Finite Element, Interface, Mohr-Coulomb, ABAQUS

INTRODUCTION

The Static Loading test (SLT) has been used for long time; it is an effective and economical method for load test in deep foundations. SLT uses to determine the contribution of ultimate bearing capacity of pile which is very important for obtaining of pile length, pile size and number of piles to a project. During the test, the axial loads are applied to the pile head. The different levels of load are conducted for each testing step. Detail of SLT can be found in many text books.

The pile load tests are mostly studied based on empirical correlation between experimental observations from laboratory and full scale in-situ testing. The in-situ investigations offer a direct ultimate bearing capacity while the empirical correlations provide roughly expected bearing capacity. However, both cases are not adequate to study deeply in its deformation. Thus, numerical methods can provide clearly pile and soil behaviors during the test in combining with the experimental observations from laboratory or full scale in-situ tests.

This study developed a 3D axisymmetric numerical analysis using Finite Elements method (FEM) to model a steel pile load test naming Empirical College Pile (ICP) in Dunkirk, France. Some mechanical behaviors of the pile and soil were obtained, such as: the pile head movement under axial loading, the development of yield zones in soil at the pile toe as well as the shaft and toe resistances of the pile.

IMPERIAL COLLEGE PILE

The pile which called Imperial College Pile (ICP) was reported by Chow (1997) using a steel pile which has a diameter of 0.102m and a length of 7.4m, close ended with a 60^0 in conical shape (Figure 1), and it was installed in homogeneous dense sand. The water table was at 4m depth below the ground surface. The detail of ICP pile can be referred in Chow (1997) and Bond (1989).

During the test, axial loads were applied by increment of 5-10% of the design load. The failure was reached when the rate of pile movement increased significantly with time. The failure axial load was 309 kN, which corresponded to a pile head displacement of 4.2 mm (Chow, 1997).

NUMERICAL MODEL OF PILE TEST

Mohr-Coulomb model

The Mohr-Coulomb plasticity model is intended for modeling granular materials such as soils under monotonic loading conditions and does not consider rate dependence. When the granular soil is subjected to the loads, the displacement will contain both the recoverable and non-recoverable components. Therefore, the failure criteria should include the elastic model to define the distribution of stress causing by plastic behavior (Figure 2). Thus, a possible envelope which is Mohr Coulomb surface can present Elasto-plastic, as shown in Figure 3.

The Mohr-Coulomb criterion assumes that yield occurs when the shear stress at any point in a material reaches a value that depends linearly on the normal stress in the same plane (*ABAQUS, 2010 User manual*). The Mohr-Coulomb model is based on plotting Mohr's circle for states of stress at yield in the plane of the major ($\sigma 1$) and minor stresses ($\sigma 3$). The yield line is the best straight line that touches these Mohr's circles (Figure 3).

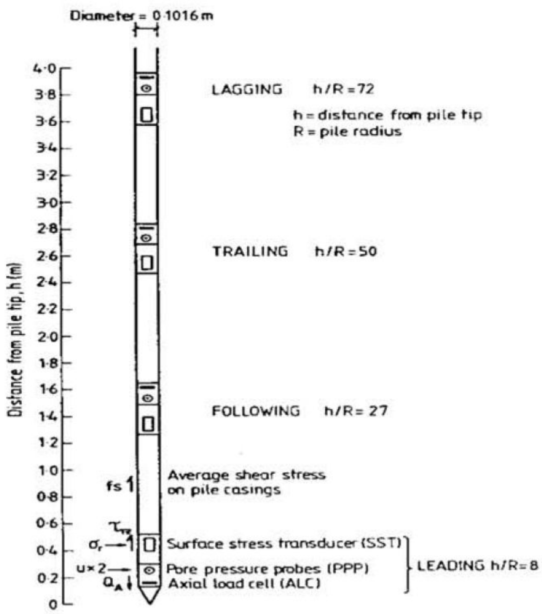
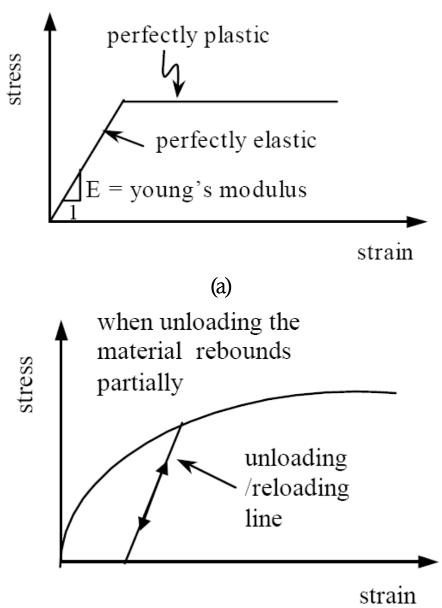


Figure 1 Model of ICP test pile (Chow, 1997)



(a)

(b)

Figure 2 Stress-strain relationship for
(a) Most structural materials. (b) An elasto-plastic material

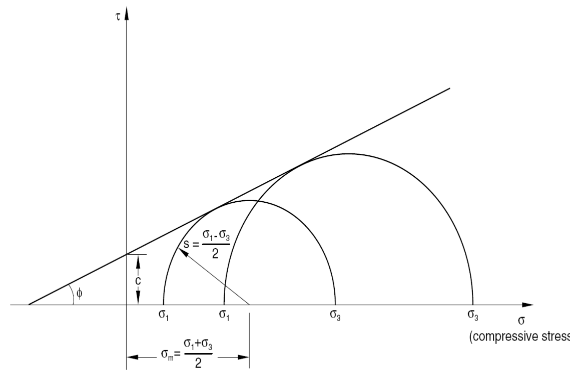


Figure 3 Mohr-Coulomb yield model

Therefore, the Mohr-Coulomb model is defined by:

$$\tau = c - \sigma_m \tan\phi \tag{1}$$

where *s* is negative in compression.

From Morh's circle:

$$\tau = s\cos\phi \tag{2}$$

$$\sigma = \sigma_m + s\sin\phi \tag{3}$$

$$s = ½(\sigma_1 + \sigma_3) \tag{4}$$

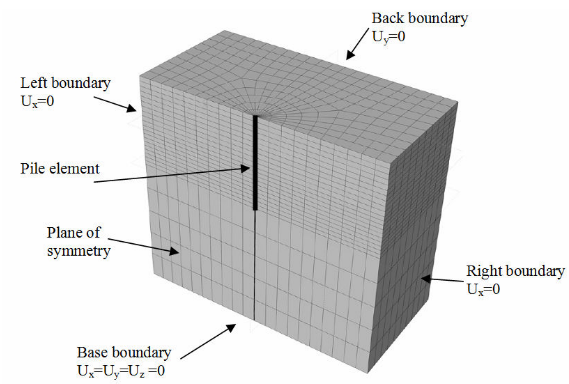
The material will behave elastically if the stress point lies within the failure envelope. However, if the stress reaches the yield surface, the material will present a degree of plastic deformation.

Finite element mesh and numerical model

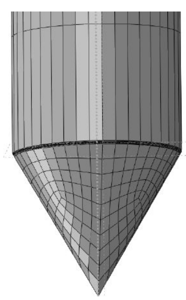
An axisymmetric model was developed by using ABAQUS (2010) with the mesh extending larger than 30 times the pile diameter (taken 20m long and 10m wide in *x* and *y* directions, respectively) laterally from the axis of symmetry. And, the height of the model was larger than 2 times the pile length (taken 18.5m height in *z* direction) extending vertically, as shown in Figure 4a. The pile toe was simulated similarly the real conical shape of ICP pile, as shown in Figure 4b. In the analysis, the pile was assumed to be perfect contact with the surrounding soil. The hexahedral elements with eight-node linear brick, reduced integration were used for the pile and soil. It is noted that, the mesh is finer in the vicinity of the pile since that zone is the one of stress concentration, as shown in Figure 4a.

The analysis was conducted in the two basic steps, as follows:

① *Initial Equilibrium condition*: The initial conditions equilibrium must be control to ensure the nonzero stress exists in the soil media as the first step of the analysis. This consists of an effective vertical stress increasing linearly with depth by the weight of material as well as the horizontal stresses which are caused by tectonic effects. In this step, the pile is assumed to have no interaction with the surrounding soil and is constrained any movement in this step. This step is necessary to ensure that the equilibrium states of stresses in the soil mass are defined.



(a) Soil model



(b) Pile model

Figure 4 Finite element discretization:

② *Loading condition*: This step conducts the simulation of static loading tests by applying the different levels of axial load to the pile head. The pile and the soil surface constraints are released from the initial conditions step, allowing contact between the pile and the surrounding soil to be established In ABAQUS, different loads are applied for each calculation process by activating the loads (pressures) to the pile head element.

Material properties

The steel pile with the diameter of 0.102m and the length of 7.4m was installed into the sandy soil. The test pile properties are given in Table 1.

Table 1 Test pile properties (Chow, 1997)

Material	Steel
Length, L (m)	7.4
Diameter, D (m)	0.102
Young's Modulus, E_s (GPa)	195
Poisson's ratio, ν	0.28

The sandy soil in Dunkirk, France, is homogenous and dense sand (Chow, 1997). The soil properties are given in Table 2.

Table 2 Sand properties at Dunkirk site (Chow, 1997)

Unit weight (kN/m^3)	γ_{unsat}=17.1 γ_{sat}=19.9
Young's Modulus, E_s (MPa)	200
Poisson's ratio, ν	0.3
Friction, ϕ (degree)	37
Dilatancy, ψ (degree)	10
Cohesion, c (kPa)	0.1

Interface between pile and soil

The interface between the pile and the soil was simulated by using penalty-type interface. The frictional ratio between the steel pile and soil was assumed to be 0.3 in this study. This type of interface uses a stiffness (penalty) method that permits some relative motion of the surfaces (an "elastic slip") when they should be sticking. While the surfaces are sticking, the magnitude of sliding is limited to this elastic slip (*ABAQUS User manual 2010*). Hence, it is capable of describing the frictional interface between the pile surface and the soil in contact.

DISCUSSIONS OF ANALYSIS

A comparison between ICP static load test and numerical analysis for the pile load-movement results is shown in Figure 5. The maximum axial load reported by Chow (1997) was 309 kN corresponding to pile head movement of 4.2mm. However, the load in the numerical model was larger than the failure load of 309 kN. If the numerical analysis failed, this means that the plastic states in the calculation was reached. The maximum simulated load test was 390 kN, corresponding to 4.3 mm of pile head movement. It can also be seen from Figure 5 that the behavior of the load-movement from FEM simulation showed a good agreement with ICP load test result.

The load-movement showed a linear relationship in the range of 260 kN applied load, corresponding to 2.6mm of pile head movement for both experimental observation and numerical analysis, which means that the steel pile and dense sand "system" for this case working in "elastic" states if the applied load does not exceed 260kN.

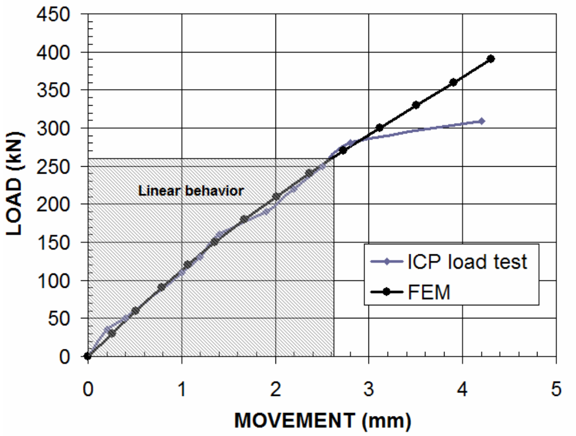


Figure 5 Comparison of measured ICP load test and numerical analysis

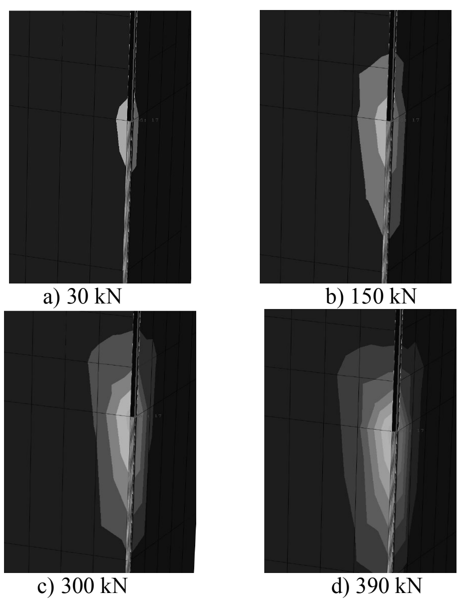


Figure 6 Development of yield zone due to different simulated load tests

Figure 6 shows the yield zones at the pile toe for different simulated loading levels. It can be seen that, as the load increases, the yield expands. Not many yield zones appear along the shaft. This suggests that the bearing capacity of pile is mostly carried out by the side resistance rather than toe resistance for this case, as shown in Table 3.

Table 3 The computed bearing capacity of pile

Toe resistance		Side resistance		Applied Load
kN	%	kN	%	(kN)
21.6	14.4	128.4	85.6	150
29.4	12.2	210.6	87.8	240
33.0	11.0	267.0	89.0	300
38.0	9.7	352.0	90.3	390

It can be seen that, the toe resistance was gradually to be fully mobilized as increments of the applied loads.

CONCLUSIONS

The numerical analysis in this study can be used to investigate soil pile system. The analysis shows a good agreement between the load-movement from numerical simulation and the ICP load test results.

The steel pile and dense sand system working in elastic behavior in case of the load test does not exceed 260 kN for this case. And during loading, the higher the load, the larger the expansion of yield zone around the pile toe which changes the elastic behavior to plastic behavior in soil mass.

This analysis also suggests that the bearing capacity of pile was mostly carried out by side resistance. The toe resistance was gradually to be fully mobilized as increments of the applied loads. The computed maximum load was 390 kN while the experimental maximum load was 309 kN.

REFERENCES

ABAQUS 6.10-1, (2010). Dassault Systèmes Simulia Corp., Providence, RI, USA.

Bond A.J. (1989). Behavior of displacement piles in over consolidated clays. *PhD Thesis*, Imperial College, London, United Kingdom.

Chow F.C. (1997). Investigation in behavior of displacement piles for offshore foundation. *PhD Thesis,* Imperial College, London, United Kingdom.

De Gennaro V., Frank F.(2002). Elasto Plastic analysis of the interface behavior between granular soil media and structure. *Computer and Geotechnics*. Vol. 29, pp: 547-572.

Fellenius.B.H, et al. (1999). O-Cell testing and FE Analysis of 28-m-deep barrette in Manila, Philippines. *Journal of Geotechnical and Geo-environment Engineering*, ASCE, Vol. 125 (7). pp566-575.

Liu.G.R. and Quek S.S. (2003). *The Finite Element Method: A practical Course*. Elsevier Science.

Jardine R., Chow F.C, Overy R. and Standing J. (2005). *ICP design method for driven piles in sands and clays*, 112. Thomas Telford Ltd.