Verification of Load –Displacement Response of Augur Cast Pile in Clay.

G P Panda, and C. Vipulanandan[1], Ph.D., P.E.
Center for Innovative Grouting Material and Technology (CIGMAT)
Department of Civil and Environmental Engineering
University of Houston, Houston, Texas 77204-4003
E-mail: gppanda@uh.edu,cvipulanandan@uh.edu Phone: (713) 743-4278

Abstract: Pile foundations are widely used in weaker soil to support super structure. Apart from laboratory and field tests, finite element method is being used to predict the settlement of piles. In this paper, some aspects of numerical analysis due to vertical loading on pile by using ABAQUS have been studied.

1. Introduction

Pile foundation is widely used in foundation engineering. In traditional approach, designing pile foundations requires checking limits. It is necessary to understand the soil structure and have an insight of interaction between soil and pile. Another alternative way to understand the complex processes is to do computer simulations. To do simulation exact geometry and material properties are required. It is important to compare a numerical solution with the results obtained by in-situ tests to know the effectiveness of model. In this paper, numerical simulation results were compared with the results of obtained in situ test.

2. **Objectives**

Evaluate the effectiveness of computer model to predict the settlement of pile by comparing it experimental results.

3**. Methods** Various constitutive model is used to describe the complex behavior of geomaterial under different loading conditions. Mohr-Coulomb failure criterion, usually named Mohr-Coulomb model is widely used in finite element analysis of geotechnical engineering due to simplicity and accuracy. The failure envelope depends upon the major and minor principle stress, is defined by cohesion and internal frictional angle.

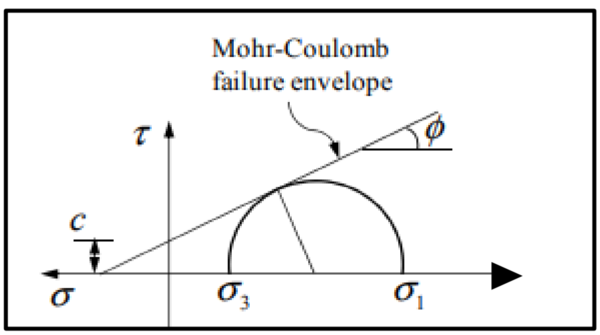

Figure 1:Mohr –Column failure criterion.

Contact behavior of pile soil Interface-Contact behavior between pile and surface is necessary to define the load transfer mechanism. Normal force is transferred only when pile and soil are in contact and this behavior is modeled by hard contact option in Abaqus.

Table 1 Material properties of Soil used for simulation

Soil	
Density	1900kg/m^3
Young Modulus	175 MPa
Poisson Ratio	0.3
Cohesion Yield Stress	200kPa
Friction Angle(deg)	20
Dilation Angle(deg)	0

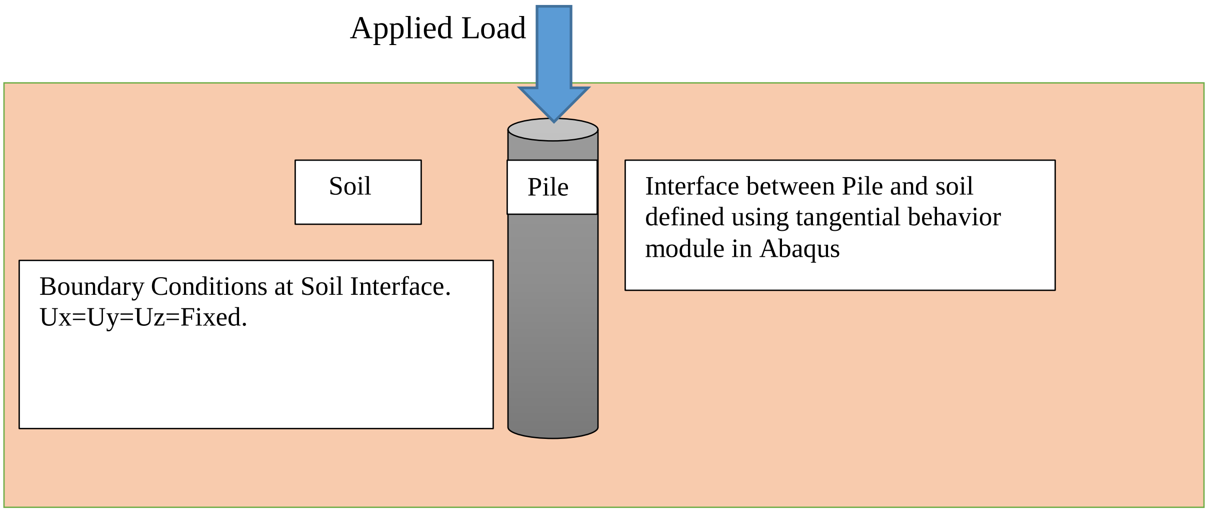

Figure 2: 3D model of pile –soil interaction taken for study.

After defining the constitutive model and contact behavior of soil, the model is submitted for analysis in ABAQUS. Finite element displacement of piles requires application of appropriate modeling of pile elements interaction with surrounding soil. The contact was modelled using master slave contact. The finite element grid was formed with 16488 elements. The pile was modeled using linear hexahedral element while the soil was modelled by quadratic tetrahedral elements are available in library if ABAQUS/standard elements.

In this study the pile settlement vs load applied is plotted. To simulate the model in Abaqus some of the conditions are assumed in the simulation. The results for the simulation in Abaqus were compared with an experimental result obtained from conventional load test of pile. The test pile was instrumented with a top load cell and settlement gauges, in order to measure the applied load and settlement during deformation. Full scale axial load was performed with ASTM D1143 “Standard Method to characterize Testing Piles under static Axial Compression Load. The test pile was loaded in 180kN increment upto to 4600kN and unloaded in four equal steps. Pile settlement results obtained from the simulation was compared with displacement results from experiment.

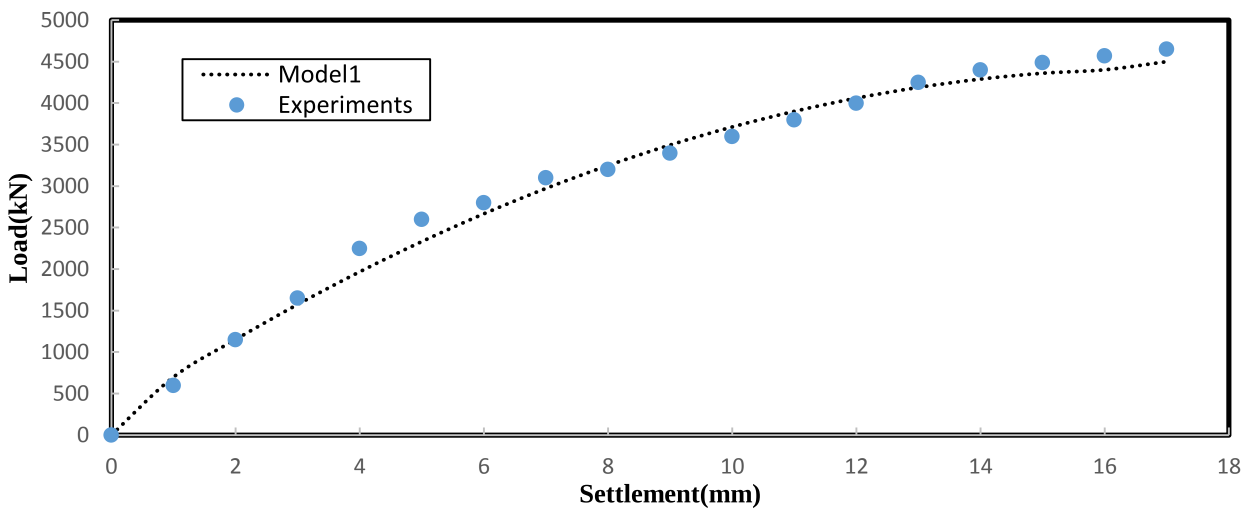

Figure 3: Vertical Displacement of pile top dependent on vertical load.

Using Vipulanandan Pile model, $(Q/Qult)=[\frac{(\frac{\delta}{d})}{(\alpha+\beta(\frac{\delta}{d})}]$ (1)

Where Q is the load and Qult is the ultimate load at very large settlement. The settlement is δ and δ_{50} is the settlement at 50% of the ultimate load. The loading model prediction using Eqn. (1) and the parameters summarized in Table 2 is shown Figure 3.

Table 2 .Summary of the Pile Load Displacement Relationship Parameters

Description	Qult(kN)	d(mm)	α	β	RMSE
Test Pile	5916	760	0.0084	1	25.3

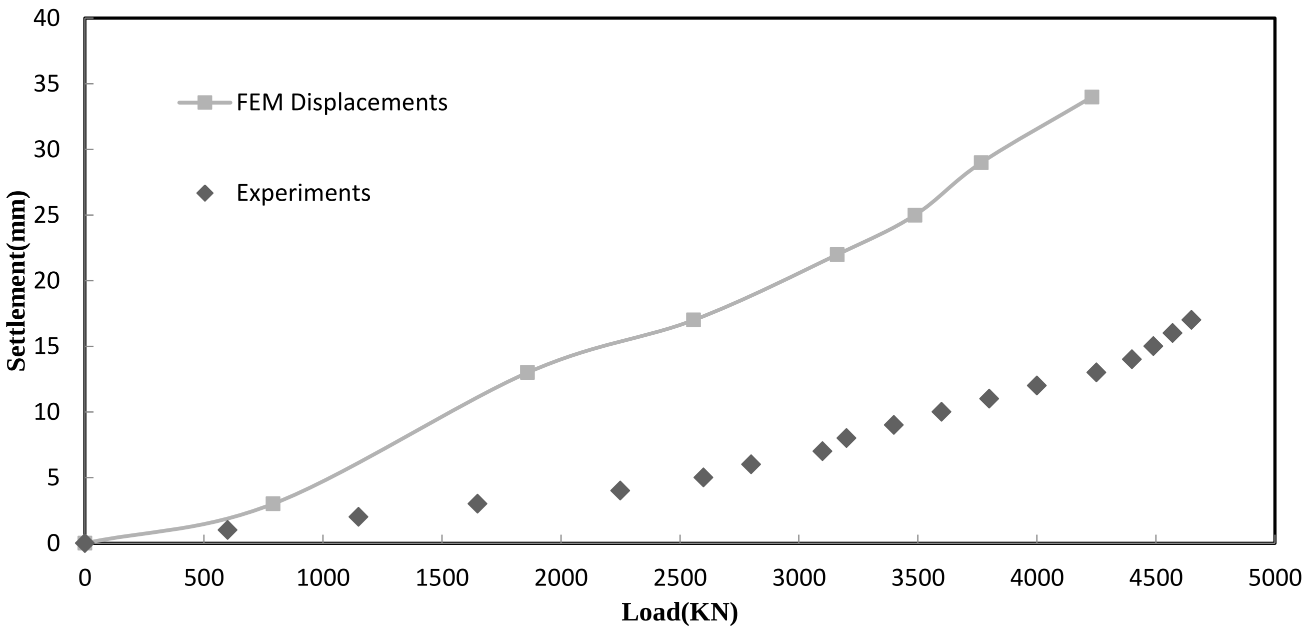

Figure 4: Comparison of pile settlement results from FEM and experimental.

The pile was loaded more than four times the design load 1150kN and displacements were comparable till 1250KN.Higher displacements were observed after further loading in case of FEM simulation. Changing the input parameters changes the displacements.

5. **Conclusion:** Numerical simulation matched with experimental results. With accurate field data of soil like density, young modulus, shear strength and cohesive strength the settlement of pile can be predicted by doing numerical simulations. It can be concluded that coulomb frictional model in conjugate with “hard contact” contact model is capable of modelling pile soil interactions.

6. **Acknowledgements:**

This study was supported by the Center for Innovative Grouting Materials and Technology (CIGMAT), University of Houston, Houston, Texas.

7. **References**

1) Wrana B, Kalisch W,Waerzonek M. “*Nonlinear Analysis of Pile Displacement Using The finite finite element method”* Technical Transactions Civil Engineering.

2) Achmus M ,Abdel K. R. “*Numerical Modelling of vertical piles in clay under inclined Compression Loading”*.

3) Zhan Y,Wang H ,Liu F. “*Modeling Vertical Bearing Capacity of Pile Foundation by using* Journal of Geotechnical Engineering. 17. 1855-1865