XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

Prediction of settlements generated by a static load test through numerical modeling using PLAXIS 2D

Pedro F. Martins[1]

[1]*Programa de Pós-Graduação em Engenharia Oceânica, Universidade Federal do Rio Grande - FURG*
Avenida Itália km 8, 96203-900, Rio Grande/Rio Grande do Sul, Brazil
pedrodafmartins@gmail.com

Abstract. When it comes to geotechnical engineering, one of the greatest concern when building any type of structure is the settlements that occurred on its foundations. As a way of predicting these settlements, static load tests can be performed. This test consists in the application of successive loading stages at the top of the pile and in the respective measurement of the settlements occurred. This article uses numerical modeling as a way to estimate the settlement of piles caused by this test. The analysis was performed through an axisymmetric two dimensional model in PLAXIS 2D AE. software, using the finite element method as a way to obtain the results. This method allows to model satisfactorily the behavior of the local stratified soil and its interaction with the pile. PLAXIS software allows the use of different constitutive models in the simulation of the soil, thus obtaining greater precision in the results. The objective of this work is to improve the numerical modeling of static load tests using the PLAXIS 2D software. Results obtained through an initial model were consistent with a static load test report carried out at the site, thus evidencing the tool's potential in predicting settlements.

Keywords: finite element methods, settlements, static load test, deep foundations, PLAXIS 2D.

1 Introduction

When building any type of structure, one of the subjects of the project that most concerns engineers is the foundations and the load transfer to the ground. This concern occurs because the soil is not formed by a single material with well-defined properties, but the union of several materials with different characteristics. In stratified soils, the prediction of behavior is even more difficult, considering that different materials may be interspersed along the depth.

As a way of avoiding uncertainties and optimizing the design of foundations, geotechnical investigations can be carried out at the study site. Standard Penetration Test (SPT) tests are widely used in Brazil as a soil analysis tool, especially for its low cost and ease of execution. Allied to these investigative tests, load proof tests can be performed on foundations. This type of test is used with the purpose of predicting permissible settlements and loads, and can be of the static and dynamic type.

A static load test (SLT) consists in the application of a progressive load on the top of a pile and the verification of the corresponding settlement resulted from that load. The procedures and methods used in this test are described in the Brazilian standard NBR 12131 [1].

When executing the design of foundations, a minimum number of SLT must be performed, respecting the requirements of NBR 6122 [2]. The number of tests to be carried out in the execution varies according to the type and quantity of pile used and the maximum allowable tension.

In the execution stage of the foundations in a port pier in the city of Rio Grande, a static test load was performed as a way to predict the behavior of the structure. The company responsible for the load test issued a report in which the loads applied to this pile are present, as well as the respective settlements.

The geotechnical parameters information from the site obtained through SPT tests, added to the results of the load proof test, were used in the generation of a computational model using the PLAXIS software. This work aims to evaluate the use of numerical modeling, through the finite element method, as a tool to predict the behavior of settlements in piles. Different models were used to make a comparative analysis of the results obtained.

2 Case study

2.1 Location and construction characteristics

The load test was carried out on one of the piles of the Honório Bicalho wharf in the city of Rio Grande, southern Brazil. This wharf was built as part of an expansion project that expected to receive oil platforms that would built in the city. The built pier consists of five modules, with 373 piles along 216 meters length. The piles, which have a diameter of 800 mm and 35.8 meters in length, are made of concrete with a characteristic resistance greater than or equal to 40 MPa.

2.2 Soil parameters

The coastal region of the city of Rio Grande is characterized by the presence of clay layers interspersed with layers of fine sand. The port region studied presented the same characteristic and, in order to verify the properties of the local soil, prospecting tests of SPT type were carried out along the length of the pier.

The soil profile used in this work has been obtained nearby the place where the load test was performed. The results of the soil penetration resistance index obtained in the SPT test were used as a means of obtaining soil parameters through correlations found in the literature.

The natural (γ) and saturated (γsat) specific weight were obtained in Godoy [3], the friction angle (φ) in Cintra [4], elasticity modulus (E) and undrained resistance (Su) in Stroud's work [5] and the cohesion (Cu) and Poisson's coefficient (υ) used were suggested by Teixeira and Godoy [6]. Table 1 presents the results obtained according to these correlations, which were later used in the construction of the model in the PLAXIS software.

Table 1. Soil parameters used in the numeric model

Layer	Soil Type	Classification	Average Nspt	γ (kN/m³)	γs (kN/m³)	φ (º)	E (kN/m²)	Cu (kN/m²)	Su (kN/m²)	υ
1	Organic Clay	Very Soft	0	13.0	15.0	17.6	600.0	3.3	1,7	0.4
2	Clayey Sand	Med. Compact	11	17.0	20.0	29.6	19200.0	106.7	53.3	0.3
3	Sand	Little Compact	11	16.0	19.0	29.8	19800.0	110.0	55.0	0.2
4	Sandy Clay	Medium	11	17.0	19.0	29.8	19800.0	110.0	55.0	0.4
5	Clayey Sand	Med. Compact	12	17.0	20.0	30.4	21375.0	118.8	59,4	0.3
6	Sandy Clay	Medium	6	17.0	19.0	26.1	11057.1	61.4	30.7	0.4
7	Sand	Very Compact	44	18.0	21.0	44.6	78750.0	437.5	218.8	0.4
8	Sandy Clay	Medium	8	17.0	19.0	27.6	14400.0	80.0	40.0	0.4
9	Sand	Very Compact	41	18.0	21.0	43.5	72900.0	405.0	202.5	0.4

2.3 Load proof test

The pile analyzed in this work belonged to the Load In/Load Out platform of the pier. The load test performed was of the static type, in which time intervals are inserted between the applications of the load in order to have their settlements stabilized.

Loads were applied using a hydraulic cylinder, along with a pump, a pressure gauge and a calibrated load cell. The reaction system of the application of the force consists of a set of metal beams anchored to the ground through eight ties. To measure the pile settlements, four strain gauges with readings measured from 0.01 mm were used.

The applied methodology can be verified in the NBR 12131 [1]. According to the guidelines, the loads must be applied in stages of a maximum of 20% of the workload of the piles and their unloading stage must consist of at least 3 stages of 15 minutes each, added to a final stage of 30 minutes. Table 2 shows the settlements results obtained in the load test report, according to Luzzardi [7] study.

Table 2. Load stages and respecive information

Stage	Duration (min)	Applied Load (kN)	Vertical Displacements (mm)
1º	30	655.0	0.58
2º	30	1256.0	1.55
3º	30	1872.0	2.35
4º	30	2523.0	3.25
5º	30	3085.0	4.3
6º	30	3703.0	5.36
7º	30	4340.0	6.52
8º	30	4950.0	7.96
9º	30	5568.0	9.37
10º	30	6240.0	11.31
11º	840	6772.0	13.34
12º	15	5022.0	12.08
13º	15	3385.0	10.57
14º	15	1713.0	6.73
15º	30	0.0	2.09

3 Numerical simulation

3.1 Finite elements method

Usually engineering problems involve complex structures and situations and are solved using differential equations. Through the numerical methods it is possible to obtain the results of these equations and the finite element method (FEM) emerged as a solution for the discretization of the problems, splitting complex situations into smaller parts. This method aims to solve differential equations through a system of algebraic equations, thus relating the sought variable to a finite number of points.

By fragmenting the problem into several small regions, complex elements and variables become more easily examined. The finite element mesh is composed of all this small elements, and the more refined it becomes, usually the more accurate the results are, with the disadvantage of the longer computational time required.

Silva [8] affirms that in situations involving interaction between soil and structure, the FEM proves to be a great tool, being able to model not only the structure but also the foundations, the soil and its particularities. The non-linear behavior of stress and deformation of the soil, the varied conditions due to heterogeneity and the complex geometry of the foundations can be modeled through this method.

Also Wegman [9] recognizes that FEM is among the most successful numerical methods in the analysis of soil-structure interaction, incorporating essential aspects such as soil stratification, area of interaction between adjacent foundations and the relative rigidity between soil and the foundation.

3.2 PLAXIS 2D software

The software used for the analysis of this work was PLAXIS 2D AE., launched in 1993 and developed at the Technical University of Delft in the Netherlands. PLAXIS uses FEM as a calculation tool for solving problems involving structures and the soil.

When performing soil modeling, several features can be implemented, such as the use of different soil constitutive models, analysis of the problem in succession of stages and dynamic loads. These tools show the software's ability to reliably model the situations encountered by engineers at the construction site.

Steps of data inputs and outputs are separated into 4 different modules that interact with each other: input (data entry), calculations (calculation steps) output (data output and information relevant to the model) and curves (generation of data and graphs with the results).

3.3 Constitutive models

When modeling a problem involving geotechnics, one of the topics of great importance is the identification and evaluation of the constitutive model of the materials to be modeled. The correct definition will generate results and information consistent with the reality of the situation.

The used version of PLAXIS has eight constitutive models, which proposal is to reproduce in the best possible way the varied materials and situations found on construction site. In this work, Mohr-Coulomb is used to model soils and Linear Elastic is responsible for modelling structures.

The linear elastic model represents Hooke's law of linear elasticity, allowing deformations without plasticity of the material. In Tab. 3 the parameters of specific weight of the material (γ), modulus of elasticity (E), shear modulus (G), oedometric modulus (Eoed) and Poisson's coefficient (υ) used in the model are presented.

Table 3. Pile material properties

Pile	γ (kN/m³)	E (kN/m²)	G (kN/m²)	E oed (kN/m²)	υ
E1G	24.0	35420000.0	16100000.0	36220000.0	0.1

The Mohr-Coulomb model, on the other hand, considers the elasticity of the material until it begins to have a plastic behavior defined by the Mohr-Coulomb rupture surface, being suggested in the PLAXIS manual as ideal for preliminary analyzes. The choice of the Mohr-Coulomb model is also advocated by Gowthaman and Nasvi [10], who showed that for loads up to 13000 kN, this choice presents better results for isolated pile settlements.

4 Situation modeling

4.1 Model properties

The modeling of the situation was carried out using an axisymmetric model of 40 meters length by 50 meters depth. The dimensions used are sufficient so that the boundary conditions do not affect the displacements generated by the loads.

The parameters adopted for the soil and for the model piles were shown in Tab. 1 and Tab. 3 respectively. Pile modeling was performed by inserting a volume of elastic material into the soil.

4.2 Mesh analysis

PLAXIS 2D has an automatic mesh generation and analysis. When generating the mesh, it is possible to choose between 5 refinements of the mesh, ranging from very coarse to very fine. After the mesh is generated, it is possible through the software to obtain information regarding its quality, indicating values of 1 for elements with high quality and 0 for elements of lower quality.

A study of the settlement obtained at the top and at the bottom of the pile was made for all refinements. This analysis presented a variation in the values between the medium and fine meshes of 1 mm, as can be seen in Tab. 4. Observing the computational time required to generate the different scenarios, the mesh defined for the model was the average, with 1590 elements and 13197 nodes. The automatic graphical analysis of the software contributed to this decision, showing a good quality for the chosen mesh.

Table 4. Comparison of mesh parameters

Displacements (mm)	Very coarse	Coarse	Medium	Fine	Very Fine
Top	15.0	14.0	14.0	13.0	13.0
Base	10.0	9.0	8.4	7.7	7.4
Nodes	9675	11305	13197	18603	26435
Elements	1155	1356	1590	2261	3227

4.3 Calculation steps

As previously mentioned, PLAXIS has a construction step simulation tool. Using this tool, it is possible to simulate tensions and displacements as they occur in reality, successively over time. In this model, the steps mentioned in table 1 were simulated in 15 progressive loading stages, and the settlements of each step were used to obtain a comparison of the parameters used.

4.4 Interfaces

In this study, 4 different situations of interaction variation between the pile and the soil were analyzed, and this interaction was simulated through the interaction coefficient (IC) of the soil. The interface element starts at the same initial height as the stake and extends for 72 centimeters after the end of the stake, following the extension suggestion of Brinkgreve and Broero [11].

Following the recommendations of Zakia, Abdelmadjid and Nahize [12], which suggests values from 0.8 to 0.9 to best represent the interaction between elements, an IC of 0.8 was used. Models with 0.6 and 0.4 interaction were also carried out, in addition to a model that does not consider the existence of interaction between soil and pile.

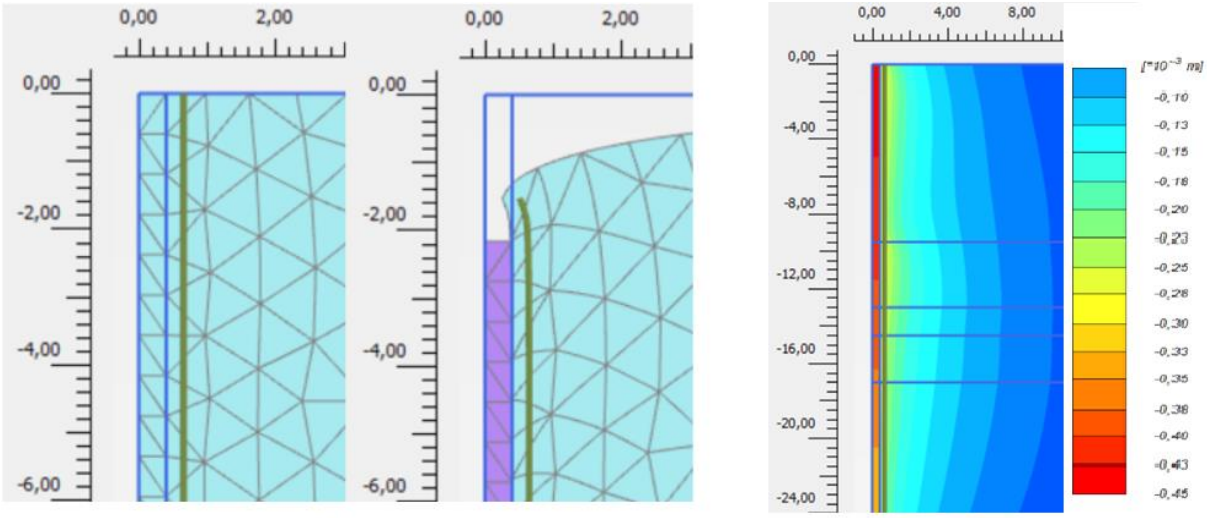
5 Results

After the creation of the model, the calculation steps were performed and the results for the settlements obtained through the different interaction parameters were analyzed. In Fig. 1(a) a detail of the initial model is presented in its initial and final stages, showing on an enlarged scale the behavior that occurred at the top of the pile when loaded.

Figure 1(b) shows a graphical representation of the total vertical settlements in the first half of the pile, demonstrating through a color gradient its dimension along the depth. It is evident that the settlements occurred at the top of the pile are greater than that occurred at the base, demonstrating its elastic shortening.

Figure 2 presents the graph of the displacement behavior according to the variation of the interaction coefficient, along with the results obtained during the field test. Although the curves have the same behavior trend, the settlement values obtained with IC of 0.4 and 0.6 were very high, discarding their use in the model.

For an IC of 0.8 and 1.0, the results were more consistent with the load test, but still showing a certain discrepancy in settlements values. While the average percentage of the settlement difference between the 0.8 interaction coefficient model and the field test was 28.95%, the difference obtained using an IC of 1.0 was 19.53%.



(a) **(b)**

Figure 1. Top of pile behavior on the modelling

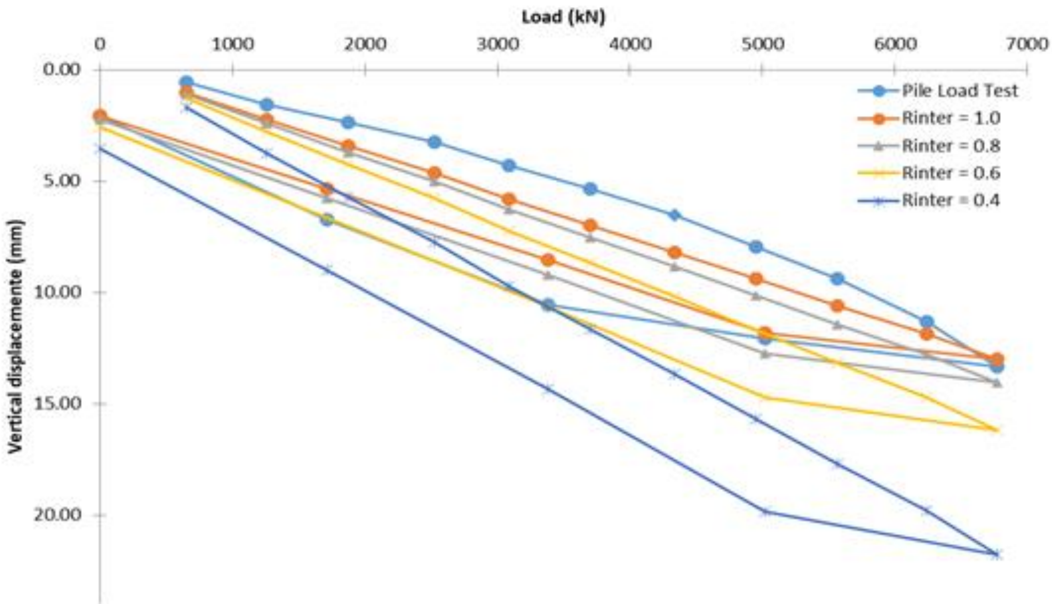


Figure 2. Vertical displacements graphic

Data analysis showed a trend of behavior common to all models. As the loads increase, the results of settlements obtained through numerical modeling approach the actual pile settlement. This may occur due to the fact that numerical model does not consider some field situations, such as the soil denting generated by the pile driving.

The average settlement difference obtained for the maximum load of 6772 kN in the models with IC 1 and 0.8 was only 3.85%, compared to the load test. The biggest difference between the displacement of the test and the modeling occurs in the 7th stage, being of approximately 1.67 mm.

6 Conclusions

Through the analysis of the results of the vertical displacements obtained in the different computational models, it is evident the discrepancy of values using very small interaction coefficients, showing that they are not adequate to model this situation. The results that came closest to the load test are those that used the interface with an IC of 1, however this configuration does not truly represent the interaction behavior between structure and soil.

Analyzing the data, it is concluded that the initial model presents satisfactory results for the prediction of the vertical displacements of the load proof test when loads above 6000 kN are applied. This model should not be used as a definitive tool, but as an auxiliary prediction method in specific situations.

This model can be optimized with the representation of the soil through different constitutive models available in PLAXIS, such as Hardening Soil. Additionally, in-depth geotechnical investigation tests can help the characterization of the local soil, improving the soil parameters used in the model and then bringing more accurate results.

Acknowledgements. This work was carried out with the support of the Coordenação de Pessoal de Nível Superior – Brasil (CAPES). Special thanks to the Post-graduate Program in Oceanic Engineering at the Federal University of Rio Grande – FURG for providing the software used in this research.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] Associação Brasileira de Normas Técnicas. NBR 12131. Estacas – Prova de Carga Estática – Método de Ensaio. Rio de Janeiro, 2006.
[2] Associação Brasileira de Normas Técnicas. NBR 6122. Projeto e Execução de Fundações. Rio de Janeiro, 2010.
[3] N. S Godoy. *Class Notes - Foundations*, Escola de Engenharia de São Carlos – USP, 1972.
[4] A. H. Teixeira and N. S. Godoy. *Fundações: teoria e prática*. Pini, 1996.
[5] M. A. Stroud. "The standard penetration test – it's application and interpretation". In: Geotech. Conf. On Penetration Testing, London, 1989.
[6] J. C. A. Cintra, N. Aoki and H. H. Albiero. *Tensão Admissível em Fundações Diretas*. RiMa, 2003.
[7] F. J. A. Luzzardi. *Análise De Confiabilidade Das Fundações Do Cais Do Estaleiro Honório Bicalho No Porto De Rio Grande (RS) Aplicando Metodologia Bayesiana*. Master thesis, Universidade Federal do Rio Grande – FURG, 2015.
[8] N. U. P. Silva. *Análise de tensões verticais em edifícios de alvenaria estrutural considerando a interação solo-estrutura*. Master thesis, Universidade Federal de Minas Gerais. 2006.
[9] M. D. Wegmann. *Análise da interação solo-estrutura em Dolfins de amarração e atracação*. 2018, Master thesis, Universidade Federal do Rio Grande – FURG, 2018.
[10] S. Gowthaman and M. C. M. Nasvi. "2D and 3D Numerical Simulation of Load-Settlement Behaviour of Axially Loaded Pile Fondations". *American Journal of Civil Engineering and Architecture*, vol. 5, n. 5, p. 187-195, 2017.
[11] R. B. J. Brinkgreve and W. Broero. PLAXIS 2D v.AE: Manual. Delft University of Technology & PLAXIS b.v., The Netherlands, 2004.
[12] K. Zakia, B. Abdelmadjid and T. Nazihe. "Modeling the Behaviour of Axially loaded and laterally loaded pile with a contact model". *Electronic Journal of Geotechnical Engineering*, vol. 16, pp. 1239-1258, 2011.