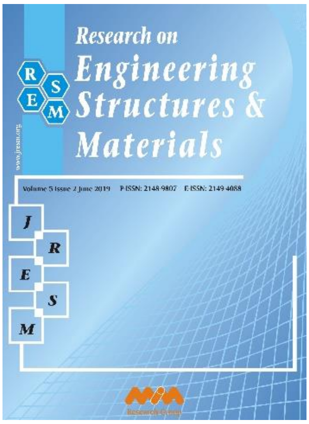
Research on
Engineering
Structures & Materials
journal homepage: http://jresm.org
RESEARCH GROUP



Estimation of the piles axial load in the Public Wharf of Porto Novo during an operation of a port crane

Matheus de Lima Nunes, Mauro de Vasconcellos Real

Online Publication Date: 07 Aug 2019
URL: http://www.jresm.org/archive/resm2019.113ms0209.html
DOI: http://dx.doi.org/10.17515/resm2019.113ms0209

Journal Abbreviation: *Res. Eng. Struct. Mater.*

To cite this article
Nunes ML, Real MV. Estimation of the piles axial load in the Public Wharf of Porto Novo during an operation of a port crane. *Res. Eng. Struct. Mater.*, 2019; 5(4): 347-353.

Research on Engineering Structures & Materials
journal homepage: http://jresm.org

Research Article

Estimation of the piles axial load in the Public Wharf of Porto Novo during an operation of a port crane

Matheus de Lima Nunes[*, a], Mauro de Vasconcellos Real[b]

Post-graduation program in Ocean Engineering, Federal University of Rio Grande – FURG, Brazil.

Article Info

Article history:
Received 09 Feb 2019
Revised 11 May 2019
Accepted 07 Aug 2019

Keywords:

Port Structures;
Computation modeling;
Wharf design;
Finite Element Method

Abstract

Port structures are usually large and are subject to heavy loads such as the stresses caused by a port crane and other equipment used during a port operation. Therefore, for a meticulous analysis of the internal forces of the port structures, with the intention to optimize and turn more effective the design, is essential the use of computational tools and advanced mathematical methods, as, for example, the Finite Elements Method. The Porto Novo of the Port of Rio Grande moves millions of tons of materials per year, becoming one of the main ports in Brazil. Its structure has gone through two stages of modernization making it possible the use of larger and more efficient port equipment, operation with ships of greater load capacity, and more quantity of mooring berths available for simultaneous operation. This study will be intended to perform the computational modeling of the Public Wharf of Porto Novo of the Port of Rio Grande using the finite element method through the ANSYS software to estimate the axial loads of the wharf piles when a Mobile Port Crane is in port operation.

1. Introduction

The maritime transport sector is growing, with bigger ships and equipment to support the terminals demand. Because of this, it becomes necessary that existing docking structures be revitalized, expanded, and the construction of new cargo terminals [1].

Port is an area for mooring boats and ships, serving as shelter from waves, winds, and currents, allowing access to the coast, aiming at port operations, by docking points. For this activity to occur safely, these areas must have mooring points for the vessels work with minimal efforts and displacements during port operations and fender systems to protect both ship and structure from collision damage. Quay structures are usually large constructions subjected to high-intensity forces [2].

Therefore, to perform a reliable design and with a low-cost construction, it is essential to carry out a rigorous structural analysis with the use of advanced mathematical techniques such as the finite element method (FEM) and the use of computational tools to solve complex structures.

The public wharf of Porto Novo is located in the city of Rio Grande / RS, and it is among the main ports in Brazil, moving millions of tons of materials. Its area has the intended use for military, tourism, roll-on/roll-off, bulks, fertilizers, and container ships. Its structure was modernized in the extension of 1575m of its wharf and increased the depth of mooring berths. This work was carried out in two stages; the first one finished in the year 2004 (450m) and the second one in 2017 (1125m). The objective of this research is to perform

[*]Corresponding author: matheusnunesrg@gmail.com
[a] https://orcid.org/0000-0002-9723-8175; [b] orcid.org/0000-0003-4916-9133
DOI: http://dx.doi.org/10.17515/resm2019.113ms0209

the computational modeling of the public wharf of Porto Novo of the Port of Rio Grande, using the finite element method through the ANSYS software to investigate the behavior of the structure when it's on the effect of a mobile port crane (Gottwald HMK280E) during a port operation.

2. The computational model

Each wharf modules have the following dimensions 75.00m x 11.20m and the characteristic compressive strength of the concrete used was 40MPa. The infrastructure consists of:

- A line of 12 mixed piles of reinforced concrete with ø71.12 cm (28 ") metal jacket, in the sea parallel to the quay line.

The union of precast structures and in-situ structures in reinforced concrete form the platform (Fig. 1) of the wharf:

- A longitudinal beam called V1, located in the front part of the platform and parallel to the line of the wharf (built part precast concrete and another part concreted in-situ), which transfers its forces to the line of piles;
- A second longitudinal beam called V2 (made in-situ), located on the back of the wharf. Its forces are transmitted to the upper part of the existing quay-wall using sliding supports;
- 22 pre-casted slabs with Pi cross-section;
- 03 blocks built in-situ where the fenders and mooring structures are located;
- A cantilever slab built in-situ after assembly of V1 beams and slabs type Pi.

Also, exist in the posterior part of the platform a vertical plate for soil containment and mobilization of frictional forces and a pavement plate, but they were not considered in the computational model. The construction of the computational model of the structure was performed using ANSYS software, which uses the finite element method for structural analysis.

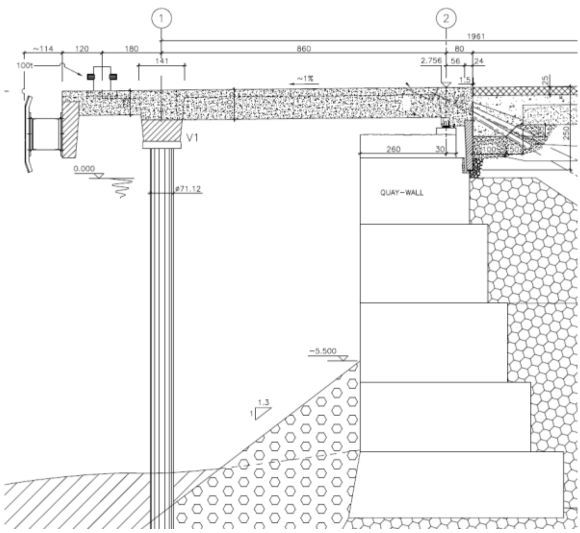


Fig. 1 Cross-section of the wharf

2.1. Platform Modeling

For the modeling of beams V1 and V2, the element BEAM189 (Fig. 2) was used. It is a quadratic element of 3 nodes, having 6 degrees of freedom in each node (displacements in x, y and z, and rotations in x, y, and z). The element BEAM189 is suitable for linear analyzes and can be applied to slender or robust beams.

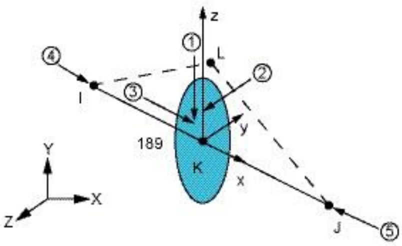


Fig. 2 The element BEAM189 [3]

It was necessary to divide the Pi slabs into one slab, and two beams, because Ansys don't have in its geometry library this type of cross-section (Fig. 3).

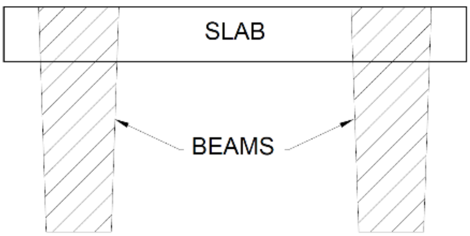


Fig. 3 Pi cross-section divided into three parts

For the beams of Pi cross-section, it was used the element BEAM189 and for the slab the element SHELL281 (Fig. 4). It was used rectangular elements of 8 nodes, having 6 degrees of freedom in each node (displacements in x, y, and rotations in x, y, and z). SHELL281 also used for the cantilever slab and Blocks. This element is suitable for analyzing thin to moderately-thick shell structures and for linear analyzes.

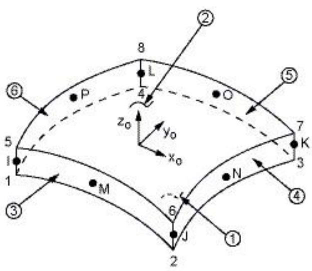


Fig. 4 The element SHELL281 [3]

2.2. Piles Modeling

The piles have a diameter of 71.12 cm and were also modeled using the BEAM 189 element. It was simulated the soil-structure interaction assuming a length, where a pile could be adopted as being perfectly fixed in the soil [4, 5].

$$L_e = L_S + L_U \quad (1)$$

$$L_S = 1.8 \cdot L_0 \quad (2)$$

$$L_0 = \sqrt[5]{\frac{E_P \cdot I}{n_h}} \quad (3)$$

Where L_e is the Cantilever length, L_s is the Buried length, L_U is the free Length of the pile (unearthed), L_0 is the Fictitious length, E_p is Modulus of elasticity of the pile material, I is the moment of inertia of the cross-section of the pile, and n_h is Coefficient of the horizontal reaction of soil [4, 5]. The depth of all the piles was considered the same (Fig. 5).

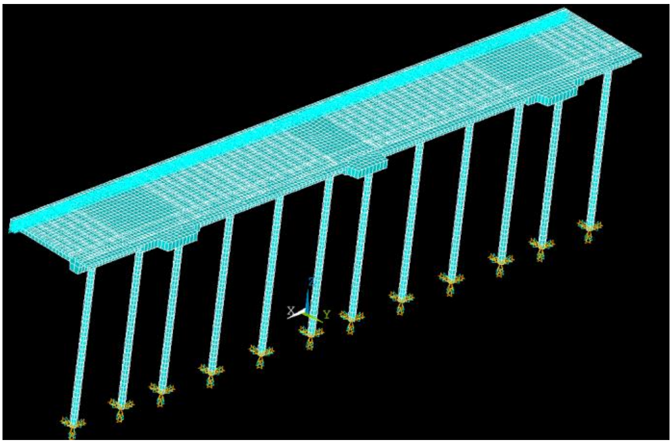


Fig. 5 The computational model

2.3. Test of Mesh Convergence

A mesh size verification test was performed to ensure that there is a convergence of the results for the elements used [6]. A test load was applied (Fz = -1000kN) in the center of the structure, carrying out simulations with mesh sizes 0.75m x 0.75m, 0.50m x 0.50m, 0.25m x 0.25m and 0.10m x 0.10m, in order to ensure that the elements used had a convergence of results.

2.4. Loadings

It was considered the deadweight of the structure, and a load of a mobile port crane model Gottwald HMK280E during a port operation. The crane can be placed in any location on the dock platform and on the deck that exists at the back of the dock. For the crane, several loading situations were tested on the quay with the boom positions of 45 °, 90 °, and 135 °, except for the 0 ° position, since the most loaded stabilizer pads are those arranged on the pavement slab, localized behind the wharf platform (Fig. 6).

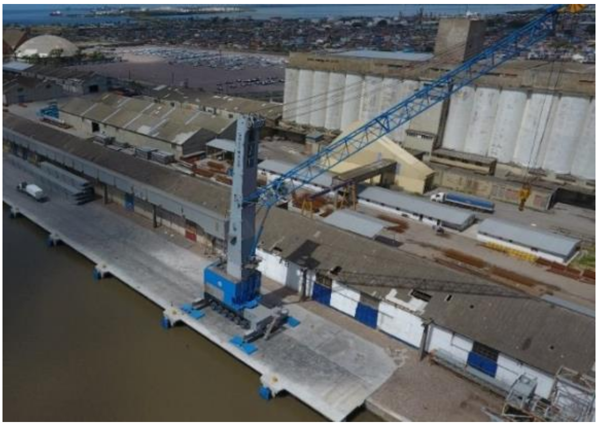


Fig. 6 Mobile Port Crane

Table 1 presents values of stabilizer pads loadings for the different positions of the boom in the quay. The stabilizer pads E1 and E2 are arranged on the pavement, and the E3 and E4 are that are on the quay platform (Fig. 7).

Table 1 - Values of stabilizer pads loadings for the different positions of the boom.

Boom angle	E1 (kN)	E2 (kN)	E3 (kN)	E4 (kN)
0°	1410	610	610	1410
45°	620	490	1140	1790
90°	420	500	1670	1460
135°	410	840	1990	800

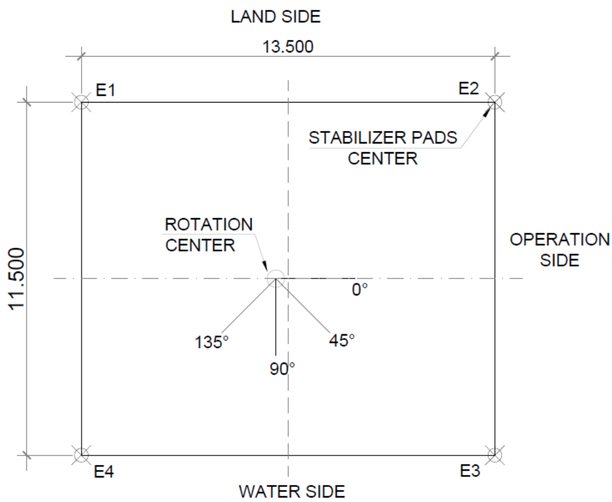


Fig. 7 Sketch of center points of the stabilizer pads and boom angle

4. Results and Discussions

The results obtained in the mesh convergence test were identical for the four mesh sizes, for that reason, the mesh with larger size the values obtained is already satisfactory. However, to get the model mesh with better precision, a value of 0.50m x 0.50m was used for the final simulation mesh.

It was simulated several positions for the port crane during operation, and the worst result is when the following situation occurs:

- The crane is in operation closest to the right corner of the dock;
- The operating side is on the left;
- The rotation center is on the left of the crane axis;
- The front stabilizer pads are over the beam V1 axis, with the stabilizer pad E3 is closest to the right edge of the wharf;
- With the boom operating in the 135° position.

In this case, was obtained the maximum value, which is a compression force of 3164kN for E12 pile (Table 2).

Table 2 Axial load (N) values of piles for the worst operating scenario of the crane

Pile	E1	E2	E3	E4	E5	E6	E7	E8	E9	E10	E11	E12
N (kN)	933	1193	1202	1031	1009	1129	1125	1008	1114	1443	1436	3164

The maximum Z-displacement of the structure for the worst scenario occurs in the region of the slab in the front part of the right edge of the quay, near the crane stabilizer pad E3, with a value of 3mm (Fig. 8).

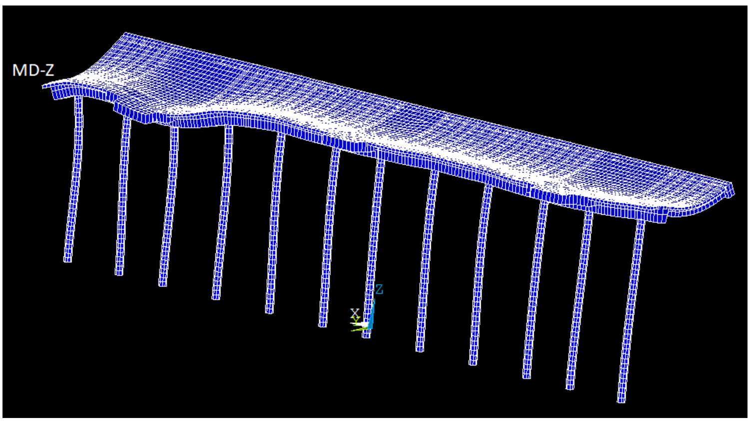


Fig. 8 Color graph of displacements in the worst condition

5. Conclusion

In this study, the computational modeling of the Public Wharf of Porto Novo was carried out with the objective of verifying the axial forces on the piles to the worst position of a port crane during an operation. The results obtained from the mesh convergence, demonstrate the crane load did not show variation for the four sizes of the element used. It is concluded that the computational model performed in the ANSYS software proved to be quite satisfactory. The worst load situation was obtained when the crane is located on the front of the platform, with the operation side in the same module where the crane is located.

References

[1] Ranga Rao AV, Sundaravadivelu R. A knowledge-based expert system for design of berthing structures. Ocean Engineering, 1999, 26, 653–673. https://doi.org/10.1016/S0029-8018(98)00013-4
[2] Alfredini P, Arasaki E. Obras e Gestão de Portos e Costas. Blucher, São Paulo, SP, Brazil, 2009.
[3] Ansys. Basic Analysis Guide.Ansys Inc, Canonsburg, PA, USA, 2009.
[4] Davisson MT, Robinson K.E. Bending and buckling of partially embedded piles. In 6th ICSMFE, 1965.
[5] Urbano AR. Dimensionamento de Fundações Profundas. Blucher, São Paulo, SP, Brazil, 2003.
[6] Viegas CHH, Real MV, Moura MW. Structural analysis of a container pier through the finite element method. Revista de Engenharia e Tecnologia, 2015, 7: 1 – 12.