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

Numerical Modeling of Post Tensioned Concrete Slab

Matheus S. V. Fernandes[1], Maria Elizabeth T. da Nóbrega[2], André Tenchini da Silva[3]

[1] *Structural Engineering Department, UERJ, State University of Rio de Janeiro*
Sao Francisco Xavier street, 524, 20550-900, Rio de Janeiro, Brazil
msvfernandes@yahoo.com.br
[2] *Structural Engineering Department, UERJ, State University of Rio de Janeiro*
Sao Francisco Xavier street, 524, 20550-900, Rio de Janeiro, Brazil
etavares@uerj.br
[3]*Structural Engineering Department, UERJ, State University of Rio de Janeiro*
Sao Francisco Xavier street, 524, 20550-900, Rio de Janeiro, Brazil
tenchini@eng.uerj.br

Abstract. The fragile failure of concrete is a complex problem where reinforcement steels are employed at regions subjected to tension. The other important solution in order to improve the structural behavior corresponds to inclusion of prestress steel reinforcement in concrete. This introduces a precompression avoiding cracks during service lifetime. Nowadays, post-tensioned concrete remains not fully controlled where the numerical modeling plays a relevant role to improve the knowledge concerning to structural behavior. Therefore, this paper focus on numerical evaluation of the post tensioning influences on the global behavior of concrete slabs. The numerical model has been created taking into account the experimental results. In details, it was modeled one simple support concrete slab that was carried out up to its ultimate loading. This test measured important aspects, such as deflections, yield lines, and cracks development. The specimen has a square geometry and uses straight prestressing wires. The numerical model has been developed using the Abaqus software. It is known that the standard solver could not advance much after cracking load because of many nonlinearities introduced. The viscosity parameter on the concrete damaged plasticity model was the only way to pass through this point, but this technique weakens the results and raises doubts about it. An attempt with explicit solver has been used to overcome those nonlinearities during the analysis and showed good results. Contact problems between concrete and steel wires, concrete damage plasticity modeling, and explicit analysis require to scrutinize many topics during pre- and post-processing analysis. This paper presents the whole process of modeling a post-tensioned concrete slab using Abaqus Explicit taking into account its applicability, advantages, disadvantages and the validation procedure. This paper presents a method to identify the cracking load in an Explicit analysis based on the plastic deformation energy.

Keywords: Prestressed Concrete, Concrete Slab, Numerical Analysis.

1 Introduction

Experimental analyzes are accurate approaches to study the behavior of particular structures with controlled parameters. The high costs and the significant amount of time invested to develop a prototype are barriers to the widespread use of those analyzes. Hence the technique of numerical modeling arouse, which is a mathematical approach to analyze real structures. A numerical application on basis of finite element method is granted accountable when there is a classical analytical formulation or an experimental test to validate it. Afterwards, through the validated numerical results it is possible to carry out a parametric study with low lost of accuracy and huge amount of information.

Nawy [1] were one of the first researches to test a slab-column connection of prestressed flat slab subjected to gravitational uniform load. These kind of tests have been developing to a combination of gravitational and lateral loading, and nowadays it evaluates the behavior of that connecton during seismic events [2]. Furthermore, more and more complex numerical tests were done with the advantage of state-of-art software. Concrete models are one example that have not taken for granted this aid, works as Souza [3] modeled a reinforced concrete beam reinforced with FRP and Ombres [4] modeled a FRC on mansory columns. However, not many prestressed concrete numerical studies had been done so far.

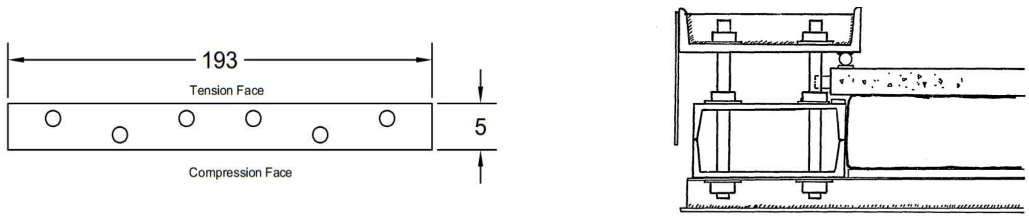
2 Experimental Test

Due to the increasing demand of post-tensioned structures and the need to understand their behavior, a numerical study of a post-tensioned flat slab with unbonded wires was carried out. The experiment of Kemp [5] was the base for this article and he carried out three experiments of post-tensioned flat slabs, tensioned on both directions, and with unbonded wires. The first two slabs had eight strands in each direction and the third one had six strands. All were test up to failure load, and their strains and displacements were measured.

The first and the second experiments were conducted normally, but they presented some technical difficulties. Whereas, the third experiment had those problems fixed and was the chosen slab to be modeled. This modeling aimed to investigate the load *versus* displacement curve, the development of cracking, the cracking load, and the post-peak behavior.

The experimental model is a 193 cm square flat slab and 5 cm deep. The distance between supports, however, is 188 cm. The prestressing wires are straight, 0.7 cm diameter and are spaced at 30,4 cm center-to-center on both directions. Strand crossing is a usual problem during modeling and to avoid interference between them, the strands were centered in one direction, and on the other direction they were altered top and down (Figure 1-a), where four strands were on the tension face and the other two on compression face.

The average strength of concrete cylinders were 43.4 MPa, modulus of rupture equal to 3.7 MPa, and in accordance to ACI 318 [6], the modulus of elasticity was 34.4 GPa. The prestressing system were Freyssinet mono-wire. The wires were low relaxation, the proportional limit was 1,651.1 MPa. The offset at 0.2% was at 1,544.4 MPa and failure stress was 1,689.2 MPa with 5% elongation.



(a) (b)

Figure 1. Wire Position (Dimension in cm) (a), Slab Support Details. (b)

Wires were stressed at 67% of failure stress and was jacked at 40.92 kN. The average stress on concrete was 2.76 MPa, after all losses the average stress reduced to 2.59 MPa. The anchorage system were made of steel with modulus of elasticity of 200 GPa.

The supports system is illustrated on Figure 1-b and was idealized as a simply supported condition. Nevertheless, this kind of support couldn't ensure a simply supported condition throughout the whole test, for twisting moment lifted up the corners of the slab. This led to the necessity of modeling the slab supports on its border.

Another problem mentioned by the author occurred during curing and drying. The slab curled after curing and it could have influenced in some ways. For instance, due to curling the slab behave as point supported up to 10% of the applied load, which represents approximately 28.5% of cracking load. The curling raised another concern about the integrity of concrete and how much it affected the concrete modulus of elasticity.

3 Numerical Model

3.1 Material Properties

The concrete model was based on Alfarrah's work [7]. It was developed specially for the concrete damage plasticity model that is already implemented on Abaqus [8]. Also, Alfarrah [7] concrete model is mesh sensitive and diminishes occurrence of problems related to the size of a finite element. The compression and tension behaviors, as well, the compressive and tension damage propagation were all modeled as Alfarrah [7] suggested and can be seen on Figures 2-a, 2-b, 3-a, and 3-b. The concrete plasticity parameters are shown on Table 1.

Table 1. Plasticity Parameters

Dilation Angle	Eccentricity	fb0/fc0	K	Viscosity Parameter
38	0.1	1.16	0.667	0

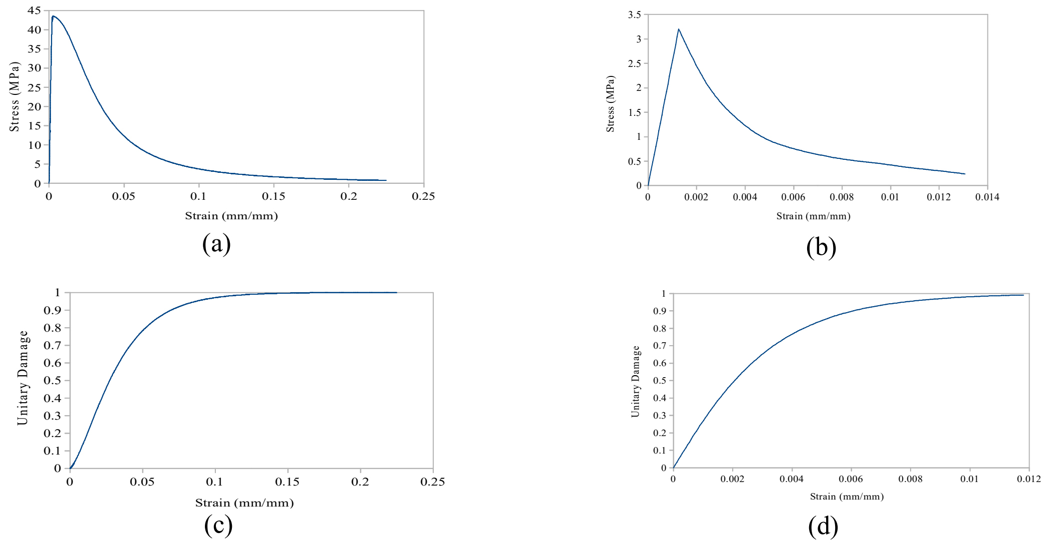


Figure 2. Concrete Behavior. (a) Compression (b) Tension, Unity Damage *x* Strain Curve. (c) Compression (d) Tension

The prestress steel behavior was modeled as a bilinear curve, where the first branch is the linear elastic response and second branch represents the inelastic response. The prestress jacking force was applied as a temperature gradient on the wires, using eq. 1.

$$\Delta T = \frac{F}{A E \alpha} \tag{1}$$

3.2 Mesh and Contact

To avoid slab/wire overclosure during analysis, every hole were made 15% bigger than the wire diameter. When wires crossed each other they got a single point in common, that caused problems during the partition of the slab. The slab was partitionate through its depth at the hole middle plane and perpendicular holes intersection.

The wire-to-anchor plate and slab-to-anchor plate contacts were modeled as Tie Constraint. The contact between slab holes and wires were modeled as general contact and their contact properties were tangential behavior with friction formulation as Frictionless, and normal behavior with pressure overclosure as Hard Contact, constraint enforcement method as Default, allowing separation after contact. The contact between slab and support had the same configurations.

The model had many parts where the finite elements were distorted, so to scale down this problem those areas were densified with reduced integration elements, instead of using smaller quantity of complete integration elements. The element used for all parts was the C3D8R from Explicit Library and their mesh size were: 12.7 mm for slab; 20.3 mm for wires; 12.7 mm for supports; and 15.4 mm for anchor plates. There were 348,096 elements and the analysis took 6 days to complete.

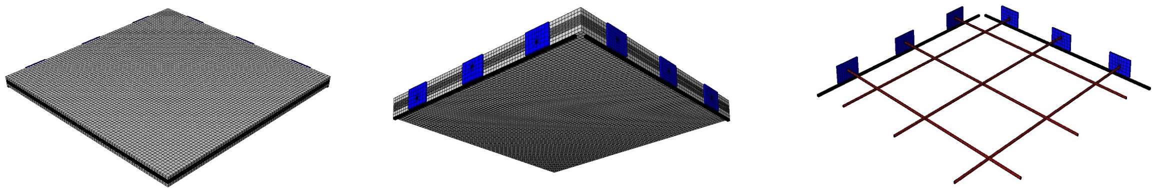


Figure 4. Meshed Parts

3.3 Boundary Condition

Running all analysis required four steps: self weight, two prestressing and applied load. Including the self weight was necessary because the edges were free to move, only resting on supports. Thus, it avoided the possibility of slab and support lose contact, preventing any unexpected edge lifting.

The support boundary condition had all displacement fixed on the face that did not touch the slab. It was used symmetry on both directions of the slab to reduce the size of the problem. The self weight was applied as a uniform pressure on the top surface with magnitude of 1.17 kN/m² and the applied load was also a uniform pressure on the top surface with magnitude of 68.95 kN/m². The gradient of temperature applied on the wires was 480.65 Celsius degree with section variation constant through region.

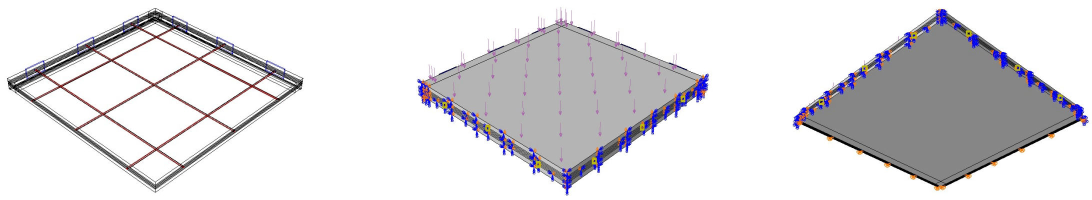


Figure 5. Boundary Conditions

3.4 Nonlinear Method

Its was developed a finite element analysis using ABAQUS. It is a complex analysis for many nonlinearities present in the model, for instance, contacts, concrete behavior and prestress steel behavior. Standart solver faced a lot of problems to overcome those nonlinearities and only through numerical tweaks became possible to run the analysis. Thus, it was decided to use the Explicit solver that allows those nonlinearities be overcome, as were done by [3] and [4]

An Explicit analysis has many peculiarities and must have a thorough control of the whole analysis. First, Explicit solver is not a static analysis one, rather a dynamic one. Therefore, the speed of the load must be controlled and this process is known as quasi static analysis. Besides the usual methods for controlling an analysis such as load *versus* displacement curves and comparison in between crack propagation of experimental and numerical models, it is necessary to control the artificial, kinetic, and internal energies.

The applied steps were: self weight, first prestressing, second prestressing, and applied load. Instantaneous amplitude is better suited for Explicit solver when temperature is applied, as the prestress jacking. Self weigh and applied load were assigned the smooth amplitude, that controls the loading speed and reduces the dynamic effects. The amplitude of self weight, prestressing and applied load were set to two, one, and five, respectively.

Mass scaling is a maneuver that helps to speed up the analysis and at the same time administer the dynamic effects. The mass scaling to self weight, prestressing, and applied load were respectively: 50, 100, and 25.

4 Results and post-processing

The results of the analysis are presented herein and they were confronted based on Kemp [1]. The identification of the cracking load based on two different approaches and they will be described. There are many methods to calibrate the numerical model, but only two will be described in this paper, crack mapping and stress in the wire.

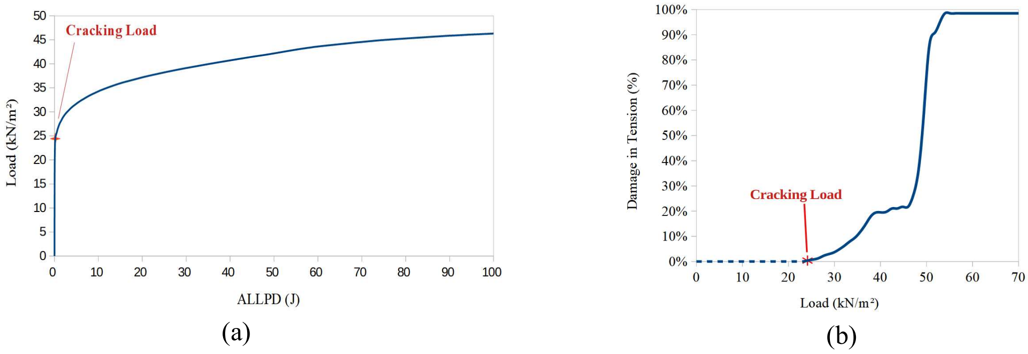


Figure 6. Load *versus* ALLPD (a); Load *versus* DAMAGET (b)

The energy of plastic deformation (ALLPD), Figure 6-a, "is the energy dissipated by rate-independent and rate-dependent plastic deformation" [4]. It is a great measure to indicate when the model starts to behave inelasticaly. On the Figure 6-a can be seen an inflection point at 24.7 kN/m² of applied load. This represents the moment where the whole model started to achieve plastic deformations. At this stage, neither wire nor compression on concrete were on inelastic behavior, only tension on concrete. On Figure 6-b is shown the development of damage in tension (DAMAGET) of concrete; it has attained the maximum damage at 24 kN/m². Those measures agree with each other and they inform the cracking load of numerical model.

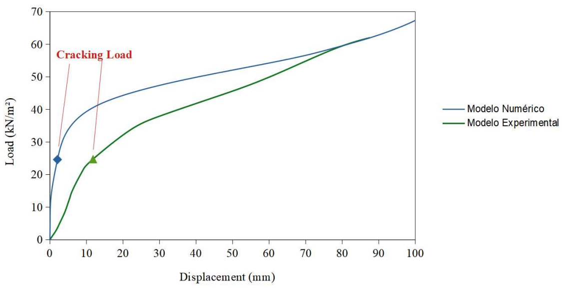


Figure 7. Load *versus* Displacement at center of slab

The load *versus* displacement, Figure 7, shows the behavior of both models at the center point of the slab. It was presented a certain level of agreement, for the cracking load occurred at same moment, further the closer they get to the ultimate load, they tend to follow the same behavior. Some problem that were mentioned at [1], might have had a major influence on results, for example, excessive shrinkage at curing curled the slab, changing the boundary condition during the beginning of the experimental test, and also it was not ensured a straight slab edge resting on supports. This last problem was cited by [1], as the main cause for the experimental results were not in good accordance to the elastic theory. Also, the elastic theory assumes that edges were not free to move.

Some parameters from the numeric model, may have influenced the result, as well. A good combination of mass scaling, time frequency, and mesh size is fundamental for they are proportional to the stable time increment. Also the concrete model for Explicit analysis are fundamental for attaining a good response from the model.

Strain gauges attached to the wires did not read significantly up to load of 31.0 kN/m². This can be seen on Figure 8, the most stressed wire; until Point C there was not any important change in stress, but after this point the stress in the wire increased exponentially. When the wire was stressed to 1,544 MPa, it started to behave inelasticaly. This was reported by [1], “wires were permanently bent at the end of test”.

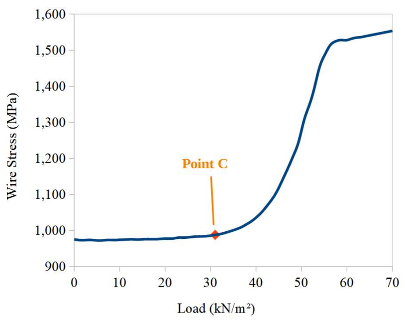
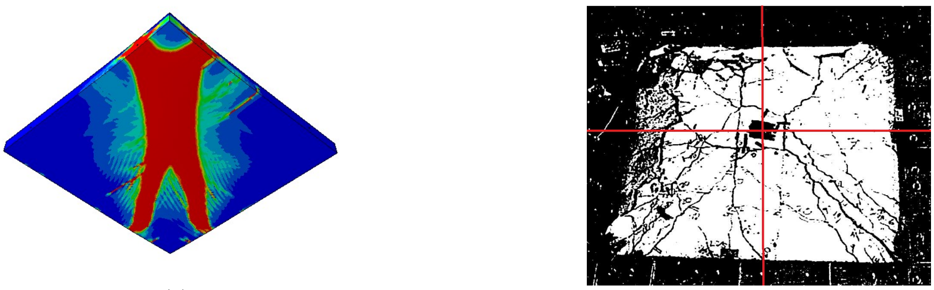


Figure. 8. Stress in the wire - Numerical model



(a) (b)

Figure 9. Crack Mapping Comparison – Numerical (a) and Experimental (b) Models

(a) (b)

(c) (d)

Figure 10. Cracking Develompment (38%, 57%, 71%; and 85% of ultimate load – a; b; c; d)

The cracks did not originate from center, as usual for reinforced concrete slabs, but rather from the corners of a 30.5 cm square central crack pattern, as can be seen on Figure 9. Figure 10 shows the development of cracking throughout the load application.

5 Conclusions

Problems such as shrinkage and unproper boundary conditions during an experimental test have great influence on results. On the other hand, some numerical problem may also present difficulties on achieving good agreement. Nevertheless, it was showed that some parameters aid on validating a numerical model. In Explict analysis is mandatory to control the energies during the whole process. The difference between Internal Energy (ALLIE) and Kinetic Energy (ALLKE) may not exceed 10% [8], and the Artificial Strain (ALLAE) may not exceed 1% to 2% of ALLIE [8].

The energy of plastic deformation (ALLPD) showed as good indication to identify the cracking load, specially in an Explicit analysis. Using other parameters, such as, DAMAGET aided on this search too. Therefore, the measure of cracking load through the Plastic Deformation Energy is good way to validate the numerical model. The crack pattern showed very good agreement to experimental analysis and its a good validation procedure.

Explicit analysis is a good approach to complex numerical problems, but it has some difficulties and should be dealt with a lot of care. However, there are many ways to control the analysis and with right choices, it can be done safely. Having an experimental program will help on the validation process. Also, controlling energies are excellent practice to understand the behavior of the model.

Acknowledgements. The authors would like to thank CAPES (Finance code 001), CNPq (305026/2017-8) and FAPERJ (E-26/203.192/2016, E26/010.1685/2016; E-26/010.100651/2018) for the financial support to this research program..

References

[1] E. G. Nawy, P. Chakrabarti, "Deflection of Prestressed Concrete Flat Plates", PCI Journal March-April, 1976.
[2] S. W. Han, H. W. Kwon, C. S. LEE, "Seismic Performance Evaluation fo Gravity-Designed Flat Plate Frames", 2013.
[3] K. L. B. Souza, "Análise Numérica da Influência do Material de Reparação em Vigas de Concreto Armado Recuperadas", Ms. Thesis, 2018.
[4] L. Ombres, S. Verre, "Numerical Modeling Approaches of FRCMs and SRG Confined Mansory Columns", 2019.
[5] G. J. Kemp, "Simply Supported, Two Way Prestressed Concrete Slabs Under Uniform Loads", Ms. Thesis, 1971.
[6] ACI 318-14, "Building Code Requirements for Structural Concrete", American Concrete Institute, 2014.
[7] B. Alfarrah, F. López-Almansa, S. Oller, "New Methodology for Calculating Damage Variables Evolution in Plastic Damage Model for RC Structures", 2016.
[8] Abaqus, "Abaqus Analysis User's Manual", 2014.

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.