XL CILAMCE
IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING
NOVEMBER 11-14, 2019
Praiamar Natal Hotel & Convention
Natal, RN-BRAZIL

NUMERICAL SIMULATION OF CONCRETE STRUCTURE PILE CAP ("D REGION") USING CONCRETE DAMAGED PLASTICITY AND NON-LINEAR ANALYSIS

Guilherme Niclewicz Saddock de Sá
Roberto Dalledone Machado
guilherme_niclewicz@hotmail.com
roberto.dalledonemachado@gmail.com
Federal University of Parana - Numerical Methods in Engineering Graduate Program
Avenida Coronel Francisco Heráclito dos Santos, 210 - Jardim das Américas, Curitiba/PR - Brazil, Zip-Code 81531-970

Abstract. The design of reinforced concrete structures aims to define the amount and distribution of steel rebars (reinforcement) required in the elements. In complex situations, the Strut and Tie Method is used for design parts that are subject to complex stress distributions (known in the literature as "D Regions"). One of the difficulties in the application of the method is the determination of the most probable distribution of tensions that the part will suffer, since it depends on its own geometry and the amount of reinforcement used. For this reason, on an initial step, elastic analyzes can be used to suggest a probable field of stresses to be used in the definition of the Strut and Tie Model. However, in the ultimate strength, the stress distribution can be affected by non-linear effects like non regular constitutive relations of the materials (concrete and steel), by damage processes that the part will suffer and also by the distribution of reinforcements. The objective of this work is to present nonlinear computational analysis of several elements of reinforced concrete, considering the distribution of reinforcement, in order to evaluate its real behavior in the ultimate state, and to compare with the results predicted by the usual methods of design, mainly the Strut and Tie method. In order reach this objective, the models of reinforced concrete are developed through the software of finite elements ABAQUS, considering Concrete Damage Plasticity. The models constructed on this paper consider the reinforcement distribution, including the constructive ones, trying to evaluate its influence in the field of tensions and its resistant capacity. Several situations are examined and the results are compared with the conventional design methods or with the Strut and Tie Model. The developed examples confirm that the stress field and the resistance capacity are affected by the reinforcement distribution. In this way, the present work intends to discuss how the Strut and Tie models are affected by non-linear behavior and by the designed reinforcements.

Keywords: Reinforced Concrete, Non-linear Analysis, Strut and Tie method, Concrete damage plasticity, D regions.

1 Introduction

Reinforced concrete is a compound material formed by the union of structural concrete and rebars of steel (called as “reinforcements”). When we analyze these two components, we notice that the response of steel is simple and easy to predict, this material has a linear and constant relationship between stress and strain in most part of his diagram (for both compressive and tension stresses). However, the concrete has some unique and irregular properties. This material is an heterogenous mixture of larges aggregates (gravel), small aggregates (sand), cement, water and additives. The concrete behavior is mostly non-linear, and its answer depends on the distribution of the components inside the element, factor that is very hard to control during the concreting. Add to that concrete has different responses to tension and compression stresses, usually the ultimate strength on tension is about 10% the resistance in compression stresses.

Since reinforced concrete has a high level of heterogeneity and a complex internal structure, it is possible to perceive that simulate the real behavior of concrete structures is not a trivial task. A proper model has to take in consideration some classic behaviors like: an initial elastic behavior, inelastic phase (when start to appear damage inside the internal structure of the material – due to cracking and micro-cracking), tension stiffening on the rebars, tension softneing (that represents the post peak phase – when tension is redistributed to other points of the structure), the response of the element on the deformed stage (factor that have a lot of influence on the ultimate strength of structures) and of course the tension behavior of concrete. We see that to construct a perfect finite element model its necessary to perform a lot of materials tests to get data and then implement a good constitutive rule to the specific material used on the models.

There are a lot of different techniques that can be used through the finite element method (FEM) to simulate this kind of problem. On this paper it will be presented a simulation, developed on a commercial FEM package called ABAQUS, of a reinforced concrete element. The model has implemented on it some techniques based on concrete damaged plasticity fundaments that will be explained on the next items. To describe the behavior of the materials it will be used two numerical stress-strain relations for concrete (compressive and tension) that will be present next and a classic bi-linear constitutive model for the steel reinforcement.

The main goal of this work is to predict the reserve of strength that the studied element has compared to the designed ultimate strength predicted by the regular methods found on international codes, specially the strut and tie method. For that it will be taken in consideration the amount of steel and its position inside the concrete. The results of the model will be compared with an experimental test developed by Campos (2007) [1].

2 Developing the constitutive models

2.1 Constitutive model for concrete

2.1.1. Concrete in tension

To a polished analysis of non-linear concrete structure, it is necessary to take in consideration the resistance of this material in tension. Usually, on international codes, most of the design methods do not take it in consideration. This happens because this material has not a good response in tension, so every stress of that nature is resisted by the reinforcement steel design.

Plain concrete submitted on tension stress has a brittle answer, but when we combine this material with steel it gains ductility and occurs two types of factors that have to be considered on the analysis to get more realistic results.

First one is the tension stiffening effect on the rebars, this phenomenon happens due to the bond action between the concrete and steel and by the gain of resistance on a flow regime, how it is explained by Noghabai [2]. This grants an extra reserve of strength to the element.

The second factor that have to be taken in consideration is the tension softening behavior of the concrete. When this material is combined with steel, its resistance does not drop studently (on a brittle behavior), instead that, it drops smoothly redistributing the high-tension concentrations to the rebars. This grants ductility to reinforced concrete elements.

On the present paper it will be used the model proposed by Nayal and Rasheed [3] and adapted by Wahalathanri et al. [4]. The authors defend that this model take in consideration both factors described previously plus the bond slip effect between concrete and steel on an approximated way. On the Fig. 1 it is shown the original and the adapted relationship of the concrete in uniaxial tension stage. Wahalathanri et. al [4] explain that the minor modifications on the constitutive model were made to avoid some problems with numeric convergence that happens when implementing on FEM. On both diagrams is possible to find a primary cracking stage that have an accentuated drop of resistance and a secondary cracking stage that drops on a smother way compared to the first one (due to redistribution of tension from the concrete to the steel).

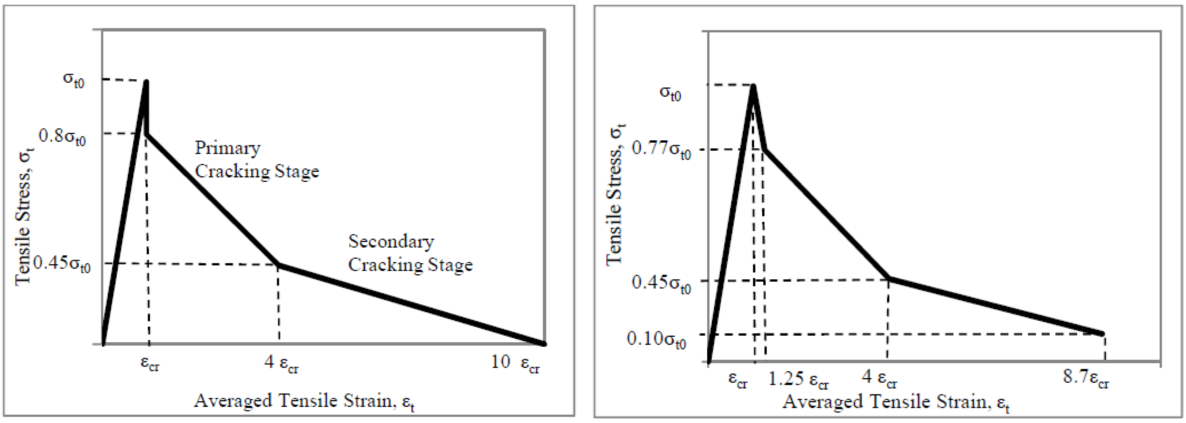


Figure 1. Original and adapted constitutive model. [4]

On the diagrams above σ_{to} is the maximum resistance on tension, designed by the NBR – 6118/2014 (Brazilian Code) by the following equation:

$$\sigma_{to} = 0.3 * \sigma_{cu}^{\frac{2}{3}} \tag{1}$$

Where σ_{cu} is the characteristic resistance of the material in compression. The limit deformation on concrete (peak stress) on tension is defined by the equation:

$$\varepsilon_{cr} = \frac{\sigma_{to}}{E_0} \tag{2}$$

Where E_0 is the Young Modulus for concrete that can be checked on equation (6).

2.1.2. Concrete in compression

The data necessary to implement a constitutive model for concrete submitted to uniaxial compression can be extracted by testing some cylinders specimens on a classical uniaxial compression test. But this material has a lot of sensibility when submitted on a multi-axial state of stress. This fact makes a lot more difficult to implement constitutive models for concrete, because it is necessary a lot more tests and data to describe the full behavior of the material.

On Fig. 2 it is illustrated a diagram of a specimen tested on a uniaxial compression state, presented by Kotsovos [5]. On this practical experiment we can perceive that the center of the cylinder, marked by the letter C, is subjected to a pure axial stress. It is possible to understand that the ultimate strength of the tested body was limited by the strength of this central zone, that happens because the friction between the cylinder and the mechanical press put the concrete on the edge (marked by the letter E) on a multi-stage of stress, granting it an addition of resistance.

It is known that when concrete is subjected by a confining pressure, like on the edge spots of the specimen on uniaxial stress, it gains an increase in its ultimate strength. Theoretically when the axial stress is equal the confining pressure (situation called in literature by hydrostatical pressure) the concrete has not an ultimate strength limit.

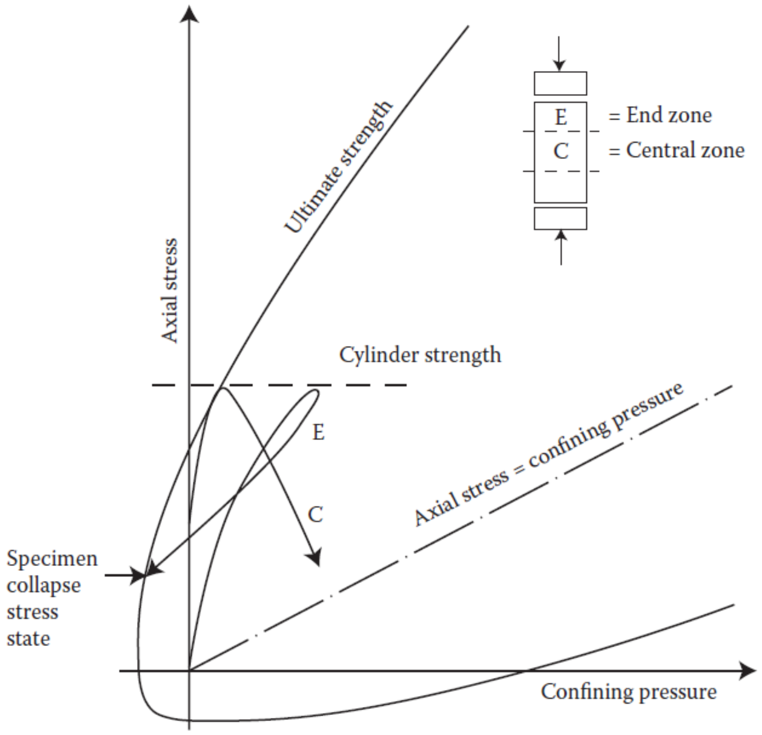


Figure 2. Schema of stress for a specimen tested on uniaxial compression essays [5]

The numerical constitutive model used on this paper's simulation for concrete on compression was the same used by Wahalathanri et. al [4], developed by Hsu Hsu [6] on 1994. This method, indicated for concrete with maximum strength up to 62 MPa, will be briefly explained on the next paragraphs.

The proposed stress strain relation by Hsu Hsu [6] has, on the beginning of the diagram, a linear response until the stress reaches 50% the ultimate strength (σ_{cu}), so on this part of the diagram the material obeys Hooke's law – Fig. 3.

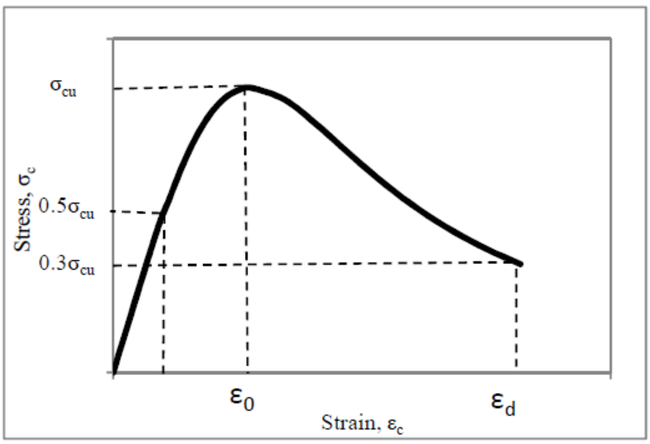


Figure 3. Numerical model for concrete proposed by Hsu Hsu. [4]

The second ascendant part of the diagram (from 0.5 σ_{cu} to σ_{cu}) presented on the Fig. 3 represents a non-linear behavior of the material, where the Young Modulus vary due to plastification and the formation of cracks. The third part of the diagram (descendent part) represents the softneing characteristic that happen after the ultimate strength (σ_{cu}) is reached. Both second and third part of the diagram can be modeled by the equation bellow:

$$\sigma_c = \frac{\beta\ ({}^{\varepsilon_c}/_{\varepsilon_0})}{\beta - 1 + ({}^{\varepsilon_c}/_{\varepsilon_0})^{\beta}} * \sigma_{cu} \tag{3}$$

On the equation (3) ε_c represent the strain at the point where the stress is equal to σ_c. The deformation at the point of ultimate strength (ε_0) can be predicted by the following equation:

$$\varepsilon_0 = 1.290836 * 10^{-5} * \sigma_{cu} + 2.114 * 10^{-3} \tag{4}$$

The beta parameter presented on equation (3) can be found by the next formula (5), and it depends on the ultimate strength (σ_{cu}), the deformation relative to it (ε_0) and the initial Young Modulus (E_0) designed by the equation number (6).

$$\beta = \frac{1}{1 - [{}^{\sigma_{cu}}/_{(\varepsilon_0 E_0)}]} \tag{5}$$

The Young Modulus designed by the Brazilian Code (NBR 6118 – 2014):

$$E_0 = \alpha_e * 5600 * \sqrt{\sigma_{cu}} \tag{6}$$

To expand the constitutive model from the uniaxial state of stress to multi-axial state on ABAQUS, it is necessary to retrieve some data from tests like bi axial and tri-axial compression tests. For the simulation presented on this paper it will be used the standard recommended parameters to perform this expansion, since tests were not made to develop a unique constitutive model.

The first parameter (σ_{b0} / σ_{c0}) measures the gain of strength for concrete when subjected to a bi-axial state of compression. Some tests presented by Kotsovos [5] show that this amount is relative to 20% of strength addition Fig. 04. On the model was used the recommended value equal to 1,16 that represents the addition of 16% of strength when in bi-axial state of compression.

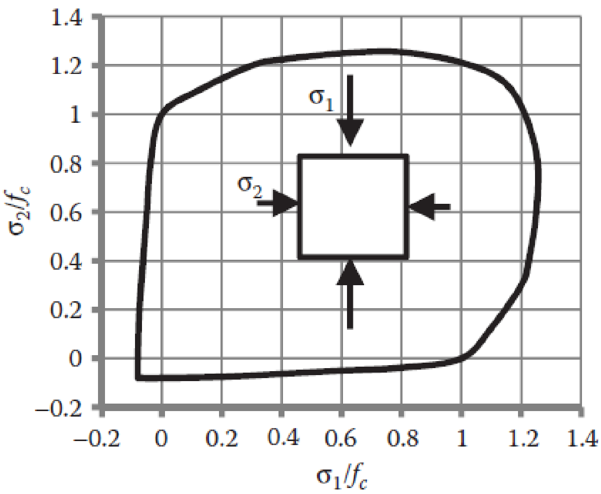


Figure 4. Complete envelope in bi-axial state of stress. [5]

The parameter K dictates the shape of the failure envelope on a tridimensional state of stress and it goes from 0,5 to 1,0. Abaqus User Manual [7] recommends to use this value equal to 0,6667 to the failure envelope gets the shape of Willian-Warnke Model – Fig 5 (extracted from Prates Aguiar [8] paper).

The Dilation Angle can be faced like the intern angle of friction of the material. Many authors studied this parameter and recommend to use a value between 32º and 41º for concrete. For the simulations presented on this paper it was used ψ = 36º.

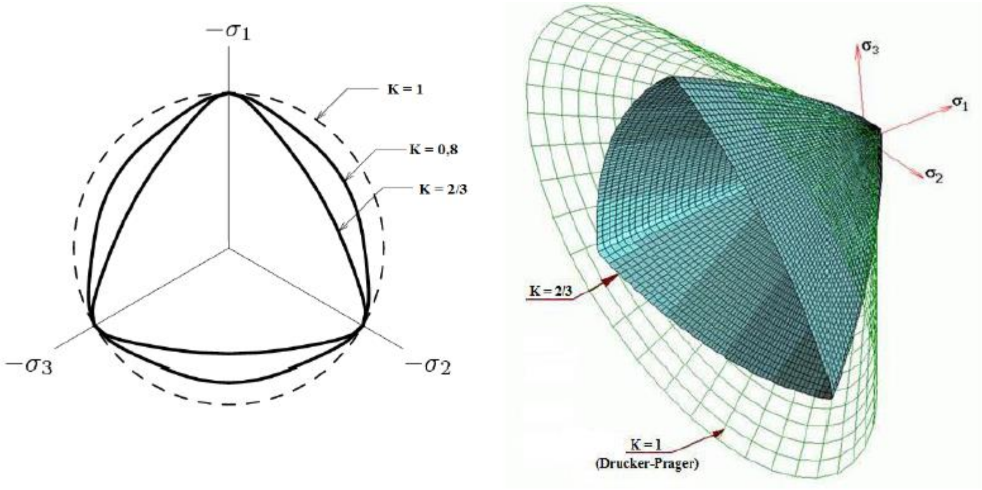


Figure 4. Influence of K parameter on the shape of the failure envelope [8]

The fourth parameter is called by eccentricity (ρ) and can be designed by the relation between the compression and tension experimental tests. This can be faced as the curvature of the failure envelope on a meridional plane. The value of this parameter considered on the analysis is equal to 0,1 – making, on a meridional plane, the envelope has the shape of a hyperbole.

The last parameter is called viscosity and is not related to the material itself. This parameter allows the stress to exceed a little bit the failure envelop to avoid some numerical errors and then facilitate the convergence of the model. The recommended value to this parameter is between 0,001 and 0,0001. Michał & Andrzej [9] studied this parameters and presented a detailed results about the variation of the viscosity – Fig 5.

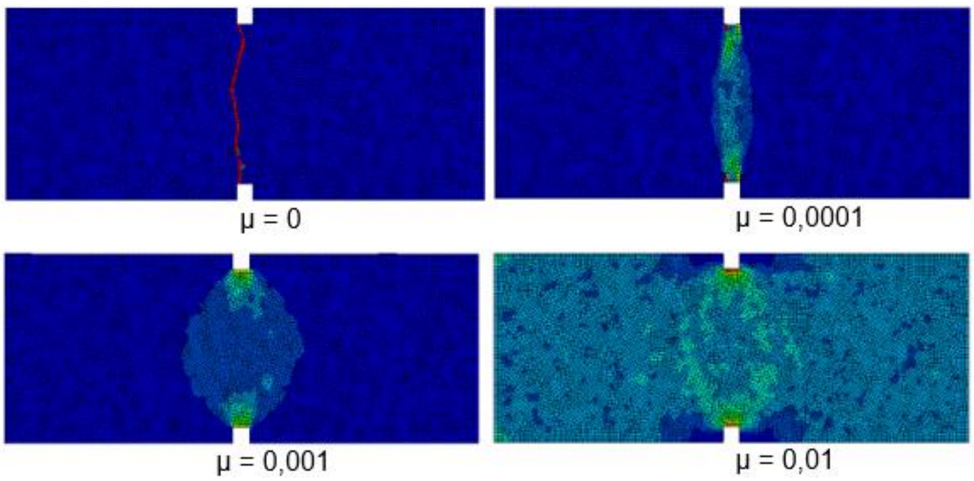


Figure 5. Influence of μ parameter on the analysis [9]

2.1.3. Concrete damage plasticity

On the Fig. 6 and Fig. 7 are illustrated the diagrams of concrete responses (in tensions and compression) with the damage of the material taken in consideration. Inelastic strain ($\varepsilon_c^{\sim in}$) is divided in two parts, the first one is a plastic ($\varepsilon_c^{\sim pl}$) portion – that is permanent or non-recoverable – and the second portion that depends on the integrity of the material (damage parameter – d_c and d_t). We can also perceive that the Young Modulus depends of the damage on the material to.

According to Lubliner [10] theory of damage plasticity, the damage parameters can be written in function of the initial Young Modulus and the current modulus (considering the plastification of the material) – equation 07.

$$\frac{E}{E_0} = 1 - \mathrm{d} \tag{7}$$

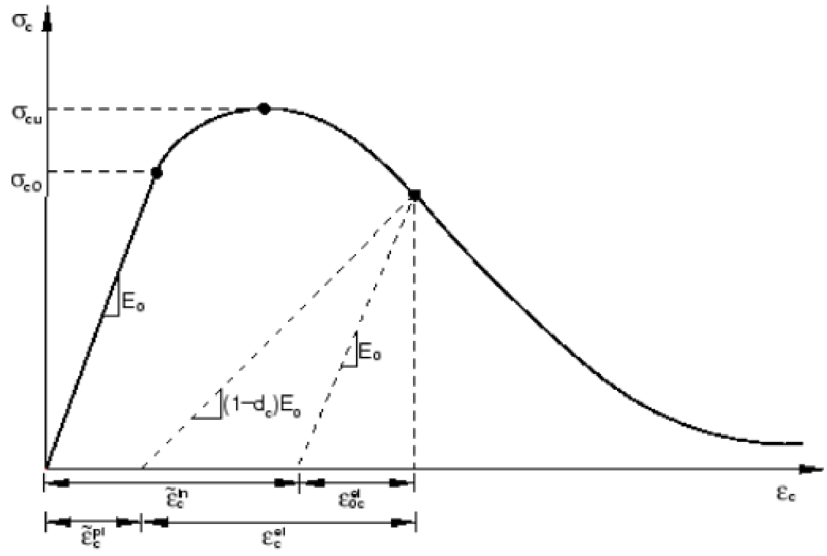


Figure 6. Stress Strain diagram for concrete in compression with the application of damage [7]

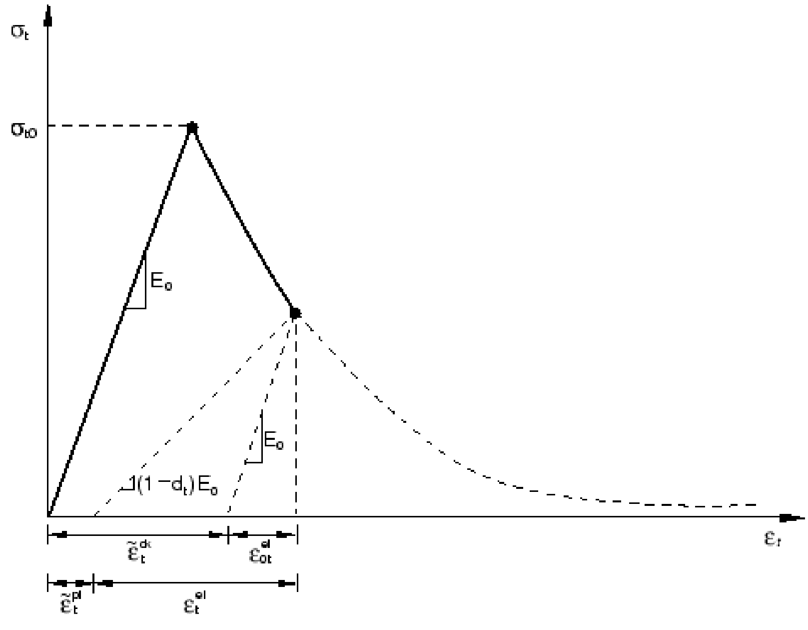


Figure 7. Stress Strain diagram for concrete in tensile stress with the application of damage [7]

To calculate the inelastic strains ($\varepsilon_c^{\sim in}$) and the cracking strain (ε_t^{ck}) we have to subtract the elastic portion of the strain from the total strain, illustrated by the equations 8 and 9 respectively. The elastic strain at any point can be found simply by dividing the stress by the initial Young modulus of the material.

$$\varepsilon_c^{\sim in} = \varepsilon_c - \ \varepsilon_{0c}^{el} \tag{8}$$

$$\varepsilon_t^{ck} = \varepsilon_t - \ \varepsilon_{ot}^{el} \tag{9}$$

Thus, the inelastic and cracking strain are the equivalent non elastic strains for compression and tensile behavior. These strains can still be divided in two portions, one represented by the plastic strain ($\varepsilon_c^{\sim pl}$) that represents the strain that is unrecoverable when the load is suspended and the damage portion of strain that measure the level of integrity of the material. That can be described by the following formula:

$$\varepsilon_c^{\sim pl} = \varepsilon_c^{\sim in} - \frac{d_c}{1-d_c} * \frac{\sigma_c}{E_0} \tag{10}$$

The previous formula has to be checked to assure that plastic strain does not get neither negative nor decreasing values, that may cause errors during the analysis. The damage parameter varies from 0 (integral state) to 1 (totally damaged state) and can be calculated by the relation of the inelastic/cracking strain and the total strain – represented on the next equations:

$$d_c = \frac{\varepsilon_c^{\sim in}}{\varepsilon_c} \tag{11}$$

$$d_t = \frac{\varepsilon_t^{ck}}{\varepsilon_t} \tag{12}$$

2.2 Constitutive model for steel reinforcement

For the steel it was used a bi-linear elastic/plastic constitutive model, with gain of endurance before the flow point (strain hardening). Here will be listed the main characteristics of the steel used on the simulations and on Fig. 8 is represented the stress strain diagram for the material:

- Young Modulus – 210 GPa;
- Poisson coefficient – 0,3;
- Ultimate Strength (elastic/linear phase) – 500 MPa;
- Strain on the beginning of flow regime – 0,002;
- Ultimate Strength (after strain hardening) – 550 MPa;
- Strain on the rupture – 0,008.

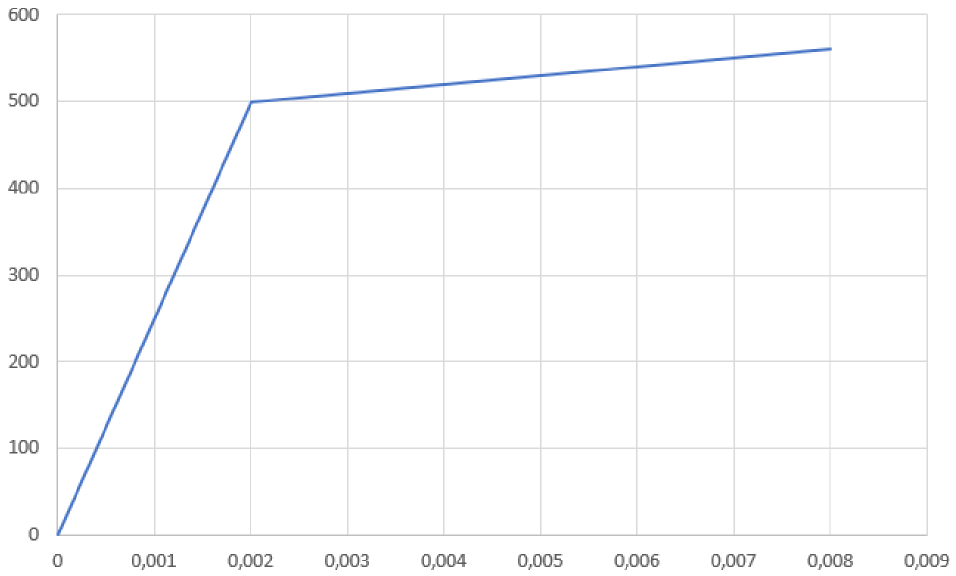


Figure 8. Stress Strain diagram for steel used on the simulations

3 Model Implementation

The model presented on this paper was inspired on experimental tests developed by Campos [1] on 2007. Campos [1] studied the ultimate strength of three pile caps foundation submitted by center force. On the Fig. 9 there is a schema of the strut and tie configuration used by the author to design the steel reinforcement on one of these elements:

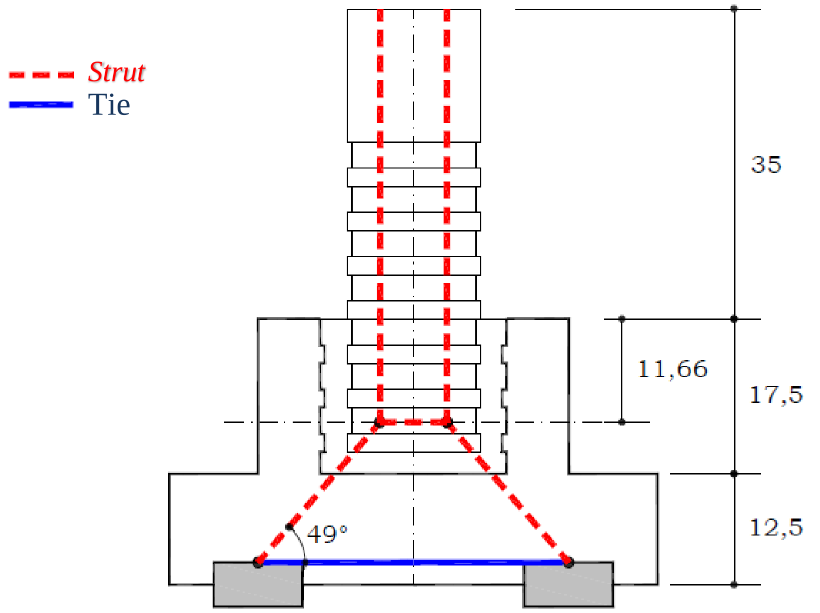


Figure 9. Equivalent Strut and Tie Model used by Campos on the design [1]

To design this element, Campos [1] used prescriptions from NBR 6118/2014 (Brazilian Code for Armed Concrete Structure) related to D regions. This element is considered a discontinuous element due to its geometry. Brazilian Code recommends that this type of region has to be studied using the strut and tie method. The geometry of the element on top view is illustrated on Fig 10 and the elevation on Fig 9 as well.

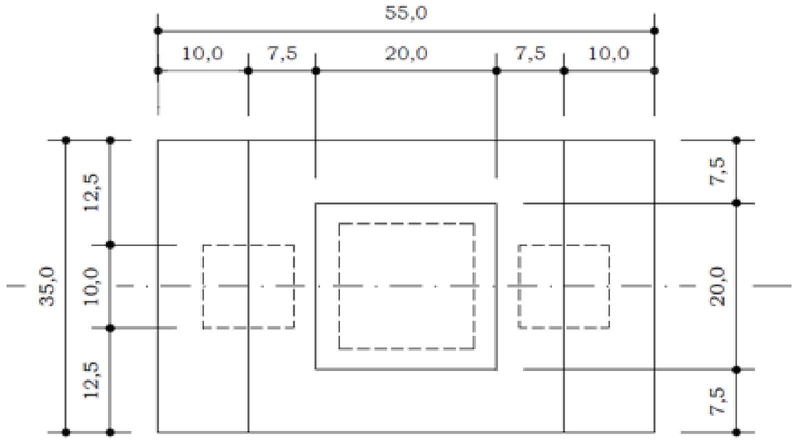


Figure 10. Geometry of the pile cap essayed by Campos [1]

The principal tie (tension rod represented by blue color on Fig. 9 schema) is reinforced with two rebars of 12.5 mm (bar number N1) of diameter plus one rebar of 10 mm (bar number N2) of diameter – this grants a reinforcement of 3,3 cm² of steel on the main tie of the element. The secondary reinforcements of the pile cap and the neck of the element can be checked on the Fig. 11.

The column that is embedded inside the neck of the pile cap was designed to resists the stresses generated by the application of the external force (by a mechanical press). The reinforcement of this element is compounded by four longitudinal rebars of 8.0 mm, stirrups of 5.0 mm each 5 centimeters and a transversal reinforcement next to the point where the load is applicated using extra stirrups each 3 centimeters.

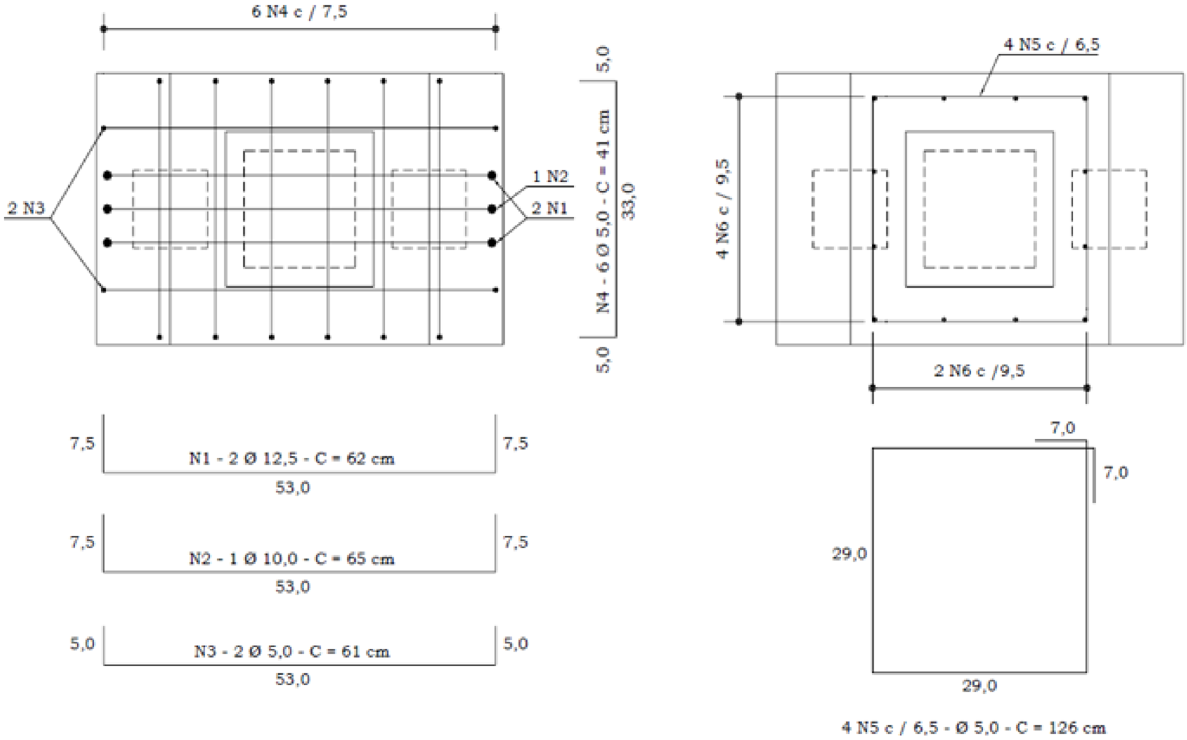


Figure 11. Schema of reinforcement embedded on the element [1]

A numerical model was constructed using all the design appointments presented by Campos [1] like: amount of reinforcement, position of rebars, class of concrete, geometry of the structure, borders condition and loads inputs.

The numerical model was built on a FEM software called ABAQUS. The rebars were inputted using a technique called embedded region – technique that consists in increase the stiffness of the elements where the wires that represents the rebars pass through. On Fig 12 is presented the numerical model compounded by 18682 hexagonal elements from type C3D8R with average dimension of 15 mm representing the concrete part and 1493 linear elements from type "beam" representing the rebars (average dimension of 15 mm too). The mesh is generated in a structured and symmetric form – the distribution of the elements on the mesh can be checked on the figure 12 isometric.

The borders condition can be checked on figure 12 as well, one of the piles have restriction on displacement on all directions (ux=uy=uz=0) and the other one has restrictions on two directions (uy=uz=0), allowing the displacement on the longitudinal axis of the element. On the top of the column was applied a force that goes from zero to the crash load. The top of the column had its displacements restricted on its transversal plane to (uy=uz=0) to simulate the friction between the mechanical press and the concrete element.

The main goal of this model simulation is to predict, through a computational method, the response of a concrete element using the concepts of concrete damage plasticity (CDP) theory, in other words, it is desired to achieve similar results founded by Campos [1] on his experimental analysis through a polished non-linear numerical analysis considering the damage of concrete, the amount and flow regime of the reinforcements (rebars), the redistribution of stress and the real deformation of the structure on an incremental analysis (second order analysis).

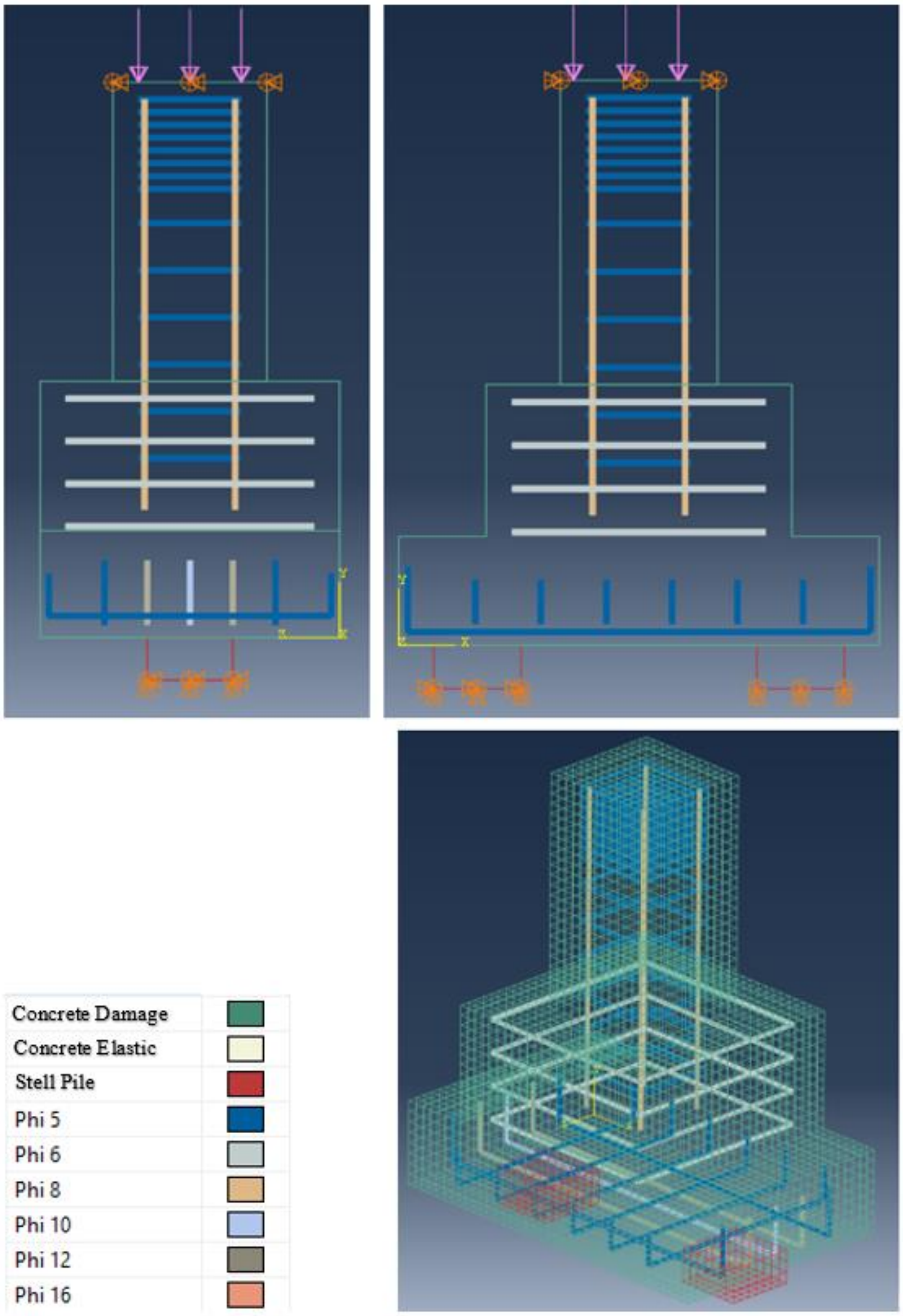


Figure 12. Computational model constructed to simulate the problem presented

The concrete has an ultimate resistance of 25 MPa and it have the constitutive concrete damage plasticity model implemented on it – all imputed parameters necessary to implement the model can be developed through the formulas presented on chapter 2. The reinforcement element follows the prescription of last chapter as well. The piles (border conditions) were implemented as a pure elastic steel material – it does not make any difference on the global behavior because the stress on these elements are far from its ultimate resistance.

4 Results and discussion

4.1 Stress

On the next figure frames, it is illustrated the stress on concrete at two different moments on the loading history. The first frame indicates the moment when the load on the column is equal to the design load of the reinforcement – load equal to 30 tf. The second frame indicates the moment of collapse/rupture of the structure – load equals to 58,73 tf. On these figures is possible to check the evolution of the stress on the inclined struts and the concentration of stress near the border conditions – all stresses are in MegaPascal – Fig 13.

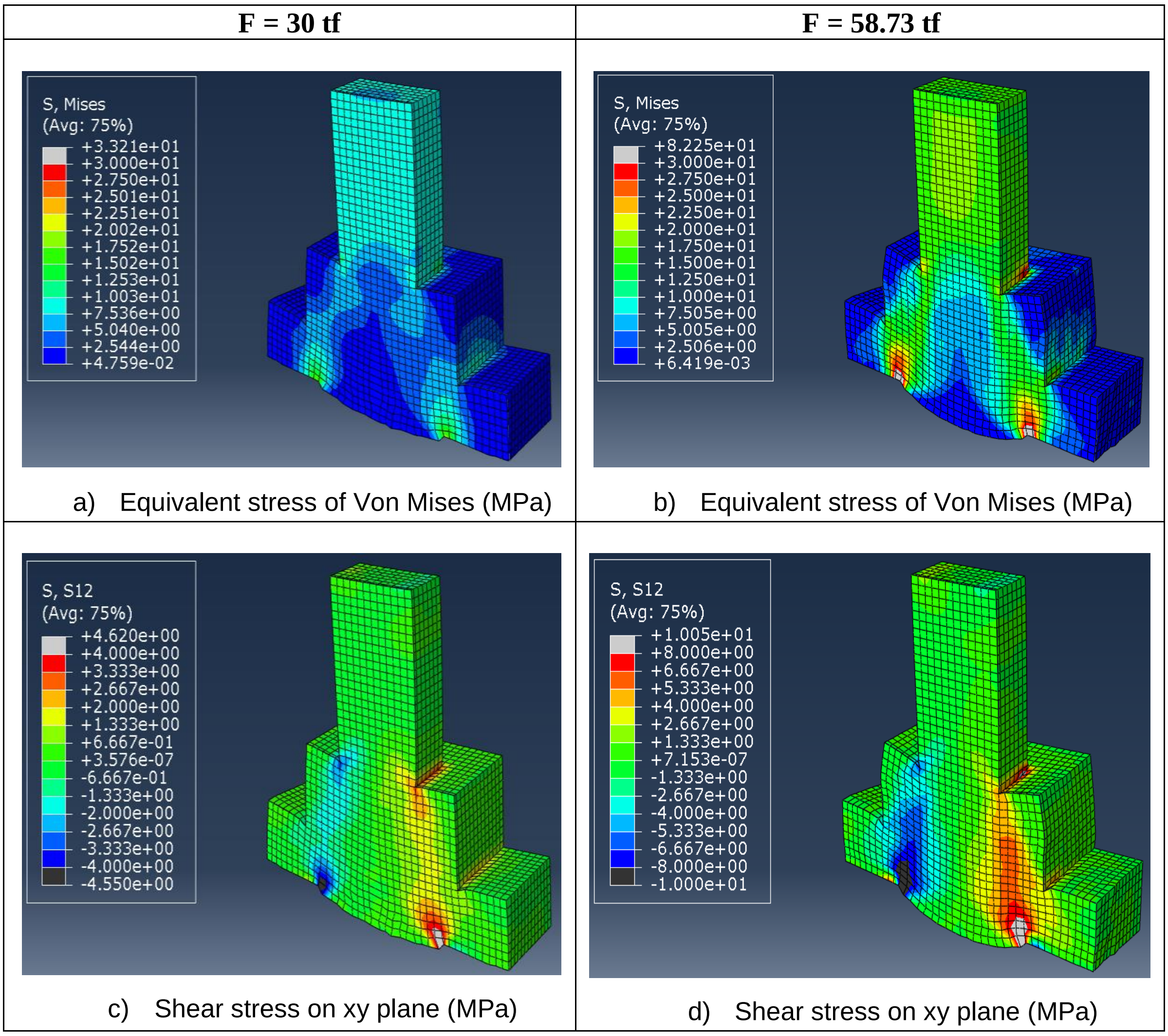


Figure 13. Von Mises and Shear Stress on the simulated pile cap

Usually, the rupture/collapse of a structure does not happen by one specific reason, but for a set of different failure spots that happen sequentially. So, the goal on this item is not to define what was the reason that take the element to collapse, but to understand how the stress evolves on the interior of the element. Seen that, on the next figure is presented the stress on the rebars reinforcement at the moment that the element collapsed. It is possible to perceive that the rebars of the main tie (on the bottom of the pile cap) are working on a stress already on the flow regime (500 MPa) – Fig 14. We can check that on this moment some elements of concrete are submitted to a Von Mises Stress Equal to 30MPa. It is also possible to check that some stirrups of the neck of the pile cap are submitted to high stress to, due to the splitting effect of these regions – high compression on struts causes tension on its transversal plane.

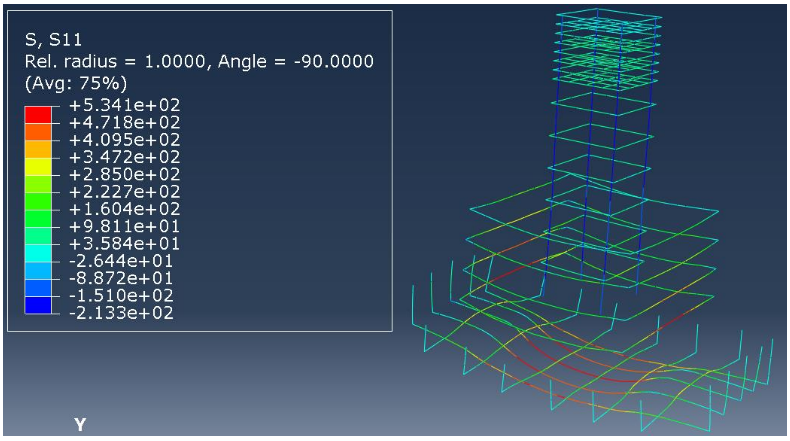


Figure 14. Normal stress on the reinforcement rebars

4.2 Load of Collapse

The pile cap essayed by Campos [1] was designed to resist a theoretical load (F_{teo}) of 30 tf, but to respect the commercial sizes of the rebars the amount of reinforcement imputed on the element was a little higher. The project load (F_{proj}) – considering the real amount of rebar cross section on the main tie – is equal to 37,96 tf, this verification was made by a strut and tie analysis without any safety coefficient.

The load resisted by the element on the experimental analysis conducted by Campos [1] was equal to 58,1 tf (F_u) – which presents a load reserve near 95 % higher when compared with the theoretical load. On the numerical model the simulation presented the collapse on the moment that the load was equal to 58,73 tf (F_u) – very close to the experimental analysis.

Both models, experimental and numerical, presented a load reserve almost 55% higher that the project load predicted by the verification. This happens because there are a lot of mechanisms that exists on reinforced concrete structures that are not considered during the design phase. Some of these mechanisms are: the concrete response on tension stresses, the effect of strain hardening on the reinforcement steel subjected to a flow regime, re-distribution of stress to other points of the element, the consideration of the deformed shape on the analysis, the response of complimentary reinforcement (constructive rebars) and the increase of concrete ultimate strength when its subjected to a multi-axial state of stress.

The resume of the loads can be checked in the table below:

Table 1. Resume of experimental and numerical loads for the simulations.

Resume of Loads					
	F_u (tf)	F_{teo} (tf)	F_{proj} (tf)	F_u/F_{teo}	F_u/F_{proj}
Experimental Model (Campos 2007)	**58,1**	30	37,96	1,936	1,530
Numerical Model	**58,73**	30	37,96	1,957	1,547
Diference	**1,07%**				

Analyzing the results, it is possible to reaffirm the argument defended by Campos [1] that the design recommendations used to detail the element and the Strut and Tie method idealized initially by Blevot and Fremy [11], lead to a conservative schema of reinforcement.

4.3 Displacements

On the experimental test, Campos [1] measured, through the use of displacement gauges, the vertical displacement on the center of the pile cap and the horizontal displacement on one of the far ends of the pile cap. The two measures were saved on the moments relative to the project load (Fproj) and for ultimate load (Fu). The displacement found on the numerical analysis on the vertical direction can be checked by Fig. 15 for both moments.

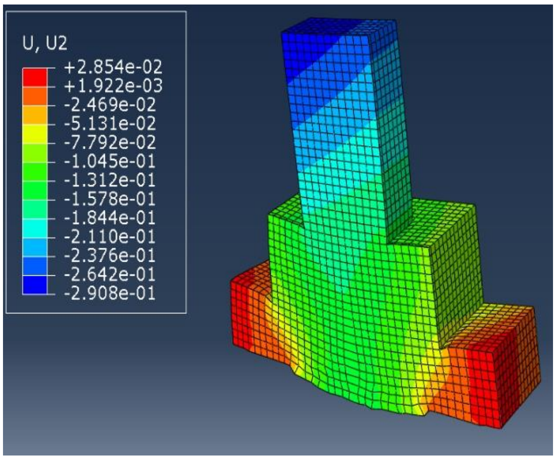
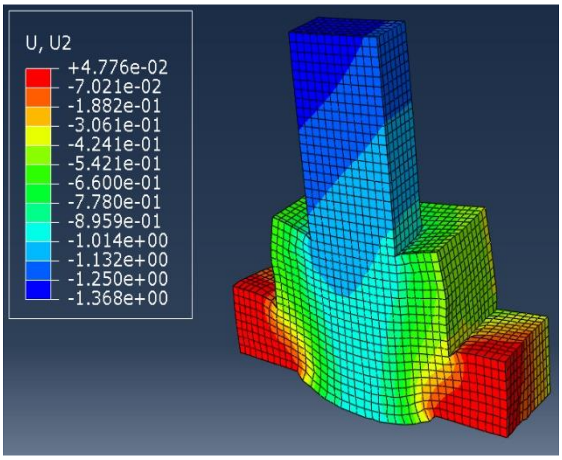
F = 30 tf	**F = 58.73 tf**
	
a) Vertical displacement – y axis (mm)	b) Vertical displacement – y axis (mm)

Figure 15. Displacements of numerical model on vertical direction

It is possible to perceive on table 2 a significant difference between the numerical and experimental models on a first look. A few reasons that can explain the difference on the results to this parameter. The first one is that on the numerical model it is not considered the truth opening of the cracks, once the damage is applied on a distributed way (approximated method) and the mesh of the FEM model is not re-generated every step. The second reason could be the non-consideration of the slip between the concrete and the rebars, because on the numerical model they are discretized with perfect bond. Both reasons could make the numerical model a little more rigid than the experimental.

Maybe on a flexural element this comparison could lead to better results, seen that we are analyzing an extremely rigid type of structure (with major shear forces acting on it). On this type of elements, the order of magnitude of the displacements are very small, which compromises a little bit the comparison of the models.

The resume of the displacements on both models for the two studied moments are presented on the next table:

Table 2. Resume of displacement on both models.

<table>
<tr><th colspan="8">Resume of Displacements</th></tr>
<tr><td></td><td>Load (tf)</td><td>Vertical displacement (mm)</td><td>Horizontal displacent (mm)</td><td rowspan="5"></td><td>F</td><td>Expv/ Numv</td><td>Expv/ Numv</td></tr>
<tr><td rowspan="2">Experimental Model</td><td>F_u = 58,1</td><td>2,64</td><td>0,43</td><td rowspan="2">F_u</td><td rowspan="2">2,61</td><td rowspan="2">0,54</td></tr>
<tr><td>F_{proj} = 37,96</td><td>1,86</td><td>0,08</td></tr>
<tr><td rowspan="2">Numerical Model</td><td>F_u = 58,73</td><td>1,01</td><td>0,8</td><td rowspan="2">F_{proj}</td><td rowspan="2">5,31</td><td rowspan="2">0,26</td></tr>
<tr><td>F_{proj} = 37,96</td><td>0,35</td><td>0,31</td></tr>
</table>

Even tough, the results putted side by side represents an expressive percentual difference. The author understand that the results are satisfactory, because both models had displacement on the millimeters order of magnitude.

4.4 Crack pattern

The crack pattern found on the numeric analysis can be checked trough the use of two different parameters. The first one called DAMAGET measure the damage (plastification) of the element on tensile stresses – it is known that damage on concrete is the representation of cracks or micro-cracks in the interior of the micro-structure of the material. This parameter does not have the goal to predict the exact place that will appear the main cracks, but it can predict what regions (on a distributed way) will suffer damage, in other words, it is a good call to predict the crack pattern. The second parameter is called PEEQT (Equivalent Plastic Strain at Integration Points) measure the plastic strain (not recoverable strain) on the elements, this parameter shows with better precision the regions that should appear the main cracks.

On the Fig. 16 is presented the results of DAMAGET and PEEQT parameters for the model. DAMAGET has values from 0 (no damage) to 1 (totally damaged). PEEQT is a strain value (mm/mm).

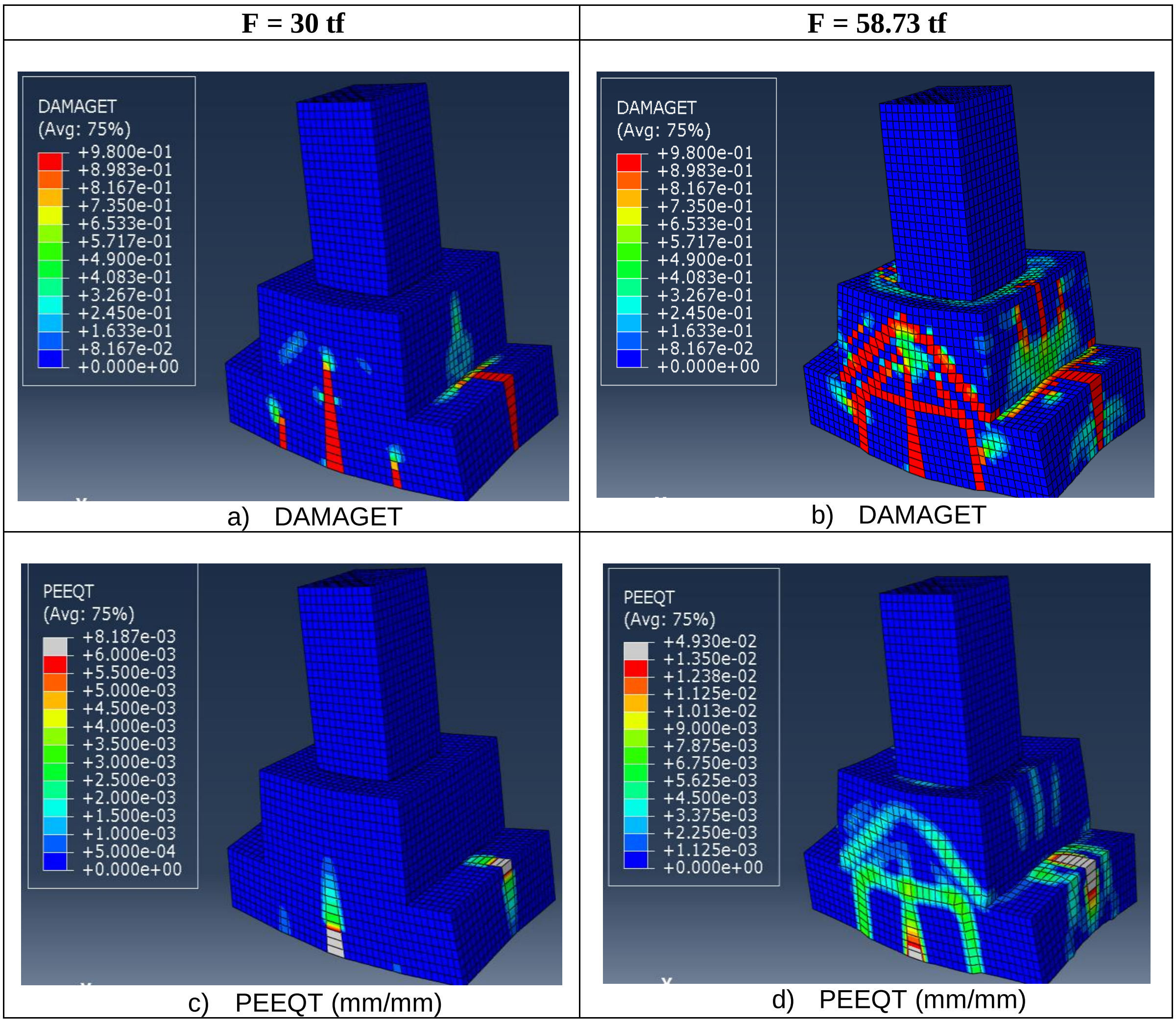


Figure 16. Crack pattern parameters

On the experimental tests, Campos [1] found a similar crack pattern that was predicted on the numerical approach. The main cracks appeared on the center of the pile cap, coming from the bottom to the top. With the increase of the load application the cracks tend to incline to the cross-section plane of the struts due to the shear force of this regions. On the Fig. 17 it is possible to visualize a photo of the essay conducted by Campos [1] and a schema made by him to present the main crack pattern.

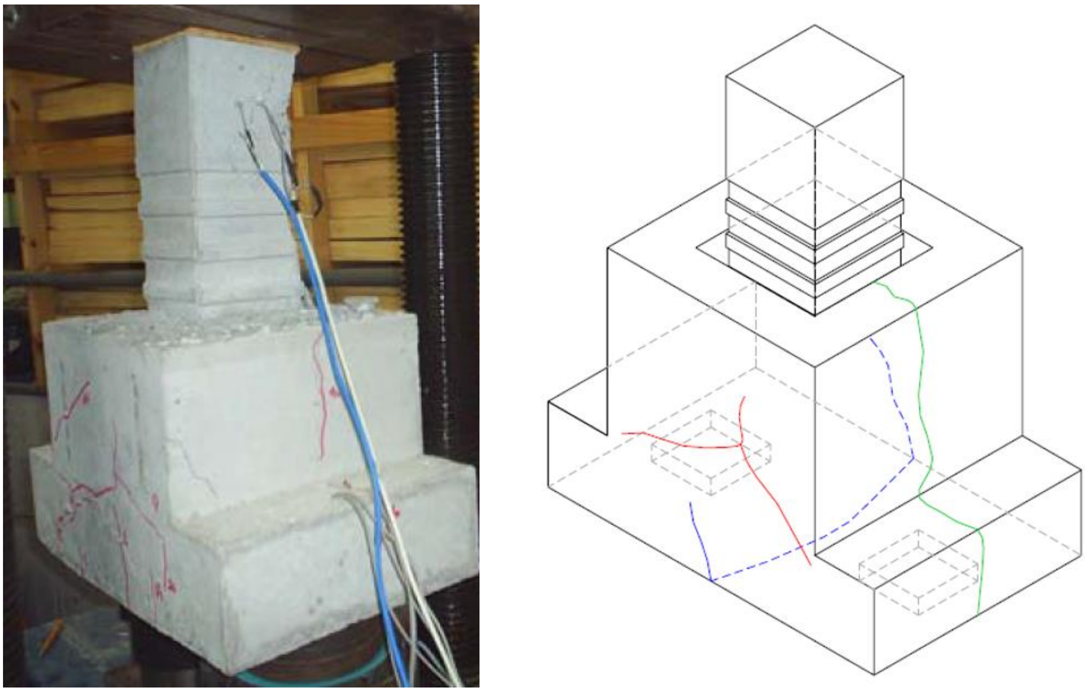


Figure 17. Crack pattern founded by Campos [1]

Just to refresh, concrete is a material very heterogenous and with a lot of uncertainties on its micro-structure organization due to the complex process of fabrication. So, it is plausible that some regions could have more stiffness than others due to this aleatory organization of the compounds. That fact on experimental models could force the cracks to appear on areas that have a minor stiffness on some cases. This explanation just wants to reinforce that predict the exact place where the crack appears and the direction that it propagates by a numerical example may be a utopia.

Despite this it is possible to correlate the similar results founded between the numerical and experimental models.

4.5 Strain on the main tie

The last parameter that is going to compared on this paper is the maximum strain on the rebars of the main tie. On the experimental model it was used strain gauges attached on the steel rebars to find out the strain on the bottom center of the pile cap. The table 3 below shows a resume of the strain found by Campos [1] and on the numerical Model.

Table 3. Resume of strain on the main tie.

Strain on the main Tie		
	Fu (tf)	**ε (‰)**
Experimental Model	58,1	**2,67**
Numerical Model	58,73	**2,41**
Diference	1,07%	**10,79%**

It's possible to check that on both models the main reinforcements are working on a flow regime (strain higher than 2‰) at the moment of the collapse, that means that the reinforcement is subjected to a load higher than 500 MPa (Ultimate Strength to steel on elastic regime). The results of the models present a difference under 11% between them, that shows a good agreement on this parameter comparison too.

5 Conclusions

This paper presents a comparison between a numerical and experimental model of a “D Region” type of concrete structure. On the numerical model was implemented a non-linear material model for concrete that intends to simulate with high precision the real element. This concrete material model has attached to itself some concepts of damage mechanics for both compression and tensile stresses. Besides that, it was considered the real amount of reinforcement and its positioning inside the element to achieve the results presented on this paper.

Besides some minors divergences due to some simplification and the uncertainties about the material, the results of the tests were very close to the expected. Next, the numerical material model used is validated and it was proved that it is possible to simulate concrete structures on a non-linear way by a numerical FEM package with good precision. The concrete material is a very though material to be simulated due its uncertainties already listed on this paper. It is certain that if it was possible to run some experimental tests on the same concrete that was effectively used on the experiments, it would be possible to retrieve data to implement a concrete model material specific for the concrete that was used. That fact could lead to better results.

References

[1] CAMPOS, L. D. A. Análise experimental de bloco de fundação sobre duas estacas, sob carga centrada, para estrutura pré-fabricada. Dissertação (mestrado) - Universidade Federal de Santa Catarina, Centro Tecnológico. Programa de Pós-graduação em Engenharia Civil.
[2] Noghabai K., Olofsson T. (2000) Effect of tension softening of concrete on the tension stiffening of rebars in plain and fibrous concrete, in First Engineering Foundation Conference. Division of structural Engineering University of Technology (Sweden).
[3] NAYAL, R., & RASHEED, H. A. Tension Stiffening Model for Concrete Beams Reinforced with Steel and FRP Bars. Journal of Materials in Civil Engineering, 18(6), 831–841. 2006. Disponível em<<https://doi.org/10.1061/(ASCE)0899-1561(2006)18:6(831)>>
[4] WAHALATHANRI , B.L.; THAMBIRATNAM, D.P.; CHAN, T.H.T.; FAWZIA, S. A material model for flexural crack simulation in reinforced concrete elements usingABAQUS,260–264.2011. Disponível em << https://doi.org/10.1007/BF02906653>>
[5] KOTSOVOS, M. D.Finite element Modelling of Structural Concrete(Vol. 41). 2015. Publisher: CRC Press; 1 edition (May 20, 2015)
[6] Hsu, L.S., & Hsu, C.-T.T. (1994). Complete stress-strain behaviour of high-strength concrete under compression. Magazine of Concrete Research, 46(169), 301-312.
[7] Abaqus Analysis User Manual – Abaqus Version 6.8. (2008). Retrieved November 5, 2010, from http://bee-pg-031941:2080/v6.8/books/usb/default.htm
[8] PRATES AGUIAR,O. Estudo do Comportamento de Conectores Crestbond em Pilares Mistos Tubulares Preenchidos com Concreto.2015.
[9] MICHAŁ, S., & ANDRZEJ, W. Calibration of the CDP model parameters in Abaqus. 2015.
[10] LUBLINER, J., OLIVER, J., OLLER, S., & Oñate, E. A plastic-damage model for concrete. *International Journal of Solids and Structures*, *25*(3), 299–326. (1988) Disponível em <<https://doi.org/10.1016/0020-7683(89)90050-4>>
[11] BLEVOT, J.; FRÉMY, R. Semelles sur pieux. Annales d’institut Technique du Bâtimentet des Travaux Publics, Paris, 1967, v.20, 230p