

Journal of Engineering Research

FINITE ELEMENT ANALYSIS OF REINFORCED CONCRETE BEAMS THROUGH ANSYS SOFTWARE

Henrique Cardoso Koch

Bruna Manica Lazzari

All content in this magazine is licensed under a Creative Commons Attribution License. Attribution-Non-Commercial-Non-Derivatives 4.0 International (CC BY-NC-ND 4.0).



Abstract: With the importance and need to advance knowledge in the project using reinforced concrete, the optimization of materials and clarification of the variables present within the project are sought. This work brings a three-dimensional computational model, through the finite element method, using the ANSYS software, version 18.0, for the study of reinforced concrete beams. As validations, the ET series beams tested by Leonhardt and Walther (1962) were used, and the numerical results were compared with the experimental values. From the analysis of the breaking stresses of the concrete, the steel bars and the stirrups, it was concluded that the results were satisfactory, showing that the proposed model brings great approximation to the projects of reinforced concrete beams.

Keywords: Reinforced concrete beams, Finite element analysis, ANSYS.

INTRODUCTION

Due to the importance of reinforced concrete structures in the context of civil construction, this subject has been the subject of many publications in the academic and professional environment (LAZZARI, 2015). These studies aim to understand the various variables present in the design of reinforced concrete, thus managing to improve its use. It is necessary to advance in the knowledge of the behavior of reinforced concrete structures, considering that the development of engineering and architecture, bring in their new approaches, bolder structures, with larger spans and slender structural parts. This project aims to gain area in the buildings, bringing a better financial return on the investments used.

As the complexity of the structures grows, so does the difficulty in studying each case. For the analysis of these reinforced concrete structures, the Finite Element Method (FEM)

is certainly the most used numerical process (AURICH, 2001). The idea of the FEM is the discretization of a continuous medium, that is, the division of structures into smaller parts, with less complexity, in order to facilitate the calculations. These parts are divided into a finite number of elements that are connected by nodes. This set of nodes is called a finite element mesh. For each node, the nodal displacements, stresses and strains inside the elements can be determined, as well as the reactions at the support points of the structure. The model allows considering the non-linearity of these materials and also the processes of concrete cracking and steel plasticization (SORIANO, 1998). Within structural engineering, the method aims to determine the state of stress and deformation of a solid, of any geometry, which is exposed to external actions (AZEVEDO, 2003).

This work aims to present an adequate modeling for reinforced concrete structures using the FEM, determining the properties of both concrete and reinforcement, allowing simulations to be carried out in different reinforced concrete structures. To reliably characterize these variables, it is necessary to define the behavior of each material, as well as the geometry to be analyzed. To create the computational model, the ANSYS program (Analysis Systems Incorporated) version 18.0 was used. According to Lazzari (2015), the great acceptance of the program in the academic environment makes it a great tool for studies related to FEM, allowing for static, dynamic, fluid analyses, whether linear or non-linear materials, attributing behaviors, which we can control: elastic, plastic, viscous or the combination that is convenient to use. Using the ANSYS software, the state of deformations and stresses of elements in reinforced concrete was analyzed after the application of an instantaneous load, through displacement increments of short duration.

The validation of the computational model for reinforced concrete structures implies the use of experimental tests or even results obtained directly through the application of the current reinforced standard for concrete (NBR 6118/2014). The results of experimental procedures must be consistent and renowned, making the utility of computational modeling proven. Among the countless possibilities of experiments for validation are the beams of Leonhardt and Walther (1962); Bresler and Scordelis (1963); the prestressed concrete beams by Tao and Du (1985); and the reinforced concrete beams with holes by Mansur (2006). Due to the wide use in academic works and its wide range of behaviors in situations of rupture, it was decided to use the beams of Leonhardt and Walther (1962) to prove the effectiveness of the proposed model.

OBJECTIVES

The objectives of this research are separated into primary and secondary, as described below.

Main objective

The main objective of this work is to perform a three-dimensional numerical analysis on reinforced concrete beams tested by Leonhardt and Walther (1962), applying the finite element method, through the ANSYS system. From the validation of the exposed model, through the comparison between the numerical and experimental results, it is understood that it is possible to perform simulations on parts of any geometry and load.

Secondary Objective

As a secondary objective has:

- a) study of ANSYS finite element program tools;
- b) description From models of behavior of materials steel and concrete;

- c) elaboration of scripts with the data entry of the analyzed parts.

DELIMITATIONS

The work presents two important limitations. The first is the fact that the proposed computational model for reinforced concrete is tested only in the finite element software ANSYS. Another factor is the validation of the exposed model, which will bring proof of its effectiveness only to the beams of the ET series, proposed by Leonhardt and Walther.

THEORETICAL REFERENCE

For the elaboration of the computational model, it was essential, in this research, to carry out a bibliographic review regarding the finite element method and also the models of the materials that were adopted. The following items present some important aspects, which were necessary for the continuity of the work.

METHOD FROM FINITE ELEMENTS

In the vast majority of cases, it is very difficult to determine the date on which an advance in knowledge occurred. In the case of the FEM, it is reported several times that the first occurrence of the term "finite elements" is the article, which dates from 1960 and is authored by Ray Clough. Major advances in terms of FEM came in the 1960s and early 1970s, bringing model analysis to the most acceptable format today. Initially, the most common finite elements were triangular and tetrahedral, later giving preference to quadrilaterals and hexahedrons. Unlike other methods that were used in the past, the FEM requires a great computational effort, due to the complexity of the mathematical equations necessary to reproduce natural phenomena. Knowing this computational effort, it is not by chance that the use of the method grew with the development of better computers

and more advanced technologies. Thus, in the mid-1990s, this method became widely used by structural designers (AZEVEDO, 2003).

ANSYS (*Analysis Systems Incorporated*)

There are many analysis programs by the MEF, among which it is possible to highlight the most used: ANSYS, ABAQUS, NASTRAN and SAP2000. For this work, ANSYS 18.0 was used, which covers numerous materials and ways of evaluating results, which makes it superior to many other analysis software by the MEF. Through this platform, the properties of materials and their behavior can be defined, as well as the geometry of the structures to be analyzed, with no single means of composition (GONÇALVES JUSTO, 2010).

ANSYS has two work environments: the APDL system (ANSYS Parametric Design Language) and the workbench system. The workbench features a more user-friendly and intuitive screen design. On the other hand, the ADPL system, despite having an older appearance, allows the user to better manipulate the simulation that he wants to do, bringing with the software a design and modeling tool in a simplified way. Another important factor that influenced the choice of this system for this work was the possibility of including commands through a text file (files with .txt extension). This file may contain all kinds of information needed for modeling as per the ANSYS command list.

COMPUTATIONAL MODELING

This item presents aspects of the modeling and properties of the finite elements of the ANSYS library, which were used to represent the beams in reinforced concrete. For the modeling of concrete, reinforcement, support points, and load application points, the following elements were used, respectively: SOLID 65; LINK 180; and SOLID 185.

Modeling of concrete

Below are the descriptions of the chosen element for representing the concrete, according to the ANSYS library. Also described are the behavior models responsible for characterizing the material within the system.

SOLID 65

For concrete modeling, the SOLID 65 element was used in this work, which is composed of eight nodes, with three degrees of freedom in each of them, as shown in figure 1. This element is commonly used for three-dimensional modeling of concrete structures, as it can simulate the behavior of the rupture surface of brittle materials. This way, it is able to represent the cracking, when subjected to tensile stress, and the crushing of the concrete when requested to compression, including plastic deformations and creep. To represent the concrete in the system, four parameters were adopted for the behavior of this material, which are : *Linear Isotropic, Multi-linear Elastic, Concrete and Density*.

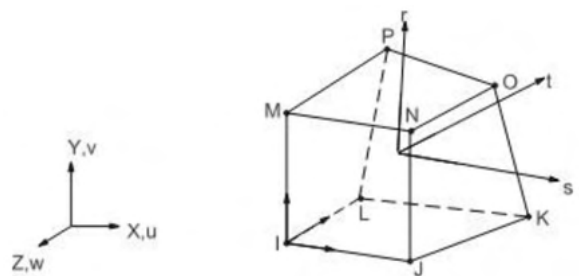


Figure 1 - geometry of the SOLID 65 element.

Source: ANSYS, 2017.

The Linear isotropic model considers a linear elastic material. It is necessary to define two variables to determine their behavior: the Poisson's ratio, which the adopted value was 0.2; and the material's modulus of elasticity, which was defined according to the Model Code *fib* 2010 (2012), presented by Equation 1.

$$E_{ci} = E_{co} \cdot \alpha_E \cdot f_{cm}^{\frac{1}{3}} \quad (\text{Equation 1})$$

Where:

E_{ci} : concrete modulus of elasticity (kN/cm²);

E_{co} : 2150 kN/cm²;

α_E : adopted 1.00 for quartz type aggregate;

f_{cm} : average compressive strength of concrete (kN/cm²).

Elastic Multilinear Model (MELAS)

The *Multi-linear Elastic model* is an elastic, non-linear constitutive model that was used to reproduce the behavior of concrete under compression. The input information is the stress-strain curve of the concrete under uniaxial compression, which has its values defined from Equations 2, 3 and 4, according to the Model Code *fib* 2010 (2012), and is represented in figure 2. Table 1 provides the

property values for C25 concrete, according to the Model Code *fib* 2010 (2012). In this way, it is possible to simulate the plasticization of concrete and the loss of its resistance capacity after the material resistance limit.

$$\frac{\sigma_c}{f_{cm}} = - \left(\frac{\kappa \cdot \eta - \eta}{1 + (\kappa - 2) \cdot \eta} \right) \text{ para } |\varepsilon| < \varepsilon_{c,lim} \quad (\text{Equation 2})$$

$$f_{cm} = \frac{E_{ci}}{E_{c1}} \cdot \eta \quad (\text{Equation 3})$$

$$\kappa = \frac{E_{ci}}{E_{c1}} \quad (\text{Equation 4})$$

Where:

ε_c : compression deformation;

ε_{c1} : deformation at maximum tension of compression

$\varepsilon_{c,lim}$: deformation last of compression;

E_{ci} : module of elasticity from concrete;

E_{c1} : secant modulus of the origin of the compression stress peak;

κ : plastic number.

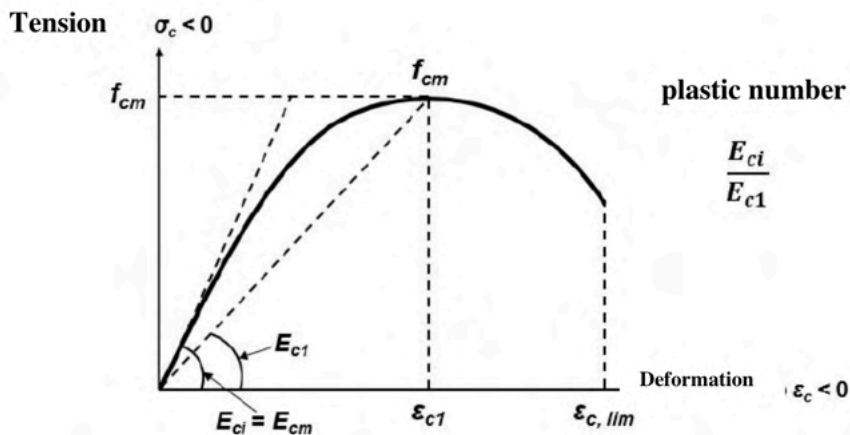


Figure 2 – Stress-strain diagram of compressed concrete.

source: Model Code *fib* 2010, 2012.

CONCRETE	f_{ck} (kN/cm ²)	E_{ci} (kN/cm ²)	E_{c1} (kN/cm ²)	S_{c1} (%)	$S_{c,lim}$ (%)	k
C25	2.5	3200	1490	-2.2	-3.5	2.15

Table 1 – Property values for C25 concrete.

Source: Model code *fib* 2010, 2012.

Model concrete (Concrete)

The model *Concrete* from ANSYS was used to represent the tensile behavior of concrete. This model uses the rupture criterion of *Willan-Warnke* according to the ANSYS manual (2017). The parameters required to use the model are:

- coefficient of transference of shear for fissure open;
- coefficient of transference of shear for fissure closed;
- ultimate uniaxial tensile strength;
- ultimate uniaxial compression strength;
- ultimate biaxial compression strength;
- ambient hydrostatic stress state;
- resistance last biaxial to compression under the state in voltage hydrostatic environment;
- resistance last uniaxial to compression under the state in voltage hydrostatic environment;
- coefficient multiplier in stiffness for condition fissured at traction.

The coefficients for shear transfer were adopted according to Kunzler (2013) where items “a” and “b” are respectively 0.2 and 0.4. The ultimate uniaxial resistance to traction is

calculated according to the recommendation of the *fib* Model Code 2010 (2012), presented for the Equation 5.

$$f_t = 0.03 \cdot (10 \cdot f_{ck})^{\text{two}} \quad (\text{Equation 5})$$

Where:

f_t : resistance to concrete traction (kN/cm²);
 f_{ck} : resistance characteristic to concrete compression (kN/cm²).

The compression limit has been turned off by applying the value -1. This value was adopted for avoid problems in convergence with others models adopted for compression.

Value gives specific mass of concrete

The model used to represent the specific mass of reinforced concrete was the *Density*, with the adopted value being 2.54e-8 t/cm³. Furthermore, it was defined within the ANSYS what value gives acceleration gives gravity would be 981 cm/s².

Modeling gives armor

Within the reinforced concrete structure, the steel bars basically resist axial efforts. Steel bars can be modeled in three ways, as shown in Figure 3: discreet; incorporated; and distributed.

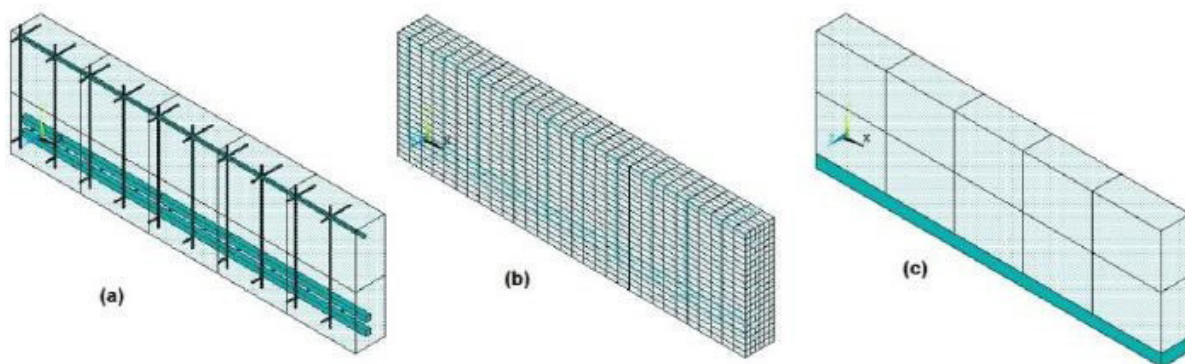


Figure 3 - Armor: (the model incorporated, (B) model discreet, (c) model distributed.

Source: LAZZARI, 2016.

The model: discrete represents at armor per elements one-dimensional, type trellis, and assumes that the reinforcement nodes are superimposed on the concrete nodes, restricting the reinforcement to the concrete mesh. Embedded model brings steel as a material line most hard inside of concrete, positioned in any point of the mesh. At the model distributed it is admitted the concrete evenly armed inside in each element, being convenient to be used at the case in plates and shells.

LINK 180

For the modeling of longitudinal bars of reinforcement and stirrups, it was used, in this work, the LINK 180 element, which is a one-dimensional element, composed of two we with three degrees in freedom in each a their according show The figure 4. This one element It is able in consider in form discrete The armor passive, considering grip perfect in between you elements of steel and in concrete.

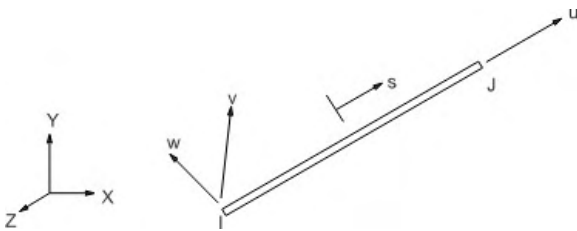


Figure 4 – Geometry of element LINK 180.

Source: ANSYS, 2017.

To represent the reinforcement, two behavior parameters were adopted being they: *Linear isotropic* and *bilinear* _ model: *Linear isotropic* Follow at same properties already described, being necessary to define two variables: the Poisson ratio, the which the value adopted was 0.3; and the elastic modulus of the material. The bilinear model is described The follow.

Model bilinear (BISO)

The model BISO (*bilinear isotropic hardening*) It is a model elastoplastic with hardening linear. At armor follow two behaviors, depending on of process in material manufacturing. For steels with well-defined yielding levels, the model elastoplastic Perfect (old class); already for steels what are hardened the cold (old class B), the elastoplastic model with linear hardening was used (figure 5). For this representation, the initial yield stress parameters and the modulus of hardening, given by Equations 6 and 7.

$$\sigma_0 = 0.85 \cdot f_y \quad (\text{Equation 6})$$

$$Et = \frac{0.15 \cdot f_y}{0.01 - \frac{f_y}{E_s}} \quad (\text{Equation 7})$$

Where:

σ_0 : voltage in flow initial;

Et : module of hardening;

f_y : voltage of flow.

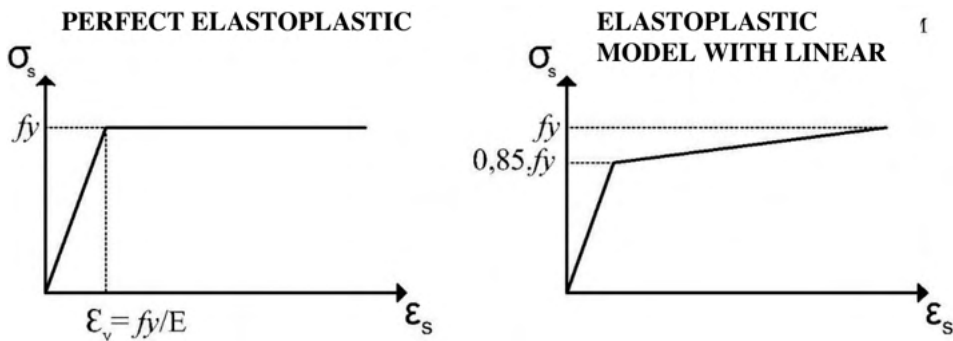


Figure 5 - Models elastoplastic Perfect and elastoplastic with hardening linear.

Source: elaborate by the author.

Modeling From supports

The modeling of the support and load application devices was carried out using the SOLID 185 element, present in the ANSYS library. This element consists of eight nodes, with three degrees of freedom in each of them, as shown in figure 6. The same is capable of simulating deformations of practically incompressible materials, reproducing the supports of beams and also you charge points, without changing you results gives analyzed beam.

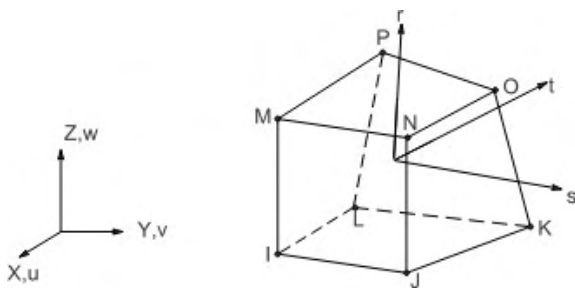


Figure 6 – Geometry of the element SOLID 185.

Source: ANSYS, 2017.

For representation of these elements, he was adopted a single parameter in behavior, the *Linear Isotropic model*. This model follows the same properties as described, being necessary to define two variables: the Poisson's ratio, which the adopted he was 0.2 and module in elasticity of material for which he was conventional, $E_{ci} = 20,000 \text{ kN/cm}^2$, an extremely high value, just so that the material does not influence the simulations.

DEVELOPMENT

With objective in prove, the efficiency of model proposed, are presented you comparisons between experimental and numerical analyses. Then, using the four beams in reinforced concrete what were rehearsed by Leonhardt and Walther (1962).

BEAMS IN LEONHARDT AND WALTHER (1962)

At four beams isostatic bi-supported in concrete armed studied are named: ET1, ET2, ET3 and ET4. Four-point bending tests were used on these beams, where two loads concentrated are applied us thirds of beams, according to the figure 7. All beams have a span of 3.0 m with the same height of 35 cm. The variations are due to the different cross sections. According to the experimental data, the resistance average The concrete compression used is in 2.42 kN/cm^2 .

All beams have the same longitudinal reinforcement, composed of cold-hardened steel (former class B). These consist of 4 lower bars of 20 mm in diameter, with the voltage in flow $f_y = 42.8 \text{ kN/cm}^2$, being two located The 3 cm and two 6 cm from the lower edge of the beam. There are also two upper bars with a diameter of 8 mm and voltage in flow $f_y = 46.5 \text{ kN/cm}^2$, located at 3 cm from the top edge gives beam.

The stirrups willing vertically, are compounds per bars in steel with toughness Natural (old class THE), with 6 mm in diameter and voltage in flow $f_y = 32 \text{ kN/cm}^2$. The distribution and bar spacing are shown in figure 7. The elastic modulus used are equals the 210 GPa for the steel class the and 195 GPa for grade steel B.

PROHIBITED IN DATA AT THE ANSYS

Launch in volumes, lines, sections and properties

To include the volumes of concrete, the support plate, and the application plate of load in the program, the command BLC4 was used, as shown in figure 8. figure it is possible to check how the parameters of this command are defined, namely: The start coordinate on the X axis, the start coordinate on the Y axis, the length on the X axis, the

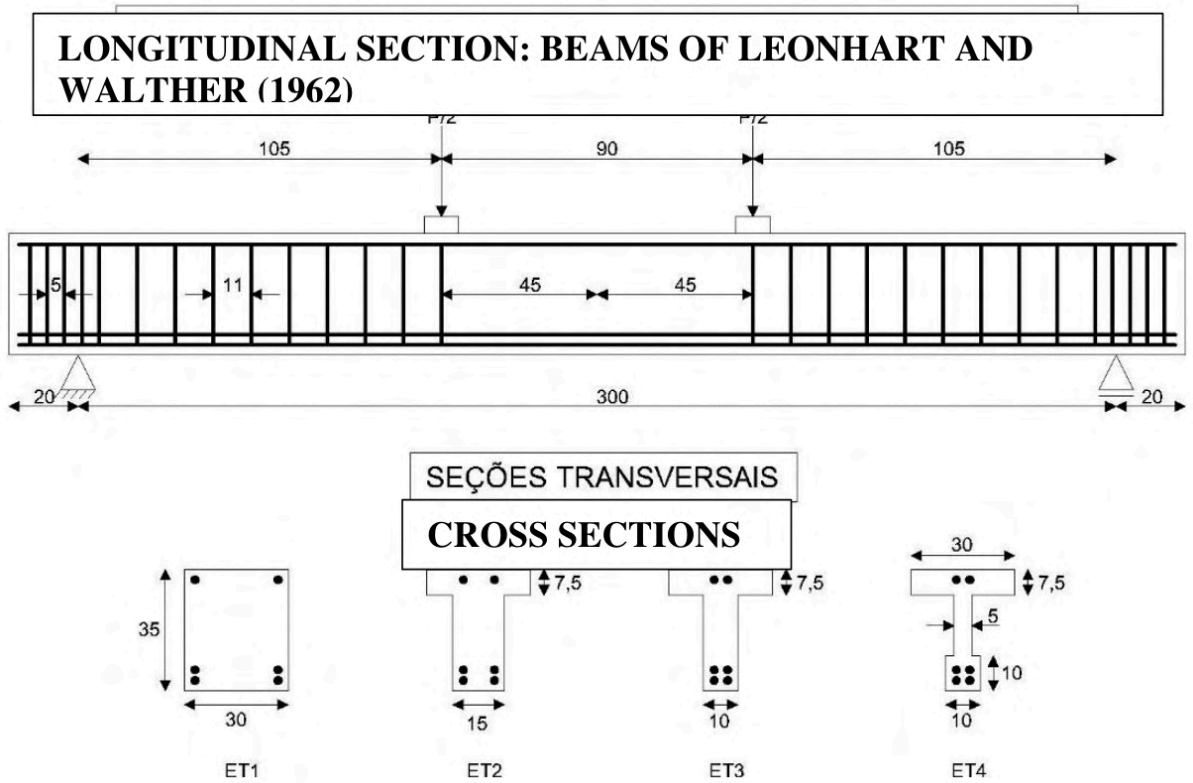


Figure 7 – beams in Leonhardt and Walther (1962) (quotas and measurements in centimeters).

Source: adapted from LEONHARDT; WALTHER, 1962.

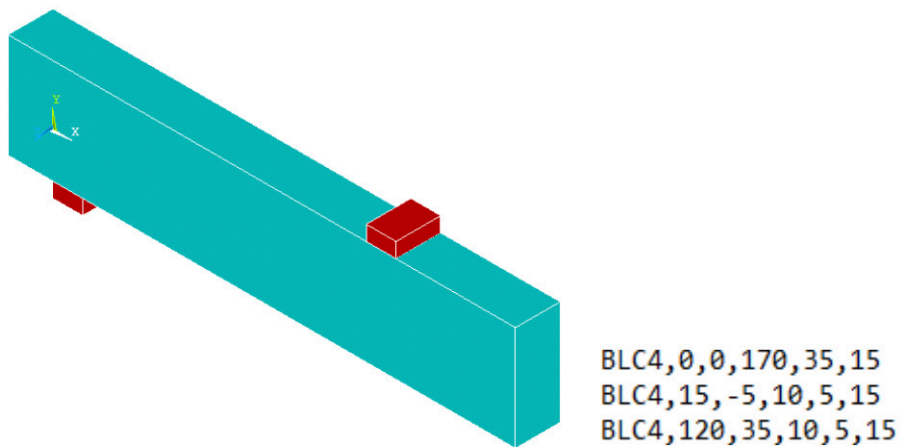


Figure 8 - Volumes in concrete and commands from launch in .txt format.

Source: elaborate by author.

length on the Y axis and finally the length on the Z axis. These five pieces of information are used for shape the desired volumes in the program.

To launch the inner lines, which formed the bars and stirrups, first *KEYPOINTS* are released. This command will create base points within the work plane of the program, where the first value is the point numbering, and the next three numbers are the localization us axes X, Y and Z, respectively. Soon after, are inserted commands for create lines in determined Score base until another. These procedures are presented at figure 9.

At three sections correspondents to the many different types in armor were included according to the commands of figure 10. On the first command line, are chosen at denominations of each section to be created and the type of element to be used is indicated. At line two, the cross-sectional areas of the bars are reported. And finally, on line three, the *SECCONTROL* are informed, where the ANSYS default values were used. In this work, sections 2, 3 and 4 indicate, respectively, the parameters adopted for the bars longitudinal lower, superior and stirrups, according presents the figure 10.

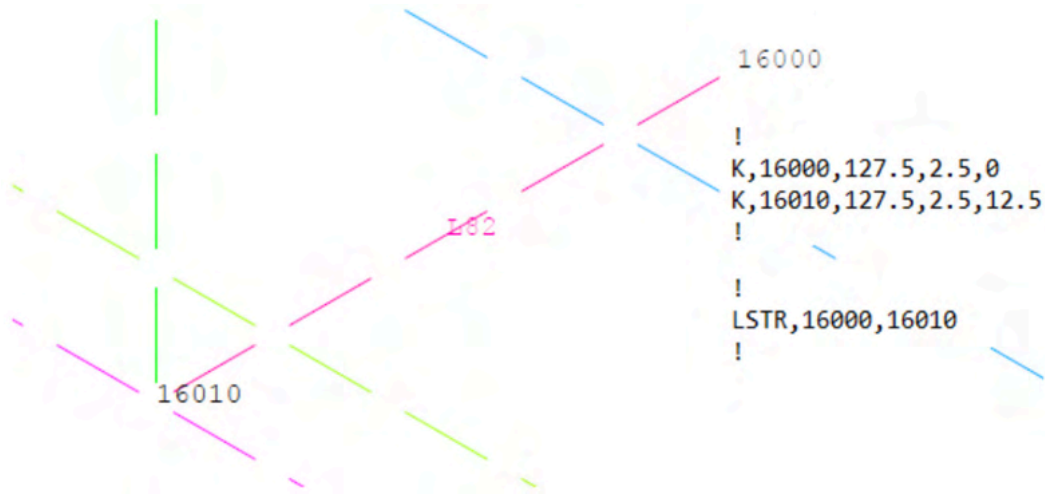


Figure 9 - Creation in lines and commands in launch in *.txt*

Source: elaborate by author.

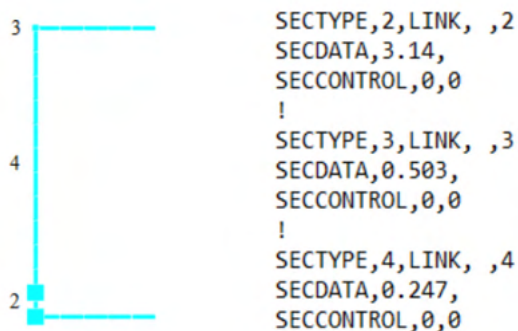


Figure 10 - Sections of bars in steel and commands in launch in Format *.txt*

Source: elaborate by author.

At the end, to include properties and sections from materials and to do the division from volumes and lines in finite elements, the *MESH command is used*. Figure 11 shows the commands for release of material properties in ANSYS. Through the *TYPE* command, you define type in element the to be adopted, according the order initial inserted at the program. At figure 11, the number 1 indicates the use of SOLID 65, and the number 2, the LINK 180. The commands *MAT*, *REAL* and *SECNUM* indicate, respectively, at properties From materials The be used;

The constant real; and The section transversal at the case of bars in armor. Soon below, command *LESIZE*, defines The line in what must to be incorporated at features and number in divisions needed for the finite element mesh. For concrete volumes, the command *MSHAPE*, defines the analysis as three-dimensional element and hexahedron, and the *VMESH* command, applies the generated parameters. As for the lines that represent the reinforcement bars, only if uses command *LMESH* _ For elaboration gives beam ET1, what It is shown at figure 12, were necessary 7370 finite elements.

```

!
TYPE, 1
MAT, 1
REAL, 1
ESYS, 0
SECNUM,
!
LESIZE,2, , ,68, , , ,1
LESIZE,4, , ,68, , , ,1
LESIZE,5, , ,68, , , ,1
LESIZE,7, , ,68, , , ,1
!
MSHAPE,0,3D
MSHKEY,1
!
VMESH,1
!
!
TYPE, 2
MAT, 4
REAL, 0
ESYS, 0
SECNUM, 4
!
! y
!
LESIZE,38, , ,12, , , ,1
LMESH,38
!

```

Figure 11 – *mesh* From volumes and lines per commands .txt

Source: elaborate by the author.

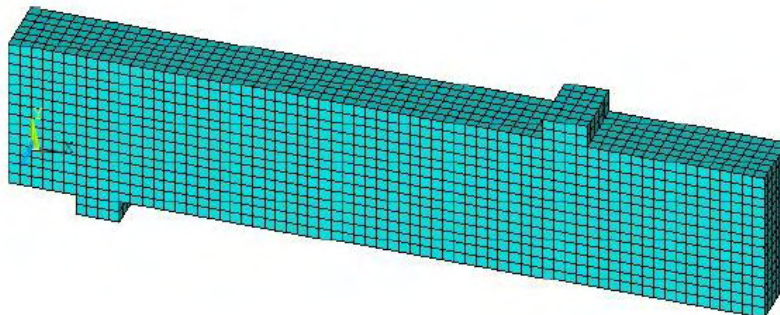


Figure 12 - Discretization of beam ET1.

Source: elaborate by the author.

Launch in properties From materials

Data in Prohibited for concrete were calculated according at equations already presented. The predicted modulus of elasticity for the concrete he was of 2886.53 kN/cm² (Equation 1). For concrete parameter, were used at properties described at table two. The shear transfer coefficients for open and closed cracks were taken from of experimental tests, according to Kunzler (2013). On the other hand, the items represented with null value, indicate that went adopted you values presets fur ANSYS. For the definition of parameter MELAS, got employed you values gives curve voltage deformation of concrete pill,

according The table 3. You values were calculated according equations already presented in item 2.3.1.2 this work.

To characterize the behavior of the steel, different values were used for each reinforcement within the *Linear Isotropic* and BISO parameters. Table 4 presents the data informed for each armor, in wake up with you values calculated and properties already clarified in the experimental process presented. The hardening modulus value of the stirrups, it was adopted, in order to eliminate the slope of the section after the yield stress initial, because the steel in question obeys to the model perfect elastoplastic.

The) Coefficient in transfer inshear for fissure open;	0.2	f) state in voltage hydrostaticenvironment;	0
B) Coefficient in transfer inshear for fissure closed;	0.4	g) Resistance last biaxial The compression under O state in voltagehydrostatic environment;	0
ç) Resistance last uniaxial Thetraction;	0.25	H) Resistance last uniaxial Thecompression under O state in voltage hydrostatic environment;	0
d) Resistance last uniaxial Thecompression;	-1	i) Coefficient multiplier instiffness for condition fissured at traction.	0
and) Resistance last biaxial Thecompression;	0		

Table 2 - Values of parameter *concrete*.

Source: KUNZLER, 2013.

Voltage(σ)	Deformation (ϵ)	Voltage(σ)	Deformation (ϵ)	Voltage(σ)	Deformation (ϵ)
0.23	0.0001	2.05	0.0013	2.38	0.0025
0.45	0.0002	2.13	0.0014	2.35	0.0026
0.65	0.0003	2.20	0.0015	2.31	0.0027
0.84	0.0004	2.26	0.0016	2.27	0.0028
1.02	0.0005	2.31	0.0017	2.22	0.0029
1.19	0.0006	2.35	0.0018	2.15	0.003
1.35	0.0007	2.38	0.0019	2.09	0.0031
1.49	0.0008	2.40	0.002	2.01	0.0032
1.62	0.0009	2.42	0.0021	1.93	0.0033
1.75	0.001	2.42	0.0022	1.84	0.0034
1.86	0.0011	2.42	0.0023	1.74	0.0035
1.96	0.0012	2.40	0.0024		

Painting 3 - Values gives curve voltage deformation of concrete pill (Units in voltage in kN/cm²).

Source: elaborate by the author.

	Longitudinal lower	longitudinal superiors	stirrups vertical
Poisson	0.3	0.3	0.3
Voltage outflow initial	36.38 kN/cm ²	39.53 kN/cm ²	32.00 kN/cm ²
Module in hardening	789.24 kN/cm ²	874.82 kN/cm ²	0
Module in elasticity	19500 kN/cm ²	19500 kN/cm ²	21000 kN/cm ²

Painting 4 – Values applied in the armor.

Source: elaborate by the author.

Restrictions

For decrease, the computational effort, enjoying at conditions in symmetry of parts shown in figure 7, only a quarter of the total volume of each beam. In the center of the support plate, displacements in the Y direction were restricted. located at the Final of bedroom in beam, were restricted you displacements at direction X. In addition, the displacements in the Z direction of all nodes belonging to the side of the ask, simulating continuity of the element in this direction.

Shipments

Were applied at routine of *software*, two phases in loading. Firstly he was applied at beam The deformation instantaneous due to Weight own of concrete armed, incorporating the acceleration of gravity in conjunction with the *density property*. Right after, for simulate The application gives charge concentrated up until The break gives beam, were applied successive displacements vertical ourselves spots in what were applied at the rehearsal experimental.

RESULTS

The diagram in charge and displacement obtained the leave of ANSYS, he was generated multiplying by four the results obtained as reactions at the support node. The values of displacements were measured at the internal node of the beam, at the end

opposite the support, where locate The a half of your they go. At figure 13 It is possible observe at curves in load-displacement gives beam ET1 and The evolution of arrows at referred beam to far away From increments in displacement, thus being exposed that the model adopted for modeling the beams is approaches of the experimental tests, proving The effectiveness thereof.

At figure 14, It is presented The deformation maximum hit at beam ET1 at the instant Where structure rupture was identified. This deformed represents the arrow hit at the moment of your collapse. The breaking load was 235 kN, with a deformation of 1.31 cm at the center of the vain.

The figure 15 bring at tensions imposed in the armor longitudinal gives beam ET1 at the instant of the break. The bar reaches the yield stress at the moment of failure, being considered so rupture per flexion.

At figure 16, they are represented at tensions active only us stirrups gives beam ET1, at the time of rupture. It was found that the stirrups did not reach the yield stress of 32 kN/cm², which proves what there was not break by shear.

Figure 17 shows the crack opening diagram for beam ET1. the observation gives direction of those cracks at the instant gives break, Assists at the process in analysis of beam behavior. Cracks are represented by

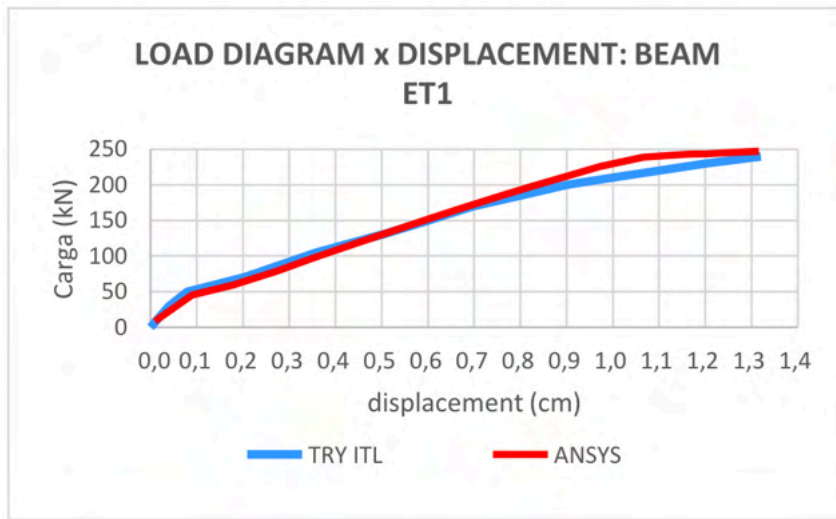


Figure 13 – Comparison gives arrow of beams experimental with computational model
 Source: elaborate by the author.

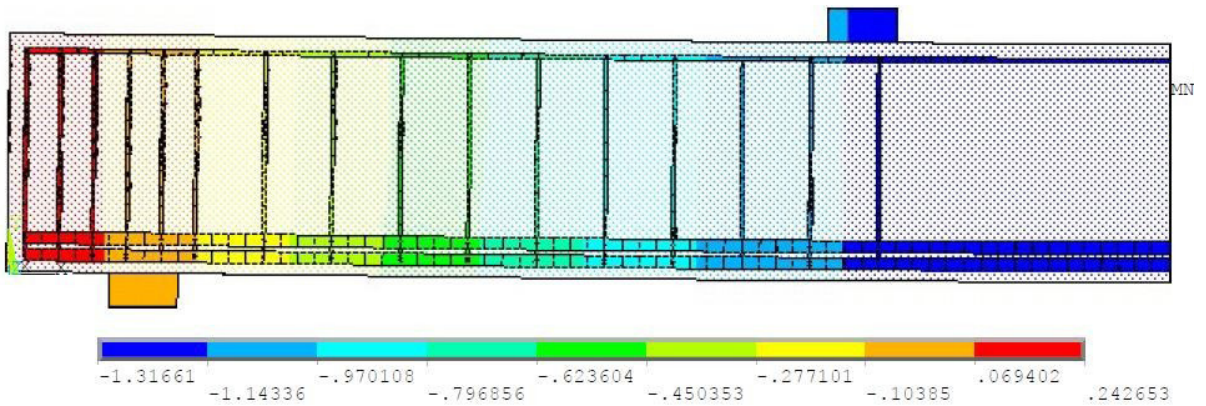


Figure 14 – deformed at the instant in break gives beam ET1.
 Source: elaborate by the author.

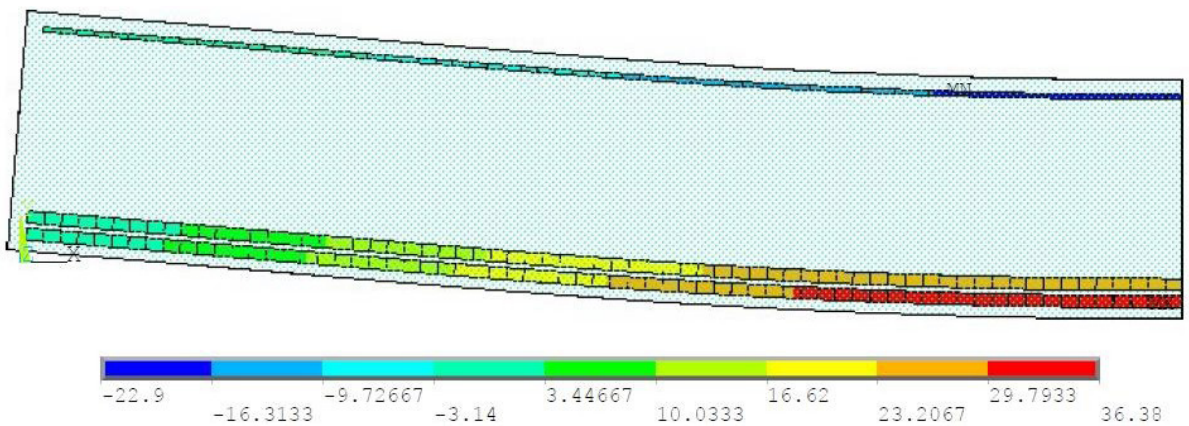


Figure 15 - Voltage in the armor longitudinal gives beam ET1.
 Source: elaborate by the author.

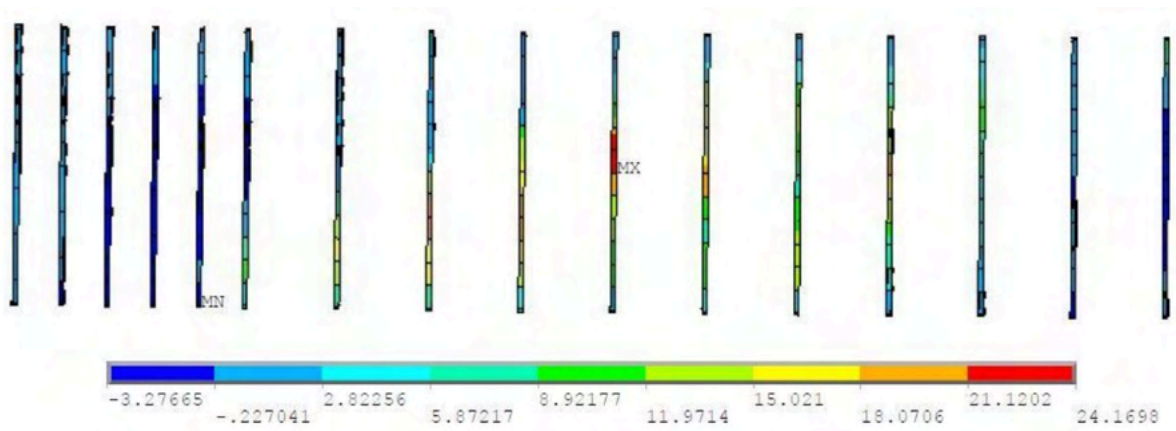


Figure 16 – Voltage in the armor transversal gives beam ET1.

Source: elaborate by the author.

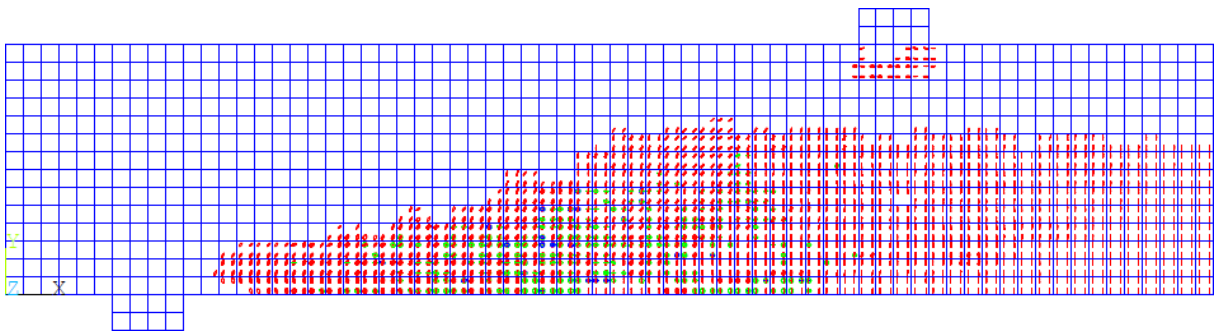


Figure 17– beam cracking diagrams ET1.

Source: elaborate by the author.

the color red, the vertical lines, are features bending, that was evidenced mainly in the center of the beam. Already the slopes are shear, observed at the points closest to the support. you can view also at this figure.

In this table, three loading stages were analyzed, in order to highlight the evolution of tensions at the concrete.

Predominant in flexion as contribution for the break gives ask. The results complete of too much beams (ET2, ET3 and ET4) they are presented in the appendix (B).

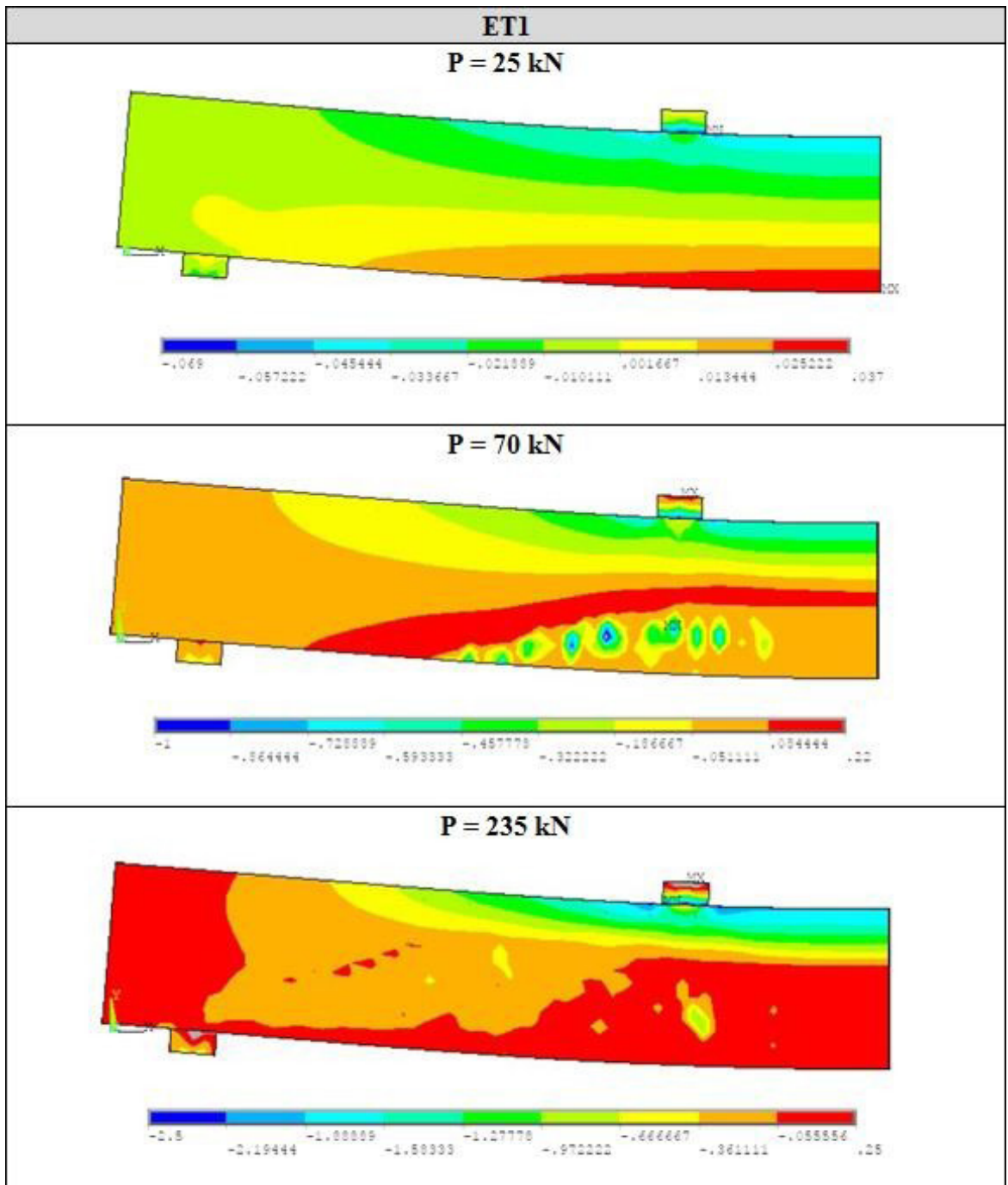
The leave of analysis made, stay evident in the mechanisms in break in each beam. For at ET1, there is bending collapse, since the lower longitudinal reinforcement is in

the level in flow While you stirrups they are with a low loading. Analyzing you results of beams ET2 and ET3, It is possible highlight what both They arrived to limit in charge per shear, then you stirrups reached The charge in break, although of bars inferior also are very close to the yield stress. And finally, ET4, shear ruptures with the flow of armor vertical, without what at bars inferior reach the voltage limit.

CONCLUSIONS

FINAL CONSIDERATIONS

This article aimed to perform a three-dimensional numerical analysis, in the beams gives series ET rehearsed per Leonhardt



Painting 5 – Evolution gives voltage σ_x at the concrete for The beam ET1 (units in kN/cm^2).

Ssource: elaborate by the author.

SUGGESTIONS FOR WORKS FUTURES

Aiming the improvement of this job, it is foreseen the use since model for analysis of other experimental trials. Among them are the tested beams: for Bresler and Scordelis (1963); the prestressed concrete beams by Tao and Du (1985); and the beams of concrete armed with holes of Mansur (2006).

As suggestions for advancing the study, it is suggested the analysis of structures with prestressing, be they adherents or no adherents to far away of time, so much at area experimental as at numeric. Other suggestion would be The application in armor incorporated and distributed with future Comparison in results with the armor discreet applied in this article.

and Walther (1962), through of *software* in analysis via finite elements, ANSYS, which achieved convergence with the experimental results already cited. The model he was developed through in launch in codes. *txt*, you which if are available in the appendix of this work, aiming to enable the improvement of the simulations.

This model was based on the work developed by Lazzari (2015) and Kunzler (2013), both inside of PPGE/UFRRGS. Were updated and revised you Data From models implemented, for simulating reinforced concrete beams, subjected to short and long duration, according to the Model Code *fib* 2010 (2012).

From this work, it was possible, within the ANSYS *software*, to create a model computation for simulation of experimental beams. With the division of the structure, being used only a quarter beam in the analysis, the computational effort was reduced and the time in analysis optimized. The program showed much efficient also per to have one library in elements and behaviors very vast and with innumerable possibilities of simulations of materials. Another point to highlight from ANSYS and the ways of viewing the results after the analysis to have been completed, owning tools graphics interesting, at which make in easy identification the results that if want to analyze.

The one coverage great in behaviors, among them, failures due to bending and shear can be highlighted. In view of the good results obtained in this job, ensure that the possibility in simulation in others structures in concrete armed.

REFERENCES

ANSYS, Inc. **Theory reference** (Version 18.0), 2017

AURICH, M. **Modelo da ligação entre concreto e armadura na análise de estruturas de concreto pelo método dos elementos finitos**. 2001. 132 f. Dissertação (Mestrado em Engenharia) – Programa de Pós-Graduação em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Porto Alegre, 2001.

AZEVEDO, F.M.A. **Método dos elementos finitos**, 1.ed. Porto: Faculdade de Engenharia do Porto, 2003.

COMITÉ EURO-INTERNATIONAL DU BÉTON. **CEB-FIP Model code 2010**. Bulletin N° 65, 2012.

GONÇALVES JUSTO, R.A.E. **Comparação de desempenho e de funcionamento de Programas Comerciais (ANSYS e SAP2000) focada à análise sísmica**. 2010. 95 f. Dissertação (Mestrado em Engenharia) – Universidade nova de Lisboa, Lisboa, 2010.

KUNZLER, P.S. **Análise paramétrica por elementos finitos de vigas de concreto armado e protendido pré-tracionadas com abertura na alma**. 2013. 112 f. Dissertação (Mestrado em Engenharia) – Programa de Pós-Graduação em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Porto Alegre, 2013.

LAZZARI, B.M. **Análise por elementos finitos de peças de concreto armado e protendido sob estados planos de tensão**. 2015. 171 f. Dissertação (Mestrado em Engenharia) – Programa de Pós-Graduação em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Porto Alegre, 2015.

LAZZARI, P.M. **Simulação numérica das etapas construtivas de pontes estaiadas através do método dos elementos finitos**. 2016. 276 f. Tese (Doutorado em Engenharia) – Programa de Pós-Graduação em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Porto Alegre, 2016.

LEONHARDT, F.; WALTHER, R. **Beiträge zur behandlung der Schubprobleme im Stahlbetonbau. Beton und Stahlbetonbau**. v. 57. n. 7. Jul. 1962.

SORIANO, H. L.; LIMA S.S. **Método dos elementos finitos em análise de estruturas**. 392 p. 1998

APPENDIX THE – SCRIPTS ANSYS ADPL 18.0

A.1 EXAMPLE IN *SCRIPT* FOR BEAM ET1 IN LEONHARDT AND WALTHER (1962)

```
! *****
!  
! Programmer: Henrique cardoso Koch  
! Date: 06/19/17  
! units: cm, kN  
!  
! *****  
!  
/NOPR      ! suppress printing of UNDO process  
/PMACRO    ! Echo following commands to log FINISH ! make sure we are at  
BEGIN level  
/CLEAR,NOSTART ! clear model sincerely no SAVE found  
/NO PR  
/PMETH,OFF,0  
/title, BEAM LEONHARDT ET1  
!  
! .....  
!      DEFINITION OF TYPE IN ANALYSIS  
! .....  
! KEYW,PR_SET,1  
KEYW,PR_STRUC,1  
!  
/GO  
!  
! .....  
!      DEFINITION ELEMENTS  
! .....  
/PREP7  
!* ET,1,SOLID65  
!* ET,2,LINK180  
!* ET,3,SOLID185  
!*  
!  
! .....  
!      PARAMETERS FROM MATERIALS  
! .....  
!  
! .....  
! CONCRETE - material 1  
! .....
```

MPTEMP,,,,,,,,,
MPTEMP,1.0
MPDATA,EX,1,,2886.53
MPDATA,PRXY,1,,0.2
TB, MELA, 1, 1, 35, 0
TBTEMP,0
TBPT,,0.0001,0.229932
TBPT,,0.0002,0.446906
TBPT,,0.0003,0.651118
TBPT,,0.0004,0.843009
TBPT,,0.0005,1.022637
TBPT,,0.0006,1.190306
TBPT,,0.0007,1.346247
TBPT,,0.0008,1.49069
TBPT,,0.0009,1.623854
TBPT,,0.001,1.745957
TBPT,,0.0011,1.857209
TBPT,,0.0012,1.957815
TBPT,,0.0013,2.047975
TBPT,,0.0014,2.127884
TBPT,,0.0015,2.197732
TBPT,,0.0016,2.257705
TBPT,,0.0017,2.307984
TBPT,,0.0018,2.348745
TBPT,,0.0019,2.380161
TBPT,,0.002,2.4024
TBPT,,0.0021,2.415626
TBPT,,0.0022,2.42
TBPT,,0.0023,2.415678
TBPT,,0.0024,2.402813
TBPT,,0.0025,2.381553
TBPT,,0.0026,2.352046
TBPT,,0.0027,2.314434
TBPT,,0.0028,2.268855
TBPT,,0.0029,2.215446
TBPT,,0.003,2.15434
TBPT,,0.0031,2.085666
TBPT,,0.0032,2.009552
TBPT,,0.0033,1.926122
TBPT,,0.0034,1.835498
TBPT,,0.0035,1.737798
TB,CONC,1,1,9,
TBTEMP,0
TBDATA,,0.20,0.40,0.25,-1,,

```

TBDATA,,,,,,,,,
MPTEMP,,,,,,,,, MPTEMP,1.0
MPDATA,DENS,1,,2.54E-8
!
!.....
! ARMOR ACTIVE (BOTTOM) - material two
!.....
!
!.....-
! PARAMETERS STEEL
!.....
! v1=0.3
ES2 = 19500
FY2 = 42.8*0.85
FY22 = 42.8
ES22 = .15*FY22/(0.01-FY2/ES2)
!
! Model Linear isotropic
!
MPTEMP,,,,,,,,,
MPTEMP,1.0
MPDATA,EX,2,,ES2 MPDATA,PRXY,2,,V1
!
! Model bilinear - goes into with The voltage initial and slope gives curve in hardening
!
TB,BISO,2,1,2,
TBTEMP,0
TBDATA,,FY2,ES2,,,,
!
!.....
! ARMOR PASSIVE (HIGHER) - material 3
!.....
!
!.....
! PARAMETERS STEEL
!.....
!
v2=0.3
ES3 = 19500
FY3 = 46.5*0.85
FY33 = 46.5
ES33 = .15*FY33/(0.01-FY3/ES3)
!
! Model Linear isotropic

```

```

!
MPTEMP,,,,,,,,
,MPTEMP,1.0
MPDATA,EX,3,,ES3
MPDATA,PRXY,3,,V2
!
! Model bilinear - goes into with The voltage initial and slope gives curve in hardening
!
TB,BISO,3,1,2,
TBTEMP,0
TBDATA,,FY33,ES33,,
!
! .....
! ARMOR STEPS - material 4
! .....
!
! .....
! PARAMETERS STEEL
! .....
!
v3=0.3
ES4 = 21000
FY4 = 32
ES44 = 0
!
! Model Linear isotropic
!
MPTEMP,,,,,,,,
,MPTEMP,1.0
MPDATA,EX,4,,ES4
MPDATA,PRXY,4,,v3
!
! Model bilinear - goes into with The voltage initial and slope gives curve in hardening
!
TB,BISO,4,1,2,
TBTEMP,0
TBDATA,,FY4,ES44,,
!
! .....
! SUPPORTS AND SPOTS IN CHARGE - material 5

!
! .....
v4=0.2

```

```

ES5 = 20000
! MPTEMP,,,,,,,,,
,MPTEMP,1.0
MPDATA,EX,5,,ES5
MPDATA,PRXY,5,,v4
!
! .....
!          GEOMETRY
! .....
!
! CONCRETE VOLUMES (coordinate X, coordinate and, compress x, compress y, length z)
!
BLC4,0,0,170,35,15
BLC4,15,-5,10,5,15
BLC4,120,35,10,5,15
!
K,1000,2.5,2.5,0 ! keypoint 1000 in x=2.5, y=2.5 and
z=0 K,1010,2.5,2.5,12.5
K,1020,2.5,5,12.5
K,1030,2.5,32.5,0
K,1040,2.5,32.5,12.5
!
K,2000,7.5,2.5,0
K,2010,7.5,2.5,12.5
K,2020,7.5,5,12.5
K,2030,7.5,32.5,0
K,2040,7.5,32.5,12.5
!
K,3000,12.5,2.5,0
K,3010,12.5,2.5,12.5
K,3020,12.5,5,12.5
K,3030,12.5,32.5,0
K,3040,12.5,32.5,12.5
!
K,4000,17.5,2.5,0
K,4010,17.5,2.5,12.5
K,4020,17.5,5,12.5
K,4030,17.5,32.5,0
K,4040,17.5,32.5,12.5
!
K,5000,22.5,2.5,0
K,5010,22.5,2.5,12.5
K,5020,22.5,5,12.5
K,5030,22.5,32.5,0

```

K,5040,22.5,32.5,12.5

!

K,6000,27.5,2.5.0

K,6010,27.5,2.5,12.5

K,6020,27.5,5,12.5

K,6030,27.5,32.5.0

K,6040,27.5,32.5,12.5

!

K,7000,37.5,2.5.0

K,7010,37.5,2.5,12.5

K,7020,37.5,5,12.5

K,7030,37.5,32.5.0

K,7040,37.5,32.5,12.5

!

K,8000,47.5,2.5.0

K,8010,47.5,2.5,12.5

K,8020,47.5,5,12.5

K,8030,47.5,32.5.0

K,8040,47.5,32.5,12.5

!

K,9000,57.5,2.5.0

K,9010,57.5,2.5,12.5

K,9020,57.5,5,12.5

K,9030,57.5,32.5.0

K,9040,57.5,32.5,12.5

!

K,10000,67.5,2.5.0

K,10010,67.5,2.5,12.5

K,10020,67.5,5,12.5

K,10030,67.5,32.5.0

K,10040,67.5,32.5,12.5

!

K,11000,77.5,2.5.0

K,11010,77.5,2.5,12.5

K,11020,77.5,5,12.5

K,11030,77.5,32.5.0

K,11040,77.5,32.5,12.5

!

K,12000,87.5,2.5.0

K,12010,87.5,2.5,12.5

K,12020,87.5,5,12.5

K,12030,87.5,32.5.0

K,12040,87.5,32.5,12.5

!

K,13000,97.5,2.5.0
K,13010,97.5,2.5,12.5
K,13020,97.5,5,12.5
K,13030,97.5,32.5.0
K,13040,97.5,32.5,12.5

!

K,14000,107.5,2.5.0
K,14010,107.5,2.5,12.5
K,14020,107.5,5,12.5
K,14030,107.5,32.5.0
K,14040,107.5,32.5,12.5

!

K,15000,117.5,2.5.0
K,15010,117.5,2.5,12.5
K,15020,117.5,5,12.5
K,15030,117.5,32.5.0
K,15040,117.5,32.5,12.5

!

K,16000,127.5,2.5.0
K,16010,127.5,2.5,12.5
K,16020,127.5,5,12.5
K,16030,127.5,32.5.0
K,16040,127.5,32.5,12.5

!

K,17010,170,2.5,12.5
K,17020,170,5,12.5
K,17040,170,32.5,12.5

!

!

! moth lines in between you spots created

!

! STEPS

!

LSTR,1000,1010
LSTR,1010,1040
LSTR,1040,1030

!

LSTR,2000,2010
LSTR,2010,2040
LSTR,2040,2030

!

LSTR,3000,3010
LSTR,3010,3040
LSTR,3040,3030

!
LSTR,4000,4010
LSTR,4010,4040
LSTR,4040,4030
!
LSTR,5000,5010
LSTR,5010,5040
LSTR,5040,5030
!
LSTR,6000,6010
LSTR,6010,6040
LSTR,6040,6030
!
LSTR,7000,7010
LSTR,7010,7040
LSTR,7040,7030
!
LSTR,8000,8010
LSTR,8010,8040
LSTR,8040,8030
!
LSTR,9000,9010
LSTR,9010,9040
LSTR,9040,9030
!
LSTR,10000,10010
LSTR,10010,10040
LSTR,10040,10030
!
LSTR,11000,11010
LSTR,11010,11040
LSTR,11040,11030
!
LSTR,12000,12010
LSTR,12010,12040
LSTR,12040,12030
!
LSTR,13000,13010
LSTR,13010,13040
LSTR,13040,13030
!
LSTR,14000,14010
LSTR,14010,14040
LSTR,14040,14030

```

!
LSTR,15000,15010
LSTR,15010,15040
LSTR,15040,15030
!
LSTR,16000,16010
LSTR,16010,16040
LSTR,16040,16030
!
! BARS
!
LSTR,1010,17010
LSTR,1020,17020
LSTR,1040,17040
!
! .....
!           SECTION
! .....
! Definition of thicknesses in bars in cm2
!
SECTYPE,2,LINK, ,2
SECDATA, 3.14,
SECCONTROL,0.0
!
SECTYPE,3,LINK, ,3
SECDATA,0.503,
SECCONTROL,0.0
!
SECTYPE,4,LINK, ,4
SECDATA,0.247,
SECCONTROL,0.0
!
! .....
!           MESH
! .....
! .....
! CONCRETE MESH:
! .....
!
! Specific at features From elements 3D:
!
TYPE, 1

```

```

MAT, 1
REAL, 1
ESYS, 0
SECNUM,
!
! x
!
LESIZE,2,,68,,,,1 !select bar "X" and divide in 68 parts
LESIZE,4,,68,,,,1
LESIZE,5,,68,,,,1
LESIZE,7,,68,,,,1
!
! and
!
LESIZE,1,,14,,,,1 !select bar "Y" and divide in 14 parts
LESIZE,3,,14,,,,1
LESIZE,6,,14,,,,1
LESIZE,8,,14,,,,1
!
! z
!
LESIZE,9,,6,,,,1 !select bar "Z" and divide in 6 part
LESIZE,10,,6,,,,1
LESIZE,11,,6,,,,1
LESIZE,12,,6,,,,1
!
! options gives mesh:
!
MSHAPE,0,3D ! define 3D problem and define hexahedron
element MSHKEY,1 ! defines mesh mapped
!
VMESH,1 !apply The mesh 1
!
!.....
! MESH SUPPORTS:
!.....
!
! Specific at features From elements 3D:
!
TYPE, 3
MAT, 5
REAL, 1
ESYS, 0
SECNUM,

```

```

!
! x
!
LESIZE,14,,4,,,,1
LESIZE,16,,4,,,,1
LESIZE,17,,4,,,,1
LESIZE,19,,4,,,,1
!
! and
!
LESIZE,13,,two,,,,1
LESIZE,15,,two,,,,1
LESIZE,18,,two,,,,1
LESIZE,20,,two,,,,1
!
! z
!
LESIZE, 21,,6,,,,1
LESIZE,22,,6,,,,1
LESIZE, 23,,6,,,,1
LESIZE,24,,6,,,,1
!
!
! options gives mesh:
!
MSHAPE,0,3D ! define 3D problem and define hexahedron
element MSHKEY,1          ! defines mesh mapped
!
VMESH,2 !apply The mesh two
!
!.....
! MESH SUPPORTS:
!.....
!
! Specific at features From elements 3D:
!
TYPE, 3
MAT, 5
REAL, 1
ESYS, 0
SECNUM,
!
! x
!

```

```

LESIZE, 26, , ,4, , , , ,1
LESIZE, 28, , ,4, , , , ,1
LESIZE, 29, , ,4, , , , ,1
LESIZE,31, , ,4, , , , ,1
!
! and
!
LESIZE, 25, , ,two, , , , ,1
LESIZE, 27, , ,two, , , , ,1
LESIZE,30, , ,two, , , , ,1
LESIZE,32, , ,two, , , , ,1
!
! z
!
LESIZE,33, , ,6, , , , ,1
LESIZE,34, , ,6, , , , ,1
LESIZE,35, , ,6, , , , ,1
LESIZE,36, , ,6, , , , ,1
!
!
! options gives mesh:
!
MSHAPE,0,3D ! define 3D problem and define hexahedron
element MSHKEY,1          ! defines mesh mapped
!
VMESH,3 !apply The mesh 3
!
!.....
! MESH ARMOR - BARS:
!.....
!
! Specific at features From elements 3D:
!
TYPE, two
MAT, 2
REAL, 0
ESYS, 0
SECNUM, two
!
! x
!
LESIZE,85, , ,67, , , , ,1
LESIZE,86, , ,67, , , , ,1
!

```

```

! options gives mesh:
!
MSHAPE,0,3D ! define 3D problem and define hexahedron
element MSHKEY,1          ! defines mesh mapped
!
LMESH,85.86
!
! Specific at features From elements 3D:
!
TYPE, two
MAT, 3
REAL, 0
ESYS, 0
SECNUM, 3
!
! x
!
LESIZE,87, , ,67, , , ,1
!
! options gives mesh:
!
MSHAPE,0,3D ! define 3D problem and define hexahedron
element MSHKEY,1          ! defines mesh mapped
! LMesh,87
!
! Specific at features From elements 3D:
!
TYPE, two
MAT, 4
REAL, 0
ESYS, 0
SECNUM, 4
!
! and
!
LESIZE,38, , ,12, , , ,1
LESIZE,41, , ,12, , , ,1
LESIZE,44, , ,12, , , ,1
LESIZE,47, , ,12, , , ,1
LESIZE,50, , ,12, , , ,1
LESIZE,53, , ,12, , , ,1
LESIZE,56, , ,12, , , ,1
LESIZE,59, , ,12, , , ,1
LESIZE,62, , ,12, , , ,1

```

LESIZE,65,,12,,,,1
LESIZE,68,,12,,,,1
LESIZE,71,,12,,,,1
LESIZE,74,,12,,,,1
LESIZE,77,,12,,,,1
LESIZE,80,,12,,,,1
LESIZE,83,,12,,,,1
!
! z
!
LESIZE,37,,5,,,,1
LESIZE,39,,5,,,,1
LESIZE,40,,5,,,,1
LESIZE,42,,5,,,,1
LESIZE,43,,5,,,,1
LESIZE,45,,5,,,,1
LESIZE,46,,5,,,,1
LESIZE,48,,5,,,,1
LESIZE,49,,5,,,,1
LESIZE,51,,5,,,,1
LESIZE,52,,5,,,,1
LESIZE,54,,5,,,,1
LESIZE,55,,5,,,,1
LESIZE,57,,5,,,,1
LESIZE,58,,5,,,,1
LESIZE,60,,5,,,,1
LESIZE,61,,5,,,,1
LESIZE,63,,5,,,,1
LESIZE,64,,5,,,,1
LESIZE,66,,5,,,,1
LESIZE,67,,5,,,,1
LESIZE,69,,5,,,,1
LESIZE,70,,5,,,,1
LESIZE,72,,5,,,,1
LESIZE,73,,5,,,,1
LESIZE,75,,5,,,,1
LESIZE,76,,5,,,,1
LESIZE,78,,5,,,,1
LESIZE,79,,5,,,,1
LESIZE,81,,5,,,,1
LESIZE,82,,5,,,,1
LESIZE,84,,5,,,,1
!
LMESH,37

LMESH,38
LMESH,39
LMESH,40
LMESH,41
LMESH,42
LMESH,43
LMESH,44
LMESH,45
LMESH,46
LMESH,47
LMESH,48
LMESH,49
LMESH, 50
LMESH,51
LMESH,52
LMESH,53
LMESH,54
LMESH,55
LMESH,56
LMESH,57
LMESH,58
LMESH,59
LMESH,60
LMESH,61
LMESH,62
LMESH,63
LMESH,64
LMESH,65
LMESH,66
LMESH,67
LMESH,68
LMESH,69
LMESH,70
LMESH,71
LMESH,72
LMESH,73
LMESH,74
LMESH,75
LMESH,76
LMESH,77
LMESH,78
LMESH,79
LMESH, 80
LMESH,81

```

LMESH,82
LMESH,83
LMESH,84
!
! options gives mesh:
!
MSHAPE,0,3D ! define 3D problem and define hexahedron
element MSHKEY,1          ! defines mesh mapped
!
! That command execute a merge From us - eliminate us repeated
! Does the renumbering From us
!
NUMMRG,NODE, , , ,LOW NUMCMP,NODE
!
! Inspect newly created reinforcing elements
!
esel,s,type,,1
! adjust the translucency level of the
! base element to reveal the embedded reinforcing
! elements
/trlcy,elem,0.9 esel, all
/view,1,1,1,1
! turn on the expanded element shapes
/eshape,1 eplot
!
! .....
!          RESTRICTIONS US SUPPORTS
! .....
!
nselect,s,loc,x,20
nselect,r,loc,y,-5
d,all,uy
!
DA,6,UX,
DA,1,UZ,
DA,7,UZ,
DA,13,UZ,
allsel
!
! .....
!          LOADING
! .....

```

```

! .....
! 1st CASE OF CHARGE - Stage 1 - Weight Own
! .....
!
/PREP7
!
ALLSEL,ALL
ACEL,0.981,0,
!
nset,s,loc,x,125
nset,r,loc,y,40
nset,r,loc,z,0
f,all,fy,-0.0
allsel, all
!
antype,0 !      Static analysis solcontrol,on
!
autots, on nlgeom,on
NCNV,2,0,0,0,0      !stop criterion to end the analysis neqit, 100
CNVTOL,F, ,0.1,2, ,
CNVTOL,U, ,0.1,2, , NSUBST,1,100,1 OTHERS, ERASE OTHERS, ALL, 1 AUTOTS,1 EQSLV,
SPAR LNSRCH,0
! RESCONTRL,DEFINE,ALL,1,1
TEAM, 1 LSWRITE,1,
!
! .....
!      2nd CASE OF CHARGE - Stage two - increment of DISPLACEMENT
! .....
!
/PREP7
!
nset,s,loc,x,125 nset,r,loc,y,40 nset,r,loc,z,0 d,all,uy,-2 allsel
!
antype,0 solcontrol,on
!
autots, on
CNVTOL,F, ,0.4,2, ,
CNVTOL,U, ,0.4,2, ,
NSUBST,25,100,25
OTHERS, ERASE
OTHERS, ALL, 1
AUTOTS,1 EQSLV,
SPAR LNSRCH,0
NCNV,2,0,0,0,0

```

```

NEQIT, 100
!
RESCONTRL,DEFINE,ALL,1,1
team, 25 LSWRITE,2,

```

A.2 EXAMPLE IN SCRIPT FOR DRAW DIAGRAM LOAD-DISPLACEMENT – POST PROCESSING

```

/POST26
!
! Routine for if to do O graphic load-displacement When if use increment in displacement
!
!
*SET,FACTOR1,-1.00 ! Factor for multiply O displacement if necessary (per example: become
positives you displacements)
*SET,FACTOR2,4.00 ! Factor for multiply The charge if required (per example: Ptotal = 4xP)
*SET,NNUY,84 ! Node number of which you want to make the load-displacement graph
(see list of we with coordinates)
!
/AXLAB,Y,LOAD Q ! label Y
/AXLAB,X,ARROW UY! label X
NSOL,2,NNUY,U,y ! Specifies the node whose Uy displacement will be plotted.
NSOL,3,1,U,Z ! Variable mute, it serves to store the sum of the reactions (corresponds to the
UZ offset of at the 1).
!
NSEL,S,LOC,x,20 ! Selects nodes whose y coordinate is equal to -2.50 (choose the coordinate
in where are at reactions).
NSEL,R,LOC,y,-5
*GET,NUMNOD,NODE,0,COUNT ! Catch you numbers From we in Where are the reactions.
*GET,CURNOD,NODE,0,NUM,MIN !Capture at coordinates From we Where are the
reactions in support.
!
*DO,I,1,NUMNOD ! bow that does The sum of reactions Fy From we selected before.
RFORCE,4,CURNOD,F,y
ADD,3,3,4,SUM CURNOD=NDNEXT(CURNOD)
*ENDDO
!
PROD,2,2,,,,,FACTOR1 ! Adjustment From displacements fur factor FACTOR 1
PROD,3,3,,,,,FACTOR2 ! Adjustment of loads by the factor FACTOR2
!
XVAR,2 ! plot you displacements in axle X
PLVAR,3 ! plots the charge total at the Y axis Y
!
ALLSEL,ALL
FINISH

```

APPENDIX B – BEAMS RESULTS ET2, ET3 AND ET4

B.1 ET2

In figure 18 it is possible to observe the load-displacement curves of the ET2 beam and the evolution of arrows at referred beam to far away From increments in displacement. Staying so exposed what model adopted for modeling of beams if approaches from essay experimental proving the effectiveness thereof.

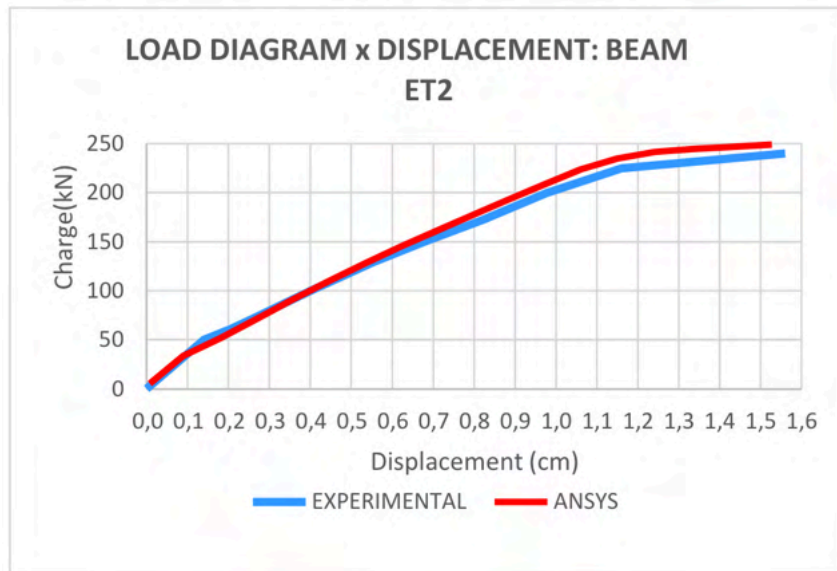


Figure 18 – Comparation gives arrow of beams experimental with computational model

Source: elaborate by the author.

At figure 19, It is presented The deformation maximum hit at beam ET2 at the instant Where structure rupture was identified. This deformed represents the arrow hit at the moment of your collapse. The breaking load was 248 kN, with a deformation of 1.53 cm at the center of the vain.

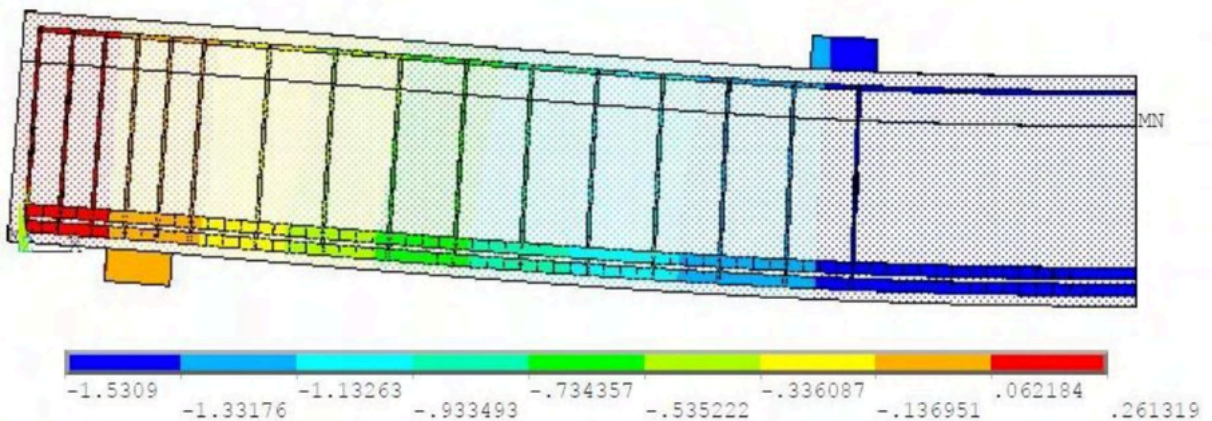


Figure 19 – deformed at the instant in break gives beam ET2.

Source: elaborate by the author.

The figure 20 bring at tensions imposed in the armor longitudinal gives beam ET2 at the instant of the break. The stress on the beam failure stresses was separated. The bar reaches 36 kN/cm², value very close to the flow.

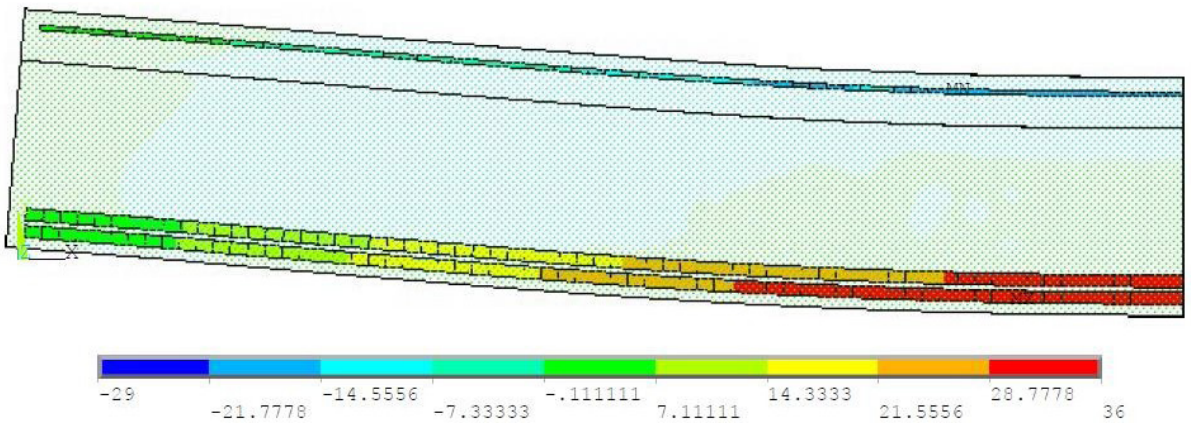


Figure 20 - tension in armor longitudinal gives beam ET2.

Source: elaborate by the author.

At figure 21, they are represented at tensions active only us stirrups gives beam ET2, at the time of rupture. It was found that the stirrups reached a yield strength of 32 kN/cm², which proves that there was break by shear.

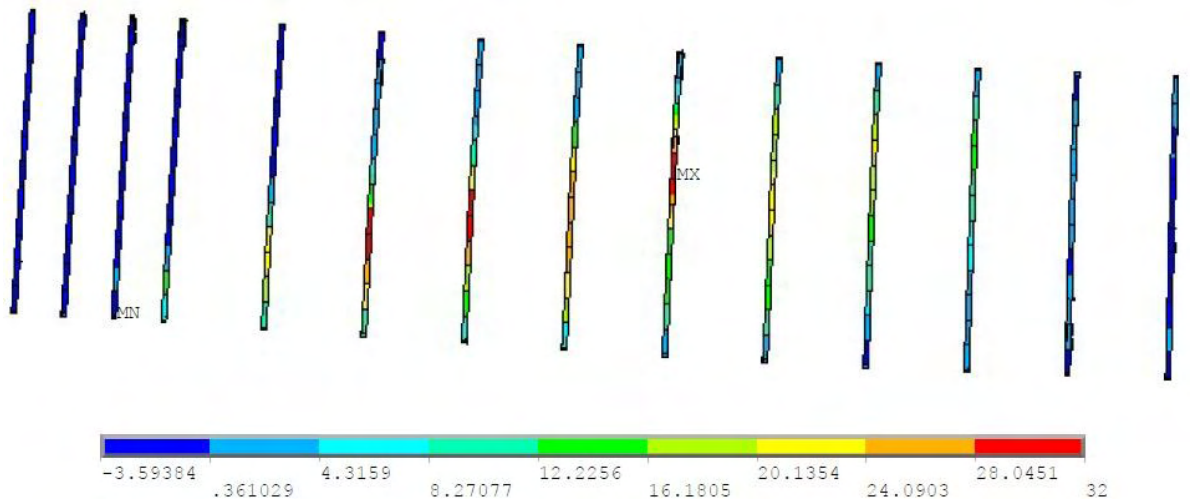


Figure 21 – tension in armor transversal gives beam ET2.

Source: elaborate by the author.

At figure 22, there has been O diagram in opening in cracks for The beam ET2. Staying so the shear failure in the central sections of the beam shown is evident in the graph, but the cracks in bending also present.

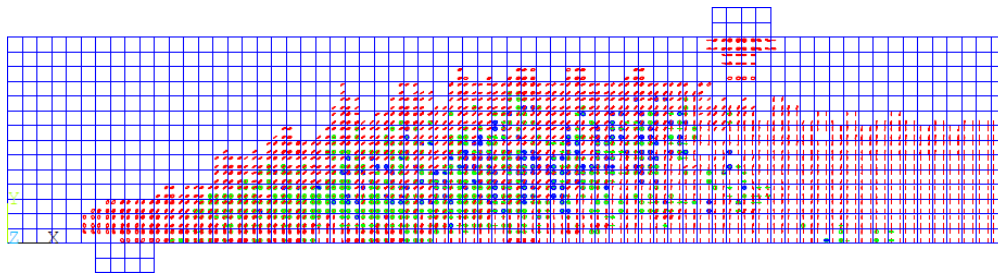
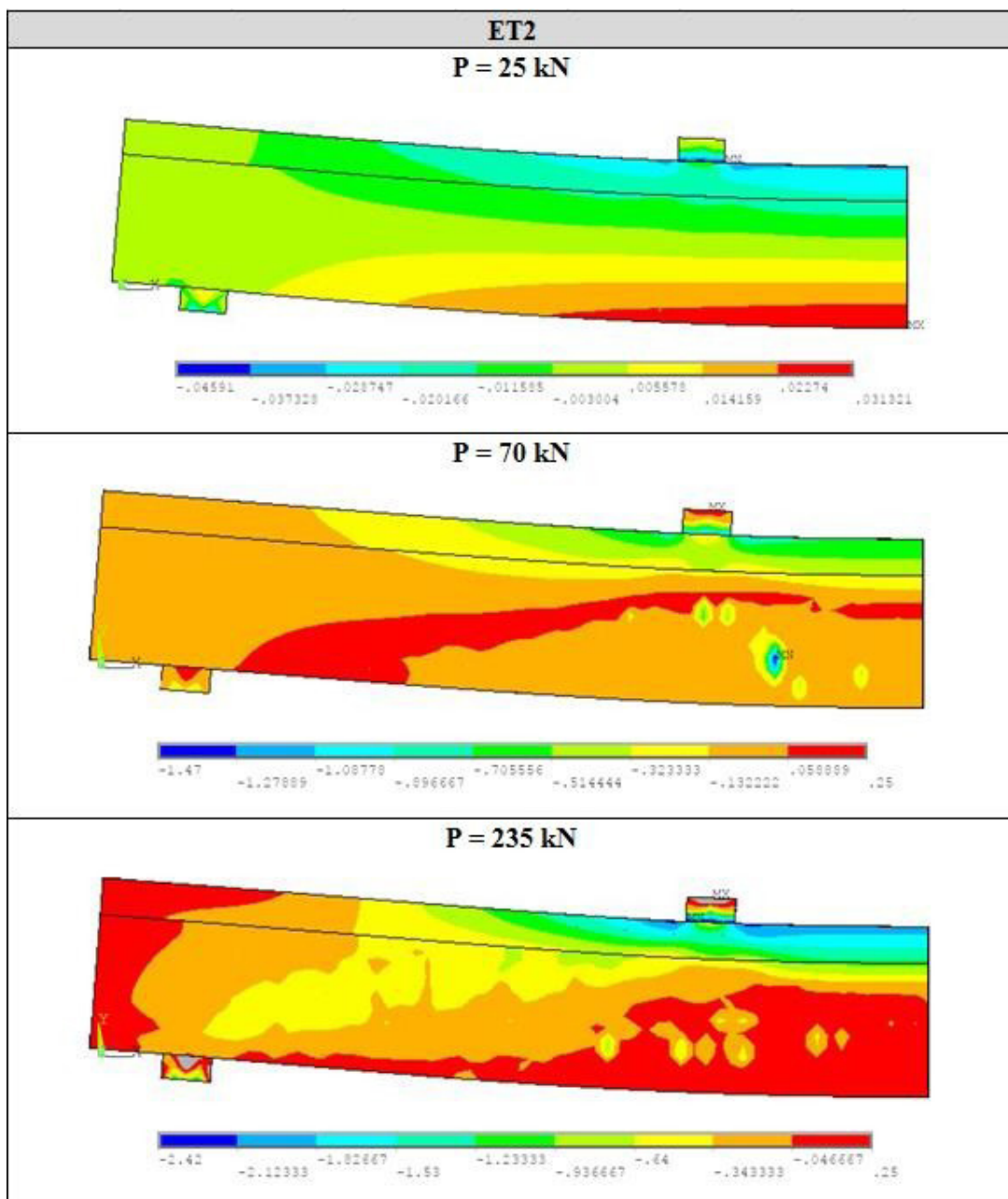


Figure 22 – diagrams in cracking gives beam ET2

Source: elaborate by the author.



Painting 6 – Evolution gives voltage σ_x at the concrete for The beam ET2 (units in kN/cm^2)
(source: elaborate by the author)

At the painting 6, were presented you Data in voltage from elements in concrete gives beam ET2. In this table, three loading stages were analyzed, in order to highlight the evolution of the tensions at the concrete.

In figure 23 it is possible to observe the load-displacement curves of the ET3 beam and the evolution of arrows at referred beam to far away From increments in displacement. Staying so exposed and what model adopted for modeling of beams if approaches from essay experimental proving the effectiveness thereof.

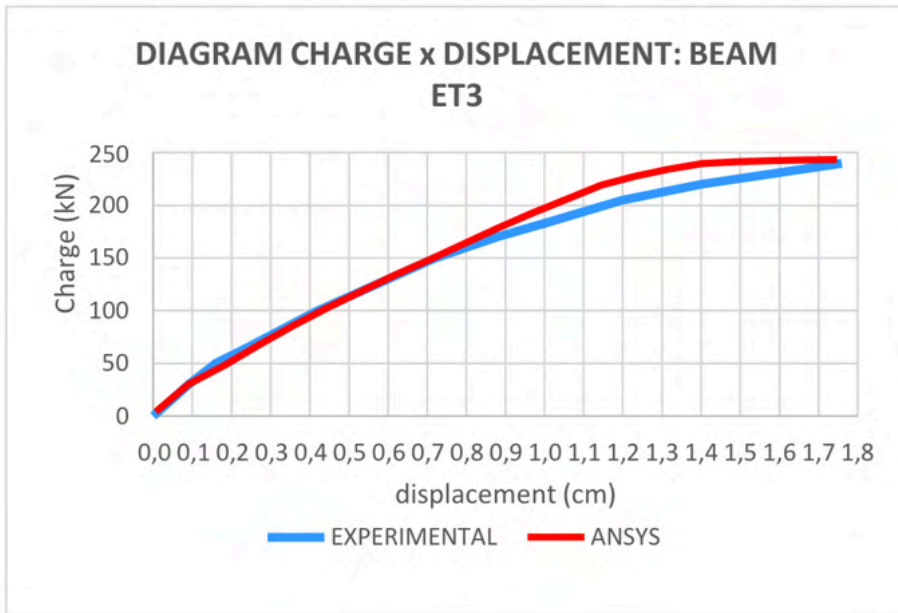


Figure 23 – Comparison gives arrow of beams experimental with computational model
(source: elaborate by the author)

At figure 24, It is presented The deformation maximum hit at beam ET3 at the instant Where structure rupture was identified. This deformed represents the arrow hit at the moment of your collapse. The breaking load was 244 kN, with a deformation of 1.74 cm at the center of the vain.

The figure 25 bring at tensions imposed in the armor longitudinal gives beam ET3 at the instant gives break. He was separate O effort in the tensions in break gives beam. THE bar he arrives The 29.3 kN/cm², value close to the drain.

At figure 26, they are represented at tensions active only us stirrups gives beam ET3, at the time of rupture. It was found that the stirrups reached a yield strength of 32 kN/cm², which proves that there was break by shear.

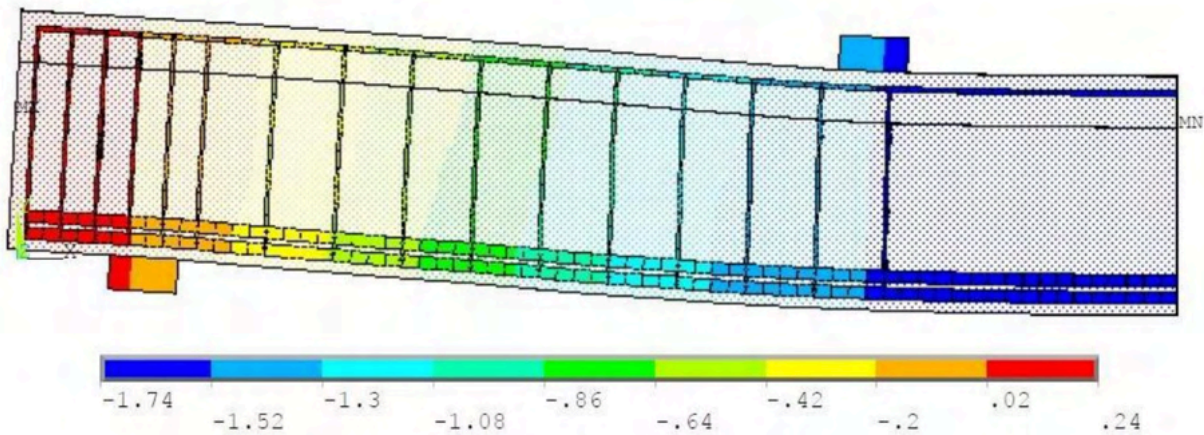


Figure 24 – deformed at the instant in break gives beam ET3.
 (source: elaborate by the author)

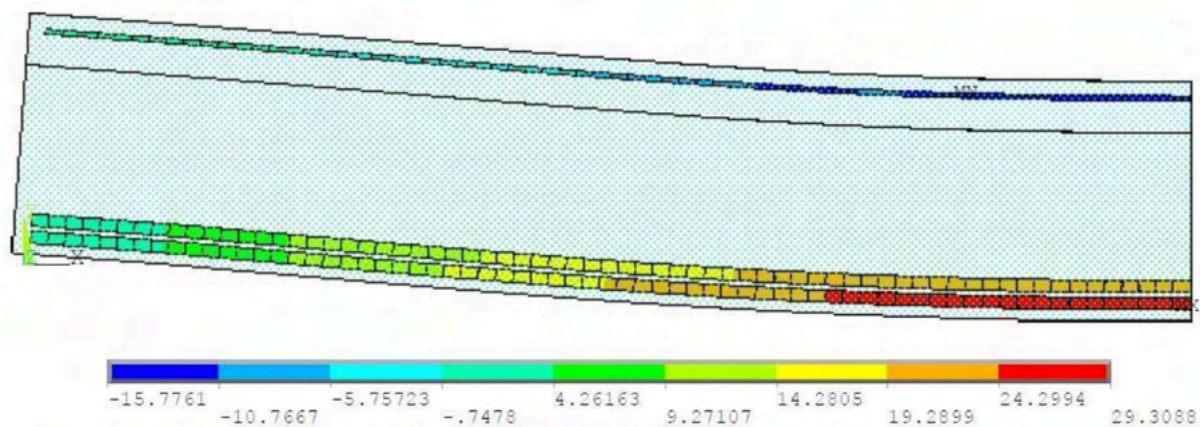


Figure 25 – tension in armor longitudinal gives beam ET3.
 (source: elaborate by the author)

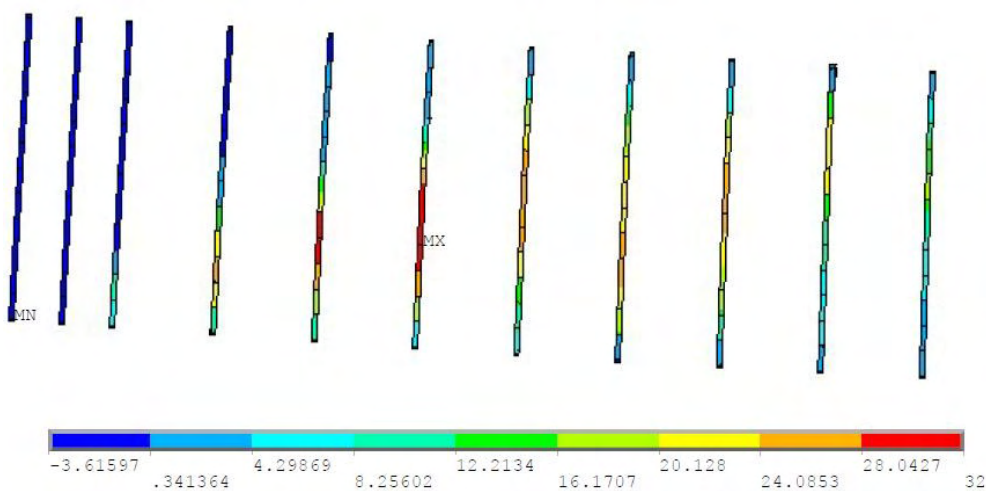


Figure 26 – tension in armor transversal gives beam ET3.
 (source: elaborate by the author)

At figure 27, there has been O diagram in opening in cracks for The beam ET3. Staying so the shear failure in the central sections of the beam shown is evident in the graph, but the cracks in flexion also present, but of form milder what at ET2.

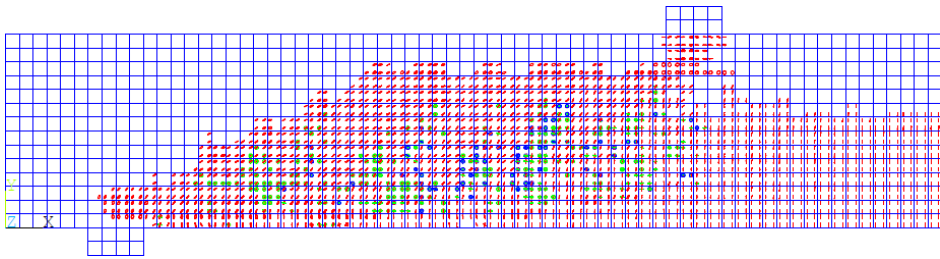


Figure 27 - diagrams in cracking gives beam ET3.

(source: elaborate by the author)

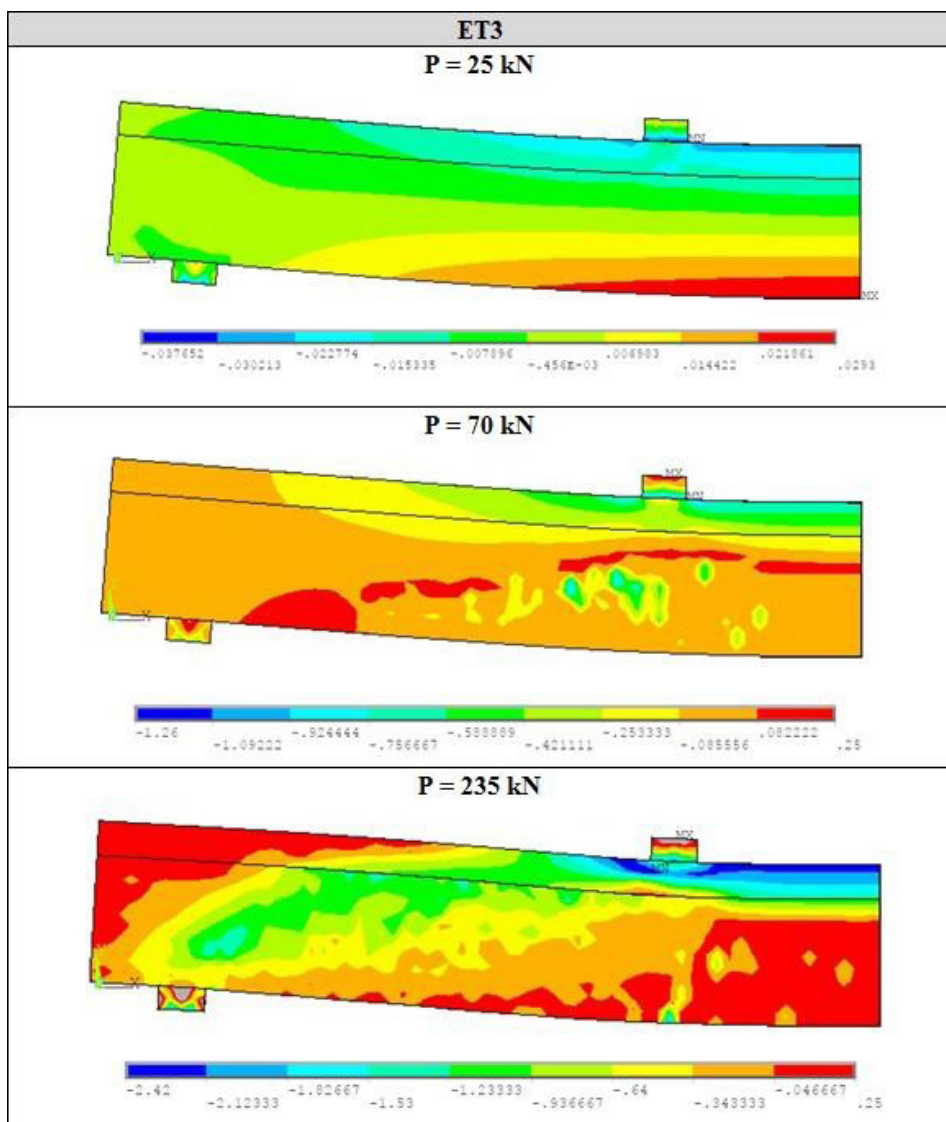


Table 7 – Evolution gives voltage σ_x at the concrete for The beam ET3 (units in kN/cm^2).

(source: elaborate by the author)

At the painting 7, were presented you Data in voltage from elements in concrete gives beam ET2. In this table, three loading stages were analyzed, in order to highlight the evolution of the tensions at the concrete. Analyzing you results presented, It is possible observe O displacement gives line neutral to from the increment of charge at structure.

B.2 ET4

In figure 28 it is possible to observe the load-displacement curves of the ET4 beam and the evolution of arrows at referred beam to far away From increments in displacement. Staying so exposed what model adopted for modeling of beams if approaches from essay experimental proving the effectiveness thereof.

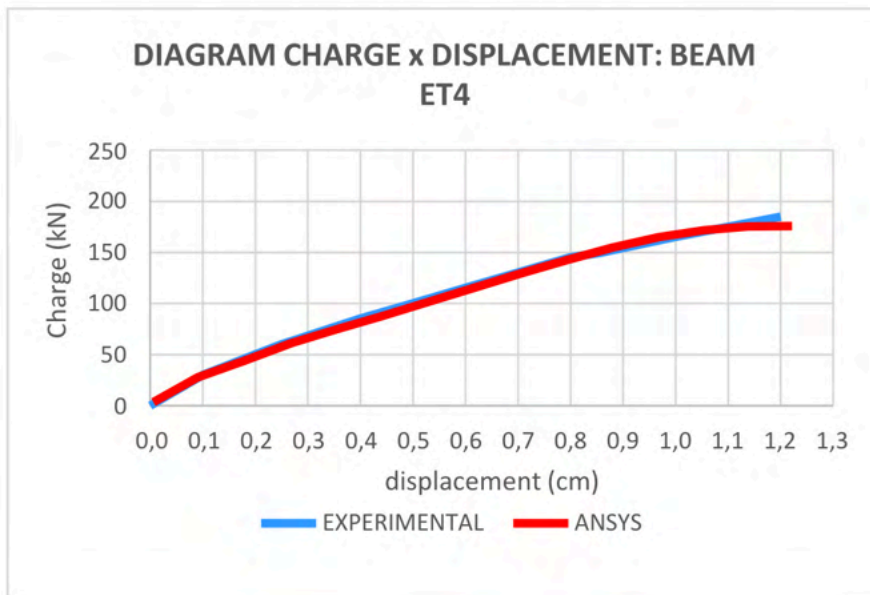
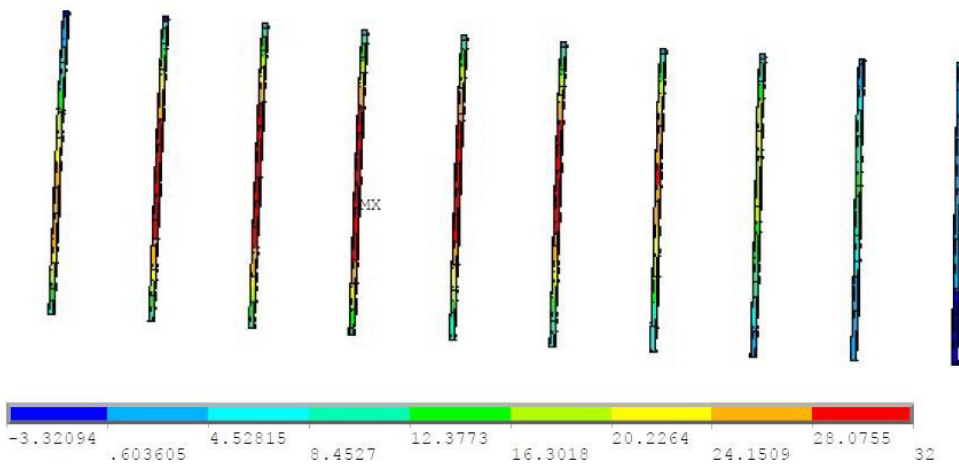
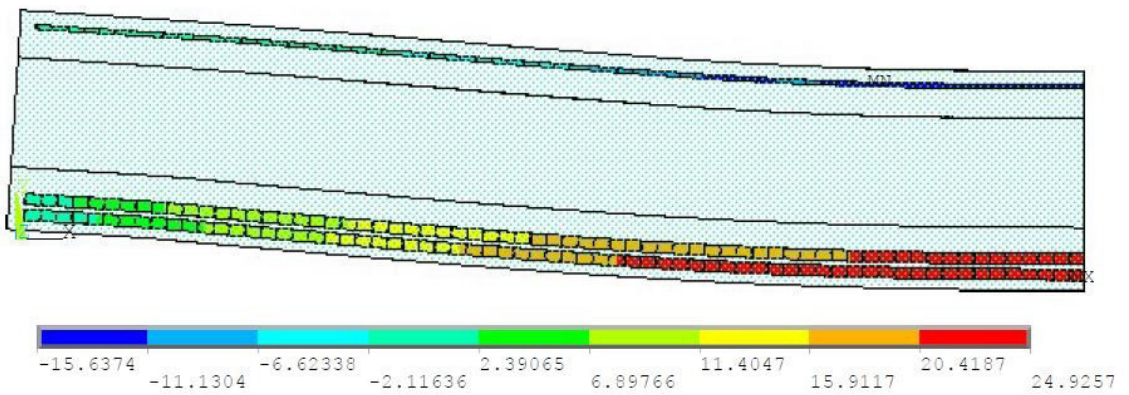
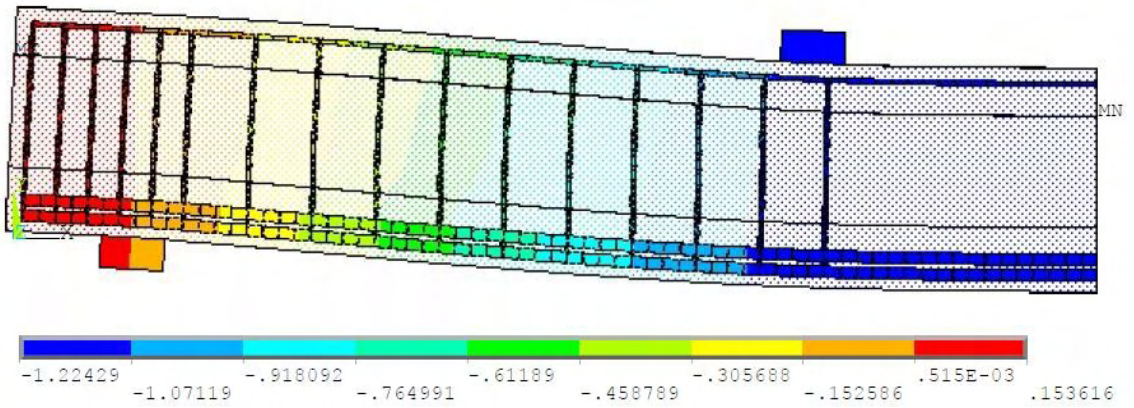


Figure 28 – Comparison of arrow of beams experimental with computational model
(source: elaborate by the author)

At figure 29, It is presented The deformation maximum hit at beam ET4 at the instant Where structure rupture was identified. This deformed represents the arrow hit at the moment of your collapse. The breaking load was 175 kN, with a deformation of 1.22 cm at the center of the vain.

The figure 30 bring at tensions imposed in the armor longitudinal gives beam ET4 at the instant gives break.

At figure 31, they are represented at tensions active only us stirrups gives beam ET4, at the time of rupture. It was found that the stirrups reached a yield strength of 32 kN/cm², which proves that there was break by shear.



Staying so evident in the graph the shear failure in the central and upper sections of the beam shown.

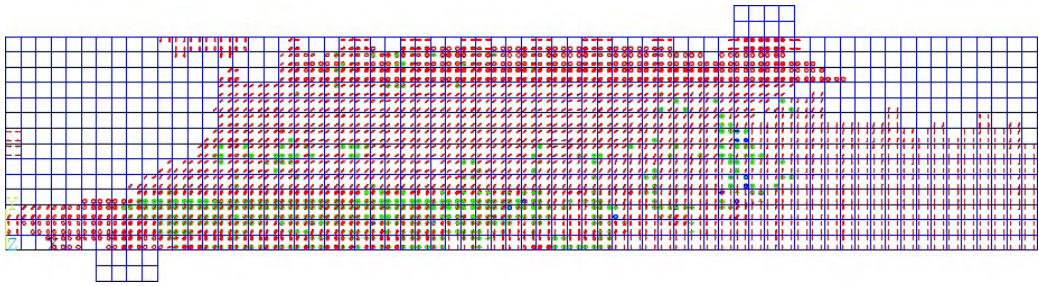
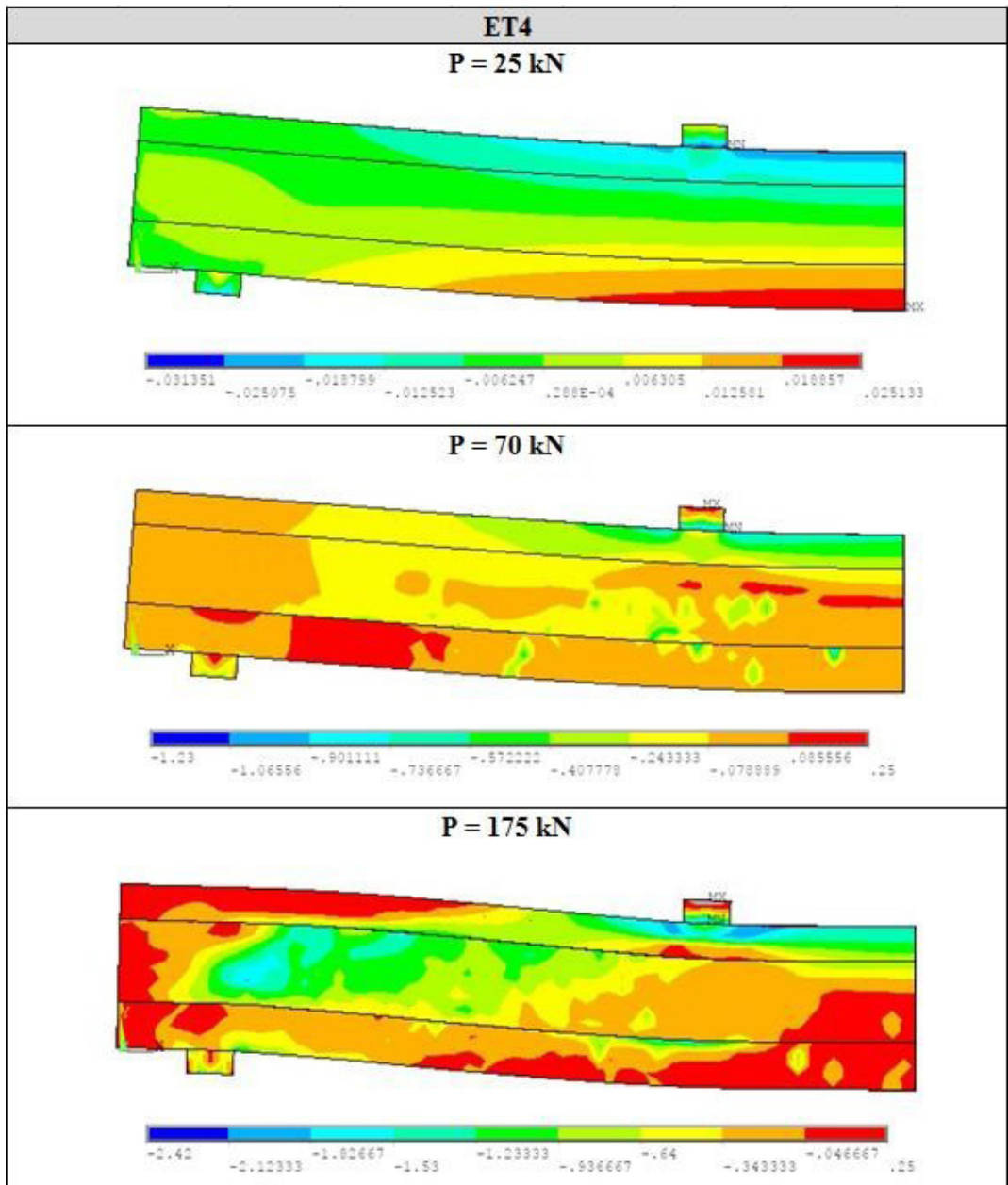


Figure 32 - diagrams in cracking gives beam ET4.

(source: elaborate by the author)



Painting 8 – Evolution gives voltage σ_x at the concrete for The beam ET4 (units in kN/cm^2)

(source: elaborate by the author)

At the painting 8, were presented you Data in voltage From elements in concrete gives beam ET4. In this table, three loading stages were analyzed, in order to highlight the evolution of the tensions at the concrete.