TYPES OF REBAR DISCRETIZATION AND THEIR CHARACTERISTICS IN COMPUTATIONAL SIMULATIONS OF CONCRETE STRUCTURES THROUGH FINITE ELEMENT METHOD

Rodrigo Borges Primieri

Alexandre Rodrigues Pacheco

rbp_95@hotmail.com

apacheco@ufrgs.br

Federal University of Rio Grande do Sul

99 Osvaldo Aranha Ave. 3rd floor, 90040-020, Porto Alegre, RS, Brazil

Paula Manica Lazzari

paula.lazzari@ufsc.br

Federal University of Santa Catarina

2700 Dr. João Colin St, 89218-035, Joinville, SC, Brazil

Abstract. *Computational modeling to simulate numerically structures is a methodology that has been increasingly important for the understanding of structures behavior, mainly when involving reinforced concrete, a material that presents a variety of complex phenomena. Even considering a quite noticeable technological advancement in the last decades on digital resources, which has been considerably increasing the possibilities, the designer's knowledge will never cease to be of fundamental importance. Therefore, in this work, focus is given to the choice of the rebar model type when applying the Finite Element Method (FEM). Among the typical choices, they could be classified as Distributed or Smeared; Discrete; and of the Element-Embedded type. In this way, a description of each model type is given, identifying main characteristics, advantages, and disadvantages, illustrating with experimental tests available in the literature, involving different structural elements in reinforced concrete. Comparisons between experimental data and computational analyses carried out with ANSYS are presented, showing the different aspects investigated. Additionally, statistics on the occurrence and typical use conditions of each model type when applied in reinforced concrete problems are also presented.*

Keywords: *Finite Element Method, ANSYS, Reinforced concrete, Rebar discretization.*

1 INTRODUCTION

Many works on modeling structures in reinforced concrete using the Finite Element Method have been done. However, few articles mention the selection criteria adopted for the computational representation of reinforcement behavior.

In the smeared representation, the characteristics of the material of the reinforcement are continuously scattered in the volume of concrete, or in part of it.

In the discrete representation, the reinforcements are defined as bar elements, being mandatory on the edges of bi or three-dimensional concrete elements.

In the embedded representation, in turn, rebar elements are one-dimensional bars that do not necessarily form part of larger element edges, and do not even share nodes with them.

Of course, each representation has its advantages and disadvantages, whose quantification and qualification are discussed here.

It will be considered: ease of use of each of the representations; quality of the result obtained and its proximity to results obtained from models tested in the laboratory; number of finite elements required; time spent by the computer for processing and solving the problem; ease of visualization of results in post-processing.

To verify these advantages and disadvantages, four beams tested in the laboratory by Leonhardt and Walther (1962), named ET1, ET2, ET3 and ET4, were modeled in the commercial ANSYS® APDL software.

For the case of the discrete reinforcement, the values obtained from Kunzler (2013) were used. In the case of the embedded reinforcement, the results of Lazzari (2015) are used, which creates a new constitutional model for the concrete within the ANSYS, in order to allow the use of the incorporated reinforcement.

However, no data was found in literature for these four beams through the smeared model. Thus, it was necessary, in this work, to perform this modeling so that the comparison between the three models could be carried out.

2 MODEL CHARACTERIZATION

In this chapter, a description of each model type is given, identifying main characteristics, advantages and disadvantages.

2.1 Smeared model

According to Gamino (2007), the smeared representation has the advantage of introducing constant reinforcement rates, according to structure orientation axes, which implies productivity in the launching of concrete reinforcement in general, provided they have rebar equally spaced.

For the case in which flexural structures are modeled, as in the case of beams, it is necessary to choose the region in which concrete and steel properties are to be mixed, in due proportion of areas. However, in larger structures with a more homogeneous distribution of reinforcement, such as slabs and shells, this care is not necessary, and the smeared model can be applied to all (or almost all) concrete elements. A preview of how this model would look is shown in figure 1.

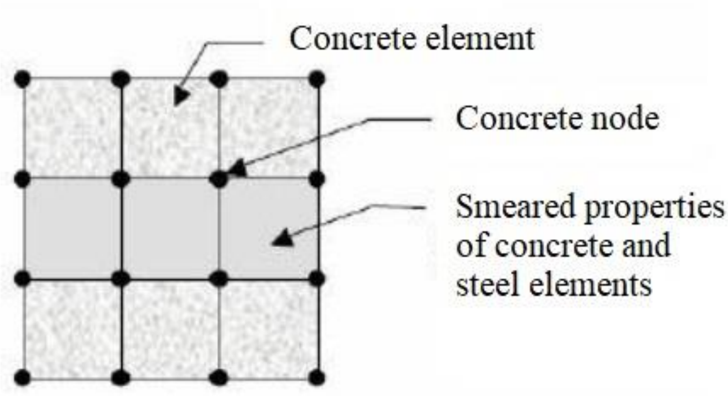

Figure 1. Smeared model (Kunzler, 2013)

This representation is easy to use, since it requires only one type of finite element, which serves to model the concrete and at the same time, the steel.

As for the number of finite elements needed to be used, it can be said that this model does not require a very refined mesh to represent well the displacements of the structure. However, in order to obtain a good visualization of the cracks in the concrete, it is recomended to use a slightly more refined mesh. It is interesting to note that the more refined the mesh, the better the results, but the calculation time increases considerably.

One of the worst characteristics of this rebar representation model is that it is not possible in the post-processing of the results to know directly how much each material (concrete and steel) absorbs the tensions generated in the structure.

2.2 Discrete model

This representation can be considered the most intuitive of all, since the rebars are modeled by lines inside the concrete volume, exactly in the position where the bars would be in laboratory models.

However, this model requires that the nodes of the bar elements that represent the reinforcement belong to the nodes of the elements that represent the concrete. This, in practice, means that rebars can only pass through where there are already edges of three-dimensional elements.

Thus, when inserting the reinforcement in the model, the user must take care to divide the volume representing the structure by planes that intercept in the position in which the reinforcement will be. Proceeding this way, the user guarantees that, when the mesh is generated, there will be edges of concrete elements in the position to be inserted the reinforcement.

However, in certain structures, the rebars may be very close together, which may require very small elements, increasing the calculation time by the computer.

Figure 2 illustrates this type of representation, making evident the interdependence between the position of the reinforcing bar and the mesh of the concrete elements.

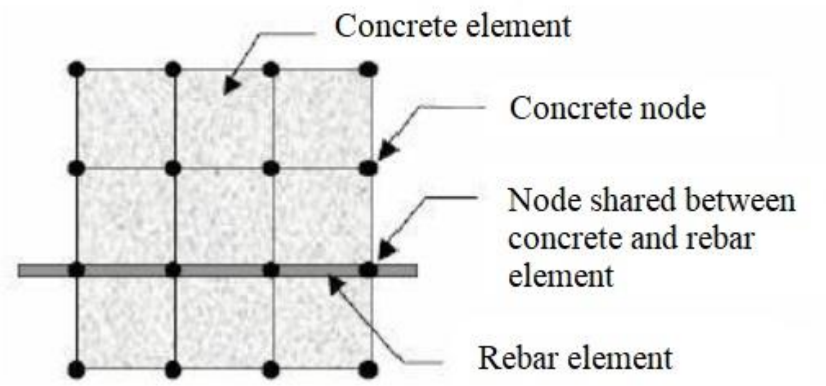

Figure 2. Discrete model (Kunzler, 2013)

An advantage of the discrete versus smeared model is that the stress generated on the rebars can be viewed directly in the post-processing. Thus, it is easy to check if there is any bar with very large tension, which would lead to its rupture.

2.3 Embedded model

Unlike the discrete model, the representation of the reinforcing bars in the incorporated model can be done without concern for the mesh, since it is assumed that the displacement of a generic point of the bar is identical to the respective displacement in the concrete element, located in the same position.

Thus, this representation is ideal for situations where the rebar arrangement is curved, or too complex, since it does not depend on the arrangement of the finite element mesh. This fact is evidenced in Figure 3.

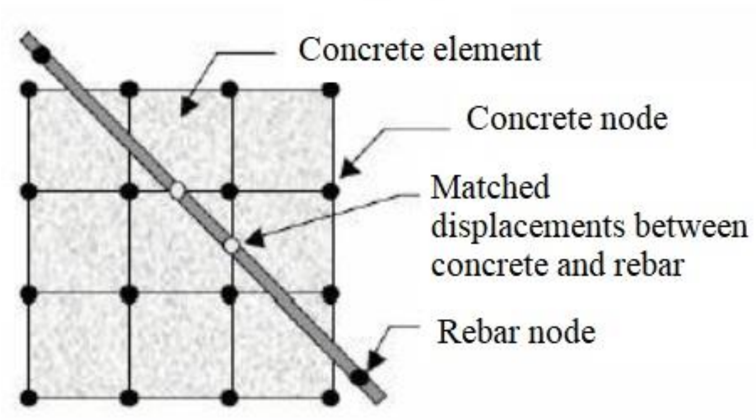

Figure 3. Embedded model (Kunzler, 2013)

In addition, the independence between reinforcement and concrete meshes allows the elements to have considerably larger sizes, limited only by the quality of the expected result.

Thus, the number of elements used and the resolution time by the computer can be considerably reduced.

Although this is the simplest representation of being launched geometrically, some large commercial software, such as ANSYS®, do not have elements capable of modeling the concrete with this rebar representation. In this way, to represent it, it is necessary for the user to create a new material model for the concrete, through the User Programmable Feature - UPF functionality. Unfortunately, creating a new material model for concrete with all the features requires a lot of time and dedication and demands a great deal of knowledge from the user of concrete failure theories. However, as the material is developed by the user, it further increases the likelihood of modeling errors.

However, even with these disadvantages, this model is the most used in Brazil in master's and doctoral research, possibly because it allows the use of a smaller number of finite elements, leading to faster solutions.

3 NUMERIC MODELS

In order to clearly discern the differences of each method, especially with regard to the precision and quality of the results, the data obtained experimentally in the work of Leonhardt and Walther (1962) were compared with the values obtained by numerical models through ANSYS® APDL software, for each of the reinforcement models.

The results presented here for the discrete reinforcement model were obtained from Kunzler (2013). For the embedded model, the data obtained from Lazzari (2015) were used.

On the other hand, for the distributed model it was necessary to do all the computational modeling in this work, due to the lack of results in the literature.

3.1 Description of the studied beams

In this item, we present the set of four bi-supported beams, called ET1, ET2, ET3 and ET4, experimentally tested by Leonhardt and Walther (1962). These beams undergo the action of two concentrated loads, whose positions can be seen in figure 5. All the beams have a span of 3.0 m and a height of 35 cm. However, these beams are differentiated by their cross sections, as shown in figure 4.

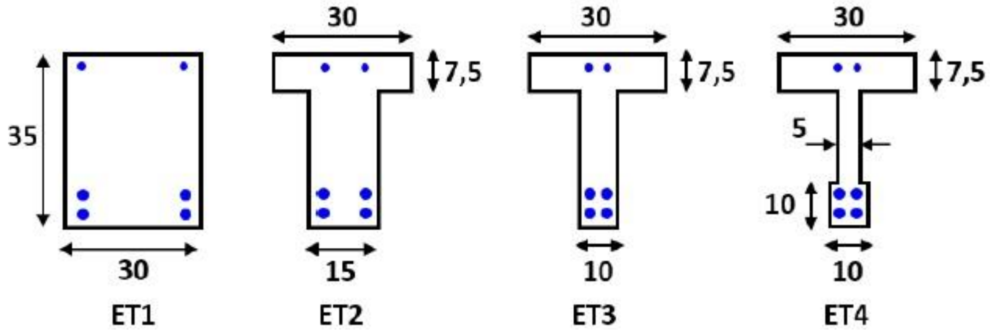

Figure 4. Transversal sections of the beams (Lazzari, 2015)

All the beams in this group are made of concrete of the same average compressive strength, equal to 2.42 kN / cm2.

The rebars are also equal between the beams. The lower longitudinal reinforcement consists of four 20 mm diameter bars (f_y = 42.8 kN / cm^2), two of which are located 3 cm from the lower edge and the other two from 6 cm from the same edge. The upper longitudinal reinforcement, in turn, consists of two 8 mm diameter bars (f_y = 46.5 kN / cm^2), located 3 cm from the upper edge. All longitudinal reinforcements are made of cold-worked steel (former class B), with ES modulus of 19500 kN / cm^2.

The vertical stirrups are formed by bars of 6 mm diameter (f_y = 32 kN / cm^2), in steel with natural hardness (former class A) and modulus of elasticity of 21000 kN / cm^2. They are spaced apart as shown in figure 5.

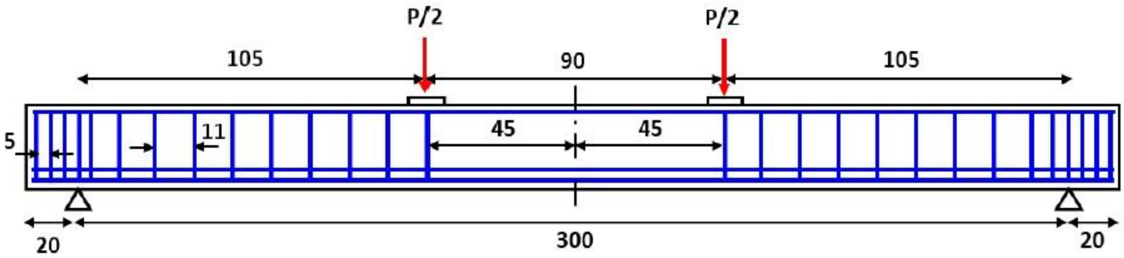

Figure 5. Longitudinal section, common to all four beams (Lazzari, 2015)

3.2 Smeared model

It was only necessary to use the three-dimensional element SOLID 65 for the representation of the concrete and the reinforcement, to model the beams by the distributed representation.

The beams were divided into three regions, along their height, as can be seen in Figure 6. Each of these regions was assigned different real constants, according to the present rebar. Thus, in the lower and upper regions, reinforcement with Class B steel in the longitudinal direction and reinforcement with Class A steel in the vertical direction was proposed, in order to represent the longitudinal reinforcement and stirrups, respectively. In the intermediate region, only the presence of reinforcement was predicted with the steel material of class A in the vertical direction, modeling the resistance of the stirrups.

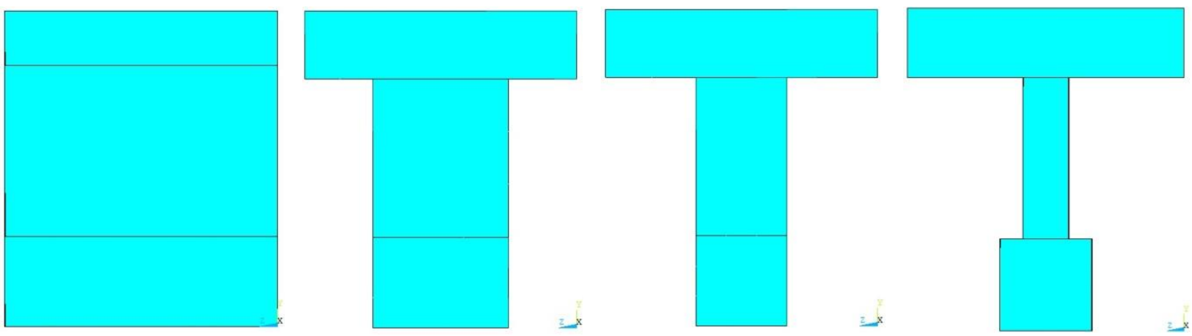

Figure 6. Cross sections divided into three regions, according to their respective reinforcement

The value adopted to obtain the modulus of elasticity was obtained through equation (3.2) of the Model Code CEB-FIP1990 (1993)

$$E_{ci} = 2{,}15 \times 10^{3} \left(f_{cm} \right)^{1/3} \tag{1}$$

where the units are given in kN / cm^2.

The concrete model was defined by the equations suggested by Macgregor (1992). For the linear region, the relation

$$E_c = \frac{f}{\varepsilon} \,. \tag{2}$$

As for the non-linear region, relations were adopted

$$f = \frac{E_c \varepsilon}{1 + \left(\frac{\varepsilon}{\varepsilon_0} \right)^2} \tag{3}$$

$$\varepsilon_0 = \frac{2 f_c'}{E_c} \,. \tag{4}$$

where f is the stress corresponding to the deformation ε, and E_c is the modulus of elasticity of the concrete.

Using the equation 1, we obtain the value of the modulus of elasticity of the concrete, which corresponds to the initial linear region of the strain-strain curve. The Poisson coefficient adopted was 0.2. These values can be observed in Table 1.

Table 1. Modulus of elasticity and Poisson’s ratio for concrete.

Concrete	
Modulus of elasticity (kN/cm²)	2886.5
Poisson’s ratio	0.2

Using the equations 2 to 4, the multilinear strain-strain relationship shown in Table 2 was obtained. Note that the previously calculated modulus of elasticity is valid for tensions up to 30% of the tensile stress.

In order to avoid convergence problems of the finite element model, the slope of the strain-strain curve was not considered.

Table 2. Concrete stress-strain relationship.

Point	Deformation	Tension (kN/cm²)
1	0.000252	0.7260
2	0.00050	1.3254
3	0.00100	2.1292

4	0.00150	2.4051
5	0.00168	2.4200
6	0.00300	2.4200

In order to simulate the concrete material, the distributed model still requires complementary information for its execution. The values entered are shown in table 3.

Table 3. Coefficients for concrete.

Coefficients for concrete	
Open shear transfer coef.	0.3
Closed shear transfer coef.	1.0
Uniaxial cracking stress (kN/cm²)	0.3064
Uniaxial crushing stress (kN/cm²)	2.42

The values for the open and closed shear coefficients used in the model were 0.3 and 1.0, respectively, these values being the most used in the literature. For the calculation of the initial cracking tension, presented in table 3 above, the formula suggested by Wolanski (2004) was used, with the conversion of units to kN / cm².

$$f_r = 5{,}171 \times 10^{-3} \times \sqrt{1450.38 \times f_c'} \, . \quad (5)$$

For the reinforcement, a bi-linear model was defined to simulate the behavior of the material. The data entered are shown in table 4.

Table 4. Coefficients used to model each of the reinforcements.

Property	Unity	Bottom reinforcement	Top reinforcement	Stirrups
Modulus of elasticity (E)	(kN/cm²)	19500	19500	21000
Poisson’s ratio (υ)	-	0.3	0.3	0.3
Yield point (f_y)	(kN/cm²)	36.38	39.52	32
Tangent modulus (E_t)	(kN/cm²)	789.2	874.8	210

For beam ET1 and ET2, the element size was defined with a maximum of 10 cm of edge (figure 7). For the other beams, this size was reduced to 5 cm, so that the mesh was composed of finite elements with cubic shape or the closest, since for these two beams the web has a smaller thickness (figure 8).

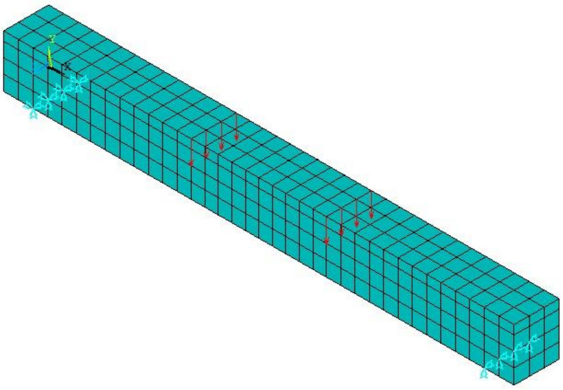

Figure 7. Finite element representation of the ET1 and ET2 beams, with elements of 10 cm

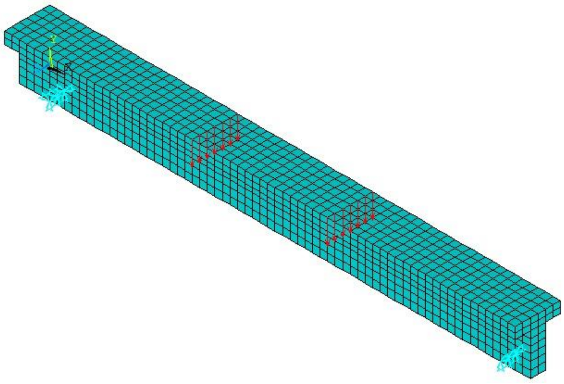

Figure 8. Finite element representation of the ET3 and ET4 beams, with 5 cm elements

The maximum load applied P was divided into small loading steps of 1kN, in order to observe the deformations and cracks at several different instants, besides ensuring convergence in the non-linear region of the concrete.

3.3 Discrete model

For the discrete representation, the data obtained by Kunzler (2013) were used. In this model, a finite element mesh of type SOLID 65, with an edge 2,5 cm was used, as can be seen in figure 9. This mesh size represents elements from 8 to 16 times smaller (in volume) than those used in the distributed model.

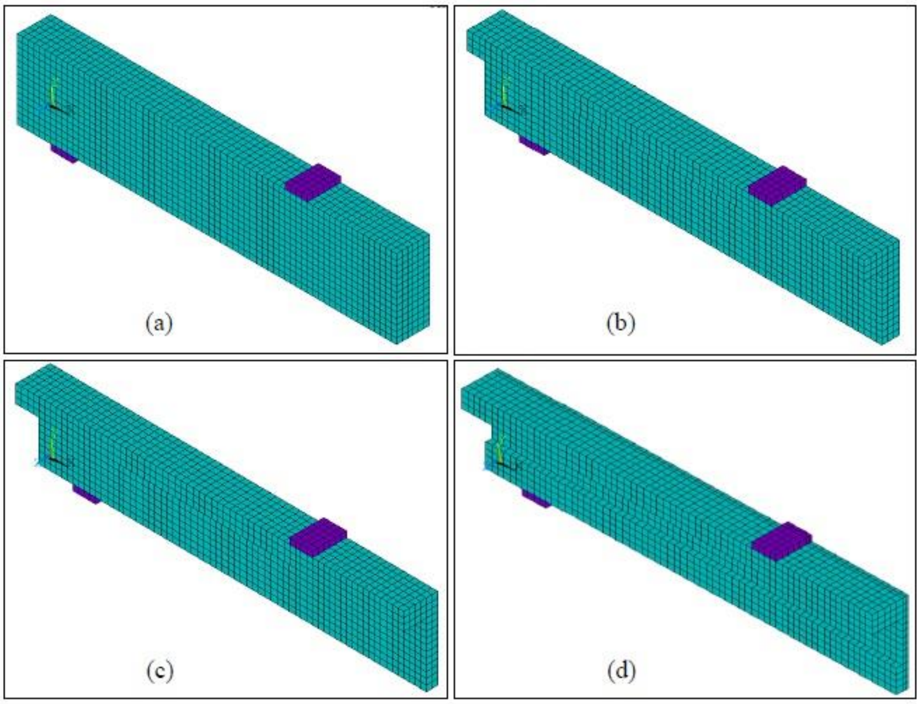

Figure 9. Finite element representation of the ET3 and ET4 beams, with 5 cm elements (Kunzler, 2013)

In order to ease the computational impact caused by such a refined mesh, only one quarter of each beam was simulated, considering the two planes of symmetry (longitudinal and central transverse).

3.4 Embedded model

For the embodied representation, the data obtained by LazzariI (2015) are analyzed. In this work, it was possible to use considerably large elements. Only half of each beam was modeled (using the symmetry of the central transverse plane), and the analysis was considered only in the XY plane. Thus, it took only 20 finite elements to model the concrete, as can be seen in figure 10.

However, as the model is incorporated, it was necessary to make all the development of the concrete material for this analysis.

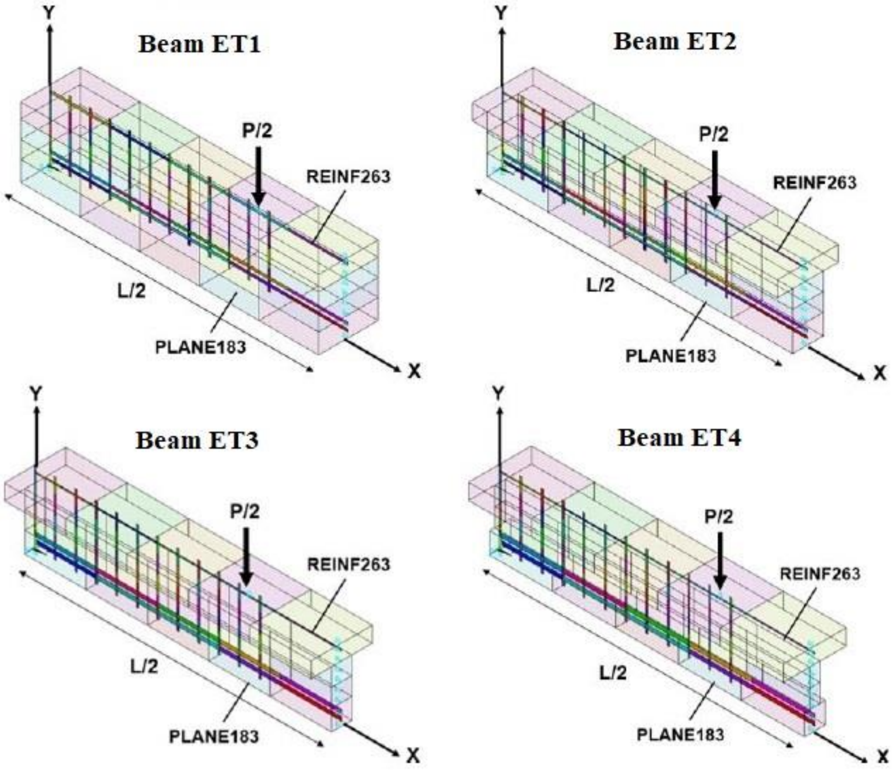

Figure 10. Representation in finite elements, considering the embedded model (Lazzari, 2015)

3.5 Comparison between results obtained

One of the great differences between the models of armor representation is the possibility of representing the results. Thus, before starting work on finite elements, it is necessary to keep in mind what results will be needed at the end, to choose the model of representation of the armature that best meets the needs of the problem in question.

In this way, in order to show the possible results that each model is able to display, the results load-displacement, cracking in concrete and stresses in the reinforcement will be compared.

Load-displacement curves. To verify the quality of the results, the load-displacement curves obtained by the computational models and the laboratory result were compared, as can be seen in figures 11 to 14. The displacements considered were measured at the lower nodes of the central cross sections of the beams.

It can be observed that there is good match of results. In the initial stretches, where the loads have a lower intensity, all the curves are presented as a straight stretch, indicating that the material is behaving elastically-linearly.

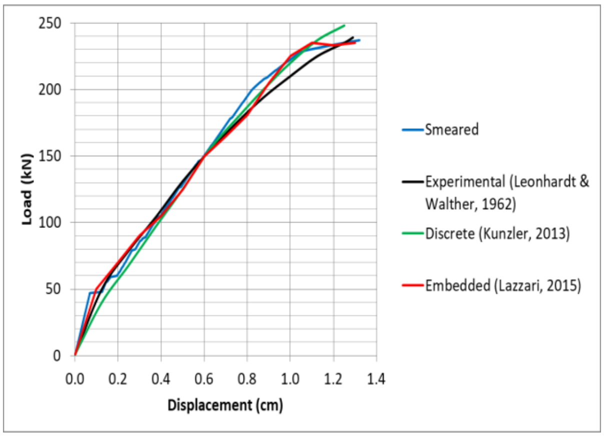

Figure 11. Load-displacement curves for ET1

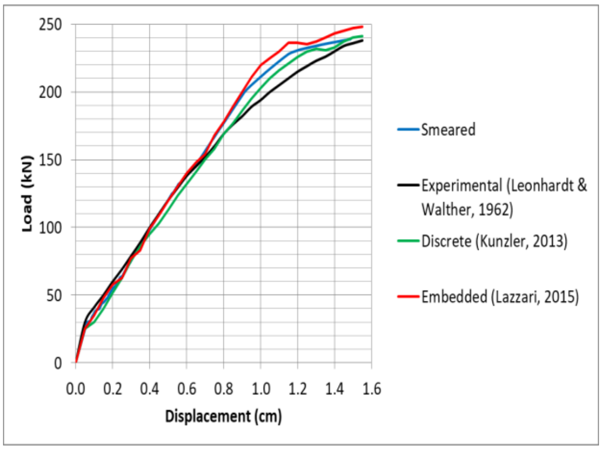

Figure 12. Load-displacement curves for ET2

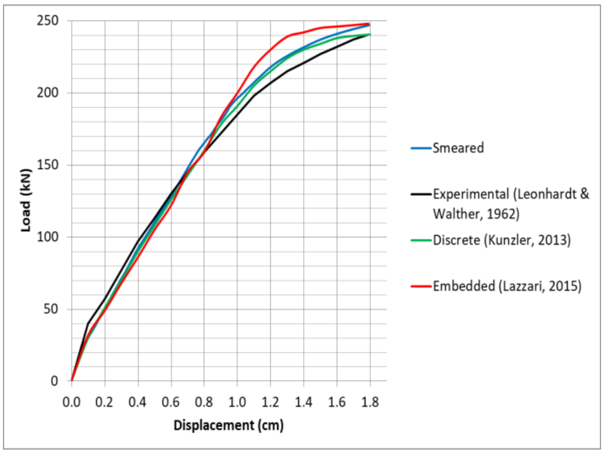

Figure 13. Load-displacement curves for ET3

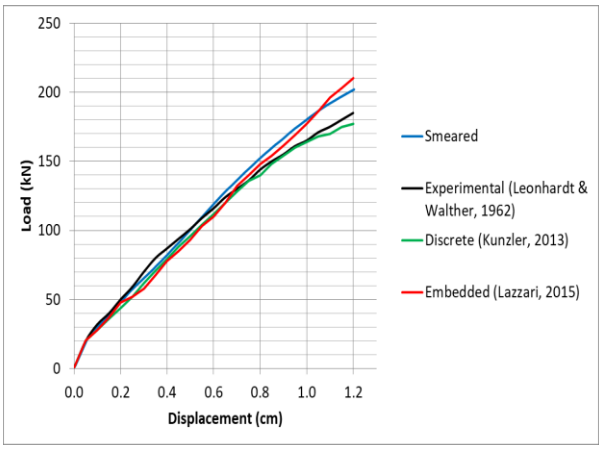

Figure 14. Load-displacement curves for ET4

As the load increases, the slopes of the load-displacement curves decrease. It is a fact that reflects the non-linear behavior of the concrete, whose stress-strain relationship decreases with increasing stresses. In addition, the increased load induces the appearance of cracks, which further reduce the resistive capacity of the parts as a whole.

The curves that present more linear behavior throughout the graph characterize the shear rupture. The curves that, close to the rupture, tend to be horizontal, characterize the stage of drainage of the reinforcement and plastification of the compressed concrete, breaking through the effect of the flexion.

Thus, it can be stated that the ET1 beam breaks by bending. However, the ET2 and ET3 beams break by shear and bending. In turn, the ET4 beam breaks through shear.

As the three models of armature representation presented results very close to each other and to the one obtained experimentally, it can be concluded that the three are valid for the modeling of the beams of the study.

Cracking. The discrete and distributed models allow the cracking to occur in the concrete. The built-in model, however, does not display cracking of parts if this feature is not preprogrammed. The display of cracking in the embedded model is complex, since it only occurs if the material model developed by the user contains this functionality. This requires a great deal of user knowledge on concrete properties as well as solid programming knowledge.

In this way, the embedded model is not indicated, in principle, for a user who wishes to obtain results in cracking but does not want to develop a new material.

An example of cracking can be seen in Figure 15.

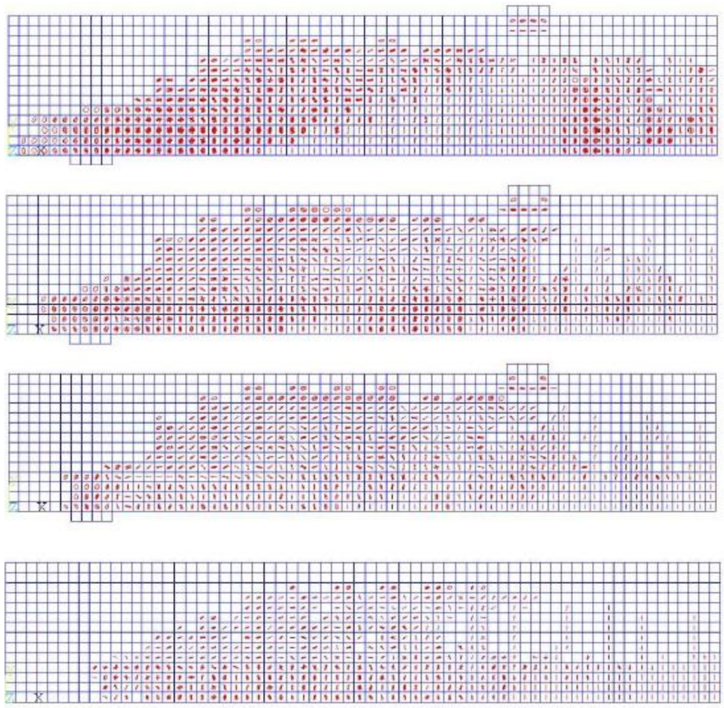

Figure 15. Cracking in ET1, ET2, ET3 e ET4 (Kunzler, 2013)

Tension in reinforcement. The smeared model does not allow the results to be shown in tensions in the armature, because the reinforcement is distributed in the concrete. Thus, if the user wishes to obtain the stress results in the steel, he must use the discrete model or the embedded model. An example of tension results in the reinforcement is shown in figure 16.

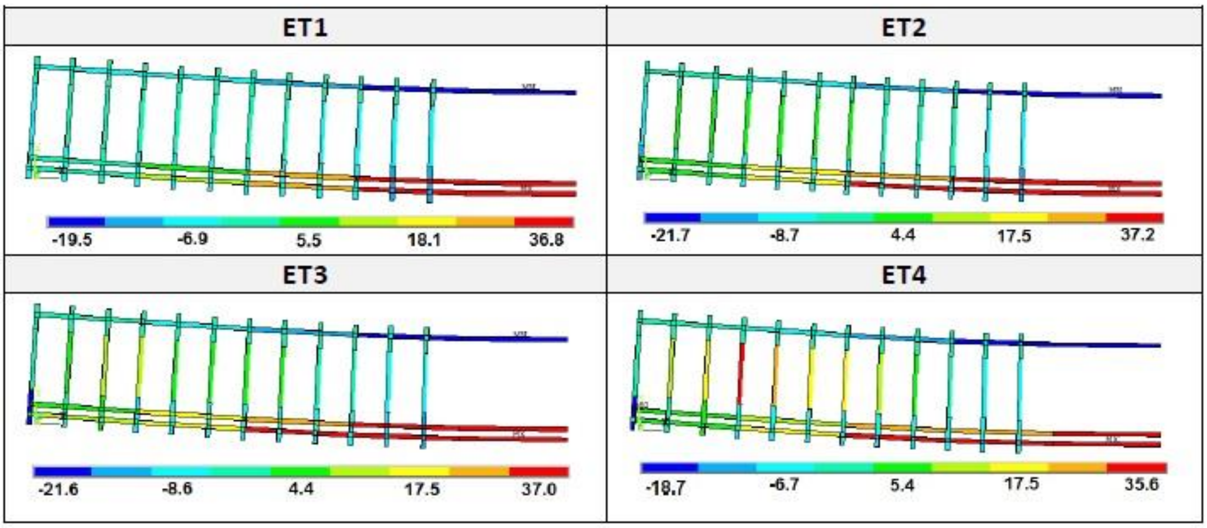

Figure 16. Tension at the reinforcement (Lazzari, 2014)

Considerations about results. As can be seen, with the three models of armature representation it is possible to obtain the load-displacement curve. However, for other results, the characteristics of each model must be considered, since the distributed model does not show results in tensions; The discrete model requires more refined mesh, and therefore takes longer to obtain the computational results; And the built-in model does not show cracking of parts if this feature is not previously programmed.

4 SURVEY OF THE OCCURRENCES OF EACH MODEL IN BRAZIL

After comparisons between the models of armor representation in reinforced concrete by finite elements have been carried out, it is also of great interest to know how much each of these models has been used in recent years.

For that, a survey was made, among all theses and dissertations published in Brazil, on the topic of finite element modeling of reinforced concrete structures, published between the years 2000 and 2016.

Among the publications found, we selected those that explicitly made clear which model of armor was used. Thus, after this selection, 45 theses and dissertations were included. Their distribution over time and according to the model used can be seen in figure 17.

An interesting fact to be observed is the presence of armor models that do not follow the pattern of the three existing classical models. In the graph of figure 17 they are represented as "others", due to the fact that there is no name in the literature for such representations.

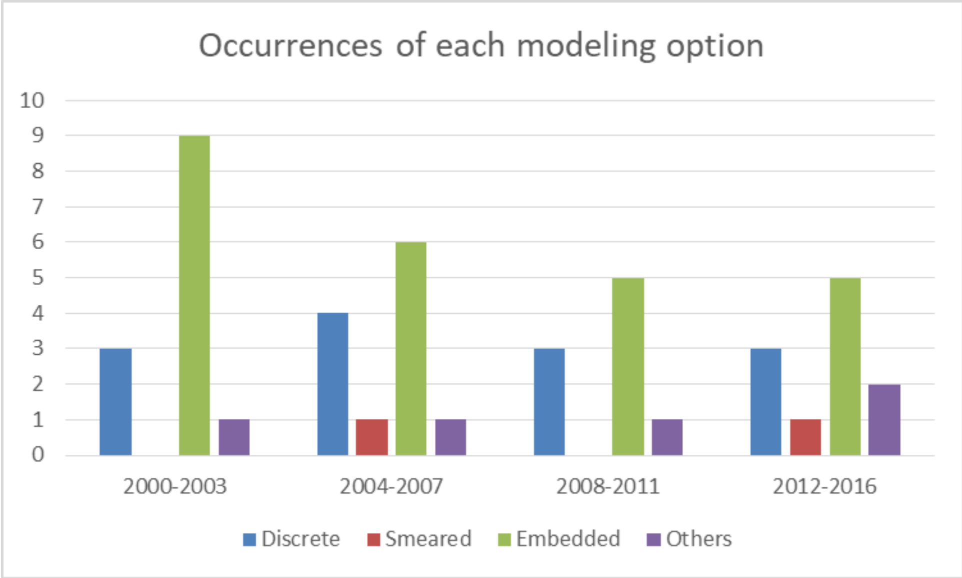

Figure 17. Occurrences of each modeling option, for the last four triennia

These models consider the rebars to be a steel cylinder within the concrete material. The finite elements of armor are, therefore, three-dimensional. This creates an advantage as it allows the user to consider the cross-resistance of the steel bar. However, such modeling causes incompatibilities between the mesh of steel elements, which have curved (generally circular) edges, and the mesh of concrete elements, which have straight edges. This condition makes the user have to be careful when choosing the element that will model the concrete, since it cannot be SOLID 65. There is also a great inconvenience that the concrete material should be modeled with linear behavior, unless that the user creates a new concrete model within the software.

Another fact to be observed is the expressive use of the incorporated reinforcement, which was present in 56% of the works considered. On the other hand, the distributed model was found in only 4%, as can be seen in Figure 18.

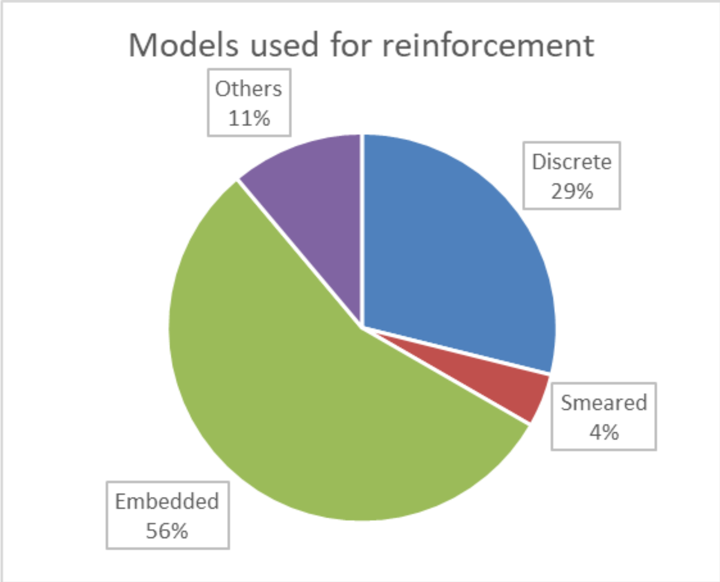

Figure 18. Occurrences of each modeling option

5 FINAL CONSIDERATIONS

The finite element method can be used through a wide variety of formulations and constitutive models for the materials adopted. The numerical models presented here meet the requirements proposed in this paper. With four beams, each of them being modeled by the three modes of armature representation, twelve representations were calculated and compared to the experimental results. It was verified that in all the simulations, there was great similarity of the results, even the beams presenting, among themselves, varied behavior and with different forms of rupture. Therefore, it is evident that the numerical model generates satisfactory results for a great variety of structures.

In general, the three numerical models presented very good data, and it can not be said that any of them are imprecise. Thus, to make the choice of one of these representations for the armature, one must pay attention to the characteristics of the structure to be modeled. It is up to the designer, therefore, to choose the model that best meets their needs, knowledge and computational availability.

REFERENCES

Comité Euro-International du Béton, 1993. *CEB-FIP Model Code 1990*. London, Thomas Telford.

Gamino, A. L., 2007. *Modelagem física e computacional de estruturas de concreto reforçadas com CFRP*. Tese de doutorado. Universidade de São Paulo, São Paulo.

Kunzler, P. S., 2013. *Análise paramétrica por elementos finitos de vigas de concreto armado e protendido pré-tracionadas com abertura na alma*. Dissertação de mestrado. Universidade Federal do Rio Grande do Sul, Porto Alegre.

Lazzari, B. M., 2015. *Análise por elementos finitos de peças de concreto armado e protendido sob estados planos de tensão*. Dissertação de mestrado. Universidade Federal do Rio Grande do Sul, Porto Alegre.

Lazzari, P. M., 2014. Análise estrutural não linear de vigas em concreto armado utilizando o ANSYS 14.5. *VII Congresso Brasileiro de Pontes e Estruturas*. Rio de Janeiro, ABPE.

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

Macgregor, J.G., 1992. *Reinforced Concrete Mechanics and Design*. Prentice-Hall, Inc., Englewood Cliffs, NJ.

Wolanski, A. J., 2004. *Flexural behavior of reinforced and prestressed concrete beams using finite element analysis*. Master thesis. Marquette University, Milwaukee.