Resilient Structures and Sustainable Construction
Edited by Pellicer, E., Adam, J. M., Yepes, V., Singh, A., and Yazdani, S.

ISBN: 978-0-9960437-4-8

USING THE ELEMENT-EMBEDDED REBAR MODEL IN ANSYS TO ANALYZE REINFORCED CONCRETE BEAMS

BRUNA MANICA LAZZARI[1], AMÉRICO CAMPOS FILHO[2],
PAULA MANICA LAZZARI[3], and ALEXANDRE RODRIGUES PACHECO[2]

[1]*Polytechnic School, Pontifical Catholic Univ. of Rio Grande do Sul, Porto Alegre, Brazil*
[2]*Department of Civil Engineering, Federal Univ. of Rio Grande do Sul, Porto Alegre, Brazil*
[3]*Mobility Engineering Center, Federal University of Santa Catarina, Joinville, Brazil*

ANSYS is software well acknowledged by professionals and academics, due to its large range of usage options, offering a number of finite element types, good possibilities of constitutive models and linear and nonlinear analysis of structures in general. For the concrete material, the software uses an elastoplastic model with the Willam-Warnke surface of rupture (1975). However, this model is only available for finite elements that do not offer the possibility of use of the element-embedded model for rebars, demanding a much larger amount of elements to discretize structures. This study is about the development of a computational model using the FEM via ANSYS platform for nonlinear analysis of reinforced concrete beams under plane stress states. The most significant advantage of this implementation is the possibility of using the element-embedded rebar model in ANSYS with its 2D quadratic element PLANE183 for discretization of the concrete together with element REINF263 for discretization of rebars and stirrups, making the solutions more efficient. For representation of the constitutive equations of the steel and the concrete, a proposed model was implemented with the help of the UPF customization tool of ANSYS, where new subroutines were attached to the main program. The numerical results are compared with experimental values for 4 reinforced concrete beams originally tested by Bresler and Scordelis (1963) to validate the proposed model, showing satisfactory results.

Keywords: Stress simulations, ANSYS, Finite element modeling, UPF.

1 INTRODUCTION

Due to its importance to the structural engineering, reinforced concrete structural elements have been object of permanent study. This is due to the complex behavior of the structural concrete as a material, since, once subjected to stresses, presents a nonlinear response.

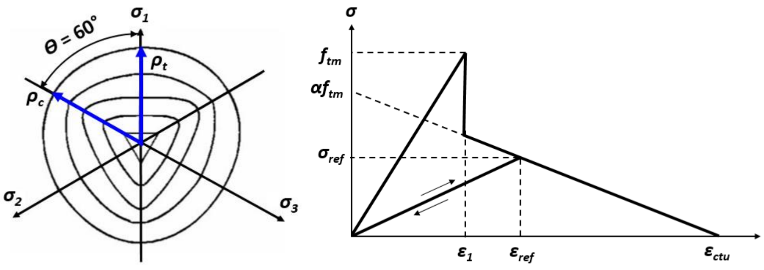
The FEM has already shown to be a worldwide-acknowledged tool due to its capabilities when analyzing complex structures and its possibilities of customization of routines for nonlinear analysis. Among commercial FEM-based computational packages, ANSYS platform is one that can be surely highlighted. This package is an important tool for nonlinear analysis of concrete structures, being frequently used in studies in this line of research.

This paper presents a computational model using ANSYS, which applies the element-embedded rebar model to simulate numerically reinforced concrete beams under plane stress states. Through an elastoplastic formulation, the proper deformed state of such structures can be found. The implementation of the constitutive model has been carried out by using the UPF customization tool of ANSYS. In order to validate the implemented subroutines interfacing the main program, reinforced concrete beams that have been experimentally tested by Bresler and Scordelis (1963) are, therefore, numerically analyzed for comparison.

2 MATERIAL CONSTITUTIVE MODELS

Since concrete behavior is extremely complex, assembling its constitutive equations is not an easy task. For a case where instantaneous loads are acting, and whether only instantaneous effects are wanted, an elastoplastic model should then be considered up to reaching the failure surface of the material. Since the main characteristic of concrete is that it is a material with a low tensile strength, relatively to its compressive strength, two different models to describe its behavior are used in this paper. An elastoplastic model with hardening is used for the concrete under compression, while an elastic linear model is used for the concrete in tension, considering the contribution of concrete between cracks for the total stiffness of the structure.

The concrete model uses the failure criterion proposed by Ottosen (1977), which is suggested by *fib* Model Code (2012). It is admitted that the compressed concrete presents an elastoplastic behavior with isotropic hardening. The concrete under tension, in its turn, is modeled as a linear elastic material with softening, i.e., before cracking occurs, the material behaves linear-elastically with softening and, after cracking, a smeared cracking model with tension stiffening is used. The cracking model used is based in the formulation presented by Hinton (1988). In Figures 1(a) and 1(b) are represented the cross sections of the failure surface and the stress-strain diagram for the concrete under tension, respectively.



(a) **(b)**

Figure 1. (a) Failure surface by Ottosen (1977); (b) Stress-strain curve for the concrete under tension.

Regarding the steel rebars, it was considered that they would resist only to axial forces and present the same behavior when under tension or compression. The behavior is then given by a bilinear stress-strain diagram. The steel rebars follow two behaviors, depending on their fabrication process. When the steel presents a yield plateau, a perfect elastoplastic model is considered, but when cold rolled steel is the case, an elastoplastic behavior with linear hardening after 85% of the yielding stress, f_y, is then used.

3 COMPUTATIONAL MODEL

The plane finite element PLANE183 was the element chosen in the list of elements of ANSYS library to model the concrete regions of the structures studied. This is an element of higher order, quadratic, two-dimensional, with eight nodes of two degrees of freedom each.

Plane stress options allow the specification of a thickness for the element through the ANSYS command “Real Constant”. Specifically, the element PLANE183 has been chosen because it gives good results with relatively coarse meshes. Besides, this element has compatibility with element REINF263, which has been chosen to model the reinforcing bars and stirrups.

The element REINF263 can be used together with certain elements of ANSYS library. This element is suited to simulate, for instance, reinforcing fibers aligned in one direction. In this case, each fiber would be individually modeled, taking into account material and cross section properties and considering only axial stiffness. The nodal coordinates, degrees of freedom, and internal connections of the reinforcing element would be identical to the basis element.

Since in this study is being considered the analysis of structural elements under plane state stresses, it is possible to lump together the rebars of the beams in the modeling, as can be seen in Figure 2. The same illustration also shows an example of beam discretization with transparency, which allows visualization of the REINF263 elements. However, code scripting is the more efficient method to add these elements, making it easier to verify possible mistakes.

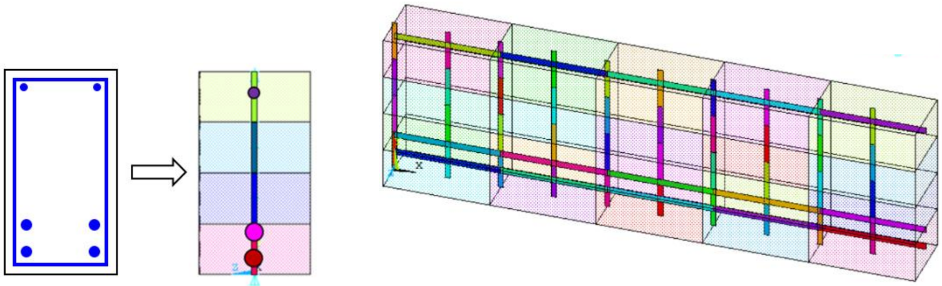


Figure 2. Example of discretization with elements REINF263.

In addition to a great variety of finite elements, ANSYS presents also a number of constitutive models to describe the behavior of materials. Regarding concrete, the program uses an elastoplastic model with the failure surface of five parameters given by Willam and Warnke (1975). Unfortunately, this model is only available for the element SOLID65, which does not present the element-embedded rebar model as an option.

With the objective of adopting the element-embedded model with elements REINF263 together with elements PLANE183, the customization tool UPF (User Programmable Features) of ANSYS had to be, therefore, employed. Using the UPF, it was possible to propose a new elastoplastic material model with cracking for the concrete based on the failure criterion given by Ottosen (1977). This new model has been implemented with the use of the programing language FORTRAN via USERMAT (User Material routine), a routine present in the customization system of ANSYS. Regarding the constitutive modelling of the reinforcing bars and stirrups, the BISO model (Bilinear Isotropic Hardening) was used, which is available within ANSYS.

4 RESULTS

To verify the efficiency of the proposed model, the study of a set of 4 simply supported reinforced concrete beams (Figure 3), originaly tested by Bresler and Scordelis (1963), is presented as follows. These 4 beams were subjected to monotonically increased concentrated loads at their midspan. Details about the cross sections, span length and concrete properties of each beam, as well as information about yielding stress and modulus of elasticity for the bottom rebars (A_s), top rebars (A'_s), and for the stirrups are presented in Tables 1 and 2.

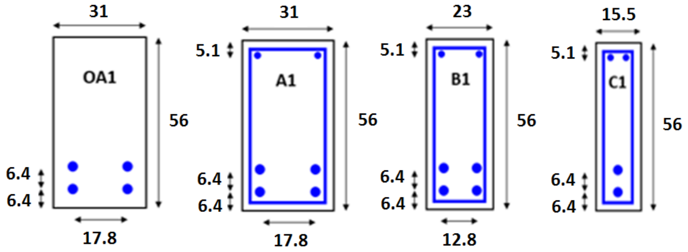


Figure 3. Sketch of the cross sections of the beams tested by Bresler and Scordelis (1963) (dimensions in cm).

Table 1. Concrete properties and cross section details of the beams.

BEAM	f_C (kN/cm²)	E_C (kN/cm²)	Span length (cm)	*h* (cm)	*b* (cm)	NOTE
OA1	2.25	2413	366	56	31	no stirrups
A1	2.41	2413	366	56	31	-
B1	2.48	2413	366	56	23	-
C1	2.96	2413	366	56	15.5	-

Table 2. Properties of the rebars and stirrups of the beams.

A_S		A'_S		STIRRUPS	
f_Y (kN/cm²)	E_S (kN/cm²)	f_y ,(kN/cm²)	E_S (kN/cm²)	f_y (kN/cm²)	E_S (kN/cm²)
55.5	21787	34.54	20133	32.54	18961

To study computationally these elements, geometrical and load symmetry were taking into consideration when modeling the beams, being specified four elements in the height of the cross sections and five elements along the half spans. Therefore, a mesh of 20 eight-node quadratic quadrangular finite elements for plane stress states was adopted (PLANE183). Inside these elements, the elements REINF263 were then embedded to model the reinforcement of the beams. Regarding their fixities, the Y direction was fixed at the bottom nodes of the element at the end of the beams, with the X direction being fixed at each node located at their midspan cross section.

In Figure 4, it is possible to observe the load-displacement curves for the four beams (OA1, A1, B1, and C1), i.e., their midspan deflection development with the applied load. Numerically, the load was applied by imposing vertical displacements at the same location of the concentrated load to simulate an instantaneous loading case up to the collapse of the beams. Thus, the load axis of the load-displacement diagram was obtained by just multiplying the values of the vertical reactions of the beams by 2. The vertical displacements were measured at the bottom node of the cross section localized at the midspan of each beam. It can be seen from the results, that the load-displacement diagrams show a good correlation between the curves presented. Based on these curves, it can be observed that the lowest rupture load value was obtained with beam C1, which is the beam with the smallest cross section. Conversely, the beam that supported the biggest rupture load was beam A1. In order to observe the stress distribution in the concrete and in the reinforcement along the structures, it is indicated in Figures 5 to 7 the stress distribution for the four beams at the instant the rupture load is reached. Throughout these diagrams, it is possible to observe that the beams collapse by bending and shear, presenting stirrups more stressed in the region of inclined cracks.

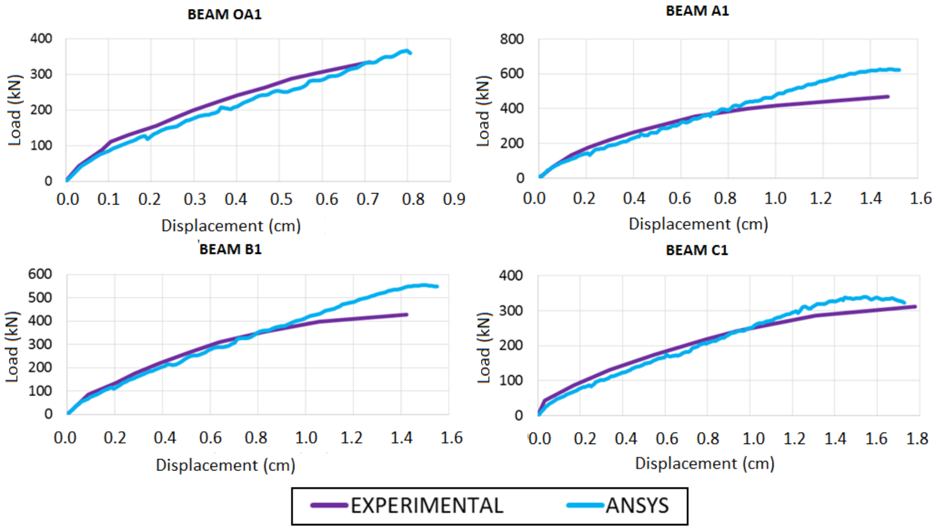


Figure 4. Comparison between midspan deflections obtained from the experiments by Bresler and Scordelis (1963) and with the proposed model in ANSYS.

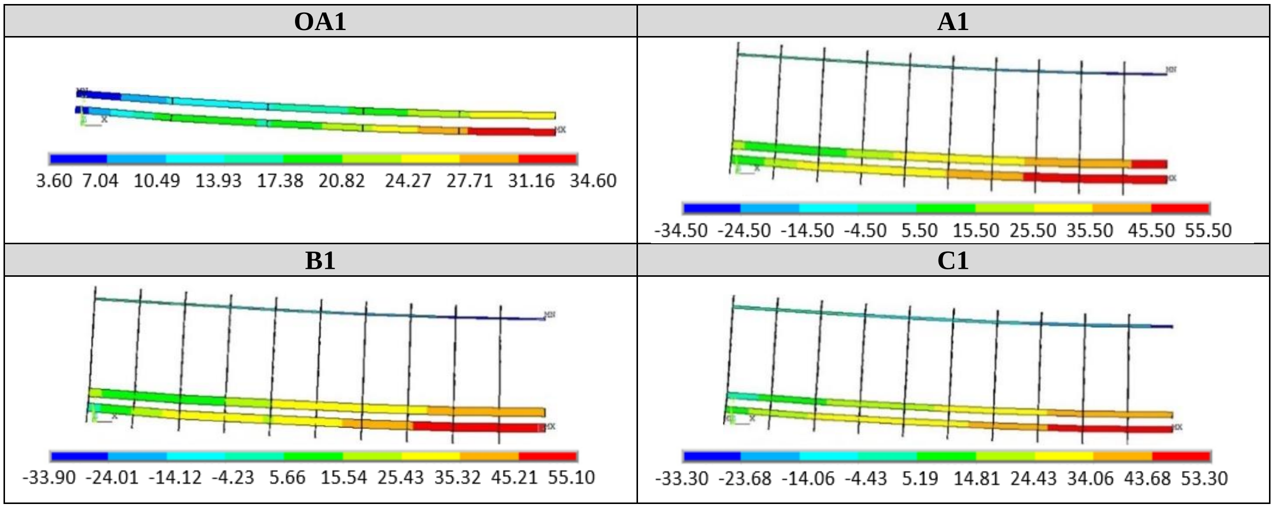


Figure 5. Stresses in rebars and stirrups of beams OA1, A1, B1, and C1, according with the computational model (units in kN/cm²).

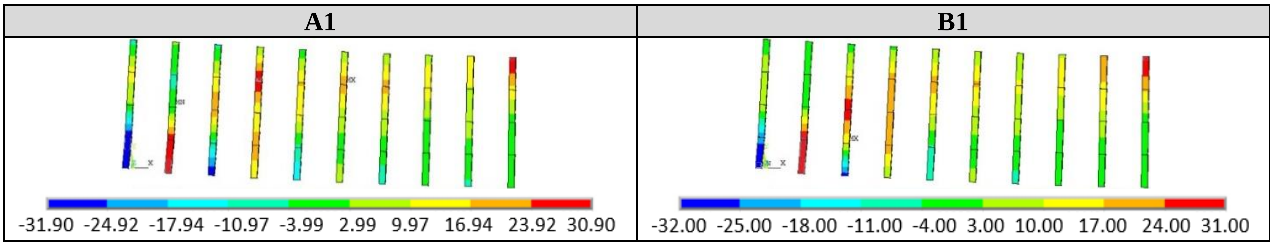


Figure 6. Stresses in stirrups of beams A1 and B1, according with the model (units in kN/cm²).

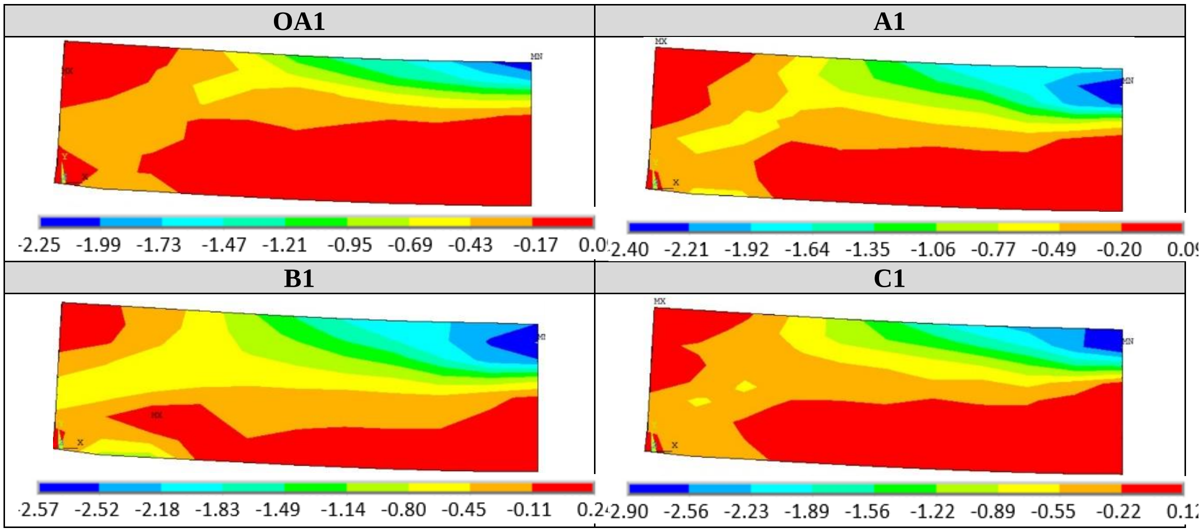


Figure 7. Stress σ_x in concrete elements of beams OA1, A1, B1, and C1, according with the computational model (units in kN/cm²).

5 CONCLUSIONS

The study herein had the main objective of presenting an elastoplastic model based on the FEM to analyze numerically reinforced concrete beams under plane state stresses. In this way, it was possible to generate in ANSYS a computational model that uses the element-embedded rebar model into concrete elements, reducing significantly the computational effort. ANSYS has shown to be a very suitable choice to implement the proposed model.

Therefore, with the good results obtained with the model proposed, the possibility of simulating computationally the real behavior of more general structural concrete elements is quite promising. Additionally, it can be also highlighted that the UPF tool available in ANSYS allows structural analysis more efficiently and precisely, with consequent optimization of materials.

A thorough analysis of the results, as well as more information regarding the models used and the scripts and codes written are presented in Lazzari (2015).

Acknowledgments

The authors gratefully thank CAPES (The Brazilian Coordination for the Improvement of Higher Education Personnel) and CNPq (The Brazilian National Council for Scientific and Technological Development) for the financial support, making the development of this work possible.

References

Comité Euro-International Du Béton, *CEB-FIP Model code 2010*. Bulletin No 65, 2012.

Hinton, E., *Numerical methods and software for dynamic analysis of plates and shells*. Swansea: Pineridge Press Limited, 550p. 1988.

Lazzari, B. M., *Finite element analysis of reinforced and prestressed concrete elements under plane stress states*. Master's Dissertation – Postgraduate Program in Civil Engineering - PPGEC, UFRGS, Porto Alegre, 2015. Website: http://www.lume.ufrgs.br/handle/10183/131037.

Ottosen, N. S., *A failure criterion for concrete*. Journal of Engineering Mechanics Division, 103(4), 527-535, 1977.

Willam, K. J. and Warnke, E. P., *Constitutive models for the triaxial behavior of concrete*. Proceedings of the International Association of Bridge Structures, 19, 1-30, 1975.