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

ANALYZING REINFORCED CONCRETE BEAMS BY USING THE UPF FEATURES OF ANSYS

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

[1]*Mobility Engineering Center, Federal University of Santa Catarina, Joinville, Brazil*
[2]*Dept of Civil Engineering, Federal University of Rio Grande do Sul, Porto Alegre, Brazil*

In this paper, the implementation of an elastoplastic constitutive model with cracking is presented for analyzing reinforced concrete structures using The Finite Element Method (FEM). This implementation was made through the UPF (User Programmable Features), a customization tool available in ANSYS (version 14.5). By using the UPF features, a new constitutive model for concrete was created (in USERMAT3D), adding new subroutines in FORTRAN language to the main program. The main advantage of the implementation of this model is the possibility of using quadratic twenty-node three-dimensional elements (SOLID186) with the element-embedded rebar model (REINF264), making the solution considerably faster and more effective, when compared to typically available options. In order to validate the subroutines added to the system, the beams tested by Leonhardt and Walther (1962) were simulated, covering a wide variety of behaviors, including bending and shear failures. Analyses of the stresses in the concrete and in the reinforcement are made and load-displacement diagrams are presented. The comparisons that are done between numerical and experimental results demonstrate the validity of the results found.

Keywords: Reinforced concrete, ANSYS, USERMAT3D, Element-embedded rebar model, UPF (User Programmable Features).

1 INTRODUCTION

Analyzing the structural behavior of reinforced concrete elements involves a variety of factors that make the task rather difficult. Among these factors, there is, for instance, the difference in behavior of concrete when under tension and under compression, its nonlinear stress-strain relation, the cracking of its matrix, and the effects on concrete due to shrinkage and creep phenomena. In this context, this work has the main objective of presenting nonlinear structural analyses of reinforced concrete beams through The Finite Element Method by using ANSYS (version 14.5). Traditionally, when concrete is modeled in ANSYS, the option CONCRETE is used to apply the model by Willam-Warnke (1975), which has five parameters to control the rupture surface. The limitation in the use of this model, however, is that it can only be used with the three-dimensional finite element SOLID65, which, in its turn, does not allow the use of the element-embedded rebar model. Considering the need of discretization of the reinforcement whether a more realistic analysis is sought, avoiding, therefore, the smeared modeling option, the alternative would be the discrete rebar model. Nevertheless, this alternative tends to demand a much higher number of finite elements for the mesh to represent satisfactorily the structure, relatively to the element-embedded option. Ultimately, the numerical simulations of structural concrete problems using the discrete alternative would become extremely slow, demanding machines with increasing computational power. In this way, to carry out computational simulations of concrete structures with the element-embedded rebar model as the option to speed up the analysis process, the customization tool of ANSYS, the UPF system (User Programmable Features), has been therefore adopted. This have also allowed a more realistic type of analysis through the implementation of an elastoplastic with cracking model to represent the concrete. This numerical model is used herein to analyze beams subjected to shear and flexural ruptures. To validate the proposed model, which has been implemented in FORTRAN programming language in the subroutine USERMAT3D, the beams experimentally tested by Leonhardt and Walter (1962) were numerically simulated for comparison. The results are commented and thoroughly analyzed herein, overall showing good adherence between numerical and experimental values.

2 CONSTITUTIVE MODELS OF THE MATERIALS

Considering the concrete modeling, two different models have been used to describe the material's behavior. An elastoplastic model with hardening has been adopted for the concrete under compression, while a linear elastic model up to the rupture condition with tension stiffening has been used for the concrete under tension, i.e., considering the contribution of the concrete between cracks. The model for the concrete under compression is composed of a rupture criterion, a plastification criterion, and a hardening rule. Regarding the rupture criterion, the surface of rupture by Ottosen has been the one adopted, since it is being currently recommended by the *fib* Model Code 2010 (2012). It is possible to visualize this surface of rupture in a three-dimensional space of stresses in Figure 1(a), being represented by two cross sections (CHEN; HAN, 1988). In this work, the compressed concrete is considered to have isotropic hardening, while the plastification surfaces are considered to have the same shape of the surface of rupture. The loading and rupture surfaces can be observed in Figure 1(b).

(a) (b) (c) (d)

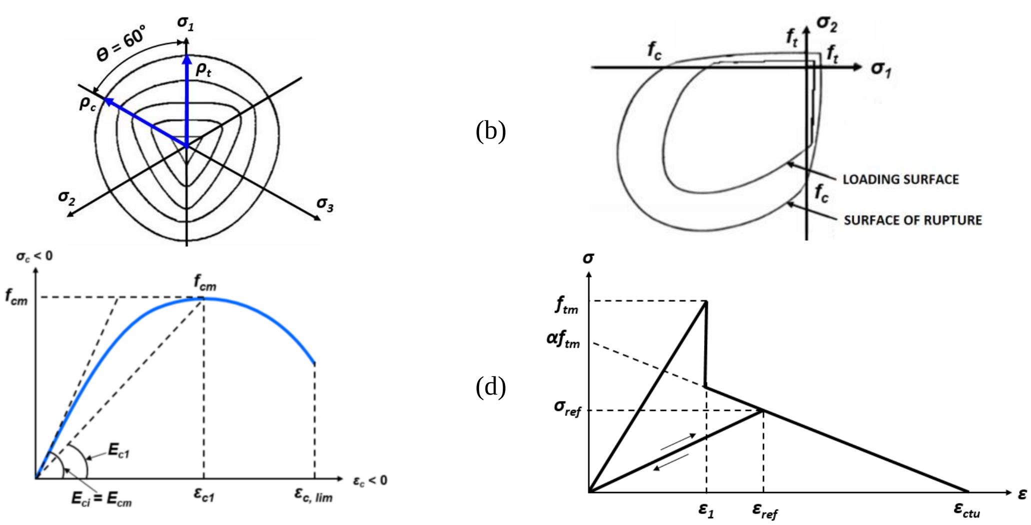


Figure 1. (a) Deviator cross sections; (b) plastification surfaces; (c) stress-strain curve for the concrete under compression (d); stress-strain curve for the concrete under tension.

The hardening rule defines how the plastification surfaces (loading surfaces) move during plastic deformation. It is determined by the effective plastic stress-strain relation, which allows extrapolation of the results from a simple uniaxial tension test to a multiaxial configuration. In this work, the stress-strain diagram proposed by the *fib* Model Code 2010 (2012), illustrated in Figure 1(c), was used to represent the uniaxial compressed concrete. The concrete under tension was modeled as an elastic material with softening. Before cracking, the concrete behaves as a linear elastic material but, after cracking, the smeared cracking model with tension stiffening is used, as indicated in Figure 1(d). The cracking model used, which is comprised of a cracking criterion, a between cracking collaboration rule, and a shear stress transfer model, is based in the formulation presented by Hinton (1988).

Considering, as usual, that steel rebars resist only axial forces, an uniaxial model has been adopted to represent their behavior. The steel is represented by a perfect elastoplastic material, presenting the same behavior both in tension and in compression. It has been considered that the reinforcing bars could present two behaviors, depending upon their fabrication process. When steels with well-defined yield plateau are considered, the perfect elastoplastic model has been adopted, but when cold rolled steels are the case, an elastoplastic behavior with linear hardening after reaching 85% of the yield strength have been the choice more appropriately considered.

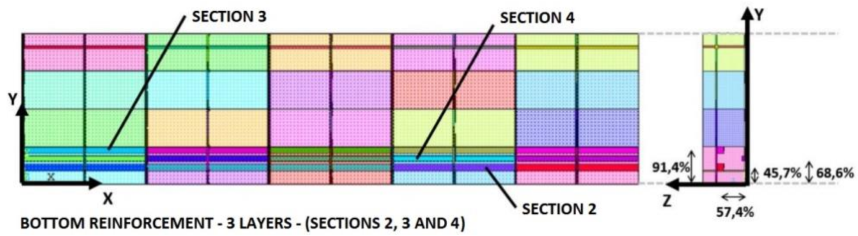
3 COMPUTATIONAL MODEL

The hexahedral element SOLID186 was used to model the concrete, a quadratic three-dimensional twenty-node element with three degrees of freedom per node (translations in X, Y, and Z directions). In this way, good results can be obtained without the need of enforcing meshes excessively refined, reducing significantly the processing time of structural analyses. Another important factor considered when choosing that element is the compatibility of the element with the element REINF264, a fundamental condition to represent the concrete with its embedded reinforcement. The reinforcing element REINF264 has been used to represent the rebars and stirrups along the structural concrete beams by means of the element-embedded type of modeling. The element REINF264 is adequate, for instance, to simulate reinforcing fibers with arbitrary directions, where each fiber is separately modeled as a segment that has only a uniaxial stiffness. Additionally, many REINF264 reinforcing fibers could be specified for one single base element. The nodal coordinates, degrees of freedom, and connectivity of element REINF264 are identical to those of the base element. Moreover, this element allows a number of models, including models for plasticity, creep, initial stresses, large displacements, and large strains. Figure 2 presents an example of an ANSYS APDL script for use of longitudinal bottom rebars on a reinforced concrete beam. The reinforcements must be indicated for each element.

4 ANALYSES OF RC BEAMS

To validate the implementation of the elastoplastic constitutive model with cracking, the results obtained through the computational model were compared with the values of four reinforced concrete beams determined experimentally by Leonhardt and Walther (1962), having been denominated beams ET1, ET2, ET3, and ET4. Two concentrated loads are applied on the beams, as illustrated in Figure 3. The mean concrete strength of the concrete, as reported for the experimental tests, was equal to 2.42 kN/cm^2.

All the beams had the same longitudinal reinforcement, being composed of 4 bottom rebars of 20 mm in diameter (f_y = 42.8 kN/cm^2), and two top rebars of eight mm in diameter (f_y = 46.5 kN/cm^2). All the rebars were made of cold rolled steel (old Class B steel in Brazil), but the stirrups were six mm in diameter (f_y = 32 kN/cm^2), and made of laminated steel (old Class A steel in Brazil). The modulus of elasticity of the steel was assumed equals to 210 GPa for the rebars and 195 GPa for the stirrups.




```
! --------------------
! BOTTOM REINFORCEMENT - 3 LAYERS -- SEC 2, 3, AND 4
! --------------------
!
! Define discrete reinforcing section 2
!
sect, 2, REINF, DISC                                  !discrete reinforcement at section 2
secdata, 2, 6.47, EDGo, 1, 0.457, 0.574, 0.457, 0.574 !material: 2; cross section area: 6.47cm2; discr. reinf. (EDGo)
!                                                     !reinf. direction (1); located at 45.7% in y and 57.4% in z
! Create reinforcing element via EREINF
!
FLST, 5, 2, 2, ORDE, 2                                !selects elements from 1 to 5
FITEM, 5, 1
FITEM, 5, -5
ESEL, S, , , P51X
SECN, 2                                               !define these elements as section 2
EREINF                                                !generates reinforcement in these elements
```

Figure 2. Exemplifying the programing of element REINF264.

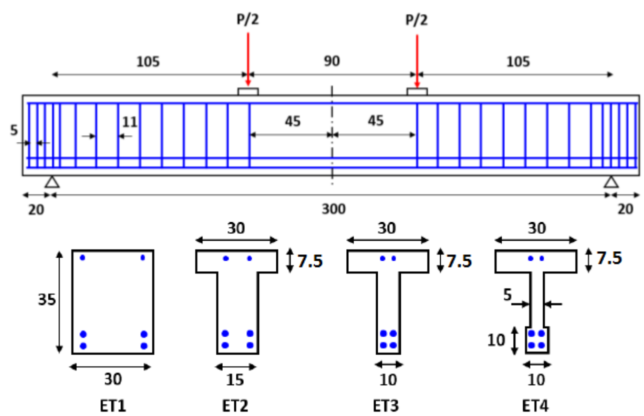


Figure 3. Geometrical characteristics (in cm) of the beams by Leonhardt and Walther (1962).

The computational study of these four beams has been carried out by dividing their heights in 4 elements and their spans in five, resulting in 20 elements in each case. A finite element mesh composed of quadratic hexahedral twenty-node elements (SOLID186) was adopted and, inside them, REINF264 elements have been added to represent the embedded reinforcement of the beams. Since the beams presented geometrical and loading symmetry, only a quarter of their volume has been modeled. Regarding their fixities, and considering an XYZ coordinate system with X along the spans (L) of the beams and Y along their heights, displacements in direction X have been restricted on the plane parallel to YZ at X equals to $L/2$ (midspan). Additionally, displacements in direction Z have also been restricted on the plane parallel to XY at Z equals to zero, as well as in direction Y at the bottom nodes of the ends of the beams at X and Y equal to zero. The loading has been incrementally applied in the form of displacements at the top surface nodes of the beams at position X equals to 105 cm. Figure 4 shows the load-displacement curves for the four beams ET1, ET2, ET3, and ET4. The displacement is measured at the bottom central nodes along the spans of the beams (midspan deflections). The results of the load-displacement diagrams present a good correlation between the presented curves, being also noticeable that the numerical rupture loads are quite close to those that were experimentally found.

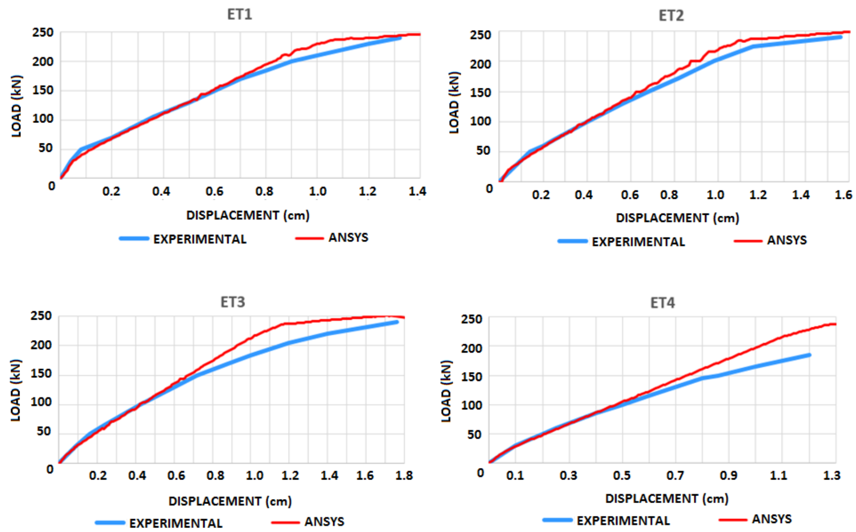


Figure 4. Load-displacement diagrams for beams ET1, ET2, ET3, and ET4.

The distribution of stresses along the structures can be observed in Figures 5 and 6, respectively for stresses in the concrete and in the reinforcement. It can be observed that narrower widths lead to bigger stress values in the stirrups, culminating with a shearing rupture type. It is important to observe that, for the rupture loads of all the beams, the maximum compression stress values are above concrete's strength. This inconsistency is due to the stress extrapolation that is typically done from the Gauss points to the nodes of the finite elements.

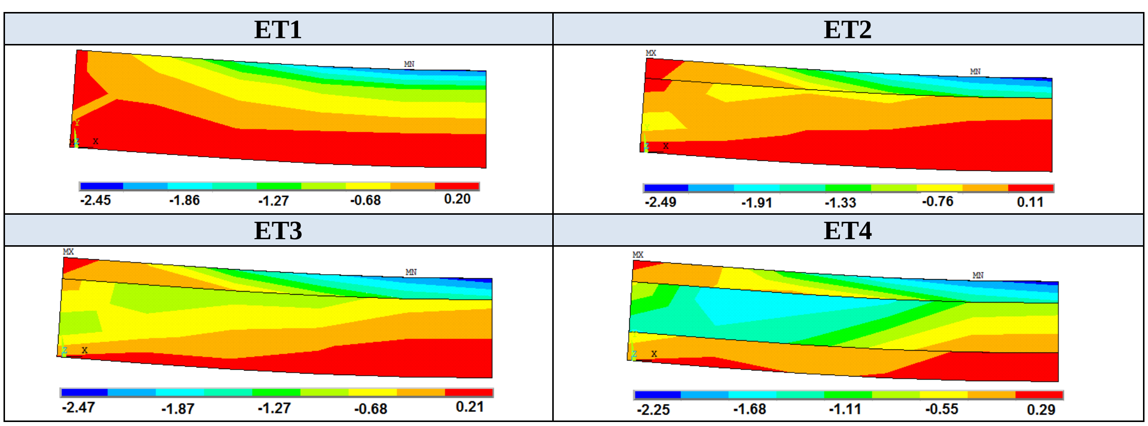


Figure 5. Stresses in the concrete for beams by Leonhardt and Walther (1962) – kN/cm².

In the diagrams showing axial stresses for the reinforcement, it can be observed that, for beam ET1, when the rupture load is reached, the bottom rebars are already at the yield plateau, while the stirrups are only mildly tensioned. The bottom rebars of beams ET2 and ET3 can also be verified to be at yield plateau, but their stirrups also are with stresses near their maximum axial stress. Finally, regarding beam ET4, it reaches maximum stress at the stirrups before the beginning of the yielding process of the bottom rebars. Therefore, it gets evident that beam ET1 collapses by bending, while beams ET2 and ET3 collapse by both bending and shear, with beam ET3 collapsing by shear only.

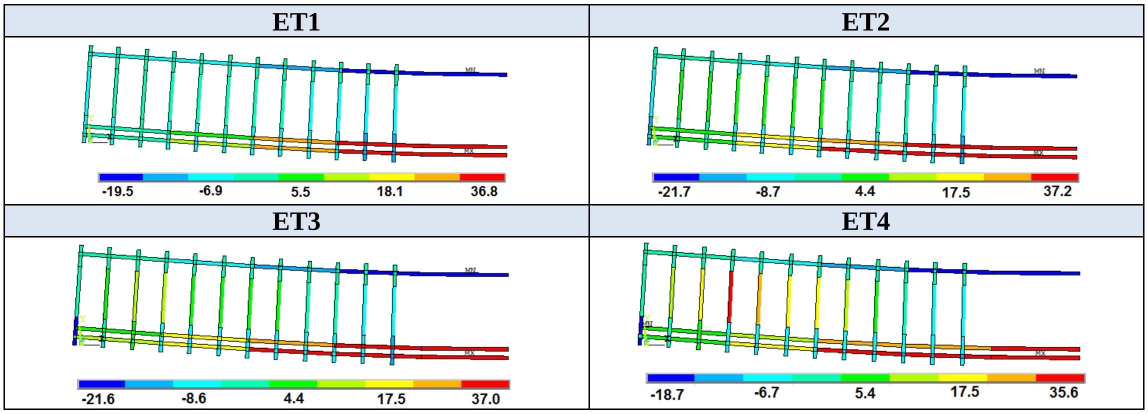


Figure 6. Stresses in the reinforcement for beams by Leonhardt and Walther (1962) – kN/cm².

5 CONCLUSIONS

By means of implementing subroutines of a new elastoplastic material behavior with cracking for concrete using the customization system of ANSYS, it was possible to apply a proposed finite element model to analyze structural concrete beams. The USERMAT3D subroutine created is validated through computational simulation of beams that were originally tested in experiments by Leonhardt and Walther (1962). The comparison between numerical and experimental results show quite satisfactory results. Thus, it can be concluded that the UPF tool that is available in ANSYS have shown to be very useful to analyze concrete structures, making it possible the modeling with the element-embedded rebar option, which minimizes computational costs. The numerical model implemented herein is fully described in the PhD dissertation by Lazzari (2016).

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

Chen, W. F. and Han, D. J., *Plasticity for structural engineers*, New York: Springer-Verlag, 606p, 1988.

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, P.M., *Numerical Simulation of Construction Stages of Cable-stayed Bridges through the Finite Element Method*, Thesis (Doctorate in Civil Engineering) – Postgraduate Program in Civil Engineering - PPGEC, UFRGS, Porto Alegre, 2016.

Leonhardt, F., Walther, R., *Beiträge zur behandlung der Schubprobleme im Stahlbetonbau*, Beton und Stahlbetonbau, 57(7), July, 1962.