

## ANSYS IMPLEMENTATION OF A REINFORCED CONCRETE MATERIAL MODEL

Vladimir Vitanov<sup>1</sup>

UDK: 666.982.2 : 004.4 ANSYS

DOI:10.14415/konferencijaGFS 2016.008

*Summary: ANSYS is one of the most widely used programs for FEM analyses of civil engineering structures in both practical and research applications. Its vast element and material libraries enable modeling of different kinds of structures made of various materials subjected to various loading conditions. One especially attractive feature of ANSYS that particularly appeals to researchers is the possibility of adding custom features that are not present in the default ANSYS installation. This paper presents an example of implementation of Darwin and Pecknold's inelastic model for cyclic biaxial loading of reinforced concrete into ANSYS. The correctness of the implementation is confirmed by comparing the numerical results of the analyses of RC members with the available experimental data.*

*Keywords: FEM, FORTRAN, programming, reinforced concrete, material model*

### 1. INTRODUCTION

The extensive research in the field of numerical modeling of reinforced concrete members and structures has led to the development and formulation of many models which are capable of predicting their behavior under various conditions. These models are put to real use when implemented in a suitable computer code. Then they can be used for both practical and research purposes. Many computer codes exist and are used daily in the construction industry, research activities and engineering education.

ANSYS [1] is one of the most widely used computer programs for structural analysis using the finite element method. Its vast material models and finite element libraries enable engineers and researchers to use it for wide range applications and not only in construction but also in other industries and scientific fields. One of the features of ANSYS that especially appeals to researchers is the possibility to customize it by adding additional features and capabilities. This facilitates the research and formulation of new material models or finite elements contributing to the expansion of the scientific knowledge and understanding in various scientific fields.

ANSYS is also used in modeling of reinforced concrete members and structures. However, at the present time, ANSYS provides only one dedicated material model for reinforced concrete – the constitutive model for triaxial behavior of concrete after

---

<sup>1</sup> Vladimir Vitanov, Phd, Ass.Prof., Ss.Cyril and Methodius University – Skopje, Civil Engineering Faculty - Skopje, R.Macedonia, tel: ++389 2 3116066, e – mail: v.vitanov@gf.ukim.edu.mk

Williams and Warnke [2]. Furthermore only one finite element can be used with that model – the three dimensional eight noded solid isoparametric element Solid65.

An example of implementation of another constitutive model of reinforced concrete into ANSYS will be shown in the following sections. The model chosen for this task is Darwin and Pecknold’s [3] inelastic model for biaxial loading of reinforced concrete.

## 2. RC MODEL DESCRIPTION

The RC model of Darwin and Pecknold uses the concept of “equivalent uniaxial strain” to determine equivalent uniaxial stress-strain curves for plain concrete under biaxial loading. Very often for such a purpose the curves are based on equation suggested by Saenz [4], (Figure 1ispod):

$$\sigma_i = \frac{\varepsilon_{ui} \cdot E_0}{1 + \left( \frac{E_0}{E_s} - 2 \right) \frac{\varepsilon_{ui}}{\varepsilon_{ci}} + \left( \frac{\varepsilon_{ui}}{\varepsilon_{ci}} \right)^2} \quad (1)$$

where  $E_0$  is the tangent modulus of elasticity at zero stress,  $E_s$  is the secant modulus at the point of maximum compressive stress  $\sigma_{ci}$ , and  $\varepsilon_{ci}$  is the equivalent uniaxial strain at the maximum compressive stress, (Figure 1.).

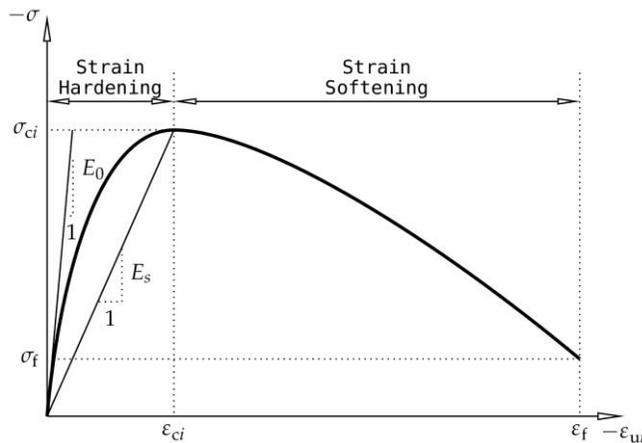


Figure 1. Equivalent Uniaxial Stress-Strain Curve for Concrete in Compression [4]

The value of the maximum compressive stress  $\sigma_{ci}$  is determined using the concrete failure surface proposed by Kupfer and Gerstle [5] based on the experimental data by Kupfer et al. [6],(Figure 2.).

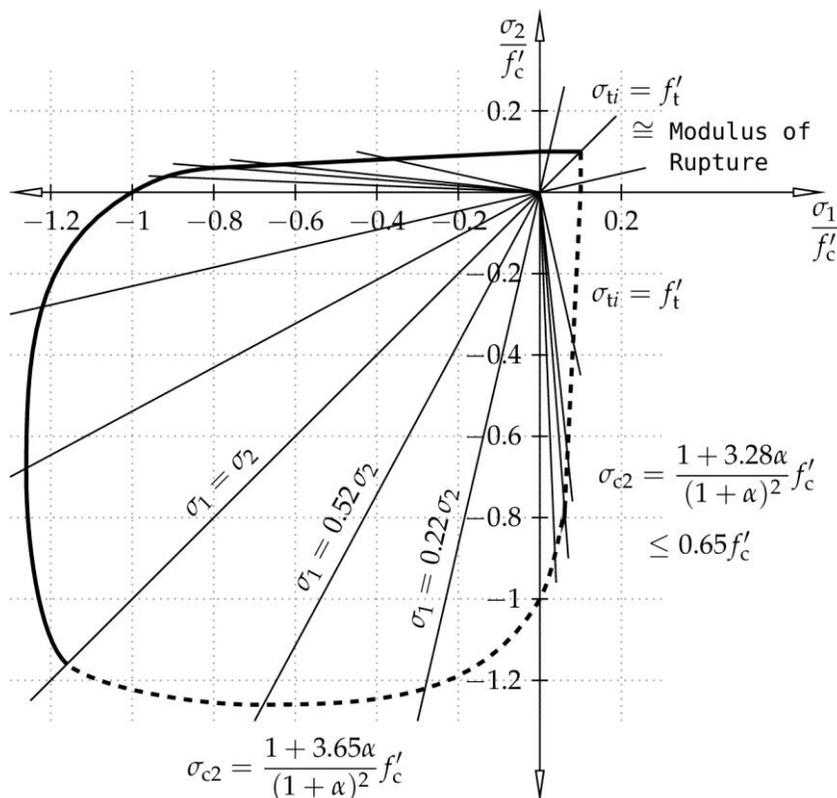


Figure 2. Analytical Biaxial Strength Envelope [5]

The model adopts the smeared crack approach in modeling the cracking behavior of the concrete as a primary mechanism for development of inelastic deformations in the material.

The reinforcing steel is uniaxial material which is also “smeared” throughout the concrete. A simplified bilinear model for the stress-strain behavior of steel is used, (Figure 3.).

### 3. ANSYS IMPLEMENTATION

The RC model described in the previous sections was implemented into the finite element method program ANSYS. ANSYS provides a set of Fortran 90 functions and subroutines that are available to extend or modify its capabilities, for ex.: customizing loads, customizing material behavior, creating new elements, modifying and monitoring existing elements, creating custom optimization routines, etc.

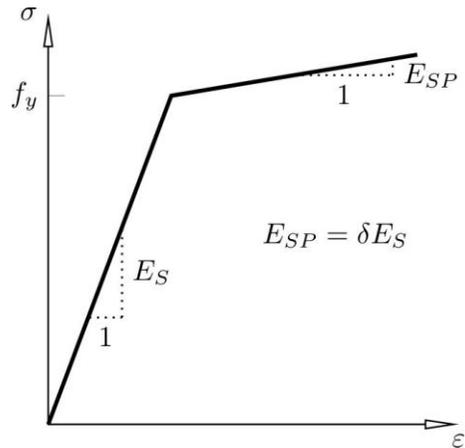


Figure 3. Idealized Stress-Strain Curve of Steel adopted in the Model

For an implementation of a custom material model, the USERMAT.F subroutine with its set of additional subroutines to be used with different stress states is provided. This subroutine can be used with the ANSYS 18x class of elements (Link180, Shell181, Plane182, Plane183, Solid185, Solid186, Beam188 and Beam189). Of these, the surface elements Shell181, Plane182 and Plane183 can be used to model members in plane stress state. Because Shell181 element formulation is based on logarithmic strain measure, only Plane182 and Plane183 are applicable with the material model used here. With the USERMAT.F subroutine modified to contain the custom material model, the main program is re-linked and recompiled which results with a custom version of ANSYS containing the user defined material model. During the analysis, for every Newton - Raphson iteration, USERMAT.F is called at every material integration point. ANSYS passes the stresses, strains and state variables at the beginning of the time increment and the current strain increment. USERMAT.F then updates the stresses and state variables at the end of the time increment and returns them to the main program. It also provides the current material Jacobian matrix, (Figure 4)

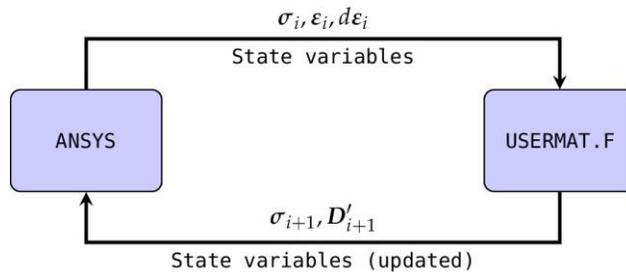


Figure 4. Input/output from ANSYS and USERMAT.F

## 4. EXAMPLE

An analysis of the behavior of shear panel (W-2) (Fig.5) tested by Červenka and Gerstle [7,8] was carried out. The tested panel is 7.62 cm (3 in) thick and has thickened ribs on the two sides. The materials parameters used for the concrete were 25.17 MPa (3650 psi) for the uniaxial compressive strength, 3.65 MPa (530 psi) for the uniaxial tension strength 20 GPa ( $2.9 \times 10^6$  psi) for the elasticity modulus, 0.2 for the Poissons ratio and -0.0025 for the equivalent uniaxial strain at the maximum compressive stress, (Eq.1, Figure 1). The material parameters used for the steel reinforcement were: 353.01 MPa (51200 psi) for the yield strength, 188.22 GPa ( $27.3 \times 10^6$  psi) for the elasticity modulus and 0.0092 for the strain hardening stiffness ratio. The reinforcement ratio for the different parts of the panel are given in Table 1.

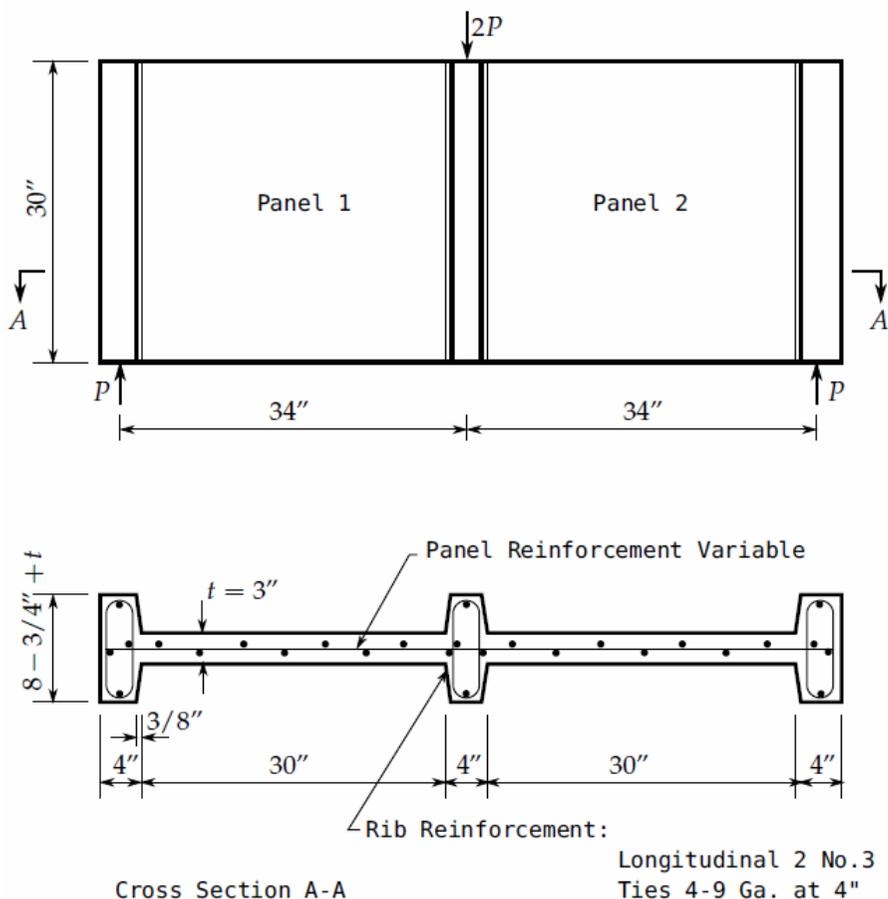


Figure 5. Shear Panel Test Specimen W-2

The FEM model of the shear panel is shown in Fig.6. The thickness of the panel is 7.62 cm (3 in), and of the ribs is 29.8 cm (11.75 in). The panel has varying reinforcement over the height – the lower part with a height of 15.24 cm (6 in) is more heavily reinforced than the upper one. The finite element mesh was created to accommodate this feature. Several analysis runs were performed, the load was applied as a vertical displacement in the lower left corner of the model. Element size was varied in order to investigate the sensitivity of the solution and the run times. The results shown in Fig.7 indicate that the solution is quite stable element-size-wise, i.e. different sized element meshed produced similar results. Conversely, run-times highly increased with the reduction of the element size.

Table 1. Reinforcement ratios of the shear panel model W-2

	Reinforcement ratio	Upper part	Lower part
<b>Panel</b>	$p_x$	0.00917	0.01833
	$p_y$	0.00917	0.00917
<b>Ribs</b>	$P_x$	0.00528	0.01326
	$p_y$	0.00468	0.00468

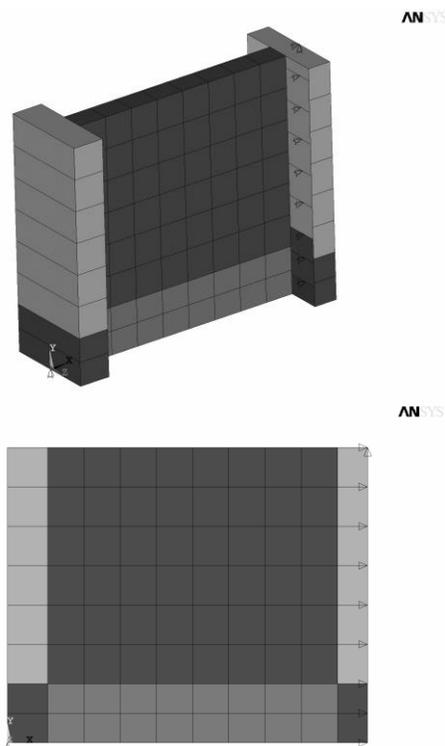


Figure 6. FEM model of Shear Panel W-2 in ANSYS

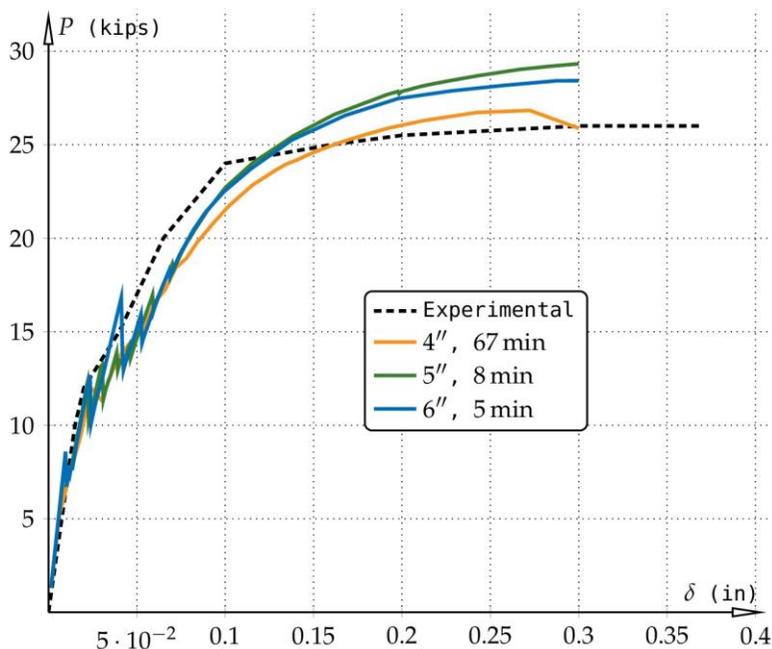


Figure 7. Monotonic Load Test – Shear Panel W-2

## 5. CONCLUSION

The inelastic model for biaxial loading of reinforced concrete of Darwin and Pecknold was implemented into the general purpose finite element code ANSYS and its results were compared to the available experimental data.

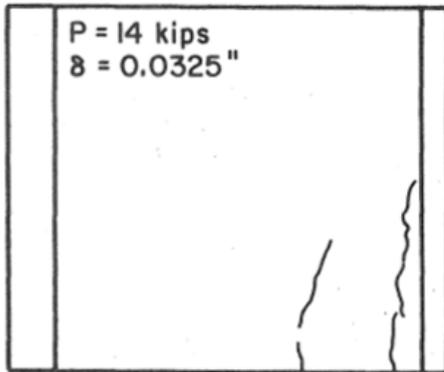
There are two notable differences in the obtained results and the experimental data. First, the ultimate force was highly overestimated, which could probably be attributed to the assumption of perfect bond between the concrete and the steel.

Also, the simulation curves don't show distinct "yielding" point as the experimental one but gradually increase until the maximum displacement is reached.

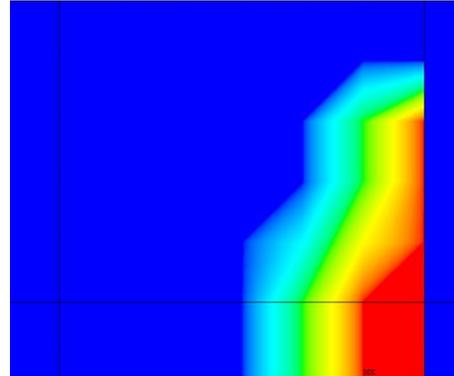
In general, it can be concluded that the acquired results show satisfactory correlation with the experimental data.

Similar conclusion can be drawn for the calculated crack patterns shown in Fig.8.

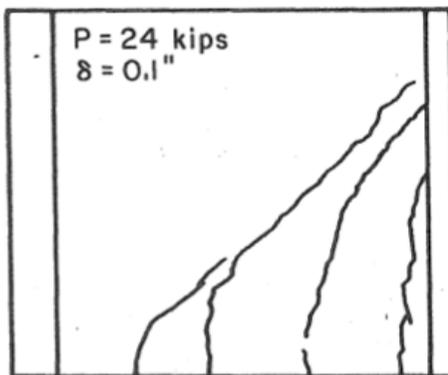
It can be seen that the crack patterns and the vertical reaction force of the numerical model correlate well with the experimentally acquired for different levels of displacement.



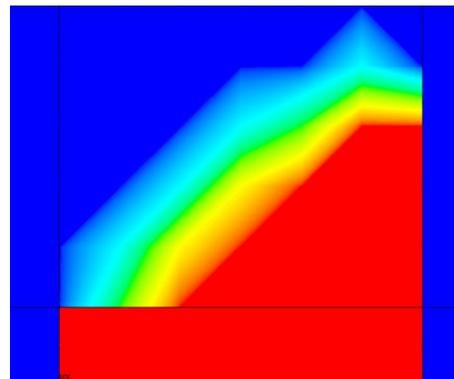
(a) experimental



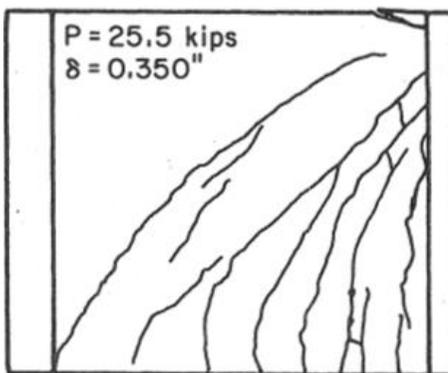
(b) Simulation,  $P=12.7$  kips



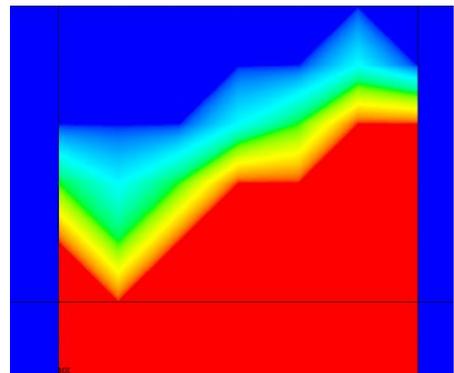
(c) experimental



(d) Simulation,  $P=21.2$  kips



(e) experimental



(f) Simulation,  $P=24.1$  kips

Figure 8. W-2 Crack propagation

### REFERENCES

- [1] ANSYS, 2004, ANSYS®Multiphysics, ANSYS Inc., Southpointe, 275 Technology Drive, Canonsburg, PA 15317.
- [2] William, K. J., and Warnke, E. P., 1974, "Constitutive model for the triaxial behaviour of concrete," Proceedings, International Association for Bridge and Structural Engineering, ISMES, Bergamo, Italy, p. 174.
- [3] Darwin, D., and Pecknold, D. A., 1974, Inelastic Model for Cyclic Biaxial Loading of Reinforced Concrete, Univeristy of Illinois, Urbana-Champaign, Illinois.
- [4] Saenz, L. P., 1964, "Discussion of 'Equation for the Stress-Strain Curve of Concrete', by Desayi and Krishman," ACI J., **61**(9), pp. 1229–1235.
- [5] Kupfer, H. B., and Gerstle, K. H., 1973, "Behavior of Concrete Under Biaxial Stresses," J. Eng. Mech. Div. ASCE, **99**(4), pp. 852–866.
- [6] Kupfer, H., Hilsdorf, H. K., and Rusch, H., 1969, "Behavior of Concrete Under Biaxial Stresses," ACI Journal Proceedings, ACI, pp. 656–666.
- [7] Červenka, V., and Gerstle, K. H., 1971, "Inelastic Analysis of Reinforced Concrete Panels, Part I: Theory," Int. Assoc. Bridge Struct. Eng. Publ., **31-II**, pp. 31–45.
- [8] Červenka, V., and Gerstle, K. H., 1972, "Inelastic Analysis of Reinforced Concrete Panels, Part II: Experimental Verification and Application," Int. Assoc. Bridge Struct. Eng. Publ., **32-II**, pp. 25–39.

## ИМПЛЕМЕНТАЦИЈА МАТЕРИЈАЛНОГ МОДЕЛА ЗА АРМИРАНИ БЕТОН У ANSYS-У

**Резиме:** ANSYS је један од најкоришћенијих софтверских пакета у анализи грађевинских конструкција коначним елементима, како у практичним тако и у научно-истраживачким циљевима. Његове простране библиотеке коначних елемената и материјалних модела омогућавају моделсање различитих типова конструкција сачињеним од различитих типова материјала, изложеним различитим утицајима. Једна посебно атрактивна особина ANSYS-а која особито одговара истраживачима је могућност додавања корисничких елемената или материјала који се не садрже у основној ANSYS инсталацији. Овај труд претставља имплементацију нееластичног материјалног модела армираног бетона Дарвина и Пекнолда у ANSYS. Тачност имплементација је проверана и потврђена поређењем нумеричких резултата са експерименталним који се могу наћи у литератури.

**Кључне речи:** МКЕ, FORTRAN, програмирање, армирани бетон, материјални модел