

Crack Identification in Reinforced Concrete Beams Using ANSYS Software

L. Dahmani,^a A. Khennane,^b and S. Kaci^a

^a Mouloud Mammeri University, Tizi-Ouzou, Algeria

^b University of Southern Queensland, Toowoomba, Australia

УДК 539.4

Анализ развития трещин в железобетонных балках с помощью программы ANSYS

Л. Дахмани^а, А. Хеннане^б, С. Кази^а

^а Университет им. Мулуда Маммери, Тизи-Узу, Алжир

^б Университет Южного Квинсленда, Тувумба, Австралия

Разработана трехмерная нелинейная конечноэлементная модель железобетонной балки. Для численного моделирования использовался стандартный конечноэлементный пакет программ ANSYS 8.0. С использованием объемных конечных элементов типа SOLID65 разрушение бетона при сжатии моделировалось с помощью алгоритма пластичности, тогда как его растрескивание в зоне растяжения описывалось нелинейной моделью материала. Учет стального армирования балки производится для различных объемных долей стали в бетоне. Для бетонной балки без армирования результаты расчетов с помощью программы ANSYS сравниваются с аналитическими расчетами. При этом получена их хорошая сходимость. Показана применимость модели для оценки критических зон растрескивания, загрузок и перемещений в железобетонных балках при различного рода нагружении.

Ключевые слова: нелинейное конечноэлементное моделирование, железобетонная балка, программа ANSYS, трещины, статический анализ.

Introduction. Concrete structural components require the understanding into the responses of these components to a variety of loadings. There are a number of methods for modeling the concrete structures through both analytical and numerical approaches [1, 2]. Finite element (FE) analysis is a numerical one widely applied to the concrete structures based on the use of the nonlinear behavior of materials. FEA provides a tool that can simulate and predict the responses of reinforced and prestressed concrete members. A number of commercial FE analysis codes are available, along with the advanced modules for complex analyses. The use of FEA has increased because of progressing knowledge and capability of computer package and hardware. Any attempts for engineering analyses can be done conveniently and fast using such versatile FE analysis packages. Nonlinear material models have been integrated in many of general purpose finite element codes, i.e., ABAQUS, ANSYS, STRAND7, or MSC.NASTRAN. Those nonlinear

models play a vital role in nonlinear response analyses since each material component tends to possess the complicated stress-strain behavior. Among those packages, ANSYS [3] provides a three-dimensional element (SOLID65) with the nonlinear model of brittle materials similar to the concrete materials. The element features a smeared crack analogy for cracking in tension zones and a plasticity algorithm to take into account the concrete crushing in compression zones. It is eight-noded solid isoparametric element with the integration points for the cracking and crushing checks. The linear elastic behavior governs the analyses until exceeding either the specified tensile or compressive strengths. Once the principal stresses at the integration points reach the tensile or compressive strength, the cracking or crushing of concrete elements can be formed. Then, the cracked or crushed regions will form in perpendicular with the locally redistributed residual stresses to the direction of principal stress. These require the nonlinear iterative solution with high performance computer [4, 5].

1. Finite Element Modeling.

1.1. **Reinforced Concrete.** An eight-node solid element (SOLID65) was used to model the concrete. The solid element has eight nodes with three degrees of freedom at each node – translations in the nodal x , y , and z directions. The element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. The geometry and node locations for this element type are shown in Fig. 1.

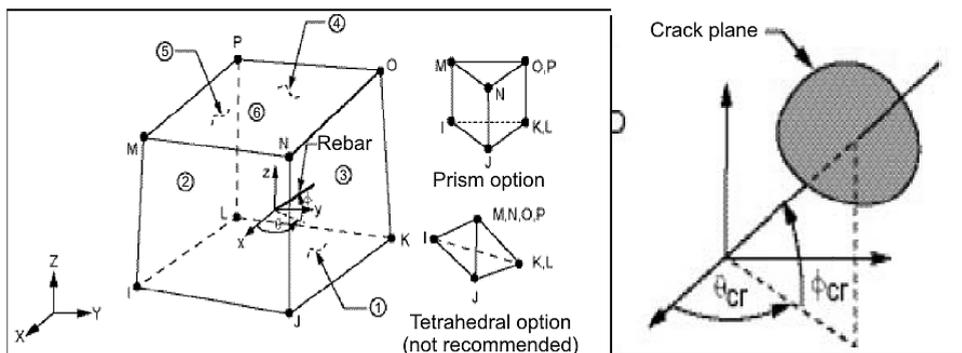


Fig. 1. SOLID65: 3D reinforced concrete solid (ANSYS 8.0).

1.2. **Steel Reinforcement.** To model concrete reinforcing, one of two methods is usually followed. In the first method, the reinforcing is simulated as spar elements with geometric properties similar to the original reinforcing (Fig. 2a). These elements can directly be generated from the nodes in the model. This method of discretization is useful in simple concrete models. The second idealization of steel reinforcing is the smeared concrete element method (used in this paper). In this case, the concrete and the reinforcing are discretized into elements with the same geometrical boundaries and the effects of reinforcing are averaged within the pertaining element (Fig. 2b). Cracks can also be idealized into either the discrete type or the smeared type.

Since the SOLID65-3D concrete element simulates tension and compression in reinforcing bars, the volumetric ratio of reinforcing steel to concrete along with the direction of the steel had to be provided for each volume, in order for the

T a b l e 1

Material Properties for ANSYS FE Model

Material	Density (kg/m ³)	Modulus of elasticity (MPa)	Poisson's ratio	F_{c28} (MPa)	F_{t28} (MPa)	F_y (MPa)
Concrete	2400	30,000	0.2	30	3	–
Reinforced steel	7850	200,000	0.3	–	–	240

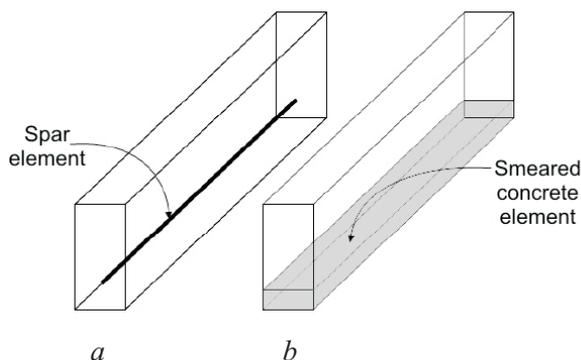


Fig. 2. Discrete vs. smeared element for concrete reinforcing.

program to account for the reinforcing steel. The required steel properties include density, modulus of elasticity, and Poisson's ratio are shown in Table 1.

1.3. **FE Model Input Data.** For concrete, ANSYS requires input data for material properties as follows: elastic modulus E_b , ultimate uniaxial compressive strength f_c , ultimate uniaxial tensile strength (modulus of rupture) f_r , Poisson's ratio ν , shear transfer coefficient β , and compressive uniaxial stress-strain relationship for concrete. The shear transfer coefficient, β , represents conditions of the crack face. The value of β_t ranges from 0 to 1.0, with 0 representing a smooth crack (complete loss of shear transfer) and 1.0 representing a rough crack (no loss of shear transfer) (ANSYS 8.0). The value of β used in many studies of reinforced concrete structures, however, varied between 0.2 and 0.5 [1, 2, 6]. A number of preliminary analyses were attempted in this study with various values for the shear transfer coefficient within this range, but convergence problems were encountered at low loads with β less than 0.2. Therefore, the shear transfer coefficient used in this study was equal to 0.3.

1.4. **Failure Criteria for Concrete.** The model is capable of predicting failure for concrete materials. Both cracking and crushing failure modes are accounted for. The two input strength parameters – i.e., ultimate uniaxial tensile and compressive strengths – are needed to define a failure surface for the concrete. Consequently, a criterion for failure of the concrete due to a multiaxial stress state can be calculated [7, 8]. In a concrete element, cracking occurs when the principal tensile stress in any direction lies outside the failure surface [1]. After cracking, the elastic modulus of the concrete element is set to zero in the direction parallel to the principal tensile stress direction. Crushing occurs when all principal stresses are compressive and

lie outside the failure surface; subsequently, the elastic modulus is set to zero in all directions [3], and the element effectively disappears.

1.5. FE Discretization. As an initial step, a finite element analysis requires meshing of the model (Fig. 3). In other words, the model is divided into a number of small elements, and after loading, stress and strain are calculated at integration points of these small elements [9]. A beam is composed of two regions; a concrete element without reinforcement and a concrete element with a smeared reinforcement (Fig. 4).

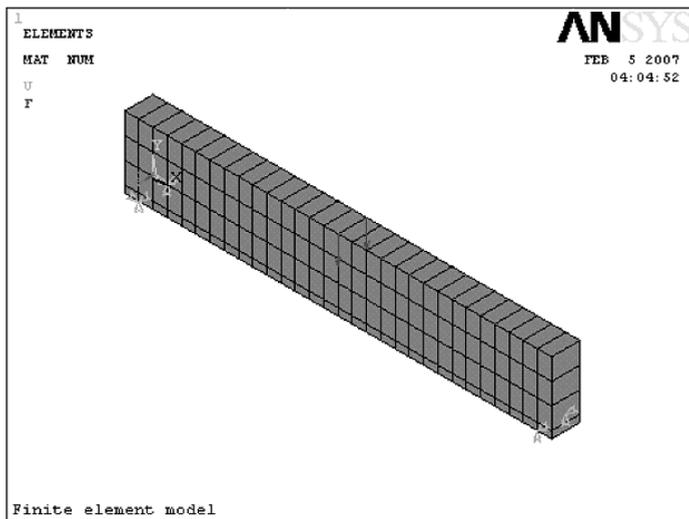


Fig. 3. Finite element model and boundaries conditions.

1.6. Nonlinear Solution. In nonlinear analysis, the total load applied to a finite element model is divided into a series of load increments called load steps. At the completion of each incremental solution, the stiffness matrix of the model is adjusted to reflect nonlinear changes in structural stiffness before proceeding to the next load increment. The ANSYS program uses Newton–Raphson equilibrium iterations [3, 10] for updating the model stiffness. In this study, for the reinforced concrete solid elements, convergence criteria were based on force and displacement, and the convergence tolerance limits were initially selected by the ANSYS program. It was found that convergence of solutions for the models was difficult to achieve due to the nonlinear behavior of reinforced concrete. Therefore, the convergence tolerance limits were increased to a maximum of 5 times the default tolerance limits (0.5% for force checking and 5% for displacement checking) in order to obtain convergence of the solutions.

1.6.1. Load Stepping and Failure Definition for FE Models. For the nonlinear analysis, automatic time stepping in the ANSYS program predicts and controls load step sizes. Based on the previous solution history and the physics of the models, if the convergence behavior is smooth, automatic time stepping will increase the load increment up to a selected maximum load step size. If the convergence behavior is abrupt, automatic time stepping will bisect the load increment until it is equal to a selected minimum load step size. The maximum and minimum load step sizes are required for the automatic time stepping.

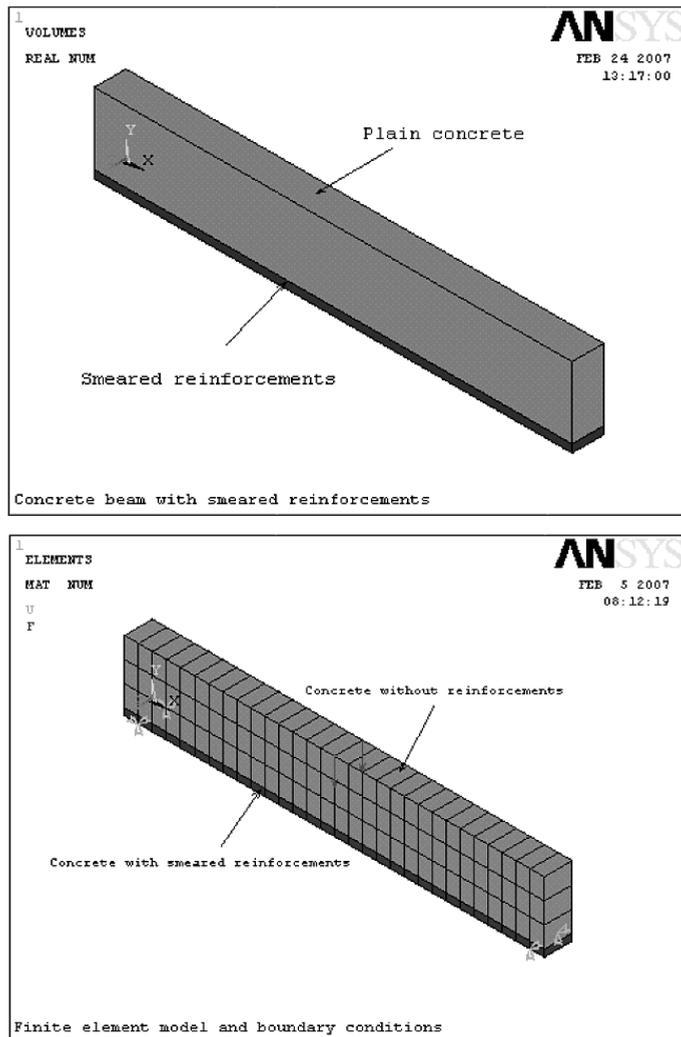


Fig. 4. Concrete element with smeared reinforcement.

2. **Results.** The goal of this study is to show:

- (i) the different phases of the FE model behavior from initial cracking, yielding of steel until failure of the concrete beam;
- (ii) the applicability of ANSYS software for analyzing and predicting of crack patterns in the reinforced concrete beam;
- (iii) the advantage of performing numerical simulation instead of experimental tests in saving time and costs.

2.1. **Behavior at First Cracking.** The analysis of the linear region can be based on the design for flexure [1, 6] for a reinforced concrete beam. Comparisons were made in this region to ensure deflections and stresses were consistent with the FE model and the beam before cracking occurred. Once cracking occurs, deflections and stresses become more difficult to predict. The stresses in the concrete and steel immediately preceding initial cracking were analyzed. The load at step 42602 was analyzed and it coincided with a load of 42,602 N applied to the beam.

Calculations to obtain the concrete stress, steel stress and deflection of the beam at a load of 42,602 N are given in Appendix. A comparison of values obtained from the FE model and Appendix can be seen in Table 2. The maximums exist in the constant-moment region of the beam during load application. This is where we expect the maximums to occur. The results in Table 2 indicate that the FE analysis of the beam prior to cracking is acceptable.

Table 2

Deflection and Stress Comparison at First Cracking

Model	Extreme tension fiber stress (MPa)	Reinforcing steel stress (MPa)	Centerline deflection (mm)	Load at first cracking (N)
Hand calculation	3.15	28.30	0.293	40,476
ANSYS	3.20	30.32	0.303	42,602

The cracking pattern(s) in the beam can be obtained using the Crack/Crushing plot option in ANSYS. Vector Mode plots must be turned on to view the cracking in the model. The initial cracking of the beam in the FE model corresponds to a load of 42,602 N that induces stress just beyond the modulus of rupture of the concrete (3.0 MPa) as shown in Table 2. This compares well with the load of 40,476 N calculated in Appendix.

The stress increases up to 3.2 MPa at the centerline when the first crack occurs. The first crack can be seen in Fig. 5, which occurs in the constant moment region, and is a flexural crack.

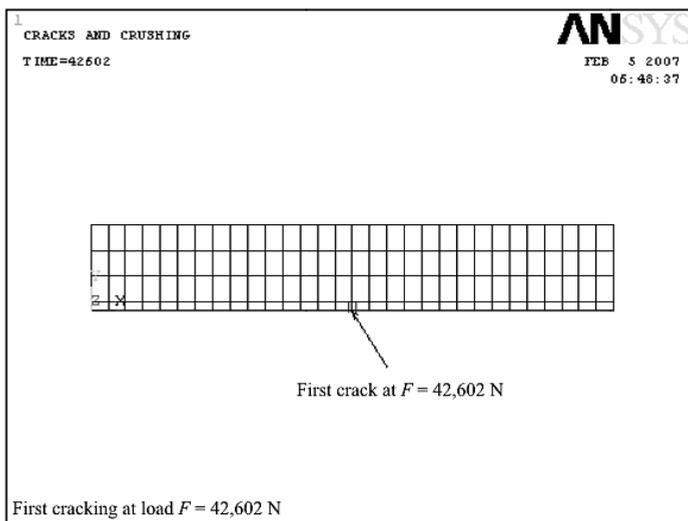


Fig. 5. First crack of the concrete model.

2.2. Behavior beyond First Cracking. In the non-linear region of the response, subsequent cracking occurs as more loads are applied to the beam. Cracking increases in the constant moment region, and the beam starts cracking out towards the supports at a load of 50,000 N.

Significant flexural cracking occurs in the beam at 60,000 N. Also, diagonal tension cracks are beginning to form in the model at load of 70,000 and 90,000 N. This cracking can be seen in Figs. 6 and 7.

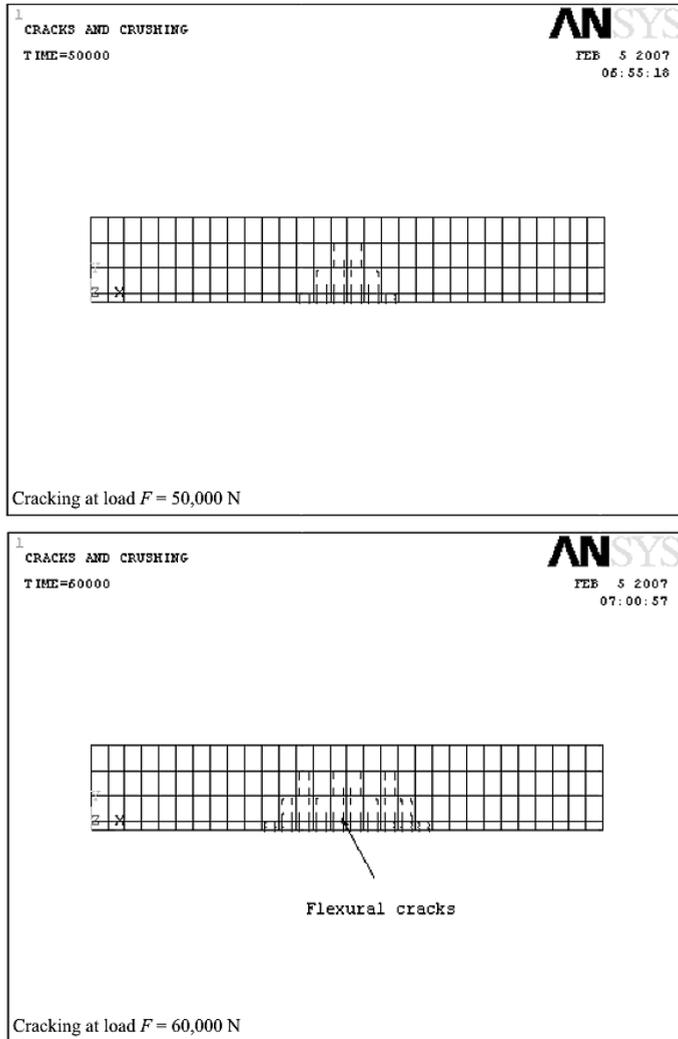


Fig. 6. Cracking at 50,000 and 60,000 N.

2.3. Behavior at Reinforcement Yielding and Beyond It. Yielding of steel reinforcement occurs when a force of 94,000 N is applied. At this point in the response, the displacements of the beam begin to increase at a higher rate as more loads are applied. The cracked moment of inertia, yielding steel and nonlinear concrete material, now defines the flexural rigidity of the member. The ability of the beam to distribute load throughout the cross section has diminished greatly. Therefore, greater deflections occur at the beam centerline. Figure 8 shows successive cracking of the concrete beam after yielding of the steel occurs. At 94,000 N, the beam has increasing flexural cracks, and diagonal tension cracks. Also, more cracks have now formed in the constant moment region. At 110,000 N, cracking has reached the top of the beam, and failure is soon to follow.

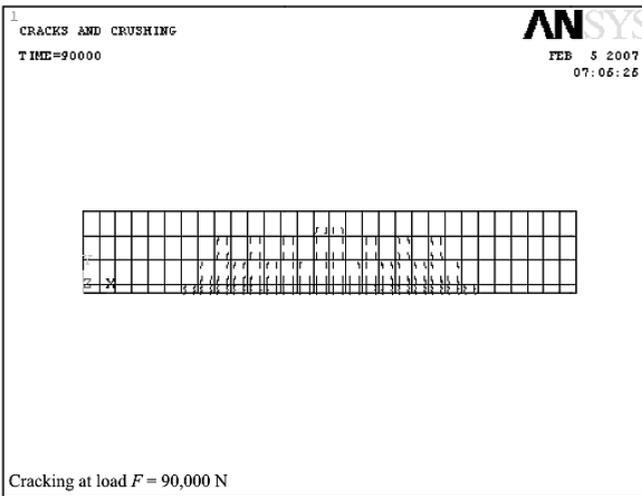
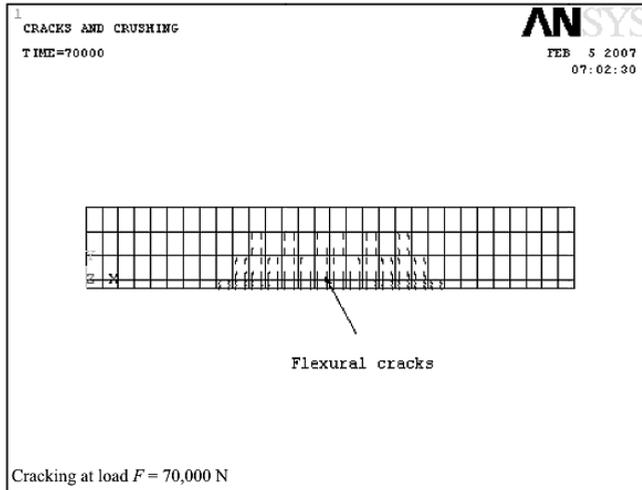


Fig. 7. Further cracking at 70,000 and 90,000 N.

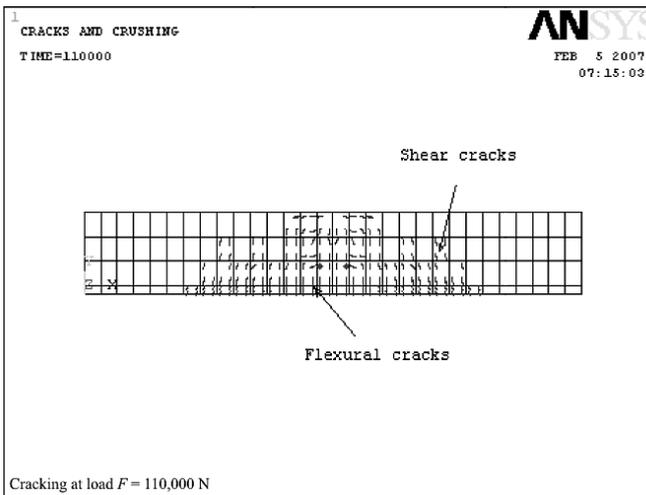


Fig. 8. Increased cracking after yielding of reinforcement.

2.4. **Strength Limit State.** At load of 114,000 N, the beam can no longer support additional load as indicated by an insurmountable convergence failure. Severe cracking throughout the entire constant moment region occurs (Fig. 9). Noteworthy is that just before the collapse few splitting cracks (compressive cracks) appear at the upper part of the beam due to crushing failure of the concrete there.

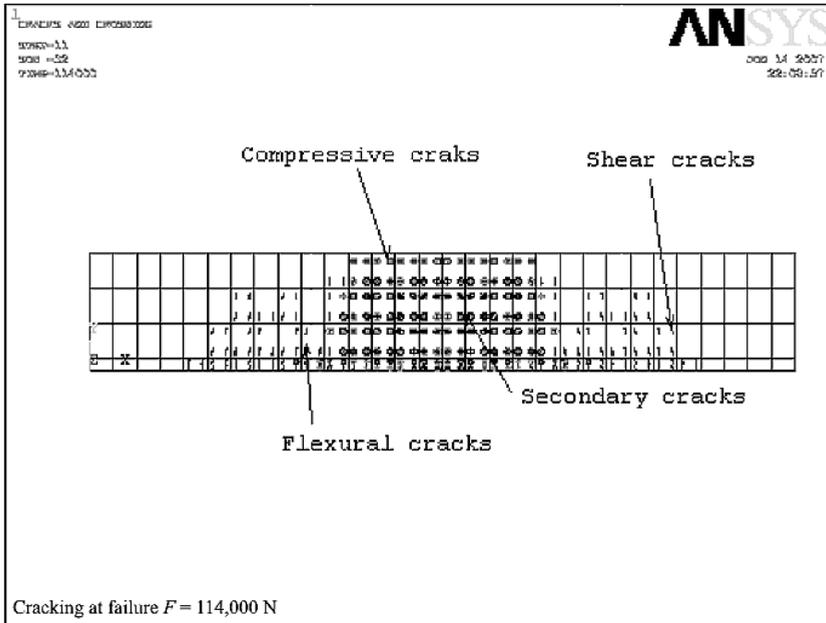


Fig. 9. Failure of the concrete beam.

2.5. **Load–Deflection Response.** Load–deflection behavior of concrete structures typically includes three stages. Stage I manifests the linear behavior of uncracked elastic section. Stage II implies initiation of concrete cracking and Stage III relies relatively on the yielding of steel reinforcements and the crushing of concrete. In nonlinear iterative algorithms, ANSYS 8.0 utilizes Newton–Raphson method for the incremental load analysis.

The full nonlinear load–deformation response is shown in Fig. 10. The response calculated using FE analysis is plotted. The entire load–deformation response of the model produced well correlates with the hand calculated results (see Appendix). This gave confidence in the use of ANSYS and the model developed.

Conclusions. Finite element models of 3.0 m ordinarily reinforced concrete beams, constructed in ANSYS V8.0 using the dedicated concrete elements have accurately captured the nonlinear flexural response of these systems up to failure.

The dedicated element employs a smeared crack model to allow for concrete cracking with the option of modeling the reinforcement in a distributed or discrete ways.

In terms of using finite element models to predict the strength of existing beams, the assignment of appropriate material properties is critical.

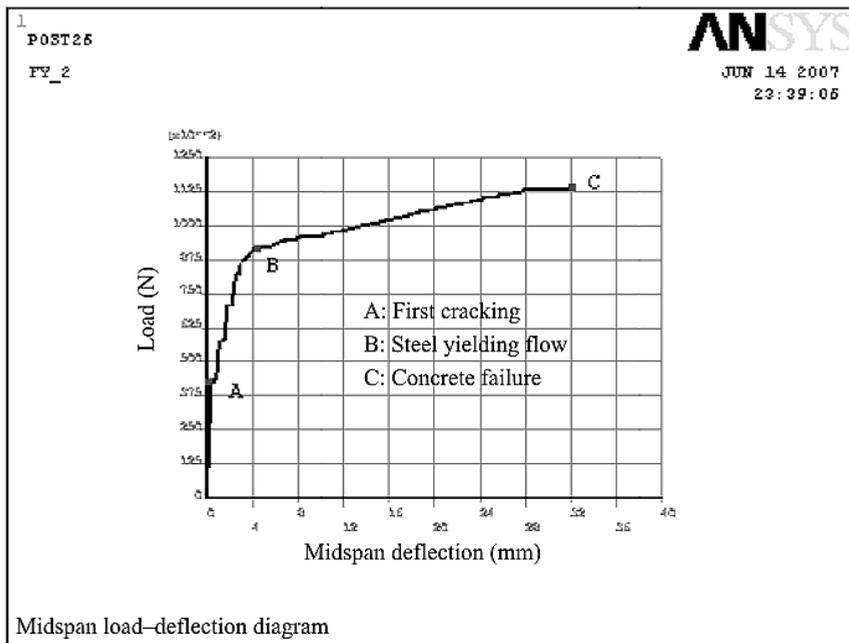


Fig. 10. Load-deflection response of the concrete beam.

The following conclusions can be stated based on the evaluation of the analyses of the reinforced concrete beam:

- the load applied to cause initial cracking of the reinforced concrete beam well correlates with hand calculations;
- flexural failure of the reinforced concrete beam is adequately modeled using a finite element package, and the load applied at failure is very close to hand-calculated results.

The good results attained suggest that, in spite of the relative simplicity of the analyzed structure and of the employed models, satisfactory prediction of the response of reinforced concrete structures may be obtained.

The failure model of concrete [8] adopted by the commercial code ANSYS is adequate to determine the nonlinear behavior of reinforced concrete structures.

Recommendations for Future Work. Usage of ANSYS smeared reinforcement approach for beams with moderate shear span failed to reproduce satisfactory results, in comparison with the hand-calculated values. That is likely to be more realistically achieved by using a discrete reinforcement approach.

The particular concrete finite element does not consider one of the most important fracture mechanics parameters – the fracture energy G_F . That means that in the case of concrete beam with no reinforcement no solution will be obtained.

The literature review and analysis procedure utilized in this paper has provided useful insight for future application of a finite element package as a method of analysis. To ensure that the finite element model yields results that can be used for study, any model should be calibrated with reliable experimental data. This will provide the proper modeling parameters needed for later use.

Appendix

Analysis of reinforced concrete beam for flexure at applied load of 42,602 N (see Fig. 11).

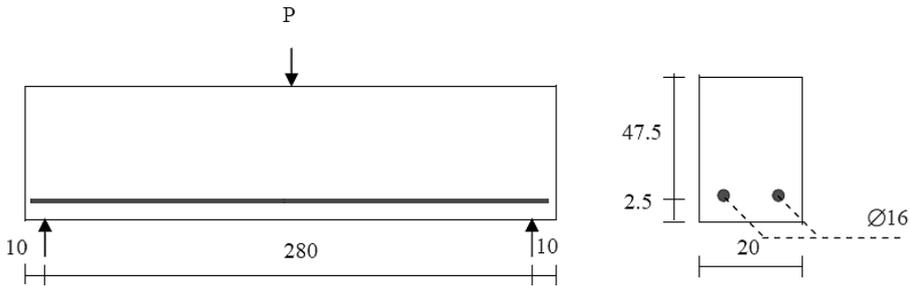


Fig. 11. Geometry, loading, and beam reinforcements (dimensions in cm).

Maximum Moment. The moment that occurs from the existing forces

$$M_{\max} = (42,602 \cdot 280) / 4 = 2982,140 \text{ N} \cdot \text{cm}.$$

Material Properties. The gross moment of inertia

$$I_G = bh^3 / 12 = (20 \cdot 50^3) / 12 = 208,333.34 \text{ cm}^4.$$

Stresses in Concrete and Steel. The stresses at the extreme tension fiber are calculated using a transformed moment of inertia of the concrete and steel reinforcement (Fig. 12).

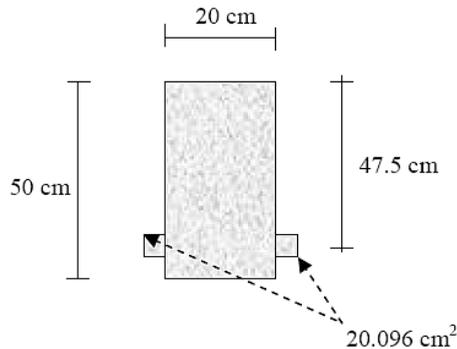


Fig. 12. Transformed cross section.

$$2\text{Ø}16 \rightarrow \rho = 0.0401 A_S = 4.0192 \text{ cm}^2, \quad n = 10.$$

Transformed Area of Steel.

$$(A_S)_t = 10 A_S = 10 \cdot 4.0192 = 40.192 \text{ cm}^2,$$

20.096 cm² distributed at each side of the concrete cross section.

Distance from the Top Fiber to the Neutral Axis of the Transformed Moment of Inertia:

$$\bar{y} = \frac{A_1 y_1 + A_2 y_2}{A_1 + A_2} = \frac{50 \cdot 20 \cdot 25 + 2 \cdot 20 \cdot 0.096 \cdot 47.5}{50 \cdot 20 + 2 \cdot 20 \cdot 0.096} = 25.87 \text{ cm.}$$

The Transformed Moment of Inertia:

$$\begin{aligned} I_{tr} &= [I_G + Bd^2]_{concrete} + [Ad^2]_{steel} = \\ &= [208,333.34 + 50 \cdot 20 \cdot 0.87^2] + [2 \cdot 20 \cdot 0.096 \cdot 21.63^2] = 227,894.4 \text{ cm}^2. \end{aligned}$$

The stress at the extreme tension fiber is then calculated

$$f_{ct} = \frac{My_b}{I_{tr}} = \frac{2982,140 \cdot 24.13}{227,894.4} = 315.8 \text{ N/cm}^2 = 3.15 \text{ N/mm}^2 \text{ (MPa).}$$

The stress in the steel at this point is calculated

$$f_s = \frac{My_b}{I_{tr}} n = \frac{2982,140 \cdot 21.63}{227,894.4} \cdot 10 = 2830.4 \text{ N/cm}^2 = 28.3 \text{ N/mm}^2 \text{ (MPa).}$$

Deflections. The deflection at the centerline of the concrete beam at load of 42,602 N:

$$\Delta_{\max} = \frac{Pl^3}{48EI} = \frac{42,602 \cdot 280^3}{48 \cdot 3 \cdot 10^6 \cdot 221,532.4} = 0.0293 \text{ cm.}$$

Loads at First Cracking. The load at first cracking:

$$f_{ct} = \frac{My_b}{I_{tr}},$$

$$3 \cdot 10^2 \text{ N/cm}^2 = P \frac{280 \cdot 24.13}{4 \cdot 227,894.4},$$

$$P = 40,476 \text{ N.}$$

Резюме

Розроблено тривимірну нелінійну скінченноелементну модель залізобетонної балки. Для числового моделювання використано стандартний скінченноелементний пакет програм ANSYS 8.0. Із використанням об'ємних скінченних елементів типу SOLID65 руйнування бетону при стиску моделювалося за допомогою алгоритму пластичності, в той час як його розтріскування в зоні розтягу описувалося нелінійною моделлю матеріалу. Урахування сталюого

армування балки проводилося для різних об'ємних долей сталі в бетоні. Для бетонної балки без армування результати розрахунків за допомогою програми ANSYS порівнювалися з аналітичними розрахунками. При цьому отримано їх хорошу збіжність. Показано використання моделі для оцінки критичних зон розтріскування, завантажень і переміщень у залізобетонних балках за різного роду навантажень.

1. M. Y. H. Bangash, *Concrete and Concrete Structures: Numerical Modeling and Applications*, Elsevier Science Publishers Ltd., London (1989).
2. Y. Hemmaty, "Modeling of the shear force transferred between cracks in reinforced and fibre reinforced concrete structures," in: Proc. of the ANSYS Conf., Vol. 1, Pittsburgh, PA (1998).
3. *ANSYS 8.0 Manual Set*, ANSYS Inc., Canonsburg, PA (1998).
4. *ANSYS Theory Reference*, Seventh Edition, Swanson Analysis Systems (1998).
5. *ANSYS – Engineering Analysis System. Theoretical Manual* (for ANSYS Revision 8.04), Swanson Analysis Systems (1998).
6. D. Kachlakev and T. Miller, *FE Modeling of Reinforced Concrete Structures Strengthened with FRP Lamiates*, Final Report SPR 316, Oregon State University (2001).
7. K. J. Willam, University of Colorado (Private communication) (1982).
8. K. J. William and E. D. Warnke, "Constitutive model for the triaxial behavior of concrete," in: Proc. of the Int. Association for Bridge and Structural Engineering, ISMES, Bergamo (1975), Vol. 19, p. 174.
9. K. J. Bathe, *Finite Element Procedures*, Prentice-Hall Inc., New Jersey (1996).
10. W. F. Chen and D. J. Han, *Plasticity for Structural Engineers*, Springer-Verlag, New York (1988).

Received 12. 09. 2007