

Numerical Modeling of Classic Concrete Beam Tests

Modelagem Numérica de Ensaios Clássicos de Vigas de Concreto Armado



A. L. GAMINO ^a
andre.gamino@uol.com.br

T. N. BITTENCOURT ^b
tulio.bittencourt@poli.usp.br

J. L. A. O. SOUSA ^c
jls@fec.unicamp.br

Abstract

The classic series of tests on reinforced concrete beams conducted by Bresler and Scordelis 40 years ago is often used to calibrate finite element analysis models. This paper describes the modeling of these tests using four finite element programs: QUEBRA2D/FEMOOP (development system), FormWorks-VecTor2-Augustus (University of Toronto, Canada), ATENA 2D (Cervenka Consulting, Czech Republic) and DIANA (TNO Building and Construction Research, The Netherlands). In the development system, appropriate finite elements and constitutive models have been implemented for the representation of steel and concrete. The performed analyses assumed perfect bond between concrete and reinforcement. However, a tension-stiffening model was considered after concrete started cracking. An anchorage system used in the experimental tests contributed to the adequacy of this assumption. Results demonstrate the efficiency of the implementations.

Keywords: beams; reinforced concrete; constitutive models; finite element method.

Resumo

A série de ensaios clássicos em vigas de concreto armado conduzida por Bresler e Scordelis há 40 anos é comumente utilizada para a calibração de modelos em análise de elementos finitos. Este artigo descreve modelagem desses ensaios com a utilização de quatro programas de análise bidimensional por elementos finitos: QUEBRA2D/FEMOOP (plataformas de desenvolvimento), FormWorks-VecTor2-Augustus (Universidade de Toronto, Canadá), ATENA 2D (Cervenka Consulting – Praga, República Tcheca), e DIANA (TNO Building and Construction Research, Delft, Holanda). Nas plataformas de desenvolvimento foram implementados modelos constitutivos e elementos finitos capazes de representar os materiais aço e concreto. As análises efetuadas admitiram aderência perfeita entre o concreto e a armadura. Considerou-se, entretanto, um modelo de tension-stiffening a partir da fissuração do concreto. A utilização de sistemas de ancoragem nas armaduras passivas levou a uma condição de aderência perfeita nas extremidades das vigas, contribuindo para a adequabilidade dessa hipótese. Os resultados obtidos demonstram a eficiência das implementações efetuadas.

Palavras-chave: vigas; concreto armado; modelos constitutivos; método dos elementos finitos.

^a Universidade de São Paulo, Escola Politécnica, Departamento de Engenharia de Estruturas e Fundações, andre.gamino@uol.com.br, Av. Prof. Almeida Prado, 271, CEP 05508-900, São Paulo-SP, Brasil.

^b Universidade de São Paulo, Escola Politécnica, Departamento de Engenharia de Estruturas e Fundações, tulio.bittencourt@poli.usp.br, Av. Prof. Almeida Prado, 271, CEP 05508-900, São Paulo-SP, Brasil.

^c Universidade Estadual de Campinas, Faculdade de Engenharia Civil, Arquitetura e Urbanismo, Departamento de Estruturas, jls@fec.unicamp.br, Avenida Albert Einstein, 951, CEP 13083-852, Campinas-SP, Brasil.

1 Introduction

In the evolution of the nonlinear analysis of reinforced concrete structures using the Finite Element Method (FEM), the Scordelis' pioneer work in the 1960's defined concepts and criteria to be followed by the research community in this area. From the many contributions of Prof. Scordelis, this paper addresses the classic laboratory tests of twelve reinforced concrete beams (Bresler; Scordelis [1]) developed with the objective of investigating the critical shear behavior of the beams and also producing experimental results aiming at support to numerical developments in finite elements.

Another significant contribution for the crack modeling was accomplished by Ngo; Scordelis [2]. Although they developed a simple model, it represented a step ahead in the field of computational development for the simulation of cracks in concrete structures.

These beams were considered as a classic test series by the scientific community. Since then, these results were used intensively as reference data for calibration and verification of numerical models for reinforced concrete structures using the FEM. Recently, in 2004 (Vecchio; Shim [3]), another experimental program developed in the University of Toronto, in Canada, reproduced the classic tests of RC beams, tracking the post-peak behavior by means of force-displacement curves.

This paper describes the numerical modeling of those beams using the programs QUEBRA2D and FEMOOP,

developed by the Structural Concrete Modeling Group (GMEC), which involves researchers from the Polytechnic School of the University of São Paulo (EPUSP), group leader, as well as researchers from the Pontifical Catholic University of Rio de Janeiro (PUC-RIO) and the State University of Campinas (UNICAMP). The modeling was developed with the purpose of detecting the capabilities of these programs to simulate the behavior of reinforced concrete structures. Simultaneously, these results were compared with those generated by the programs developed by Prof. Vecchio's group in the University of Toronto, and with the behavior of the beam OA3 in the reference (Vecchio; Shim [3]). These results were also confronted with those obtained with the programs ATENA 2D and DIANA, developed, respectively, by Cervenka Consulting, in the Czech Republic and TNO Building and Construction Research, in The Netherlands.

2 Programs FEMOOP and QUEBRA2D

The program QUEBRA2D (Figure 1) is an interactive graphical simulator of the damage evolution of structural elements, under development as a cooperative project of the Structural Concrete Modeling Group (GMEC), involving the Computational Mechanics Laboratory (LMC) of the Polytechnic School of the University of São Paulo (EPUSP) and TECGRAF of Pontifical Catholic University of Rio de Janeiro (PUC-RIO).

Figure 1 – QUEBRA2D user interface

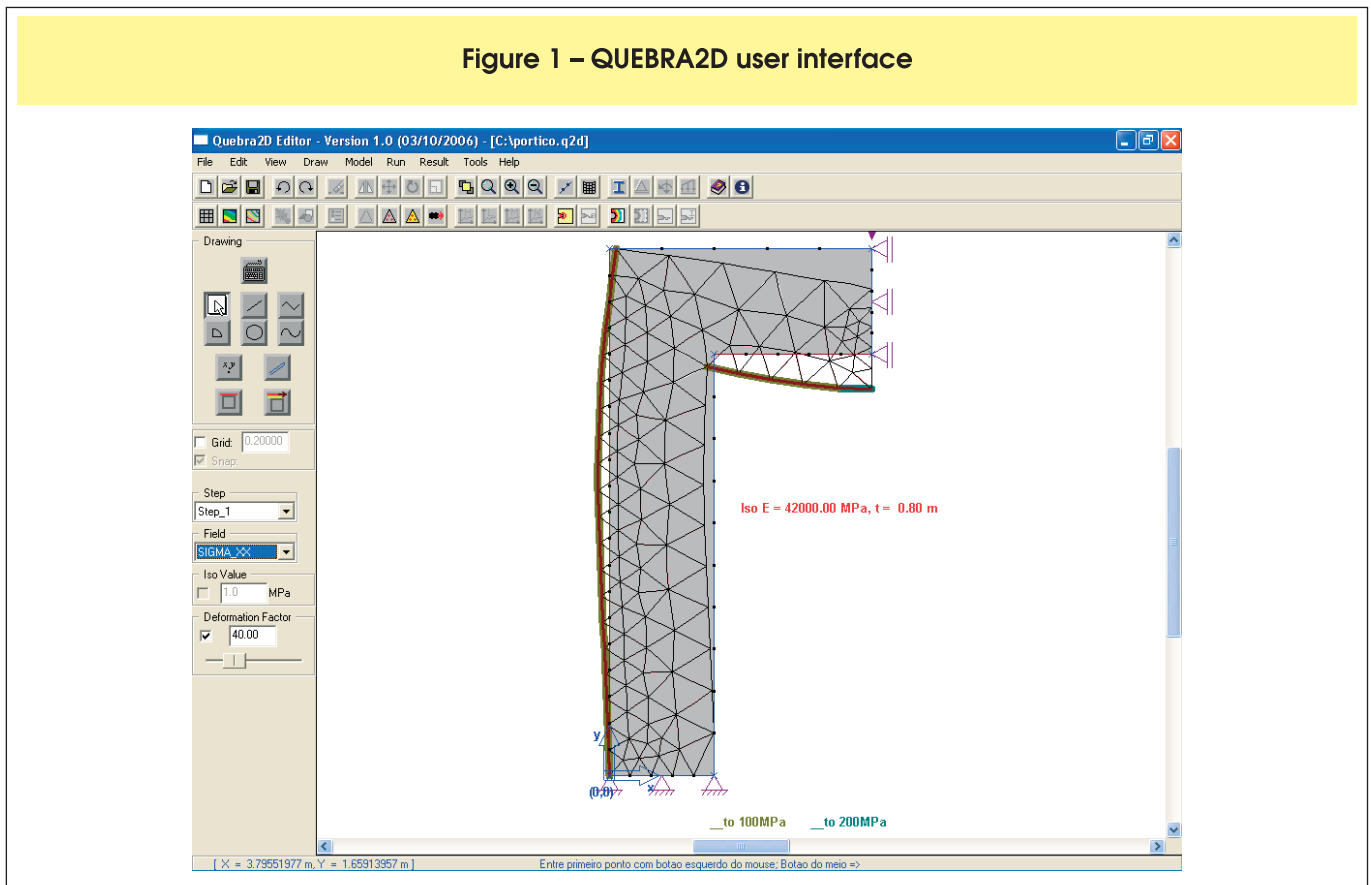
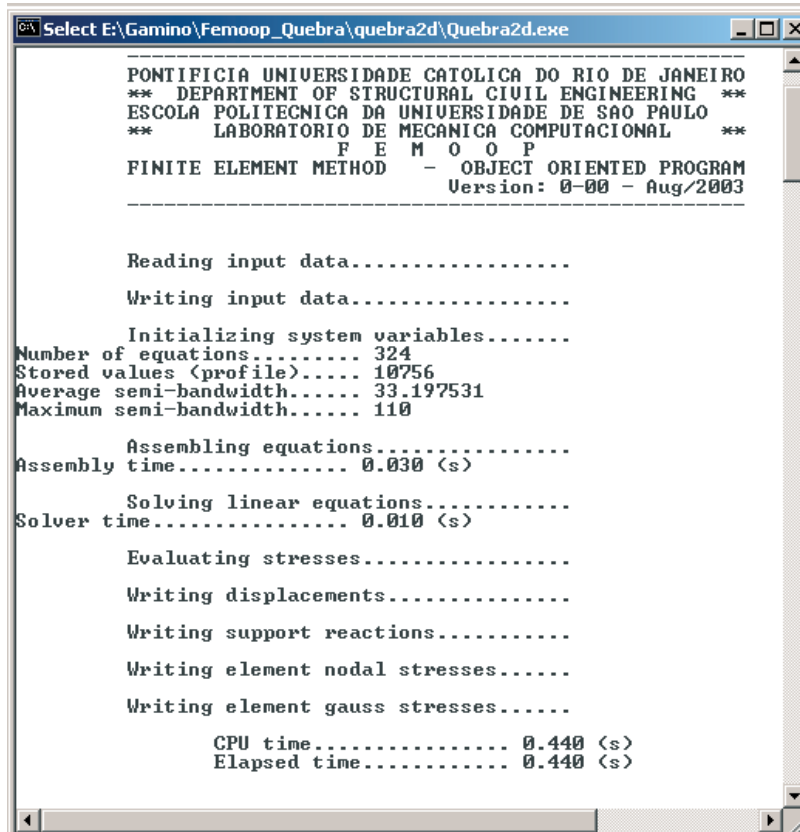


Figure 2 – FEMOOP user interface



```

Select E:\Gamino\Femoop_Quebra\quebra2d\Quebra2d.exe
-----
PONTIFICIA UNIVERSIDADE CATOLICA DO RIO DE JANEIRO
** DEPARTMENT OF STRUCTURAL CIVIL ENGINEERING **
ESCOLA POLITECNICA DA UNIUERSIDADE DE SAO PAULO
** LABORATORIO DE MECANICA COMPUTACIONAL **
      F E M O O P
FINITE ELEMENT METHOD - OBJECT ORIENTED PROGRAM
                        Version: 0-00 - Aug/2003
-----

Reading input data.....
Writing input data.....

Initializing system variables.....
Number of equations..... 324
Stored values (profile).... 10756
Average semi-bandwidth..... 33.197531
Maximum semi-bandwidth..... 110

Assembling equations.....
Assembly time..... 0.030 (s)

Solving linear equations.....
Solver time..... 0.010 (s)

Evaluating stresses.....
Writing displacements.....
Writing support reactions.....
Writing element nodal stresses.....
Writing element gauss stresses.....

CPU time..... 0.440 (s)
Elapsed time..... 0.440 (s)

```

This program acts as input/output manager of the fracture processes data, as well as an adaptive mesh generator. Post-processing is entirely performed in QUEBRA2D. The Finite Element Method formulation is implemented the program FEMOOP (Finite Element Method - Object Oriented Programming), which is used together with QUEBRA2D. FEMOOP (Figure 2) is based on the paradigm of Object Oriented Programming and developed in C++ language. In both computational platforms the necessary routines for the simulation of concrete structures were implemented. QUEBRA2D works as pre and post-processor, and is responsible for introduction and edition of all the pertinent analysis attributes. FEMOOP is the solver module of the system, and is responsible for processing the analysis attributes data file generated by QUEBRA2D, named neutral file, saving the results in a post-processing file, which returns to QUEBRA2D for the visualization of results.

3 Details of Bresler and Scordelis' Beams

The twelve beams tested were divided in four groups of three: the group OA (only bottom longitudinal steel reinforcement, without stirrups or longitudinal top reinforcement) and groups A, B and C with stirrups and longitudi-

nal top steel reinforcement. The beams belonging to each group had different spans and similar height and width dimensions. Table 1 presents the summary of the tested beams data. Table 2 presents the physical and mechanical properties of the materials steel and concrete.

The beams were tested in a three-point load setup. The load steps started in 40 kN and, close of the rupture, reduced to 20 kN. All the beams were tested at the age of 13 days.

4 Finite Element Analysis

4.1 Constitutive Model: Uncracked Concrete

The constitutive model adopted for the two-dimensional analyses and implemented in the programs QUEBRA2D/FEMOOP was Ottosen's model [4]. This is a four-parameter model ("A", "B", "K₁", "K₂") whose functional is represented by:

$$F = A \frac{J_2}{f_{cm}^2} + \lambda \frac{\sqrt{J_2}}{f_{cm}} + B \frac{I_1}{f_{cm}} - 1 = 0 \quad (1)$$

Table 1 – Details of the beams tested by Bresler e Scordelis

Beam	b_w (mm)	h (mm)	d (mm)	L (mm)	Span(mm)	A_s	A'_s	A_{sw}/s
OA1	310	556	461	4100	3660	4 n° 9	---	---
OA2	305	561	466	5010	4570	5 n° 9	---	---
OA3	307	556	462	6840	6400	6 n° 9	---	---
A1	307	561	466	4100	3660	4 n° 9	2 n° 4	n° 2 c/ 210
A2	305	559	464	5010	4570	5 n° 9	2 n° 4	n° 2 c/ 210
A3	307	561	466	6840	6400	6 n° 9	2 n° 4	n° 2 c/ 210
B1	231	556	461	4100	3660	4 n° 9	2 n° 4	n° 2 c/ 190
B2	229	561	466	5010	4570	4 n° 9	2 n° 4	n° 2 c/ 190
B3	229	556	461	6840	6400	5 n° 9	2 n° 4	n° 2 c/ 190
C1	155	559	464	4100	3660	2 n° 9	2 n° 4	n° 2 c/ 210
C2	152	559	464	5010	4570	4 n° 9	2 n° 4	n° 2 c/ 210
C3	155	554	459	6840	6400	4 n° 9	2 n° 4	n° 2 c/ 210

where:

A, B, K₁ e K₂: model parameters, experimentally determined

f_{cm}: average concrete strength,

I₁: first invariant of the stress tensor,

J₂: second invariant of the deviatoric stress tensor and

λ: computed using K₁, K₂ and the invariant cos(3θ).

The failure surface is described by this functional. The parametric calibration was performed based on tests according to the proposals of Ottosen [4], CEB [5] and Dahl [6]. According to tests conducted by Hartl [7] in biaxial failure of concrete, the best results were reached according to the formulation of Dahl [6], whose parameters can be calculated through:

$$\begin{aligned}
 x &= \frac{f_{cm}}{100 [MPa]} \\
 A &= -1.66 x^2 + 3.49 x + 0.73 \\
 B &= -0.19 x^2 + 0.41 x + 3.13 \\
 K_1 &= 0.46 x^2 - 0.97 x + 11.89 \\
 K_2 &= -0.02 x^2 + 0.04 x + 0.974
 \end{aligned} \tag{2}$$

In the T-T (pure tensile) zone, the failure surface Ottosen [4] approaches Rankine's model. The definition of the failure surface is made through associative plasticity, and the energy balance is made according to the return algorithm proposed by Owen; Hinton [8].

In the programs of the University of Toronto (FormWorks-VecTor2-Augustus) the constitutive model of Hognestad (VecTor2 [9]) was used for the compressed concrete (pre-peak), defined by the expression:

$$\sigma = \sigma_c \left[\frac{2\varepsilon}{\varepsilon_c} - \left(\frac{\varepsilon}{\varepsilon_c} \right)^2 \right] \tag{3}$$

where σ_c e ε_c are respectively the tensile stress and strain limits of the concrete.

The model for the concrete in compression in the post-peak region follows:

- The modified Park's model. Further details of this model can be accessed in VecTor2 [9].
- The softening model used for compression is described in Vecchio [10].

A model to evaluate the concrete contribution between cracks in the tensile region was used (Bentz [11]).

In the program ATENA 2D, a biaxial model named SBETA, described in Chen; Saleeb [12] was used, with non-linear behavior in the compression in agreement with the model of Kupfer et al. [13]. This criterion establishes, in the C-C (compression only) region, that:

$$\sigma_c = \frac{1 + 3,65 a}{(1 + a^2)} f_c; \quad a = \frac{\sigma_{c1}}{\sigma_{c2}} \tag{4}$$

where f_c is the concrete strength, σ_{c1} e σ_{c2} are the principal stresses in the concrete and σ_c is the stress in the structural member.

In DIANA, combined plasticity models are used when the simulated structure presents tensile and compressive zones simultaneously. In this case, the model for the tensile concrete follows Rankine's proposal. For the concrete

under compression, Drucker-Prager or Mohr-Coulomb can be used.

4.2 Constitutive Model: Cracked Concrete

Coupled with the model of physical integrity, a model of rotational smeared crack was implemented. The cracking criterion is related to the tensile stresses (from Ottosen's model [4]) in the Gauss points in comparison with the tensile strength of the material. After the first crack, a linear softening model was used. The same was used in the modeling with ATENA 2D.

In the system (FormWorks-VecTor2-Augustus), the model described in Vecchio [10] for softening in compression, and linear softening model in tension were used. Cracks were modeled with a smeared rotating crack model.

In DIANA the tensile post-peak behavior was modeled with a linear softening model, and the cracking with a smeared rotating crack model. However, for hardening in compression, Thorenfeldt's model was used (Diana User's Manual [14]). Using a linear softening model in tension, the crack opening can be obtained through:

$$w^{cr} = \frac{2G_f}{f_t}; \text{ ou } w^{cr} = \varepsilon^{cr} \cdot h^{cr} \quad (5)$$

where w^{cr} is the crack width, G_f the fracture energy in mode I, ε^{cr} the crack strain and h^{cr} the crack band width.

The crack opening displacements were computed from the crack strains, which depend on the failure models adopted for the concrete and the steel reinforcement. The reinforcement ratios used influences the stress level in the tensile portion of the structure, and thus the crack opening displacements. Therefore, the higher the reinforcement ratio, the smaller the crack opening displacements. Crack opening displacements depend also on the tension-stiffening model adopted.

4.3 Constitutive Model: Steel Reinforcements

For the reinforcements, von Mises plasticity model was used. In this case the envelope that defines yielding surface is:

$$F = \lambda \frac{\sqrt{J_2}}{f_y} - 1 = 0; \lambda = \text{constant} \quad (6)$$

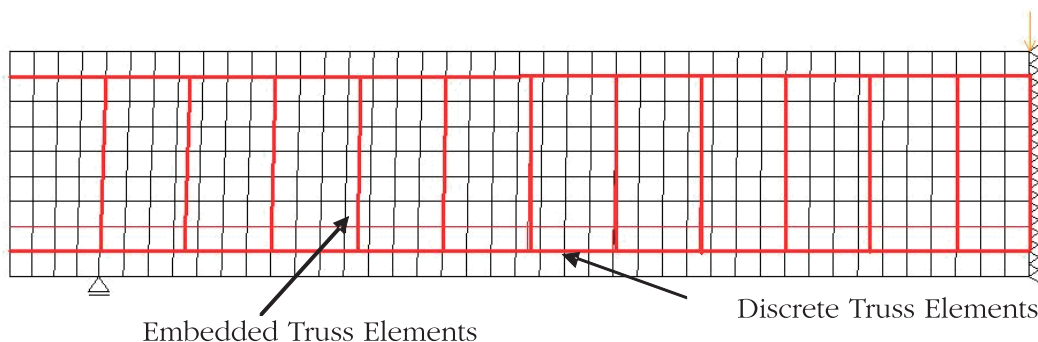
where f_y is the yield stress.

For a uniaxial condition, an isotropic model, comparing axial stresses directly to the yield stress limit of the material, would be sufficient. A linear hardening model, which accounts for the yielding and the rupture limits of the steel reinforcements, can also be used. The typical mesh used is illustrated in Figure 3. In this mesh, part of the reinforcement was inserted as discrete bars and part was incorporated in the mesh. Quadratic isoparametric Q8 finite ele-

Table 2 – Materials properties of the beams tested by Bresler e Scordelis

Steel						
Bar	Diameter (mm)	Area (mm ²)	f_y (MPa)	f_u (MPa)	E_s (MPa)	
n° 9	28.7	645	555	933	218000	
n° 4	12.7	127	345	542	201000	
n° 2	6.4	32.2	325	430	190000	
Concrete						
Beam	f_c (MPa)		f_t (MPa)			
OA1	22.6		3.97			
OA2	23.7		4.34			
OA3	37.6		4.14			
A1	24.1		3.86			
A2	24.3		3.73			
A3	35.1		4.34			
B1	24.8		3.99			
B2	23.2		3.76			
B3	38.8		4.22			
C1	29.6		4.22			
C2	23.8		3.93			
C3	35.1		3.86			

Figure 3 – Typical mesh for numerical modeling



ments were used for the concrete and linear isoparametric truss elements for the reinforcements. All analyses were performed under plane stress assumption.

5 Important Aspects for a FEM Program Validation

In general, load-displacement curves are used by researchers for validating programs or implementations in finite element analyses concerning the ability to predict the behavior of concrete structures. However, in addition to the global behavior, the behavior of each material employed in the structure should be evaluated. Thus, in addition to the load-displacement curves, other

criteria should be used, such as:

- Obtained crack width and cracking pattern, compared with experimental response;
- Stress or strain in the reinforcement bars, and their evolution along the loading process;
- Strain in the concrete;

Therefore, validation requires that the numerical result match reasonably well the experimental results according to the aforementioned criteria.

6 Results

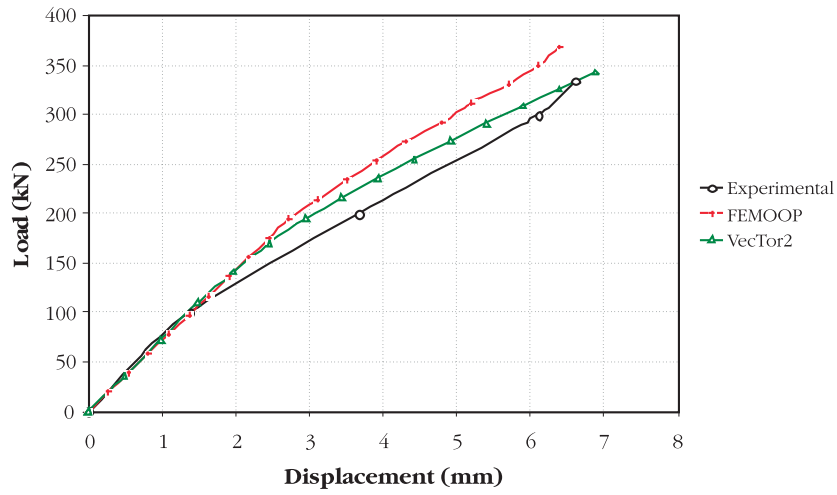
Table 3 illustrates the obtained results in terms of load and displacements at failure predicted by the four programs

Table 3 – Results obtained for the beams tested by Bresler e Scordelis

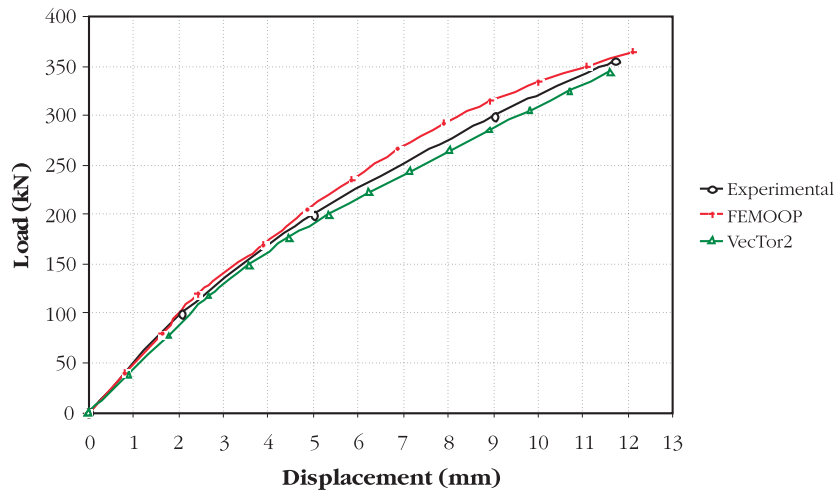
Beam	P _u (kN)			Δ(%)		δ _u (mm)			Δ(%)	
	Exp.	VecTor2	FEMOOP	VecTor2	FEMOOP	Exp.	VecTor2	FEMOOP	VecTor2	FEMOOP
OA1	334	343	368	2.69	9.24	6.6	6.9	6.4	4.55	-7.25
OA2	356	344	365	-3.37	2.47	11.7	11.6	12.1	-0.85	4.31
OA3	378	359	380	-5.03	0.53	27.9	29.6	27	6.09	-8.78
A1	468	533	439	13.89	-6.61	14.2	14.8	7.56	4.23	-48.92
A2	490	529	509	7.96	3.73	22.9	21.4	19.2	-6.55	-10.28
A3	468	430	456	-8.12	-2.63	35.8	35.9	34.2	0.28	-4.74
B1	446	506	415	13.45	-7.47	13.7	14.7	10.2	7.30	-30.61
B2	400	346	406	-13.50	1.48	20.8	20.5	22.5	-1.44	9.76
B3	356	358	360	0.56	1.11	35.3	35.9	38.2	1.70	6.41
C1	312	332	328	6.41	4.88	17.8	17.8	14.8	0.00	-16.85
C2	324	287	314	-11.42	-3.18	20.1	19.6	21.6	-2.49	10.20
C3	270	256	222	-5.19	-21.62	36.8	37.4	31.9	1.63	-14.71

Observations: the numerical results of the ultimate load and ultimate displacement obtained for the beam OA3 with the programs ATENA 2D and DIANA were respectively: 400 kN (5.82%) and 30.2 mm (8.24%), 402 kN (6.35%) and 29.9 mm (7.17%); the percentile variation of the results was obtained using: (%) = {((numerical result) - (experimental result)) / (experimental result)}.100%.

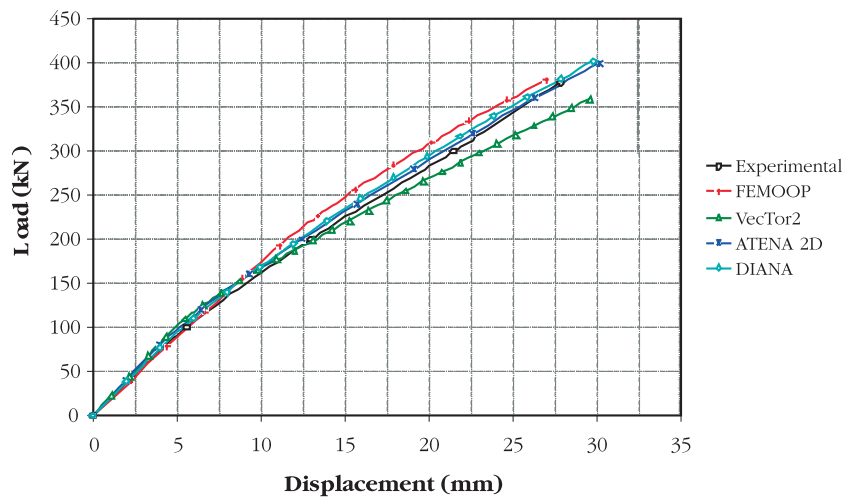
Figure 4 – Results obtained for series OA



a) Beam OA1

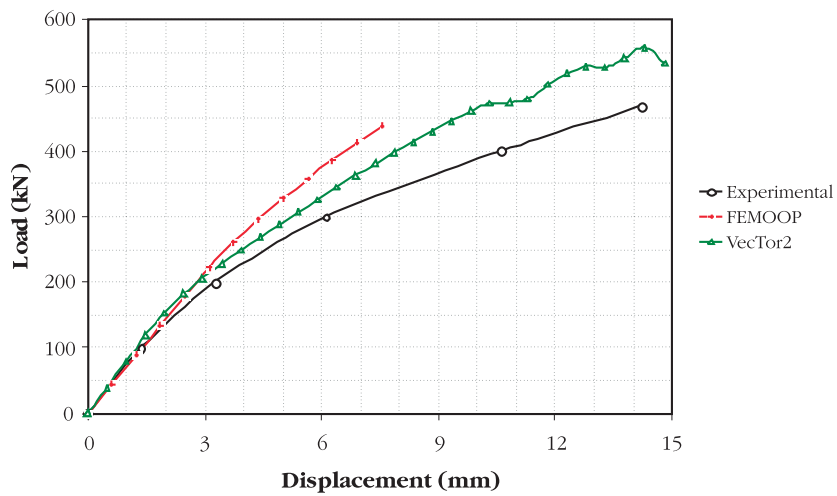


b) Beam OA2

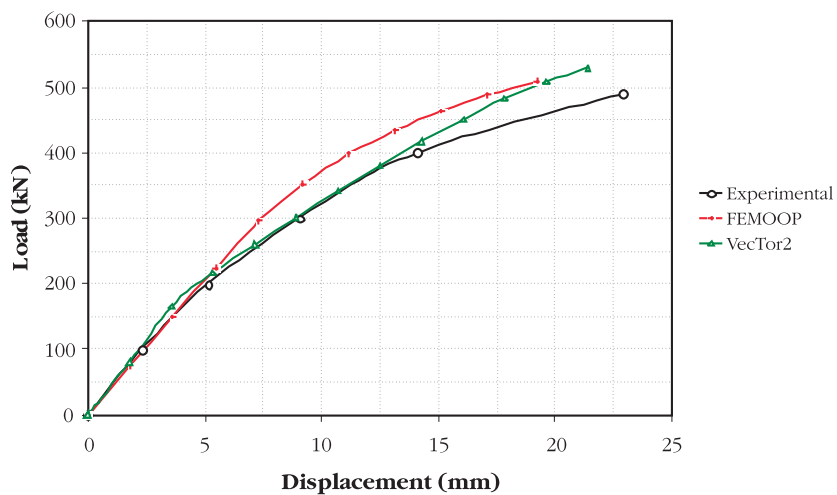


c) Beam OA3

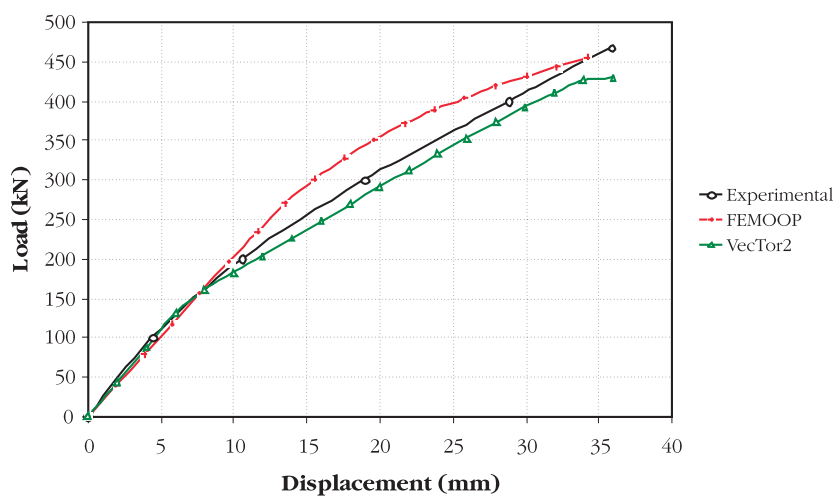
Figure 5 – Results obtained for series OA



a) Beam A1

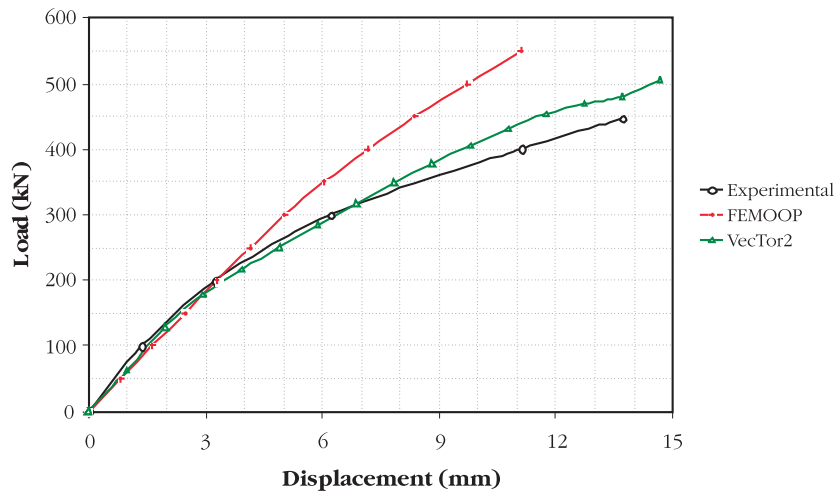


b) Beam A2

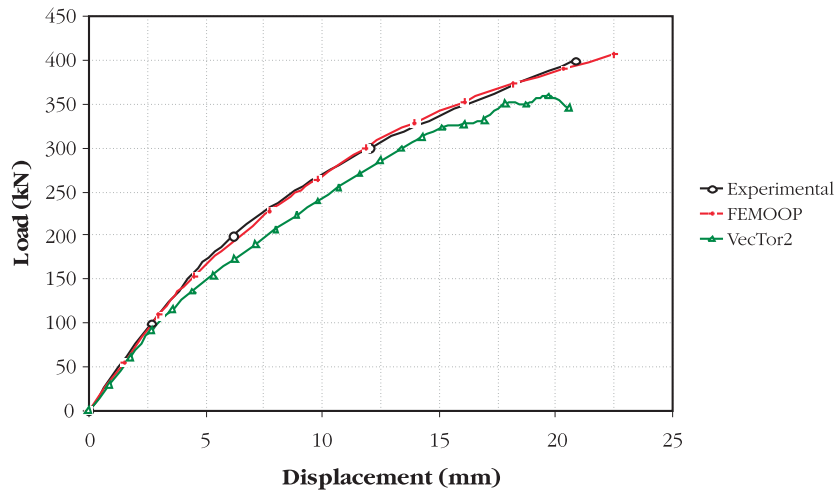


c) Beam A3

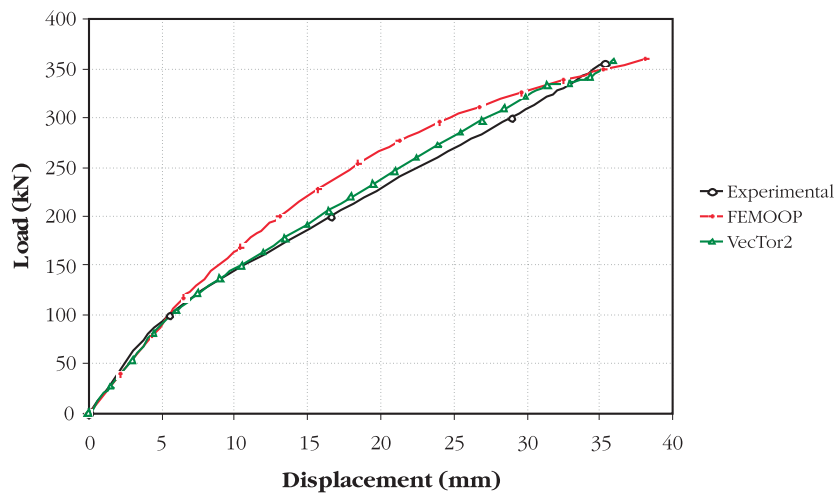
Figure 6. Results obtained for series B



a) Beam B1

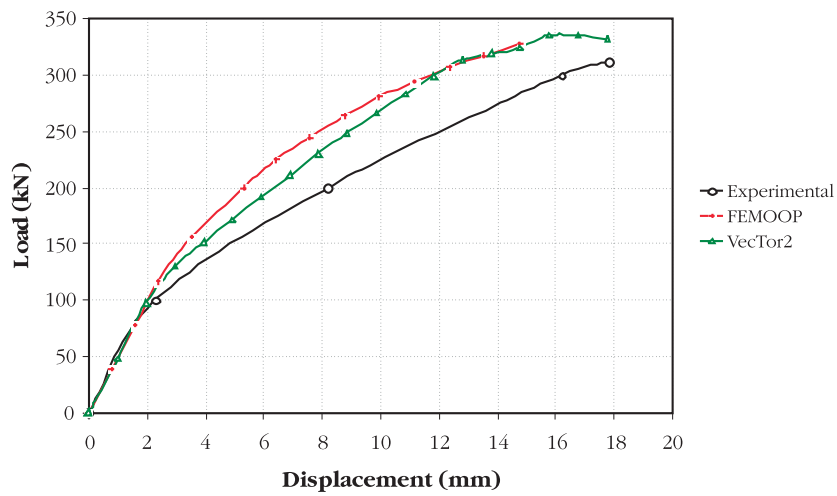


b) Beam B2

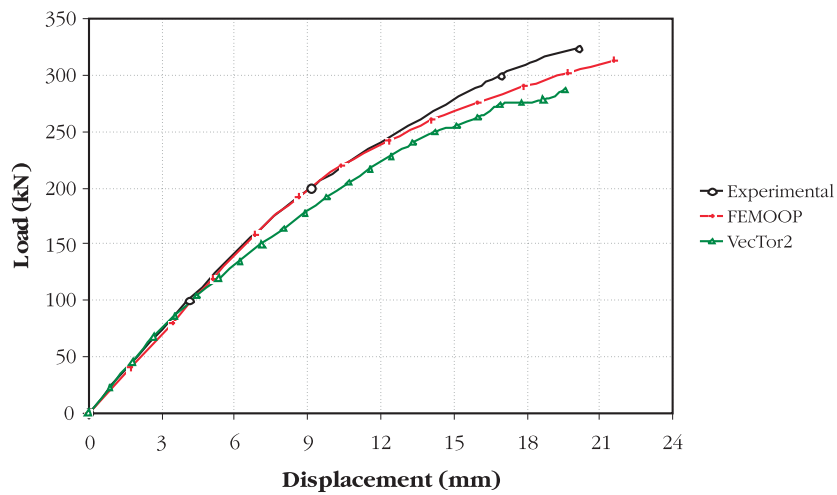


c) Beam B3

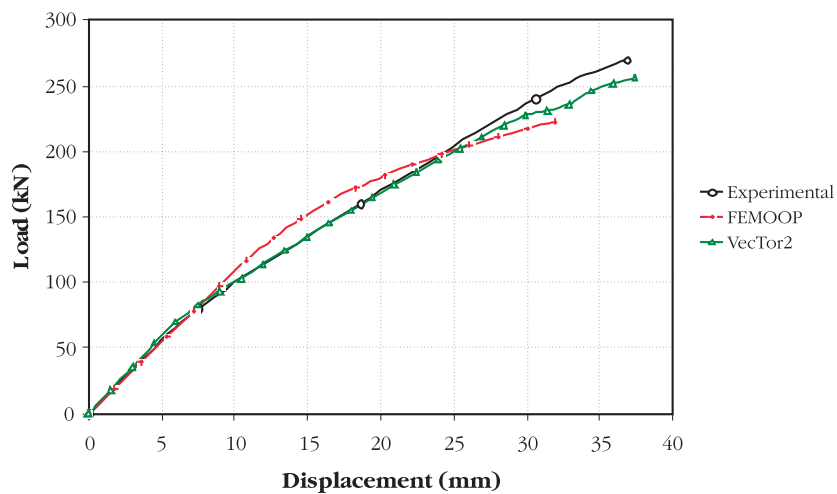
Figure 7 – Results obtained for series C



a) Beam C1



b) Beam C2



c) Beam C3

used in the analyses. The obtained load-displacement curves are illustrated in Figures 4 through 7.

Considering overall results, good correlation was obtained between the numerical response and the experimental results of Bresler and Scordelis. Displacements calculated with FEMOOP were particularly close to those from the experimental tests, especially regarding the beams without steel stirrups (OA1, OA2 and OA3), and also for the beams A3, B1, B2 and C2. In terms of failure load, the numerical values from FEMOOP presented good correlation to the experimental tests for all the beams, except for the beam C3.

The program VecTor2 produced results which are more coherent with those of Bresler and Scordelis in terms of displacements than in terms of failure load. The use of different constitutive models led to varied results predicted by FEMOOP and VecTor2. For the beam C1, the displacements obtained by the numerical simulation were the same as observed in the experimental tests.

It is important to point out that, for Bresler and Scordelis' beams, with large bending reinforcement ratios and large bar sizes, the use of a linear softening model with constant parameters has a significant influence in the results. Usually, parameters for these linear tension-stiffening models are established for beams with moderate or low bending reinforcement ratios, with bars sizes smaller than those used in the tests.

With the purpose of investigating other numerical responses produced by the programs, crack widths (Figure 8), and tensile stress in bending reinforcements (Figure 9) with the strain evolution (Figure 10) for the beam OA3, are visualized. The other two programs, ATENA 2D and DIANA, were also used. For this beam, all the numerical responses in terms of load-displacement curves were close to the experimental observations; particularly, the responses obtained by ATENA 2D and DIANA were practically the same. The numerical crack patterns that best approached the experimental tests (Figure 8) were generated by FEMOOP and ATENA 2D.

In Figure 8, the numerically obtained crack pattern shows cracks closely spaced, while the experimental crack pattern shows fewer cracks but more spaced. The beams of Bresler and Scordelis presented a large bending reinforcement ratio and were reinforced with large bar sizes. This makes difficult to approach the crack pattern with a linear tension-stiffening model with constant parameters. This model would have to take into account bar sizes and reinforcement ratio to simulate appropriately the contribution of the concrete between cracks. Therefore, the divergence among crack patterns lead to load-displacement curves which are more rigid than the corresponding experimental curves.

In FEMOOP a maximum crack width 0.29 mm was obtained, while 0.30 mm was obtained by VecTor2, 0.28 mm by DIANA and 0.51 mm by ATENA 2D. Shear stresses obtained for the concrete in the region of the supports were respectively 3.10 MPa and 3.21 MPa for FEMOOP and VecTor2. For ATENA 2D and DIANA these values were,

respectively, 2.68 MPa and 3.29 MPa. The maximum numerically computed strains in the bending reinforcement bars were 1.86‰ and 1.91‰ for FEMOOP and VecTor2, respectively. For ATENA 2D and DIANA, the strains in the reinforcement bars were, respectively, 2.0‰ and 1.94‰. This shows the tendency of proximity of responses in the computational analyses using the four platforms.

Table 4 presents the numerical results of crack opening displacements compared to the experimental values by Vecchio; Shim [3] (because Bresler and Scordelis' tests present only load-displacement information). The comparison between crack opening displacements would make sense for service load, but were available only for the failure load. Such beams presented characteristics similar to the beams of Bresler and Scordelis but with different bar sizes (however, with same bending reinforcement ratio). This inhibited the comparison in terms of stress/strain in the reinforcement bars. The concrete and other characteristics of the beams were similar.

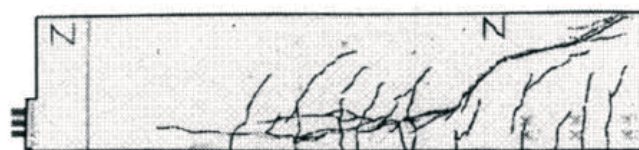
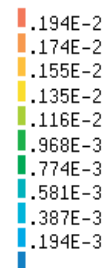
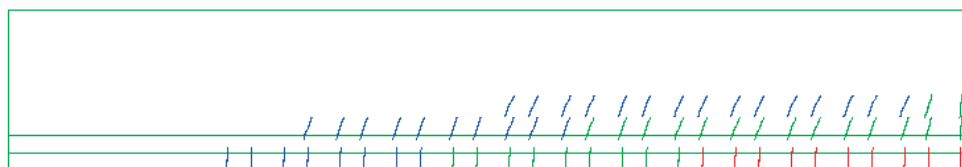
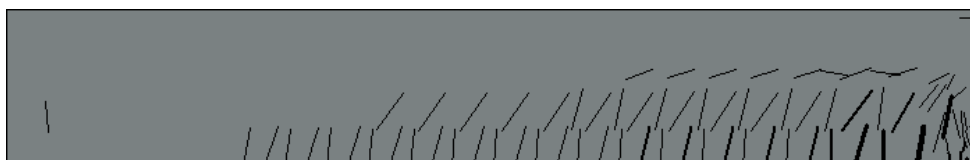
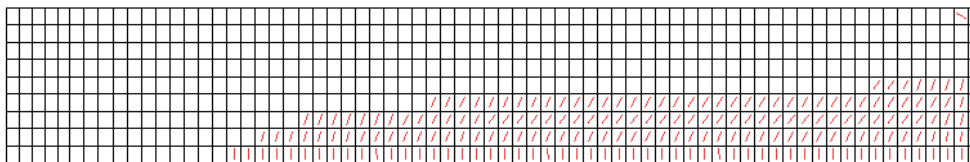
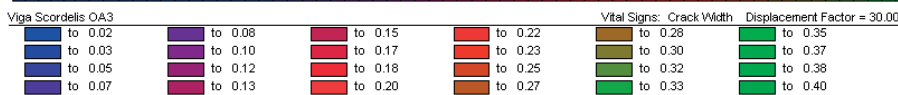
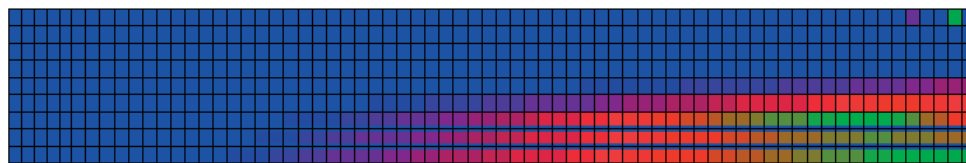
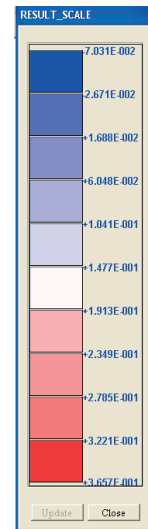
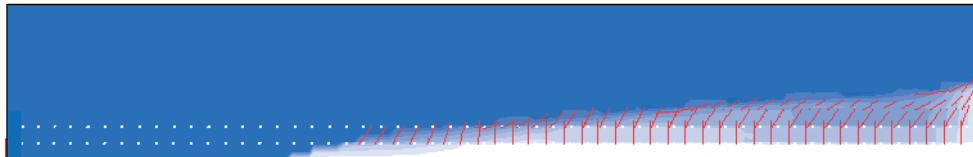
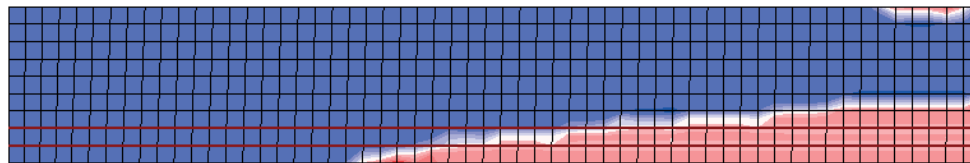
In general, results were satisfactory. However, the programs were not capable to predict the larger crack opening displacements of the beams A2, A3, B2 and B3.

7 Conclusions

This work attempted to reproduce numerically classic tests of reinforced concrete beams in order to validate a finite element program named FEMOOP, in continuous development by the research group. The responses produced by FEMOOP were compared with VecTor2, developed in the University of Toronto, with ATENA 2D, developed by Cervenka Consulting, and DIANA, developed by the TNO Building and Construction Research. The following conclusions can be drawn:

- a) In a general way, good correlation was obtained between the numerical response and the experimental observations of Bresler and Scordelis;
- b) The displacements computed with FEMOOP were very close to the experimental observation, especially for the beams without stirrups (OA1, OA2 and OA3), and for the beams A3, B1, B2 and C2.
- c) In terms of failure load, the numerical results from FEMOOP had good correlation to the experimental observations for all the beams, except for the beam C3.
- d) The program VecTor2 produced responses which were more coherent with the results of Bresler and Scordelis in terms of displacements than in terms of failure load. For the beam C1 the displacements obtained by the numerical simulation were the same as those observed in the experimental tests.
- e) In the OA3 beam all the numerical responses in terms of load-displacements were close to the experimental observations; particularly the responses obtained by ATENA 2D and DIANA, which were practically the same.
- f) In a general manner the numerical results, in terms of crack opening displacements for all the beams, were reasonable. However, the programs were not capable of predicting the larger crack opening displacements observed in the beams A2, A3, B2 and B3.

Figura 8 – Crack patterns obtained for beam OA3 tested by Bresler and Scordelis

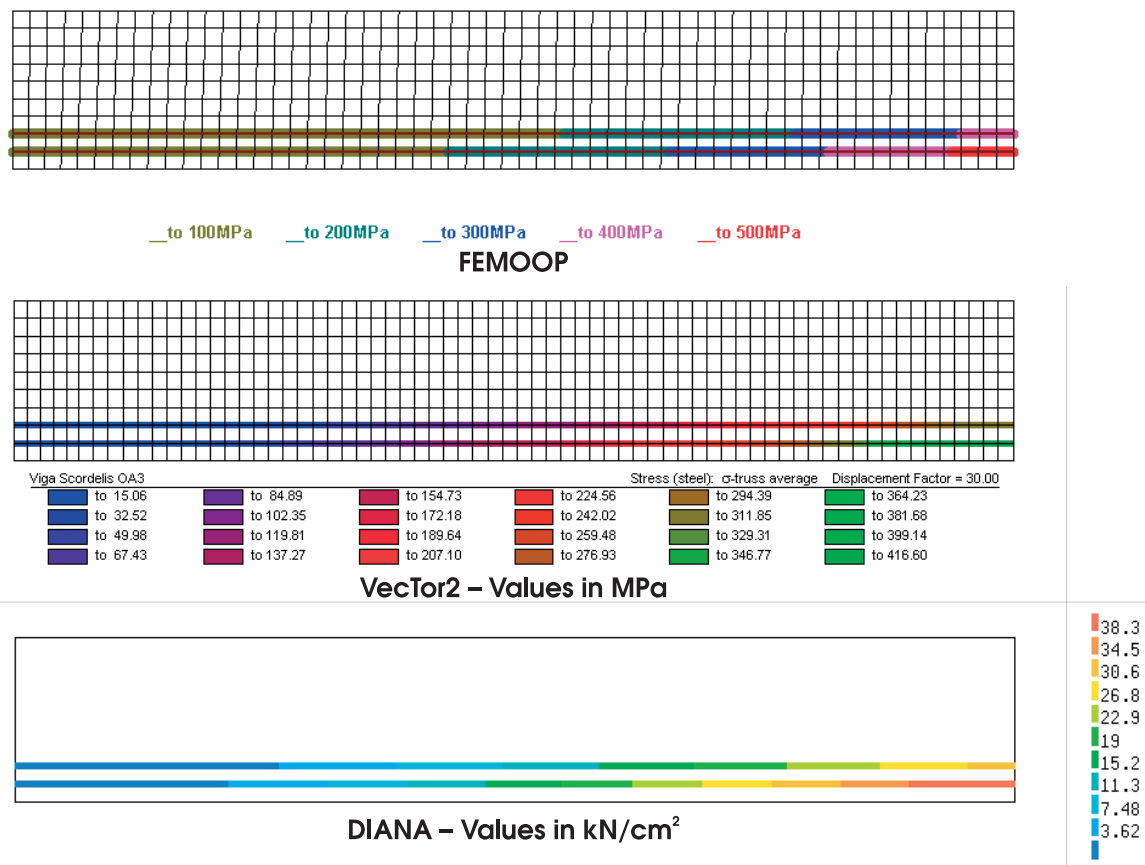


g) The obtained results provide strong indication that the program FEMOOP is capable of representing with sufficient realism the behavior of concrete structures using the implementations described in this work.

8 References

- [01] Bresler, B., Scordelis, A.C., Shear strength of reinforced concrete beams. Journal of American Concrete Institute, v. 60, n. 1, pp.51-74, 1963.
- [02] Ngo, D., Scordelis, A.C., Finite element analysis of reinforced concrete beams. ACI Journal, v.64, n.3, pp.152-163, 1967.
- [03] Vecchio, F.J., Shim, W., Experimental and Analytical Re-examination of Classic Concrete Beam Tests. ASCE Journal of Structural Engineering, v.130, n.3, pp.460-469, 2004.
- [04] Ottosen, N.S., A failure criterion for concrete. Journal of the Engineering Mechanics Division, v.103, n.4, pp.527-535, 1977.
- [05] Comité Euro – International du Béton, Model Code for Concrete Structures. CEB – FIP MC 90, 437p., 1993.
- [06] Dahl, K.K.B., A failure criterion for normal and high strength concrete. Technical University of Denmark, Lyngby, 1992.
- [07] Hartl, H., Development of a continuum-mechanics-based for 3D finite element analysis of reinforced concrete structures and application to problems of soil-structure interaction. Tese de Doutorado, Technische Universität Graz, Áustria, 250p, 2002.
- [08] Owen, D.R.J., Hinton, E., Finite Elements in Plasticity – Theory and Practice. Pineridge Press, 1980.
- [09] VecTor2 & FormWorks, User’s manual, August, 232p., 2002.
- [10] Vecchio, F.J., Finite element modeling of concrete expansion and confinement. ASCE Journal of Structural Engineering, v.18, n.9, pp.2390-2406, 1992.
- [11] Bentz, E.C., Sectional analysis of reinforced concrete members. Tese de Doutorado,

Figure 9 – Axial stresses distribution in bending steel reinforcements obtained for beam OA3 tested by Bresler and Scordelis



Departamento de Engenharia Civil,
Universidade de Toronto, 310p., 2000.

[12] Chen, W.F., Saleeb, A.F., Constitutive equations for engineering materials. John Willey & Sons, 1982.

[13] Kupfer, H., Hilsdorf, H.K., Rüşh, H., Behavior of concrete under biaxial stress. ACI Journal, v.66, n.8, pp.656-666, 1969.

[14] Diana User's Manual – Material Library, TNO Building and Construction Research, 454p., 2002.

Figure 10 – Axial strain evolution for the flexure steel reinforcements obtained for beam OA3 tested by Bresler and Scordelis

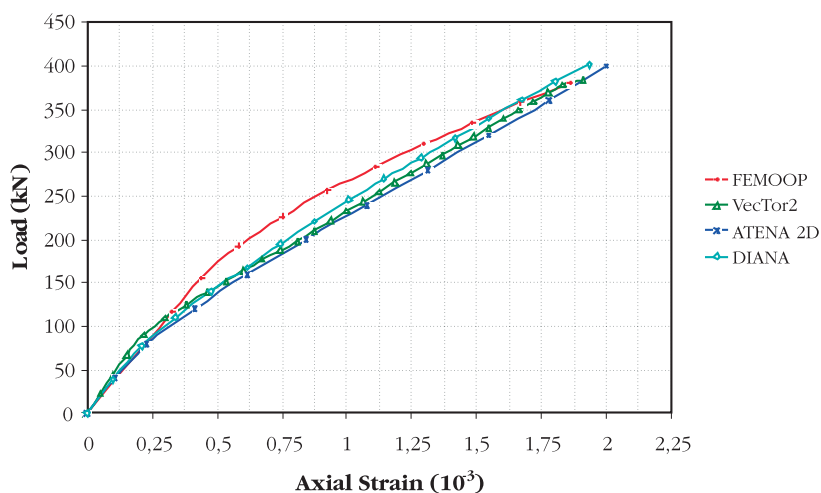


Table 4 – Results obtained for the crack opening displacements in the tested beams by Vecchio; Shim (2)

Beam	Experimental	w^{cr} (mm)		Δ (%)	
		VecTor2	FEMOOP	VecTor2	FEMOOP
OA1	0.40	0.30	0.29	-25.00	-27.50
OA2	0.30	0.25	0.22	-16.67	-26.70
OA3	0.35	0.27	0.58	-22.86	65.71
A1	0.50	0.51	0.57	2.00	14.00
A2	1.40	0.79	0.59	-43.57	-57.86
A3	1.60	0.49	0.46	-69.38	-71.30
B1	0.75	0.88	0.61	17.33	-18.67
B2	1.60	0.83	0.69	-48.13	-56.87
B3	1.60	0.58	0.71	-63.75	-55.63
C1	2.50	3.11	2.71	24.40	8.40
C2	0.40	0.70	0.52	75.00	30.00
C3	0.90	0.37	0.59	-58.89	-34.44