Alexandre Joseph Tokka

Response of reinforced concrete beams: numerical simulation and code prediction

São Paulo

2018

Alexandre Joseph Tokka

Response of reinforced concrete beams: numerical simulation and code prediction

Monograph presented to the Polytechnic School at the University of São Paulo as a requirement for obtaining the Bachelor Degree in Civil Engineering

São Paulo

2018

Catalogação-na-publicação

Tokka, Alexandre Joseph
Response of reinforced concrete beams: numerical simulation and code prediction / A. J. Tokka -- São Paulo, 2018.
59 p.

Trabalho de Formatura - Escola Politécnica da Universidade de São Paulo. Departamento de Engenharia de Estruturas e Geotécnica.

1.Vigas de concreto armado 2.Modelo computacional 3.Fissuras 4.Elementos finitos 5.interação concreto-aço I.Universidade de São Paulo. Escola Politécnica. Departamento de Engenharia de Estruturas e Geotécnica II.t.

Alexandre Joseph Tokka

Response of reinforced concrete beams: numerical simulation and code prediction

Monograph presented to the Polytechnic School at the University of São Paulo as a requirement for obtaining the Bachelor Degree in Civil Engineering

Supervisor: Prof. D.Sc. Luís A. G. Bitencourt Jr.

São Paulo

2018

Acknowledgment

A special gratitude goes out to Prof. Luís Bitencourt for supervising and highly contributing to broaden my knowledge in concrete and numerical modeling. I am also grateful to Engª Yasmin T. Trindade to for her patience and her dedicated time to explain many concepts and how to calibrate the numerical models.

Contents

Abstract

This work presents and compares different methodologies to predict the behavior of reinforced concrete beams. Numerical, analytical and experimental results are compared to evaluate the accuracy of the model. The modeling procedure relies on: (i) a damage model to describe the concrete behavior, (ii) coupling finite elements to simulate the interaction between concrete and steel reinforcement, (iii) the concrete-steel interaction law from the fib Model Code and (iv) a predictive formula for crack width.

Keywords: Reinforced concrete beam, numerical modeling, crack width, bond-slip, coupling finite elements, damage models.

Resumo

Este trabalho apresenta e compara diferentes metodologias para prever o comportamento de vigas de concreto armado. Resultados numéricos, analíticos e experimentais são comparados para avaliar a correlação entre eles. O procedimento de molagem foi baseado em: (i) modelo de dano para descrever o comportamento do concreto, (ii) acoplamento concreto-aço para simular a interação entre os dois materias através do uso de elemenots finitos de acoplamento, (iii) a interação aço-concreto descrita pela lei proposta pelo fib Model Code e, por fim, (iv) uma fórmula para estimar o valor das aberturas de fissuras.

Palavras-Chave: Vigas de concreto armado, modelagem numérica, abertura de fissuras, lei de aderência, elementos finitos de acoplamento, modelos de dano.

List of Figures

List of Tables

1 Introduction

Modeling the nonlinear behavior of reinforced concrete beams with accuracy requires appropriate models to describe the behavior of concrete, rebars and concrete-rebars interaction. The research published by Ngo and Scordelis (1967) was the first work published in literature about numerical modeling of reinforced concrete beams by finite element method. For modeling the concrete is necessary to use a model that presents distinct behavior under tension and compression. In the last years, continuum damage models have been used for many researchers due to the simplicity of these models. Rebars (steel reinforcing bars) are usually represented by truss finite elements such behavior is described by one-dimensional elastoplastic model. Finally, the modeling of the concrete-rebars interaction is very important since the crack width and spacing are directly influenced by the type of adherence chosen. As examples of strategies adopted for modeling this characteristic are the works developed by Ngo and Scordelis (1967), E-Mezaini and Citipitioglu (1991), Manzoli and Oliver (2008) and Bitencourt Jr. et al. (2018).

1.1 Objectives

The main objective of this work is to predict the behavior of reinforced concrete beams by the finite element method. Thus, the following secondary objectives should be performed:

- To use the numerical model proposed by Bitencourt Jr. et al. (2018) to predict the Ultimate and Serviceability Limit States of reinforced concrete beams.
- To predict crack width of reinforced concrete beams by numerical model;
- To predict the behavior of reinforced concrete beams by *fib* Model Code 2010 (2013).
- To compare the results obtained by *fib* Model Code 2010 (2013) and numerical analyses.

1.2 Methodology

This work uses the set of software for simulation of concrete structures by finite element method developed in the last two years by the research group of Professor Luís A.G. Bitencourt Jr. The input parameters are defined graphically in the pre-processing stage. The results are explored in the post-processing stage in terms of displacement, stress, strain, etc. The solver developed in MATLAB language is reponsible by the analysis by finite element method. For analyses of reinforced concrete structures, more details about the mathematical formulation can be seen in Bitencourt Jr. (2015) and Bitencourt Jr. et al. (2018).

1.3 Monograph structure

The document is organized in the following sections. In chapter 2 the numerical model is presented. The design predictions according to fib Model Code is described in chapter 3. In chapter 4 the numerical examples are performed. And finally, the conclusions of this research are presented in chapter 5.

2 Numerical model

This section describes the numerical model used to simulate reinforced concrete beams.

The mesh refinement is performed using basic triangle elements with linear interpolation functions (CST - constant strain triangles) which provides results fair enough and a lower computing time. The steel reinforcements (rebars) are descretized using truss finite elements and their behavior is described by one-dimensional elastoplastic constitutive model.

The problemas are simulated in 2D in order to avoid high computational costs. For all the cases, plane stress condition is applied.

The basic procedure adopted in the simulations is: the geometry of the problem is constructed, by defining the lines that will be used to represent the rebars. Then, the problem is discretized in finite elements, by adopting the apropriate constitutive model to describe the behavior of each component. The coupling between the reabrs and concrete is performed by the introduction of coupling finite elements. Fir these elements an appropriate constituive model is also adopted to define the bond-slip law between them. Details about those three components, concrete, rebars and concrete-rebars interactions are given in the following sections.

2.1 One-dimensional elastoplastic model for reinforcing bars

The reinforcing bars (rebars) are represented by linear elements since their cross section is smaller than its length. The behavior of these elements are represented by the one-dimensional elastoplastic model depicted in Figure 1.

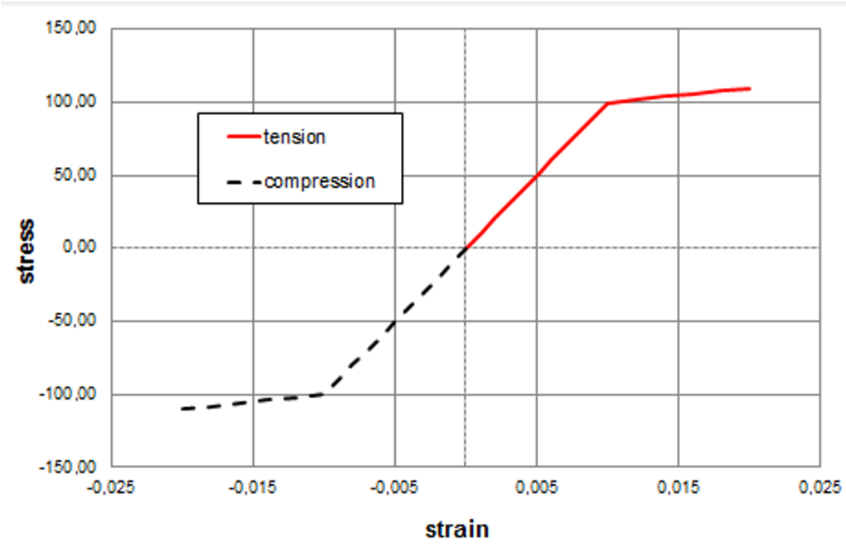


Figure 1: One-dimensional elastoplastic model for reinforcing bars

2.2 Concrete modeling

A continuum isotropic damage model developed by Cervera et al. (1996) is adopted do describe the concrete behavior in tension and compression (see Figure 2) by using two independent scalar damage variables. It is important to mention that in this model the fracture energy (G_f), which is the amount of energy required to create a tensile crack of unit area, is an important parameter to calculate the tensile behavior and expressed in term of N/m.

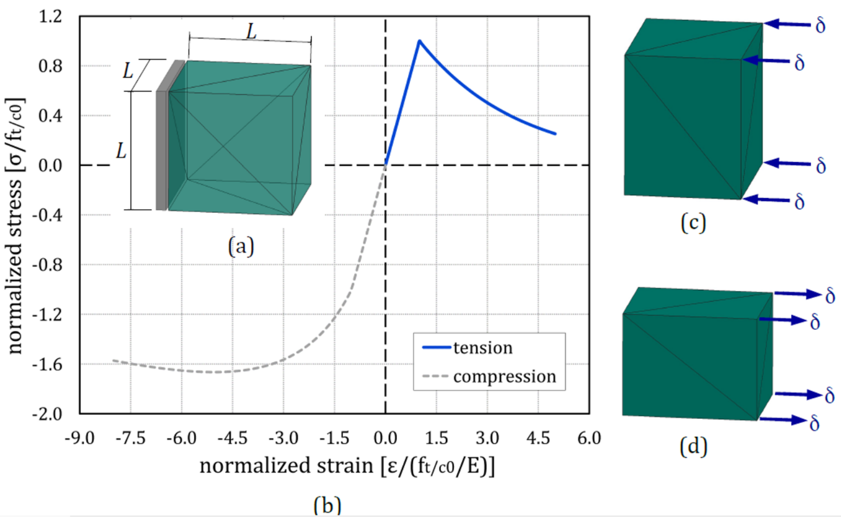


Figure 2: Isotropic damage model proposed by Cervera et al. (1996) with deformed configuration in the case of tension and compression loading (from Bitencourt Jr. et al. (2018))

2.3 Concrete-reinforcing bars interaction

The complexity involved in modeling of reinforced concrete structures is due the coupling between the concrete and reinforcements. Each one with its own function and behavior (physical properties).

The advantage of reinforced concrete is precisely the combination of concrete that has a high compression resistance and the rebars with good response under tensile loading. Modeling reinforced concrete behavior lies on the interaction between these two materials. In this study the interaction is modeled by the coupling scheme proposed by Bitencourt Jr. (2015) (see Figure 3). This strategy allows to described the interaction using the formulas given by the *fib* Model Code 2010 (2013).

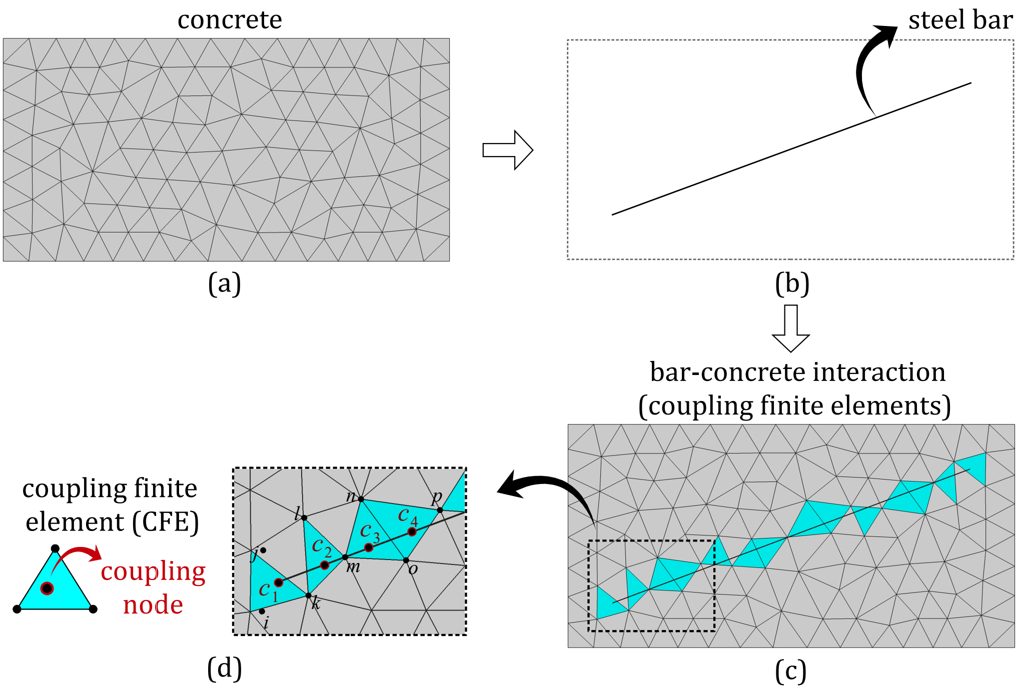


Figure 3: Coupling procedure (adapted from Gravina et al. (2018))

In the coupling procedure using coupling finite elements (CFEs) there is no need to know the length of the bar confined in a concrete element. This fact is an advantage in terms of time-consumption. For each loose node of the rebars a coupling finite element is introduced in the mesh using as reference the concrete elements without the addition of degrees of freedom to the problem. As a consequence, no further adaptation is require to standard finite element procedure. In other words, the global internal force vector is equal to the sum internal force components from concrete, rebars and coupling finite elements. The same procedure is applied for the stiffness matrix.

$$F_{int} = F^{c}_{int} + F^{s}_{int} + F^{ce}_{int}$$

$$K = K^{c} + K^{s} + K^{ce}$$

2.3.1 Perfect bond

The creation of coupling finite elements is based on the concrete elements, in which the loose node belongs to its domain. This procedure is performed in a preprocess stage using the information obtained by the propgram GID.[1]. A Matlab programa has been developed to generate these CFEs and [2] to connect truss finite steel elements to the nearest triangular concrete element as shown in the Figure 4. The generation of these elements has the following informations: nodes *id* of the concrete triangle, node *id* of the steel element and the bond-slip relationship *id* to simulate the corresponding concrete-steel interaction. According to these strategy by adopting high value for the coupling parameter by defining a perfect coupling between rebars and concrete is adopted.

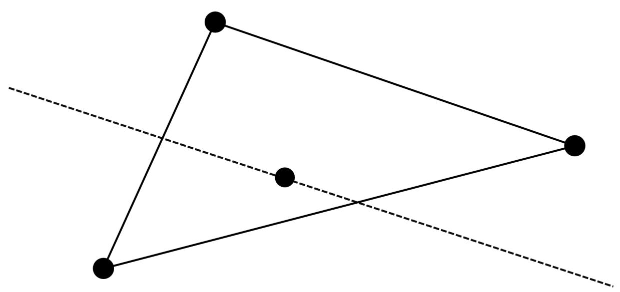


Figure 4: Definition of a coupling finite element to couple concrete-rebar interaction

[1]Pre-pos processing software used in this work. The software has been developed by the *International Center for Numerical Methods in Engineering.*

[2]Program for generation of Coupling Finite Element developed by *L. Bitencourt and Y. Trindade.*

2.3.2 Local bond–slip

The bond-slip relation is applied using the model proposed by fib Model Code, as illustrated in Figure 5. This model is applied through a continuum damage model with a scalar damage variable and IMPL-EX integration scheme, in order to avoid problems of convergence.

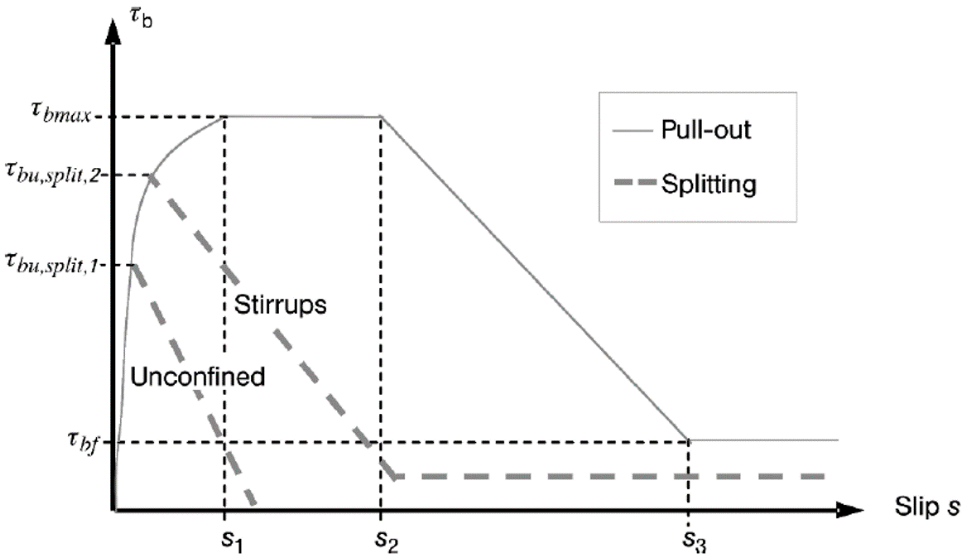


Figure 5: Illustration of the concrete-steel interactions (from MC2010)

$$\tau_b(s) = \begin{cases} \tau_{b,max}(\frac{s}{s_1})^{\alpha} & \text{if } 0 \leq s \leq s_1 \\ \tau_{b,max} & \text{if } s_1 \leq s \leq s_2 \\ \tau_{b,max} - (\tau_{b,max} - \tau b, f)\frac{s-s_2}{s_3-s_2} & \text{if } s_2 \leq s \leq s_3 \\ \tau_{b,f} & \text{if } s_3 \leq s \end{cases}$$

2.3.3 Estimating the bond-slip parameters according to *fib* Model Code 2010 (2013)

The bond stress–slip relation is based on following parameters[3] depicted in Table 1:

- $\tau_{b,max}$: the maximum bond stresses between concrete and reinforcing bar;

[3]See Table 6.1-1 of *fib* Model Code 2010 (2013)

- $\tau_{b,min}$: the minimum bond stresses between concrete and reinforcing bar;
- s_1, s_2, s_3: slip parameters;
- α: exponent;
- c_{clear}: mean distance between ribs;

These parameters depend on the failure mode (pull-out or splitting) and for different bond conditions.

Table 1: Concrete-rebar interaction parameters (from MC2010)

	1	2	3	4	5	6
	Pull-out (PO)		**Splitting (SP)**			
	$\varepsilon_s < \varepsilon_{s,y}$		$\varepsilon_s < \varepsilon_{s,y}$			
	Good bond cond.	**All other bond cond.**	**Good bond cond.**		**All other bond cond.**	
			Unconfined	**Stirrups**	**Unconfined**	**Stirrups**
τ_{bmax}	$2.5\sqrt{f_{cm}}$	$1.25\sqrt{f_{cm}}$	$2.5\sqrt{f_{cm}}$	$2.5\sqrt{f_{cm}}$	$1.25\sqrt{f_{cm}}$	$1.25\sqrt{f_{cm}}$
$\tau_{bu,split}$	—	—	$7.0 \cdot \left(\frac{f_{cm}}{25}\right)^{0.25}$	$8.0 \cdot \left(\frac{f_{cm}}{25}\right)^{0.25}$	$5.0 \cdot \left(\frac{f_{cm}}{25}\right)^{0.25}$	$5.5 \cdot \left(\frac{f_{cm}}{25}\right)^{0.25}$
s_1	1.0 mm	1.8 mm	$s(\tau_{bu,split})$	$s(\tau_{bu,split})$	$s(\tau_{bu,split})$	$s(\tau_{bu,split})$
s_2	2.0 mm	3.6 mm	s_1	s_1	s_1	s_1
s_3	c_{clear} 1)	c_{clear} 1)	$1.2 s_1$	$0.5 c_{clear}$ 1)	$1.2 s_1$	$0.5 c_{clear}$ 1)
α	0.4	0.4	0.4	0.4	0.4	0.4
τ_{bf}	$0.40 \tau_{max}$	$0.40 \tau_{max}$	0	$0.4 \tau_{bu,split}$	0	$0.4 \tau_{bu,split}$

Columns 1 and 2 (pull-out failure) in Table 1 is valid for confining reinforcement. The values in columns 3 to 6 (splitting failure) the values are derived from a empirical equation for the reinforcement stress (f_{stm}) in MC2010 and calculate according to Eq. 1 for $\phi = 25$ mm, $c_{max}/c_{min} = 2.0$, $c_{min} = \phi$ and $K_{tr} = 0.02$ in the case bar confined by stirrups or $K_{tr} = 0$ for unconfined situation. The bond stress for split failure for the other cases is given by:

$$\tau_{bu,split} = \eta_2 \cdot 6.5 \cdot \left(\frac{f_{cm}}{25}\right)^{0.25} \left(\frac{25}{\phi}\right)^{0.20} \left[\left(\frac{c_{min}}{\phi}\right)^{0.33} \left(\frac{c_{max}}{c_{min}}\right)^{0.10} + k_m \cdot K_{tr}\right] \tag{1}$$

where:

- Bond conditions[4]:
  - $\eta_2 = 1.0$ for stirrups: in every case all bars have an inclination between 45° and 90° to the horizontal during casting;
  - $\eta_2 = 0.7$ for horizontal reinforcement;
- f_{cm} is the mean cylinder concrete compression strength (N/mm^2);
- ϕ is the diameter of the bar (mm);
- $c_{min} = min(c_s/2, c_x, c_y)$ (see Figure 6);
- $c_{max} = max(c_s/2, c_x)$ (see Figure 6);
- k_m represents the efficiency of confinement from transverse reinforcement;
- K_{tr} coefficient:
  - $K_{tr} = 0.02$ if the bars are confined by stirrups;
  - $K_{tr} = 0$ for unconfined situation.

In MC2010 the parameter k_m varies from 0 to 12. In this work for safety evaluation it was defined as equal to zero which corresponds to a situation where splitting occurs for a lower value of strength.

[4]See section 6.1.3.2 of *fib* Model Code 2010 (2013)

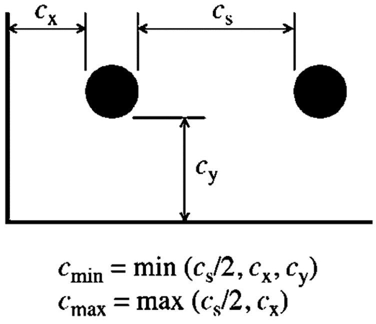


Figure 6: Spacing parameters c_s, c_x, c_y (from MC2010)

Furthermore, to calculate $s(\tau_{bu,split})$ is necessary to have s_1 as reference. As can be seen in Figure 5 the first branch of the curve has the equation type: $\tau_b = \tau_{b,max}(s/s_1)^{\alpha}$. Consequently, $s(\tau_{bu,split})$ can be calculates as follows:

$$s(\tau_{bu,split}) = s_1 \left(\frac{\tau_{bu,split}}{\tau_{b,max}} \right)^{1/\alpha}$$

where the value of s_1 is considered equal to the two first columns of Table 1, $s_1 = 1$ mm for good bond conditions (stirrups) or 1.8 mm for all other bond conditions.

2.4 Material and geometric equivalent properties

All the studied beams are model in two dimension, therefore some adjustment must be applied.

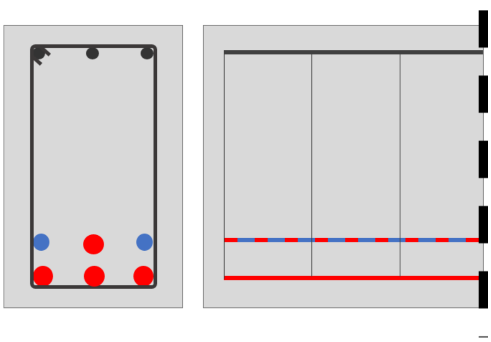


Figure 7: Geometric equivalent properties in 2D analysis

In some cases rebars can be aligned in the cross section of a beam, as illustrated in Figure 7. Consequently, in 2D modeling a unique rebar must have the equivalent properties. Then, the cross section area of the equivalent bar (A^*) is equal to the sum of the area of the real rebars (A_i):

$$A^* = \sum A_i$$

For the cases where the equivalent material properties must be calculated for different rebars sizes aligned and different properties, such as Young's modulus and yield stress, the equivalent propriety (X^*) is calculated as the sum of the individual properties (X_i) weighted by the areas (A_i):

$$X^* = \frac{\sum A_i \cdot X_i}{\sum A_i}$$

For the constitutive model to describe concrete-rebar interaction, the rebar equivalent perimeter (P^*) is calculated considering an equivalent diameter (D^*) as:

$$P^* = \pi \cdot D^* = \pi \sqrt{\frac{4 \cdot \sum A_i}{\pi}}$$

2.5 Matlab code to estimate crack width

Initially proposed by Hillerborg et al. (1976) the crack width can be estimated as:

$$w = \varepsilon_1 \cdot l_{cs} \tag{2}$$

where ε_i is the principal strain in the corresponding concrete element and l_{cs} is the structural characteristic length.

The structural characteristic length can be estimated according to Rots (1988) as

$$l_{cs} = \sqrt{2A} \tag{3}$$

were A is the area of the concrete finite element.

A Matlab code was developed to estimate crack width using the results of numerical analysis in terms of the principal strain ε_1 and the structural characteristic length l_{cs}. The code consists on:

(i) search the truss element (rebar) where there is the maximum stress;

(ii) find the corresponding coupling finite element (CFE) associated with the rebar;

(iii) find the corresponding concrete finite element associated with the CFE;

(iv) if the damage factor of the concrete element is higher than 0,9, the principal strain of this element is stored;

(v) the area of the concrete element is calculated;

(vi) the crack width is obtained.

This procedure is repeated for every load step. More details of the Matlab code are presented in Appendix B.

3 Design predictions according to *fib* Model Code 2010 (2013)

The material properties calculated according to the formulas of the *fib* Model Code 2010 (2013) (herein MC2010) are defined in this section. In addition, the design process of a beam according to MC2010 is also described.

3.1 Material properties

3.1.1 Poisson's ratio

According to the MC2010[5] for the Poisson's ratio ν is attributed the value of 0.20. This value respects the criteria $0.14 \leq \nu \leq 0.26$ and is recommended in the case of experimental value is not given.

3.1.2 Tensile strength

The concrete tensile strength (f_{cm}) is calculated[6] as:

$$f_{ct} = 0.3 \cdot f_c^{2/3} \tag{4}$$

where f_c is the compression strength.

3.1.3 Fracture energy

The fracture energy (G_F) can be estimated as:

$$G_F = 78 \cdot f_{cm}^{0.18} \tag{5}$$

[5]See section 5.1.7.3 of *fib* Model Code 2010 (2013)
[6]See section 5.1.5.1 of *fib* Model Code 2010 (2013)

where f_{cm} is the mean compressive strength in MPa and G_F in (N/m) according to MC2010[7].

3.2 Design of RC beam according to the *fib* Model Code 2010 (2013)

3.2.1 Design for flexure

The design bending moment $M_{R,max}$ is function of the area and resistance of concrete material or steel material.

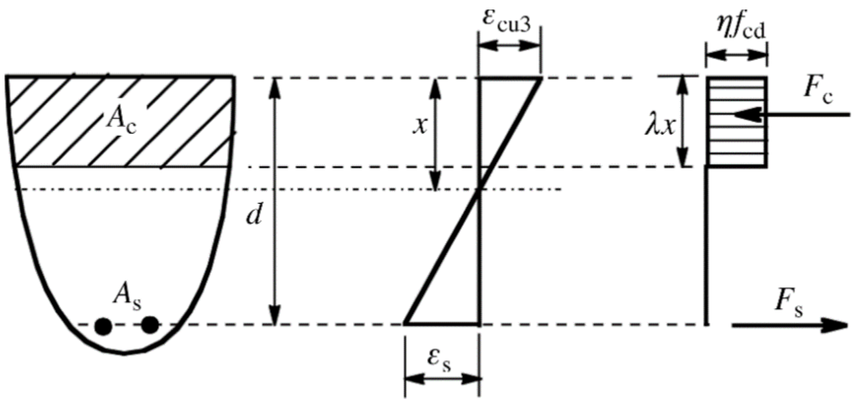


Figure 8: Rectangular stress distribution for bending design (from fib *Model Code 2010 (2013)*

The bending moment[8] resulting from the active concrete area strength after simplification authorized by the model is equal to:

$$M_{Rc} = \eta \cdot f_{ct} \cdot \lambda \cdot x \cdot b_w \cdot (d - 0,45x)$$

where $\lambda = 0,80$ and $\eta = 1$ for concrete strength inferior to 50 MPa, b_w is the width of the beam and x defined the real depth of concrete that is resisting.

The equivalent bending moment resulting from the steel reaction is:

$$M_{Rs} = A_s \cdot f_{yd} \cdot (d - 0,45x)$$

[7]See section 5.1.5.2 of *fib* Model Code 2010 (2013)
[8]See section 7.2 of *fib Model Code 2010*

where A_s is the total area of the cross section of the steel reinforcing bars. Then, equalizing the resisting bending moment to the solicitation $M_{S,max} = M_{Rs} = M_{Rc}$, the variable x defining the active concrete area can be calculated, and consequently, the quantity of reinforcing steel bar necessary to resist the solicitation.

3.2.2 Design for shear

The design shear force V_{Rd} depends on both of the resistance of concrete V_{Rc} and steel V_{Rs}, the inclination of the stirrups α, the inclination of the compressing stress field θ and the resistance properties of each material.

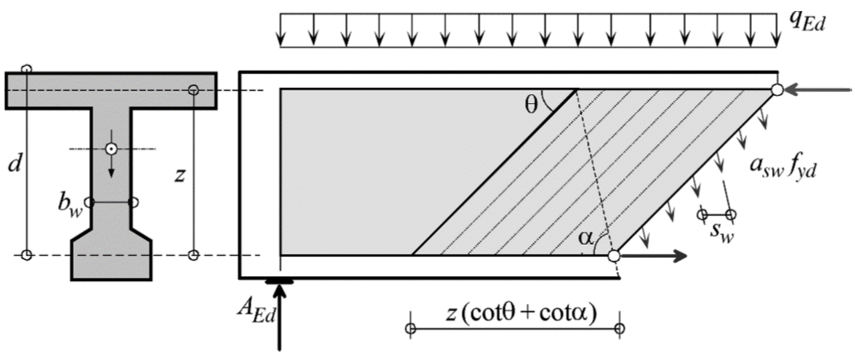


Figure 9: Geometry and definitions for shear (from fib *Model Code 2010 (2013)*

The formulas are:

1. design shear force: $V_{Rd} = V_{Rc} + V_{Rs} = k_c \cdot \frac{f_{ck}}{\gamma_c} \cdot b_w \cdot z \cdot \sin\theta \cos\theta$
2. strength reduction factor: $k_c = 0,55 \cdot \eta_{fc}$
3. factor: $\eta_{fc} = \left(\frac{30}{f_{ck}}\right)^{1/3} \leq 1,0$
4. inclination of the stirrups: $\alpha = \pi/2$ for reinforcing concrete members.
5. shear resistance provided by the stirrups: $V_{Rs} = \frac{A_{sw}}{s_w} \cdot z \cdot fywd \cdot \cot\theta$
6. shear resistance provided by the concrete: $V_{Rc} = k_v \cdot \frac{\sqrt{f_{ct}}}{\gamma_c} \cdot b_w \cdot z$ where $\sqrt{f_{ct}} \leq 8,0\ MPa$
7. factor: $k_v = \frac{0,4}{1+1500\epsilon_x}\left(1 - \frac{V_S}{V_{Rd}}\right) \geq 0$
8. the strain: $\epsilon_x = \frac{1}{2E_sA_s}\left(\frac{M_S}{z} + V_S + N_S\left(\frac{1}{2} \pm \frac{\Delta e}{z}\right)\right)$

Solving and rearrange those equations one may calculate the steel rate per meter needed A_{sw}/s_w.

3.2.3 Calculation of crack width

The predicted crack width w_d in function of the steel stress σ_s in the reinforcing steel bar is given by the formulas[9]:

1. crack width: $w_d = 2l_{s,max}(\epsilon_{sm} - \epsilon_{cm} - \epsilon_{cs})$, where $l_{s,max}$ is the length other which slip between concrete and steel occurs, ϵ_{cm} is the average strain of concrete over $l_{s,max}$, ϵ_{sm} is the average strain of steel over $l_{s,max}$, and ϵ_{cs} is the strain of the concrete due to shrinkage;

2. the length: $l_{s,max} = k \cdot c \cdot + \frac{1}{4} \cdot \frac{f_{ctm}}{\tau_{bms}} \cdot \frac{\phi_s}{\rho_{s,ef}}$, where k is equal to 1, c is the concrete cover, τbms the mean bond stress and $\rho_{s,ef} = As/A_{c,ef}$ the effective steel-concrete rate;

3. the relative main strain: $\epsilon_{sm} - \epsilon_{cm} - \epsilon_{cs} = \frac{\sigma_s - \beta \cdot \sigma_{sr}}{E_s} - \eta_r \cdot \epsilon_{sh}$, where σ_s is the steel stress in the crack, and σ_{sr} is the maximum steel stress in the crack for pure tension;

4. the maximum steel stress: $\sigma_{sr} = \frac{f_{ctm}}{\rho_{s,ef}}(1 + \alpha_e \cdot \rho_{s,ef})$, where $\alpha_e = E_s/E_c$ is the modular ratio;

5. for short term loading and instantaneous loading: $\beta = 0,6$, $\eta_r = 0$ and $\tau_{bms} = 1,8 \cdot f_{ctm}$.

[9]See section 7.6.4.3 Limitation of crack width of *fib* Model Code 2010 (2013)

4 Numerical examples

In this chapter three examples are numerically analyzed. For the first part, the structural response of a beam experimentally tested by Vecchio and Shim (2004) is studied. In the second part the crack width prediction of RC beams is studied in two examples: beams experimentally tested by Ma and Kwan (2015) and beams designed according to *fib* Model Code 2010 (2013).

4.1 Structural response of a beam tested by Vecchio and Shim (2004)

In the 60's Bresler and Scordelis (1963) tested 12 different beams in order to investigate reinforced concrete behavior and the results became a benchmark to calibrate finite element models. Later, Vecchio and Shim (2004) published an reexamination of the previous paper (herein named Toronto beams) comparing the experimental and analytical results of similar 12 types of beams.

The set of 12 beams are divided in 4 different categories: varying the amount of longitudinal span length, cross-section dimensions, concrete strength and longitudinal and shear reinforcements. In this study, one type of beam (A1) was chosen to be numerically analyzed. The geometrical and material properties are described as follow.

4.1.1 Geometrical properties

Table 2 shows the geometrical characteristics of beam A1: name, width (b), height (h), length (L) and the *Span* between the supports as illustrated in Figure 10.

Beam	b *mm*	h *mm*	d *mm*	L *m*	Span *m*	Bottom steel	Top steel	Stirrups
A1	305	552	457	4,10	3,66	2 M30,2 M25	3 M10	D5 at 210

Table 2: Geometrical characteristics of the specimen

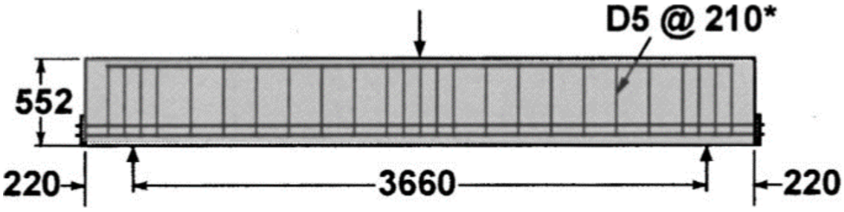


Figure 10: Geometry and reinforcement details of the beam A1 (from Vecchio and Shim (2004))

The reinforcement details of the beam are shown in Table 2 and the reinforcement configuration is depicted in Figure 11. The information in the last column (stirrups) of Table 2 refers to the name and the longitudinal spacing between them. Additional stirrups are located in critical regions as shown in Figure 10.

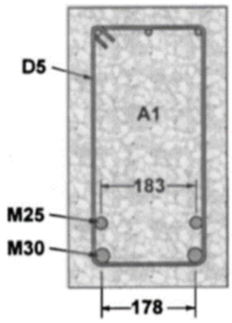


Figure 11: Cross-section of beam A1 (from Vecchio and Shim (2004))

4.1.2 Material properties

The material properties of steel reinforcement and concrete obtained from Vecchio and Shim (2004) and estimated using MC2010 are shown in Table 3 and Table 4, respectively.

The numerical model based on the Modified Compression Field Theory (MCFT) used in Vecchio and Shim (2004) estimated the tensile strength as $f_{ct} = 0.33 \cdot \sqrt{f_c}$, where f_c is the compression strength of concrete. However, the expression used for the numerical simulation of this study was the one presented in Section 3.1.2 from MC2010. Note that in Table 4 the value of the tensile strength f_{ct} calculated using MC2010 is 53% higher than in Vecchio and Shim (2004).

Bar size	Diameter	Area	perimeter	f_y	f_u	E_s
	mm	mm^2	mm	MPa	MPa	MPa
M10	11,30	100	35,50	315,00	460,00	200000
M25	25,20	500	79,17	445,00	680,00	220000
M30	29,90	700	93,93	436,00	700,00	200000
D5	6,40	32	20,11	600,00	649,00	200000

Table 3: Reinforcing material properties for beam A1

Beam	f_c	E_c	f_{ct} [Toronto]	f_{ct} [fib]	G_F [fib]	$f_{ct}^{[fib]}/f_{ct}^{[Toronto]}$
	MPa	MPa	MPa	MPa	N/m	%
A1	22,60	36500	1,57	2,40	127,96	153

Table 4: Concrete material properties for beam A1

Table 5 shows the equivalent geometrical and physical properties for the steel equivalent diameter (D^{eq}), perimeter (P^{eq}), tensile yield strength (f_y^{eq}), ultimate yield strength (f_u^{eq}) and modulus of elasticity (E_s^{eq}) calculated for the 2D analysis.

Steel	D^{eq}	P^{eq}	P^{eq}_{axis}	f_y^{eq}	f_u^{eq}	E_s^{eq}
	mm	mm	mm	MPa	MPa	MPa
1 M25	25,20	79,17	-	-	-	-
1 M30	29,90	93,93	-	-	-	-
2 D4	7,40	23,25	11,62	-	-	-
2 D5	12,80	40,21	20,11	-	-	-
2 M25	50,40	158,34	-	-	-	-
2 M25, M30	80,30	252,27	-	441,29	688,24	211764,71
2 M30	59,80	187,87	-	-	-	-
3 M10	33,90	106,50	-	-	-	-
3 M30	89,70	281,80	-	-	-	-

Table 5: Steel reinforcement material properties

Table 6 illustrates the coupling parameters to describe the interaction steel-concrete for each rebar considering equivalent properties for 2D analysis. Note that k_x corresponds to the axis bar direction.

Beam	Steel	k_x	k_y	k_z	$\tau_{bu,max}$	τ_{bf}	s_1 mm	s_2 mm	s_3 mm	α	P mm
A1	3 M10	1E+03	1E+09	1E+09	11,88	0,00	0,86	0,86	1,03	0,4	106,50
	2 M25	1E+03	1E+09	1E+09	11,88	0,00	0,81	0,81	0,97	0,4	158,34
	2 M30	1E+03	1E+09	1E+09	11,88	0,00	0,78	0,78	0,94	0,4	187,87
	2 D5	1E+09	1E+03	1E+09	11,88	5,39	1,05	1,05	1,18	0,4	40,21

Table 6: Coupling properties for beam A1

4.1.3 Experimental results

The following observations of both Bresler-Scordelis (1963) and Vecchio and Shim (2014) experimental results have been obtained:

- mode failure: shear compression (V-C);
- critical cracks: formed at 60% of the ultimate load, splitting in the compression zone, without splitting in the tension reinforcement;
- time: cracks during later load stage;
- support's region: cracks beneath and adjacent to the support occurring before any shear distress.

4.1.4 Finite element analysis

Mesh sensitivity study

In order to evaluate the most appropriate mesh size that gives precise results without being time-consuming, a mesh sensitivity study has been performed. Three different meshes were generated: 20 mm, 30 mm and 40 mm. The results were evaluated in terms of load *vs.* deflection as depicted in Figure 12.

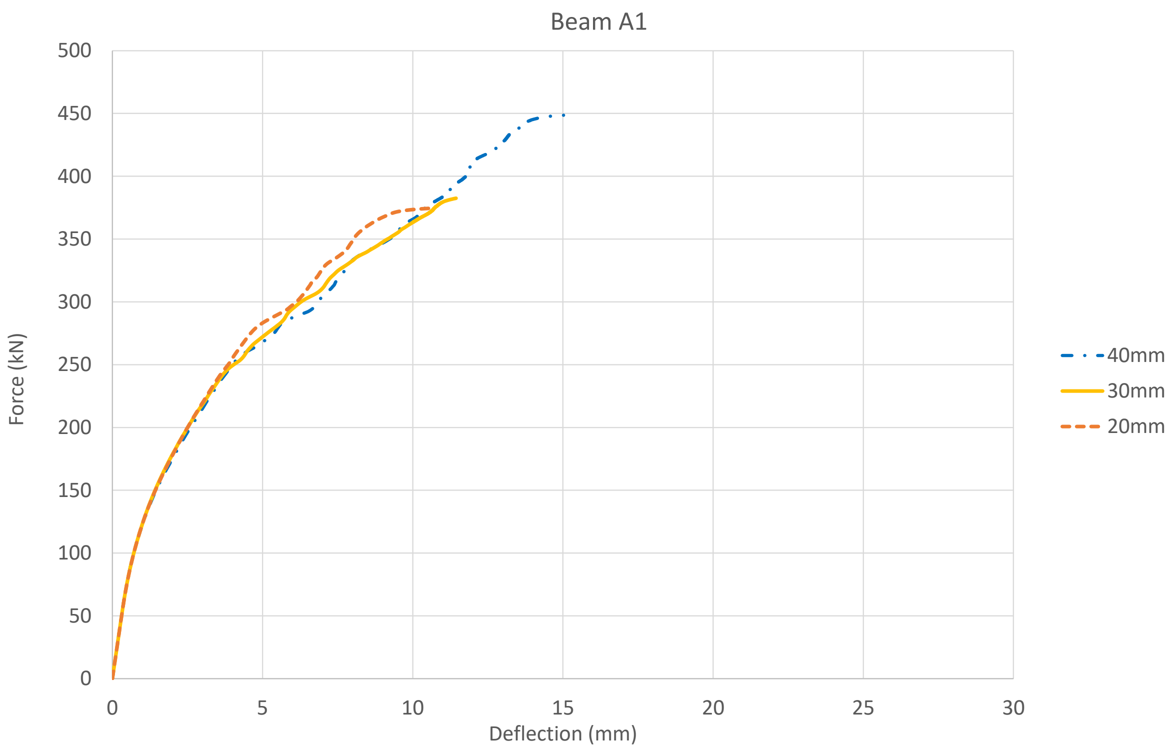


Figure 12: Results of load vs. deflection for the mesh sensitivity study

Figure 12 shows that a mesh of 40 mm returns results with higher values than the meshes of 30 and 20 mm. For this reason, for the next analyses, a mesh of 30 mm was employed due to the good results without increasing the calculation time.

Analysis and results

This section shows the numerical results obtained for beam A1 in terms of ultimate load (P_u), ultimate displacement (δ_u), crack pattern and curve of load *vs.* displacement. In addition, the numerical analyses are compared against the Vecchio and Shim (2004) results. Later, some parameters were modified to show the influence in the structural response (Beam A1.0, A1.2, A1.3 and A1.4).

- **Beam A1.0**

The first beam numerical analyzed is named Beam A1.0 whose properties are shown in Table 6. The crack pattern for both numerical and experimental is illustrated in Figure

13. As can be seen, the cracks appear in the same regions for both results. A large diagonal crack due to shear stresses converge in the direction of the point of application of the load. Smaller and vertical cracks go up as well. Those cracks are due to the bending of the beam.

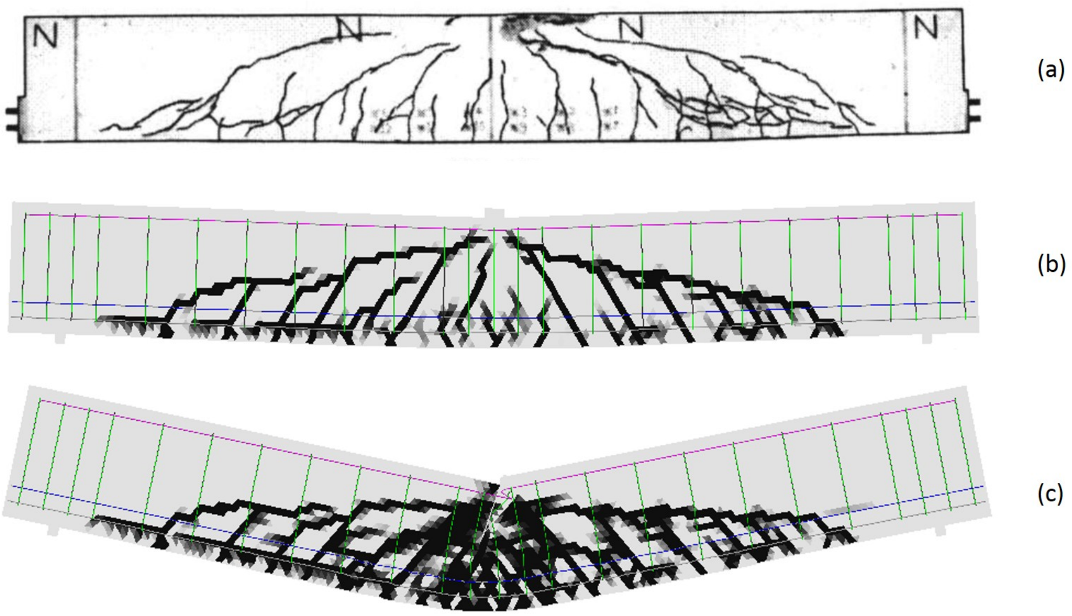


Figure 13: Beam A1 at rupture: (a) Toronto study, (b) model just before rupture, (c) model just after rupture

Figure 14 shows the experimental and numerical results of Vecchio and Shim (2004), the experimental results of Bresler and Scordelis (1963) and the numerical results of this study. In general the results are in good agreement. However, the ultimate load obtained in the numerical analysis differs from the other curves.

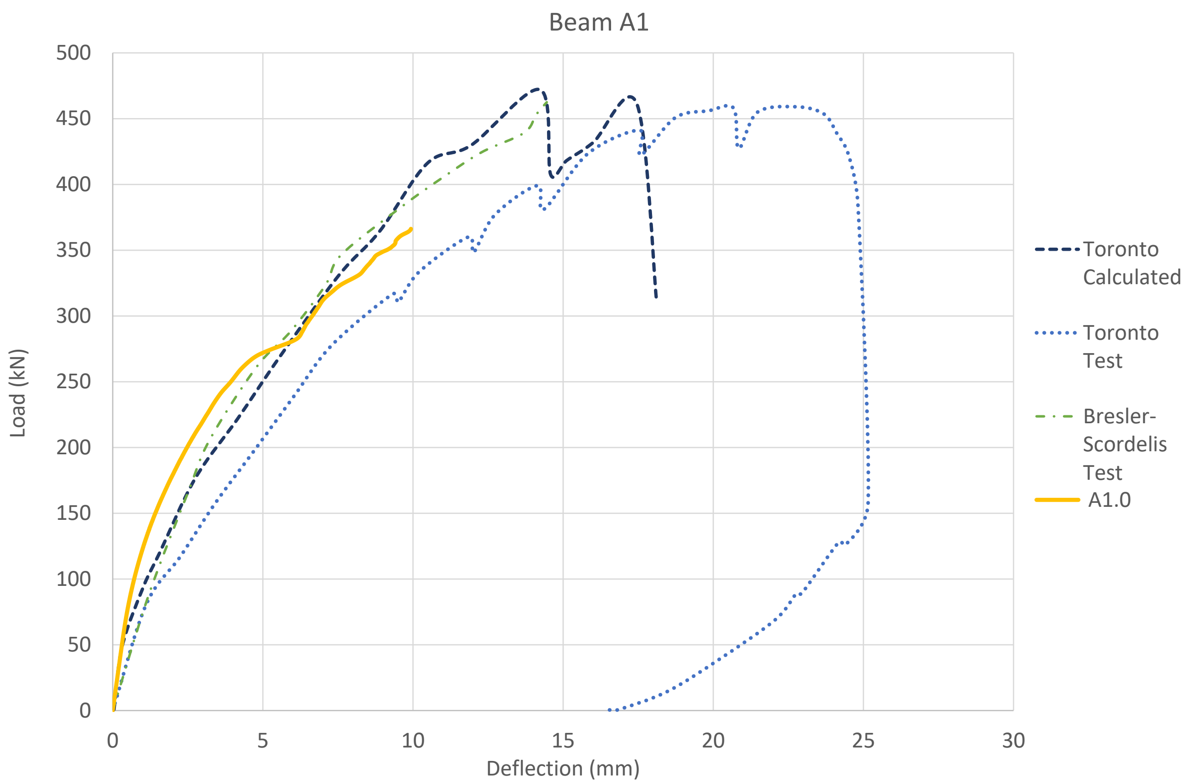


Figure 14: Load-deflection response compared with Toronto study

In order to verify the influence of some parameters in the overall response of the beam, the following values were modified from Beam A1.0: (i) the bond slip parameters (Beam A1.1), (ii) the fracture energy G_f (Beam A1.2), (iii) the plastic behavior of reinforcing bars (Beam A1.3) and (iv) the tensile stress f_{ct} (Beam A1.4).

Table 7 presents the values of the ultimate load and the corresponding deflection and Figure 15 the load vs. deflection for the different situations.

	P_u^{BS} kN	δ_u^{BS} mm	P_u^{VS} kN	δ_u^{VS} mm	P_u^{num} kN	δ_u^{num} mm	ΔP_u^{BS}	$\Delta\delta_u^{BS}$	ΔP_u^{VS}	$\Delta\delta_u^{BS}$
Beam A1.0	467	14,2	459	18,8	365	9,9	1,28	1,43	1,26	1,90
Beam A1.1	467	14,2	459	18,8	372	11,0	1,26	1,29	1,23	1,71
Beam A1.2	467	14,2	459	18,8	367	11,0	1,27	1,29	1,25	1,71
Beam A1.3	467	14,2	459	18,8	478	11,0	0,98	1,29	0,96	1,71

Table 7: Reinforcing material equivalent properties

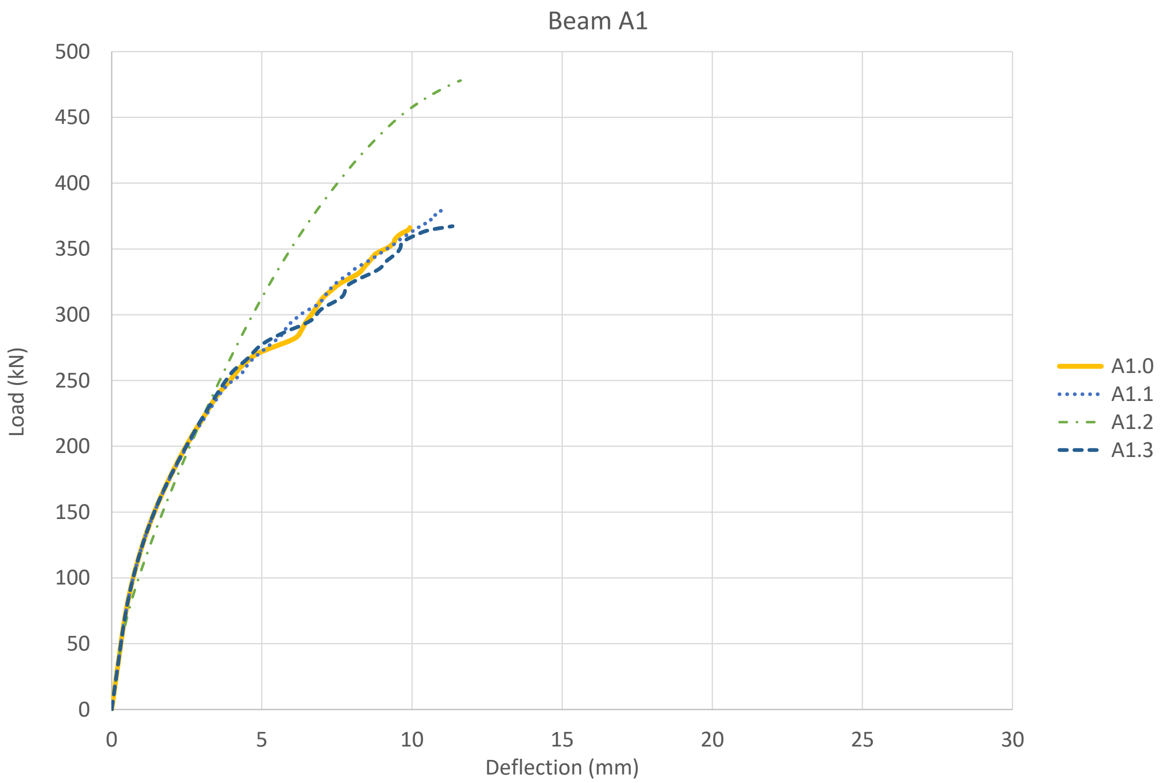


Figure 15: Load vs. deflection response for different parameters adopted.

For each parameter studied, it is important to make the following considerations:

- **Beam A1.1:** The bond-slip parameters

To verify the accuracy of the model, the interaction between steel and concrete was considered as perfect adherence. The result was not very different from the original model because the bottom longitudinal bars were extended past the ends of the beam and anchored.

- **Beam A1.2:** The fracture energy (G_f)

By increasing the energy required to propagate fractures, loading bearing capacity of the beam is enhanced. However, as illustrated in the Figure 16, the crack patterns are quite different from the ones present in the Toronto beam.

- **Beam A1.3:** Ultimate strength of steel (f_u)

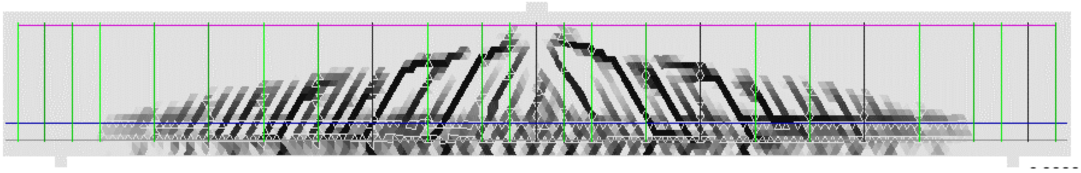


Figure 16: Increasing the fracture energy (G_f) for Beam A1

In order to verify whether or not the ultimate strength of steel has a major part in the beam loading capacity, all values of yield stress f_y were replaced by the values of rupture f_u. In this case, no significant results were obtained, as can be seen in 15.

- **Beam A1.4:** Tensile stress (f_{ct})

Given that the tensile stress in MC2010 was already higher than the one obtained by the formulation proposed in Vecchio and Shim (2004), no change is observed.

4.2 Crack width prediction of RC beams

This section focuses on the prediction of crack width in beams. Two distinct studies have been performed. First, an experimental-numerical comparison based on the results of Ma and Kwan (2015) and, secondly, an analytical-numerical comparison, where the analytical results were obtained based on the MC2010 recommendations for beam design.

The crack width for the numerical results were calculated applying the Matlab code presented in Section 2.5.

4.2.1 Beams experimentally tested by Clark (1976)

In this application the experimental results obtained by Clark (1976) and numerical results presented by Ma and Kwan (2015) are both compared with the results of crack width using the numerical approach described in this work.

Geometrical and material properties

In Clark (1976) each beam were designed and represented by a serie of numbers in the form A–B–C–D, where A is the depth of cross-section (in inches), B is the width of cross-section (in inches), C is the bar size number and D is a serial number. In this work, the beams 15-6-8-1 and 15-6-6-1 were chosen. The geometrical and material properties of the beams are presented in Figure 17 and Table 8, respectively.

Considering that for some properties, the paper (Ma and Kwan, 2015) gives a range of values for each beam [10], the analyses have been performed using the interpolated values.

The two beams (15-6-8-1 and 15-6-6-1) were simulated using the same dimensions and properties as given in Ma and Kwan (2015). In addition, for Cervera et al. (1996), the constants values $A = 0,89$ and $B = 1,16$ of the concrete have been also employed.

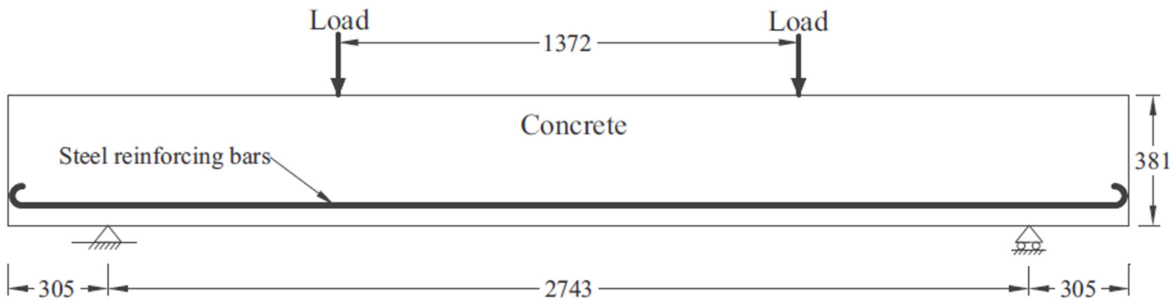


Figure 17: Beam test setup (from Ma and Kwan (2015)) (all dimensions in mm)

[10]See Table 1 and 2 of Ma and Kwan (2015)

Name	15-6-8-1	15-6-6-1	
Geometry			
Beam Length	3353,00	3353,00	mm
Mesh dimension	15,00	15,00	mm
Number of bars	1,00	2,00	
Steel bar diameter	25,40	19,10	mm
Steel perimeter	79,80	60,00	mm
Steel section	506,71	286,52	mm^2
Effective depth	330,00	339,90	mm
width	152,00	152,00	mm
Reinforcement ratio	0,01	0,01	%
Concrete			
Concrete strength	26,80	30,10	MPa
Uniaxial tensile strength	2,89	3,10	MPa
Initial elastic modulus	24,46	26,00	GPa
Poisson's ratio	0,20	0,20	1
Fracture toughness	1,26	1,26	$MNm^-2/3$
Steel reinforcement			
Yield strength	275,70	275,70	MPa
Ultimate tensile strength	482,60	482,60	MPa
Initial elastic modulus	200,00	200,00	GPa
Tensile strain at start of strain hardening	1,00	1,00	%
Ultimate tensile strain	10,00	10,00	%
Steel reinforcement-concrete bond–slip properties			
Peak bond stress	10,47	11,20	MPa
Slip at start of peak bond stress	0,60	0,60	mm
Slip at end of peak bond stress	0,60	0,60	mm
Slip at start of residual bond stress	2,50	2,50	mm
Residual bond stress	1,57	1,68	MPa
Alpha	0,40	0,40	

Table 8: Beams properties (adapted from Ma and Kwan (2015).

Crack width prediction from Ma and Kwan (2015)

The paper published by Ma and Kwan (2015) presents a different approach to estimate crack width[11]. Once the cracking criterion is attained, the crack width (w) is calculated as the sum of the displacement (d_j, d_k) of two of the three nodes of the triangles concrete element perpendicular to the main tensile strength evaluated as:

[11] See page 213, Eq. 10, in Ma and Kwan (2015)

$$w = |d_j + d_k| \tag{6}$$

and illustrated in Figure 18.

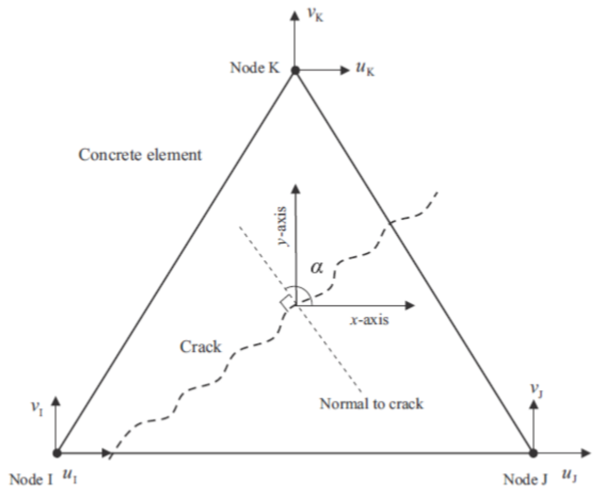


Figure 18: Crack formation (from Ma and Kwan (2015))

FE results

The numerical results obtained in this work are compared against the experimental and numerical results from Clark (1976) for both beams named 15-8-6-1 and 15-6-6-1.

The crack width values for beam 15-8-6-1 for different values of steel stresses are shown in Table 9 and in Figure 19.

Steel Stress MPa	Measured Crack Width HK mm	Numerical Crack Width HK mm	Numerical Crack width mm	Error HK %	Error %
103,35	0,05	0,14	0,00	157%	0%
137,69	0,12	0,21	0,21	83%	82%
172,69	0,17	0,24	0,27	45%	62%
207,34	0,21	0,28	0,38	32%	76%
241,65	0,24	0,34	0,43	35%	70%
276,64	0,28	0,38	0,00	30%	0%
Average Error				64%	73%

Table 9: Experimental and numerical results - Beam 15-8-6-1

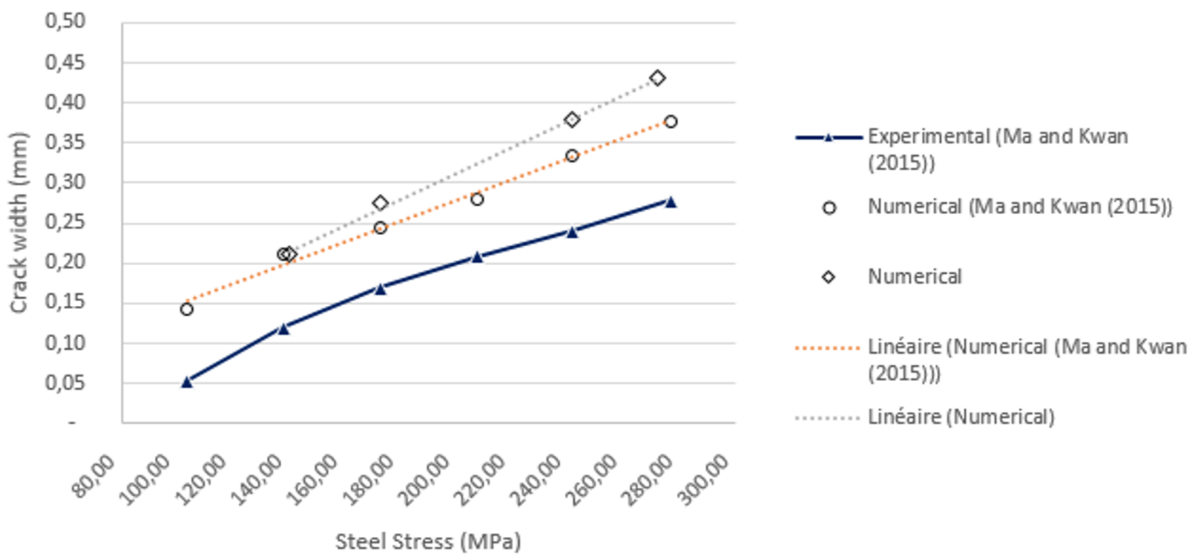


Figure 19: Stress vs. crack width: experimental and numerical results for beam 15-8-6-1

For the beam 15-6-6-1 the crack width values for different values of steel stresses are shown in Table 10 and the crack width vs. steel stress is depicted in Figure 20.

Steel Stress MPa	Measured Crack Width HK mm	Numerical Crack Width HK mm	Numerical Crack width mm	Error HK %	Error %
103,28	0,06	0,13	0,06	129%	7%
137,70	0,11	0,15	0,13	37%	15%
172,46	0,17	0,20	0,19	15%	11%
206,89	0,23	0,25	0,24	9%	6%
241,31	0,27	0,29	0,33	6%	20%
276,07	0,32	0,29	0,38	11%	17%
Average Error				34%	13%

Table 10: Values of experimental and numerical results - Beam 15-6-6-1

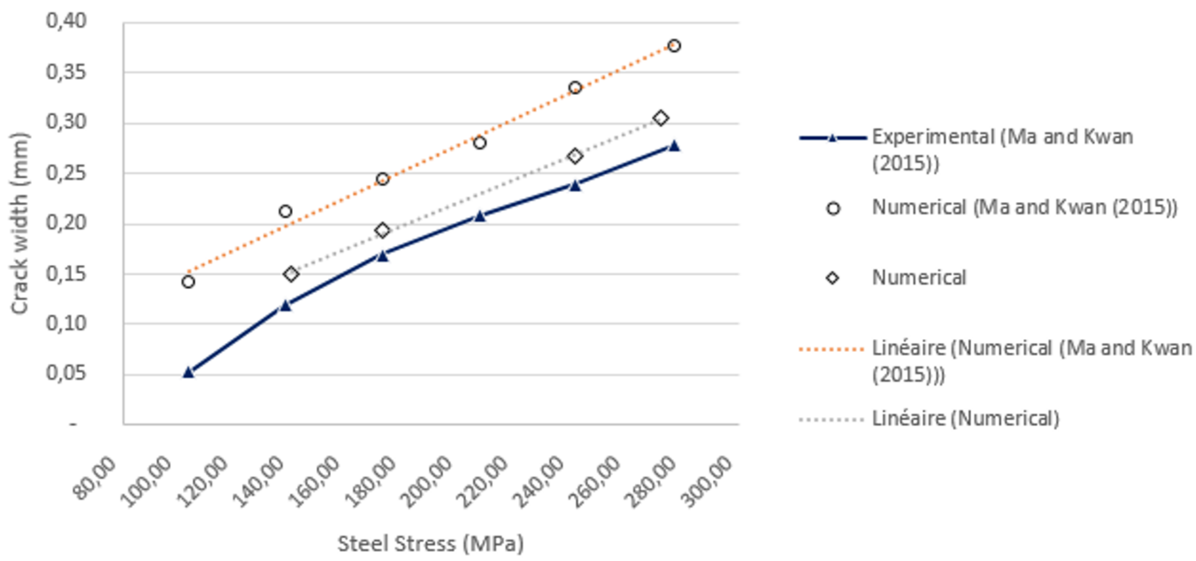


Figure 20: Stress vs. crack width: experimental and numerical results for the beam 15-6-6-1

For both beams, the numerical values almost always overestimate the values of the experimental one. By comparing the values of crack width calculated from Rots (1988) (using $l_{cs} = \sqrt{2A}$) with those produced by Ma and Kwan (2015), smaller values for the beam 15-8-6-1 can be observed and higher values for the beam 15-6-6-1.

Furthermore, it can be noted that the error increases with the value of the steel stress: the greater the tension in the longitudinal bar more diverging is the calculated crack width.

It is important to mention that if the characteristic length (l_{cs}) is calibrated as the square of the concrete element area ($l_{cs} = \sqrt{A}$) (Cervera et al., 1996) and not as the square of the double of the area ($l_{cs} = \sqrt{2A}$), as adopted in Section 2.5, the results for beam 15-8-6-1 should be better estimated and an error equal to 23% should be obtained instead of 73% (see Table 9).

As can be seen in Figure 21 the crack patterns are similar for both numerical studies, except for the stage corresponding to a steel stress equal to 65 MPa. In the last stage, the number of cracks is higher for Ma and Kwan (2015), 11 cracks against 9 in the numerical results. In the intermediaries stages the number of cracks and spacings is similar.

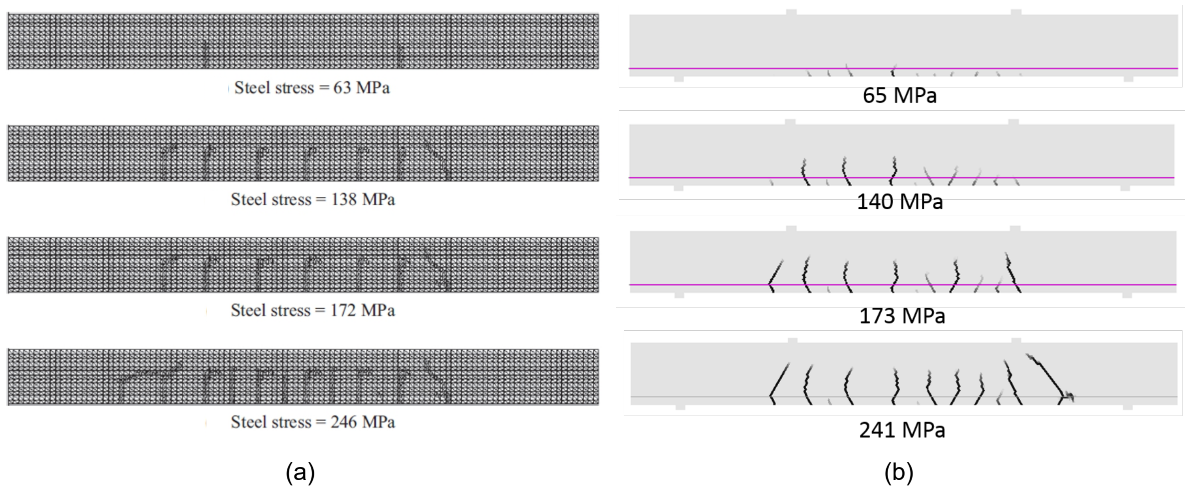


Figure 21: Crack patterns evolution for the corresponding steel stress in beam 15-8-6-1: (a) numerical model from Ma and Kwan (2015) (b) numerical results

4.2.2 Beam designed according to fib Model Code (2010)

In this section, the crack width obtained for a beam designed following MC2010 predictions is compared against the result of the numerical analysis. Different rebar-concrete bond conditions are studied, since this variable can influence the values obtained.

Geometrical and material properties

The geometrical and material properties adopted are summarized in Table 11. The rebars adopted are common types in the Brazilian market.

Geometry		
Beam Length	4,10	m
Thickness	0,22	m
Width	0,40	m
Covering material	0,03	m
Concrete		
Concrete class	C25	
fck	25,00	MPa
γ_c	1,50	
fcd	16,67	MPa
fcm	33,00	MPa
fctm	2,56	MPa
Ec	28,00	GPa
Ecm	32,01	GPa
G	139,23	N/m
Steel bars and stirrups		
Steel class	CA50	
Es	200,00	GPa
fyk	50,00	kN/cm^2
γ_s	1,15	
fyd	43,48	kN/cm^2

Table 11: Beam designed using fib Model Code properties

Design of the beam

The beam are designed applying the MC2010 recommendations described in Section 3 by considering a uniform load equal to 50kN/m and a safety factor equal to $1,4$. Therefore, the resulting bending moment $M_{S,max}$ is equal to $147,09$ kN $\cdot$ m and the shear force $V_{S,max} = 143,50$ kN.

The calculated longitudinal rebars and stirrups are described in Table 12 and Table 13, respectively.

Longitudinal rebars		
Number	4,00	
Diameter	20,00	mm
Total area	628,00	mm^2
Equivalent Perimeter	89,00	mm

Table 12: Longitudinal rebars properties

Stirrups		
Number	27,00	
Spacing	150,00	mm
Diameter	8,00	mm
Length	342,00	mm
Cross section area	100,00	mm^2
Equivalent Perimeter	63,00	mm

Table 13: Stirrups properties

The final configuration of the designed beam is illustrated in Figure 22.

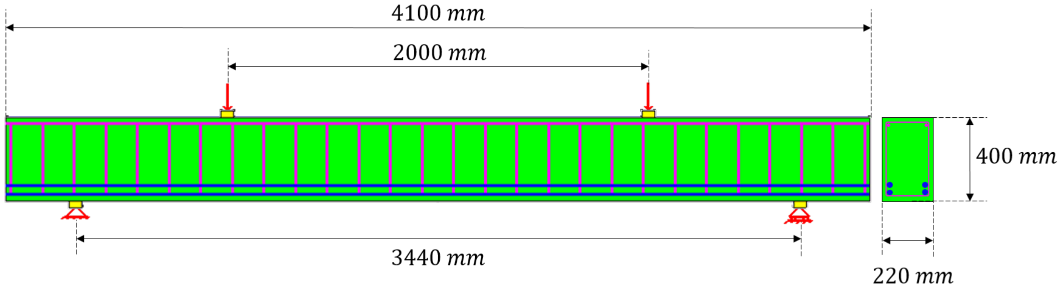


Figure 22: Beam designed following MC2010 recommendations: (i) green: concrete material, (ii) blue: tensile longitudinal rebars and (iii) pink: stirrups and compressive longitudinal bars

FE results

The designed beam is numerically simulated considering three situations: (i) bond-slip law using Eq. 1 (herein Bond Condition); (ii) good bond condition and (iii) all other bond conditions both based on the parameters of Figure 1. The results are shown in Table 14 and plotted in Figure 23.

Steel Stress	Theoretical Crack Width	Design Bond Cond. Crack Width	Good Bond Cond. Crack Width	All Other Bond Cond. Crack Width
MPa	mm	mm	mm	mm
100,77	0,010	0,016	0,028	0,060
126,77	0,056	0,032	0,046	0,103
159,25	0,113	0,058	0,073	0,161
174,82	0,141	0,069	0,084	0,187
196,16	0,179	0,090	0,099	0,218
225,69	0,231	0,115	0,116	0,244
249,99	0,274	0,129	0,130	0,261
275,28	0,319	0,142	0,145	0,278
300,80	0,364	0,153	0,160	0,000
Average error		17	15	-6

Table 14: Crack width and longitudinal rebar stress values considering different bond conditions

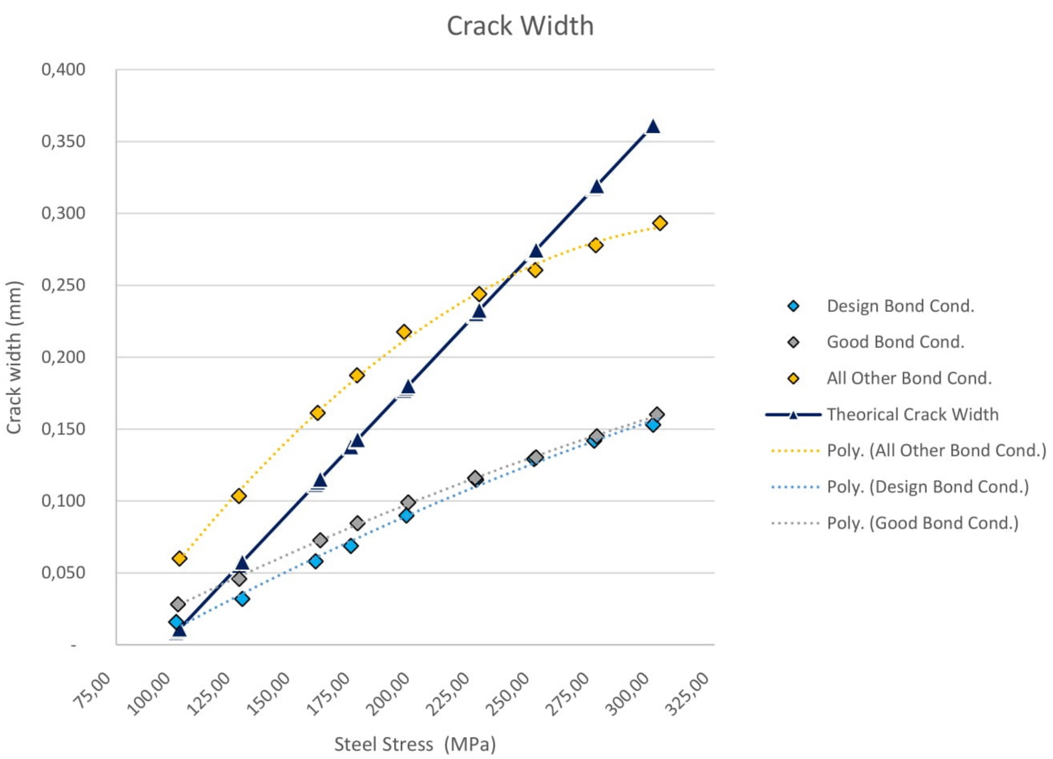


Figure 23: Crack width vs. steel stress values for different bond conditions

The first two bond-slip conditions (*design - bond condition* and *good bond condition*) underestimate the crack width predicted by MC2010, whereas *all other bond conditions* overestimated. In addition, the last simulation (*all other bond conditions*) presented the best result.

Figure 24 illustrates for the beam with *good bond conditions* the results in terms of load vs. displacement curve and the crack pattern evolution.

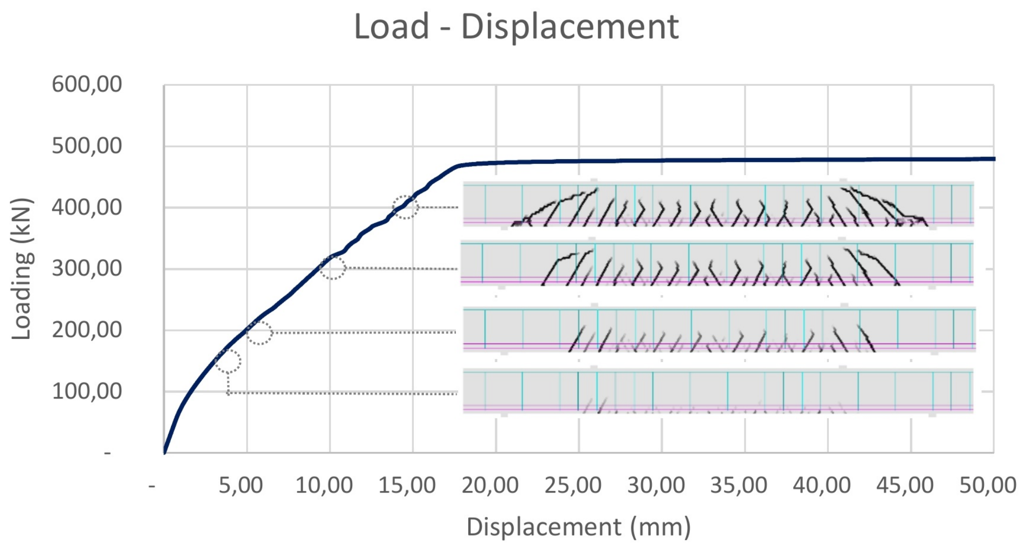


Figure 24: Load vs. displacement curve for good bond conditions

5 Conclusion

In this work the numerical strategy for modeling reinforced concrete beams developed by Bitencourt Jr. et al. (2018) is used for modeling the beam A1 experimentally tested by Vecchio and Shim (2004) and two beams 15-6-6-1 and 15-8-6-1 experimentally tested by Ma and Kwan (2015). In addition, a reinforced concrete beam was also designed according to *fib* Model Code 2010 (2013) and numerically simulated in order to compare the results in terms of crack width and Ultimate and Serviceability Limit States.

For the beam A1 the numerical and experimental results are in good agreement until the load of 370kN. After this point, the behavior is governed by crushing of the concrete and could be observed that the constitutive model adopted to represent the concrete behavior does not represent in this example the appropriate response. Thus, the numerical result is plotted until the peak load, just before the rupture by crushing of the concrete. In this example, the influence of the following parameters has been investigated: bond-slip law, fracture energy, ultimate strength of steel and tensile stress of steel.

For crack width prediction of reinforced concrete beams via finite element method a characteristic length equals to $l_{cs} = \sqrt{2A}$ as proposed by Rots (1988) is adopted, where A is the area of the concrete finite element. Adopting this approach, the better the results have been obtained in the numerical modeling of beam 15-6-6-1, with an error of approximately 13% regarding the experimental result. For the beam 15-8-6-1 the numerical results overestimated about 73% the experimental result, showing that the proposed formula should not be the most suitable for this case.

In the last example, for the beam designed according to *fib* Model Code 2010 (2013), the numerical results have been obtained for three different types of adherence: good bond conditions, all other bond conditions and using the formula given by Eq. 1. The better result has been obtained for all other bond conditions, while for the other two cases, the numerical results underestimated the designed ones.

References

L. A. G. Bitencourt Jr. *Numerical Modeling of Failure Processes in Steel Fiber Reinforced Cementitious Materials*. PhD thesis, Polytechnic School at the University of Sao Paulo, University of Sao Paulo, Brazil, 2015.

L. A. G. Bitencourt Jr., O. L. Manzoli, Y. T. Trindade, E. A. Rodrigues, and D. Dias-da-Costa. Modeling reinforced concrete structures using coupling finite elements for discrete representation of reinforcements. *Finite Elements in Analysis and Design*, 149: 32 – 44, 2018. ISSN 0168-874X. doi: https://doi.org/10.1016/j.finel.2018.06.004.

B. Bresler and A. C. Scordelis. Shear strength of reinforced concrete beams. (English). *J. Am. Concr. Inst*, 60(1):51–72, 1963.

M. Cervera, J. Oliver, and O. Manzoli. A rate-dependent isotropic damage model for the seismic analysis of concrete dams. (English). *Earthquake Engineering and Structural Dynamics*, 25(9):987–1010, 1996. doi: 10.1002/(SICI)1096-9845(199609)25:9⟨987::AIDEQE599⟩3.0.CO;2-X.

A. P. Clark. Cracking in reinforced concrete flexural members. *ACI Journal Proceedings*, 52(4):773 – 781, 1976. ISSN 0008-8846. doi: 10.1016/0008-8846(76)90007-7. URL `https://doi.org/10.14359/11653`.

N. E-Mezaini and E. Citipitioglu. Finite element analysis of prestressed and reinforced concrete structures. (English). *Journal of Structural Engineering*, 117(10):2851–2864, 1991. doi: 10.1061/(ASCE)0733-9445(1991)117:10(2851).

fib Model Code 2010. *fib Model Code for Concrete Structures 2010*. International Federation for Structural Concrete (*fib*), Berlin, Germany, 2013.

R. Gravina, L. Bitencourt Jr., and L. Meneghetti. Coupling finite elements to model steel to concrete bond. In N. Powers, D. M. Frangopol, R. Al-Mahaidi, and C. Caprani, editors, *Proceedings of the 9th International Conference on Bridge Maintenance, Safety and Management (IABMAS 2018)*, pages 2658 – 2663, London, United Kingdom, 2018. ISBN 9781138730458.

A. Hillerborg, M. Modéer, and P.-E. Petersson. Analysis of crack formation and crack growth in concrete by means of fracture mechanics and finite elements. *Cement and*

Concrete Research, 6(6):773 – 781, 1976. ISSN 0008-8846. doi: 10.1016/0008-8846(76)90007-7. URL `https://doi.org/10.1016/0008-8846(76)90007-7`.

F. Ma and A. Kwan. Crack width analysis of reinforced concrete members under flexure by finite element method and crack queuing algorithm. (English). *Elsevier*, 105(1):1–11, 2015. doi: https://doi.org/10.1016/j.engstruct.2015.10.012.

O. Manzoli and A. D. G. Oliver, J.; Huespe. A mixture theory based method for three-dimensional modeling of reinforced concrete members with embedded crack finite elements. (English). *Computers and Concrete*, 5(6):401–416, 2008. doi: 10.12989/cac.2008.5.4.401.

D. Ngo and A. C. Scordelis. Finite element analysis of reinforced concrete beams. (English). *ACI Journal*, 64(3):152–163, 1967.

J. G. Rots. *Computational modeling of concrete fracture*. PhD thesis, Delft University of Technology, Delft, Netherlands, 1988.

F. J. Vecchio and W. Shim. Experimental and Analytical Reexamination of Classic Concrete Beam Tests. (English). *Journal of Structural Engineering*, 2004. doi: 10.1061/(ASCE)0733-9445(2004)130:3(460).

A Procedure for modeling a beam using the FE platform

1. Observations:
   (a) Dimension: length mm, load N, strength N/mm;
2. With GID:
   (a) *New file*
   (b) Geometry:
      i. *Create line* [left side] > *command* [down side] > enter the points to delimite half of the beam (0,0) > *Esc* to apply changes
      ii. Create the surface for concrete: *Create NURBS surface* > select the beam lines > *Esc*
      iii. Create a surface geometry for the supports apply high value of the Young's Modulus (E) to avoid deformation;
      iv. Create the lines that represent the longitudinal rebars
      v. Create the lines that represent the stirrups. Obs.: for the rebar mesh make sure that the distance between the longitudinal rebars and the extreme node of a stirrups is great enough to not have coupling between the two element during the coupling procedure
   (c) *Data* [top side] > *Problem type* > *femoop* (must be installed/add previously to GiD problem type)
   (d) Boundary conditions:
      i. *Point Constraints* > *Y and X Constraints* > select the bottom point of the support that represent the pinned support > *Esc*
      ii. *Point Constraints* > *X Constraints* > select the bottom point of the support that represent the roller support > *Esc*
      iii. *Point force load* > *Y force* > enter the value and the sign of the applied force > select the node at the top of the symmetry-axis-line > *Esc*
      iv. To see the final configuration of the applied boundary conditions: *Draw* > *all conditions* > *include local axes*

(e) Material:

i. Select material model > Enter the properties > select line/surface to apply properties > *Esc*

ii. To see the final configuration of the applied materials: *Draw* > *all materials*

iii. Note that for the supports the *elastic isotropic model* with $E = 10^6$ and $\nu = 0.3$ was used.

(f) Problem data: femoop

i. Title: beam name

ii. Analusis Process: Structural

iii. Mechanical problem type: Non-linear Static

iv. Analysis type: Plane stress

v. Thickness: enter the problem thickness (for 2D analysis)

vi. Iteration order: 1x1x1

vii. Non-linear algorithm:

viii. Analysis maximum steps: 10000

ix. Analysis step factor: 0.0001 (inverse of the Analysis maximum steps)

x. Analysis tolerance: 0.001

xi. Analysis maximum iterations: 100

xii. Analysis print steps: 100

xiii. Analysis update: IMPL-EX2 (use if there is reinforcement)

xiv. [X] reinforced concrete with or without steel fibers:

- Note that the order of the entered areas are note necessary the same order of the material attribution. Make sure in the date file that areas and materials are correctly associated
- Given that the model is in 2D, if there is several bars on the same level of the cross-section, the equivalent area equal to the sum of all bars areas must be used. Same procedure is used for stirrups.

xv. Postproces Program Option: GiD (select which result you want to be generated by the solver)

xvi. [X] Print Mesh GiD

xvii. [X] Print Option Basic Results

xviii. [ ] Print Option All Results

xix. [X] Displacement

xx. [X] Load Factor

xxi. [ ] Stress Field

xxii. [ ] Strain Field

xxiii. [X] Damage Variable

xxiv. [ ] Yield Surface

xxv. [X] Reaction Support Evolution

xxvi. [X] Print Element Gauss Evolution: Element: (number of the element), Gauss-Point: 1

X Print Node Evolution: Node: (number of the node)

xxvii. *Accept* > *Close*

(g) Mesh

i. Rebars: *Mesh* > *mesh criteria* > *mesh* > *line* > select the lines that represent longitudinal rebars and stirrups

ii. Rebars: *Mesh* > *structured* > *line* > *assign size* > size of discretization > *assign* > select lines. Obs.: for the rebars mesh make sure that the distance between the longitudinal rebars and the extreme node of a stirrups is great enough to not have coupling between the two element during the coupling procedure.

iii. Concrete and support: *Mesh* > *unstructured* > *surface* > *assign size* > size of discretization > *assign* > select surface

iv. *Mesh* > *generate mesh*

B Matlab code to estimate crack width

Function: Crack Width breaklines

```
function matrix = CrackWidth(mflFilePath, resFilePath, mshFilePath, barId)
    %{
        PROGRAM STEP :
        0. CALCULATE THE AREA OF A TRIANGLE ELEMENT :
            A = 1/2 * det( [    1 X1 Y1;
                                1 X2 Y2;
                                1 X3 Y3;     ]);
        1. FIND THE BAR ELEMENT WITH THE GREATEST STRESS
        2. FIND THE COUPLED TRIANCLE ELEMENT
        3. CHECK OUT IF THE DOMAGE VALUE > 0.9 (FISSURE CRITERIA)
        4. IF TRUE,
        5.      GET CONCRETE PRINCIPAL STRAIN S1
        6.      CALCULATE CRACK WIDTH :
                    W = A^0.5*S1
        7. ITERATE ON THE FOADINGS
        8. PRINT GRAPH AND TABLE
    %}

    %VARS
    crackCriteria = 0.9;
    %OPEN FILES
    mflFile = OpenFile(mflFilePath); %GET COUPLING ELEMENT INFORMATION
    resFile = OpenFile(resFilePath); %GET DOMMAGE AND STRESS INFORMATION
    mshFile = OpenFile(mshFilePath); %GET ELEMENT GEOMETRY INFORMATION
    %BUILDING MATRIX
    couplingData = ExtractLibrary('COUPLING_ELEM_DATA',mflFile);
    domageData = ExtractLibrary('DOMAGE_DATA',resFile);
    stressData = ExtractLibrary('STEEL_STRESS_DATA',resFile);
    strainData = ExtractLibrary('CONCRETE_STRAINS_DATA',resFile);
    geometryData = ExtractLibrary('TRIANGLE_ELEM_GEOMETRY',mshFile);
    concreteData = ExtractLibrary('TRIANGLE_ELEM_DATA',mshFile);
    %GET NUMBER OF ITERATIONS
    N = size(stressData);
    N = N(2);
    %LOOP
    for j = 2 : N
        %GET ITERATION VALUE
        iter = stressData(1,j);
        if 1 == 1
            %FIND THE BAR ELEMENT WITH THE GREATEST STRESS
            [val, idx] = max(stressData(2:end,j));
            idSteelElem = stressData(idx+1,1)
            barId = 0;
%         else
            %FIND THE BAR ELEMENT WITH THE GIVEN ID
%             idSteelElem = stressData(barId+1,1);
        end
        %FIND THE COUPLED TRIANCLE ELEMENT
        couplingData(:,end)
        idNodes = couplingData(couplingData(:,end) == idSteelElem,5:7)
        [tf, idTriElem]=ismember(idNodes,concreteData(:,2:4),'legacy')
        idTriElem = idTriElem(1)
        %CHECK OUT IF THE DOMAGE VALUE > 0.9 (FISSURE CRITERIA)
        domageValue = domageData(domageData(:,1) == idTriElem,2);
        if domageValue >= crackCriteria
            %CALCULATE THE AREA OF A TRIANGLE ELEMENT
            nodes = concreteData(idTriElem,2:4);
            [X1,Y1] = geometryData(geometryData(:,1)==nodes(1),2:end);
            [X2,Y2] = geometryData(geometryData(:,1)==nodes(2),2:end);
            [X3,Y3] = geometryData(geometryData(:,1)==nodes(3),2:end);
            A = 1/2 * det( [    1 X1 Y1;
                                1 X2 Y2;
```

```
                                    1 X3 Y3;      ]);
                %GET CONCRETE PRINCIPAL STRAIN S1
                SXX = strainData(strainData(:,1)==idTriElem,iter,1);
                SYY = strainData(strainData(:,1)==idTriElem,iter,2);
                SXY = strainData(strainData(:,1)==idTriElem,iter,3);
                vp = eig([SXX SXY;
                            SXY SYY]);
                S1 = vp(1);
                %CALCULATE CRACK WIDTH
                W = (2*A)^0.5*S1;
            end
            %OUTPUT DATA
            array = [iter A idSteelElem val idTriElem domageValue S1 W];
            if j == 2
                matrix = array;
            else
                matrix = [matrix; array];
            end
        end
    end
```

Function: Extract specific data breaklines

```
function extraction = ExtractLibrary(caseName,txtMatrix)
    %USE THE FUNCTION 'EXTRACT' FOR SPECIFIC CASES OF THE CRACK WIDTH
    %STUDY.
    switch caseName
        case 'COUPLING_ELEM_DATA'
            fistLine = '$----------------------------------------------------------------------GENERAL_DATA';
            delta_f = 3;
            lastLine = '$--------------------------------------------------------------------------SETS_DATA';
            delta_l = 2;
            formatSpec = '%d %d %d %d %d %d %d %d';
            sizeA = [8 Inf];
%            extraction = Extract(txtMatrix,fistLine,delta_f,lastLine,delta_l,formatSpec,sizeA);
            extraction = ExtractSscanf(txtMatrix,fistLine,delta_f,lastLine,delta_l,formatSpec,sizeA);
        case 'TRIANGLE_ELEM_DATA'
            fistLine = 'MESH dimension 2 Elemtype Triangle  Nnode 3';
            delta_f = 2;
            lastLine = 'end elements';
            delta_l = 1;
            formatSpec = '%d %d %d %d %d';
            sizeA = [5 Inf];
            indexList = ExtractLoop(fistLine,lastLine,txtMatrix);
            for k = 1 : length(indexList)
                i = indexList(k,2);
                j = indexList(k,3);
                intermediary = Extract(txtMatrix(i:j),'Elements',delta_f,'end elements',delta_l,formatSpec,sizeA);
                if k == 1
                    extraction = intermediary;
                else
                    extraction = [extraction; intermediary];
                end
            end
         case 'TRIANGLE_ELEM_GEOMETRY'
            fistLine = 'MESH dimension 2 Elemtype Triangle  Nnode 3';
            delta_f = 2;
            lastLine = 'end elements';
            delta_l = 1;
            formatSpec = '%d %f %f';
            sizeA = [3 Inf];
            indexList = ExtractLoop(fistLine,lastLine,txtMatrix);
            for k = 1 : length(indexList)
                i = indexList(k,2);
                j = indexList(k,3);
                intermediary = Extract(txtMatrix(i:j),'Coordinates',delta_f,'end elements',delta_l,formatSpec,sizeA);
                if k == 1
                    extraction = intermediary;
                else
                    extraction = [extraction; intermediary];
                end
            end
        case 'DOMAGE_DATA'
            fistLine = 'Result "Damage //Tensile" "Load Analysis"';
            delta_f = 3;
            lastLine = 'End Values';
            delta_l = 1;
            formatSpec = '%d %f';
            sizeA = [2 Inf];
            indexList = ExtractLoop(fistLine,lastLine,txtMatrix);
            for k = 1 : length(indexList)
                id = indexList(k,1);
                i = indexList(k,2);
                j = indexList(k,3);
                intermediary = Extract(txtMatrix(i:j),fistLine,delta_f,lastLine,delta_l,formatSpec,sizeA);
                if k == 1
                    intermediary = [0 id; intermediary];
                    extraction = intermediary;
```

```
                else
                    intermediary = [id; intermediary(:,end)];
                    extraction = [extraction intermediary];
                end
            end
        case 'STEEL_STRESS_DATA'
            fistLine = 'Result "Stresses on Elements Linear" "Load Analysis"';
            delta_f = 3;
            lastLine = 'End Values';
            delta_l = 1;
            formatSpec = '%d %f';
            sizeA = [2 Inf];
            indexList = ExtractLoop(fistLine,lastLine,txtMatrix);
            for k = 1 : length(indexList)
                id = indexList(k,1);
                i = indexList(k,2);
                j = indexList(k,3);
                intermediary = Extract(txtMatrix(i:j),fistLine,delta_f,lastLine,delta_l,formatSpec,sizeA);
                if k == 1
                    intermediary = [0 id; intermediary];
                    extraction = intermediary;
                else
                    intermediary = [id; intermediary(:,end)];
                    extraction = [extraction intermediary];
                end
            end
        case 'CONCRETE_STRAINS_DATA'
            fistLine = 'Result "Strains//On Gauss Points" "Load Analysis"';
            delta_f = 3;
            lastLine = 'End Values';
            delta_l = 1;
            formatSpec = '%d %f %f %f %f';
            sizeA = [5 Inf];
            indexList = ExtractLoop(fistLine,lastLine,txtMatrix);
            for k = 1 : length(indexList)
                id = indexList(k,1);
                i = indexList(k,2);
                j = indexList(k,3);
                intermediary = Extract(txtMatrix(i:j),fistLine,delta_f,lastLine,delta_l,formatSpec,sizeA);
                if k == 1
                    M = [0 id; intermediary(:,1:2)];
                    M = cat(3,M, [0 id; intermediary(:,1) intermediary(:,3)]);
                    M = cat(3,M, [0 id; intermediary(:,1) intermediary(:,4)]);
                    extraction = M;
                else
                    M = [id; intermediary(:,2)];
                    M = cat(3,M, [id; intermediary(:,3)]);
                    M = cat(3,M, [id; intermediary(:,4)]);
                    extraction = [extraction M];
                end
            end
        otherwise
            disp('Specified extraction impossible')
    end
end
```

Function: Extract data routine breaklines

```
function data = ExtractLoop(header_,footer_,file)
    %IN A GIVEN FILE IF YOU WANT TO EXTRACT SUCESSIVELY INFORMATIONS THE
    %BEGINNING WITH SIMILARY HEADERS AND ENDING WITH SIMILARY HEADERS. THE
    %OUTPUT IS A MATRIX : [ID FIRSTLINE LASTLINE; ...]

    %VARS
    headers = find(~cellfun(@isempty,strfind(file,header_)));
```

```
    footers = find(~cellfun(@isempty,strfind(file,footer_)));
    list = zeros(length(headers),3);
    for i = 1 : length(headers);
        %GET THE Ith VALUE OF HEADERS LIST
        k = headers(i);
        %GET THE ENTIRE LINE OF THE K LINE IN THE GIVEN FILE
        headerLine = file(k);
        %EXTRACT FROM THIS LINE THE ITERATION STEP VALUE
        txt = regexp(headerLine,'\d+','match');
        txt = txt{1}(1);
        id = str2double(txt);
        %FIND THE FOOTERS THAT ARE AFTER THE Ith HEADER
        footers_i = find(footers > k);
        %APPEND DATA
        list(i, 1) = id;
        list(i, 2) = k;
        list(i, 3) = footers(footers_i(1));
    end
    data = list;
end
```

Function: Extract data from file breaklines

```
function extraction = ExtractSscanf(txtMatrix,fistLine,delta_f,lastLine,delta_l,formatSpec,sizeA)
    %GET THE INDEX OF THE FIRST LINE TO EXTRACT
    i = find(~cellfun(@isempty,strfind(txtMatrix,fistLine)));
    if length(i)>1
        %IF VARIOUS ELEMENTS WERE FOUND, KEEP ONLY THE FIRST ONE
        i = i(1);
    end
    i = i + delta_f;
    %GET THE INDEX OF THE LAST LINE TO EXTRACT
    j = find(~cellfun(@isempty,strfind(txtMatrix,lastLine)));
    if length(i)>1
        j = j(1);
    end
    j = j -delta_l;
    %RENAME txtMatrix(i:j)
    mat = txtMatrix(i:j);
    %GET TXTMATRIX(i:j) SIZE
    N = size(mat);
    N = N(1);
    %GET NUMBER OF COLUMN OF THE OUTPUT MATRIX
    M = sizeA(1);
    %LOOP
    for k = 1 : N
        %EXTRACT STRING DATA FROM ROW k
        row = char(mat(k,:));
        %CONVERT DATA IN INT ARRAY
        array = sscanf(row,formatSpec,sizeA)';
        %GET THE SIZE OF THE ARRAY
        P = size(array);
        P = P(2);
        %IF P<M PUT 0s AT THE END OF THE ARRAY
        if P < M
            array = [array zeros(1,M-P)];
        end
        %APPEND OUTPUT MATRIX
        if k == 1
            matrix = array;
        else
            matrix = [matrix; array];
        end
    end
    %EXTRACT FORM matrix THE ROWS THAT DO NOT HAVE 0 ON THE LAST COLUMN
    extraction = matrix(matrix(:,end)>0,:);
end
```

C Poster

MODELO NUMÉRICO DE ABERTURAS DE FISSURAS

Professor Orientador : Luís A. G. Bitencourt Jr.
Estudante : Alexandre J. Tokka

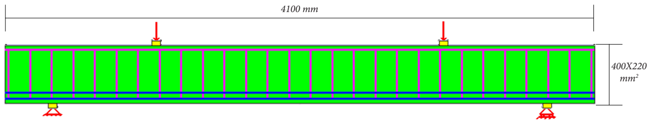
1. INTRODUÇÃO

Modelar com precisão vigas de concreto armado - cujo comportamento é não linear - requer, primeiro, modelos de concreto e de reforços de aço adequado e, segundo um modelo capaz de representar a interação entre os dois materiais. Para o aço foi adotado um comportamento elástico-plástico, para o concreto o modelo de *Cervera* [1] e para a interação o modelo da *fib Model Code* [2].

O propósito deste trabalho de formatura II, foi, a partir de um programa em matlab já desenvolvido para resolver modelos com elementos finitos, implementar um código para estimar as abertura de fissuras baseando-se na pesquisa da *Politecnico di Milano* [3]. Os resultados numéricos são comparados resultados analíticos de uma viga viga dimensionada com a *fib Model Code*.

2. METODOLOGIA

O aço se modela facilmente adotando um comportamento elástico-plástico. Modelar concreto é mais subtil, vários modelos foram desenvolvidos para representar o melhor possível dos comportamentos diferentes em relação a compressão ou tração. A fib Model Code propôs em seguida um modelo para simular a interação entre os dois materiais e estimar a aberturas de fissuras em vários estados de carregamento da viga.



	Geometria			Propriedades	
Concreto					
Cobrimento	0,025	*m*	Categoria	C25	
			Ec	28,00	*GPa*
Barras Longitudinais					
Numero	4		Categoria	CA50	*GPa*
Diametro	20,00	*mm*	Es	200,00	
Estribos					
Espacemento	150,00	*mm*	Categoria	CA50	
Diametro	8,00	*mm*	Es	200	*GPA*

Após escolher as dimensões da viga, foi (1) dimensionada as barras longitudinais e estribos em função das recomendações das *fib Model Code.*

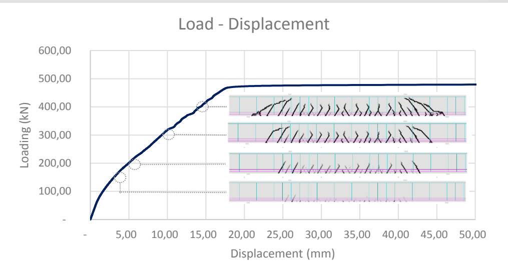
A mesma fonte permitiu (2) uma estimação da abertura das fissuras. O cálculo depende de vários fatores, tais como: o da tensão na barra longitudinal onde o concreto rompeu, a taxa de aço, as resitências do concreto e do aço e outros parâmetros que dependem, entre outros, da interação concreto-aço.

Após a simulação numérica foi utilizado a formulação da *Politecnico di Milano* para estimação da abertura de fissuras. Essa formula é função da área do elemento de concreto onde há uma fissura (cf. elementos pretos na figura Load - Displacement) e da deformação principal.

3. RESULTADOS

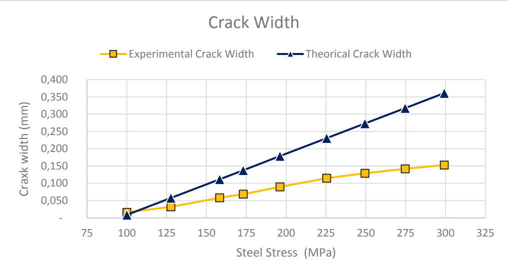
A viga foi discretizada em uma dezena de milhares de elementos de 20 mm de comprimento. A cada passo a viga recebe um carregamento adicional. Assim, podemos conhecer as características (deformação e tensão) de vários passos do carregamento. A figura *Load-Displacement* mostra justamente a flecha da viga para vários carregamentos até a ruptura caracterizada pela porção quase-horizontal da curva.

Para vários pares de força aplicada-deslocamento foi ilustrado o estado da viga. A visualização permite enxergar a proliferação das fissuras. As fissuras cuja tendência é relativamente vertical são devidas a flexão enquanto as diagonais são devidas à força cortante.



A tabela a seguir mostra a comparação entre os resultados analíticos e numéricos. O gráfico Crack Width mostra os mesmos resultados. A curva amarela ilustra os resultados numéricos (ou seja, coma formulação da *Politecnico di Milano),* *e a azul, os resultados numericos (do fib Code Model).*

Steel Stress	Theorical Crack Width	Numerical Crack Width	Error
MPa	*mm*	*mm*	*%*
100,04	0,009	0,016	-81%
127,67	0,058	0,032	44%
158,28	0,112	0,058	48%
173,07	0,138	0,069	50%
196,19	0,179	0,090	50%
225,41	0,230	0,115	50%
249,51	0,273	0,129	53%
274,74	0,318	0,142	55%
299,32	0,361	0,153	58%



A diferença entre os valores numéricos e analíticos é cada vez mais importante quando o valor da tensão na barra longitudinal vai aumentando. Na média o erro é de mais ou menos 50% o que pode parecer importante. Entretanto outros métodos numéricos mostram uma diferença relativamente significativas, como por exemplo no artigo de *F.J. Ma, A.K.H. Kwan* da *University of Hong Kong.*

Além disso o mais importante para deduzir e concluir de maneira pertinentes se os modelos são adequados seria ter a possibilidade de ter resultados experimentais de laboratório. O ideal seria ter experimentos para várias vigas com características diferentes.

A diferença pode ser explicada também pela presença de estribos. A contribuição dos estribos na formulação da fib Model Code não é tão precisa como a contribuição das barras longitudinais, porém eles podem ter uma contribuição.

4. BILBLIOGRAFIA

[1] O. Manzoli M. Cervera J. Oliver. «A rate-dependent isotropic damage model for the seismic analysis of concrete dams. (English)». In: *Earthquake Engineering and Structural Dynamics* 25.9 (1996), pp. 987{1010. doi: 10.1002/(SICI)1096-9845(199609) 25:9<987::AIDEQE599>3.0.CO;2-X.

[2] Special Activity Group 5 :Walraven et al. The *fib Model Code for Concrete Structures 2010*. Ernst Sohn, 2010. isbn: 978-3-433-03061-5. *(1)7.3.3.3Memberswithshearreinforcement,(2)7.6.4.3Limitationofcrackwidth*

[3] Marco Di Prisco, Matteo Colombo e Isabella G. Colombo, *The Role Of The Structural Characteristic Length In FRC Structures*, Politecnico di Milano, ITALY, DOI 10.21012/FC9.308