Numerical model of high-strength reinforced concrete columns subjected to the ultimate limit state of instability

Nicolle L. D. Guerra[1], Ricardo Carrazedo[1]

[1]*Dept. of Structural Engineering, São Carlos School of Engineering, University of São Paulo*
Av. Trabalhador São Carlense, 400, 13566-59, São Carlos, SP, Brazil
n.guerra@usp.br, carrazedo@usp.br

Abstract. High-strength reinforced concrete columns have greater tendency to slenderness and to higher compressive axial loads. Both aspects contribute to a significant magnified moment, and to a potential context to lead those columns to instability failure. Although the second order effects can be determined by simplified methods in some circumstances, the general method allows to identify the Ultimate Limit State of Instability clearly and to evaluate the second order effects precisely. Thus, it is proposed the employment of general method in two distinct two-dimensional models on finite element Abaqus software to obtain a load-deflection diagram. The first model is a plane stress element with embedded reinforcement, with the Concrete Damaged Plasticity representing the constitutive behavior of the concrete, and the second model adopts a beam element with the rebar embedded into concrete section using Abaqus/Standard, with the Cast Iron Plasticity as a simplified representation of the concrete. The adequacy of the models is validated by experimental data from literature. The results have shown that two-dimensional model is effective to simulate the uniaxial bending High-strength concrete column subjected to the Ultimate Limit State of Instability. In addition, the most simplified model, which adopts beam element with rebar and neglects tensile strength, provides suitable load-deflection diagram. Therefore, this model is recommended to represent such analysis.

Keywords: Instability. High-strength concrete. Slenderness. Finite Element.

1 Introduction

High-strength concrete is widely used when it is required to design reduced cross-sections columns with a satisfying carrying capacity. As a result, those columns tend to be slender – which increase the deflection – and to have high compressive axial load. Thus, the consequence is a significant magnified moment, which may lead to a stability failure when occurs the loss of equilibrium of the deformed structure, that is, when the column has already failed by instability although the section has not reached the ultimate strength. Therefore, it is important to represent a structural analysis in the deformed configuration of the structure, so the geometric nonlinearity enable the establishment of the second orders effects, which is also affected by the material nonlinearity.

Nonetheless, when a simplified method is proposed to introduce the second order effects on high-strength concrete columns, it neglects the tendency of these columns to the Ultimate Limit State of Instability. Thus, besides of proposing a refined analyses by implementing the geometric and material nonlinearity, the general method data allows to predict the kind of failure occurs on the column, as proposed on MacGregor et al [1] interaction diagrams. Henceforth, this paper aims to generate a model in the finite element Abaqus software to simulate columns with a suitable load-deflection diagram when compared with the tests results.

To achieve this objective, two different approaches to simulate the reinforced concrete column is accomplished. The first model is a plane stress element with embedded reinforcement, with the Concrete Damaged Plasticity representing the constitutive behavior of the concrete and the elastoplastic describing the steel behavior. The second model adopts a beam element with the rebar embedded into concrete section using Abaqus/Standard, and although the constitutive model of steel remains the same, it was adopted Cast Iron Plasticity as a simplified representation of the concrete. Even though it is usual to neglect concrete tensile strength in design, it is considered on each model due to convergence problems. The adequacy of the models is validated by the experimental simulations from Kim and Yang [3], Germain and Espion [4] and Pallarés et al [5], in which all columns had reached instability failure according to the authors. The efficiency of the models is rated by analyzing preprocessing and processing effectiveness.

2 Finite element analysis

Preprocessing a numerical simulation involves the definition of the geometry, material properties, finite elements mesh, steps, and boundary conditions. Depending on the purpose of the analysis results, it is possible to create simplified models to predict the problem behavior. As the load-deflection diagram provides significant information to consider the second-order effects and to evaluate the possibility of occurrence of the Limit State of Instability, two-dimensional models offer an adequate analysis.

Regardless of the element type or material property, columns subjected to uniaxial bending forces must have load eccentricity on model. For this, reference points connected to the ends of the columns are used, and it is where the boundary conditions are applied. Besides of displacement restraints, it is possible to apply on that point a displacement capable of representing a load. By applying a displacement to the column instead of a load, it is possible to obtain the column's equilibrium path, that is, a load-deflection diagram that provides the peak load and its post peak behavior. Moreover, it is important to assign the restraints on initial step and to create a step with geometric nonlinearity to apply displacements in increments.

2.1 Plane stress element with embedded reinforcement

The four-node plane stress element type with linearly interpolation and reduced integration with automatic hourglass control (CPS4R in Abaqus notation), illustrated on Fig. 1, is suitable to simulate a concrete column. Considering the order of magnitude of the experimental columns, the mesh sizes adopted are 20 mm x 20 mm. In addition, the thickness section is defined by a solid section, which corresponds to the cross-section dimension perpendicular to the eccentricity direction. The reinforcement steel, however, is well represented by a two-node linear displacement element type (T2D2 in Abaqus notation), illustrated on Fig. 1, employing a 20 mm mesh size and truss section.

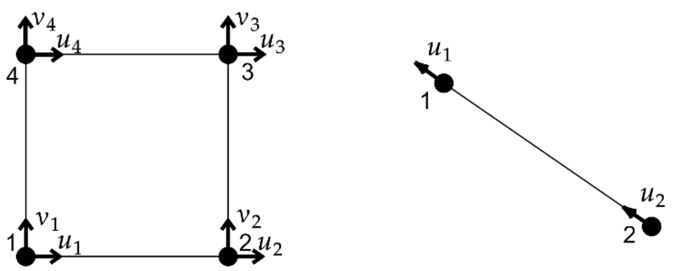

Figure 1. CPS4R (left) and T2D2 (right) element types

The constraint between both element is accomplished when performed the embedment of the reinforcement to the concrete column. It is used a rigid body at both ends, so the motion of the end column would be governed by the motion of the reference point, with tie nodes, which have both translational and rotation degrees of freedom.

2.2 Beam element with rebar

The classical approach to beam theory, Euler-Bernoulli assumption, is considered on B23 element type, which uses cubic interpolation. The illustration of the beam element is exhibited on Fig. 2, with six degrees of freedom. The mesh size adopted is also 20 mm and the dimensions are defined by beam sections.

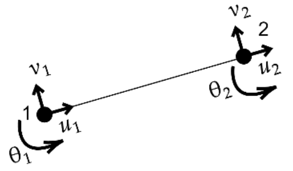

Figure 2. B23 element type

The implementation of the reinforcement steel, however, must be modeled as discrete rebar embedded into the concrete section in Abaqus/Standard, as illustrated on Fig. 3. Both lines indicated on parenthesis must be written on part instance level to represent each rebar. Consequently, the number of codes is the same number of reinforcement layers, due to two-dimensional model. Also, it is used a MPC constraint to provide a rigid beam between the reference point and the end column, to constrain the displacement and rotation to one node.

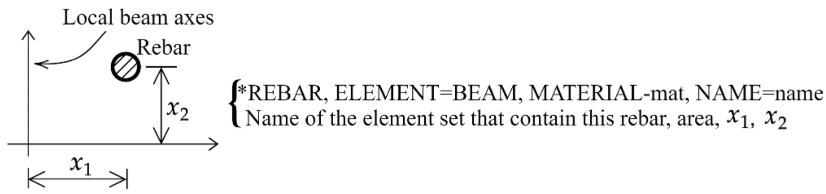

Figure 3. Rebar in beam elements

2.3 Constitutive models for concrete and reinforcement

The constitutive models describe the material mechanical behavior; thus, it is important to choose it properly. While the reinforcement has almost an Elastic-Perfectly Plastic behavior (constitutive model chosen to represent the steel, both in truss section and beam section), the concrete has a different nonlinear material. Despite of Concrete Damaged Plasticity being a consolidated model on Abaqus software, it demands rigorous parameters calibration. Thus, a simplified analysis could be adopted by Cast Iron Plasticity when suitable. In both cases is necessary to define a stress strain diagram to describe the concrete strength. A stress-train relation for nonlinear structural analysis proposed by *fib* Model Code 2010 (MC2010) [6] is illustrated on Fig. 4. The code also gives the eq. (1) to determine the elastic modulus of the concrete and the eq. (2) to determine the tensile strength of concretes above 50 MPa, and the compressive strength (f_{cm}) must be written in MPa.

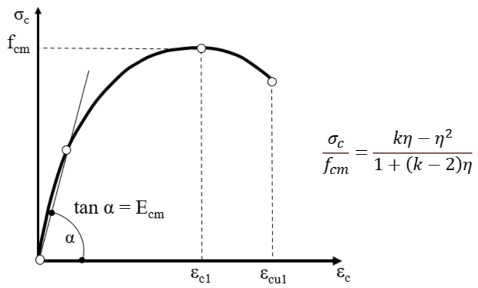

Figure 4. Stress-strain diagram given by *fib* Model Code 2010 (MC2010) [6]

$$E_{cm} = 21500 \cdot \left(\frac{f_{cm}}{10}\right)^{\frac{1}{3}}. \quad (1)$$

$$f_{ct} = 2.12 \cdot \ln(1 + 0.1 f_{cm}). \quad (2)$$

It is important to emphasize, however, that Ibracon [7] allows the use of this stress-strain diagram, but with some compatibilities. As concrete in this paper have higher strength as allowed in ABNT NBR6118:2014 [8] and the aim of this study is to validate the numerical simulations, *fib* Model Code 2010 (MC2010) [6] data was used.

Concrete Damaged Plasticity represents the inelastic behavior of concrete by combining the concept of isotropic damaged elasticity with isotropic tensile and compressive plasticity, according to Simulia [2]. It is a modification of the Drucker-Prager strength hypothesis and had been modified by Lubliner et al [9] and Lee and Fenves [10], which enable the failure surface in the deviatoric cross section to be modified by a K_c parameter. Moreover, the state of the material is described by the eccentricity, which adjust the shape of the plane's meridians in the stress space, by the ratio of the strength in the biaxial state to the strength in the uniaxial state (fb0/fc0) and by the dilatation angle, the concrete internal friction angle. The default values arranged by Kmiecik and Kaminski [11] on their research paper are shown on Tab. 1, though with different viscosity parameter, to describe the values used on the development of the numerical simulations from this work.

Table 1. Concrete Damaged Plasticity parameters

Dilatation angle	Eccentricity	fb0/fc0	K_c	Viscosity parameter
36°	0.1	1.16	2/3	10^{-5}

According to Kmiecik and Kaminski [11], as the viscosity parameter collaborates with solution convergence by regularizing the constitutive equations, it is necessary to match this value with its influence on problem solution. When the tensile strength was close to zero, it was onerous determine a viscosity parameter. Hence, on Concrete Damaged Plasticity models the tension strength of the concrete was considered. Thus, this constitutive model was applied on solid section with plane stress element, resulting a more complete numeric simulation.

To simplify the analyses, although cast iron plasticity is a constitutive model to simulate the elastoplastic behavior of gray cast iron, it is suitable to represent the concrete behavior under some situations, due to the possibility to adopt different yield strength in tension and in compression. The plastic Poisson's ratio, equal to the absolute value of the ratio of the transverse to longitudinal plastic strain under uniaxial tension, becomes insignificantly in conditions from this paper. Therefore, Cast Iron Plasticity was applied on beam section with B23 element type, resulting on a simplified numeric simulation, with concrete tensile strength close to zero.

3 Experimental data

Numerous papers available on literature which study slender high-strength concrete columns contribute with experimental data. On Kim and Yang [3], Germain and Espion [4] and Pallarés et al [5] research, some tested columns have had reached their maximum load by instability. Therewith, one column from each author was select to be performed on Abaqus software and validated. The chosen columns were those that have had failed by instability and that the authors had provided the load-deflection diagram, being just Germain and Espion [4] column tested with displacement control. To illustrate the columns parameters, Fig. 5 is displayed bellow.

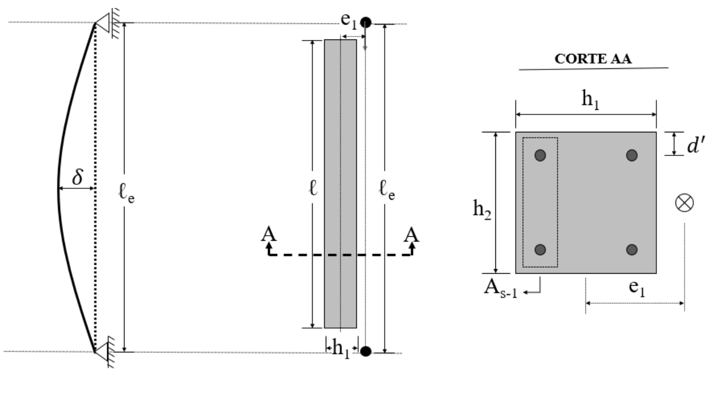

Figure 5. Illustration of columns parameters

The three columns are pin-ended and subjected to uniaxial bending, which results on single curvature columns with larger deflection (δ) in the mid-height. When test's setup uses plates on columns end, the effective length (ℓe) is different from column length (ℓ). The cross-section dimension (h_1) is parallel to eccentricity (e_1), and the thickness of the cover plus the half of bars diameters (d') is uniform along cross-section dimension. Furthermore, as simulation is accomplished on two-dimensional finite model, the sum of steel area which represents a layer (A_{s-1}) is an important information.

Data from tests are presents on Tab. 2. In addition to the geometric information mentioned above, Tab. 2 shows columns slenderness (λ), given by the ratio between $\sqrt{12} \cdot \ell_e$ and h_1. The compressive strength obtained on papers (f_{cm0}) from specimens with different size than the cylinder specimen with diameter of 150 mm and height of 300 (f_{cm1}) mm, were converted by a factor given by Montoya [12]. This parameter, along with the elastic modulus of concrete (E_c) assures the development of the stress strain diagram from *fib* Model Code 2010 (MC2010) [6] to represent the compressive behavior of the concrete.

Tabela 2. Experimental data

References	ℓ (mm)	ℓ_e (mm)	h_1 (mm)	h_2 (mm)	e_1 (mm)	λ	A_{s-1} (mm^2)	d' (mm)	f_{cm-0} (MPa)	f_{cm-1} (MPa)
Kim and Yang [3]	2400	2420	80	80	24	104.8	62.3	18.2	86.2	83.6
Germain and Espion [4]	3780	4380	180	180	10	72.7	226.2	37.0	85.9	85.9
Pallarés et al [5]	2000	2000	200	100	40	34.6	157.1	24.0	107.0	107.0

Therefore, it was simulated on Abaqus software three distinct columns in both proposed model: plane stress element with embedded reinforcement, in which the concrete constitutive model was Concrete Damaged Plasticity and beam element with rebar, in which the concrete constitutive model was Cast Iron Plasticity. On both models, steel constitutive model was Elastic-Perfectly Plastic. After that, six load-deflection diagrams were obtained from mid-height lateral deflection and columns load reaction to be validated with experimental data.

4 Results

The load-deflection diagrams obtained from numerical simulation on Abaqus software are given and discussed below. It is illustrated the plane stress element with embedded reinforcement model considering tensile strength (Plane TS), the beam element with rebar model (Beam), and the experimental data (EXP). Load-deflection diagrams from Kim and Yang [3] columns data are shown on Fig. 6, and it is possible to observe an extra result, which represents the first model with concrete tensile strength close to zero (Plane). The load-deflection diagrams show a suitable result when adopted the beam element with rebar model, but not when adopted plane stress element with embedded reinforcement model. Thus, Plane model shows a dissonant result when compared with Plane TS model, but suitable to experimental data. This unusual result, given a complete model represented by Plane TS, demonstrate the importance of not taking tensile strength into account to have safer results.

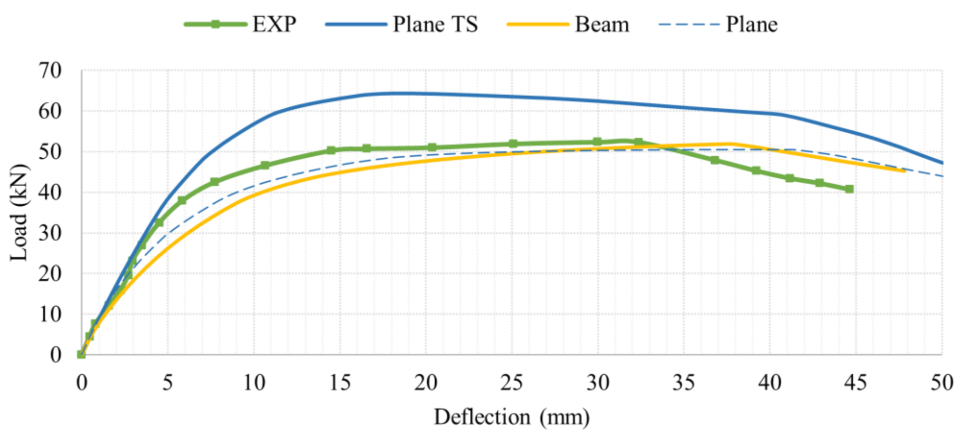

Figure 6. Load-deflection diagram compared with 100-H2-1 column from Kim and Yang [3]

Another column, with Germain and Espion [4] data (GE-2005), is shown on Fig. 7. The load-deflection diagrams show a suitable result when adopted the beam element with rebar model and the plane stress element with embedded reinforcement model. Even with Plane TS model considering the tension strength on its constitutive model, while it was almost neglected on Plane model, the load-deflection was similar on both models and gave a good prediction of the column behavior. As experimental tests had a displacement control, it would be consistent to have results close to a numerical simulation.

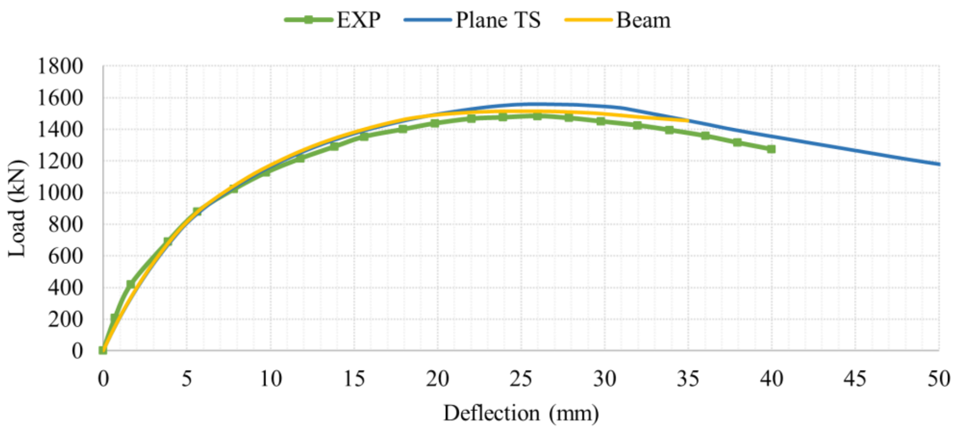

Figure 7. Load-deflection diagram compared with A-1/18-R2 column from Germain and Espion [4]

Finally, Load-deflection diagrams from Pallarés et al [5] ae shown on Fig. 8. The load-deflection diagrams show a suitable result when adopted the beam element with rebar model, but even more when adopted plane stress element with embedded reinforcement model.

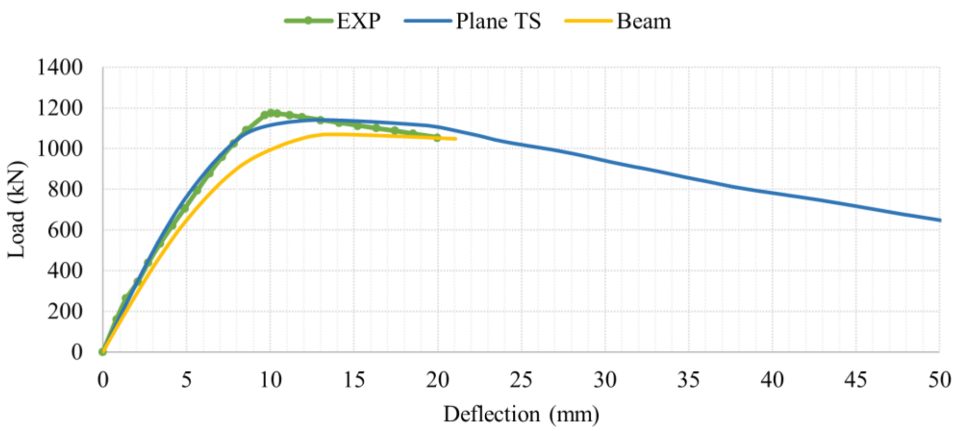

Figure 8. Load-deflection diagram compared with λ20_α0_ζ 2 column from Pallarés et al [5]

5 Conclusion

After simulating two distinct models of uniaxial bending high-strength concrete column subjected to the Ultimate Limit State of Instability in Abaqus software and validate load-deflection diagrams with experimental data from Kim and Yang [3], Germain and Espion [4] and Pallarés et al [5], it is noted that second order effects are satisfactory computed by using two-dimensional finite element model as general method. Besides, the defined load-deflection diagram can be used to predict the type of the column failure by using interaction diagrams as proposed by MacGregor et al [1]. Nonetheless, plane stress element with embedded reinforcement model and Concrete Damaged Plasticity has a higher time preprocessing cost due to parameters calibration, mostly viscosity parameter, which must match the value with its influence on the problem solution. When poorly adopted, may raise processing time and result in wrong solutions. It is even more evident when reducing tension strength, due to the complex stiffness matrix as a result of model embedded.

Thus, beam element model with Cast in Place Plasticity is a simple model which can give suitable results of load-deflection diagram. Although it is necessary to use Abaqus/Standard to apply the rebar, it is an easy tool to allow addition of integral points in section to represent the reinforcement. Another issue is the impossibility to see the reinforcement behavior as a separate element from the concrete column, which is not a problem when the aim of the general method is to understand the column behavior by the load-deflection diagram. Consequently, beam element model with Cast in Place Plasticity is a recommended model to simulate in Abaqus software a uniaxial bending high-strength concrete column subjected to the Ultimate Limit State of Instability and obtain its load-deflection diagram, and it is safer to neglect tensile strength.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] J. G. MacGregor, J. E. Breen and E. O. Pfrang, "Design of slender concrete columns". *ACI journal proceedings*, vol. 67, n. 2, pp. 6–28, 1970.
[2] Simulia. Abaqus analysis user's manual. Dassault Systèmes, version 6.11.
[3] J. K. Kim and J. K. Yang, "Bucking behavior of slender high strength concrete columns". *Engineering Structures*, vol. 17, n. 1, pp. 39–51, 1995.
[4] O. Germain and B. Espion, "Slender high-strength RC columns under eccentric compression". *Magazine of concrete research*, vol. 57, n. 6, pp. 361–370, 2005.
[5] L. Pallarés, J. L. Bonet, P. F. Miguel and M. A. Fernández Prada, "Experimental research on high strength concrete slender columns subjected to compression and biaxial bending forces". *Engineering Structures*, vol. 30, n. 1, pp. 1879–1894, 2008.
[6] Fédération internationale du béton (*fib*). *fib* Model Code for concrete structures 2010. Lausanne, 2010.
[7] Ibracon. *ABNT NBR 6118:2014 – Comments and application examples*. Ibracon, 2015.
[8] Associação Brasileira de Normas Técnicas (ABNT). *NBR 6118: Design of concrete structures – Procedure*. Rio de Janeiro, 2014 (in Portuguese).
[9] J. Lubliner, J. Oliver, S. Oller, and E. Oñate, "A plastic-damage model for concrete". *International Journal of Solids and Structures*, vol. 25, n. 3, pp. 299–329, 1989.
[10] J. Lee and G. L. Fenves, "Plastic damage model for cyclic loading of concrete structures". *Journal of Engineering Mechanics*, vol. 124, n. 8, pp. 892–900, 1998.
[11] P. Kmiecik and M. Kaminski, "Modellig of reinforced concrete structures and composite structures with concrete strength degradation taken into consideration". *Archives of civil and mechanical engineering*, vol. 11, n. 3, pp. 623–636, 2011.
[12] P. J. Montoya, A. G. Meseguer and F. M. Cabré. *Hormigón armado*. Editorial Gustavo Gili, 2000.