XIII SIMMEC
Simpósio de Mecânica Computacional
29 de Outubro a 1º de Novembro de 2018
UFES - Campus Goiabeiras - Vitória, ES

AMBIENTE GRÁFICO INTERATIVO PARA SIMULAÇÃO COMPUTACIONAL DE ESTRUTURAS EM CONCRETO ARMADO

Interactive Graphical Environment for Computational Simulation of Reinforced Concrete Structures

Pamela Daniela Nogueira (1); Leandro Lopes da Silva (2)

(1) Estudante de Engenharia Civil, Universidade Federal de Minas Gerais, Belo Horizonte - MG, Brasil.
(2) Ms. Prof., Universidade Federal de Minas Gerais, Belo Horizonte - MG, Brasil.
E-mail para Correspondência: pameladanielanogueira@gmail.com; (P) Apresentador

Resumo: O uso de estruturas em concreto armado é bastante difundido e, desse uso, surge a necessidade de desenvolver modelos de análise específicos para este tipo de estrutura que forneçam resultados confiáveis. Para o desenvolvimento destes modelos, deve-se considerar que o concreto armado é composto por dois tipos de materiais distintos, o que torna o comportamento do conjunto mais complexo. Neste contexto, modelos baseados no Método dos Elementos Finitos têm se mostrado recursos eficientes por serem capazes de fornecer resultados satisfatórios para a análise deste tipo de estrutura. Usualmente, a modelagem de estruturas em concreto armado considera que há compatibilidade de deformações entre o aço e o concreto. No entanto, esta premissa não é suficiente para representar o comportamento de algumas estruturas, nas quais existem regiões em que as tensões de aderência são muito altas e ocorre a danificação do concreto nas proximidades da interface aço-concreto, o que leva ao escorregamento da armadura. Nestes casos, ocorre a perda de aderência entre os dois materiais, comprometendo a transferência de esforços. De forma a atender à necessidade de considerar o fenômeno da perda de aderência, o presente trabalho apresenta uma interface gráfica interativa implementada no sistema INSANE, que suporta a análise de estruturas em concreto armado com modelos de armadura discreta e embutida, considerando ou não a perda de aderência. O sistema INSANE (I*Nteractive Structural ANalysis Environment)* é um software livre desenvolvido no Departamento de Engenharia de Estruturas da Universidade Federal de Minas Gerais.

Palavras chaves: concreto; armadura; aderência; MEF; escorregamento.

Abstract: The widespread use of reinforced concrete structures leads to the need of developing analysis models that are specific for this type of structure and can provide reliable results. For the development of these models, it should be considered that the reinforced concrete is composed by two different materials, which leads to a more complex behavior. In this context, models based on the Finite Element Method have been proving to be efficient resources by being able to provide satisfactory results for the analysis of this type of structure. Usually, the modeling of reinforced concrete structures considers that there is compatibility of strains between steel and concrete. However, this premise is not sufficient to represent the behavior of some structures, in which there are regions that the bond stress is high and occurs damage of the concrete in the vicinity of the steel-concrete interface, which leads to the reinforcement slip. In this cases, the phenomenon of the bond-slip takes place, compromising the transfer of efforts. In order to address the need to consider the bond-slip phenomenon, the present work present an interactive graphic interface implemented in the INSANE system, which supports the analysis of reinforced concrete structures applying the discrete and embedded reinforcement models. The INSANE system (INteractive Structural ANalysis Environment) is a free software developed at the Department of Structural Engineering of the Federal University of Minas Gerais.

Keywords: *concrete; reinforcement; adherence; FEM; bond-slip.*

1 INTRODUCTION

The advent of computer methods combined with the large amount of studies related to the structural engineering field, and, more specifically, related to reinforced concrete structures, leads to the development of numerical models that improve the prediction of the behavior of this type of structure. These models must consider that reinforced concrete structures are composed by two different materials, which leads to a more complex behavior of the system.

In this context, models based on the Finite Element Method (FEM) have shown to be powerful tools for the analysis of reinforced, simple and prestressed concrete structures. Generally, the modeling of reinforced concrete structures considers that there is compatibility of strain between the reinforcement and the concrete. However, this simplifying assumption is not able to represent adequately the behavior of certain structures in which the bond-slip phenomenon is more pronounced (Castro, 2013). To include the reinforcement in the analysis, three models can be cited: the discrete reinforcement model, the embedded model and the smeared model. It is important to notice that only in the first two models cited the bond-slip phenomenon can be included.

In this work, resources implemented in the interactive graphic interface of the INSANE system that allow the modeling of reinforced concrete structures considering bond-slip will be presented.

The INSANE system (INteractive Structural ANalysis Environment), where this project was held, is a free software, implemented in JAVA and based on object-oriented programming, developed at the Department of Structural Engineering of the Federal University of Minas Gerais.

2 REINFORCEMENT MODELS

To represent the reinforcement steel in the analysis, three models can be used: the smeared reinforcement model, the discrete reinforcement model and the embedded reinforcement model.

The smeared model considers that the reinforcement is distributed in a given orientation, not allowing changes in the geometry of the reinforcement layer along the thickness of the element. Considering that this model does not support the inclusion of the bond-slip phenomenon, it will not be addressed in this work. Next, the other two models will be presented in more details.

2.1 Discrete reinforcement model

The discrete reinforcement model was the first model proposed for the analysis of reinforced concrete structures by the Finite Element Method. This model was first addressed in the work of Ngo e Scordelis (1967), and it considers a plane or a three-dimensional element to represent the concrete and bar elements for the reinforcement (Figure 1).

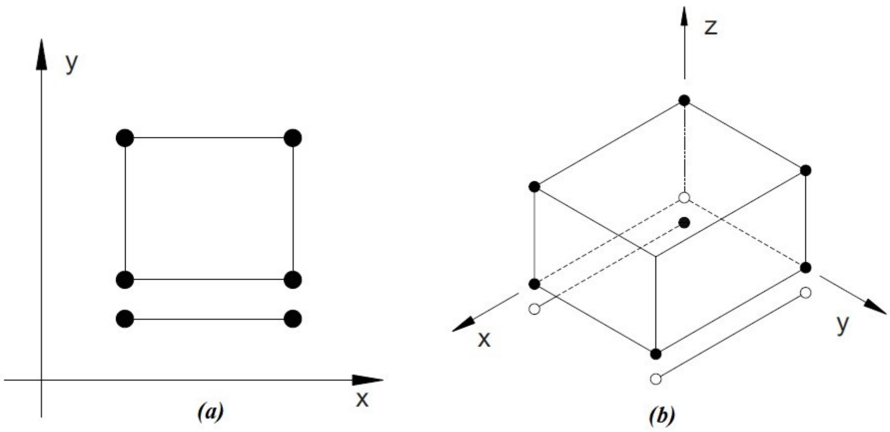

Figure 1: Two-dimensional (a) and three-dimensional (b) discrete reinforcement model.

Source: (Castro, 2013)

This model allows the inclusion of the bond-slip phenomenon by means of special elements that connect the concrete and the reinforcement elements. These special elements will be detailed in Section 3. In the discrete model, the positioning of the reinforcement bars is restricted by the mesh geometry, leading to an increase in the numbers of elements required.

2.2 Embedded reinforcement model

The embedded reinforcement model considers the reinforcement bar as an one-dimensional element incorporated to the concrete element, which can be two-dimensional or three-dimensional.

The first elements developed in this model supposed perfect adherence between the reinforcement and the concrete. Due to the limitations of this assumption, others elements were proposed to solve this problem. In this work, the element proposed by Elwi and Hudrey (1989) was applied. In this element, a reinforcement bar with a given geometry is inserted in the two-dimensional concrete element, as depicted in Figure 2. It is important to highlight that this element can have its formulation expanded from the two-dimensional to the three-dimensional case and includes in its formulation the assumption of loss of bond strength.

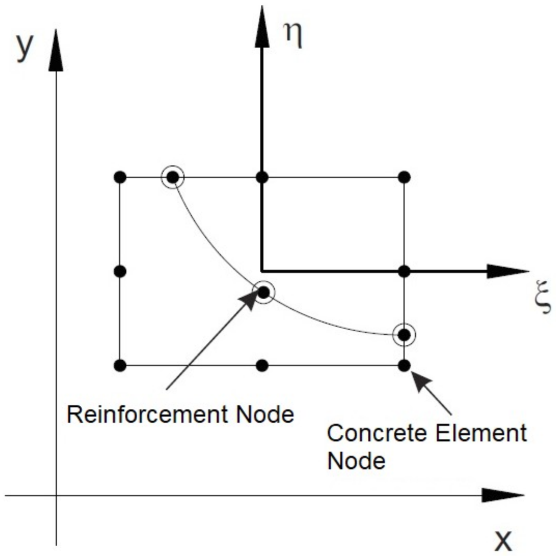

Figure 2: Representation of a concrete element with an element of embedded reinforcement.

Source: (Castro, 2013)

3 ADHERENCE MODELS

Usually, in the modeling of reinforced concrete structures, it is adopted the simplifying assumption that there is perfect adherence between the concrete and the reinforcement, which leads to the premise of compatibility of strain. However, in certain cases, the loss of bond strength must be considered. To consider the bond-slip between reinforcing steel and concrete, special elements associated to an appropriated bond stress-slip relationship must be used. For bond stress-slip relationships the work of the following authors can be cited: Eligehausen et al. (1983), Dörr (1978), and Hawkins et al. (1982).

In the case of the embedded reinforcement model, the bond-slip phenomenon can be included in the formulation of the element. However, in the discrete reinforcement model, special elements, connecting the nodes of the concrete element to the nodes of the steel elements, allow the inclusion of the bond-slip phenomenon in the analysis. Two different elements can treat this inclusion: the bond link element and the contact element, also know as bond zone element.

3.1 Bond link element

This model was proposed by Ngo and Scordelis (1967), and the bond-slip phenomenon is represented by orthogonal spring with no physical dimensions that connect the reinforcement nodes to the concrete nodes. The behavior in the interface of the two materials is defined by the chosen bond stress-slip relationship.

3.2 Contact element

This model was first presented by Hoshino (1974) and Shafer (1975), and it proposes the use of a continuous element to represent the concrete-steel bond-slip. An example of a contact element with a linear approximation can be seen in Figure 3.

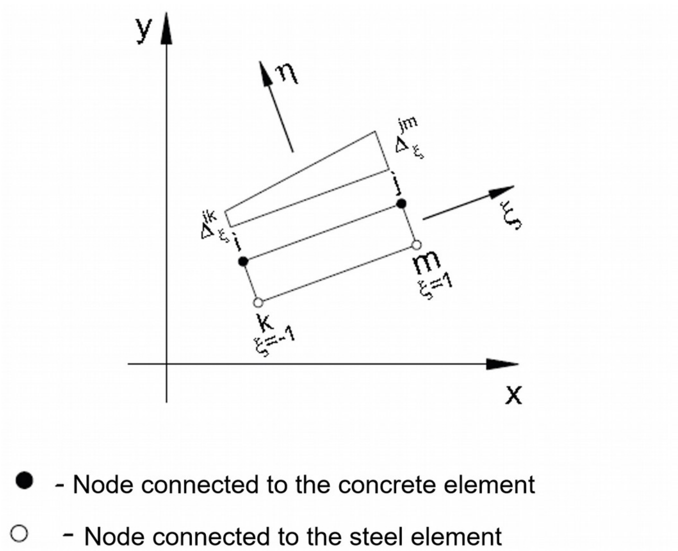

Figure 3: Contact element with a linear approximation.

Source: (Castro, 2013)

4 INSANE INTERACTIVE GRAPHICAL APPLICATIONS

According to Gonçalves (2004), three phases must be followed to develop an application based on the object-oriented programming. The first is the analysis phase, where the objects are defined and described, setting the problem scope. Following, the logic elements are delineated, characterizing the attributes and the methods involved, defining the project phase. In the last step, the components are implemented in a programming language that allows the adoption of an object-oriented system.

The INSANE system uses the JAVA programming language, as well as various APIs (Application Program Interface) and graphical packages available for this language.

4.1 Software design patterns

Software design patterns can be defined as the basic structure for a software project, allowing the creation of the components of the system to be developed (Gonçalves, 2004).

The INSANE system has a structure in layers that result from the merging of three design patterns: the Model-View-Controller (MVC), the Observer, and the Command. In the MVC, the Model stores the model data, the View represents the model, and the Controller intermediate the communication between the View and the Model. The Observer propagates the changes made in the model to the related View-Controller. At last, the Command organizes all the commands required, packing them in separated classes. This process promotes the segmentation of the program, and it favors its expansion.

4.2 Data structure

According to Penna (2007), to propitiate the development of geometrical models, it is important to store the data in an organized and structured manner, where it can be easily accessed, accomplishing a less costly process. To achieve this goal, the graphic applications of INSANE put to use the half-edge data structure, where one edge is represented by two half-edges, each one storing the information from one vertex of the original edge. This data structure is based in the following hierarchy: Solid, Face, Loop, Half-Edge and Vertex. Del Savio et al. (2004) proposed the inclusion of an intermediary level, the Edge, connecting directly the edge and the half-edge related. This hierarchy is depicted in Figure 4.

5 IMPLEMENTATION OF THE GRAPHIC INTERFACE

The analysis of problems considering steel reinforcement bars, with the possibility to include the bond-slip phenomenon, was only possible in the INSANE system thought an XML file (Extensible Markup Language). The use of this type of file requires following certain rules for its writing, and it is necessary the knowledge of the structure that reads and interprets it for the correct use of the available resources. The numerical implementation of this type of analysis was presented by Castro (2013), where the resources for the study of such problems were implemented in the numerical core of the INSANE system.

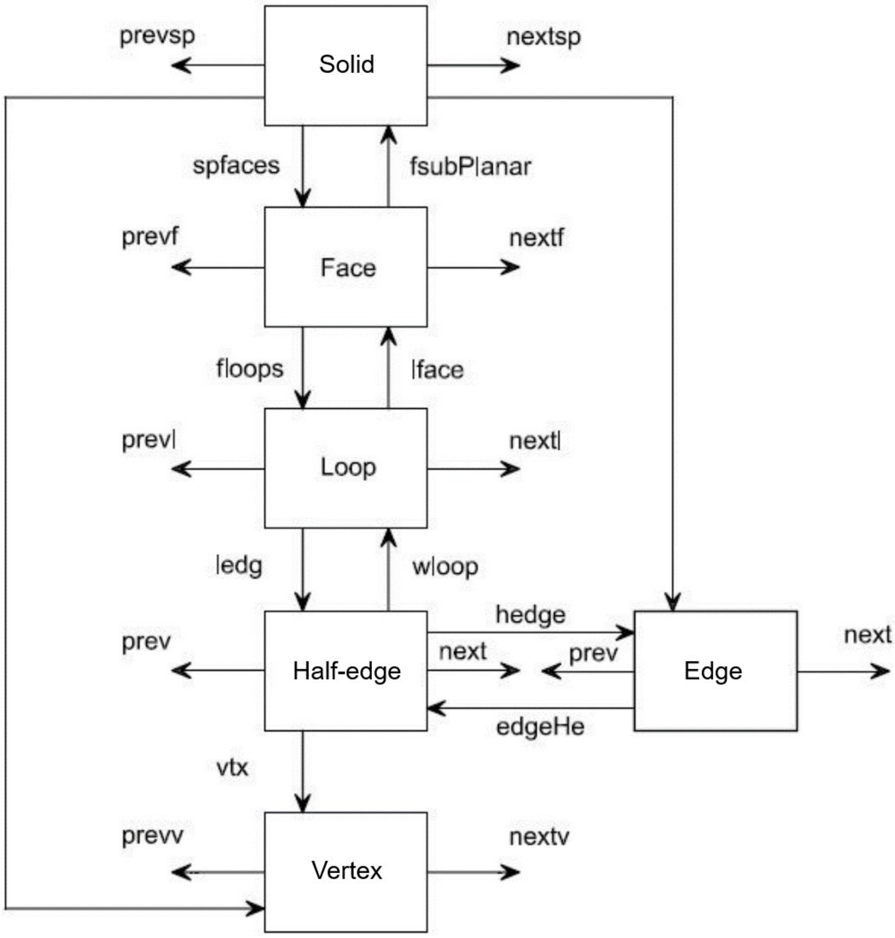

Figure 4: Half-edge data structure.

Source: (Del Savio et al. , 2004)

This work aimed to develop an interactive graphical application to support the modeling of reinforced concrete structures. To achieve this goal, initially, alterations in the graphical interface were necessary to include a button and a menu item that allow the creation of reinforcement bars. These items communicate with the class responsible for the creations of the objects that can be interpreted by the numerical core of the system, allowing the analysis.

The alterations here presented took place in the Preprocessing application of INSANE. The Preprocessing is a graphical interface connected to other two applications: the Processing, that represents the numerical core of the system; and the Postprocessing, that allows the graphical visualization of the results obtained by the processing application. The Preprocessing is based in the design pattern described previously, and it is divided in three modules: Geometry module, where the geometry of the problem is defined; Mesh module, where the geometry is discretized; and the Attributes module, where the attributes of the model are specified, as the constitutive model, the materials, the sections and the boundary conditions.

6 EXAMPLES

In order to illustrate the analysis process by means of the graphical application implemented, three examples will be presented. The first is the simulation of the test carried by Elwi and Hudrey (1989) and adapted in the work of Castro (2013) considering the discrete reinforcement model. In the second example, the embedded reinforcement model was adopted to simulate the test conducted by Mazars and G. Pyaudier-Cabot (1989). Lastly, the pull-out test regulated by RILEM/CEB/FIB (1983) will be simulated, also employing the embedded model.

6.1 Example 1

This example involves the analysis of a quarter ring with one reinforcing layer, as illustrated in Figure 5, with three different approaches: applying the discrete reinforcement model disregarding the bond-slip phenomenon, using nodal springs to simulate the phenomenon, and, lastly, using contact elements. The problem was modeled using 12 plane elements, linear-elastic materials and the following parameters: $A_s/L = 0,025$, $E_s/E_c = 8$, and $\nu = 0,25$.

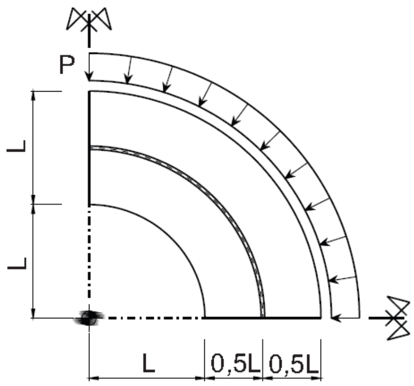

Figure 5: Quarter ring with one reinforcing layer.

Source: (Castro, 2013)

To carry this test using the INSANE system, the geometry and the mesh must be defined through the Geometry and the Mesh modules, followed by the definition of the analysis type and the shape of the finite elements to be used in the simulation. After this phase, the constitutive model, the materials, the sections, and the boundary conditions are specified in the Attribute module (Figure 6). In this stage, the possibility to include reinforcing bars was implemented.

The creation of reinforcing bars is allowed through the keyboard and through the mouse. To consider the bond-slip phenomenon, the user must create specific materials using the dialog that allows the definition of the materials involved in the analysis. After the definition of all the parameters required for the analysis, the problem is processed and the results can be visualized in the Postprocessing application. The results for the simulation can be seen in Figure 7 in a graph that compares the values obtained for the steel stress in the three simulations conducted.

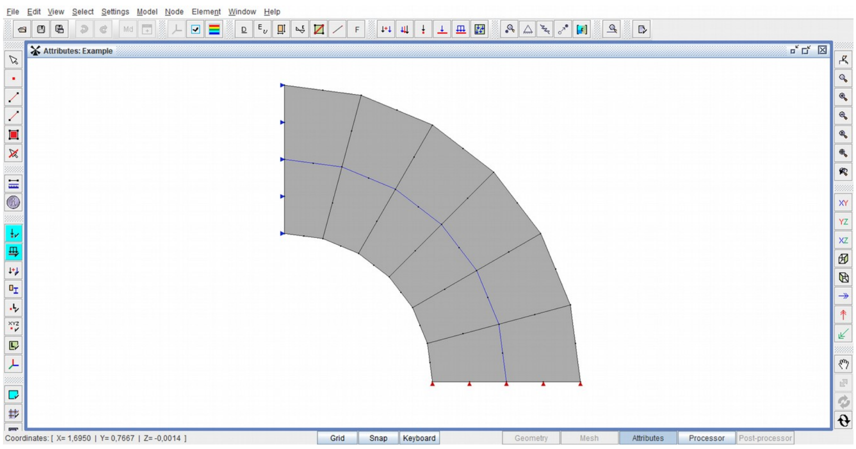

Figure 6: Example 1 - attributes module.

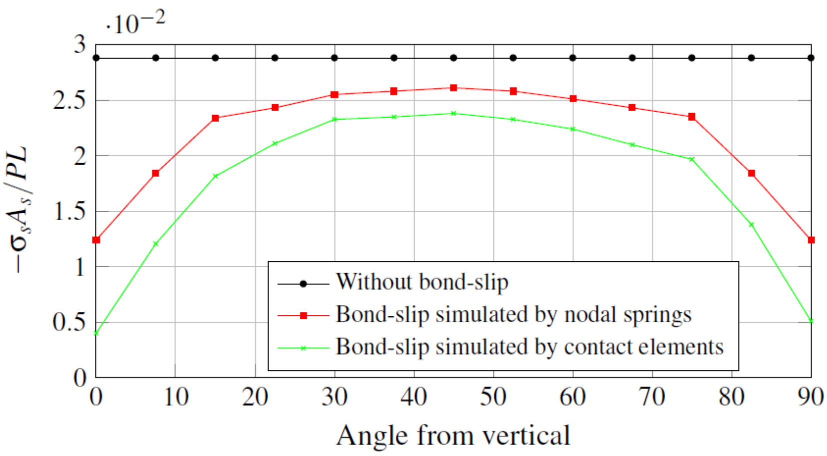

Figure 7: Steel stress, one-layer quarter ring problem.

6.2 Example 2

In this example, the simulation of a beam subjected to a three-point flexural test was conducted with the geometric parameters and the finite elements model shown in Figure 8. The results were compared to the experimental results of Mazars and G. Pyaudier-Cabot (1989). The parameters used in this analysis were: E = 30.000 MPa and ν = 0,20 for the concrete; E = 210.000 MPa for the reinforcing steel.

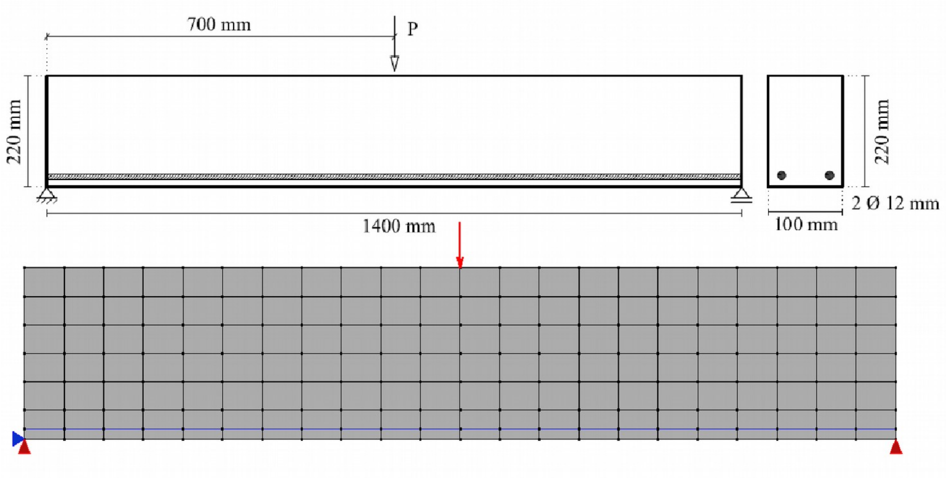

Figure 8: Concrete beam subjected to a three-point flexural test studied by Mazars and G. Pyaudier-Cabot (1989): geometry and finite elements model.

Source: (Adapted from Penna, 2011)

For the concrete modeling, the smeared cracked model with stress-strain law proposed by Carreira and Chu (1985) and Boone and Ingraffea (1987) was adopted with the following parameters: $f_t = 2{,}5$ N/mm^2; $f_c = 25$ N/mm^2; $G_F = 0{,}1$ N/mm; h = 50 mm; $\beta_r = 0{,}05$. The equilibrium path was obtained with the generalized displacement control with the initial load factor of 550, a tolerance of 1×10^{-3} and P = 1N. To represent the reinforcement, the embedded model was applied and for the bond-slip phenomenon, the law proposed by Eligehausen et al. (1983) considering: $w_{b1} = 0{,}5$ mm; $w_{b2} = 1$ mm; $w_{b3} = 5$ mm; $\alpha = 0{,}4$; $\tau_{max} = 22$ MPa; $\tau_f = 9$ MPa. The equilibrium path obtained for the

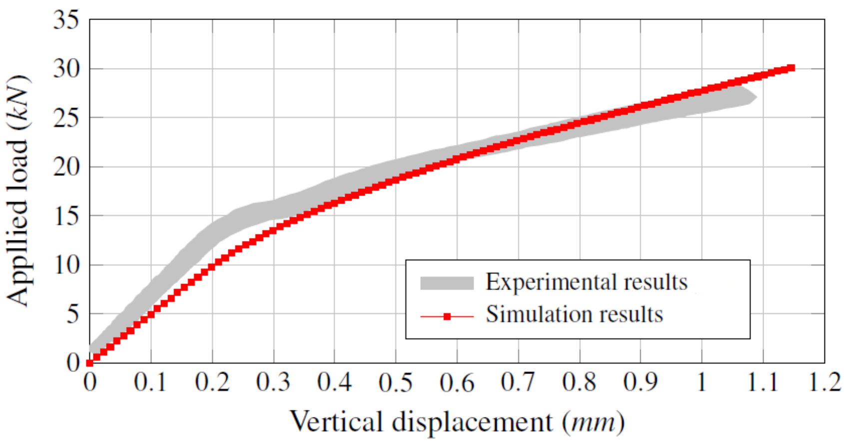

Figure 9: Vertical displacement in the point of application of the load (mm). Experimental results by Mazars and G. Pyaudier-Cabot (1989).

vertical displacement of the node where the load was applied is displayed in comparison with the experimental results of Mazars and G. Pyaudier-Cabot (1989).

6.3 Example 3

In this example, the pull-out test regulated by RILEM/CEB/FIB (1983) will be simulated. The test consists of pulling out a steel reinforcing bar solidarity with a concrete specimen. The numerical model aimed to simulate the test presented in the work of Silva (2010), with the geometry depicted in Figure 10.

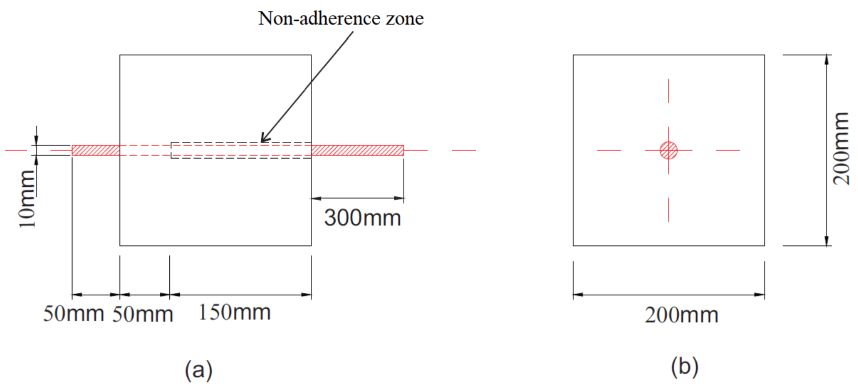

Figure 10: Geometry of the pull-out test.

Source: (Adapted from Castro (2013)).

The example was modeled with 441 quadrilateral elements with four nodes in plane stress state, where 21 elements are elements with embedded reinforcing bars. The finite element model adopted is presented in Figure 11.

For the concrete, smeared cracked model with stress-strain law proposed by Carreira and Chu (1985) and Boone and Ingraffea (1987) was adopted with the following parameters: $f_t = 3{,}28$ N/mm^2; $f_c = 27{,}80$ N/mm^2; $G_F = 0{,}0066$ N/mm; h = 50 mm; $\beta_r = 0{,}2$. Besides that, for the concrete it was adopted $E_0 = 36.100$ MPa and, for the steel, $E_s = 210.000$ MPa. For the bond-slip phenomenon, the law proposed by Eligehausen et al. (1983) was applied considering: $w_{b1} = 0{,}5$ mm; $w_{b2} = 2$ mm; $w_{b3} = 5$ mm; $\alpha = 0{,}4$; $\tau_{max} = 21$ MPa; $\tau_f = 6$ MPa.

For the non-linear analysis, the generalized displacement control method was adopted with initial load factor of 0,5, tolerance for convergence of 1×10^{-3} and reference load of P = 35 kN. The results for the bond-stress x bond-slip obtained for the control point are presented in Figure 12 along with the experimental results of Silva (2010).

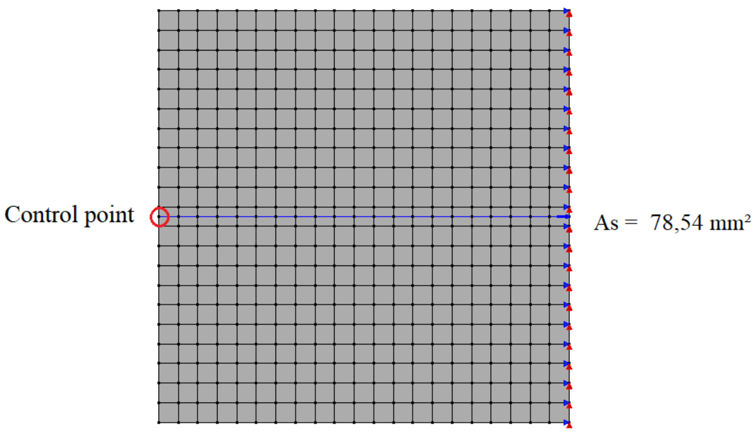

Figure 11: Finite element model adopted.

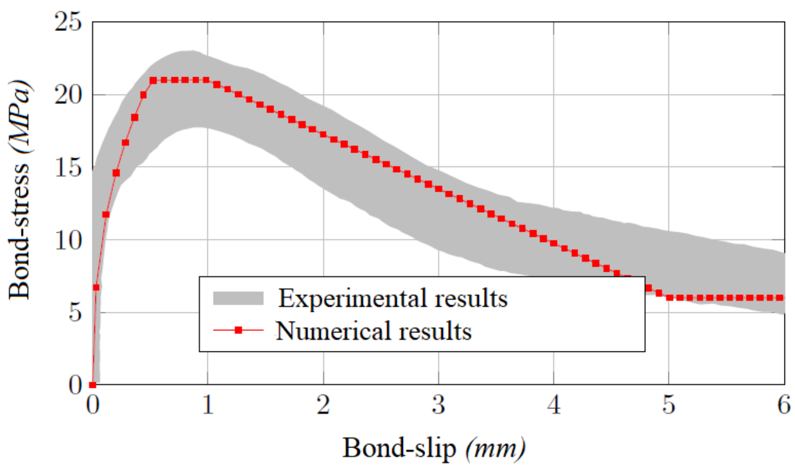

Figure 12: Bond-stress versus bond-slip.

7 CONCLUSIONS

The analysis of problems considering reinforcing bars with the possibility to include the bond-slip phenomenon was implemented in the numeric core of the INSANE system in the work of Castro (2013). However, to access this analysis, it was necessary the use of an XML file, that requires a well defined structure for its interpretation. In the present work, the implementation of the graphic interface that allows this type of analysis was presented.

In the Preprocessing application, the needed resources for the inclusion of reinforcing bar were made available, being possible to adopt the embedded reinforcement model as well as the discrete model.

By means of the simulations results here presented, it can be noticed that the models presented consisted results when compared to existent experimental results. It can be concluded that the graphic interface implemented works properly and wields coherent results, and also allows the input of the necessary data for the analysis of reinforced concrete structures in a friendly manner, without requiring the knowledge of the structures of the XML file.

AKNOWLEDGMENTS

The authors would like to thank the important support of FAPEMIG (Fundação de Amparo à Pesquisa do Estado de Minas Gerais) and CNPq (Conselho Nacional de Desenvolvimento Científico e Tecnológico).

REFERENCES

Boone, T. and Ingraffea, A. R., 1987. Simulation of the fracture process at rock interfaces, in *Proceedings of the fourth international conference in numerical methods in fracture mechanics*, pp. 519-531.

Carreira, D. J. and Chu, K., 1985. Stress-strain relationship for plain concrete in compression. *ACI Journal*, vol. 82, pp. 797-804.

Castro, S. S., 2013. *Framework teórico e computacional para estruturas de concreto armado – implementação de modelos de armadura e aderência*. Masters dissertation, Universidade Federal de Minas Gerais, Belo Horizonte, MG, Brasil.

Del Savio, A. A., Santi, M. R. e Martha, L. F., 2004. Traçado de curvas offset para auxílio na geração de malhas. *XXV Congresso Ibero Latino Americano de Mecânica Computacional - CILAMCE,* Recife, Pernambuco.

Dörr, K., 1978. Bond behavior of ribbed reinforcement under transverse pressure. *IASS symp. Darmstadt*, vol. 1, pp. 13-14.

Eligehausen, R., Popov, E. P. and Bertoro, V. V., 1983. Local bond stress-slip relationships of deformed bars under generalized excitation. *Technical Report 82/83*, Earthquake Engineering Research Center, University of California, Berkeley.

Elwi, A. E. & Hudrey, T. M., 1989. Finite element model for curved embedded reinforcement. *Journal of Engineering Mechanics*, vol. 115, pp. 740-754.

Gonçalves, M. A. B., 2004. Geração de malhas bidimensionais de elementos finitos baseada em mapeamentos transfinitos. Masters dissertation, Universidade Federal de Minas Gerais, Belo Horizonte, MG, Brasil.

Hoshino, M., 1974. *Ein Beitrag zur Untersuchung des Spannungszustandes an Arbeitsfugen mit Spannglied-Kopplungen von abschnittsweise in Ortbeton hergestellten Spannbetonbriicken.* Doctoral thesis, Technischeltoch Schale, Darmstadt, Germany.

Mazars, J. e G. Pyaudier-Cabot, S. M., 1989. Continuum damage theory applications to concrete. *J. Eng. Mech. ASCE*, vol. 115, pp. 345–365.

Ngo, D. & Scordelis, A. C., 1967. Finite element analysis of reinforced concrete beams. *Journal of the American Concrete Institute*, vol. 64, pp. 152-163.

Penna, S. S., 2007. Pós-processador para modelos bidimensionais não-lineares do método dos elementos finitos. Masters dissertation, Universidade Federal de Minas Gerais, Belo Horizonte, MG, Brasil.

Penna, S. S., 2011. *Formulação Multipotencial para Modelos de Degradação Elástica - Unificação Teórica, Proposta de Novo Modelo, Implementação Computacional e Modelagem de Estruturas de Concreto*. Doctoral Thesis, Universidade Federal de Minas Gerais, Belo Horizonte, MG, Brasil.

RILEM/CEB/FIB, (1983), Bond test for reinforcing steel: pull-out test, *Technical report*, Comité Euro-International Du Béton, Paris.

Shafer, H., 1975. A contribution to the solution of contact problems with the aid of bond elements. *Computer Methods in Applied Mechanics and Engineering*, vol. 6, pp. 335-354.

Silva, B. V., 2010. *Investigação do potencial dos ensaios apulot e pull-out para estimativa da resistência a compressão do concreto.* Masters dissertation, Universidade Estadual Paulista – Júlio Mesquita, Ilha Solteira, SP, Brasil.