

Structural analysis of a prestressed segmented girder using contact elements in ANSYS

Paula M. Lazzari^{*1,2}, Américo Campos Filho^{2a}, Bruna M. Lazzari^{2b}
and Alexandre R. Pacheco^{2c}

¹Infrastructure Engineering Graduate Program, Federal University of Santa Catarina, 2700 Dr. João Colin St, 89218-000, Joinville, SC, Brazil
²Civil Engineering Graduate Program, Federal University of Rio Grande do Sul, 99 Oswaldo Aranha Ave, 90035-190, Porto Alegre, RS, Brazil

(Received March 9, 2017, Revised April 19, 2017, Accepted May 3, 2017)

Abstract. Studying the structural behavior of prestressed segmented girders is quite important due to the large use this type of solution in viaducts and bridges. Thus, this work presents a nonlinear three-dimensional structural analysis of an externally prestressed segmented concrete girder through the Finite Element Method (FEM), using a customized ANSYS platform, version 14.5. Aiming the minimization of the computational effort by using the lowest number of finite elements, a new viscoelastoplastic material model has been implemented for the structural concrete with the UPF customization tool of ANSYS, adding new subroutines, written in FORTRAN programming language, to the main program. This model takes into consideration the cracking of concrete in its formulation, being based on fib Model Code 2010, which uses Ottosen rupture surface as the rupture criterion. By implementing this new material model, it was possible to use the three-dimensional 20-node quadratic element SOLID186 to model the concrete. Upon validation of the model, an externally prestressed segmented box concrete girder that was originally lab tested by Aparicio *et al.* (2002) has been computationally simulated. In the discretization of the structure, in addition to element SOLID186 for the concrete, unidimensional element LINK180 has been used to model the prestressing tendons, as well as contact elements CONTA174 and TARGE170 to simulate the dry joints along the segmented girder. Stresses in the concrete and in the prestressing tendons are assessed, as well as joint openings and load versus deflection diagrams. A comparison between numerical and experimental data is also presented, showing a good agreement.

Keywords: prestressed concrete; segmented girder; contact elements; ANSYS; USERMAT

1. Introduction

Prestressed concrete segmented box girders are considerably used in bridges, viaducts, and elevated walkways due to their constructive advantages. In order to analyze structurally this type of solution, the Finite Element Method (FEM), has been used, since it is a worldwide-recognized methodology by structural designers and researchers. The use of this method allows the study of structural concrete structures involving all factors that typically make analyses more difficult. Among these factors, can be mentioned, for instance, the difference in behavior observed in concrete when in tension and in compression, the concrete's nonlinear stress-strain relation, its cracking, creep and shrinkage, as well as steel's relaxation. Besides, a methodology based on FEM is capable of analyzing quite complex structures, simulating

different geometrical configurations, loading types, and boundary conditions. Among a diversity of available commercial computational platforms for structural analyses, ANSYS has been one of the most used, mainly because of its continuous development. Many researchers have been using this program in their works and some can be cited as of importance to this work. For instance, there is the study of Vasudevan and Kothandaraman (2015), where ANSYS was used to assess the effects of flexural strengthening of RC beams with external rebars using different anchorages. Another work worth mentioning is the study of Demir and Husem (2015), where different modeling methods were used to simulate bonding losses in RC elements. Another one is the work by Oner *et al.* (2015), where contact elements were studied analytically and numerically. In the paper of Ergun and Ates (2015) FEM is used to investigate stress analysis of a shear wall which is subjected to concentrated loads and fundamental principles of stress analysis of the shear wall are presented by using matrix displacement method. Others studies on concrete structures using ANSYS are also found in the works of Shaheen (2016), Anil and Uyaroğlu (2013), Amiri *et al.* (2012), Bulut *et al.* (2011) and Kazaz (2011).

In this work, the main objective is to present a computational simulation, applying FEM with ANSYS (version 14.5), of the behavior of the prestressed segmented box girder that was experimentally tested by Aparicio *et al.* (2002). The results obtained herein are in terms of stresses

*Corresponding author, Professor
E-mail: p.manica.lazzari@gmail.com

^aProfessor
E-mail: americo@ufrgs.br

^bPh.D. Student
E-mail: bruna.ml@gmail.com

^cProfessor
E-mail: apacheco@ufrgs.br

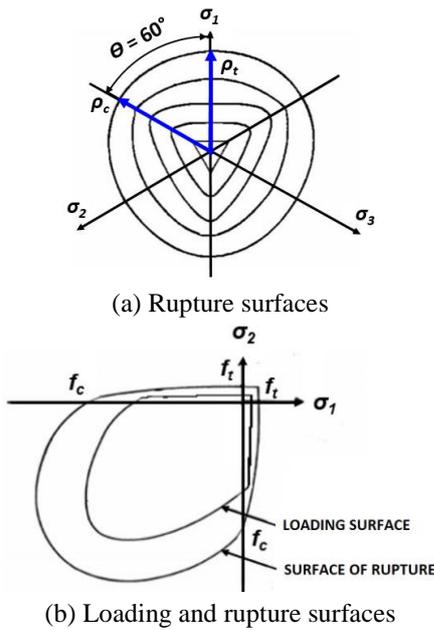


Fig. 1 Rupture and plastification criteria for concrete (Ottosen 1977)

in the concrete and in the prestressing tendons. In order to carry out this computational modeling, taking into consideration the least number of finite elements, it was necessary to use the UPF customization tool (UPF-User Programmable Features), which is available in ANSYS. This tool allowed the implementation of a new model to analyze concrete and prestressing steel, and therefore being able to analyze both reinforced and prestressed concrete.

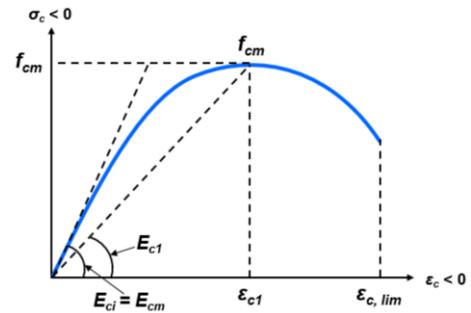
Additional validation of the created model was carried out through numerical simulations of reinforced and prestressed beams that have also been originally tested in the lab and are presented in Lazzari (2015), Lazzari (2016), and Lazzari *et al.* (2017).

2. Material constitutive models

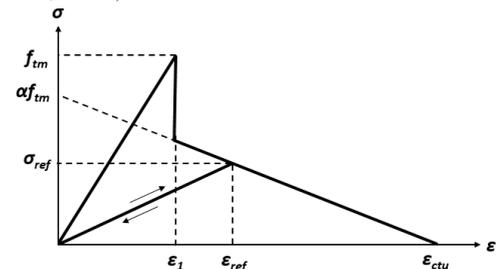
Two different material models have been used to describe concrete's behavior. An elastoplastic model has been adopted for concrete under compression and, for the concrete under tension, a linear elastic behavior up to rupture has been the option, when a model considering the contribution between cracks takes place.

The model for concrete under compression is composed by a rupture criterion, a plastification criterion, and a hardening rule. The rupture criterion used is based upon Ottosen's model, which is recommended by *fib* Model Code 2010 (2012). In Fig. 1(a), this rupture surface is illustrated in a three-dimensional stress space by its cross sections (Chen and Han 1988). In this work, the compressed concrete is considered to have isotropic hardening and that the plastification or loading surfaces have the same shape as the rupture surface. In Fig. 1(b), the loading and the rupture surfaces can be observed.

The hardening rule defines how the plastification or loading surfaces move during plastic deformation. It is



(a) Stress-strain curve for compressed concrete (*fib* Model Code 2010, 2012)



(b) Stress-strain curve for tensioned concrete (Hinton 1988)

Fig. 2 Concrete's behavior under compression and under tension

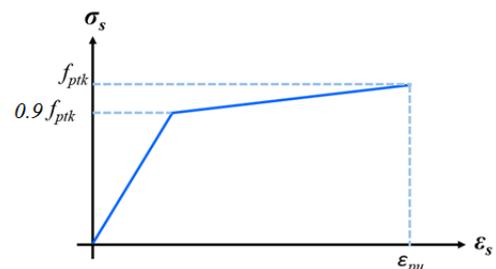


Fig. 3 Elastoplastic model with linear hardening for prestressing strands

determined by the effective plastic stress-strain relation, which can be used to extrapolate simple uniaxial test results to a multiaxial situation. In this work, the uniaxial stress-strain diagram for compressed concrete proposed by *fib* Model Code 2010 (2012), which is illustrated in Figure 2(a), has been adopted for that purpose.

Concrete under tension is modeled as an elastic material with softening. Before cracking, concrete behaves as a linear elastic material, but after cracking, the Smeared Cracking model with tension stiffening is used, as indicated in Fig. 2(b). The cracking model used, which involves a cracking criterion, a between cracks collaboration rule for the concrete, and a shear stress transfer model is based on the formulation presented by Hinton (1988).

Considering, for simplification, that steel rebars resist only axially, a uniaxial model is used to represent their behavior. Steel is represented as a perfect elastoplastic material that shows the same behavior when either under tension or compression. When active reinforcement is considered (prestressing), steel is modeled as a linear material up to 90% of its strength (rupture stress, f_{ptk}). Upon reaching this value, it presents a linear hardening behavior,

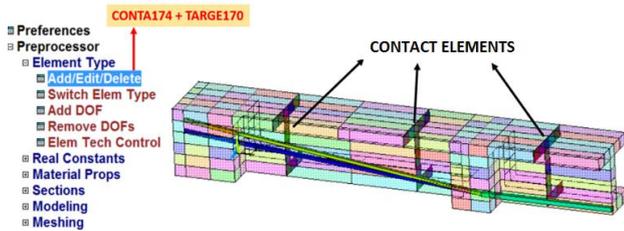


Fig. 4 Discretization example with elements CONTA174 and TARGE170

as illustrated in Fig. 3.

3. Computational model

The computational analysis conducted has been carried out over the finite element platform ANSYS, in its 14.5 version. This type of numerical analysis allowed the consideration of the behavioral nonlinearities of the materials steel and concrete, including cracking in concrete, as well as plastification of both concrete and steel.

3.1 Types of finite elements used

In order to model the concrete, hexahedral element SOLID186 is used, which is a three-dimensional 20-node quadratic element with three degrees of freedom per node (translations in X, Y, and Z directions). In this way, good results can be reached without the need of an extremely refined mesh, decreasing significantly the time for a structural analysis. In addition to a hexahedral configuration, element SOLID186 can be pyramidal, prismatic, or even tetrahedral, allowing better adaptation to complex geometries. There is no need to assign constants for this element and, even being used herein to represent concrete, this element type is not compatible with model CONCRETE, available in ANSYS, which is explained later. Element LINK180 can be used to represent bonded and unbonded prestressing reinforcement, allowing introduction of either strains or initial stresses. It is a unidimensional element with three degrees of freedom in each node (translations in X, Y, and Z directions), where plasticity, viscoelasticity, and large deformations can be considered. In the case of bonded prestressing, element REINF264 can be used (admitting an initial stress in the Element-Embedded reinforcement) or element LINK180 (using the Discrete type of modeling for the reinforcement). The reinforcing element REINF264 can be used together with elements of the type “bar”, “plate”, “shell”, or even with solid elements. This element is appropriate for simulations of randomly distributed reinforcing fibers, where each fiber is modeled separately as a segment that has only axial stiffness. It is possible to specify several reinforcing fibers REINF264 for a single base element. The nodal coordinates, degrees of freedom, and connectivities of element REINF264 are identical to those of the base element (in this work, element SOLID186). Details of this element and usage examples can be found in Lazzari (2016). It is important to point out that using element LINK180 may imply in mesh constraints

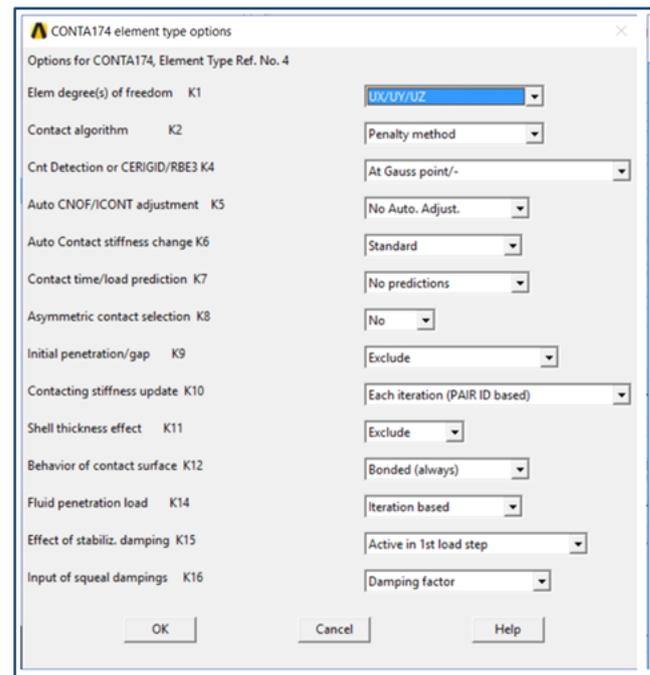


Fig. 5 Window options for CONTA174

for the concrete because of the reinforcement location and distribution. This tends to occur because the discretization of the reinforcement with element LINK180 is of the Discrete type, with nodes having to necessarily coincide with the nodes of element SOLID186.

Contact elements CONTA174 and TARGE170 are used herein to represent the dry joints of the prestressed concrete segmented girder by Aparicio *et al.* (2002). CONTA174 is a three-dimensional eight-node element (surface-to-surface contact), used to represent the slipping contact between a deformable surface (composed by CONTA174 elements) and a target surface (composed by TARGE170 elements). These two elements must be used always together, with element CONTA174 pertaining to one of the dry joint surfaces and element TARGE170 to the other. The target surface is modeled by a set of elements of the target segment (TARGE170) and is coupled with its contact surface through a set of REAL CONSTANTS. CONTA174 is located on the surface of solid elements or three-dimensional shells with central nodes, presenting the same geometrical characteristics of the connection surface. In Fig. 4, an example is presented, as well as showing how to select such elements in the main menu.

Figs. 5 and 6 shows the window of options for element CONTA174 and a programming code to add such contact elements. The information being given in the figures is still related to the example of the segmented girder by Aparicio *et al.* (2002). The information has been given by following indications by Wahab (2014), and adjusted accordingly with the boundary conditions of the example. The REAL CONSTANTS of element CONTA174 have been considered as negative at options 3 and 12, zeroing the starting properties related to tension stresses and shear, respectively. This procedure allows application of a specified constitutive model to represent dry joint

```

APARICIO.txt - Bloco de notas
Arquivo Editar Formatar Exibir Ajuda
-----
CONTATO - CONTA174 + TARGE170 - 3 e 4
-----
Et,3,TARGE170
|
Et,4,CONTA174
|
Keyopt,4,2,1 ! Penalty function
Keyopt,4,9,1 ! Exclude both initial penetration or gap
Keyopt,4,10,2 ! Contact stiffness update each iteration based
Keyopt,4,12,5 ! bonded (always)
|
-----
REAL CONSTANTS
-----
|
R1, CONCRETE
R2, INCLINED PART OF PRESTRESSING TENDON
R3, HORIZONTAL PART OF PRESTRESSING TENDON
R4, CONTACT ELEMENT
|
R,1, , , , , ,
RMORE, , , , , ,
RMORE, ,
|
R,2,1.81453, 0,0
R,3,3.62906, 0,0
|
R,4
RMODIF,4,3,-1e1 !FKN - Normal stiffness
RMODIF,4,12,-1e1 !FKT - Tangent stiffness
|
!A negative value is used as an absolute
!value, which overrides the default
|

```

Fig. 6 Scripting for elements CONTA174 and TARGE170

connections, allowing further adjustment of the values, if necessary.

According to Wahab (2014), the Cohesive Zone Model (CZM) is a model that presents bilinear behavior with linear softening, being characterized by a maximum tension and a maximum separation. This model would be the most indicated to represent the separation process in dry or bonded joints. In ANSYS, this model can be used together with contact elements (CONTA174 and TARGE170), giving, for instance, joint opening values and resulting stresses on the surfaces in between segments. A detailed formulation for this constitutive model for joints is presented in the book by Wahab (2014).

3.2 Modeling the concrete

The simulation of the structural concrete is implemented through the new viscoelastoplastic material model with cracking, which is based on the failure criterion by Ottosen (1977), and which is recommended by *fib* Model Code 2010 (2012). The constitutive equations for this material have been implemented through the UPF customization system (User Programmable Features) of ANSYS in its subroutine USERMAT (User Material Routine).

The proposition of the new model has been necessary because the only option available until then was the model CONCRETE, an elastoplastic model with cracking based upon the criterion by Willam and Warnke, issuing a considerable disadvantage. The CONCRETE model can only be used with element SOLID65, which, in its turn, does not allow the use of the Element-Embedded model for the reinforcement, demanding a much larger amount of finite elements to represent the structure, ultimately complicating the convergence control. Therefore, numerical simulations in structural concrete would become extremely slow, demanding machines with increased computational

```

-----
CONCRETE - USER - MATERIAL 1
-----
Constitutive Model of user material
tb,user,1,1,5 ! Material ID: 1, 1 temperature,
! 5 constants(prop)
tbtemp,1,0 ! First temperature
tbdata,1,E1, v, fc,4, 21.002 ! Temperature 1, E, Poisson, fc, aggregate,
! loading steps, initial date
tb,state,1,,9 ! Define 9 state variables
|

```

Fig. 7 Example of use of subroutine USERMAT for concrete

power unnecessarily.

In order to make the UPF system available, ANSYS should be installed with option “ANSYS Customization Files” activated, automatically creating folders “Custom” and “Customize” in the path “C:\Program Files\ANSYS Inc\v145\ansys”. In this way, it is possible to access subroutine USERMAT to write new constitutive equations for a customized material. Considering the case of concrete, the new model has been created in the subroutine USERMAT3D, which is called from subroutine USERMAT when three-dimensional elements are considered. Three other editable subroutines can also be called: USERMATPS, for elements under plane stress states; USERMATBM, for three-dimensional BEAM elements; and USERMAT1D, for unidimensional elements.

The use of subroutine USERMAT, containing the new concrete model implemented, is preceded by carrying out its compilation and linkage with the main program ANSYS, creating a Dynamic-Link Library (DLL). In this work, these tasks have been carried out with programs MS Visual Studio and the FORTRAN Intel compiler.

The USERMAT subroutine is used in any ANSYS analysis that involves mechanical behavior, being called in every Newton-Raphson iteration. In the starting time increment, ANSYS stores stresses, displacements, and the necessary variables that are updated at the end of the time increment. The input data needed by the new constitutive model are given in an input data file defined by the command “TB, USER”. This is exemplified in Fig. 7, where a data input script for the new concrete model implemented for the segmented girder is presented. The command requires only five variables: modulus of elasticity, Poisson’s ratio, compressive concrete strength, aggregate type and loading step sequence, with other parameters being calculated internally.

Regarding the variable “aggregate type”, four types of aggregate can be considered internally in USERMAT3D to calculate the modulus of elasticity. The identification of each type is as follows: 1-basalt; 2-quartz; 3-limestone; and 4-sandstone. The parameter that corresponds to the “loading step sequence” describes the loading sequence with two different procedures used to simulate the viscoelastoplastic behavior of concrete: STEP 1 and STEP 2. In STEP 1, the time dependent structural behavior is determined through a time increment process. The time interval used has been of one day. In this step, a viscoelastic behavior is adopted for the concrete, taking into consideration effects of creep, shrinkage, and relaxation. In STEP 2, a structural equilibrium is sought after applying a load increment for an instantaneous load case. This step corresponds to a static

```

-----
| REINFORCEMENT - USER - MATERIAL 2
|-----
| Constitutive Model of user material
|-----
tb,user,2,2,5          ! Material ID:2. 2 temperatures.
                       ! 5 constants (prop)
tbtemp,1.0            ! First temperature
tbdata,1,19500, 0.3, 190, 87.6,28 ! Temperature 1, E, Poisson, Fptk,
                                   ! prestressing tension,
                                   ! date of prestressing
tb,state,1,,9         ! Define 9 state variables
|-----

```

Fig. 8 Example of use of subroutine USERMAT for the prestressing steel

state, where a structural elastoplastic behavior is analyzed. In the computational model, as many loading cases as needed are allowed to be applied, executing successively each step accordingly to the dates specified in the loading cases. In the example presented in Fig. 7, the parameter that indicates the amount and the sequence of steps is represented by the number 21.002. The digits on the left of the period represent the sequence of calculation steps, from right to left; while the digits on the right of the period represent the total number of load cases. Therefore, in this example, two calculation steps are to be realized in the following sequence: STEP 1+STEP 2. The number of days considered in STEP 1 is specified with the variable “nsubst” of ANSYS, corresponding to the number of load increments that is indicated for each loading case in the input data script.

3.3 Modeling the steel

Since the segmented girder presents prestressing tendons only of the external type, a model to allow specification of an initial stress in the reinforcement has been created with the UPF system and subroutine USERMAT1D. In addition, a new material model with time dependent properties to consider the effects of prestressing steel relaxation, as recommended by *fib* Model Code 2010 (2012), has been also implemented. In Fig. 8, an example of script is presented with the data input for the new reinforcement model implemented with command “TB, USER”. This command requires five variables: modulus of elasticity, Poisson’s ratio, strength stress, initial prestressing stress, and the date of prestressing.

4. Analysis of the prestressed segmented concrete girder

In the next items, details for the simulation of an externally prestressed segmented girder that was experimentally tested by Aparicio *et al.* (2002) are presented. In order to validate the model, the numerically obtained results are compared with the experimental data in the form of load versus displacement diagrams; load versus tendon stress diagrams; and even by observing joint openings.

4.1 Characteristics of the girder

The simply supported prestressed concrete girder

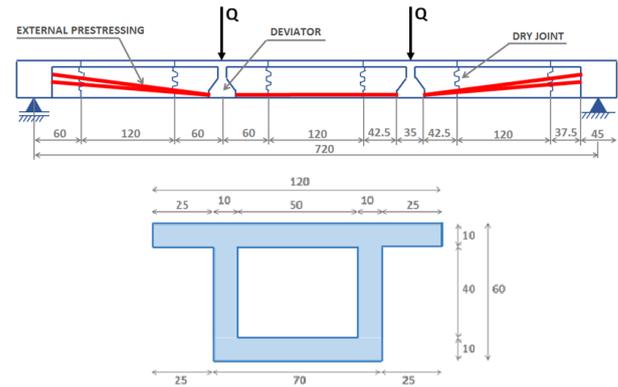


Fig. 9 Elevation and cross section views of the box segmented girder (dimensions in cm)

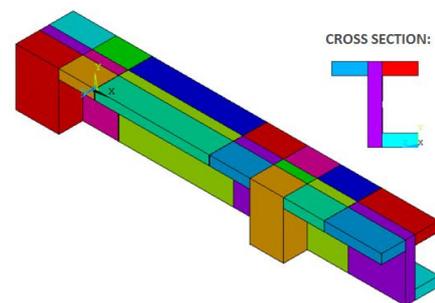


Fig. 10 Volume mesh-segmented girder

presents a box cross section and spans over a length of 7.2 m. The girder is composed of 7 concrete box segments connected by dry joints (cross section in Fig. 9). The prestressing tendons are composed of four strands of 15.2 mm in diameter of steel CP-190 RB (Brazilian specification), with an applied initial stress of 876 MPa. The modulus of elasticity of the strands is equal to 195000 MPa and its tensile strength is equal to 1900 MPa, with mean compressive strength for the concrete of 45 MPa.

The concrete discretization in finite elements for this segmented girder resulted in a mesh with 192 quadratic hexahedral elements of twenty nodes, the element SOLID186. In order to take into account the symmetry in loading and geometry, only a quarter of the girder is considered in the analyses. The modeling started with the creation of 26 volumes, considering geometrical conditions and the regions with dry joints, as shown in Fig. 10. These volumes are discretized with concrete finite elements, with the final mesh being illustrated in Fig. 11. Additionally, there are metal plates at the ends of the girder to avoid stress concentration in the concrete elements that result from the anchorage points of the prestressing tendons.

In addition to the three-dimensional solid elements, three LINK180 elements are included on the internal surfaces of the box cross section, representing the horizontal and the inclined portions of the unbonded prestressing tendons. These portions are connected to the concrete only at the two ends of the girder, as well as at the deviators, allowing slippage of the tendon along its own length. The discretized deviator is located near the bottom of the girder, at position $X = 240$ cm and, in the same vertical, at the top nodes, the load is applied incrementally.

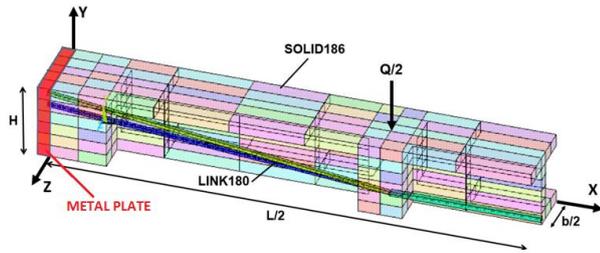


Fig. 11 Finite element mesh for the computational analysis

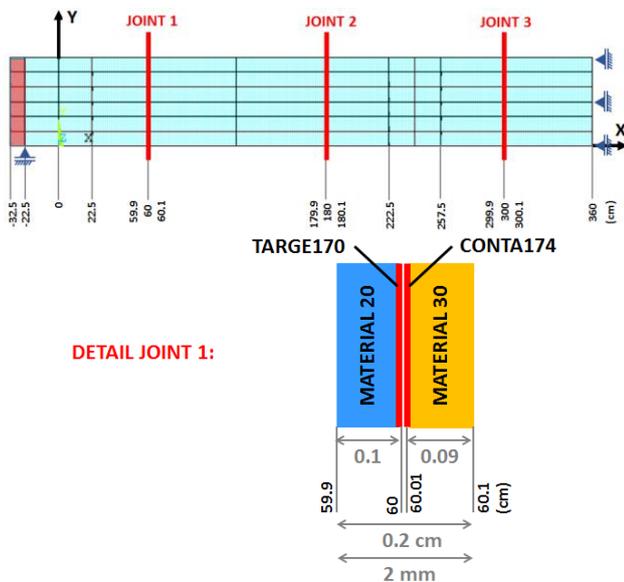


Fig. 12 Modeling details of the dry joints (dimensions in cm)

Regarding boundary conditions, restrictions in the X direction are applied on the plane surface YZ at $X = L/2$; restrictions in the Z direction on the plane surface XY at $Z = 0$; and restrictions in the Y direction at the bottom nodes of the supported ending ($X = 0$ and $Y = 0$). In Fig. 11, it is possible to visualize the discretization of the concrete elements, of the metal plate, and of the prestressing elements in the model.

The new models implemented through subroutines USERMAT3D and USERMAT1D are used for the concrete and for the prestressing reinforcement, respectively. However, for the metal plate, the elastic linear material available in ANSYS is the one chosen, considering a Poisson's ratio of 0.3 and a longitudinal modulus of elasticity equals to 200000 MPa.

The three dry joints are modeled according with the illustration presented in Fig. 12. In order to avoid concentration stress problems, a linear elastic material with approximately 1mm in thickness is added to the dry joint surfaces. Additionally, according to the drawing presented, it can be observed also the use of the contact elements TARGE170 and CONTA174 on each joint surface. These elements are separated by a gap as small as 0.1 mm, which is necessary to facilitate the addition of the contact elements in ANSYS.

In Fig. 13, the discretization of the contact elements for the three dry joints of the segmented girder is presented.

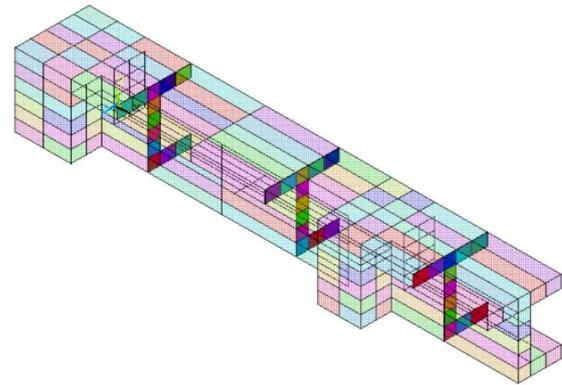


Fig. 13 Discretization of contact elements TARGE170 and CONTA174

The mesh for these elements follows the same discretization defined for the concrete elements. Since elements TARGE170 and CONTA174 are separated by too small a gap, visualization of their surfaces becomes difficult to fulfill. During the analysis of the results, however, it is going to be easier to observe their behavior.

Materials numbered as 20 and 30 are being used to represent the linear elastic behavior of the dry joint elements, which are set with a Poisson's ratio of 0.2 and a longitudinal modulus of elasticity equals to 20000 MPa. These identification numbers are adopted for convenience, allowing a better visualization of the elements.

The Cohesive Zone Model (CZM) is used with contact elements TARGE170 and CONTA174, a model that is available in the material library of ANSYS and that is indicated by Wahab (2014) to represent the behavior of contact elements. Among the available options, a bilinear behavior with linear softening that allows maximum separation between contact elements is also chosen.

The deviators have been considered in the model through command "CP" of ANSYS, allowing slippage only in the axial direction. When using this command, it is necessary to select a node from the girder and a node from the deviator, then indicating the direction that would be fixed.

After finishing the modeling, the structural analyses are carried out by considering a concrete curing period of 28 days, together with the reinforcement prestressing. In the next item, the results of the analyses are presented, considering two possibilities: active deviators, which allow tendon slippage, and deviators without any slippage.

4.2 Analysis of results

In this item, numerical results are compared with the experimental data obtained by Aparicio *et al.* (2002). The validation of the numerical analyses is made through load versus displacement curves, load versus prestressing stress, and concrete stress diagrams.

Regarding the analysis steps, firstly, the prestressing is applied at 28 days, together with the girder's dead load. The next step is the application of the instantaneous loading on the girder up to its failure. In order to better analyze the results, two cases are numerically studied: simulations with

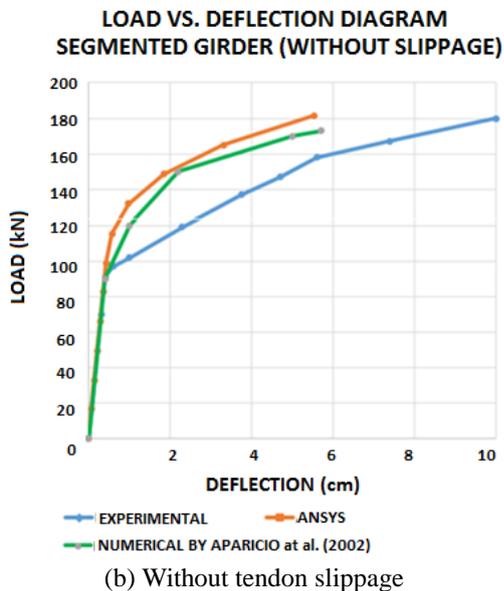
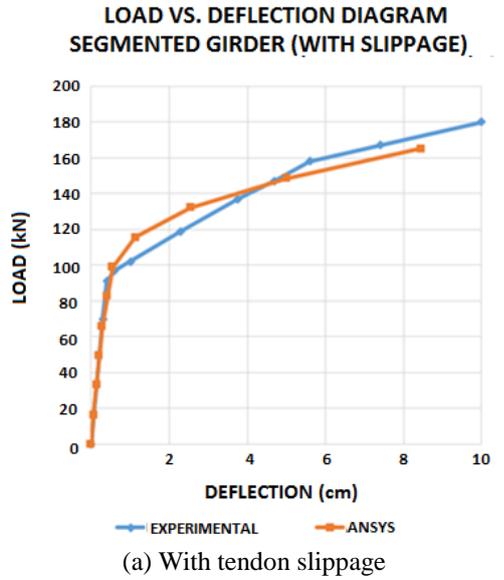


Fig. 14 Load versus displacement diagrams for the segmented girder

and without slippage of tendons at the deviators.

In the article by Aparicio *et al.* (2002), an experimentally obtained load versus displacement diagram for the case with slippage and a numerically calculated load versus displacement diagram for the case without slippage are presented. In Fig. 14, a comparison between the results obtained herein numerically with ANSYS and the two diagrams presented by Aparicio *et al.* (2002) can be observed. The displacement is measured at the bottom of the girder's midspan and the load axis of the load versus displacement diagram is obtained by multiplying the value of the support vertical reaction by two.

In general, the results, in terms of load versus displacement diagrams, present a good agreement between the showed curves, as well as presenting numerical failure loads quite similar to the one found experimentally. When analyzing the load versus displacement diagram for the segmented girder without tendon slippage (Fig. 14(b)), it is

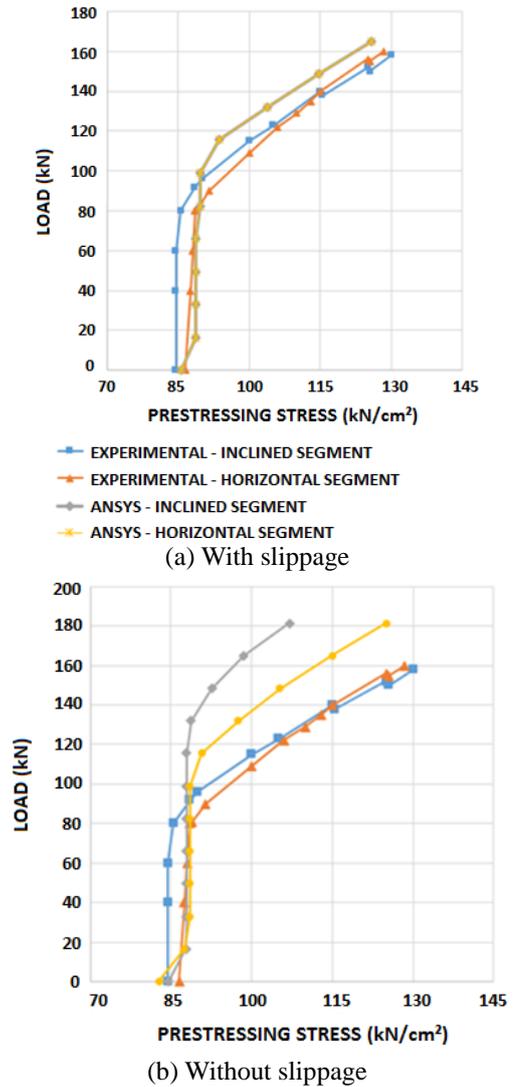


Fig. 15 Load versus prestressing stress diagram for the segmented girder

possible to notice a good agreement between the numerical curve obtained with ANSYS and the numerical curve presented in the article by Aparicio *et al.* (2002). Additionally, from the graphs, it can be observed how the consideration of slippage changes the numerical response for the girder. When the axial displacement is allowed along the prestressing tendons at deviators, the girder experiments higher vertical displacements, as well as presenting a lower strength, when compared with the case when the tendon is fixed at the deviators.

In Fig. 15, the load versus prestressing stress diagrams are presented for the segmented girder with and without tendon slippage. When tendon slippage is simulated (Fig. 15(a)), the stresses both in the horizontal and in the inclined tendon segments present the same value, since friction at the deviators are being neglected. Therefore, only one curve represents the numerical results obtained with ANSYS for both the horizontal and the inclined tendon segments. When observing the experimental results, however, where tendon slippage is present, it can be noticed that each tendon segment has its own curve slightly apart from each other.

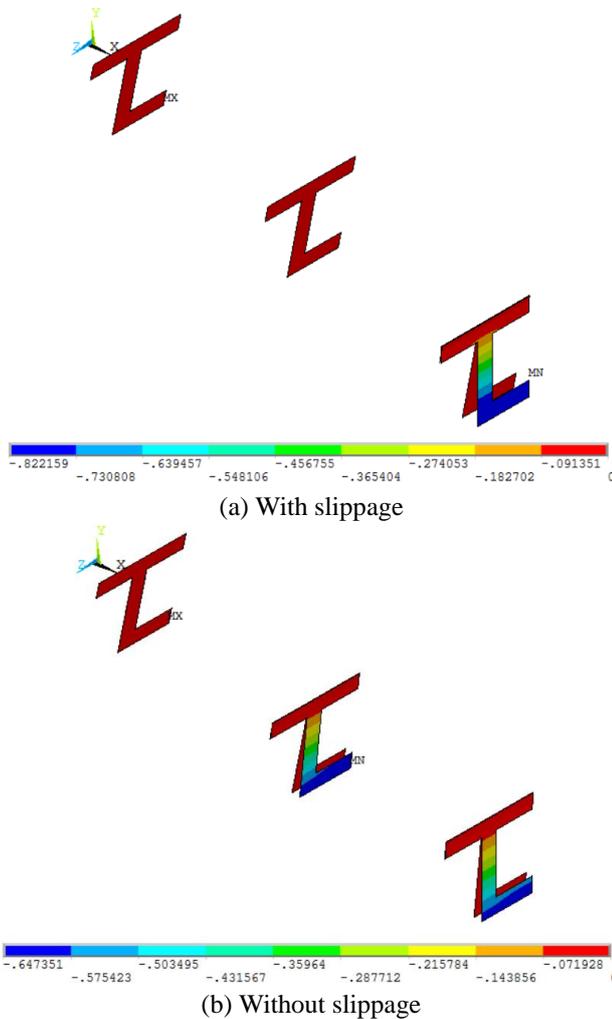


Fig. 16 Representation of the final joint openings for the segmented girder

This effect, therefore, is clearly due to the friction at the deviators.

In both graphs, an important increase in tendon stresses can be observed due to joint openings. In Fig. 15(b), it is possible to notice that stresses obtained with ANSYS for the inclined tendon segment are different from the ones obtained for the horizontal segment. These stresses present higher values when compared with the ones obtained for the case without slippage. This occurs because, when there is no slippage, the girder's strength gets slightly higher, the deflections are lower and, consequently, the joints tend to open after a higher load.

The development of the deformed shape for the segmented girder with and without slippage begins with cambering of the girder (dead load and prestressing are acting), and ends with the opening of the joints, not presenting a monolithical type of behavior anymore. When slippage is considered, only the joint nearest to the midspan opens, while a joint opening also follows at the midspan when there is no slippage. The final opening values for each joint can be observed in Fig. 16, where an amplification factor of 10 has been applied for a better visualization.

The stress distribution in the concrete of the segmented

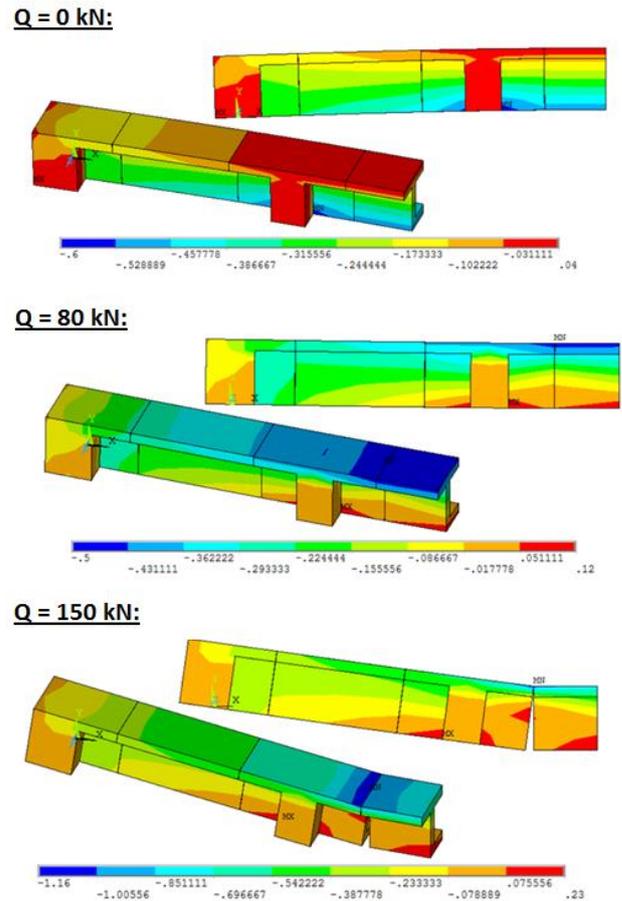


Fig. 17 Diagrams for stress component σ_x in the concrete with slippage (kN/cm^2)

girder with and without tendon slippage is presented in Figs. 17 and 18, respectively. It can be observed that, with the increase in the load, an increase is also observed in the compressive stresses. However, the tensile stresses tend to increase when the girder approaches its Ultimate Limit State, culminating with joint openings. When the joints open, the stresses decrease to zero on their surfaces.

5. Conclusions

It is presented a validation of a viscoelastoplastic model, which is implemented through ANSYS customization system, for a prestressed concrete segmented girder. The results from the numerical analyses show values that satisfactorily agree with those obtained experimentally in the work by Aparicio *et al.* (2002).

The use of the subroutine USERMAT, where the constitutive equations to represent the behavior of the concrete and of the prestressing steel is added, allowed the use of the three-dimensional quadratic element SOLID186, resulting in a model with a lower amount of finite elements and, consequently, in faster and more efficient analyses. Additionally, with USERMAT, it was possible to analyze a structural concrete element with the Element-Embedded Rebar model, i.e., with considerable reduction in mesh size and, consequently, in computational cost, when compared

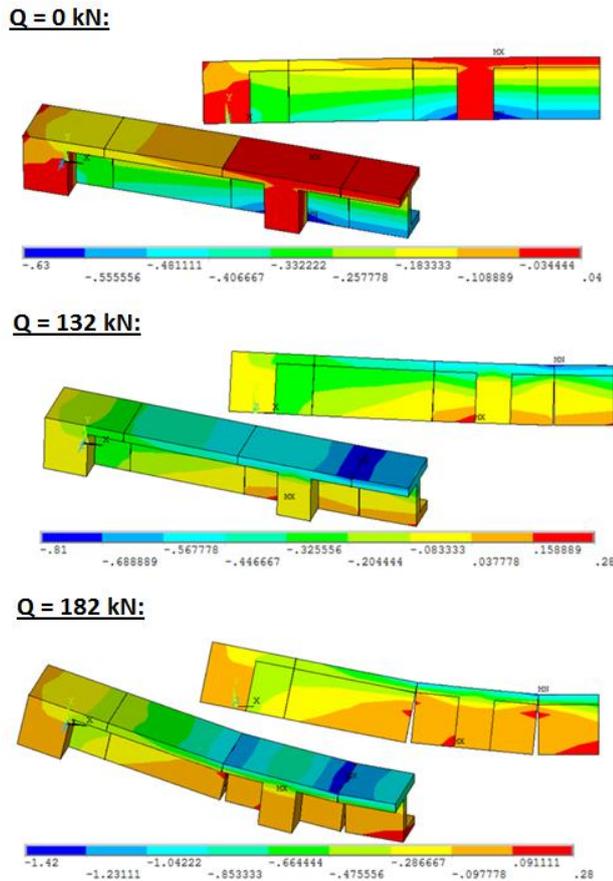


Fig. 18 Diagrams for stress component σ_x in the concrete-without slippage (kN/cm^2)

with the typical choice based on the Discrete Rebar model.

Considering the results obtained with the developed model, it can be observed that the UPF customization tool available in ANSYS allows a structural analysis of the behavior of such structures in a shorter time and in a more precise fashion, optimizing, therefore, the intervenient materials. Therefore, this tool shows to be very useful for computational simulations of reinforced and prestressed concrete structures, making it possible to simulate numerically different elements in structural concrete.

Acknowledgments

The authors wish to acknowledge the financial support given by the Civil Engineering Graduate Program-PPGEC of the Federal University of Rio Grande do Sul-UFRGS, and by the Brazilian governmental research institutions CAPES and CNPq.

References

- Amiri, G.G., Jahromi, A.J. and Mohebi, B. (2012), "Determination of plastic hinge properties for static nonlinear analysis of FRP-strengthened circular columns in bridges", *Comput. Concrete*, **10**(5), 435-455.
- Anil, Ö. and Uyaroglu, B. (2013), "Nonlinear finite element

- analysis of loading transferred from column to socket base", *Comput. Concrete*, **11**(5), 475-492.
- Aparicio, C.A., Gonzalo, R. and Casas, R.J. (2002), "Testing of externally prestressed concrete beams", *Eng. Struct.*, **24**(4-8), 73-84.
- Bulut, N., Anil, Ö. and Belgin, C.M. (2011), "Nonlinear finite element analysis of RC beams strengthened with CFRP strip against shear", *Comput. Concrete*, **8**(6), 717-733.
- Chen, W.F. and Han, D.J. (1988), *Plasticity for Structural Engineers*, Springer-Verlag, New York, U.S.A.
- Comité Euro-International du Béton (2012), *CEB-FIP Model Code 2010*, Bulletin N. 65.
- Demir, S. and Husem, M. (2015), "Investigation of bond-slip modeling methods used in FE analysis of RC members", *Struct. Eng. Mech.*, **56**(2), 275-291.
- Ergun, M. and Ates, S. (2015), "The stress analysis of a shear wall with matrix displacement method", *Struct. Eng. Mech.*, **53**(2), 205-226.
- Hinton, E. (1988), *Numerical Methods and Software for Dynamic Analysis of Plates and Shells*, Pineridge Press Limited, Swansea, Wales, U.K.
- Kazaz, I. (2011), "Finite element analysis of shear-critical reinforced concrete walls", *Comput. Concrete*, **8**(3), 143-162.
- Lazzari, B.M., Campos Filho, A., Lazzari, P.M. and Pacheco, A. R. (2017), "Using element-embedded rebar model in ANSYS for the study of reinforced and prestressed concrete structures", *Comput. Concrete*, **19**(4), 347-356.
- Lazzari, B.M. (2015) "Finite element analysis of reinforced and prestressed concrete elements under plane stress states", M.S. Dissertation, Federal University of Rio Grande do Sul, Porto Alegre, RS, Brazil.
- Lazzari, P.M. (2016) "Numerical simulation of construction stages of cable-stayed bridges through the finite element method", Ph.D. Dissertation, Federal University of Rio Grande do Sul, Porto Alegre, RS, Brazil.
- Oner, E., Yaylaci, M. and Birinci, A. (2015), "Analytical solution of a contact problem and comparison with the results from FEM", *Struct. Eng. Mech.*, **54**(4).
- Ottosen, N.S. (1977), "A failure criterion for concrete", *J. Eng. Mech. Div.*, **103**(4), 527-535.
- Shaheen, Y.B.I., Mahmoud, A.M. and Refat, H. (2016), "Structural performance of ribbed ferrocement plates reinforced with composite materials", *Struct. Eng. Mech.*, **60**(4), 567-594.
- Vasudevan, G. and Kothandaraman, S. (2015), "RC beams retrofitted using external bars with additional anchorages-a finite element study", *Comput. Concrete*, **16**(3), 415-428.
- Wahab, M.A. (2014), *The Mechanics of Adhesives in Composite and Metal Joints: Finite Element Analysis with ANSYS*, DEStech Publications, Inc., Pennsylvania, U.S.A.

CC