cet
47: (2024)
ISSN: 1668-9178

DOI: https://doi.org/10.70198/cet505

Optimización de anclajes y acopladores para sistemas de postensado mediante modelos numéricos

CORDERO, Mariela [1]; APARICIO BENGOECHEA, Ángel [2]

[1] MK4 World Wide, S.L., Polígono Industrial Can Nadal, Calle Can Nadal s/n, Nave 1-A, 08185 Lliçà de Vall, Barcelona, España.
[2] Universidad Politécnica de Cataluña (UPC), ETSI Caminos, C.- y P., Dpto. Ingeniería Civil y Ambiental, Edificio C1, Calle Jordi Girona 1-3, 08034 Barcelona, España.

Resumen

En la actualidad, la mayoría de los sistemas de postensado se basan más en resultados experimentales que en un estudio científico exhaustivo que combine la teoría con la experimentación. El objetivo de este trabajo es presentar la optimización de un modelo numérico existente para diseñar cabezas de anclaje y acopladores de la empresa MK4 World Wide, S.L. Con este proyecto se pretende reducir las dimensiones de estos productos, lo que contribuye al ahorro de material y mejora la calidad. Aquí se presentan los resultados obtenidos del anclaje 12/0.6” y del acoplador 24/0.6” pertenecientes a la familia de postensado MK4 TK (López et al., 2008; Oñate, 1995).

Palabras clave: postensado, placas y acopladores de anclaje, modelo numérico

Abstract

Currently, most post-tensioning systems are based more on experimental results than on an exhaustive scientific study combining theory with experimentation. The objective of this paper is to present the optimization of an existing numerical model to design anchor heads and couplers of the company MK4 World Wide, S.L. This project aims to reduce the dimensions of these products, which contributes to material savings and improves quality. Here the results obtained from the anchorage 12/0.6” and the coupler 24/0.6” belonging to the MK4 post-tensioning family are presented (López et al., 2008; Oñate, 1995).

Key words: *post-tensioning, anchor head and coupler, numerical model*

Introduction

Innovation and quality improvements are the new challenges in the civil sector. Improving the design of post-tensioning systems fosters their advantages in a sustainable way in the medium and long term (López et al., 2008; Oñate, 1995). Numerical models and their comparative with experimental results are exposed in this paper (Oliver and Agelet de Saracibar, 2003).

This work presents part of the results obtained in an extensive project that was developed together with the UPC, Barcelona Tech. Numerical models and their comparative with experimental results are exposed ins this paper. These results are applied for the optimization of two existing anchorages of the Mk4 post-tensioning system.
For this study, a commercial mesh GiD developed at the International Center for Numerical Methods in Engineering (CIMNE), was used. The finite element code DRAC,

developed at the Department of Ground Engineering of the Polytechnic University of Catalonia was used for the calculations.

Model for anchorage–wedge–strand

A post-tensioning anchorage is formed for three basic components: anchor head, wedge, prestressing steel strand. These three elements have very different mechanical properties. Other property difficult to estimate is the friction coefficient between these components.

In a first phase of the study, a reliable and verifiable model is fitted based on experimental results. For this, two geometrical models are developed:

a) Model 1: wedge and strand are modeled as a single dummy material with conveniently proposed material properties. The contact area between this material and the anchor head is modeled as a transition material, with great shear deformability.

b) Model 2: each element is modeled separately. Two contact areas are defined: one between anchor head and wedge, the other between wedge and strand. The contact area between the wedge and the strand is model with a great deformability at shear stress.

This study is focuses on the behavior of the post-tensioning anchorage and not on the mechanism to transmit the stresses to the structure. Because of this, no analysis about concrete cracking is done.

The anchor head is manufactured with carbon steel C45, this material has to guarantee a yield stress > 350 MPa.

Model 1

Three materials formed the first model (Fig. 1). The anchor head material (in green color) is carbon steel C45 with elasticity modulus E = 210 GPa. The wedge material and the contact area (in magenta and blue, respectively) are designed with parameters to be calibrated.

The blue material is modeled with a great shear deformation and 4mm crown radius. This material is acting as a transition volume.

Finally, the strand and the wedge are modeled with a dummy material (magenta color). This material is more stiffness than the contact area but less than the carbon steel of the anchor head. The friction coefficient for the three materials is 20%.

The characteristics of the mesh used in the finite element model is:

- Nodes: 4000
- Elements: 18000

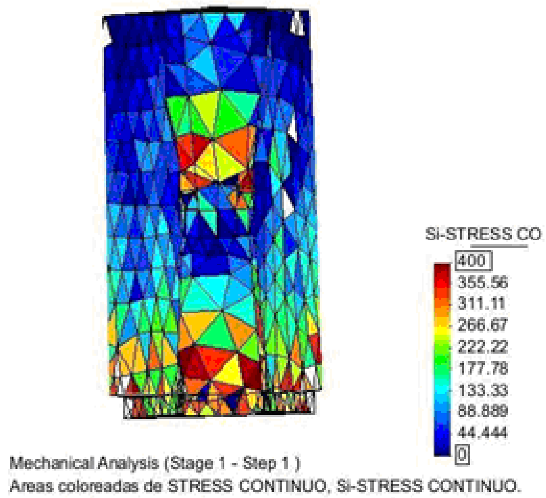

Figure 1: Model 1 – Post-tensioning anchorage

The elements are linear tetrahedrons, with an average size of 4 mm. The material is assumed working in the elastic zone. This behavior has to be corroborated.

Firstly, the model is calibrated using the experimental tests results (Marques, 1984). In its final position, the distance between the upper faces of the wedge and the anchor head is 2,5mm. Calibrate a model is equivalent to determine the elasticity modules of each material to satisfy this final position.

After many calculations using different values, the best result is:

- Elasticity modulus for dummy material = 9,5 GPa
- Elasticity modulus for contact area = 2 GPa
- Elasticity modulus for anchor head = 210 GPa

In this model, the post-tensioning force has been introduced at the top of the dummy material, within a radius of 10 mm, and for 115% of the breaking load to guarantee a higher level of stress. In this way, the force considered for a strand is:

$$F\_max=1.15*140*1860=299460N \quad (1)$$

This force is distributed among the nodes corresponding to the application area.

The deformation distribution is practically non-existent (0.35mm) in the transition material (Fig. 2) making the dummy material stiff enough to simulate the deformations. The fact of considering two fictitious materials does not

cet
47: (2024)
ISSN: 1668-9178

extract additional significant information, and one of them can be dispensed with without loss of veracity. From a conservative point of view, only one dummy material with elasticity modulus = 9 GPa will be used.

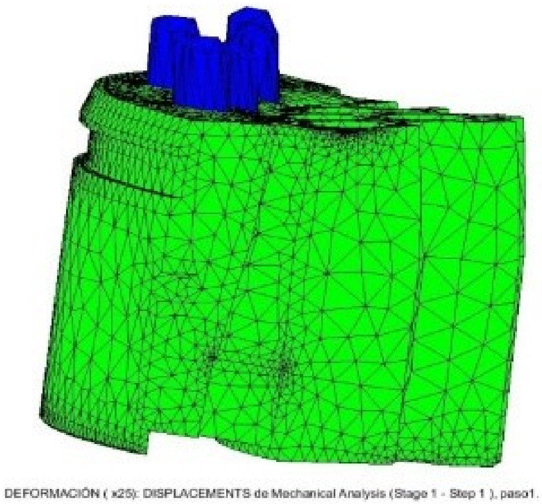

Figure 2:. Displacement in the upper face (Model 1).

In all the simulations the maximum tensile stress is less than 350 MPa so the element does no plasticize and the model is consistent. This model is considered valid for the characteristics given.

Indeed, the main limitation of this model is that it does not correctly collect the stresses in the contact area, since it is simulated with a fictitious material. However, it is useful to establish the stresses in the anchor head. Characteristic that will be the main concern in the optimization that arises.

Model 2

The model 2 consider each element of an anchorage by separate and two contact areas or transition zones (Fig. 3). Therefore, the model has three elements:

a) Anchor head = green color
b) Steel strand = brown color
c) Wedge = magenta color

And two contact areas with high shear stress deformation:

a) Between anchor head and wedge = blue color
b) Between wedge and steel strand = light green

These areas will represent the slippage of the wedge.

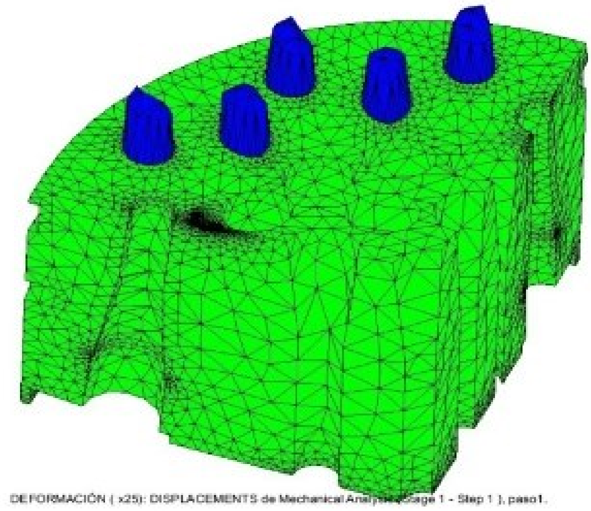

Figure 3: Model 2 – Post-tensioning anchorage.

The characteristics of the mesh used in the finite element model is:

- Nodes: 9000
- Elements: 38000

This mesh is denser because of the contact areas. These areas are taken from 1mm. The elements are linear tetrahedrons with an average size 0 2.5mm. A very relevant point in the definition of the model lies in the definition of the transition zones, which percentage must be sufficient to avoid influencing on the stresses. It is believed that in this model, with transitions of approximately 30% of the wedge thickness, this condition is fulfilled, in compromise with the complexity of meshing areas of small relative size. It is estimated that in this model, where the transitions are approximately 30% of the wedge thickness, this condition is fulfilled, in compromise with the complexity of meshing areas of small relative size.

In this model, the material characteristics are:

- Elasticity modulus for anchor head = 210 GPa
- Elasticity modulus for steel strand = 180 GPa
- Elasticity modulus for wedge = 210 GPa

All the Poisson modulus are equal to 20% as in the first model. The contact areas are calibrated in the same way that in the first model. The same properties are applied to the both

cet
47: (2024)
ISSN: 1668-9178

transition materials because are considered identical. The stress obtained for these materials is 350MPa.

Forces are introduced in the model by imposing actions on the underside of the strand. Deformations are amplified because the length of the strand. For a slippage of the wedge imposed, the deformations are shown in Fig. 4.

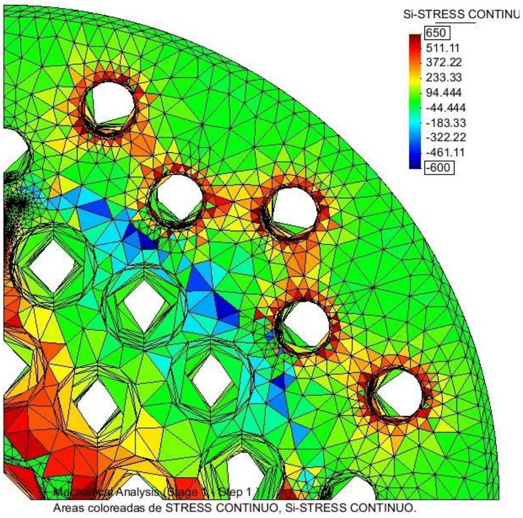

Figure 4: Displacements in the upper face (Model 2).

In this case, the geometry of the deformed mesh is very representative (Fig. 5), since it simulates the wedge penetration phenomenon appropriately. The result obtained in the simulation is very close to 2.5mm, this value coincides with the experimental results (starting hypothesis).

The stresses obtained with model 2 are very similar to the results with model 1. In both cases the stresses in the anchor head are less than 350 MPa. Model 2 represents the contact between wedge – strand and between wedge – anchor head, this is the significant difference with model 1.

This model is very useful to simulate displacements and stresses in the anchorage but in other hand, incorporated numerical and geometry complexity to the model. Unlike the model 1, in this one it is not possible to eliminate the transition elements because they are the basis for the veracity of the model. For this reason, model 2 can be useful in some particular cases where information about contact areas are needed. This model is too complex to evaluate the structural integrity of the anchor head or coupler.

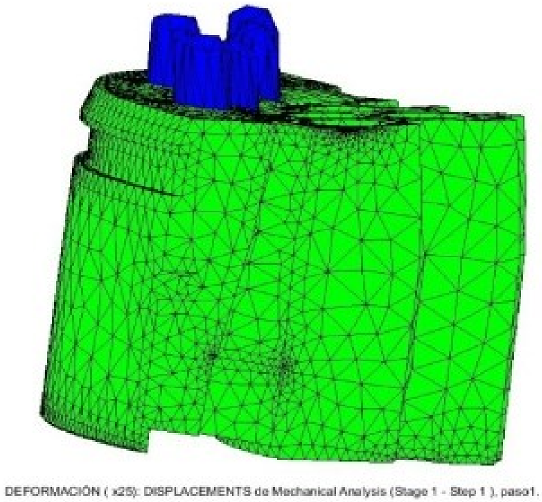

Figure 5: Deformed mesh (Model 2).

Finally, the calibrations performed are used to establish the loads for specifics anchorages and couplers.

Post-tensioning coupler 12/0.6” model

Taking advantage of the symmetry of the coupler 12/0,6”, its structural behavior is modeled by means of 1/8 piece. The respective restraints are simulated in the parallel faces. This model has two materials as model 1:

a) Anchor head = green color
b) Dummy material = blue color

Where their properties are:

- Elasticity modulus for anchor head = 210 GPa
- Elasticity modulus for dummy material = 9 GPa
- Poisson modulus for both materials = 20%

In order to collect the areas of small thickness, the mesh proposed has:

- Nodes: 7000
- Elements: 37000

The elements are linear tetrahedrons with an average size = 3 mm. As in the previous cases, a linear behavior will be assumed and then the stresses will be analyzed.

To simplify the calculation and eliminate the local effects of the lower wedges, the support on the underside of the coupler is modeled (Fig. 6). In Fig. 7, the anchor head displacements and its relatively little flexion are shown. In these

cet
47: (2024)
ISSN: 1668-9178

results, it is observed that the displacements in the support compression area are less than in the rest of coupler. However, what is really important in this case is the differential movements between some areas and others, because the fixed points are not so clearly defined.

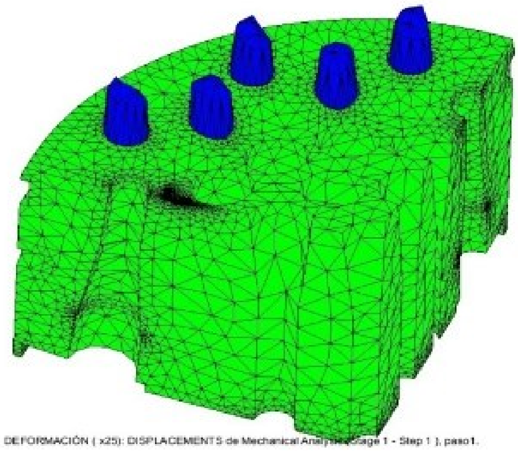

Figure 6: Upper and lower views of the model.

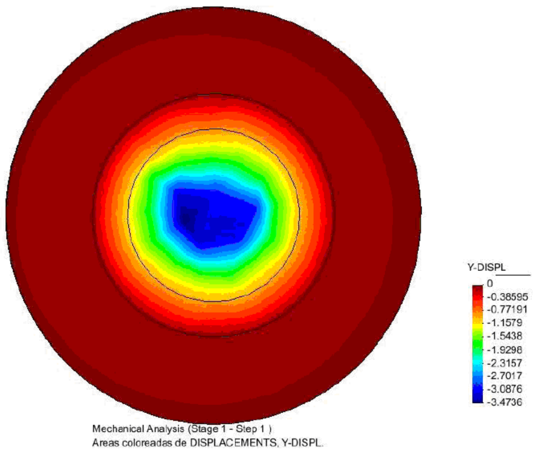

Figure 7: Displacements obtained with the finite element model (1/8 coupler 12/0.6”).

After analyzed the deformations in the piece of coupler, the stresses are studied. A shear failure through tangential stresses on a cylindrical surface perpendicular to the bearing plane is presumed.
A vector flow diagram of maximum tensile stresses is presented to illustrate the stresses mentioned (Fig. 8).
In Fig. 8, it can be seen that the flow generates a zone of high tensile stresses between the two holes of the strands. These stresses reached the tensile stresses of the support. These tensile stresses in the support zone represent a mathematical result of balance due to the imposed conditions. In reality, this area would be supported by two compressions, one on the upper face and one on the lower face, both in contact with the concrete.

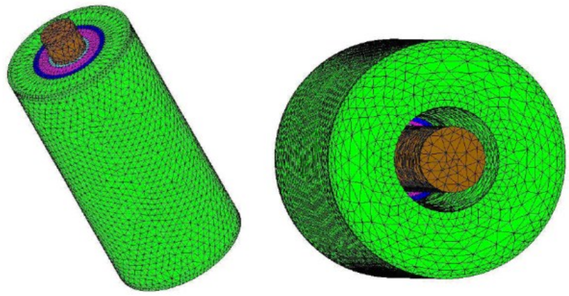

Figure 8: Flow diagram of maximum tensile stresses.

The principal stresses obtained with this model are shown in Fig. 9.

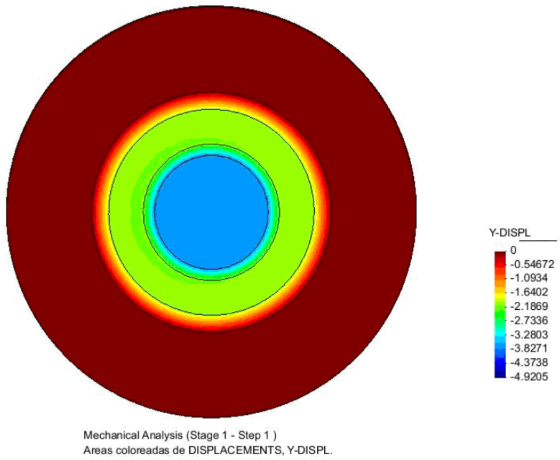

Figure 9: Flow diagram of maximum compression stresses.

Logically, in this scheme the complementary information to Fig. 7 can be observed: the low compression stresses are in the cited plane, while the central area of coupler is requested at large compressions. It should be noted that the intermediate main stress has very small values and is not relevant in this study.
The maximum tensile stresses appeared in the cylindrical plane shown before (Fig. 10). The highest stresses are small and they are lower

47: (2024)
ISSN: 1668-9178

than the material yield stress so the model is stable.

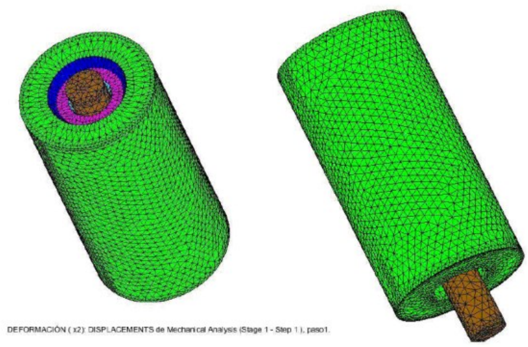

Figure 10: Flow diagram of maximum compression (detail).

Comparing these results with the test ones, it concludes:

- Load applied to de model = 1,15 times strand breaking load
- Stable lineal model

The relation between the test results at 80% loaded and the numerical calculation are:

$$\frac{Exp\ results}{Numerical\ results} = \frac{0{,}8}{1{,}15} = 0{,}6970 \sim 70\% \qquad (2)$$

Then, the numerical stresses are equal to 70% respected experimental results. dummy) and Poisson ratio 0.20.

Post-tensioning coupler 24/0.6" model

The study of ¼ piece of post-tensioning coupler 24/0,6" is presented. The same symmetry criteria, external loads and restraints conditions as in the previous model are applied.

The respective restraints are simulated in the parallel faces. This model has two materials:

a) Anchor head = green color
b) Dummy material = blue color

Where their properties are:

a) Elasticity modulus for anchor head = 210 GPa
b) Elasticity modulus for dummy material = 9 GPa
c) Poisson modulus for both materials = 20%

In order to collect the areas of small thickness, the mesh proposed has:

- Nodes: 7000
- Elements: 37000

The elements are linear tetrahedrons with an average size = 4 mm. In this model, the mesh in the central zone has larger elements because it does not provide relevant information about the behavior of the coupler (Fig. 11). Instead, the mesh is densified in the areas of the holes because these are the most stressed zone. As in the previous cases, a linear behavior will be assumed and then the stresses will be analyzed.

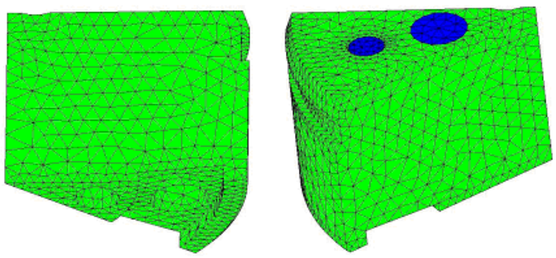

Figure 11: Upper and lower views of the model (24/0.6").

For this modeling, several support or restraints conditions have been studied. A circular support surface in the lower area with a width = 100 mm is adopted because is considered the most consistent with reality. As in the previous studies, the load considered for the calculations is:

$$F_{max} = 1.15 * 140 * 1860 = 299460\text{N} \qquad (3)$$

This load is the steel strand breaking load. In Fig. 12, the coupler's mesh deformed is shown. In the same figure, the bend zone is observed too but these results are not relevant. The importance of these results lies in the relative deformations between points of the mesh around the holes.

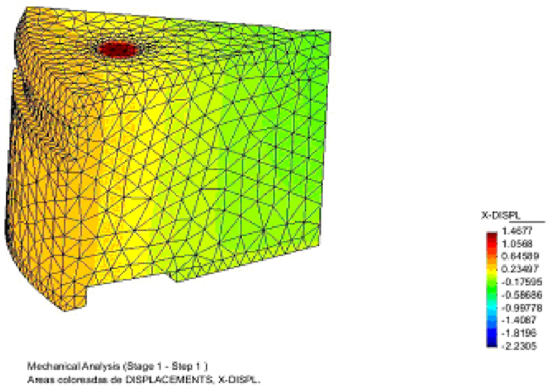

Figure 12: Front and side view of the deformed mesh.

In Fig. 13, the magnitude of the maximum tensile stresses obtained in the calculations are observed. The coupler`s maximum stresses are near 650 MPa, in the tests results with the specimens plasticized the stresses arrived to 450MPa. Thus, this fact must be corroborated by means of a study with an elastoplastic behavior of the material.

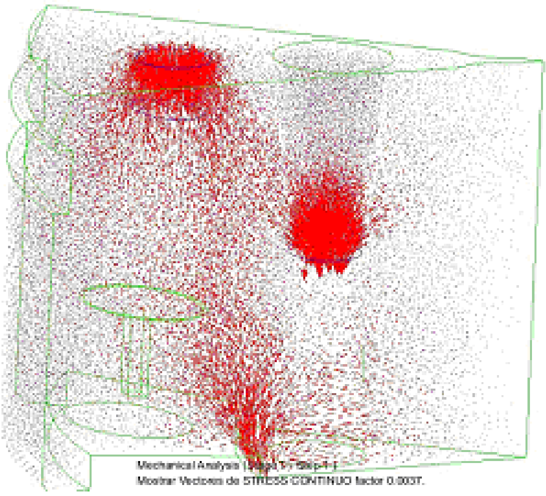

Figure 13: Maximum tensile stresses on the model surface.

In front to these results, it can be concluded that the main susceptibility of the coupler is in the area between strands with same sign. In figure 13, the interaction between the external strands can be seen.
Lastly, it should be mentioned that the stresses in the development of the x-axis (depth) have been studied to see their affectation, and they do not present great variations.

Conclusions

The linear model is a simplification that works well on a qualitative level and is useful for understanding how anchorages and couplers work (López et al., 2008; Oñate, 1995). The models give information about how stresses are distributed within them.
They can be enough to validate existing designs and help to propose optimization strategies, showing which are the critical areas of the pieces.
As an example, Table 1 shows an advance of results of the designs presented in the text and how they have been rated after the numerical study for further optimization.
To specify these optimization strategies, it will be necessary to develop a calculation model that includes elastoplastic materials.

Table 1: Couplers results.

Coupler	Theor.max. Stress	Exp.max. stress	Conclusions
12 strands	360 MPa	250 MPa	Material can be saved by reducing dimensions
24 strands	650 MPa	— (*)	In the absence of a more detailed elastoplastic study, the improvement strategy should be aimed at widening the gap between outer holes.

(*) The real behavior is not linear above 350 MPa.

ACKNOWLEDGEMENTS

The authors wish to thank the Department of Civil and Environmental Engineering of the Technical University of Catalonia (UPC) for their collaboration and technical support in this research.

REFERENCES

López, C.M., Carol, I. & Aguado, A. (2008). “Meso-structural study of concrete fracture using interface elements. I: numerical model and tensile behaviour.” Materials and Structures, 41, 583–599. https://doi.org/10.1617/s11527-007-9314-1

Marques, J. (1984). “Stress computation in elastoplasticity.” Engineering Computations, 1, 42–51. doi not available.

Oliver, J. & Agelet de Saracibar, C. (2003). Mecànica de medis continus per a enginyers. Edicions UPC, Barcelona. doi not available.

Oñate, E. (1995). Cálculo de estructuras por el método de los elementos finitos: análisis estático lineal. Centro Internacional de Métodos Numéricos en Ingeniería (CIMNE), Barcelona. doi not available.

This work was developed during 2024 in the Department of Civil and Environmental Engineering of the Technical University of Catalonia (UPC), Spain.

AUTHORS

CORDERO VERGE, Mariela
Is PhD in Civil Engineering and Director of R&D at MK4 World Wide, S.L., a company specialized in structural systems for bridges and buildings. Her work focuses on product innovation, certification, and numerical modelling of post-tensioning systems, bearings, and expansion joints.
mcordero@mekano4.com

APARICIO BENGOECHEA, Ángel
Is Professor at the Technical University of Catalonia (UPC), specialized in structural engineering and bridge design. His research covers experimental analysis and finite element modelling of post-tensioned structures.
angel.carlos.aparicio@upc.edu