Flexural response of polypropylene fiber reinforced concrete using the fiber composite model

Luís Felipe Ribeiro[1,2], Marcello Congro[1,2], Cristian Mejia[1,2], Deane Roehl[1,2]

[1]Department of Civil and Environmental Engineering / Pontifical Catholic University of Rio de Janeiro – PUC-Rio – Brazil
[2]Multiphysics Modeling and Simulation Group / Tecgraf Institute / Pontifical Catholic University of Rio de Janeiro – PUC-Rio – Brazil
Rua Marquês de São Vicente, 225 – Gávea – Rio de Janeiro – Brazil
luissantos@tecgraf.puc-rio.br[1], marcellocongro@tecgraf.puc-rio.br[2], crisms@tecgraf.puc-rio.br[3], deane@tecgraf.puc-rio.br[4]

Abstract. In the last decades, several laboratory tests have been carried out to investigate the mechanical behavior of fiber-reinforced concrete (FRC), especially regarding their flexural response for several structural applications in Engineering. Modern developments concerning these materials have improved the number of numerical models capable of predicting mechanisms at the material scale and the load-displacement behavior of the composite. Therefore, the preliminary stages of advanced cement material design can profit from mathematical models since the numerical experiments can consider different material working conditions before prototyping. This paper proposes the numerical modeling of polypropylene fiber reinforced concrete beams using a mesoscale formulation called the fiber composite model. This model consists of the coupling of uniaxial fiber finite elements with continuum cementitious elements through cinematic constraint equations, where the fiber degrees of freedom are eliminated at the element level. Additionally, a cohesive zone with a continuum damage behavior is inserted into the formulation to simulate the post-cracking material flexural response. Three-point bending experimental tests reported in the literature are modeled. The results show good agreement between the experimental references and the reinforced cohesive element model to predict fracture patterns for FRC beams.

Keywords. *Fiber-reinforced concrete; Damage; Fiber composite model; Numerical modeling.*

1 Introduction

In recent years, the use of cement composite materials has gained relevance in structural engineering due to the development of composite materials with excellent mechanical properties. Moreover, in a search to combine a sustainable practice with the development of high-performance materials, polypropylene fiber reinforced concrete (PFRC) has been widely employed due to considerable improvements in strain capacity, toughness, and crack control of structural components (Naaman *et al*. [1], Neville [2]).

In order to understand the behavior of PFRC structural elements and determine their load capacity, several modeling techniques can be carried out. At the macroscale level, the PFRC is often considered homogeneous, and the heterogeneity is not considered. At this scale, most studies do not require highly refined and complex methodologies. The rule of mixtures is widely used to determine homogenized properties applied to the Finite Element Method (FEM) of structural components (Mora *et al*. [3]). However, to investigate local effects such as those from fracture propagation, special formulations, such as using Embedded Strong Discontinuities (Häussler-Combe *et al*. [4]), cohesive models (Lópes *et al*. [5]), lattice models (Kozizki and Tejchman [6]), and particle models (Jirásek and Bazant [7]) are required.

At the mesoscale, the numerical modeling of PFRC adopts the explicit representation of three constituents: matrix, fibers, and interfacial transition zone (ITZ). This modeling strategy is important because it makes possible to incorporate each material effect separately and the behavior of the interaction between them. Several studies are presented in the literature for analyzing concrete at the mesoscale. In a recent research, Congro *et al*. [8] present a formulation with cohesive interface elements to represent the fibers and simulate fracture propagation. Bittencourt Jr. et al. [10] adopt coupling elements to join the fiber elements with the matrix. This coupling was adopted considering a penalty variable to simulate the adhesion between components. In this case, the fiber is considered perfectly bonded to the cement matrix. Häussler-Combe et al. [4] present a more in-depth formulation where the Strong Discontinuity Approach (SDA) is used to simulate discrete fractures, and bond laws are employed via bond elements. This approach allows the simulation of all nonlinear effects concerning fiber, matrix, and the interface zone.

This paper proposes the numerical investigation of the mechanical behavior of polypropylene fiber reinforced concrete beams using the fiber composite model initially proposed by Congro *et al*. [9]. In this model, the fiber is embedded in the continuous matrix element, taking into account fiber orientation, Young's modulus, and cross-section area. A reinforced fracture element is also incorporated into this formulation to simulate the fiber axial forces during the crack opening mechanism.

In order to extend the validation of the fiber composite model to polypropylene fibers, mesoscale analyses are performed to study the influence of random fibers dispersion in the post-cracking stage on three-point bending tests, observing peak load, maximum displacement, and fracture parameters. Finally, fiber composite model results are compared to experimental reference data available in the literature.

2 Methodology

2.1 Fiber Composite Model

The fiber composite model proposed by Congro *et al*. [9] consists of a finite element capable of simulating the mechanical behavior of cement composite materials at the mesoscale, as presented schematically in Fig 1(a). The formulation couples the fiber stiffness matrix (K_f) of a bar element with the concrete stiffness matrix (K_m) (plane stress continuum element) to obtain the stiffness of the fiber-reinforced element (K_{fm}), as presented in Fig. 1(b). The fiber stiffness K_f is transformed to the global orientation using the general transformation matrix (T_e) in terms of the fiber orientation with respect to the horizontal axis (α). Next, the fiber contribution is merged in K_m using a coupled matrix (T_g) by a multifreedom constraint method, Eq. 1. The coupling matrix T_c is composed of 1D shape functions considering the position of the initial and final intersection points between fibers and the edge of continuous elements. The matrix is considered homogeneous and for both constituents we adopt linear elastic behavior.

In order to simulate the fracture patterns and consider the material nonlinearities, cohesive elements are included at the interface between the continuum elements (Figure 1(c)). The fiber element is considered with a spring crossing the cohesive interface element. This inclusion allows the simulation of normal stress transfer mechanisms across the fracture. The constitutive model for the cohesive elements consists of an initial linear elastic behavior up to the peak load. Then, when cohesive strength is reached, a softening branch with exponential damage law is adopted for the critical displacement.

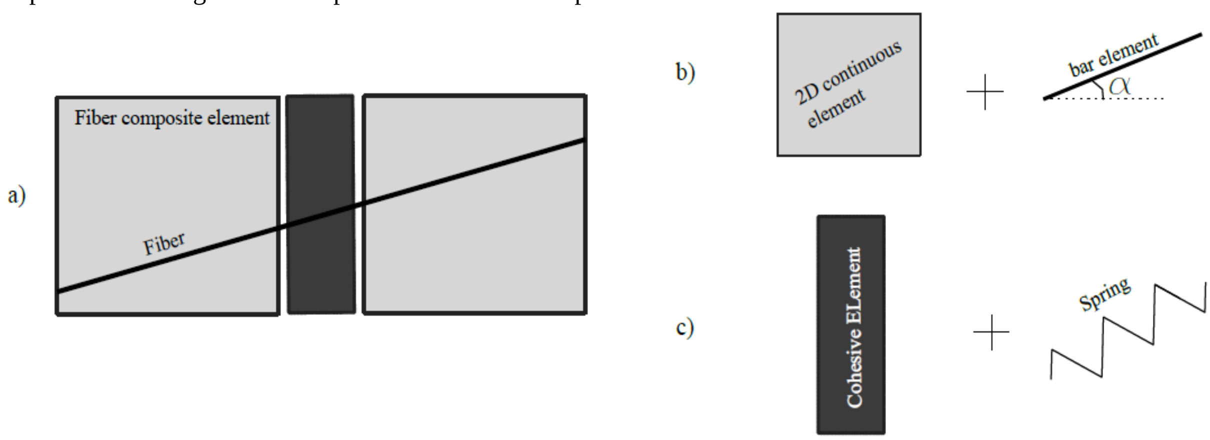

Figure 1. Fiber composite model. (a) general representation of FRC; (b) fiber-matrix element; (c) reinforced cohesive element (Adapted from Congro et al. [9]).

$$K_{fm} = K_m + T_g^T \cdot (T_e^T \cdot K_{fm} \cdot T_e) \cdot T_g \tag{1}$$

The fiber composite model implementation was carried out in framework GeMA (Mendes *et al*. [11]). Random fiber dispersion is included in the numerical model by an external subroutine developed in MATLAB ®. This script is able to create fibers considering an initial structured mesh and identify the intersection points between fibers and the edges of continuous elements. It should be noted that the fiber nodes and the continuous mesh do not need to be overlapped, making the process of mesh generation straight forward.

2.1 Experimental background

The validation of the fiber composite formulation presented in this article adopts two numerical simulations based on experimental tests proposed by Manfredi and Silva [12] and Cifuentes *et al*. [13]. The properties and geometry of three-point bending tests are presented in Tab.1 and Fig.2, respectively.

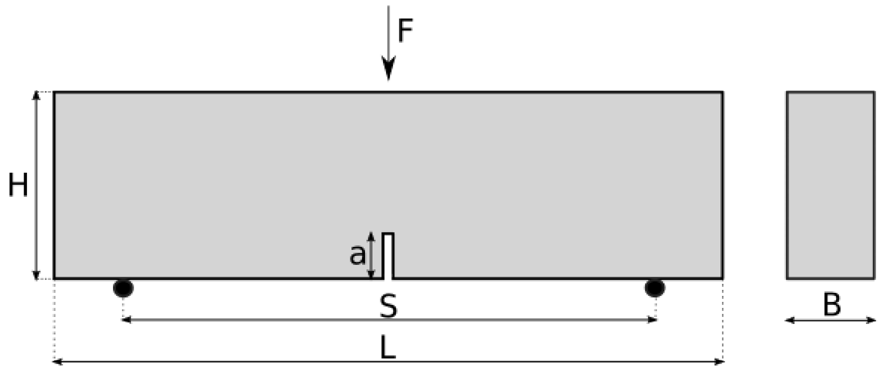

Figure 2. Geometry of the three-point bending tests.

Table 1. Specimen dimensions of the experimental beams.

Experimental test	*L* (mm)	*S* (mm)	*H* (mm)	*B* (mm)	an (mm)
Manfredi and Silva [12]	550	500	150	150	25
Cifuentes *et al*. [13]	540	480	120	60	6

Tab. 2 presents the experimental parameters of each test. In this way, E refers to the Young modulus of the cement matrix; *v* refers to the Poisson ratio; E_f refers to the fiber Young Modulus, A_f is the fiber cross-section area, and R_f refers to the fiber volumetric fraction.

Table 2. Physical input parameters of experimental test.

Experimental test	*E (GPa)*	*v*	$E_f (GPa)$	$A_f (mm^2)$	$R_f (\%)$
Manfredi and Silva [12]	30	0,2	9,5	0,221	0,32
Cifuentes *et al*. [13]	30	0,2	10	0,00085	0,05

The cohesive interface elements consider the normal (K_n) and tangential (K_t) stiffness parameters. These terms are calibrated considering the ratio between Young Modulus and the interface element thickness, as proposed by Manzoli *et al*. [14]. The stiffness parameters of cohesive elements are presented in Tab.3.

Table 3. Physical input parameters of experimental test.

Experimental test	*t (mm)*	*Kn (GPa/m)*	*Kt (GPa/m)*
Manfredi and Silva [12]	120	250	250
Cifuentes *et al.* [13]	60	500	500

3. Results e Discussions

3.1 Manfredi and Silva [12]

The 2D plane stress finite element model is a structured mesh with 240 quadrilateral linear elements (Q4) in the outer regions and an unstructured mesh in the middle region with 2046 linear triangular elements (T3), as presented in Fig. 3. Cohesive elements were introduced at the edges between the continuum elements in the central region of the beam.

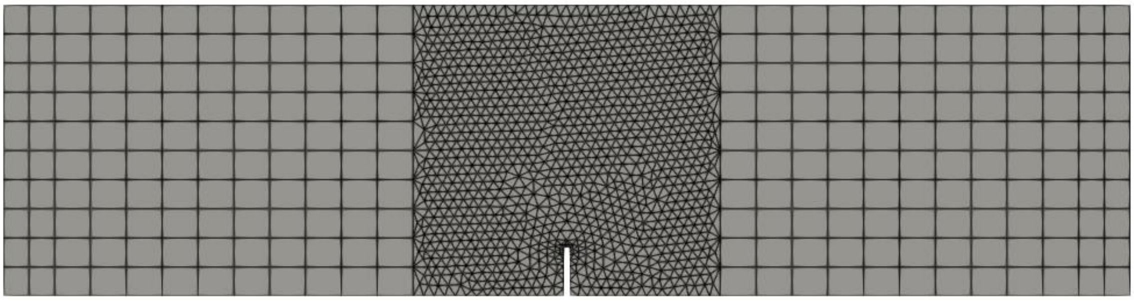

Figure 3. Finite element mesh for the experimental test of Manfredi and Silva [12].

Figure 4 presents the stress-CMOD global response comparing the numerical and experimental curves. Good accuracy can be observed between the pre-peak curve for both results. In the softening branch, close responses are observed. The inclusion of fibers promotes a less accentuated post-peak behavior than the plain concrete model. In addition, the most extensive area under the curve can be observed in the model with fibers. It indicates a non-immediate failure, demonstrating the higher energy absorption capacity of the composite.

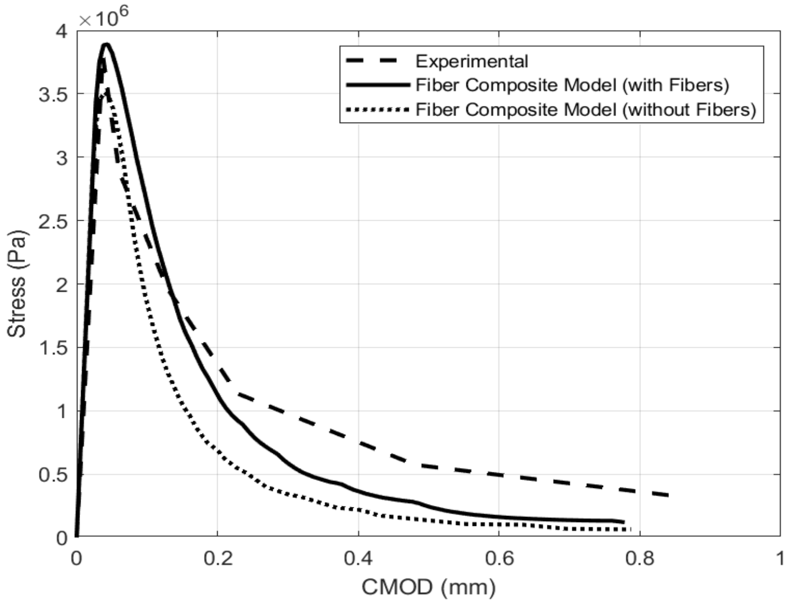

Figure 4. Stress-CMOD curves for the three-point bending test of Manfredi and Silva [12] comparing experimental results and composite numerical model.

3.2 Cifuentes *et al.* [13]

The second application considers an experimental test carried out by Cifuentes *et al*. [13]. The geometry of the prismatic beam is presented in Tab. 1. A 2D plane stress model is adopted with 1082 linear triangular elements containing 3246 nodes (Fig. 5). The crack path was simulated using cohesive elements at the interface between continuum elements. The normal and shear stiffness parameters are presented in Tab. 2.

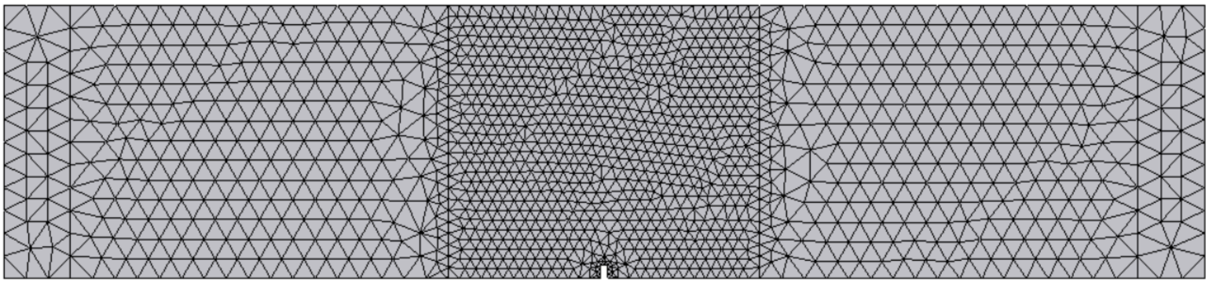

Figure 5. Finite element mesh for the experimental test of Cifuentes *et al*. [13].

Figure 6 presents the load-CMOD curve to verify the suitability of the the fiber composite model. One higher load peak, greater ductility, and energy absorption capacity gain represented by the bigger area under the softening branch. In this sense, it is possible to see the crack control proposed by the inclusion of the fibers by the difference between the softening branches presented in the load-displacement curves.

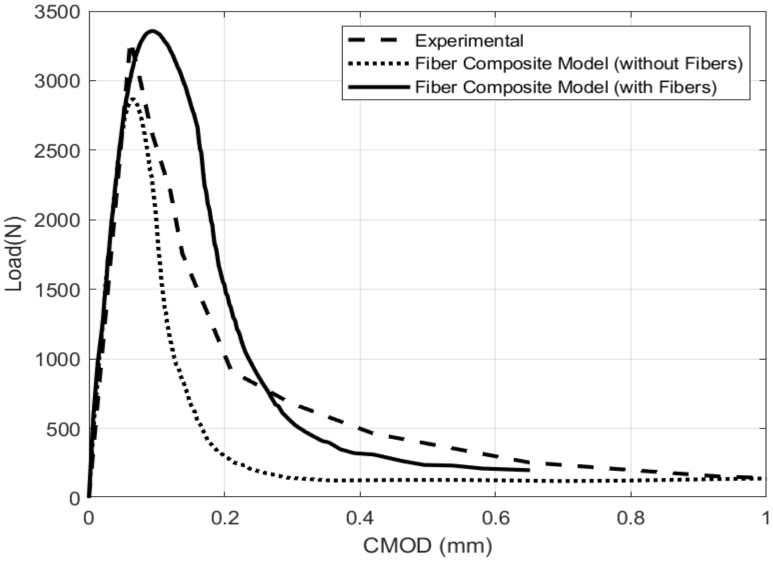

Figure 6. Load-CMOD curves for the Three-point bending test of Cifuentes *et al*. [13].

Figure 7 presents the crack propagation patterns obtained with the fiber composite model using enriched cohesive elements in the middle region of the beam. Four stages with the crack path evolution provide by fibers are shown in Figure 7(a) to (d).

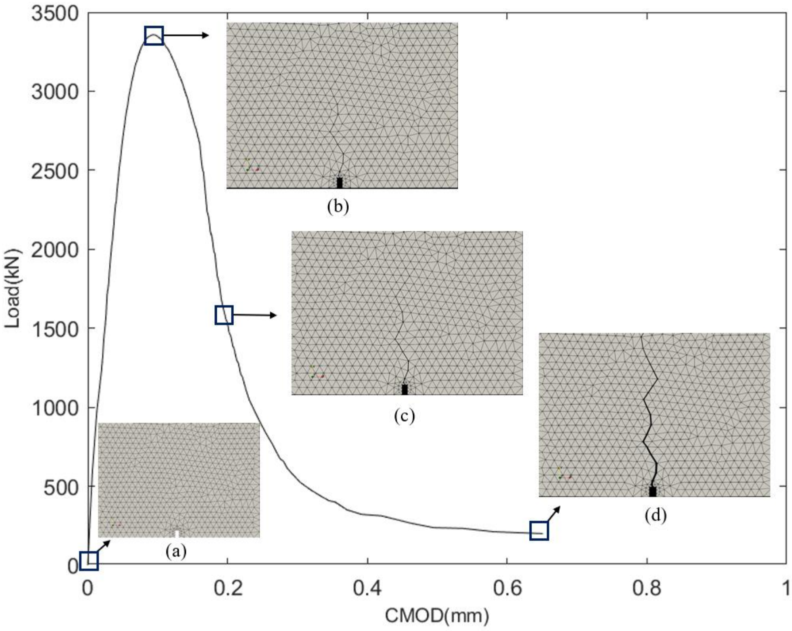

Figure 7. Fracture patterns identified in the load-CMOD composite behavior for the three-point bending test carried out by Cifuentes *et al*. [13].

Figures 8(a) to 8(d) show the crack pattern evolution in the beam. A major crack grows in the central region of the specimen with small changes in its trajectory due to the unstructured mesh and the presence of fibers. The crack path response is in close agreement with the reported experimental fracture pattern (Figures 8(b) and 8(d)).

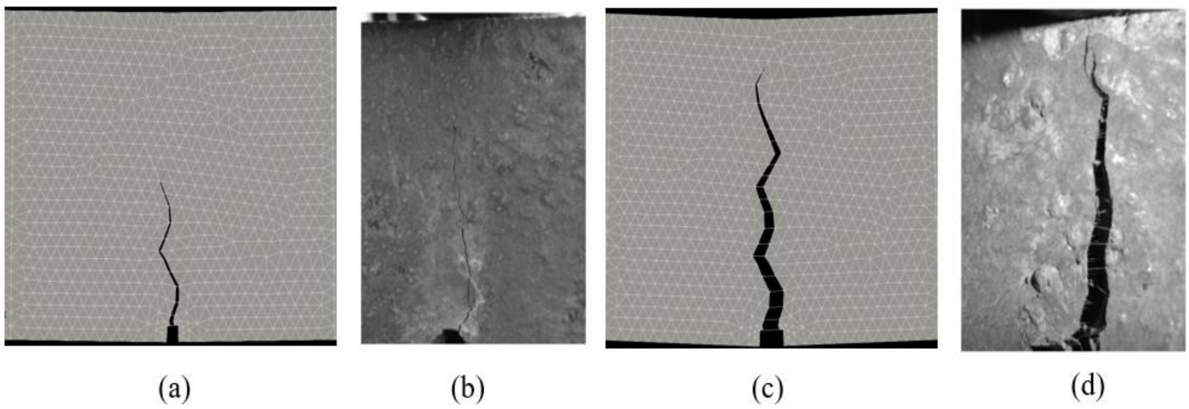

Figure 8. Crack propagation patterns: (a) of the numerical analysis using fiber composite model; (b) of the experimental test of Cifuentes *et al.* [13] (c) at the end of the numerical analysis using the fiber composite model; (d) at the end of the experimental test of Cifuentes *et al*. [13].

4. Conclusions

This paper proposes a numerical model to describe the mechanical behavior of PFRC using the fiber composite model originally proposed by Congro *et al.* [9]. The simulations performed in this article seek to extend the validation process of the fiber composite model for polypropylene fibers. Until then, they have only been tested for experimental applications considering steel fibers.

The fiber composite model is able to represent adequately the flexural response of cement composite materials. The load-CMOD results are close to the experimental curves in terms of global and local fracture behavior. In general, the inclusion of fibers promotes a higher peak load, greater fracture energy, and ductility.

All these points have been captured by the numerical model. Moreover, the crack pattern was observed and compared with the experimental reference, reaching good levels of similarity.

Finally, although the formulation considers only the linear elastic behavior for fiber and matrix, the model can represent the PFRC behavior as demonstrated. However, for a more realistic simulation, an elastoplastic constitutive model can be included in the formulation to simulate the mechanical behavior of the polypropylene fiber-reinforced concrete. Moreover, the fiber composite model does not consider fracture mode II or a mixed crack opening displacement behavior. Therefore, a future improvement of the formulation considers the inclusion of fiber slip and debonding effects.

Acknowledgments. This study was financed in part by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior – Brasil (CAPES) – Finance Code 001, Conselho Nacional de Pesquisa (CNPq) – Grant 870122/1997-5.

Authorship statement. The authors hereby confirm that they are the solely 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] Naaman, A. E.; Argon, A.; Moavenzadeh, F. A Fracture Model for Fiber-Reinforced Cementitious Materials. Cement and Concrete Research, v. 3(4), p. 397-411, 1973.
[2] Neville, A. M. Properties of Concrete. 5ª edição, Longman, 2011.
[3] D.F. Mora, J. Oliver, and Huespe A.E. Computational modeling of fiber-reinforced cement composites as a complex material. In *Congress on Numerical Methods in Engineering 2011.*
[4] Häussler-Combe, U., Shehni, A. and Chihadeh, A., 2020. Finite element modeling of fiber-reinforced cement composites using a strong discontinuity approach with explicit representation of fibers. *International Journal of Solids and Structures*, *200*, pp.213-230.
[5] Carlos M. Lópes, Ignacio Carol, and Antonio Aguado. Meso-structural study of concrete fracture using interfacial elements. II: compression, biaxial and brasilian test. *Materials and Structures*, 41(3):601-620,2008
[6] J. Kozizki and J. Tejchman. Modeling of fracture process in concrete using a novel lattice model. *Granular Matter, 10(5):377-388,2008.*
[7] Milan Jirásek and Z.P. Bazant (1995). Particle model for quasi-brittle fracture and application to sea ice. *Journal of Engineering Mechanics, 121(9): 1016-125,1995*
[8] Congro, M., Sanchez, E. C. M., Roehl, D., & Marangon, E. (2019). Fracture modeling of fiber reinforced concrete in a multiscale approach. Composites Part B: Engineering, 174, 106958.
[9] Congro, Marcello, Deane Roehl, and Cristian Mejia. "Mesoscale computational modeling of the mechanical behavior of cement composite materials." *Composite Structures* 257 (2021): 113137.
[10] Bitencourt Jr, L.A., Manzoli, O.L., Bittencourt, T.N. and Vecchio, F.J., 2019. Numerical modeling of steel fiber reinforced concrete with a discrete and explicit representation of steel fibers. International Journal of Solids and Structures, 159, pp.171-190.
[11] Mendes CAT, Gattass M, Roehl D. The GeMA framework: an innovative framework for the development of multiphysics and multiscale simulations. In: Proceedings of VII European Congress on Computational Methods in Applied Sciences and Engineering. Crete Island, Greece; 2016. p. 7886–7994. https://doi.org/10.7712/100016.2383.6771
[12] Manfredi, Raphael Palmier, and Flávio de Andrade Silva. "Test methods for the characterization of polypropylene fiber reinforced concrete: A comparative analysis." *KSCE Journal of Civil Engineering* 24.3 (2020): 856-866.
[13] Cifuentes, Héctor, et al. "Influence of the properties of polypropylene fibres on the fracture behaviour of low-, normal- and high-strength FRC." *Construction and Building Materials* 45 (2013): 130-137.
[14] Manzoli, OL, Maedo, MA, Bitencourt, LAG, Rodrigues, EA, 2016. On the use of finite elements with a high aspect ratio for modeling cracks in quasi-brittle materials. Eng. Fract. Mech. 153, 151–170, (2015).