XXXVI Ibero-Latin American Congress on Computational Methods in Engineering

CILAMCE

Rio de Janeiro, 22-25 Nov 2015

GFEM NONLINEAR ANALYSIS USING MICROPLANE CONSTITUTIVE MODELS

Anderson Renato Vobornik Wolenski

andersonwolenski@gmail.com

Anelize Borges Monteiro

ane.lize.eng@gmail.com

Roque Luiz da Silva Pitangueira

roquepitangueira@gmail.com

Samuel Silva Penna

spenna@dees.ufmg.br

Felício Bruzzi Barros

feliciobbarros@gmail.com

Universidade Federal de Minas Gerais. Av. Antonio Carlos, 6627, Belo Horizonte, MG, Brazil

Abstract. *One of the most broadcast methods to the nonlinear analysis of structures is the Finite Element Method (FEM). However, there are phenomena whose behavior is not satisfactorily treatable by the conventional form of FEM, fact which has quickened the development of new strategies such as the Generalized Finite Element Method (GFEM), understood as a variation of the FEM. In parallel, nonlinear analysis of concrete structures requires the use of constitutive models that represents the nucleation and propagation of cracks. In this paper it has been adopted a Microplane Constitutive Model to represent the concrete under the GFEM approach. Such model is incorporated in the Unified Theoretical and Computational Environment to Constitutive Models implemented on the INSANE system (INteractive Structural ANalysis Environment). Based on the experimental responses obtained by Petersson (1981), numerical simulations are carried out aiming to discuss the influence of the GFEM enrichment strategy.*

Keywords: *INSANE, Physically Nonlinear Analysis, Anisotropic Constitutive Model.*

1 INTRODUCTION

Several engineering problems use partial differential equations relating field variables inside a particular domain. To obtain the analytical solution it is used some numerical method since such problems have complex geometry and boundary conditions. In this context, it was developed the Finite Element Method (FEM), which is an efficient numerical resource to solve boundary value problems.

Nonetheless, there are phenomena whose behavior cannot be satisfactorily described by conventional FEM and this fact has motivated the development of new strategies. According to Kim and Duarte (2015), the difficulties with the direct incorporation of a strong discontinuity into the element interior have been tackled by the Generalized Finite Element Method (GFEM). Such method relies on the construction of the so-called patch approximation spaces in a node-by-node manner, and hierarchically enriches the standard FEM approximation space using the Partition of Unity method (PU).

In this paper it has been used a computational platform named INteractive Structural ANalysis Environment (`INSANE`) (Fonseca, 2006). Such system has been expanded by Alves (2012) to enclose the GFEM formulation with minimum impact in the code structure and with requirements for extensibility and robustness. Monteiro *et al*. (2014) has performed an expansion of this GFEM implementation to physically nonlinear analysis.

The current version of the `INSANE` system presents many resources, such as: (1) extensive library of analysis models, constitutive models and methods for the solution of nonlinear models; (2) discrete models of Finite Element Method, Generalized Finite Element Method, Boundary Element Method and Mesh Free Methods; and (3) an Unified Theoretical and Computational Environment for Constitutive Models.

In respect to the Unified Theoretical and Computational Environment for Constitutive Models developed by Penna (2011), it has been included on this environment, by Wolenski (2013), a model class named Microplane Constitutive Model, based on work of Leukart (2005). Such model is applied as a strategy to describe the anisotropy of the material and it will be used in the simulations of this paper.

Based on this context, it is presented the results of numerical simulations on a concrete beam subjected to the three-point bending using GFEM and nonlinear analysis, comparing the responses with the experimental ones obtained by Petersson (1981). The main purpose is discussing the influence of the enrichment strategy of GFEM that can influence the numerical responses.

2 GENERALIZED FINITE ELEMENT METHOD

In comparison with the conventional FEM, the GFEM presents some advantageous characteristics, such as the improvement of shape functions to treat specific problems. For instance, aiming to reproduce a high gradient field, special functions can be explicitly introduced by enriching the original approximate solution. Additionally, the use of polynomial enrichment functions associated with each node separately provides a very flexible p-refinement strategy (named here p-enrichment). Different regions of the mesh can be enriched by different sets of polynomial functions, depending on the kind of phenomenon to be simulated.

Nevertheless, according to Duarte *et al.* (2000), the use of this enrichment function can result in obtaining an equation system whose stiffness matrix is positive semi-definite and it occurs due to enrich a PU with monomials. To overcome this problem, the application of the numerical strategy proposed by Strouboulis *et al.* (2000) is recommended.

The enrichment used in GFEM consists of employing functions of the Partition of Unity (PU) type that are improved by their multiplication by additional terms (not necessarily polynomials) specially chosen to the kind of analyzed problem. Those PU functions are defined over a cloud, or patch of elements that enriched define the shape functions. The conventional functions of FEM (such as Lagrangian functions) facilitates the application of the GFEM and, differently from the meshless methods, directly verifies the boundary conditions (Barros, 2002).

The clouds are formed by sets of finite elements that share the same nodal points x_j (on Fig. 1 represented by ω_j). For example, in $\mathbb{R}^1$ the PU is formed with the use of linear Lagrangian functions (on Fig. 1 represented by $\mathcal{N}_j(x)$).

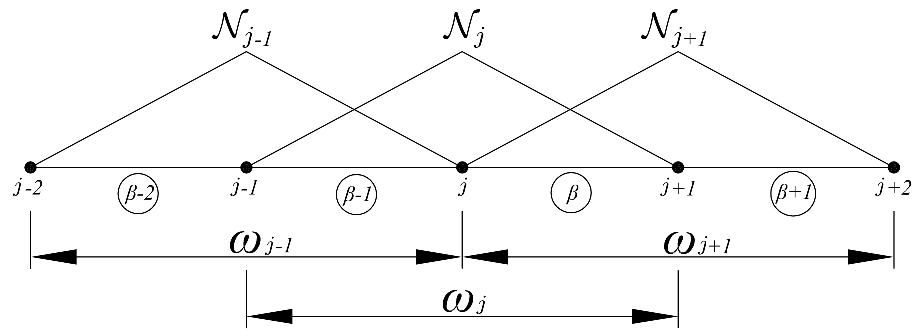

Figure 1: Definition of the PU from one-dimensional finite elements.

The Lagrangian Finite Element functions N_j, associated with each one of the n nodes, can be considered as a PU because for any position x:

$$\sum_{j=1}^{n} \mathcal{N}_j(x) = 1 \ . \tag{1}$$

In summary, it is considered a conventional mesh of finite element defined from a set of n nodal points $\{x_j\}_{j=1}^{n}$, according to Fig. 2(a), in $\mathbb{R}^2$. It is defined a patch or cloud ω_j formed by all elements that share the nodal point x_j.

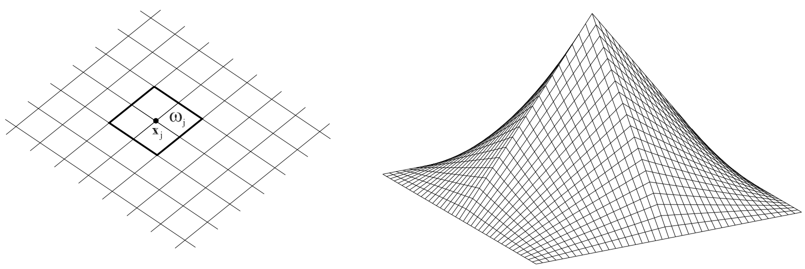

Figure 2: PU functions to the cloud ω_j: (a) Mesh with cloud ω_j; (b) PU function $\mathcal{N}_j(x)$ (Barros (2002)).

The set of interpolative Lagrangian functions associated with the node x_j defines the function $\mathcal{N}_j(x)$ whose support corresponds to the region ω_j, according to Fig. 2(b).

A set of enrichment functions, so-called *local approximation functions*, is composed by q_j linearly independent functions defined to each node x_j with support on the cloud ω_j:

$$\mathcal{I}_j = \{L_{j1}(x), L_{j2}(x), ..., L_{jq}(x)\} = \{L_{ji}(x)\}_{i=1}^{q}, \;\; with \; L_{ji}(x) = 1 \; . \tag{2}$$

Finally, the shape functions $\phi_{ji}(x)$ of GFEM, shown on Fig. 3(a), associated with the node x_j are built through the enrichment of the PU functions by the components of the set $\mathcal{I}_j$.

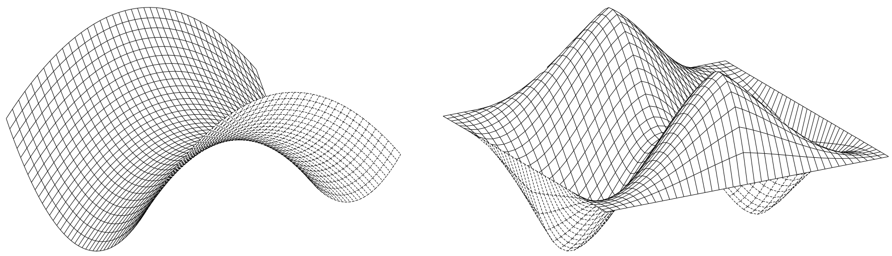

Figure 3: Local approximation and enrichment of the cloud ω_j: (a) Local approximation L_{ji}; (b) $\phi_{ji}(x) = \mathcal{N}_j(x) \times L_{ji}$ (Barros (2002)).

Thus, according to the Eq. (3), $\phi_{ji}(x)$ can be obtained by the product between the basic functions that form the PU (Fig. 2(b)) and the enrichment functions (Fig. 3(a)).

$$\{\phi_{ji}\}_{i=1}^{q} = \mathcal{N}_j(x) \times \{L_{ji}\}_{i=1}^{q} \;\; (no \; summation \; on \; j). \tag{3}$$

The functions of the Eq. (2) can be polynomial or not depending on the problem analyzed. The use of the functions of FEM as the PU simplifies the implementation and avoids, according to Barros (2002), problems related to the numerical integration and to the imposition of the boundary conditions.

Thus, a generic approximation $\tilde{u}$ is obtained by the following linear combination of the shape functions:

$$\tilde{u}(x) = \sum_{j=1}^{N} \mathcal{N}_j(x) \left\{ u_j + \sum_{i=2}^{q} L_{ji}(x) b_{ji} \right\}, \tag{4}$$

where u_j and b_{ji} are nodal parameters associated with standard ($\mathcal{N}_j$) and GFEM ($\mathcal{N}_j \times L_{ji}(x)$) shape functions, respectively.

Furthermore, aiming to minimize round-off errors, Duarte *et al.* (2000) suggest that a transformation should be performed over the $L_{ji}(x)$ functions, when they are of polynomial type. In such case, the coordinate x is replaced as follows:

$$x \rightarrow \frac{x - x_j}{h_j}, \tag{5}$$

in which h_j is the diameter of the largest finite element sharing the node j.

It is obtained the product function that presents the approximative characteristics of the local approximation function while inherits the compact support of the PU.

3 MICROPLANE CONSTITUTIVE MODEL

The application of Microplane Theory (Mohr (1900) and Taylor (1938) *apud* Leukart (2005)) to modeling concrete structures is quite pertinent because the association between solid structure of the heterogeneous material (cementitious matrix with aggregates of different particle sizes) and the existence of multiple plans of discontinuities positioned at the interfaces of its grains (Figure 4).

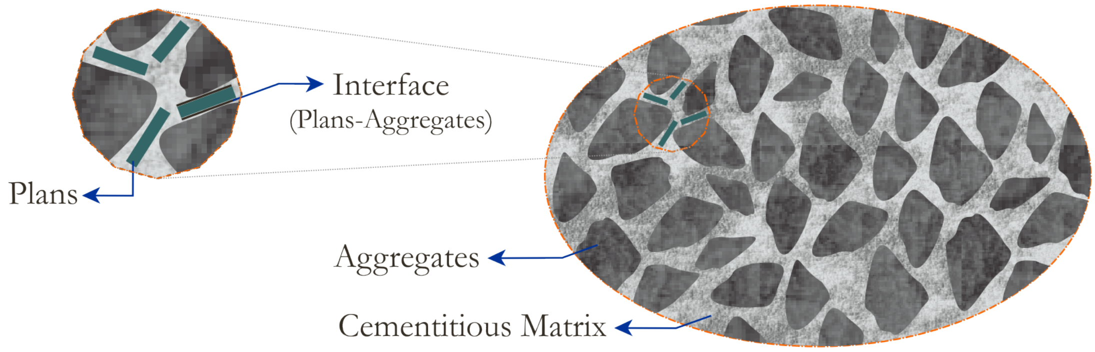

Figure 4: Discontinuity plans inside concrete microstructure.

Although this association has been understood on a micro scale, Bažant and Gambarova (1984) formulated a macro scale model based on such association. The model prescribes constitutive behavior in individual and independent microplanes and relates this local behavior with macro stresses and strains in order to represent the inelastic response of the material. After this pioneer work, many other microplane models has been proposed such as Carol *et al.* (2001), Ožbolt *et al.* (2001), Leukart and Ramm (2002), Leukart and Ramm (2003), D'Addetta (2004), Bažant and Caner (2005), Leukart (2005), Leukart and Ramm (2006) e Fuina (2009). All of them follow the schema showed in Figure 5.

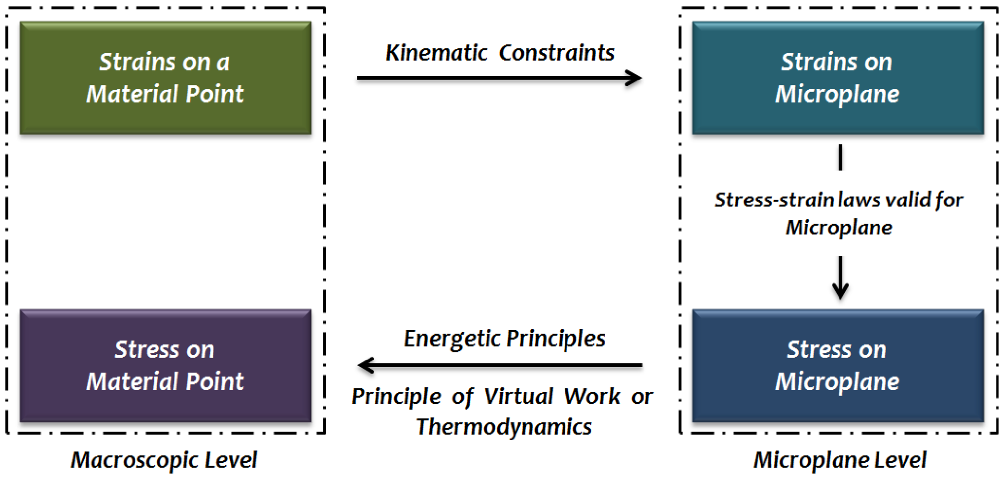

Figure 5: Synthesis of the Microplane Models formulation.

In a very general way, the formulation of Microplane Models follows three main steps, as can be seen in Figure 5. Given the macroscopic strain tensor at a material point, kinematic constraints are imposed in order to calculate the strains on the microplanes. The direction of each microplane is defined to be normal to the surface of a sphere centered at the material point (Figure 6). After imposing such constraints, a local constitutive behavior is assumed in order to evaluate the local stresses as well as a microplane measure of the material degradation. In the final stage, macroscopic stresses as well as the global constitutive tensor are evaluated after using an energy principle.

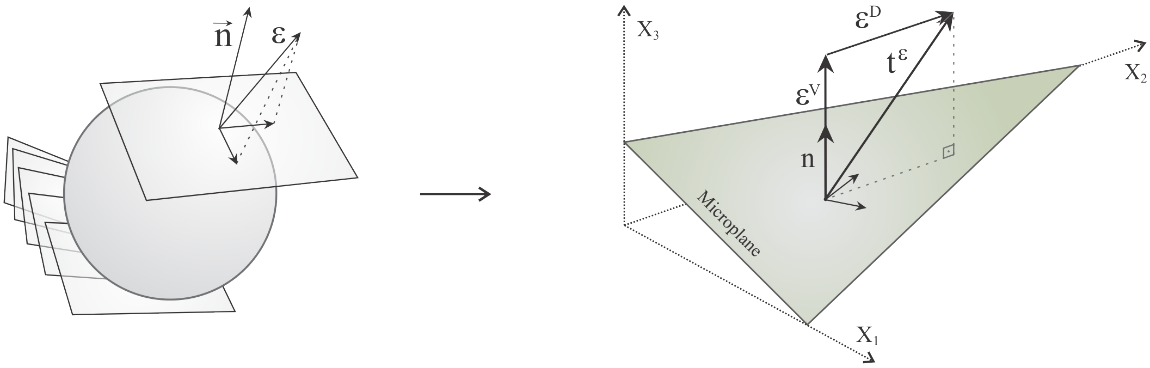

Figure 6: Microplane positions and kinematic constraints.

The model proposed by Leukart (2005) adopts (step 1) a decomposition of the macroscopic strain tensor into its volumetric and deviatoric components (V-D split); (step 2) that the damage process is the main dissipation mechanism which describes the degradation on the material and that degradation is evaluated through a single equivalent strain combined with a single damage law; (step 3) that the free energy on the microplanes exists and its integral over all microplanes is equal to the macroscopic free energy of *Helmholtz*.

Wolenski (2013) generalizes the computational implementation of Leukart (2005) proposition, in order to allow any microplane equivalent strain measure and any damage law. Such an improvement has been implemented in the context of the Unified Computational Environment, proposed by Penna (2011), on the INSANE system.

The numerical simulations presented in this paper use one of the options of the Unified Environment for microplane models of the INSANE system. Specifically, the simulations uses volumetric-deviatoric strain split proposed by Leukart (2005) and the equivalent strain defined by de Vree *et al.* (1995), according to:

$$\eta_{Vree} = 3k_1\varepsilon^V + \sqrt{(3k_1\varepsilon^V)^2 + 3/2\ k_2\varepsilon_p^D\varepsilon_p^D}, \tag{6}$$

where η_{Vree} is equivalent strain measure, ε^V is volumetric part of the strain tensor, ε_p is the p component of the deviatoric strain tensor and k_1 and k_2 are material parameters that relate to tensile and compression of concrete, and an exponential damage law, given by:

$$d^{mic} = 1 - k_0/k\left\{1 - \alpha + \alpha e^{[\beta(k_0-k)]}\right\}, \tag{7}$$

where d^{mic} is the damage measure, k is the current equivalent strain, k_0 is a material parameter that specifies a limit for k referring to the beginning of the damage process, and α and β are others material parameters.

4 INSANE SYSTEM

The INSANE (INteractive Structural ANalysis Environment) system is an open source software implemented in Java, an Object Oriented Programming (OOP) language. It contains three great applications: pre-processor, processor and post-processor. The pre and post-processor are interactive graphical applications that provide resources to build different discrete representations of a structural problem as well as the visualization of the results. The processor is the numerical core of the system and it is responsible for performing the numerical analysis. The persistence of data among such three segments is performed by structured data files written in eXtensible Markup Language (XML) format (Fowler and Scott, 2000).

This system has advantageous resources and its large library allows independent and simultaneous implementations. To achieve it, there is a constant concern in conceiving the implementations at the highest level of generalization, permitting future applications with minimum changes in the pre-existing code.

The items below provide an overview of the INSANE system, mainly in relation to the GFEM formulation to physically nonlinear analysis. Other aspects of the implementation mentioned are presented in Alves (2013) and Monteiro *et al.* (2014).

4.1 General View

The general structure of the numerical core is composed by interfaces and classes able to represent different abstractions of the solution by discrete models. The organization of the core is based on relations between the interfaces Assembler and Persistence and abstract classes Solution and Model. Figure 7 shows these modulus and their communication.

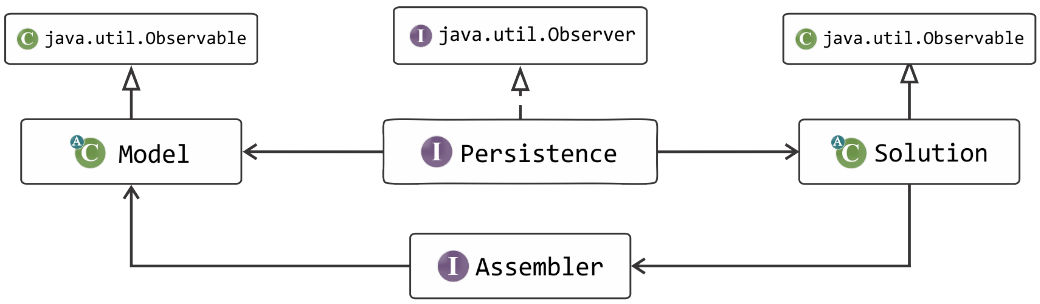

Figure 7: General organization of the numerical core of INSANE computational system.

The interface Assembler is responsible for mounting the second order matrix system presented on Eq. (8) that can represent a wide range of discrete models.

$$\mathbf{A}\,\ddot{\mathbf{X}} + \mathbf{B}\,\dot{\mathbf{X}} + \mathbf{C}\,\mathbf{X} = \mathbf{D}, \tag{8}$$

where $\mathbf{X}$ is the vector of state variables of the problem; $\dot{\mathbf{X}}$ and $\ddot{\mathbf{X}}$ are respectively the vectors that represent the first and second variations of state quantities; $\mathbf{A}$, $\mathbf{B}$ and $\mathbf{C}$ are the matrixes of coefficients and $\mathbf{D}$ is the vector with independent terms of this system.

The abstract class Solution is responsible for triggering the solution process and it contains necessary resources to the resolution of the matrix system (Eq. (8)). The abstract class `Model` contains the data related to discrete model to be analysed and it provides to `Assembler` information to mounting the model equation whose resolution is performed by `Solution`. `Model` and `Solution` communicate with the interface `Persistence` that reads input data and provides output data to other applications whenever there are modifications on the state of the discrete model.

Horstmann and Cornell (2008) affirmed that this observation strategy occurs according to the *Observer-Observable* design pattern, which is a change propagations mechanism. When an object of type *observer* (which implements the interface *java.util.observer*) is created, it is added to a list of observers of other objects of type *observable* (which extends the class *java.util.observable*). When a modification in the state of any observed object occurs, the change propagation mechanism is triggered and the observer objects are notified to update themselves.

This process guarantees the consistence and communication between the observer and the observed components, according to Fonseca (2006). In `INSANE`, the observer component is the interface `Persistence` and the observed components are the `Model` and `Solution` interfaces respectively.

4.2 Physically Nonlinear Analysis

On the computational solid mechanics, the equation system (8), for the case of a static analysis, is reduced to:

$$\mathbf{C} \cdot \mathbf{X} = \mathbf{D} \, , \tag{9}$$

where $\mathbf{C}$ is the global stiffness matrix, $\mathbf{X}$ is the vector of nodal displacements and $\mathbf{D}$ is the vector of nodal forces of the model. If the analysis is nonlinear, matrix $\mathbf{C}$ depends on the displacements $\mathbf{X}$. In this case, the roots of the Eq. (9) are obtained through a incremental iterative process whereby the system of equations (9) is rewritten as:

$$\mathbf{K} \cdot \delta\mathbf{U} = \delta\lambda \cdot \mathbf{P} + \mathbf{Q} \, , \tag{10}$$

where $\mathbf{K}$ is the incremental stiffness matrix, $\delta\mathbf{U}$ is the vector of incremental displacements, $\delta\lambda$ is an increment of the load factor, $\mathbf{P}$ is the vector of reference loads and $\mathbf{Q}$ is the vector of residual loads of the model. The vector $\mathbf{Q}$ is obtained by the difference between the vector of external loads and the vector of equivalent loads to the internal stresses, $\mathbf{F}$.

In GFEM, as in FEM, the matrix $\mathbf{K}$ and the vector $\mathbf{F}$ are assembled from the contribution of each finite element. This process is based on two fundamental operations:

$$\mathbf{K}^e = \int_V \mathbf{B}^T \mathbf{E} \mathbf{B} \, dV \quad and \quad \mathbf{F}^e = \int_V \mathbf{B}^T \boldsymbol{\sigma} \, dV \, , \tag{11}$$

where $\mathbf{K}^e$ is the stiffness matrix, $\mathbf{F}^e$ is the vector of equivalent loads to the current stress state, $\mathbf{B}$ is the matrix of the strain-displacement relations, $\mathbf{E}$ is the constitutive matrix, and σ are the stresses of finite element, being all integrated on a volume $\mathbf{V}$ of each element.

The class `Solution` is responsible for solving Eq. (10). On `INSANE`, the incremental iterative process is implemented on a method of the `StaticEquilibriumPath` class, derived from `Solution` class, as can be seen in Fig. 8.

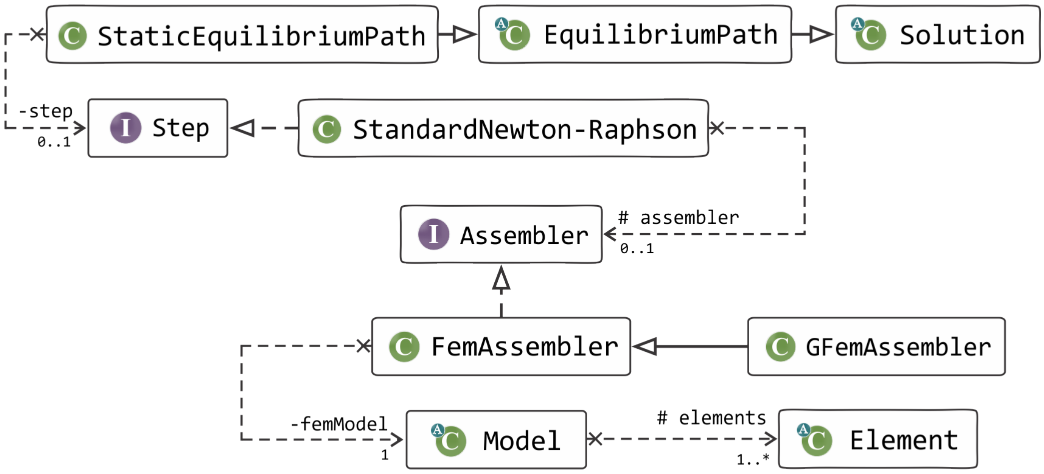

Figure 8: Classes to obtain the incremental stiffness matrix, assembled from FEM or GFEM.

It is also shown that `StaticEquilibriumPath` is supported by the `Step` class that implements the standard version of the Newton-Raphson method. To perform the Newton-Raphson process, class `Step` requires `GFEMAssembler` for mounting all the matrices and vectors of Eq. (10). Aiming this, class `GFEMAssembler` uses its `Model` object to ask class `Element` to integrate matrix K^e and vector F^e of Eq. (11).

5 NUMERICAL EXAMPLE

Petersson (1981) experimentally studied concrete beams subjected to the three-point bending. The experimental results obtained by that author were used here to compare with the numerical simulations performed with GFEM approach using linear and quadratic enrichment functions compared with an approach without enrichment.

The material parameters used by Petersson (1981) and adopted to the numerical simulations are shown in Table 1.

Table 1: Material parameters experimentally obtained by Petersson (1981).

Young's modulus	$E = 30000\ MPa$
Poisson Ratio	$\nu = 0,20$
Uniaxial Yield Stress	$\sigma_t = 3,3 \approx 4,3\ MPa$
Fracture Energy	$G_f = 0,115 \approx 0,137\ N/mm$

The approximation functions, with monomials expressed in coordinates x and y, used to the analysis are defined by:

- $P0$ (No Enrichment):

$$\phi_j^T(x) = \begin{bmatrix} \mathcal{N}_j(x) & 0 \\ 0 & \mathcal{N}_j(x) \end{bmatrix} \tag{12}$$

- $P1$ (Linear Enrichment):

$$\phi_j^T(x) = \mathcal{N}_j(x) \begin{bmatrix} 1 & 0 & \left(\frac{x-x_j}{h_j}\right) & 0 & \left(\frac{y-y_j}{h_j}\right) & 0 \\ 0 & 1 & 0 & \left(\frac{x-x_j}{h_j}\right) & 0 & \left(\frac{y-y_j}{h_j}\right) \end{bmatrix} \tag{13}$$

- $P2$ (Quadratic Enrichment):

$$\phi_j^T(x) = \mathcal{N}_j(x) \begin{bmatrix} 1 & 0 & \left(\frac{x-x_j}{h_j}\right) & 0 & \left(\frac{y-y_j}{h_j}\right) & 0 & \left(\frac{x-x_j}{h_j}\right)^2 & 0 & \left(\frac{y-y_j}{h_j}\right)^2 & 0 \\ 0 & 1 & 0 & \left(\frac{x-x_j}{h_j}\right) & 0 & \left(\frac{y-y_j}{h_j}\right) & 0 & \left(\frac{x-x_j}{h_j}\right)^2 & 0 & \left(\frac{y-y_j}{h_j}\right)^2 \end{bmatrix} \tag{14}$$

Figure 9 depicts the geometric data of the beam and the mesh of 115 four-noded quadrilaterals elements (with 4 x 4 Gauss Quadrature) used in the numerical simulations.

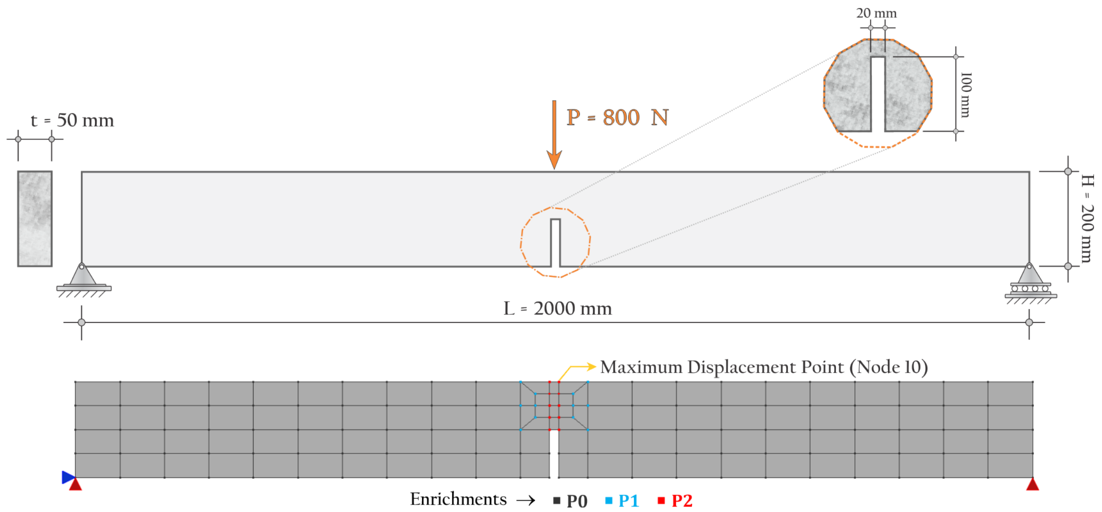

Figure 9: Three-point bending: geometry and mesh used in the numerical simulations.

The numerical simulations are done with the GFEM approach, without enrichment ($P0$, equation 12) and with GFEM applying linear ($P1$, equation 13) and quadratic ($P2$, equation 14) enrichment functions highlighted at the nodes of the Figure 9.

From Table 1, it is possible to obtain the numerical parameters to the exponential damage law (equation 7). These parameters are described in Table 2 and they are grouped according to the equivalent strains defined by de Vree (1995) (equation 6).

Table 2: Parameters for the simulation of the beam tested by Petersson (1981).

Exponential	$\alpha^{mic} = 0,960$	$\beta^{mic} = 500,00$	$\kappa_0^{mic} = 0,0002$

In possession of such parameters, the nonlinear analyses have been performed under plane stress conditions and with the adoption of the generalized displacement control method, with load factor equal to $0,021$, error tolerance of 1×10^{-4}, reference load of $P = 800\ N$.

Figure 10 shows the numerical results of the vertical displacement of the node 10 (Figure 9), together with the experimental results obtained by Petersson (1981).

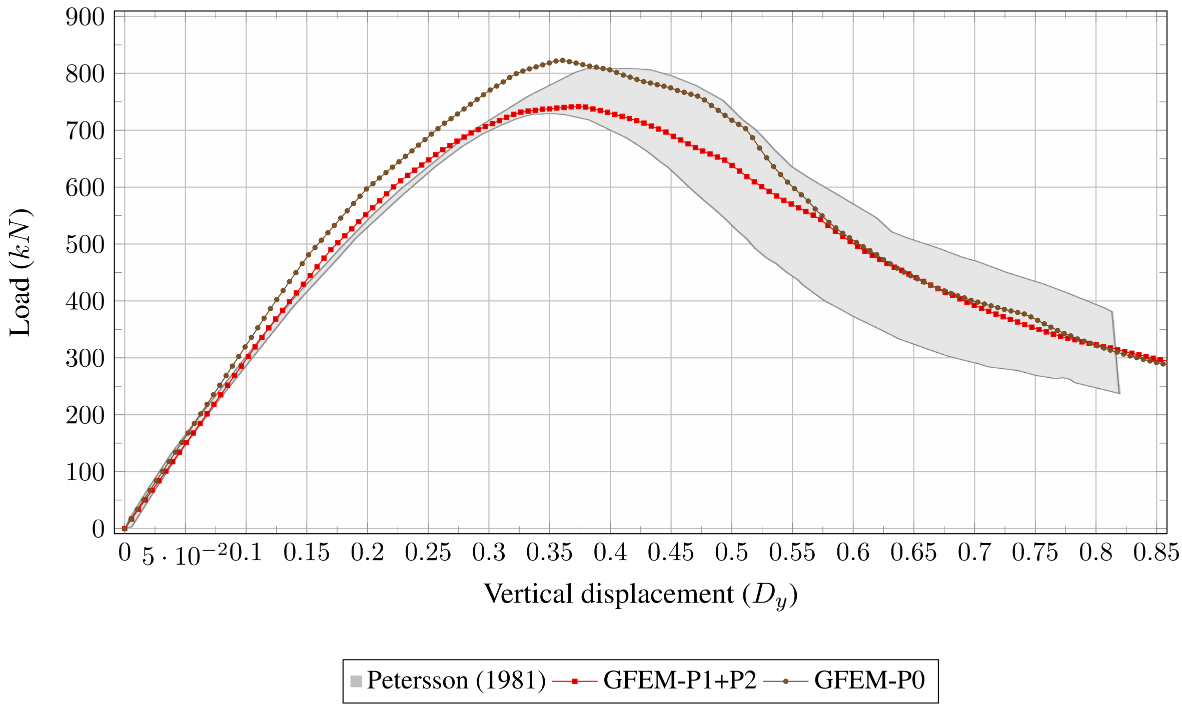

Figure 10: Simulations of the beam of Petersson (1981) to the exponential damage law.

The results presented show good concordance with the experimental one obtained by Petersson (1981) mainly with the application of the GFEM and the enrichment functions. It was possible to estimate the maximum load and to describe stiffness in the inelastic regime.

6 CONCLUSION

This paper presents numerical simulations of a concrete beam subjected to the three-point bending using GFEM approach using polynomials enrichment functions combined with a Microplane Constitutive Model to represent the concrete behavior aiming to compare the results with the experimental obtained by Petersson (1981).

The simulations were carried out with two meshes, one using GFEM and linear ($P1$) and quadratic ($P2$) enrichment functions and the other applying GFEM without enrichment ($P0$).

With the results presented in Figures 10 it is possible to observe that the enrichment strategy under the GFEM approach provided stability to the equilibrium pathes. It should be recognized that the use of the GFEM approach without enrichment (that simulated the use of conventional FEM) provides a worse description of the equilibrium path due to the poorer discretization adopted.

Using a finer mesh or elements with higher polynomial order formulation should improve the nonlinear numerical response similarly to GFEM performance. In the case of a finer mesh, concentrating the smaller elements around the notch requires special attention to avoid element distortion. On the other hand, using higher order elements only on that region would include irregular nodes leading to a constrained approximation. The polynomial enrichment strategy provided by GFEM allows to improve the quality of the approximation around the notch in a very simple and straightforward way.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the important support of the Brazilian research agencies CNPq (in Portuguese ”Conselho Nacional de Desenvolvimento Científico e Tecnológico” - Grants 309005/2013-2, 486959/2013-9, and 308785/2014-2) and FAPEMIG (in Portuguese ”Fundação de Amparo à Pesquisa de Minas Gerais” - Grant PPM-00669-15).

REFERENCES

Alves, P. D., 2012. *Estratégia global-local aplicada ao método dos elementos finitos generalizados*. Masters thesis, Universidade Federal de Minas Gerais.

Barros, F. B., 2002. *Métodos Sem Malha e Método dos Elementos Finitos Generalizados em Análise Não-Linear de Estruturas*. PhD thesis, Universidade de São Paulo.

Bažant, Z. P. & Gambarova, P. G., 1984. Crack shear in concrete: crack band microplane model. *Journal of Structural Engineering*, vol. 110, n. 9, pp. 2015-2035.

Bažant, Z. P. & Prat, P. C., 1988. Microplane model for brittle-plastic material: I. theory. *Journal of Engineering Mechanics*, vol. 114, n. 10, pp. 1672-1688.

Bažant, Z. P. & Caner, F. C., 2005. Microplane model M5 with kinematic and static constraints for concrete fracture and anelasticity. I: Theory. II: Computation. *Journal of Engineering Mechanics*, vol. 131, pp. 1672-1688.

Carol, I., Jirsek, M. & Bažant, Z. P. 2001. A thermodinamically consistent approach to microplane theory. I. Free energy and consistent microplane stresses. *International Journal of Solids Structures*, vol. 38, pp. 31-40.

D'Addetta, G. A., 2004. *Discrete Models for Cohesive Frictional Materials*. PhD thesis, Stuttgart University.

de Vree, J. H. P.; Brekelmans, W. A. M. & van Gils, M. A. J., 1995. Comparison of nonlocal approaches in continuum damage mechanics. *Computer and Structures*, pp. 581-588.

Duarte, C. A., Babuška I. & Oden, J. T., 2000. Generalized finite element methods for three-dimensional structural mechanics problems. *Computer and Structures*, vol. 77(2), pp. 215-232.

Fonseca, M. T., 2006.*Aplicação orientada a objetos para análise fisicamente não-linear com modelos reticulados de seções transversais compostas*. Masters thesis, Universidade Federal de Minas Gerais.

Horstamnn, C.S. & Cornell, G., 2008. *Core Java - Fundamentals*. vol. 1, 8 ed., Sun Microsystems Press.

Fowler, M. & Kendall, S., 2000. *UML Essencial - Um breve guia para a linguagem-padrao de modelagem de objetos*. Bookman.

Fuina, J. S., 2009. *Formulações de modelos constitutivos de microplanos para cont nuos generalizados*. PhD thesis, Universidade Federal de Minas Gerais.

Kim, D.-J. & Duarte, C. A., 2015. A new generalized finite element method for two-scale simulations of propagating cohesive fractures in 3-D. *International Journal for Numerical Methods in Engineering*. DOI: 10.1002/nme.4954.

Leukart, M. & Ramm, E., 2002. An alternative split within the microplane material model. *Proceedings of the IV World Congress on Computational Mechanics*, WCCM Congress, Vienna University of Technology, Vienna, Austria.

Leukart, M. & Ramm, E., 2003. A comparison of damage models formulated on different material scales. *Computational Materials Science*, vol. 28, pp. 749-762.

Leukart, M., 2005. *Kombinierte anisotrope Schdigung und Plastizitt bei kohsiven Reibungsmaterialien*. PhD thesis, Stuttgart University.

Leukart, M. & Ramm, E., 2006. Identification and interpretation of microplane material laws. *Journal of Engineering Mechanics*, vol. 132, n. 3, pp. 295-305.

Mohr, O., 1900. Welche umstnde bedingen die elastizittsgrenze und den bruch eines materiales? *Zeitschrift des Vereins Deutscher Ingenieure*, vol. 46.

Monteiro, A. B., Wolenski, A. R. V. W, Penna, S. S., Barros, F. B. & Pitangueira, R. L. S., 2014. Application of the Generalized Finite Element Method on the physically nonlinear analysis of Structures. *XXXV CILAMCE - Ibero Latin American Congress on Computational Methods in Engineering*

Ožbolt, J., Li, Y., & Kožar, I., 2001. Microplane model for concrete with relaxed kinematic constraint. *International Journal of Solids Structures*, vol. 38, pp. 2683-2711.

Ramm, E., 1981. Strategies for tracing the nonlinear response near limit points. In Wunderlich, W., Stein, E., & Bathe, K. J., eds, *Nonlinear Finite Element Analysis in Structural Mechanics*, pp. 63-89. Springer-Verlag.

Penna, S. S., 2011. *Formulação Multipotencial para Modelos de Degradao Elástica: Unificao Teórica, Proposta de Novo Modelo, Implementação Computacional e Modelagem de Estruturas de Concreto*. PhD thesis, Universidade Federal de Minas Gerais.

Petersson, P. E., 1981. Crack growth and development of fracture zones in plain concrete and similar materials. *Tech Report TVBM-1006, Lund Institute of Technology - Division of Building Materials*, Lund, Sweden.

Strouboulis, T., Babuška, I. & Copps, K., 2000. The design and analysis of the generalized finite element method. *Computer Methods in Applied Mechanics and Engineering*, vol. 181(1-3), pp. 43-69.

Strouboulis, T., Copps, K. & Babuška, I., 2000. The generalized finite element method: an example of its implementation and illustration of its performance. *International Journal for Numerical Methods in Engineering*, vol. 47(8), pp. 1401-1417.

Taylor, G. I., 1938. Plastic strains in metals. *Journal of the Institute of Metals*, vol. 62, pp. 307-324. In Leukart and Ramm (2006).

Wolenski, A. R. V., 2013. *Ambiente Teórico-Computacional Unificado para Modelos Constitutivos: Inclusão de Modelo de Microplanos*. Masters thesis, Universidade Federal de Minas Gerais.