XXXVI Ibero-Latin American Congress on Computational Methods in Engineering

CILAMCE

Rio de Janeiro, 22-25 Nov 2015

A FINITE ELEMENT/GENERALIZED FINITE ELEMENT METHOD CODE-COUPLING FOR LINEAR ELASTIC FRACTURE MECHANICS PROBLEMS

Mohammad Malekan

Felicio B. Barros

Roque L. S. Pitangueira

malekan@dees.ufmg.br

felicio@dees.ufmg.br

roque@dees.ufmg.br

Graduate Program in Structural Engineering from School of Engineering (PROPEEs), Federal University of Minas Gerais (UFMG), Av. Antonio Carlos, 6627, Pampulha, 31270-901, Belo Horizonte, MG, Brazil

Abstract. *This paper presents a procedure for the coupling of a Generalized Finite Element Method (GFEM) code and a commercial Finite Element (FE) package to solve a linear elastic fracture mechanics problem. The code coupling process consists of a GFEM code embedded into the computational platform INSANE (INteractive Structural ANalysis Environment) with the commercial FE package ABAQUS for coupled FEM/GFEM analyses. The problem domain is decomposed into two sub-domains: FE domain (safe domain) and GFEM domain (crack domain). Theoretical background information is provided on the GFEM and the stress intensity factors computation. Main steps of the coupling procedure are explained and necessary information is provided. Details of the implementation are presented and important aspects of using this procedure are highlighted in the numerical example.*

Keywords: *Generalized Finite Element Method (GFEM), eXtended Finite Element Method (XFEM), Object Oriented Programming (OPP), Code-coupling, Fracture Mechanics*

1 INTRODUCTION

The finite element method (FEM) is a powerful method to solve various physical problems. This method is robust and has been thoroughly developed to solve engineering and applied science problems such as linear and non-linear stress analysis of solid and structures, electromagnetism, fluid flows and similar fields. Fracture analysis using standard FEM is quite tedious because the crack geometry needs to conform to element boundaries. In order to model crack propagation, remeshing is always needed to match the new geometry of the crack. The generalized FEM (GFEM) has been proposed to facilitate the modeling of arbitrary crack geometry and its evolution and to eliminate need for remeshing and conformity to element boundaries. In GFEM (Melenk and Babuska, 1996; Strouboulis et al., 2000b; Duarte et al., 2000; Strouboulis et al., 2001), as in FEM, the approximation is built over a mesh of elements using interpolation functions. However, the approximation is associated with nodal points as in the Meshless methods, and it is enriched in the same fashion as in the *hp*-cloud method (Duarte and Oden, 1995; Duarte, 1996; Liszka et al., 1996; Oden et al., 1998). Special functions multiply the original FEM functions and smooth and non-smooth solutions can be modeled independently of the mesh. The enrichment strategy of GFEM is similar to the one used in the extended finite element method (XFEM), introduced in (Belytschko and Black, 1999).

The FEM has proved to be a very qualified numerical technique for solving field problems. Due to the structure and the properties of the resulting system of equations, namely symmetry, sparseness and positive definiteness, highly efficient solvers can be used. Also, GFEM is an interesting alternative for the investigation of problems with high stress concentrations and it has been embedded into INSANE (Interactive Structural ANalysis Environment) by Alves et al. (2013). The INSANE is an open source software available at http://www.insane.dees.ufmg.br and written in Java language. The current GFEM implementation in INSANE is able to solve different types of structural problems with material nonlinearities and it can easily extended to incorporate any kind of enrichment function. Such capabilities can be improved if INSANE is integrated with a high performance FEM package. There are some FE to GFEM/XFEM code-coupling works done by others such as: the coupling of the XFEM with an existing FE analysis software (Bordas and Moran, 2006; Wyart et al., 2007, 2008), XFEM coupled with face-based strain smoothing technique for three-dimensional fracture problems (Jiang et al., 2015), non-intrusive coupling global-local GFEM with a FE commercial package (Gupta et al., 2012). Thus, a combined FEM/GFEM analysis allows the user to benefit from the advantages of both methods, one from available capabilities of FEM commercial package and the other from GFEM code.

In this paper the coupling of a FEM code with a GFEM code is explained. The coupling procedure was implemented using the in-house GFEM code, INSANE, and the commercial FE package ABAQUS. A set of Python codes were developed, which allows the coupling procedure to be run semi-automatically. This means that the user has to create only one model consisting of FEM and GFEM domains. The problem domain is decomposed into two sub-domains: FE domain (safe domain) and GFEM domain (crack domain). The implementation is conducted through the development of some Python codes in order to simulate initial model in ABAQUS commercial package and solve a part of the problem using GFEM code. It allows that force or displacement can be applied as boundary conditions for GFEM parts. Details of the implementation are discussed and important aspects of using these strategies are highlighted in numerical example. This work is organized as follows: A brief explanation of GFEM/XFEM is presented in Sect. 2. The INSANE class organization, including GFEM method is discussed in Sect. 3. Sect. 4 explains code-coupling strategy conducted in this work. In the Sect. 5, numerical example is presented, and final section is devoted to concluding remarks and discussion.

2 THE GENERALIZED/EXTENDED FINITE ELEMENT METHOD

A brief explanation of GFEM/XFEM is described in this section. Further details can be found in (Duarte et al., 2000; Oden et al., 1998; Strouboulis et al., 2001; Melenk and Babuška; Babuska and Melenk, 1997; Belytschko and Black, 1999). The GFEM was developed for modeling structural problems with discontinuities. The origins of this method can be summarized as:

- Research done by Babuška and co-workers – initially named as special finite element method by Babuška et al. Babuska et al. (1994) and later as partition of unity method (PUM) by Melenk and Babuska (1996); Melenk (1995).
- Then, as a meshless formulation in the *hp*-cloud method (Duarte and Oden, 1995) and later as an hybrid approach with the FEM (Oden et al., 1998).

Furthermore, it can be considered an instance of the PUM, in the sense that it employs a set of PU functions to guarantee interelement continuity. Such strategy creates conforming approximations which are improved by a nodal enrichment scheme. This basic idea shares the same characteristics from XFEM proposed by Belytschko and Black (1999), as it is observed in Fries and Belytschko (2010).

Here this procedure is summarized following Barros et al. (2004, 2013) and considering two-dimensional space. A conventional finite elements mesh can be considered for which $\{\mathcal{K}_e\}_{e=1}^{NE}$ is a set of NE elements, defined by N nodes,$\{\boldsymbol{x}_j\}_{j=1}^{N}$.

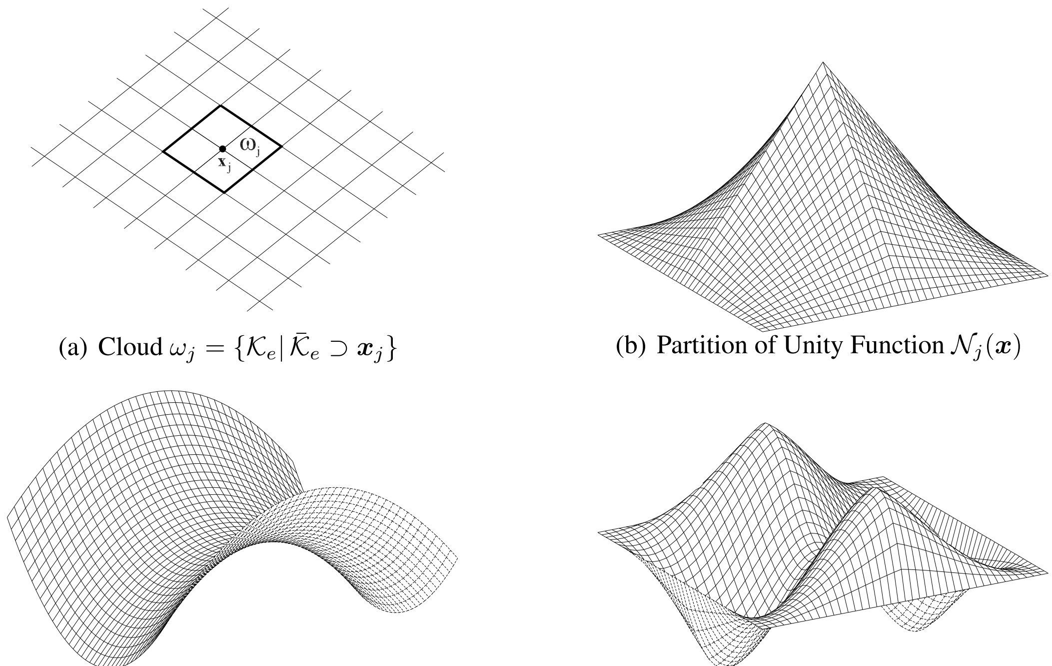
A generic patch of elements or cloud $\omega_j \in \bar{\Omega}$ is obtained by the union of finite elements sharing the vertex node $\boldsymbol{x}_j$, Fig. 1(a). The assemblage of the interpolation functions, built at each element $\mathcal{K}_e \subset \omega_j$ and associated with node $\boldsymbol{x}_j$, composes the function $\mathcal{N}_j(\boldsymbol{x})$ defined over the support cloud ω_j. As $\sum_{j=1}^{N} \mathcal{N}_j(\boldsymbol{x}) = 1$ at every point $\boldsymbol{x}$ in the domain $\bar{\Omega}$, the set of functions $\{\mathcal{N}_j(\boldsymbol{x})\}_{j=1}^{N}$ constitutes a partition of unity (PU). A set of q linearly independent functions is defined at each cloud ω_j as:

$$\mathcal{I}_j \overset{\text{def}}{=} \{L_{j1}(\boldsymbol{x}), L_{j2}(\boldsymbol{x}), \ldots, L_{jq}(\boldsymbol{x})\} = \{L_{ji}(\boldsymbol{x})\}_{i=1}^{q} \quad L_{j1}(\boldsymbol{x}) = 1 \tag{1}$$

The generalized finite element shape functions are determined by the enrichment of the PU functions, which is obtained by the product of such functions by each one of the components of the set $\mathcal{I}_j$ at the generic cloud ω_j:

$$\{\phi_{ji}\}_{i=1}^{q} = \mathcal{N}_j(\boldsymbol{x}) \times \{L_{ji}(\boldsymbol{x})\}_{i=1}^{q} \tag{2}$$

The enrichment scheme is obtained by multiplying a PU function of C^0 type with compact support ω_j, Fig. 1(a), by the function $L_{ji}(\boldsymbol{x})$, Fig. 1(c), named in (Strouboulis et al., 2000a) as a local approximation (also called enrichment function). The resulting shape function $\phi_{ji}(\boldsymbol{x})$, Fig. 1(d), inherits characteristics of both functions, i.e., the compact support and continuity of the PU and the approximate character of the local function.



(a) Cloud $\omega_j = \{\mathcal{K}_e | \bar{\mathcal{K}}_e \supset \boldsymbol{x}_j\}$

(b) Partition of Unity Function $\mathcal{N}_j(\boldsymbol{x})$

(c) Enrichment Function $L_{ji}(\boldsymbol{x})$

(d) $\phi_{ji}(\boldsymbol{x}) = \mathcal{N}_j(\boldsymbol{x}) \times L_{ji}(\boldsymbol{x})$

Figure 1: Enrichment scheme for a cloud ω_j, from Barros et al. (2007)

As a consequence, the generalized global approximation, denoted by $\tilde{\boldsymbol{u}}(\boldsymbol{x})$, can be described as a linear combination of the shape functions associated with each node:

$$\tilde{\boldsymbol{u}}(\boldsymbol{x}) = \sum_{j=1}^{N} \mathcal{N}_j(\boldsymbol{x}) \left\{ \boldsymbol{u}_j + \sum_{i=2}^{q} L_{ji}(\boldsymbol{x}) \boldsymbol{b}_{ji} \right\} \tag{3}$$

where $\boldsymbol{u}_j$ and $\boldsymbol{b}_{ji}$ are nodal parameters associated with standard FE shape function, $\mathcal{N}_j(\boldsymbol{x})$, and GFEM shape functions, $\mathcal{N}_j(\boldsymbol{x}) \cdot L_{ji}(\boldsymbol{x})$, respectively. An example of the enrichment function, L_{ji}, based on polynomials is:

$$L_{ji}(\boldsymbol{x}) = \left(\frac{x - x_j}{h_j} \right)^m \times \left(\frac{y - y_j}{h_j} \right)^p \tag{4}$$

where where (x_j, y_j) are the coordinates of node $\boldsymbol{x}_j$, m and p are degree of polynomials in x and y directions, respectively, and h_j is an scaling factor. Another example for the enrichment function by considering the singularities can be defined as (Barros et al., 2013):

$$\{{}^{x}L_{ji}(\boldsymbol{x})\}_{i=1} = \frac{A_1}{2G} r^{\lambda_1} \{[\kappa - Q_1(\lambda_1+1)] cos\lambda_1\theta - \lambda_1 cos(\lambda_1 - 2)\theta\} \tag{5}$$

$$\{{}^{y}L_{ji}(\boldsymbol{x})\}_{i=1} = \frac{A_1}{2G} r^{\lambda_1} \{[\kappa + Q_1(\lambda_1+1)] sin\lambda_1\theta + \lambda_1 sin(\lambda_1 - 2)\theta\} \tag{6}$$

where λ_1, Q, and A_1 are some coefficients related to the crack and they are a function of relative position of crack to global coordinate system. Also, $\kappa = 3 - 4\nu$ for plane stress analysis and $G = \frac{E}{2(1+\nu)}$.

3 INTERACTIVE STRUCTURAL ANALYSIS ENVIRONMENT

The INSANE environment, (Fonseca and Pitangueira, 2007; Alves et al., 2013), is an open source software implemented in Java, an OOP language. The INSANE computational environment is composed by three great applications: pre-processor, processor and post-processor. The persistence of data among these three segments of INSANE is performed by data files written in extensible markup language (XML) format. Here, a summary of several modules of the numerical core application (the processor) are presented aiming to introduce the INSANE system, corresponding to the standard FEM/GFEM approach, and also to show the generalization performed here to enclose the $GFEM^{gl}$ formulation. More information can be found in (Alves et al., 2013).

The INSANE numerical core is composed by the interfaces *Assembler*, *Model* and *Persistence* and the abstract class *Solution*. Figure 2 shows the unified modeling language (UML) diagram from numerical core of INSANE.

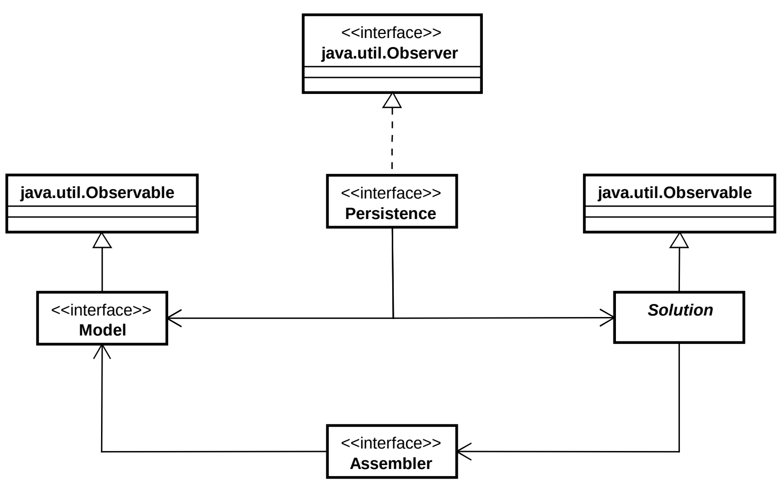


Figure 2: Organization of numerical core of INSANE

Persistence treats the input data and persists the output data. The *Assembler* interface has the necessary methods for assembling the matrix system. Class *Solution* is responsible for solving the final system of matrices. The *Model* interface contains the data of the discrete model and provides to *Assembler* informations to assemble the final matrix system. Both *Model* and *Solution* communicate with the *Persistence* interface, which treats the input data and persists the output data to the other applications, whenever it observes a modification of the discrete model state. The Model interface is composed by the following main attributes:

– *A list of Nodes*

  It is designed to manage the geometric representation of a node entity as well as the information from the discrete model. In addition to information about the node coordinates, an object of this class holds lists of identifiers such as type of degrees of freedom, state variables and type of nodal constraint.

– *A list of Element*

  This class is extends the standard *Element* class in order to encapsulate special characteristics of GFEM, when the numerical enrichment is used in the so-called Global-local strategy (Duarte and Kim, 2008; Kim et al., 2009, 2010, 2012).

– *A list of EnrichmentTypes*

  It is responsible for storing and manipulating information related to the nodal enrichment strategy. The object of class *GFemModel* holds a list of objects of class *EnrichmentType*. Each one of these objects stores the necessary information to perform a different type of enrichment strategy. The class *EnrichmentType* is, in fact, an abstract class.

– *ProblemDriver*

  Informs to *Assembler* all the necessary data for assembling the final system of the model.

4 CODE-COUPLING METHODOLOGY

As explained before, the problem domain is decomposed into two sub-domains: FE domain (safe domain) and GFEM domain (crack domain). The FE domain is modeled using ABAQUS commercial package and GFEM domain is modeled using INSANE in-house code. Based on authors' knowledge, there are two language program that can communicate with ABAQUS and has access to its output files, *C* and *Python* programming languages. The interaction between these two codes is handled using a Python-based application which plays an interface role between them. The reason behind selecting the Python language to act as an interface role is that there are lots of Python-based command available in ABAQUS in order to extract or update its output data files. The python application is designed to manage the whole things, such as modeling initial problem by FE code, solving problem using GFEM code, and visualizing final solutions. Figure 3 shows sequences for code-coupling approach.

According to Fig. 3, the coupling procedure may be split into two sets of actions. The first pertains to the commercial code, the second to the GFEM code. In the commercial code, the following operations are performed:

(1) Create the geometry.

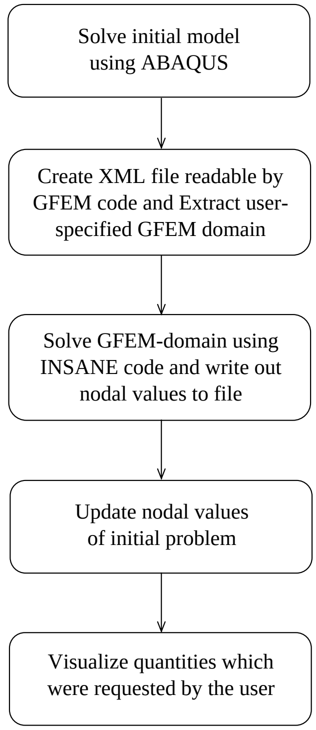


Figure 3: Code-coupling procedure sequence

(2) Specify the crack region.

(3) Mesh the geometry.

(4) Solve the initial problem and write out nodal values to a file readable by GFEM code.

Then, the following operations are carried out by the GFEM code, INSANE:

(1) Create GFEM XML file from model data extracted from FEM files.

(2) Apply Dirichlet/Neumann boundary condition around specified domain (crack domain) by user.

(3) Solve the crack problem.

(4) Update nodal values written by FEM code.

Finally, the Python interface code carries out the following tasks:

(1) Call FEM package to solve initial problem.

(2) Extract model data from FEM files and write to some readable files by GFEM code.

(3) Call INSANE code to create XML file usable by GFEM code.

(4) Call GFEM code to solve crack problem.

(5) Visualized user-defined quantities by calling FEM-visualized module, ABAQUS/Viewer.

Following section shows these whole interactions by solving a linear elastic fracture mechanic problem.

5 NUMERICAL EXAMPLE

This section presents a linear-elastic problems in $\Re^2$. This example considers a cracked plate submitted to an axial stress, as shown in Fig. 4. The cracked zone produces singular stress field near the crack tip. The objective of this problem is to illustrate the use of code-coupling approach for fracture mechanics problems.

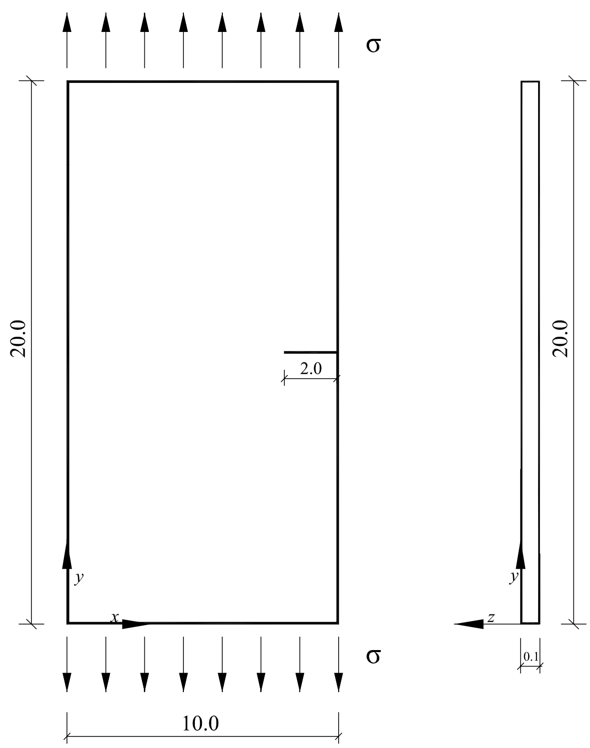


Figure 4: ***Geometry and loading of a cracked plate submitted to an axial stress***

This problem is analyzed, under plane stress state, has the following parameters (in consistent units):

- Modulus of elasticity $E = 1.0$;
- Poisson ratio $\nu = 0.3$;
- Stress $\sigma = 1.0$.

This problem is discretized into 50 quadrilateral elements using commercial software ABAQUS, as it can be seen in Fig. 5.

The solution of this problem is obtained using three different approaches, but using the same mesh discretization shown in Fig. 5:

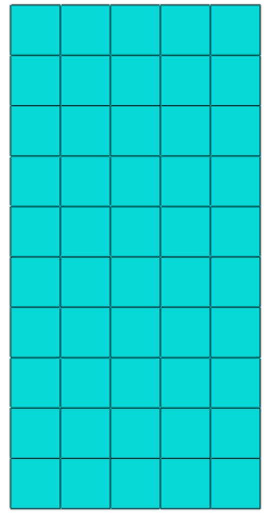


Figure 5: ***Discretization of cracked plate***

- **FEM analysis:**

  The FEM analysis is done using ABAQUS commercial package. The elements are four-node quadrilateral elements. The bottom side of the plate is fixed in y direction and stress field is applied on the top side.

- **GFEM analysis:**

  The GFEM analysis is carried out using INSANE in-house code. The geometry, material, loading and boundary conditions is considered the same as FEM analysis.

- **FEM/GFEM coupling analysis:**

  As mentioned before, the problem will be split into two sub-domains to do the FEM/GFEM coupling analysis. Figure 6 shows these two sub-domains. To solve the GFEM sub-domain, the displacement boundary condition will be applied on its boundaries. The information for displacement boundary condition is obtained using results from initial FEM analysis using ABAQUS software. Then, a combination of singular (Eqs. (5) and (6)) and polynomial (Eq. (4)) enrichment functions is used to enrich GFEM sub-domain nodes. The singular enrichment strategy is used to enrich the crack-tip node. Thus, as an advantage of singular enrichment function, we can capture high stress concentration around crack tip. The parameters in Eqs. (5) and (6) for the current crack configuration are considered as: $\lambda_1 = 0.544483737$, $Q_1 = 0.543075579$, and $A_1 = 1.0$ (Barros et al., 2013).

Results for σ_{xx} stress field for the cracked plate are presented in Fig. 7. This figure contains results for FEM, FEM/GFEM coupling, and GFEM analyses. The FEM analyses was done for two different mesh discretization: mesh with 50 elements (FEM with coarse mesh) and mesh with

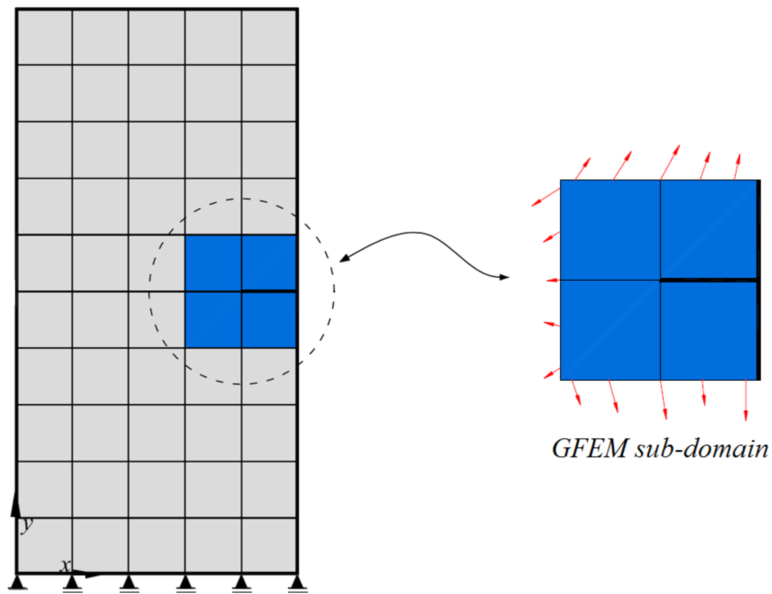
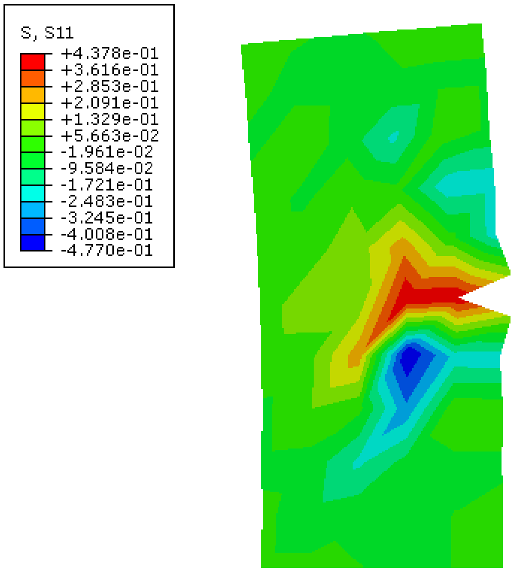


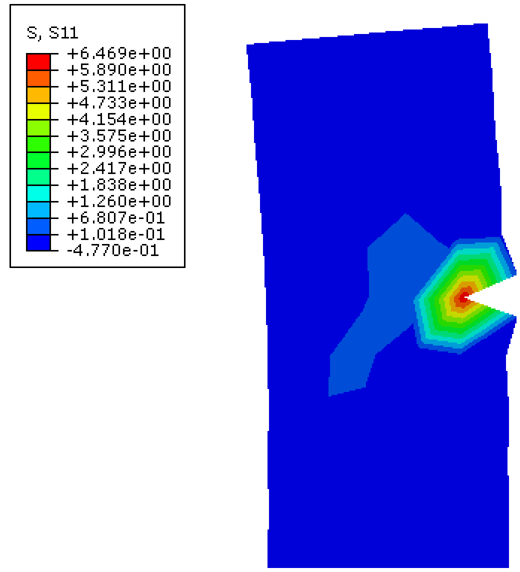
Figure 6: ***Split problem into two sub-domains: FEM sub-domain (Gray) and GFEM sub-domain (Blue)***

215168 regularly-distributed elements (FEM with fine mesh). It can be seen from this figure that the maximum σ_{xx} stress captured by FEM with coarse mesh, FEM with fine mesh, FEM/GFEM coupling, and GFEM analyses are 0.4378, 6.38, 6.469, and 6.857, respectively.

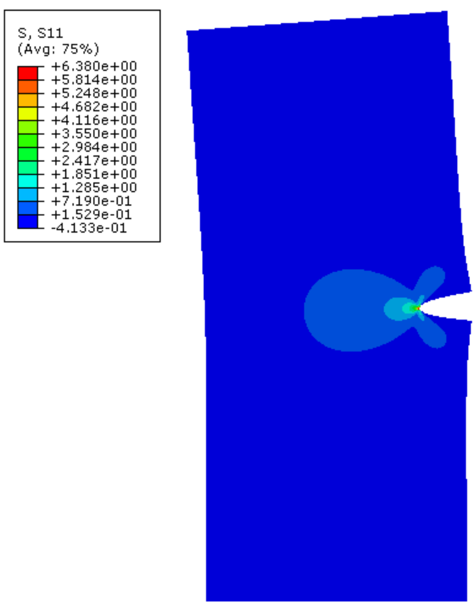
Also, Fig. 8 shows results for σ_{yy} stress field for the cracked plate. Again, the maximum σ_{yy} stress captured by FEM with coarse mesh, FEM with fine mesh, FEM/GFEM coupling, and GFEM analyses are 1.98, 9.181, 8.987, and 9.161, respectively. It is evident that when the singular enrichment is used, the maximum value at the crack tip is infinity. Thus, the values at Gauss points are used for the visualization.



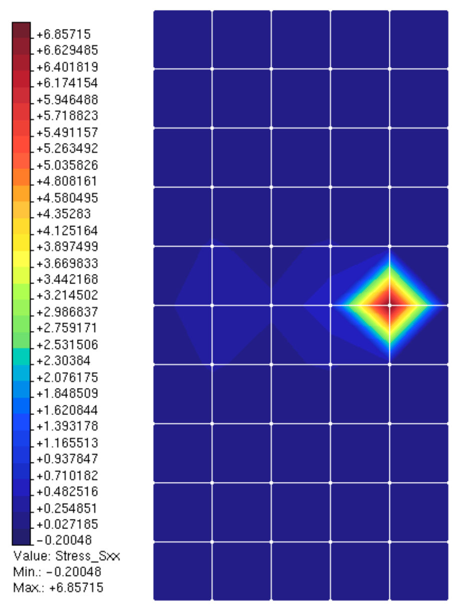
(a) FEM analysis using ABAQUS discretized with 50 elements



(b) FEM/GFEM coupling analysis

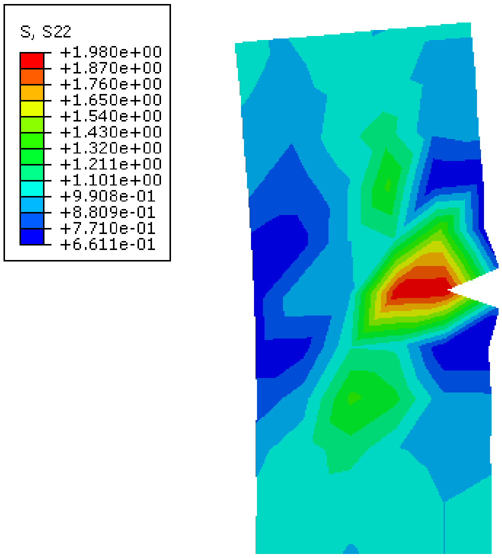


(c) FEM analysis using ABAQUS discretized with 215168 elements (regular distribution)

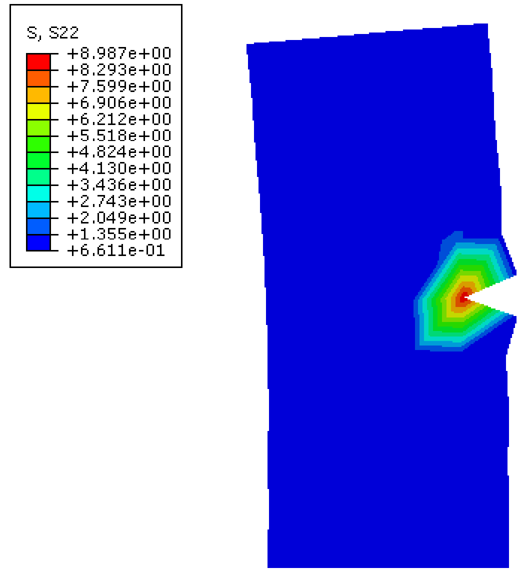


(d) GFEM analysis

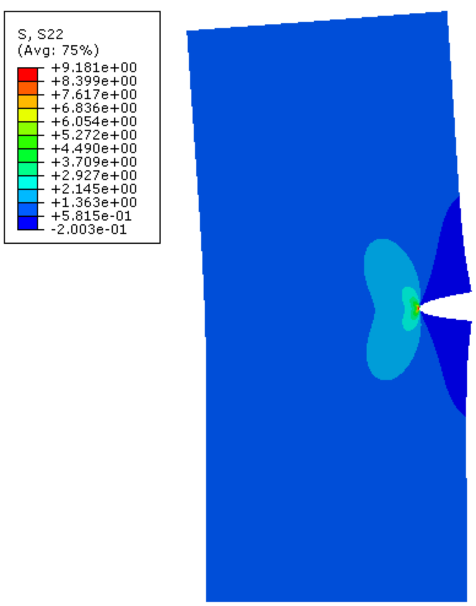
Figure 7: ***Stress component contour for σ_{xx} for FEM analysis, FEM/GFEM coupling analysis, and GFEM-only analysis***



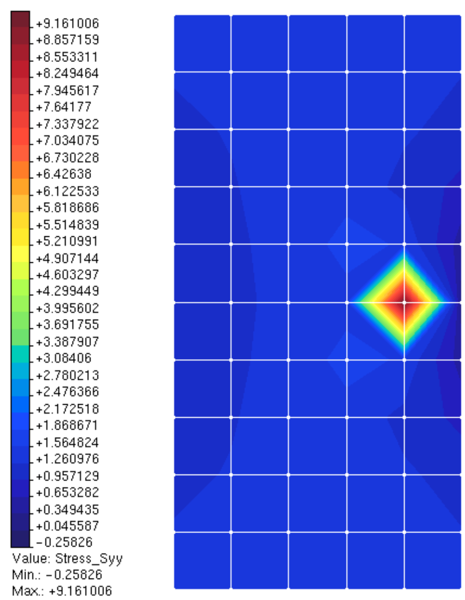
(a) FEM analysis using ABAQUS discretized with 50 elements



(b) FEM/GFEM coupling analysis



(c) FEM analysis using ABAQUS discretized with 215168 elements (regular distribution)



(d) GFEM analysis

Figure 8: ***Stress component contour for σ_{yy} for FEM analysis, FEM/GFEM coupling analysis, and GFEM-only analysis***

It can be observed from the results shown in Fig. 7 and Fig. 8 that the FEM/GFEM coupling strategy can deliver a good result for the singular stress field close to the crack tip.

Another parameter that can be evaluated here is the Stress Intensity Factor (SIF). The reference solution of SIF for current problem is obtained from following formulation (Pilkey, 2005):

$$K_I = \sigma \cdot \sqrt{\pi a} \cdot F(a/b)$$
$$F(a/b) = \sqrt{\frac{2b}{\pi a} tan(\frac{\pi a}{2b})} \times \frac{0.752 + 2.02(a/b) + 0.37[1 - sin(\frac{\pi a}{2b})]}{cos(\frac{\pi a}{2b})} \tag{7}$$

where K_I is SIF for mode I of fracture (opening mode), σ is the applied stress, a is the crack length, and b is the width of plate. These parameters for the current problem are as follows: $\sigma = 1.0$, $a = 2.0$, and $b = 10.0$. Therefore, the SIF for mode I is obtained equal to $K_I = 3.42571$. Table 1 shows normalized results of stress intensity factor for mode I. The FEM results obtained by ABAQUS itself. The GFEM and FEM/GFEM results were computed using the displacement extrapolation method (Aliabadi, 2002) that was adapted here for the GFEM approach and was presented before in (Reis et al., 2014).

Table 1: Normalized results for mode I SIF, K_I

Analysis Type	K_I
FEM with coarse mesh	1.156
FEM with fine mesh	0.994
GFEM	0.634
FEM/GFEM	0.66

It can be seen from these results that FEM analysis with fine mesh provides a good result for SIF. Also, by comparing GFEM and FEMGFEM analysis, it can be concluded that the FEM/GFEM analysis is capable to provide better result for SIF.

6 CONCLUSIONS

An overview of the FEM/GFEM coupling strategy is given with emphasis on fracture mechanics problems. The focus of this work is to present a coupling approach between a FEM commercial package, ABAQUS, a GFEM in-house code, INSANE. The proposed approach allows to solve any type of problems with discontinuities. The main advantage of using this strategy is that customized functions and different refinements levels can be used to obtain a more precise local solution by solving the local problem in INSANE (with the GFEM approach). Also, besides obtaining a more precise solution, the need of mesh refinement around the crack tip can be avoid, as the crack propagates through the domain. This strategy is illustrated by numerical example presented for Solid Mechanics applications, including Linear Elastic Fracture Mechanics problems. The numerical example showed that the proposed approach can capture better the high stress concentration around the crack tip than FEM-only approach.

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), FAPEMIG (in Portuguese "Fundação de Amparo à Pesquisa do Estado de Minas Gerais" - Grant PPM-00669-15) and CAPES (in Portuguese "Coordenação de Aperfeiçoamento de Pessoal de Nível Superior").

References

Aliabadi M.H. *The Boundary Element Method, Volume 2, Applications in Solids and Structures*. John Wiley and Sons, 2002.

Alves P.D., Barros F.B., and Pitangueira R.L.S. An object oriented approach to the generalized finite element method. *Advances in Engineering Software*, (59):1–18, 2013.

Babuska I., Caloz G., and Osborn J.E. Special finite element method for a classe of second order elliptic problems whith rough coefficients. *SIAM Journal on Numerical Analysis*, 31(4):745–981, 1994.

Babuska I. and Melenk J.M. The partition of unity method. *International Journal for Numerical Methods in Engineering*, 40:727–758, 1997.

Barros F.B., de Barcellos C.S., and Duarte C.A. p-Adaptive Ck generalized finite element method for arbitrary polygonal clouds. *Computational Mechanics*, 41:175–187, 2007.

Barros F.B., de Barcellos C.S., Duarte C.A., and Torres D.F. Subdomain-based error techniques for generalized finite element approximations of problems with singular stress fields. *Comput. Mech.*, 52:1395–1415, 2013.

Barros F.B., Proença S.P.B., and de Barcellos C.S. On error estimator and p-adaptivity in the generalized finite element method. *International Journal for Numerical Methods in Engineering (in press)*, 2004.

Belytschko T. and Black T. Elastic crack growth in finite elements with minimal remeshing. *International Journal for Numerical Methods in Engineering*, 45:601–620, 1999.

Bordas S. and Moran B. Enriched finite elements and level sets for damage tolerance assessment of complex structures. *Engineering Fracture Mechanics*, 73:1176–1201, 2006.

Duarte C.A. *The hp-cloud method*. PhD thesis, The University of Texas at Austin, 1996.

Duarte C.A., Babuska I., and Oden J.T. Generalized finite element methods for three-dimensional structural mechanics problems. *Computers & Structures*, 77(2):215–232, 2000.

Duarte C.A. and Kim D.J. Analysis and applications of a generalized finite element method with global-local enrichment functions. *Comput. Methods Appl. Mech. Eng.*, 197:487–504, 2008.

Duarte C.A. and Oden J.T. Hp clouds - a meshless method to solve boundary-value problem. Technical Report, TICAM, The University of Texas at Austin, 1995. Technical Report.

Fonseca F.T. and Pitangueira R.L.S. An object oriented class organization for dynamic geometrically non-linear. 2007.

Fries T.P. and Belytschko T. The extended/generalized finite element method: An overview of the method and its applications. *International Journal for numerical methods in engineering*, 2010.

Gupta V., Kim D.J., and Duarte C.A. Analysis and improvements of global-local enrichments for the generalized finite element method. *Computer Methods in Applied Mechanics and Engineering*, 245-246:47–62, 2012.

Jiang Y., Tay T.E., Chen L., and Zhang Y.W. Extended finite element method coupled with face-based strain smoothing technique for three-dimensional fracture problems. *International Journal for Numerical Methods in Engineering*, 102:1894–1916, 2015.

Kim D.J., Duarte C.A., , and Proença S.P.B. A generalized finite element method with global-local enrichment functions for confined plasticity problems. *Computational Mechanics*, 50:1–16, 2012.

Kim D.J., Duarte C.A., and Proença S.P.B. Generalized finite element method with global-local enrichments for nonlinear fracture analysis. *International Symposium on Mechanics of Solids*, pages 317–330, 2009.

Kim D.J., Pereira J.P., and Duarte C.A. Analysis of three-dimensional fracture mechanics problems: A two-scale approach using coarse-generalized fem meshes. *International Journal for Numerical Methods in Engineering*, 81:335–365, 2010.

Liszka T.J., Duarte C.A.M., and Tworzydlo W.W. hp-meshless cloud method. *Computer Methods in Applied Mechanics and Engineering*, 139:263–288, 1996.

Melenk J.M. *On Generalized Finite Element Methods*. PhD thesis, University of Maryland, College Park, 1995.

Melenk J.M. and Babuska I. The partition of unity finite element method: Basic theory and applications. *Computer Methods in Applied Mechanics and Engineering*, 39:289–314, 1996.

Melenk J.M. and Babuška I. The partition of unity finite element method. Technical Report, ????

Oden J.T., Duarte C.A., and Zienkiewicz O.C. A new cloud-based hp finite element method. *Computer Methods in Applied Mechanics and Engineering*, 153:117–126, 1998.

Pilkey W.D. *Formulas for stress, strain, and structural matrices*. John Wiley and Sons, 2 edition, 2005.

Reis A.P.S., Alves P.D., Barros F.B., Pitangueira R.L.S., and Andrade M.P. Numerical experiments with the generalized finite element method in two-dimensional elasticity problems. *CILAMCE 2014, Fortaleza, CE, Brazil, November 23-26*, 2014.

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

Strouboulis T., Copps K., and Babuska I. The generalized finite element method: an example of its implementation and illustration of its performance. *International Journal for Numerical Methods in Engineering*, 47:1401–1417, 2000b.

Strouboulis T., Copps K., and Babuska I. The generalized finite element method. *Computer Methods in Applied Mechanics and Engineering*, 190:4081–4193, 2001.

Wyart E., Coulon D., Duflot M., Pardoen T., Remacle J.F., and Lani F. A substructured FE-shell/XFE-3d method for crack analysis in thin-walled structures. *International Journal for Numerical Methods in Engineering*, 72:757–772, 2007.

Wyart E., Duflot M., Coulon D., Martiny P., Pardoen T., Remacle J.F., and Lani F. Substructuring FE-XFE approaches applied to three-dimensional crack propagation. *Journal of Computational and Applied Mathematics*, 215:626–638, 2008.