XLVI CILAMCE
VITÓRIA - ES - BRAZIL - 2025
XLVI IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING

# Automation of the pre-and postprocessing of analyses performed by the nonintrusive coupling between Abaqus and an in-house code

Rafaella Terezinha Cardoso Paula[1], Flávia Mello de Oliveira Campos[1], Túlio Roberto Eládio Marques[1], Ana Clara Pedras Bueno[1] , Felício Bruzzi Barros[1]

[1]*Dept. of Structural Engineering, Federal University of Minas Gerais*
*Av. Antônio Carlos, 6627– Pampulha, 31270-901, Minas Gerais, Brazil*
*rafaellaterezinha@ufmg.br, flavia-mello12@ufmg.br, tulioeladio10@ufmg.br, anaclarabueno@ufmg.br, felicio@ufmg.br*

**Abstract.** This work presents the development of two Python scripts that automate the preprocessing and postprocessing steps of a non-intrusive coupling strategy between the commercial software Abaqus and the INSANE (INteractive Structural ANalysis Environment) computational environment. This coupling is performed using the Generalized Finite Element Method with Global-Local enrichment (GFEM$^{gl}$), which divides the analysis into global, local, and mesoscale domains. The preprocessing script extracts the necessary data from the Abaqus model and generates the INSANE input file in XML format, facilitating model preparation and reducing manual errors. The postprocessing script incorporates the results obtained in Abaqus into the final file generated by INSANE, ensuring a unified and comprehensive representation of the multiscale structural behavior. These automations significantly reduce manual effort, optimize workflow, and enhance the practical applicability of the coupling strategy.

**Keywords:** Generalized Finite Element Method; Non-intrusive coupling; Multiscale modeling.

## 1 Introduction

Currently, to obtain a consistent structural analysis, many industry software packages model complex structures using numerical methods, such as Abaqus, which employs the Finite Element Method (FEM). However, when localized physical phenomena such as cracks, holes, or other discontinuities are present, it becomes necessary to use an even more refined mesh to achieve accurate results. This, however, leads to a high computational cost, making such analysis financially unfeasible for commercial software. On the other hand, in the academic environment, there are already solutions that, although computationally expensive, produce satisfactory results for localized physical phenomena, such as INSANE (INteractive Structural ANalysis Environment), developed at the Department of Structural Engineering of the Federal University of Minas Gerais (UFMG) [1].

In view of this, Bueno [2] enhanced the work of Filho [3] who, based on Li et al. [4,5], proposed a non-intrusive implementation of GFEM$^{gl}$ (Generalized Finite Element Method with Global-Local enrichment) [6], coupling the INSANE environment, developed in Java , with the Abaqus software, which uses Python scripts for interaction. This method aims to enable reliable analysis of structures with localized phenomena without incurring excessively high computational costs, and consists of performing multiscale modeling by solving three different scales: global, local, and mesoscale.

The global scale, which represents the complete structure but without the local phenomena, is solved in Abaqus using FEM. The local scale is modeled in INSANE and includes only the region where the phenomena occur. The mesoscale enables communication between the global and local scales. The solution process of this method consists of solving the global model using FEM and, through the Iterative Global-Local (IGL) strategy [7], coupling the global model with the mesoscale. In this step, the displacements obtained from the global scale are transferred as boundary conditions to the mesoscale, and the solution of the mesoscale is coupled with the

global problem by transfer the difference of the forces along the interface between these two scales. The mesoscale and the local model interact through the GFEMgl strategy, generating global-local enrichment functions that are applied to the global model.

This work aims to contribute to the automation of the non-intrusive coupling process between the Abaqus and INSANE software, through the implementation of two complementary codes: one responsible for generating the input file for INSANE, and the other dedicated to the post-processing of the obtained solutions.

# 2 Solution Procedure for the Non-Intrusive Coupling

The coupling of the final solution obtained from Abaqus and INSANE occurs iteratively, with both transferring generalized forces and displacements. This coupling can be performed according to two user-defined solution strategies: monolithic or staggered.

The methodology of the monolithic approach follows these steps: initially, IGL (Iterative Global-Local strategy) iterations are performed, in which the global model is analyzed; then, the prescribed displacements are applied to the mesoscale, which is subsequently solved. The local model analysis is performed using the previously obtained global solution as boundary conditions. From this local model, the enriched mesoscale is solved, followed by global-local MEFG-gl cycles, which are repeated until the final solution converges. From the solutions obtained at the global and enriched meso scales, the residual vector is calculated and convergence is checked. If convergence is not achieved, the process is repeated.

In the staggered approach, the convergence of the IGL itertations between the mesoscale and global scale models is performed first. Then, the residual vector is calculated, and convergence is verified. Only after this convergence is the local scale model solved, and the consequent global-local enrichment of the mesoscale model is performed.

# 3 Methodology Used for the Creation of Pre- and Postprocessing Scripts

This section presents the methodology adopted for the development of the automation scripts. The proposed approach consists of a sequence of steps that enable non-intrusive coupling between the global and local domains, integrating Abaqus and INSANE. To facilitate understanding, Fig. 1 presents a flowchart summarizing the main stages of the process, from the initial modeling to the visualization of the coupled results.

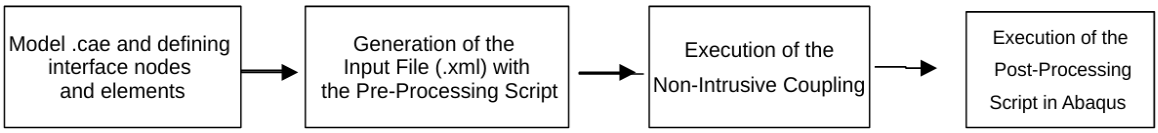



Figure 1. Workflow for the non-intrusive coupling process between Abaqus and INSANE

## 3.1 Script related to the automation of the Pre-Processing stage

The automation of the input file generation for INSANE from Abaqus was carried out through a script developed in Python. This script is executed within the Abaqus interface after the definition of the global model, where the user configures the problem's boundary conditions, such as geometry, materials, and loads, without considering localized phenomena. The identification of the regions comprising each mesoscale is performed directly in the Assembly environment using the Set tool, which allows the grouped selection of nodes and elements. This information is then used to automatically generate the corresponding .xml file for the mesoscales, which will be used as input for INSANE.

The code development was based on the Abaqus Scripting Reference Guide, which provides a comprehensive description of the Abaqus Scripting Interface. Through it, it was possible to access information such as nodal coordinates, element connectivity, material properties, and concentrated load applications.

Furthermore, the script presents interactive tables to the user, pre-filled with default values that can be modified during execution. The first table requests information such as the .cae file name, step name, instance, model name, and part name of interest, as well as additional parameters to be recorded in a .txt file, including the job name, IGL nodes Set Name, and the number of processors (CPUs) to be used. Subsequently, the second table is displayed, allowing the user to input parameters related to the coupling, such as the number of interfaces created,

the value of Kappa, and the global and local Solver Types, as illustrated in Fig. 2. Based on the specified number of interfaces, the code automatically generates fields for the user to provide the name of the sets corresponding to the nodes and elements of each defined mesoscale. Finally, the third table requests information about the local model, such as the crack coordinates and the type of enrichment to be applied in each case.

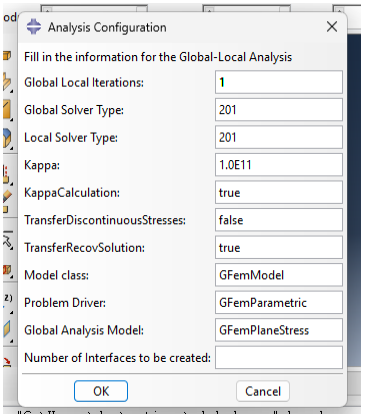



Figure 2. Global-Local Analysis Configuration Table

After filling in the tables, the script automatically generates a .txt file containing information that enables coupling between INSANE and Abaqus, as well as the .xml files referring to the global model and the specified mesoscales. The interface nodes of the mesoscale model have their degrees of freedom constrained in the input file, since during the solution stage these nodes will receive prescribed displacements coming from the global model solution. With the generated files, the .cae model, and the Python scripts managing the coupling created by Filho [3], it is possible to perform the coupling between Abaqus and INSANE.

### 3.2 Script related to the automation of the Postprocessing stage

The post-processing file consists of a .xml file that will be opened in INSANE. It is formed by the combination of the solutions from Abaqus and INSANE. After the coupling, the results from the model performed in the Abaqus .odb file and the mesoscales created with the latest applied displacements are generated. The post-processing file comprises the local domain (mesoscale) and the complementary domain (part of the global domain that doesn't coincide com the local domain).

For the generation of this file, the Abaqus Scripting Guide Reference was used in order to extract information from the .odb file, such as the names of the sets that form the mesoscales, the elements and their connectivity, labels and their coordinates, stresses, strains, and displacements.

Based on this information, and using the mesoscale result files generated by INSANE, the final solution file is structured. This file is written in the format required by INSANE, containing the information of nodes (nodelist) and elements (elementList) of the structure in addition to the results in terms of displacement, stress and strain fields. The lists is initially filled with information regarding the mesoscale models, and at the end of each list, information referring to the complementary domain is added, resulting in a single consolidated file.

Finally, the results can be visualized in INSANE through the "View Results" option, where it is possible to explore displacement, strain, and stress fields in both the global and local domains in a unified manner. This visualization allows the evaluation of the multiscale behavior of the coupled model, identification of regions with higher stress concentration, and validation of the information transfer between scales.

## 4 Results

In order to demonstrate the practical application of the proposed methodology, a two-dimensional frame was modeled in Abaqus, as illustrated in Fig. 3. The structure is subjected to a concentrated load of $2{,}0\times10^{3}$ applied along the element's axis of symmetry and does not exhibit any localized physical phenomena. The material was

considered elastic, with Young's modulus $E=2.0\times10^7$ and Poisson's ratio $\nu = 0.3$, adopting consistent units.

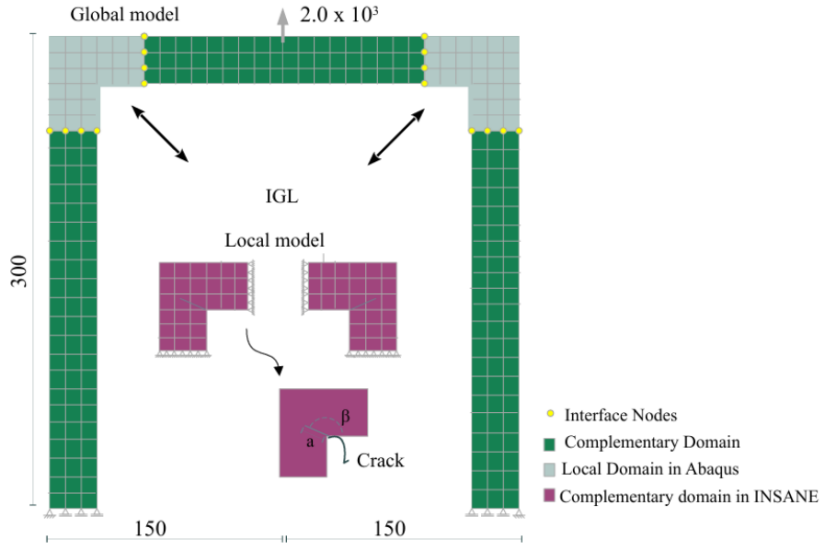



Figure 3. Two-dimensional frame model created in Abaqus with domains that compose the post-processing file

During the modeling process, sets were created to define the interface nodes of each mesoscale region, as well as the elements comprising them. Based on these sets, input files were generated using a preprocessing code, in which the user provides the crack parameters associated with each mesoscale. The cracks in the model are characterized by a length a =7.9057 and an orientation β=2.8198.

Subsequently, the non-intrusive coupling was executed, and the output files were generated. These include the .cae and .odb files from the global Abaqus model, containing force and displacement fields, as well as .xml files corresponding to each mesoscale. The final result file was produced through a post-processing script. To execute it, a script was run within the Abaqus interface, prompting the user to specify information such as the number of mesoscales, the set name of each mesoscale, and the respective solution files.

The results of the coupled analysis of the global and local domains are visualized in INSANE. Fig. 4 illustrates the displacement component in y-direction in which the continuity of the displacement fields in the mesoscale regions can be observed, evidencing the effectiveness of the proposed multiscale approach.

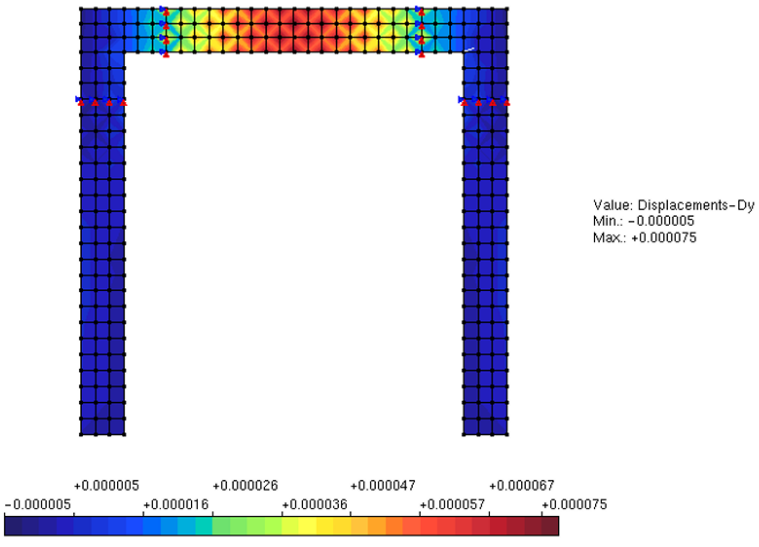



Figure 4. Distribution of the displacements yy obtained from the coupled analysis, visualized in the INSANE

# 5 Conclusion

The results obtained so far demonstrate the feasibility and effectiveness of automating the non-intrusive coupling between Abaqus and INSANE. The replacement of manual steps by Python scripts significantly reduced the likelihood of operational errors and the execution time of the multiscale process.

Data transfer between the global and local domains was observed to occur consistently, ensuring continuity in the displacement fields and compatibility in the exchange of forces at the interfaces. The integrated visualization in INSANE confirms the correct coupling between scales, highlighting the methodology's capability to accurately represent localized effects with high resolution.

**Acknowledgements.** The authors gratefully acknowledge the important support of the Brazilian research agencies FAPEMIG (in Portuguese “Fundação de Amparo à Pesquisa de Minas Gerais”), CNPq (in Portuguese “Conselho Nacional de Desenvolvimento Científico e Tecnológico” - Grant 308444/2022-1 and Undergraduate Research Scholarship).

**Authorship statement.** The authors hereby confirm that they are the sole 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] F.T. Fonseca, R.L.S. Pitangueira, An object oriented class organization for dynamic geometrically nonlinear fem analysis, in: Iberian Latin American Congress on Computational Methods in Engineering, CILAMCE, 2007.
[2] BUENO, A. C. P.; BARROS, F. B. Expansão da Implementação não intrusiva do Método dos Elementos Finitos Generalizados Global-Local. Dissertação (Mestrado) — Universidade Federal de Minas Gerais, Belo Horizonte, MG, Brasil, Junho, 2024.
[3] FILHO, N. A. da S. Implementação não intrusiva do método dos elementos finitos generalizados global-local. Dissertação (Mestrado) — Universidade Federal de Minas Gerais, Belo Horizonte, MG, Brasil, Março 2023.
[4] Li, H., O’Hara, P. e Duarte, C. (2021), Non-intrusive coupling of a 3-d generalized finite element method and Abaqus for the multiscale analysis of localized defects and structural features, Finite Elements in Analysis and Design vol. 193, 103554. URL: http://dx.doi.org/10.1016/j.finel.2021.103554.
[5] H. Li, J. Avecillas-Leon, N. Shauer, and C. A. Duarte (2022). A non-intrusive iterative generalized finite element method for multiscale coupling of 3-d solid and shell models. Computer Methods in Applied Mechanics and Engineering, vol. 402, pp. 115408, 2022.
[6] C. A. Duarte and D. J. Kim. Analysis and applications of a generalized finite element method with global-local enrichment functions. Computer Methods in Applied Mechanics and Engineering, vol. 197, n. 6–8, pp. 487–504, 2008.
[7] J. D. Whitcomb. Iterative global local finite element analysis. Computers & Structures, vol. 40, n. 4, pp. 1027–1031, 1991.