Global-local plug-in for high-fidelity composite stress analysis in Femap/NX Nastran

Global-local plug-in for high-fidelity composite stress analysis in Femap/NX Nastran

E. Carrera*[1], A.G. de Miguel†[1], M. Filippi‡[1], I. Kaleel§[1], A. Pagani¶[1], M. Petrolo‖[1], and E. Zappino**[1]

[1]MUL[2] Group, Department of Mechanical and Aerospace Engineering, Politecnico di Torino, Turin, Italy

Abstract

The prediction of the actual stress fields in real structural applications is still not completely resolved, especially when dealing with composite structures. The geometrical complexity of laminate parts and the multiple length scales which are involved in the problem lead to a severe trade-off between accuracy and computational costs. Consequently, to keep the size of the numerical problem below a certain limit, stress engineers tend to use classical laminate elements in their FEM simulations, which cannot provide the interlaminar stress solutions through the stack of plies with enough accuracy. To extend the capabilities of Nastran, this work proposes an global-local method to extract the 3D strain and stress fields from the 2D elements. The code, named MUL2@GL, has been developed as a user-friendly plug-in for Femap and requires a minimum training to be operated. By just selecting the critical elements in Femap, the code automatically generates a dedicated model for an advanced composite FEM code which is embedded in the plug-in. This solver provides the distribution of the 3D solutions across the thickness of the element. The proposed code is highly convenient for the evaluation of the failure onset in critical areas of the structure

*Professor of Aerospace Structures and Aeroelasticity, erasmo.carrera@polito.it
†Research Fellow, alberto.garcia@polito.it
‡Assistant Professor, matteo.filippi@polito.it
§Research Fellow, ibrahim.kaleel@polito.it
¶Assistant Professor, alfonso.pagani@polito.it
‖Associate Professor, marco.petrolo@polito.it
**Assistant Professor, enrico.zappino@polito.it

such as cut-outs, corners or free edges, where the hypotheses of laminate plate elements no longer apply. As an example, we show the capabilities of the method in a benchmark composite wing and the free-edge stresses in a cut-out.

Keywords: Global-local analysis, Stress analysis, Nastran, Laminates, Free-edge.

1 Introduction

Global-local approaches for stress analysis of composite structures are typically used by engineers to perform more detailed analysis of smaller components of the structure. In the finite element method (FEM), the reason for this approach is the impossibility of making a detailed numerical simulation of the macro-scale structure due to computational limits. This is especially evident in the case of composite laminates, in which the scale ratio between the overall dimension of the part and the ply thickness can be high as 10^4. For this reason, the term 'global-local' is typically applied to submodeling approaches, in which a certain component of the structure in the model is extracted for different purposes, for instance the inclusion of smaller details that could not be added in the global model, or just to refine the mesh density in a critical zone. In both cases, the boundary conditions applied to the local model are mapped from the outputs of the global model.

Classical global-local models as described above are a step towards a more reliable stress analysis of a given composite structure. However, they still do not resolve the issue of the computation of the 3D state of stress at the ply level, which is of paramount importance for the prediction of the onset of failure in laminates. It is known that if the interlaminar stresses and strains are to be accurately predicted, classical laminate elements are not sufficient due to their simple kinematic assumptions [9]. In order to predict well-known composite effects such as the zig-zag displacements or the stretching deformations [13], a distribution of solid elements must be placed across the thickness. Indeed, some authors suggest that at least three solid elements should be placed across the thickness of each ply [4] to get a proper distribution of the interlaminar components. Making some simple calculations, if we consider a typical lay-up of aeronautic applications with some tens of layers, and we respect the aspect ratio constraints for 3D FEM, the structural analysis of a simple squared plate of 1 $\times$ 1 m requires about 10^9 solid elements. The numerical problem becomes so large that such kind of refined models are usually restricted to research purposes [11], and, therefore, composite designers are behold to stick to simple rules based on best engineering practices instead of actual stress analyses.

One of the reasons for the scarcity of stress-based composite design approaches is the fact that the mechanical problem of computing the 3D stresses in laminates is multiscale in nature. This means that the structural analysis must be able to account for details which are two or three orders of magnitude different in scale, from the component size to the ply thickness. A variety of multiscale methods [1] are available to solve this problem and some can be found in many commercial software. However, all these methods make use of strong hypotheses, such as the periodicity in infinite media, to decouple the problem in macroscale and local analysis, and, therefore, important information about the actual boundary conditions is lost in the interface.

This paper proposes a link between these two approaches (global-local and multiscale methods) for the fast computation of the interlaminar stresses in FEM models based on laminate plate elements. The method remains essentially a one-way global local approach, but targets the level of detail of multiscale approaches for laminate analyses. For this purpose, the proposed tool makes use of advanced layer-wise (LW) models [2, 3] for the evaluation of the local areas, which are defined as single elements in the Nastran model. This solution, denoted to as *element-wise* (EW), allows stress engineers to perform multiscale analysis accounting for ply-level details with the actual boundary conditions obtained from the FEM model. In this manner, the only assumptions in the analysis are restricted to the imposition of plate-like displacement fields at the interface between the global and the local, thus enabling for the recovery of 3D state of stress in the central area of the element.

2 The element-wise (EW) approach

The aim of this development is to deliver a user-friendly tool to link the stress analysis of large composite parts with the most refined methods for the prediction of the interlaminar fields. Therefore, the tool must be available in well-known environments for structural engineers to use it, and it should require as few steps as possible in the analysis process. In addition, it must keep the potential to be embedded in more complex analysis runs, such as optimization loops. Our method to address these requirements is based on a simple global-local approach with one-way coupling, meaning that the information passes from the global FEM analysis to the selected local model. In other words, the boundary conditions applied in the local model are the displacement and rotation values obtained from the global analysis in Nastran.

The classical approach in the industry for the evaluation of the onset of failure and margins of safety of the structure is to compute a certain failure index from the strain and stress outputs of the FEM model. A well-known shortcoming of this approach is due to the inability of laminate elements to provide accurate solutions for the transverse components of these fields, which play a significant role in the structural response of composite laminates. Consequently, in most cases these failure indexes only reflect the likelihood of the structure to fail under in-plane modes, whereas the transverse shear and the peeling modes are usually neglected. Moreover, in global-local analyses the selection of the critical zone for the detailed study depends mostly on the experience of the stress engineer and oftentimes requires a time-consuming process for the generation of the local model and the handling of the interface conditions.

Our method to avoid these issues is to define the single laminate elements as independent local areas. In this manner, the local model can be created automatically from the element information located in the analysis input file, and no further actions are required from the user. Furthermore, the EW approach is consistent with the outputs delivered by 2D linear elements, which include a single value of the strain/stress solutions at the centroid. The left-hand image of Fig. 1 shows an example of typical failure evaluation performed on a composite wingbox using Femap/NX Nastran and the Hoffman stress criteria [6]. The FEM analysis provides an *element-wise* distribution of the

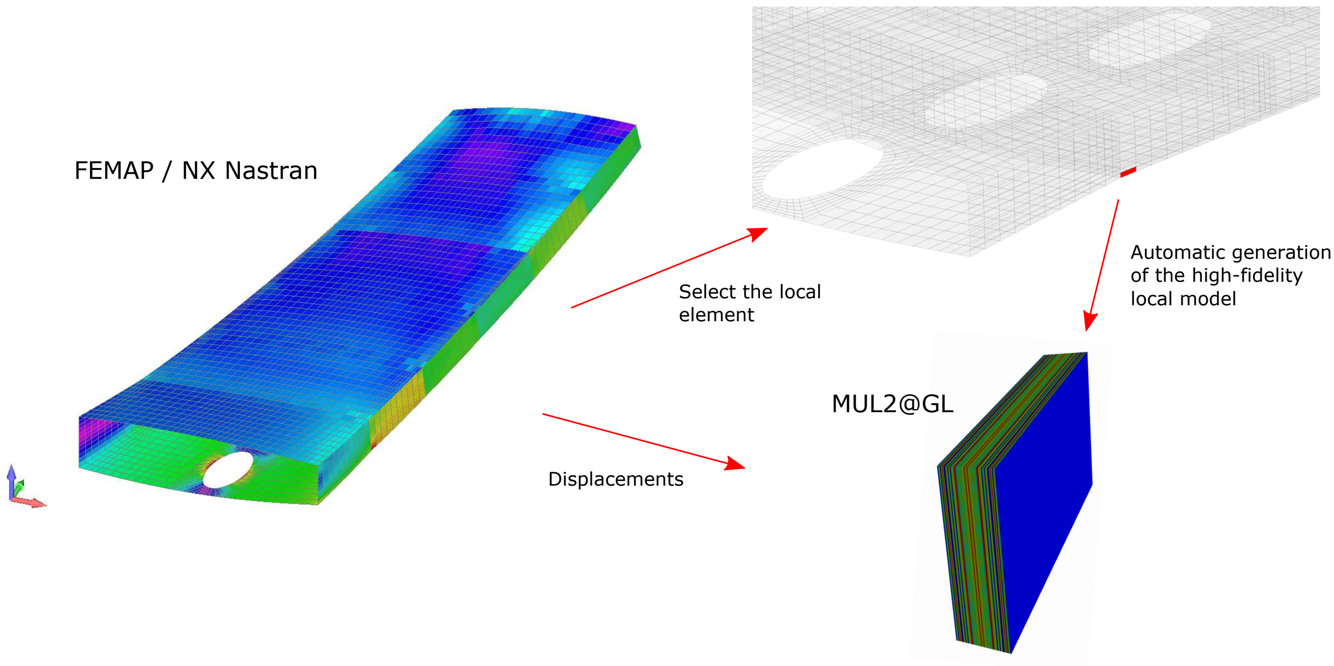


Figure 1: View of a composite wing section and failure index contour. On the right side a zoom showing the critical element highlighted in red, and the local model generated by the MUL2@GL tool.

ply stresses and strains over the surface of the plate element. In this context, the EW plug-in is introduced as a mean to extract the 3D solutions at the centroid of the element for generic cases, thus offering a more realistic prediction of the mechanical unknowns across the stack of plies in a straightforward manner.

3 The plug-in interface for Femap and NX Nastran

The majority of the interface code between Femap/Nastran and the MUL2 solver is written externally as a Python script. The installation of the plug-in can be done simply by adding the dedicated folder in the API directory of Femap. The current version of the plug-in supports the following inputs of NX Nastran:

- Element type: CQUAD4.
- Property: PCOMP and PSHELL.
- Material type: MAT1, MAT2, MAT8 and MAT11.

3.1 Reference systems

This section describes all the reference systems involved in the global-local analysis of generic laminated structures. The interface code reads the data from the bulk data file. For more information about the bulk data files in Femap/Nastran, the reader is referred to the user manuals [7, 8]. Once the code identifies the selected local element from the user inputs, the plug-in generates an

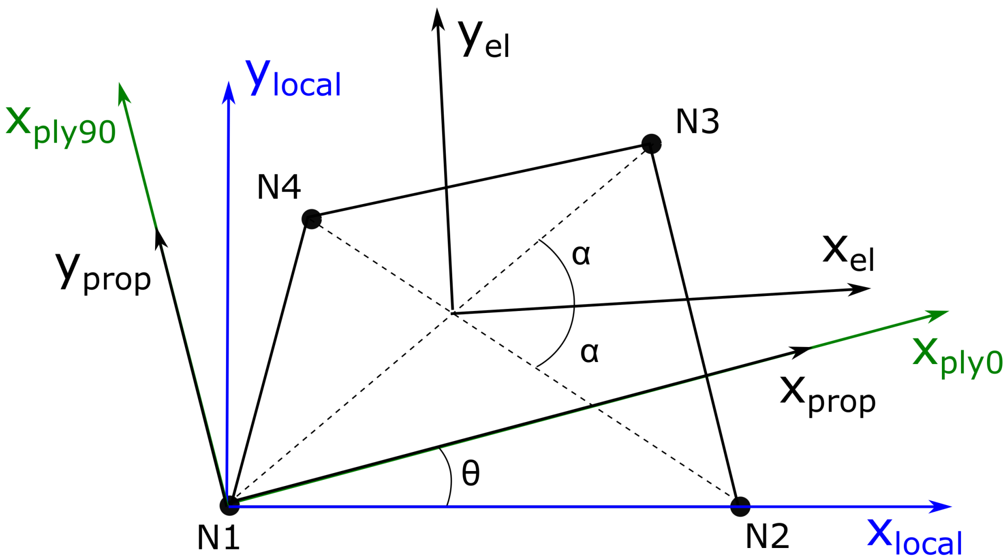


Figure 2: Reference system of the Nastran QUAD4 element and the local model.

optimized LW model for the 3D stress analysis of the laminated part. The lay-up and materials are obtained from the property number and the material coordinate system is taken directly from the bulk data file. Figure 2 shows the reference systems of the CQUAD4 element and those used by the plug-in to solve the system equations and plot the solutions.

The local system is defined as follows: the x_{local} versor lays on the first edge and points to N2, the y_{local} versor is normal to the latter and lays on the plane of the element. The z_{local} versor is normal to the element plane. On the other hand, the material orientations are computed from the reference system of the property and the angle of each ply defined in the PCOMP card, which also provides the thickness of each layer. The boundary conditions of the local model are elaborated from the displacement solutions obtained from the output file (op2). Once the local model is created and the displacement boundary conditions applied, the MUL2 solver automatically solves the linear problem and produces the 3D stresses, as shown in Figure 3. The new solutions are also stored in different formats for an easy access and manipulation.

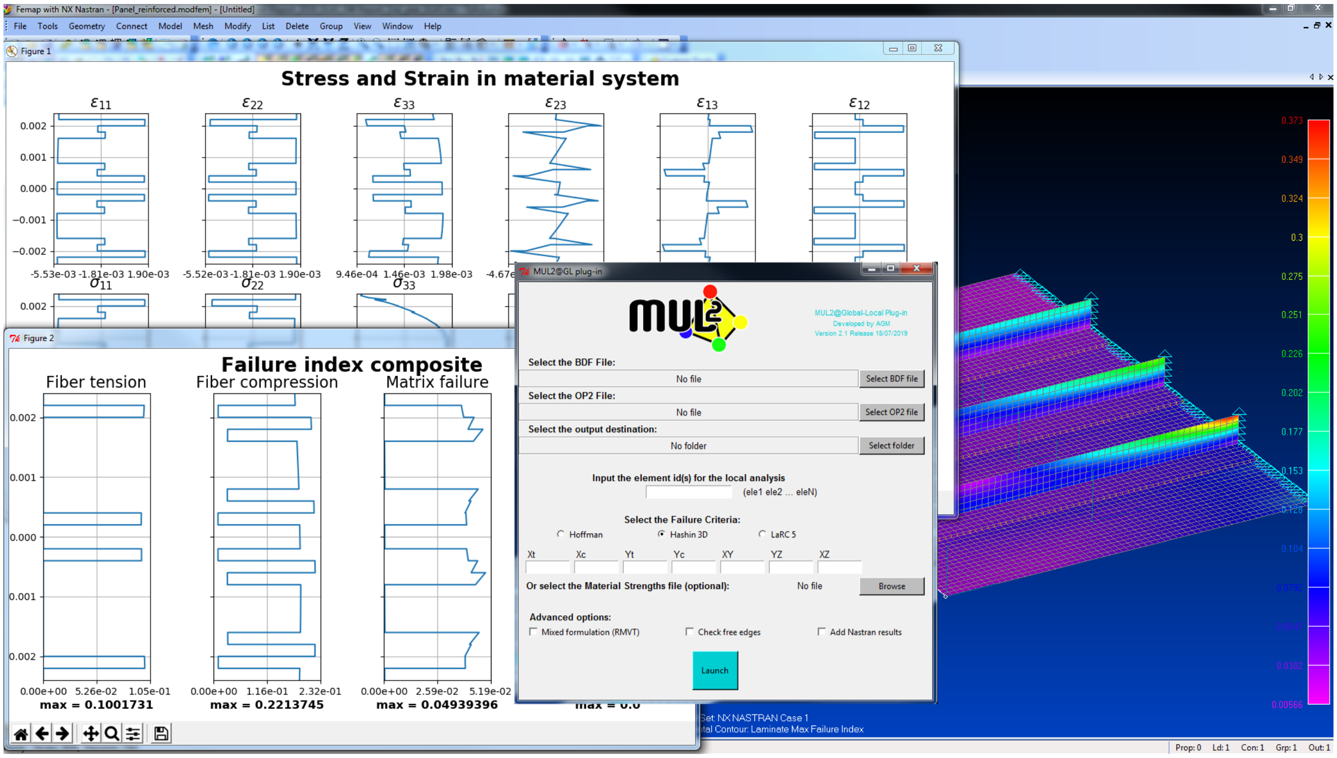


Figure 3: Visualization of 3D solutions in Femap.

4 Example

To demonstrate the capabilities of the proposed method we evaluate the stress fields in a benchmark composite wingbox. The FEM model is shown in Fig. 4 and it is made of 1D and 2D elements, including beams for stringers and reinforcements, and plate and laminate elements for ribs, spars and skins. Different lay-ups of unidirectional tape are used over the skin and spar surfaces, ranging from 28 at the spar webs to 62 plies at top and bottom skins nearby the wing root. The material properties selected are those of a IM7-8552 carbon fiber tape. The nodes located at the root section are clamped, and several point loads are applied in different directions at the center of the ribs.

Figure 1 shows the structure deformed under the given boundary conditions. An arbitrary element of the top skin near the clamped edge is picked for the global/local analysis (highlighted in red in Fig. 4). By selecting the element, the MUL2@GL automatically passes the information from the FEM model to the high-fidelity local analysis and generates the 3D solutions for strains, stresses and required failure indexes. Figures 5 and 6 show the in-plane and transverse shear values of the stress tensor at the centroid of the local element (blue lines). The solutions provided by the full model of Femap/Nastran are also included for comparison. The stress values are plotted in the ply coordinate system, i.e., 1 is parallel to the fibers at each ply, 2 is normal to the fibers and 3 is the thickness direction. One can notice that, as expected, the values of the in-plane stresses agree well due to the membrane/bending capabilities of the shell elements. On the contrary, when it

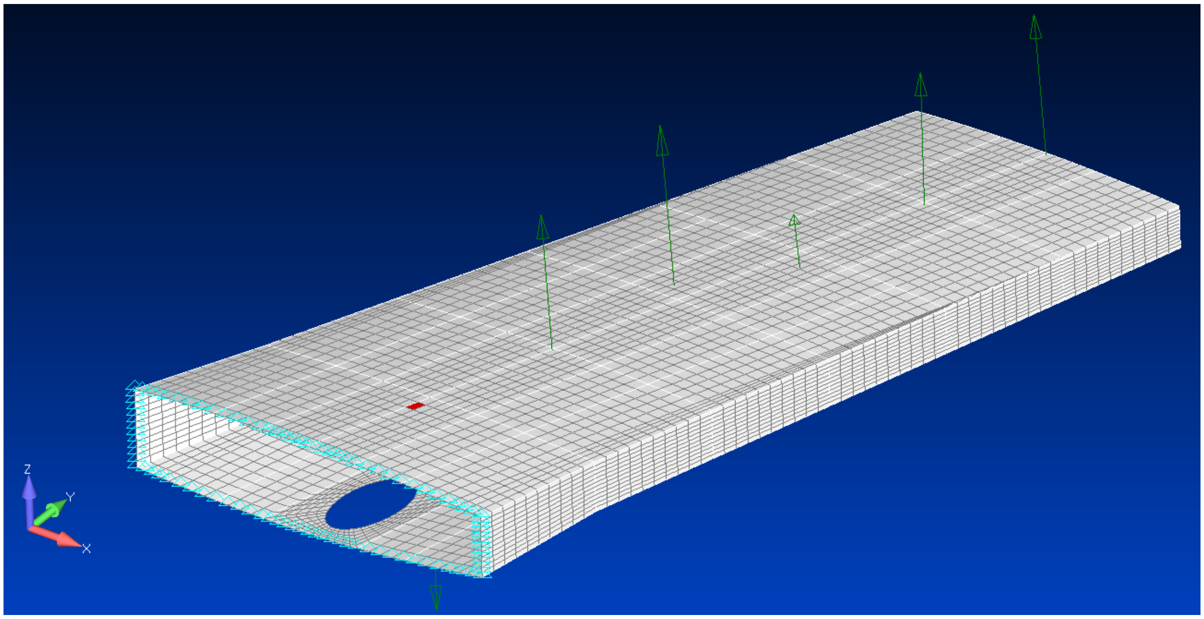
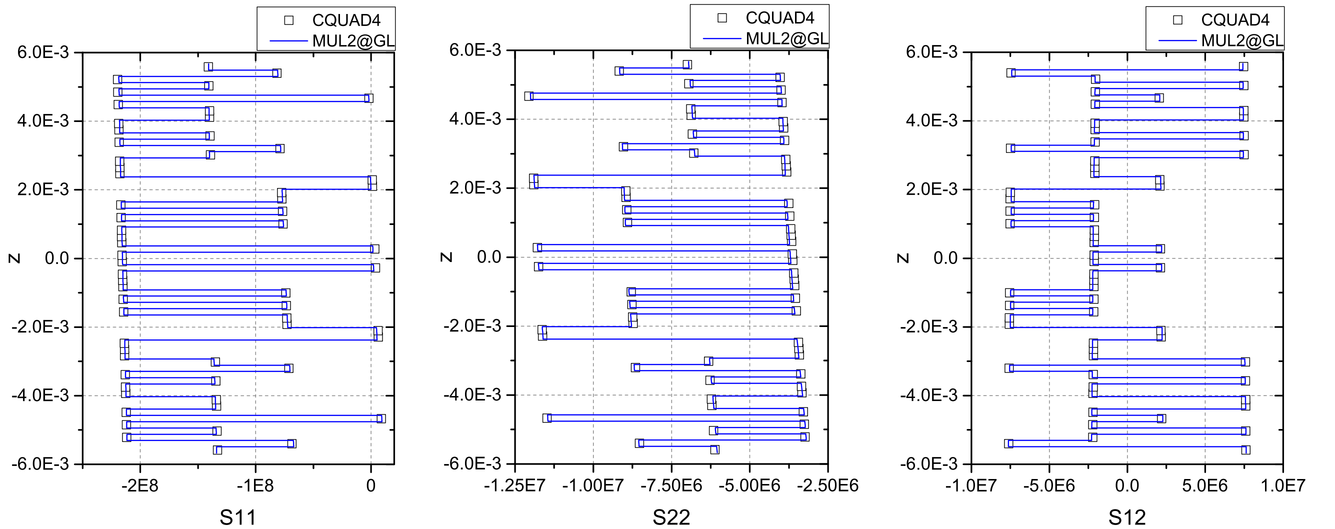


Figure 4: View of the FEM model of the wing structure in FEMAP.

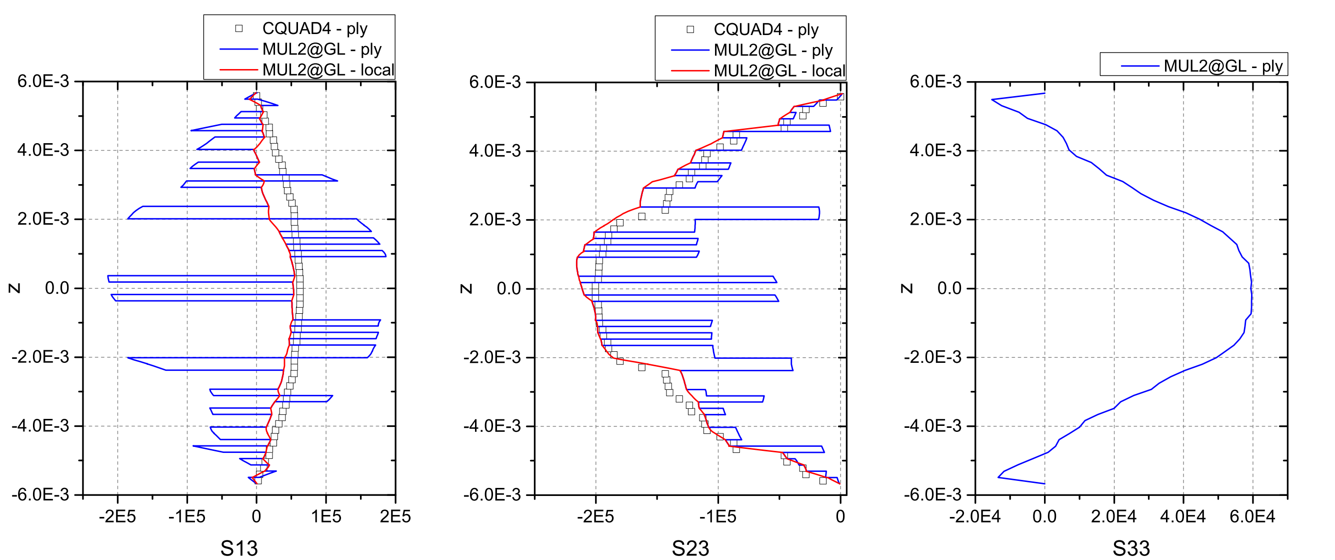
comes to the out-of-plane stress prediction the differences between the laminate plate outputs and the local model are substantial. Indeed, the code NX Nastran for laminate elements performs a stress recovery technique to provide an approximation of the transverse shear components which is based on the assumption of no coupling between the stresses in the x and y directions. Moreover, the MUL2@GL also computes the peeling stresses, as shown in Fig. 6 (c).

The assumptions adopted by laminate plates fairly hold for thin structures and simple laminations. However, the lay-ups typically employed in the aeronautic field are considerably thick in the areas of maximum loading, therefore the plane stress hypotheses no longer apply. In addition, the plate elements cannot predict the complex states of stress that develop around holes, cut-outs and over the edges of the laminated structure [8]. These shortcomings raise many questions about the validity and accuracy of composite structural analysis as commonly carried out in the industry, since these regions may constitute the most critical parts of the structure. Figure 7 shows the bottom view of the wing model considered here and the contour plot of the maximum laminate failure index based on the Hoffman criteria. One can see that the most critical elements are placed at the edges of the first and second hole, as expected. We select the most critical element to show the capabilities of the MUL2@GL code (see the element marked in yellow in Fig. 7). In free-edge zones, the plug-in can provide a 3D approximation of the interlaminar stress peaks that develop due to the mismatch of the mechanical properties between layers [5, 10]. Figure 8 shows the distribution of SZZ and SXZ across the stack of plies at the free edge in the element coordinate system (X is parallel to the wingspan). It also shows the same values computed at the centroid of the element by the local model (red line) and NX Nastran (square dots). One can observe how the interlaminar stresses increase by one or two orders of magnitude from the centroid to the free



(a) (b) (c)

Figure 5: In-plane stress fields through the thickness of the laminate at the selected element of the top skin. The stress values are plotted in the ply coordinate system, being 1 the fiber direction.



(a) (b) (c)

Figure 6: Out-of-plane shear stresses at the selected element of the top skin.

edge. Finally, Table 1 compares the values of the failure index computed using different criteria in the same element. It is demonstrated that the use of advanced composite failure indexes such as LaRC5 [12] is only justified if an accurate distribution of 3D stresses is available.

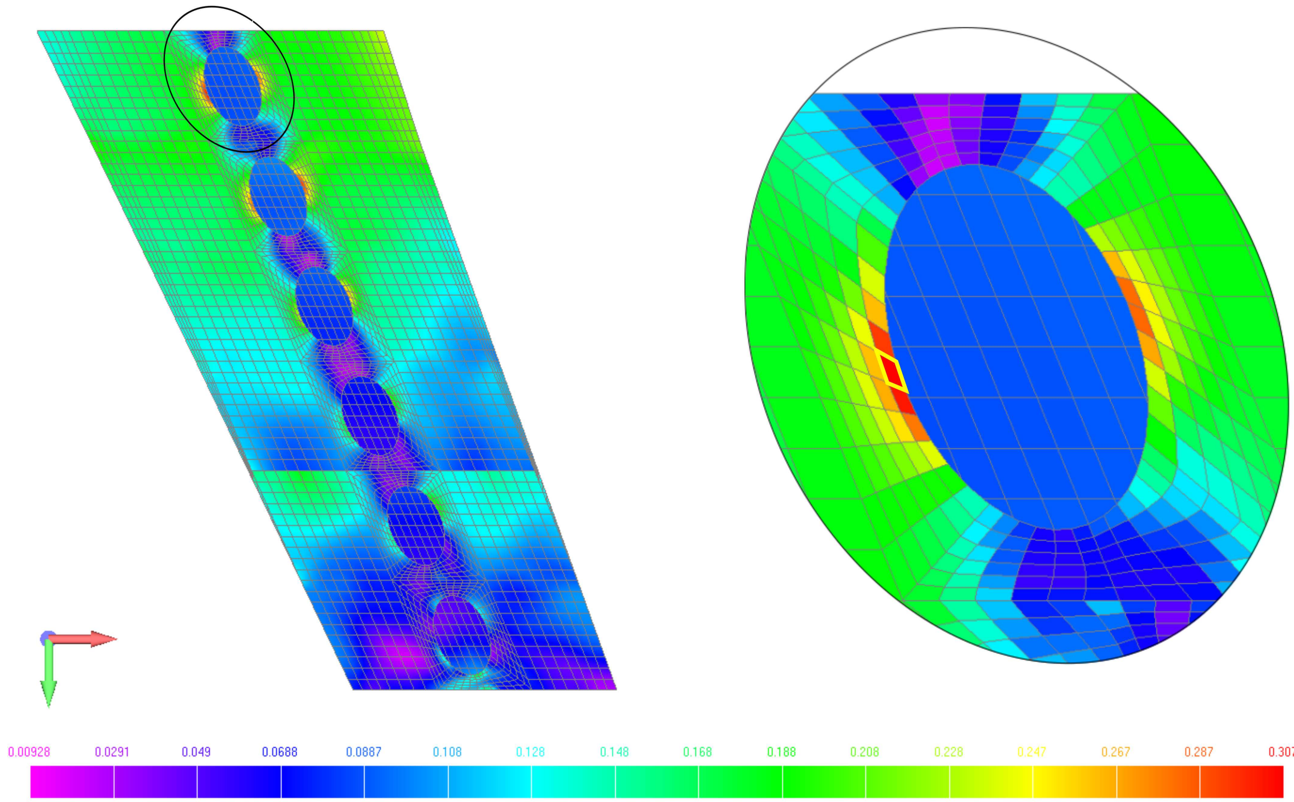
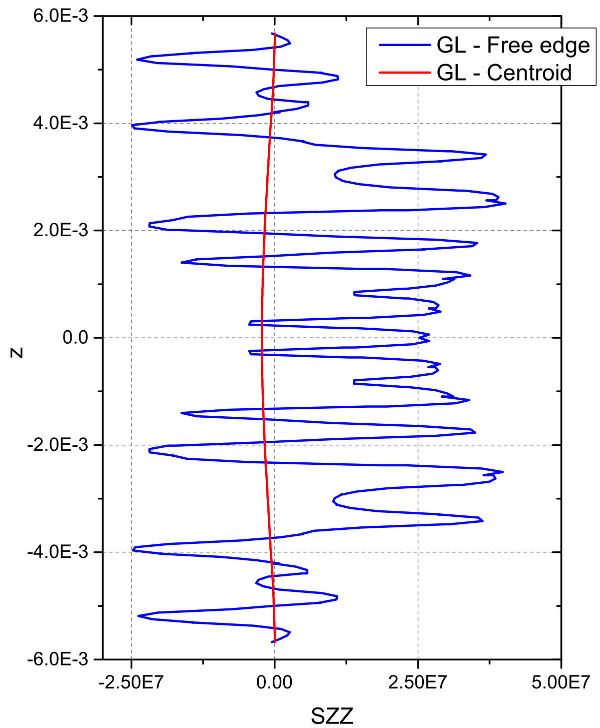


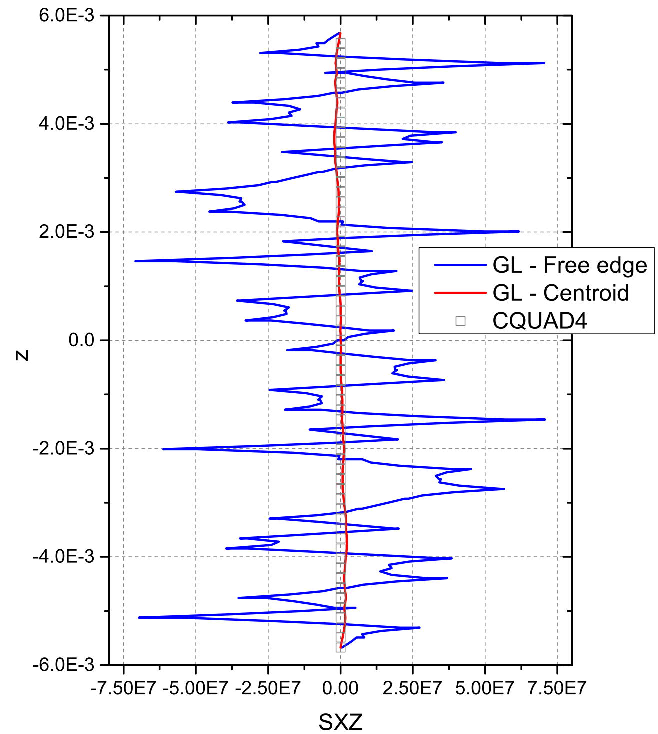
Figure 7: Contour of the maximum laminate failure index according to the Hoffman criteria. The zoomed image shows the area around the first cut-out, where the maximum value appears. The selected element for the global-local analysis is marked in yellow.

5 Conclusions

This paper introduces a user-friendly tool for the obtention of 3D stress solutions in generic composite analysis using the finite element method (FEM) in NX Nastran. The tool serves to link the advanced composite models developed in the university with the commercial FEM codes employed by the industry. The code can easily be installed in Femap as a plug-in, and allows structural engineers to complement their stress analyses with more accurate and reliable solutions. The availability of all the components of the stress tensor allows for the consistent use of advanced composite failure criteria which can predict various failure modes of the laminate structure.



(a)



(b)

Figure 8: Normal (a) and transverse shear (b) stress distributions across the thickness of the laminate in the critical element.

Table 1: Comparison of the failure indexes obtained from the stress results of the shell model and the MUL2@GL method.

Criteria	Shell Nastran Centroid	Mul2@GL Centroid	Mul2@GL Free Edge	Difference %
Hoffman	0.307	0.307	0.510	66
LaRC5 Fiber Tension	-	0.208	0.206	1
LaRC5 Fiber Kinking	-	0.079	0.542	580
LaRC5 Matrix Failure	-	0.029	0.657	2165

References

[1] J. Aboudi, S. Arnold, and B. Bednarcyk. *Micromechanics of Composite Materials*. Butterworth-Heinemann, Oxford, 2013.

[2] E. Carrera, M. Cinefra, E. Zappino, and M. Petrolo. *Finite Element Analysis of Structures Through Unified Formulation*. John Wiley and Sons, Ltd, 2014.

[3] E. Carrera, G.A. Fiordilino, M. Nagaraj, A. Pagani, and M. Montemurro. A global/local approach based on cuf for the accurate and efficient analysis of metallic and composite structures. *Engineering Structures*, 188:188 – 201, 2019.

[4] A.G. de Miguel, I. Kaleel, M.H. Nagaraj, A. Pagani, M. Petrolo, and E. Carrera. Accurate evaluation of failure indices of composite layered structures via various fe models. *Composites Science and Technology*, 167:174 – 189, 2018.

[5] A.G. de Miguel, A. Pagani, and E. Carrera. Free-edge stress fields in generic laminated composites via higher-order kinematics. *Composites Part B: Engineering*, 168:375 – 386, 2019.

[6] O. Hoffman. The brittle strength of orthotropic materials. *Journal of Composite Materials*, 1(2):200–206, 1967.

[7] Siemens Product Lifecycle Management Software Inc. *Element Library Reference*, 2014.

[8] Siemens Product Lifecycle Management Software Inc. *NX Nastran User's Guide*, 2014.

[9] R.D. Mindlin. Influence of rotary inertia and shear flexural motion of isotropic, elastic plates. *Journal of Applied Mechanics*, 18:31–38, 1951.

[10] C. Mittelstedt and W. Becker. Interlaminar stress concentrations in layered structures: Part i - a selective literature survey on the free-edge effect since 1967. *Journal of Composite Materials*, 38(12):1037–1062, 2004.

[11] P. Kerfriden O. Allix, P. Gosselet and K. Saavedra. Virtual delamination testing through non-linear multi-scale computational methods: some recent progress. *CMC:Computers, Materials and Continua*, 32(2):107 – 132, 2012.

[12] S.T. Pinho, R. Darvizeh, P. Robinson, C. Schuecker, and P.P. Camanho. Material and structural response of polymer-matrix fibre-reinforced composites. *Journal of Composite Materials*, 46(19-20):2313–2341, 2012.

[13] J. N. Reddy. *Mechanics of laminated composite plates and shells. Theory and Analysis*. CRC Press, 2^{nd} edition, 2004.