UNIFIED FORMULATION FOR COMPOSITE PLATE STRUCTURES: IMPLEMENTATION AND EVALUATION

Gregório Felipe Oliveira Ferreira[a], Humberto Almeida Jr. [b], Marcelo Leite Ribeiro[a], António Joaquim Mendes Ferreira[c], Volnei Tita[a]

[a] Department of Aeronautical Engineering, São Carlos School of Engineering, University of São Paulo
Av. João Dagnone, 1100 - Jardim Santa Angelina
São Carlos, SP, Brazil - 13563-120
gregferr@gmail.com, malribei@usp.br, voltita@sc.usp.br

[b] Department of Mechanical Engineering, Aalto University
Konetekniikka 1, Otakaari 4, 02150 Espoo, Finland
humberto.almeida@aalto.fi

[c] Department of Mechanical Engineering, Faculty of Engineering of University of Porto,
R. Dr. Roberto Frias s/n, 4200-465 Porto, Portugal
ferreira@gcloud.fe.up.pt

Keywords: Composite plates, Unified Finite Element Formulation, Carrera Unified Formulation (CUF).

Abstract. *The high design freedom of laminated composite structures makes them the best candidates to be used in aeronautic structures, in which, for instance, the coupling can be tailored by properly arranging the stacking sequence of the laminate. The usage of numerical methods for designing composite structures has become crucial to decrease costs related to the design process, manufacturing and testing. Within the Finite Element (FE) method, high order formulations may provide accurate predictions of in-plane and out-of-plane stress and strain fields. This work consists on the application of Finite Element Unified Formulation on composite plates via implementing Carrera's Unified Formulation (CUF) as a User Element Subroutine (UEL), which is linked to Abaqus FE software. Two classical benchmarks are herein considered. After comparing results from Abaqus native shell, user elements (UEL) and literature data, the potentialities and limitations of the used strategy is presented discussed.*

1. INTRODUCTION

Composite materials enable reducing structural weight of aircrafts, increasing payload and aircraft range. For designing these structures, the FE method is very efficient for solving engineering problems, when complex geometries and/or phenomena are involved. However, as shown by Caliri Jr. et al [1], high order finite element formulations are helpful to predict with more precision the response of composite structures under transverse loads. In this context, several FE formulations have been created for improving the results of classical FE method theory by considering many kinds of kinematics hypothesis. Unified formulations (UF) have been developed on the attempt to group different through thickness assumptions existent in the literature. Different theories of plate and shells were developed by [2,3]. He applied his formulation to study multilayered plate and shell structures allowing users to choose the order of expansion in the thickness direction. Carrera Unified Formulation (CUF) has the characteristic of the expansion order to be the same for all directions (u, v, and w). More recently, other works using UF can be found in the literature [4,5]. Ferreira et al. [4] proposed an Abaqus User Element subroutine (UEL) which CUF elements were implemented for evaluating stress-profiles. Thus, a UEL CUF tutorial was showed by performing static applications on composite laminates under bending loads.

In addition, Ribeiro et al. [5] simulated free vibration response of laminated plates, comparing numerical results to experimental observations by using CUF.

The present work aims at evaluating results from application of UEL subroutine and native Abaqus elements, moreover, literature key outcomes were used as comparison parameters to obtain more precisely strain fields and stress profiles.

2. FINITE ELEMENT VIA UNIFIED FORMULATION

The Unified Formulation (UF) [6-9] gives the possibility to implement any high order plate theory. For example, it is possible to use several combinations of displacement field $\boldsymbol{u}$ in order to improve the quality of stress profiles and strain fields. In other words, the displacement field can be expressed as $u = u_1 + zu_2 + z^2 u_3$. Thus, the compact notation for $\boldsymbol{u}$ displacements is given by $u = F_\tau u_i^T$, where $F_\tau = \begin{bmatrix} 1 & z & z^2 \end{bmatrix}$. And, using standard isoparametric FE approach, the displacement field is $u = N_i u_i^T$, where N_i are shape functions. Therefore, based on the PVD (Principle of Virtual Displacement), internal virtual strain energy can provide the fundamental nucleus for high order finite element formulations investigated in the present work, and their components are written as shown in [2,3]:

$$
\begin{aligned}
K_{xx}^{k\tau sij} &= Z_{pp11}^{k\tau s} < N_{i,x} N_{j,x} >_\Omega + Z_{pp16}^{k\tau s} < N_{i,y} N_{j,x} >_\Omega + Z_{pp16}^{k\tau s} < N_{i,x} N_{j,y} >_\Omega + \\
&\quad + Z_{pp66}^{k\tau s} < N_{i,y} N_{j,y} >_\Omega + Z_{nn55}^{k\tau_{,z} s_{,z}} < N_i N_j >_\Omega \\
K_{xy}^{k\tau sij} &= Z_{pp12}^{k\tau s} < N_{i,x} N_{j,y} >_\Omega + Z_{pp26}^{k\tau s} < N_{i,y} N_{j,y} >_\Omega + Z_{pp16}^{k\tau s} < N_{i,x} N_{j,x} >_\Omega + \\
&\quad + Z_{pp66}^{k\tau s} < N_{i,y} N_{j,x} >_\Omega + Z_{nn45}^{k\tau_{,z} s_{,z}} < N_i N_j >_\Omega \\
K_{xz}^{k\tau sij} &= Z_{pn13}^{k\tau s_{,z}} < N_{i,x} N_j >_\Omega + Z_{pn36}^{k\tau s_{,z}} < N_{i,y} N_j >_\Omega + Z_{nn55}^{k\tau_{,z} s} < N_i N_{j,x} >_\Omega + \\
&\quad + Z_{nn45}^{k\tau_{,z} s} < N_i N_{j,y} >_\Omega \\
K_{yx}^{k\tau sij} &= Z_{pp12}^{k\tau s} < N_{i,y} N_{j,x} >_\Omega + Z_{pp26}^{k\tau s} < N_{i,x} N_{j,x} >_\Omega + Z_{pp26}^{k\tau s} < N_{i,y} N_{j,y} >_\Omega + \\
&\quad + Z_{pp66}^{k\tau s} < N_{i,x} N_{j,y} >_\Omega + Z_{nn45}^{k\tau_{,z} s_{,z}} < N_i N_j >_\Omega \\
K_{yy}^{k\tau sij} &= Z_{pp22}^{k\tau s} < N_{i,y} N_{j,y} >_\Omega + Z_{pp26}^{k\tau s} < N_{i,x} N_{j,y} >_\Omega + Z_{pp26}^{k\tau s} < N_{i,y} N_{j,x} >_\Omega + \\
&\quad + Z_{pp66}^{k\tau s} < N_{i,x} N_{j,x} >_\Omega + Z_{nn44}^{k\tau_{,z} s_{,z}} < N_i N_j >_\Omega \\
K_{yz}^{k\tau sij} &= Z_{pn23}^{k\tau s_{,z}} < N_{i,y} N_j >_\Omega + Z_{pn36}^{k\tau s_{,z}} < N_{i,x} N_j >_\Omega + Z_{nn45}^{k\tau_{,z} s} < N_i N_{j,x} >_\Omega + \\
&\quad + Z_{nn44}^{k\tau_{,z} s} < N_i N_{j,y} >_\Omega \\
K_{zx}^{k\tau sij} &= Z_{nn55}^{k\tau s_{,z}} < N_{i,x} N_j >_\Omega + Z_{nn45}^{k\tau s_{,z}} < N_{i,y} N_j >_\Omega + Z_{np13}^{k\tau_{,z} s} < N_i N_{j,x} >_\Omega + \\
&\quad + Z_{np36}^{k\tau_{,z} s} < N_i N_{j,y} >_\Omega \\
K_{zy}^{k\tau sij} &= Z_{nn45}^{k\tau s_{,z}} < N_{i,x} N_j >_\Omega + Z_{nn44}^{k\tau s_{,z}} < N_{i,y} N_j >_\Omega + Z_{np23}^{k\tau_{,z} s} < N_i N_{j,y} >_\Omega + \\
&\quad + Z_{np36}^{k\tau_{,z} s} < N_i N_{j,x} >_\Omega \\
K_{zz}^{k\tau sij} &= Z_{nn55}^{k\tau s} < N_{i,x} N_{j,x} >_\Omega + Z_{nn45}^{k\tau s} < N_{i,y} N_{j,x} >_\Omega + Z_{nn45}^{k\tau s} < N_{i,x} N_{j,y} >_\Omega + \\
&\quad + Z_{nn44}^{k\tau s} < N_{i,y} N_{j,y} >_\Omega + Z_{nn33}^{k\tau_{,z} s_{,z}} < N_i N_j >_\Omega
\end{aligned} \tag{1}
$$

where,

$$
\begin{aligned}
&\left(Z_{pp}^{k\tau s}, Z_{pn}^{k\tau s}, Z_{np}^{k\tau s}, Z_{nn}^{k\tau s} \right) = \left(C_{pp}^k, C_{pn}^k, C_{np}^k, C_{nn}^k \right) E_{\tau s} \\
&\left(Z_{pn}^{k\tau s_{,z}}, Z_{nn}^{k\tau s_{,z}}, Z_{np}^{k\tau_{,z} s}, Z_{nn}^{k\tau_{,z} s}, Z_{nn}^{k\tau_{,z} s_{,z}} \right) = \left(C_{pn}^k E_{\tau s_{,z}}, C_{nn}^k E_{\tau s_{,z}}, C_{np}^k E_{\tau_{,z} s}, C_{nn}^k E_{\tau_{,z} s}, C_{nn}^k E_{\tau_{,z} s_{,z}} \right) \\
&\left(E_{\tau s}, E_{\tau s_{,z}}, E_{\tau_{,z} s}, E_{\tau_{,z} s_{,z}} \right) = \int_{A_k} \left(F_\tau F_s, F_\tau F_{s_{,z}}, F_{\tau_{,z}} F_s, F_{\tau_{,z}} F_{s_{,z}} \right) dz
\end{aligned} \tag{2}
$$

$C_{pp}{}^k$, $C_{pn}{}^k$, $C_{np}{}^k$ and $C_{nn}{}^k$ are elastic stiffness coefficients for each layer k. The symbol $< >_\Omega$ means an integral over the finite element domain and the indexes τ and s are related to the applied kinematic hypothesis of displacement and the number of terms used in the polynomial expansion F_τ and F_s, as previously shown. Thus, the Z-type coefficients (for each layer k) are calculated multiplying the elastic properties by integrals for each combination of τ and s along the thickness A_k, and they are called $E_{\tau s}$. Here, incomplete polynomials $F\tau = [1\ z\ z^2\ z^4]$ and cubic $F\tau = [1\ z\ z^3]$ through-thickness expansions are considered.

3. RESULTS AND DISCUSSION

3.1. Implementation of UEL subroutine

The UEL subroutine starts by the importation of all Abaqus model parameters, i.e. geometry, loads, standard elements, and boundary conditions from an input data in ".inp" format. In this file, Abaqus standard element must be changed by the User Element information. For the elastic UEL (Fig. 1), the simulation starts from the load of input data file ".inp". Thus, the first increment (i) can be initiated. If the increment is minor than the number of steps set in the ".inp", the simulation continues. Inside the UEL subroutine, the Z-CUF integrals are calculated by numerical integration and the generalized constitutive matrix (Cbar) is computed as well. After that, the in-plane numerical integration using the Gauss quadrature points for an 8-node element is carried out. The strain vector (ε) is calculated rom the multiplication of the derivative forms of shape functions and nodal displacements given by Abaqus. Thus, the vector of stresses (σ) can be calculated applying the constitutive matrix (Cbar) in the strain. On the other hand, the stiffness matrix (AMATRX) is calculated by the composition of CUF terms (Eqs. 1-4). Also, the force vector (F) is obtained from AMATRX and nodal displacements (U). After, the residual vector (RHS) could be found by the difference between force (F) and the previous value of RHS, and a new step increment ($i + 1$) is performed. In the UEL, there is another convergence criterion that considers the RHS value and its tolerance (tol). It is verified that the simulation continues for the incrementation step $j + 1$, when the $tol > 5 \times 10^{-3}$ [10]. Thus, the subroutine verifies both criteria: the internal j step (RHS tolerance) and external i (total step) one.

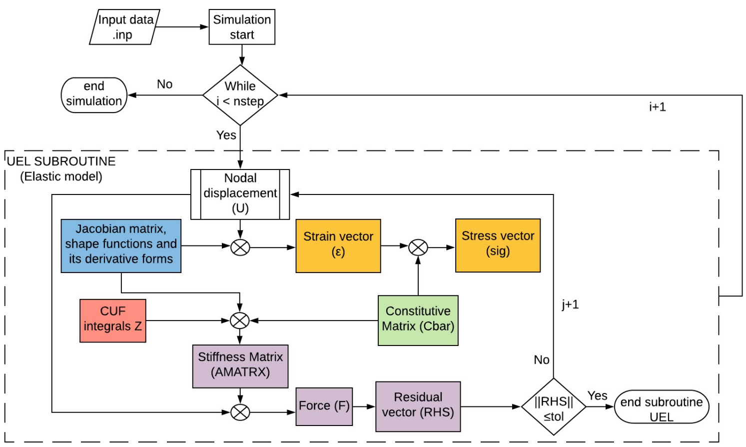

Figure 1- UEL elastic subroutine scheme

3.2. Angle ply laminate with a central load $[-45/45]_s$

A simply supported plate with a central load (Fig. 2) was considered. The plate is square with dimensions of 20 mm and 2 mm of thickness. Moreover, a symmetric angle-ply orientation of [-45/45/45/-45] and a center load of 1 N were considered. The properties used in the analyses was equal to that used in Tita [11] and are equal to, E_{22}=8570 MPa, E_{11}=105000 MPa, G_{12}=4390 MPa, G_{13}=G_{12}, G_{23}=3050 MPa, v_{12}=0.34, v_{13}=0.34 and v_{23}=0.306.

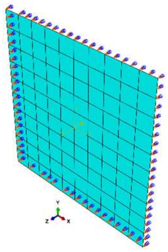

Figure 2 – Concentrated load on composite plate.

For this benchmark, a mesh with 100 elements composed of 8-node native finite element from Abaqus (S8R). This element has reduced integration and there were no numerical problems on its utilization. Moreover, a simple linear-elastic analysis was considered. Then, considering an eight-node element, UEL-CUF used a polynomial expansion for calculating the central displacement u_3, which was [1, z, z^2, z^4]. Based on this, it was possible to show the results of transversal displacement for two analyses – via native Abaqus Fig.3-a) and UEL CUF elements Fig. 3-b).

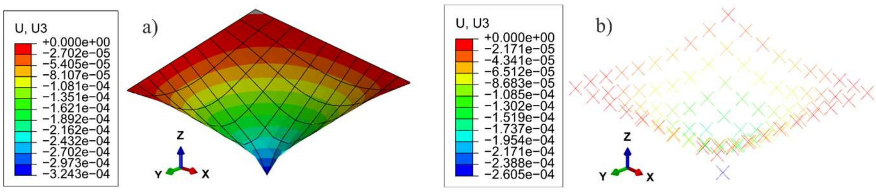

Figure 3 -a) Abaqus native and b) UEL elements

Using S8R element analysis in Abaqus, it was obtained maximum displacement of -3.243×10^{-4} mm (Fig. 3a). Regarding the implemented UEL subroutine solution, a similar maximum displacement of -2.605×10^{-4} mm was found (Fig. 3b). For Abaqus and UEL results, a scale factor of 103 was used to give a consistent visual comparison with two calculated displacements. On the other hand, it was necessary to verify the in-plane and out-of-plane stresses values for evaluating the accuracy of applied methodology. For this reason, elements showed in Fig. 4 and Fig. 5 were chosen. First element is located near central point and was used to calculate in-plane stresses. Profiles presented in Fig. 4 show good correlation between results of native Abaqus S8R and UEL S8 elements. The best agreement is observed from top and bottom layers for the stress $\boldsymbol{\sigma_{22}}$, i.e. UEL S8 equal to 0.11892 MPa and 0.11302 MPa for Abaqus S8R.

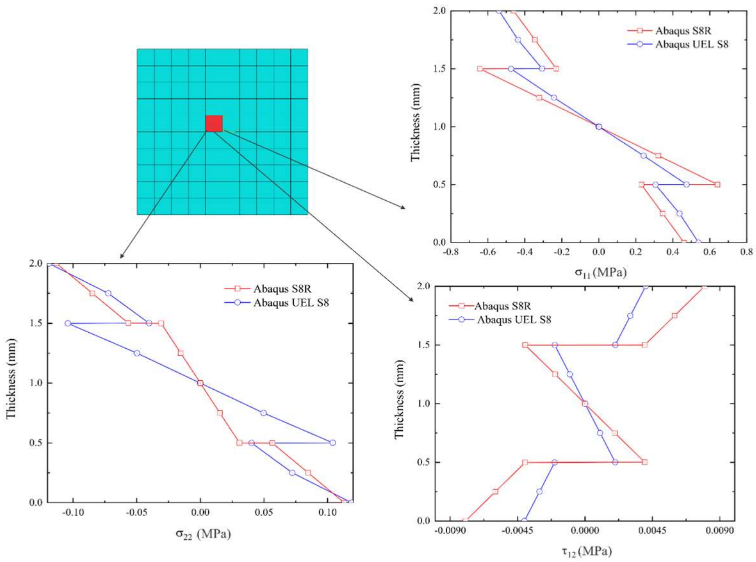

Figure 4- Profile of in-plane stresses

On the other hand, out-of-plane stresses τ_{13} and τ_{23} were evaluated considering the element chosen as Fig. 5 below. It is observed that the best correlation was obtained at the central point of plate, for stress τ_{23}, where maximum values of 0.26 MPa and 0.23 MPa could be found for Abaqus S8R and Abaqus UEL S8, respectively.

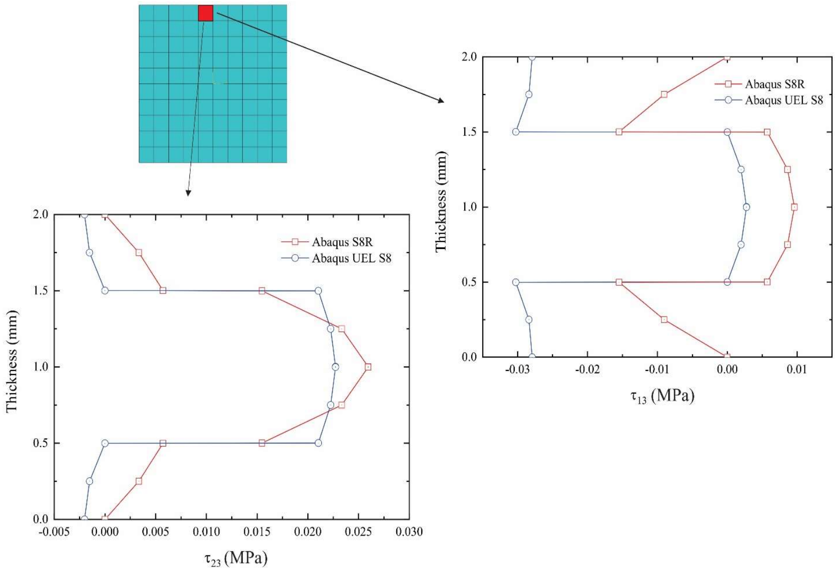

Figure 5 - Out-of-plane stress profiles

3.3. Sandwich plate under distributed load

For this benchmark (Fig. 6), a simple supported square plate of 1000 mm and thickness h=0.1 mm with a distributed load of q=1N/mm^2 was considered according to [12].

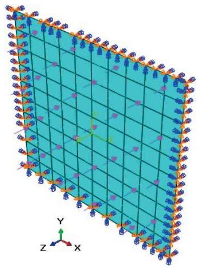

Figure 6- Sandwich plate under distributed load

Core constitutive matrix was also obtained from Ferreira [12], as follow:

$$\overline{Q}_{core} = \begin{bmatrix} 0.999781 & 0.231192 & 0 & 0 & 0 \\ 0.231192 & 0.544886 & 0 & 0 & 0 \\ 0 & 0 & 0.262931 & 0 & 0 \\ 0 & 0 & 0 & 0.266810 & 0 \\ 0 & 0 & 0 & 0 & 0.159914 \end{bmatrix}$$

It is noteworthy, the relation between skin and core matrices, can be considered according to Equation, $\overline{Q}_{skins} = R \cdot \overline{Q}_{core}$, where the factor R=5 was used. CUF polynomial expansion used for calculating the central displacement u_3 was the incomplete sequence [$\mathbf{1}\ \boldsymbol{z}\ \boldsymbol{z^3}$]. This expansion is well-known as a good approach to investigate the transverse displacements on plates as well as transverse shear effects and, for this reason, it was used in this simulation.

The same strategy used in first study case, for evaluating potentialities of CUF elements implemented within UEL subroutine was performed in this benchmark. Qualitatively, Fig. 6 shows results from two approaches applied.

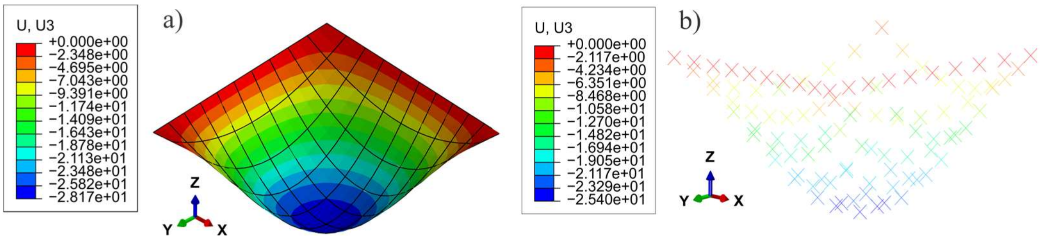

Figure 7- Plate deflection for a composite sandwich plate: a) S8R Abaqus native elements b) Abaqus UEL S8 elements

It can be observed from Fig.7-a) that the usage of Abaqus native elements S8R yield a maximum displacement of −26.17 mm. This result reveals a good agreement with UEL elements that generated -25.4 mm. Furthermore, in-plane and out-of-plane stress were obtained considering the same elements positions used previously in angle-ply studied case, as shown by Fig. 4 and Fig. 5. Thus, results could be organized in Tab. 1, considering normalized expressions as follow [12]:

$$\overline{w} = w \cdot \frac{0.999781}{hq}, \qquad \overline{\sigma}_{\alpha\beta} = \frac{\sigma_{\alpha\beta}}{q}, \qquad \overline{\tau}_{\alpha\beta} = \frac{\tau_{\alpha\beta}}{q} \tag{3}$$

In Tab. 1, literature values of High Order Shear Theory (HOST), Classical Laminate Theory (CLT), among other formulations, could be compared to Abaqus standard and CUF elements. It can be observed that the best approximation was obtained for $\boldsymbol{\sigma}_{11}$ equal to 62.91 and 62.38 from HOST. This value is coherent, mainly because the CUF expansion used shows a high order kinematic hypothesis which corroborates also with out-of-plane τ_{13} stress obtained i.e. 3.089 MPa for HOST and 3.89 MPa for Abaqus UEL S8.

Table 1- Results comparison for the sandwich composite plate

Method	$\overline{w}$ **(a/2, a/2, 0)**	$\overline{\sigma}_{11}$ **(a/2, a/2, h/2)**	$\overline{\sigma}_{22}$ **(a/2, a/2, h/2)**	$\overline{\tau}_{13}$ **(0, a/2, 0)**
HOST	256.13	62.38	38.93	3.089
FOST	236.10	61.87	36.65	3.313
CLT	216.94	61.141	36.622	4.5899
Ferreira [12]	258.74	59.21	37.88	3.593
Analytical [13]	258.94	60.353	38.491	4.3641
Abaqus S8R	281.66	69.01	42.44	3.089
Abaqus UEL S8	253.94	62.91	39.33	3.89

4. CONCLUSIONS AND FUTURE WORKS

From the analyses carried out for the two case studies, it was possible to verify the potential of the strategy of using a UEL subroutine with CUF along with the Abaqus FE package. The main advantage observed using this methodology is that of calculating the out-of-plane stresses without using solid elements, which in most of cases turns the FE analysis slow. The application of classical problems, such as two composite plates under transverse load served as an important parameter for evaluating the strategy of UEL-CUF used. Thus, once the potential of methodology is verified, the usage of this methodology along with progressive damage models is a good alternative for future works, since CUF has a good approach for calculating in-plane and out-of-plane stresses with more accuracy.

ACKNOWLEDGEMENTS

The authors acknowledge the financial support of the Coordination for the Improvement of the Higher Level Personnel (CAPES) and National Council for Scientific and Technological Development (CNPq process numbers: 401170/2014-4 and 310094/2015-1). This material is based upon work supported by the Air Force Office of Scientific Research under award number FA9550-16-1-0222.

REFERENCES

[1] Caliri Jr, Mauricio F.; Ferreira, Antonio JM; Tita, Volnei. A review on plate and shell theories for laminated and sandwich structures highlighting the Finite Element Method. *Composite Structures*, v. 156, p. 63-77, (2016).
[2] Carrera, Erasmo. Theories and finite elements for multilayered, anisotropic, composite plates and shells. *Archives of Computational Methods in Engineering*, v. 9, n. 2, p. 87-140, (2002).
[3] Carrera, Erasmo. Theories and finite elements for multilayered anisotropic, composite plates and shells: A unified compact formulation with numerical assessment and benchmarking. *Archives of Computational Methods in Engineering*, v. 10, n. 3, p. 215-296, (2003b).
[4] Ferreira, Gregório FO, José Humberto S. Almeida Jr, Marcelo L. Ribeiro, António JM Ferreira, and Volnei Tita. Development of a finite element via Unified Formulation: Implementation as a User Element subroutine to predict stress profiles in composite plates. *Thin-Walled Structures* v.157, p. 107107 (2020).
[5] Ribeiro, M.L., Ferreira, G.F., de Medeiros, R., Ferreira, A.J. and Tita, V. Experimental and numerical dynamic analysis of laminate plates via Carrera Unified Formulation. *Composite Structures*, 202, pp. 1176-1185 (2018).
[6] Demasi L., *An invariant model for any composite plate theory and fem applications: the generalized unified formulation*, 50th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, 1-35, (2009).
[7] Cinefra M., Carrera E., Valvano S., Variable kinematic shell elements for the analysis of electromechanical problems, *Mechanics of Advanced Materials and Structures*, 22, 77–106, (2014).
[8] Carrera E., Cinefra M., Petrolo M., Zappino E., *Finite Element Analysis of Structures through Unified Formulation*, Wiley, (2014).
[9] Ferreira A.J.M., Ferreira A.J.M., Araújo A.L., Neves A.M.A., Rodrigues J.D., Carrera E., Mota Soares C.M. A finite element model using a unified formulation for the analysis of viscoelastic sandwich laminates, *Composites Part B*, 45, 1258–1264, (2013).
[10] Abaqus, *Documentation*. Pawtucket: Hibbitt, Karlsson & Sorensen, (2016).
[11] Tita, V. *Contribuição ao estudo de danos e falhas progressivas em estruturas de material compósito polimérico*. Tese de doutorado, Escola de Engenharia de São Carlos, Universidade de São Paulo, São Carlos, SP, Brasil, (2003).
[12] Ferreira AJ. Análise de placas laminadas compósitas através de funções de base radial e polinómios de Legendre. *Revista internacional de métodos numéricos para cálculo y diseño en ingeniería*, vol. 21, núm. 1. (2005).
[13] Srinivas, S. A refined analysis of composite laminates. *Journal of Sound and Vibration* 30, no. 4 495-507 (1973).

RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.