ДГКС ASES | E-13
Kongres 2026
Congress 2026

doi: 10.67563/ASESCONF2026090P

Svetislav Popović[1], Marina Rakočević[2], Olga Mijušković[2]

## A CAD-BASED REFINED ZIG-ZAG THEORY FINITE ELEMENT FRAMEWORK USING GMSH SOFTWARE

**Summary:** A computational framework for the structural analysis of laminated composite plates is presented. The framework combines geometry and mesh generation in Gmsh software with a Refined Zig-Zag finite element solver written by the author. The imported mesh provides the discretization of the computational domain, while the laminate definition, material data, stacking sequence, RZT kinematics, and analysis procedures remain part of the structural solver of the proprietary software. The paper describes data transfer, mesh topology recognition, local coordinates, external definition of boundary conditions and loads, and Visualization Toolkit (VTK)-based post-processing in ParaView software. The proposed approach provides a practical basis for the analysis of plates and flat-facet shell models with arbitrary CAD-based geometry.

**Keywords:** refined zig-zag theory, Gmsh, CAD geometry, finite element mesh, laminated plates, sandwich plates

## MODEL KONAČNIH ELEMENATA ZASNOVAN NA CAD-U PREMA UNAPRIJEĐENOJ CIK-CAK TEORIJI UZ KORIŠĆENJE SOFTVERA GMSH

**Rezime:** Predstavljeno je računsko okruženje za strukturnu analizu slojevitih kompozitnih ploča. Opisani okvir povezuje generisanje geometrije i mreže u programu Gmsh sa solverom konačnih elemenata zasnovanim na Unaprijeđenoj cik-cak teoriji (RZT). Importovana mreža obezbeđuje diskretizaciju proračunskog domena, dok definicija laminata, materijalni podaci, redosled slaganja slojeva, RZT kinematika i procedure analize ostaju dio strukturnog solvera sopstvenog softvera. Rad opisuje prenos podataka, prepoznavanje topologije mreže, lokalne koordinate, eksterno zadavanje graničnih uslova i opterećenja te postprocesiranje u ParaViewu zasnovano na formatu Visualization Toolkit (VTK). Predloženi pristup predstavlja praktičnu osnovu za analizu ploča i modela ljuski aproksimiranih ravanskim elementima sa proizvoljnom CAD geometrijom.

**Ključne reči:** unaprijeđena cik-cak teorija, Gmsh, CAD geometrija, mreža konačnih elemenata, slojevite ploče, sendvič ploče

[1] Ingenieurbüro Stengel, Munich, Germany, svetislav.popovic@rcstengel.com
[2] Faculty of Civil Engineering, University of Montenegro

## 1. INTRODUCTION

The development of numerical analysis, also including a relatively complex mechanical framework of laminated composite and sandwich plates, often assumes initial implementation using simple rectangular domains. Such benchmark models are important because they make it possible to validate the implemented mechanical theory. However, practical plate or shell-type structures are rarely limited to domains that can be adequately represented by simple rectangular geometries. Cut-outs, curved boundaries, local reinforcements and load introduction regions often require a more flexible definition of the geometry domain and thus more freedom withing the meshing procedure.

For an individual researcher, a direct implementation of a complete CAD and meshing environment inside a finite element code does not represent an efficient development strategy. A more rational approach is to separate the initial step of the geometry and mesh preparation from the practical side of the mechanical solver. In the present framework, Gmsh is used for the geometric and meshing part, while the proprietary solver is used. The analysis in this framework is based on the implementation of the Refined Zig-Zag Theory (RZT) [1] within the Finite Element Analysis (FEA). In this way, the solver remains focused on the mechanical formulation, whereas Gmsh provides the geometric flexibility required for arbitrary Q4, T3, and mixed Q4/T3 meshes.

Gmsh [2-3] is an open-source finite element mesh generator with built-in geometry, mesh, solver-coupling and post-processing modules [2-3]. It can be controlled through its graphical interface, command-line interface, scripting language or application programming interface. The geometry module is based on a boundary representation of model entities, while the mesh module can generate triangular (T3), quadrilateral (Q4) and mixed surface meshes of various orders. For the present work, Gmsh is used primarily as a pre-processor, although its modular structure also provides a possible direction for future integration with a graphical user interface.

The aim of this paper is to describe the computational framework that connects a Gmsh-generated mesh with a RZT-implemented finite element solver. The material properties, layer sequence, layer thicknesses, material orientations, RZT functions and numerical analysis procedures remain defined within the proprietary software which governs the analysis of the structural model.

## 2. GENERAL STRUCTURE OF THE FRAMEWORK

The middle surface of the plate or shell-type structure is denoted by $\Omega$. In the CAD-based framework, $\Omega$ is replaced by a discrete surface $\Omega_h$imported from a mesh file:

$$\Omega \approx \Omega_h = \bigcup_{e=1}^{N_e} \Omega_e\,, \qquad \Omega_i \cap \Omega_j = \emptyset \tag{1}$$

for $i \neq j$ except at common edges or nodes.

The numerical model is therefore described by the properties referring to the set of nodal coordinates, the element connectivity and the element topology. For node $i$ the coordinate vector is:

$$\mathrm{x}_i = [x_i \quad y_i \quad z_i]^T \tag{2}$$

An element is defined by its topology and by the ordered list of its nodes:

$$\Omega_e = \{\mathrm{type}_e, I_e\}, \qquad I_e = [i_1 \quad i_2 \quad \cdots \quad i_{n_e}] \tag{3}$$

The number of nodes $n_e$ determines the finite element type. In the present stage, 1st order three-node triangular (T3) and four-node quadrilateral (Q4) layered surface elements are considered:

$$\text{type}_e = T3 \quad \text{if } n_e = 3, \qquad \text{type}_e = Q4 \quad \text{if } n_e = 4 \tag{4}$$

The complete computational loop may be divided into the following stages: definition or import of the CAD geometry in Gmsh; generation of a surface mesh and optional recombination into quadrilateral or mixed elements; export of nodal coordinates, element connectivity and selected node or element groups; reading of the mesh and recognition of element topology in the RZT solver; assignment of laminate, material, layer and orientation data from the structural input model; application of boundary conditions and loads using external node or element lists; solution of the required static, vibration, dynamic or buckling problem; export of generalized displacements, reconstructed fields and stresses in VTK format.

This practical separation within the framework is important from a software-development point of view. Gmsh as a software which is widely used within the scientific community is responsible for geometry and mesh generation, while the RZT solver is responsible for the creation of mechanical model. The data exchange layer is deliberately simple: it transfers points, elements and sets. This also represents a natural basis for a future graphical interface, since such an interface would essentially generate the same model-definition record sets.

## 3. GEOMETRY AND MESH GENERATION IN GMSH

In Gmsh, a model is defined by topological entities. A surface is bounded by curves, curves are bounded by points, and physical groups may be used to assign engineering meaning to selected regions. For the present layered-plate element formulation, the imported geometry is interpreted as the reference middle surface of the laminate. The thickness is not meshed explicitly; it is introduced in the process by the laminate definition.

Two practical geometry definitions are relevant. The first one is a native Gmsh geometry, created by points, curves and surfaces in a .geo script or interactively in the graphical interface. The second one is an imported CAD geometry, for example from a STEP/ACIS/SAT file through the OpenCASCADE kernel. In both cases, the final object required by the RZT solver is a surface mesh.

Based on the software manual [2], the mesh generation procedure in Gmsh is bottom-up. Curves are discretized first, then surfaces are meshed using the already generated boundary mesh. This is useful for structural models because the analyst can control mesh density along loaded or supported boundaries before the surface mesh is generated. The surface mesh may be triangular, quadrilateral or mixed. Quadrilateral meshes may be obtained by structured transfinite meshing, extrusion-based procedures or recombination of triangular meshes.

The choice of algorithm is problem-dependent. Delaunay-type algorithms are efficient for large plane surfaces and complex mesh size fields. Frontal-Delaunay algorithms are useful when high element quality is desired. MeshAdapt-type algorithms rely on local mesh modifications, such as edge swaps, splits and collapses, and are often robust for complex curved surfaces. In the present framework the meshing algorithm is thus not embedded in the structural solver; it is selected in Gmsh according to the geometry and desired element topology.

## 4. IMPORTED MESH DATA

The imported mesh contains a list of nodes and a list of elements. Let the total number of nodes be $N_n$. Each node carries seven generalized RZT degrees of freedom:

$$\mathrm{d}_i = [u_i \quad v_i \quad w_i \quad \theta_{xi} \quad \theta_{yi} \quad \psi_{xi} \quad \psi_{yi}]^T \tag{5}$$

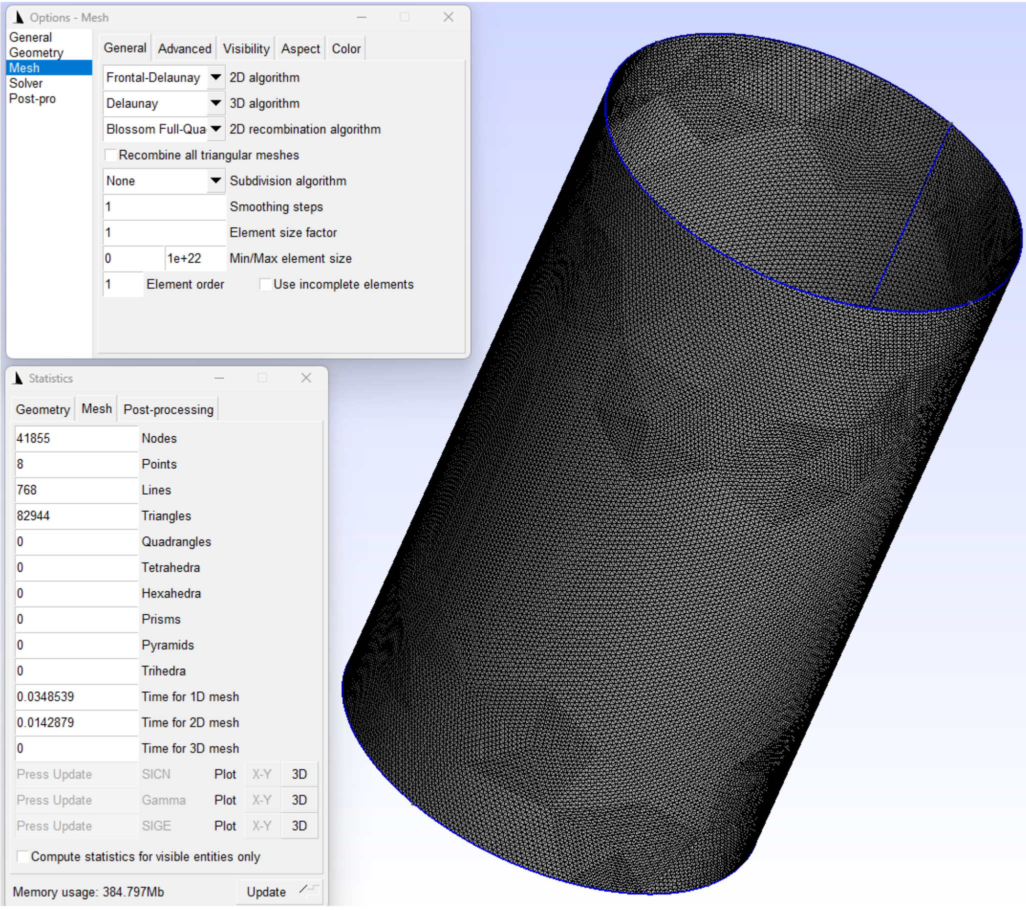


*Figure 1: Mesh creation inside Gmsh software*

The element vector is obtained by collecting the nodal vectors of the element:

$$\mathrm{d}_e = \left[\mathrm{d}_{i_1}^T \quad \mathrm{d}_{i_2}^T \quad \dots \quad \mathrm{d}_{i_{ne}}^T\right]^T \tag{6}$$

The corresponding Boolean assembly operator $\mathbf{A}_e$ maps the element vector to the global displacement vector $\mathbf{d}$:

$$\mathrm{d}_e = \mathrm{A}_e \mathrm{d} \tag{7}$$

The imported mesh does not need to have a regular row-column structure. The only requirements are that the nodal coordinates are unique within a selected tolerance, that the element connectivity is valid, and that the element orientation is consistent. If coincident nodes are present in the mesh file, they must be merged or treated explicitly, because duplicated nodes create artificial separation of the finite element domain.

For a flat plate mesh, the z-coordinate of all nodes is often constant. For a flat-facet shell approximation, the nodes are not coplanar globally, but each finite element is treated as a locally planar plate element. The element normal and local in-plane axes are then constructed from the imported element geometry.

Due to the nature of the laminate composite materials, which have layers with arbitrarily oriented fibers, the mechanical meaning of the material axes must be separated from the global CAD axes. When working with a rectangular plate aligned in the global xy-plane, it's common to align the local element axes with the global axes. However, for imported geometries – particularly shell-type surfaces – this alignment may not be adequate for effectively creating the model. Thus, each element must have a local orthonormal basis.

Let the local element basis be defined by the unit vectors $e1$, $e2$ and $e3$. The vector $e3$ is normal to the element surface, while $e1$ and $e2$ span the local tangent plane:

$$\mathbf{e}_3 = \frac{\mathbf{a}_1 \times \mathbf{a}_2}{\|\mathbf{a}_1 \times \mathbf{a}_2\|}, \qquad \mathbf{e}_2 = \mathbf{e}_3 \times \mathbf{e}_1 \tag{8}$$

Here a1 and a2 are two independent geometric vectors constructed from the element nodes. The direction $e1$ may be obtained by projecting a prescribed reference vector $r$ onto the local tangent plane:

$$\mathbf{e}_1 = \frac{\mathbf{r} - (\mathbf{r} \cdot \mathbf{e}_3)\mathbf{e}_3}{\|\mathbf{r} - (\mathbf{r} \cdot \mathbf{e}_3)\mathbf{e}_3\|} \tag{9}$$

If the projection in Eq. (9) becomes too small, an alternative reference vector must be used. This situation can occur when the selected reference vector is nearly parallel to the local element normal. Thus, certain validation concepts need to be included within the analysis. With this construction, the material direction 1 is associated with $e_1$, direction 2 with $e_2$, and the thickness direction with $e_3$. The usual ply angle is then measured in the local tangent plane, not necessarily in the global xy-plane. In this way, it is possible to create a laminate with arbitrarily oriented layers. This approach is particularly important for spatial CAD-based models. For example, if a cylindrical shell is meshed by flat facets, the local normal changes from element to element. The thickness direction follows the local normal, while the selected material reference vector defines the preferred in-plane direction. This enables the appropriate laminate definition to be used on faceted shell surfaces.

## 5. RZT KINEMATICS ON THE IMPORTED SURFACE

The imported mesh defines the in-plane $xy$-finite element domain. The through-thickness coordinate $z$ is reconstructed locally along $e_3$. For the $k$-th layer, the refined zig-zag displacement field is written as [1]

$$\begin{aligned} u^{(k)}(x,y,z) &= u_0(x,y) + z\theta_x(x,y) + \varphi_x^{(k)}(z)\psi_x(x,y), \\ v^{(k)}(x,y,z) &= v_0(x,y) + z\theta_y(x,y) + \varphi_y^{(k)}(z)\psi_y(x,y), \\ w(x,y,z) &= w_0(x,y) \end{aligned} \tag{10}$$

The unknowns $u_0$, $v_0$ and $w_0$ are the reference-surface displacements, $\theta_x$ and $\theta_y$ are the first-order rotations, and $\psi_x$ and $\psi_y$ are the zigzag amplitudes. The functions $\varphi_x^{(k)}(z)$ and $\varphi_y^{(k)}(z)$ are the layerwise zigzag functions. Their role is to enrich the through-thickness distribution of the in-plane displacement field without introducing independent displacement unknowns for every layer. Practical implementation of the RZT kinematics within the family of quadrilateral elements can be taken from [4].

Over an imported element, the generalized field is interpolated as

$$q(\xi,\eta) = \sum_{i=1}^{n_e} N_i(\xi,\eta)q_i, \qquad q_i \in \{u_i,\ v_i,\ w_i,\ \theta_{xi},\ \theta_{yi},\ \psi_{xi},\ \psi_{yi}\} \tag{11}$$

The interpolation functions $N_i$ directly depend on the element topology. For a quadrilateral element (Q4), bilinear functions are used. For a triangular element (T3), linear area-coordinate functions are used. The mesh import procedure therefore affects the element interpolation but does not change the generalized RZT nodal variables.

The in-plane $\varepsilon_p^{(k)}$ and transverse shear strains $\gamma_s^{(k)}$ are expressed in matrix form:

$$\varepsilon_p^{(k)} = \mathrm{B}_p^{(k)}\mathrm{d}_e, \qquad \gamma_s^{(k)} = \mathrm{B}_s^{(k)}\mathrm{d}_e \tag{12}$$

with $B_p^{(k)}$ and $B_s^{(k)}$ being the strain-displacement matrices associated with the in-plane and transverse shear strain components, respectively. These matrices depend on the finite element interpolation functions and their derivatives with respect to the local element coordinates. Therefore, their explicit form depends on the adopted element topology. In the case of T3 elements, this difference is especially important in the transverse shear part of the formulation. Since the derivatives of the linear triangular interpolation functions are constant over the element, the standard $\mathbf{B}_s^{(k)}$ matrix may lead to an overly stiff transverse shear response. For this reason, additional T3 shear formulations, such as element-level shear scaling and mixed interpolation of tensorial shear components, can be introduced while preserving the same RZT nodal degrees of freedom.

The corresponding layer constitutive equations are

$$\sigma_p^{(k)} = \overline{\mathrm{Q}}_p^{(k)}\varepsilon_p^{(k)}, \qquad \tau_s^{(k)} = \overline{\mathrm{Q}}_s^{(k)}\gamma_s^{(k)} \tag{13}$$

with $\overline{\mathrm{Q}}_p^{(k)}$ and $\overline{\mathrm{Q}}_s^{(k)}$ being the ply-dependent transformed stiffness matrices of the $k$-th layer [5].

## 6. ELEMENT MATRICES AND ASSEMBLY

The finite element solver was developed in the C# programming language within the .NET 8 environment. Element stiffness matrices and load vectors are assembled into sparse global matrices. After imposing the essential boundary conditions, the reduced linear system is solved using a sparse direct solver. For symmetric positive-definite systems, sparse Cholesky factorization with minimum-degree ordering is employed.

Numerical integration is performed over the finite element surface. The Q4 element is formulated as a bilinear isoparametric element, in which the same shape functions are used to interpolate the geometry and the seven generalized RZT displacement variables. Complete integration is performed using a 2 x 2 Gauss-Legendre rule, while one-point reduced integration may be applied to the transverse shear contribution. The T3 element is a linear isoparametric simplex based on barycentric shape functions. Its complete surface integration uses a three-point triangular quadrature rule, whereas reduced integration is performed at the element centroid. The laminate constitutive matrices are evaluated layer by layer through the thickness. Since the material properties are constant within each layer and the RZT zigzag functions are piecewise linear, the corresponding thickness integrals are evaluated exactly for the adopted polynomial fields.

The element stiffness matrix $K_e$ is obtained by integrating the strain energy over the element surface and through the laminate thickness. Since the material properties, ply-angle and zigzag functions may vary from layer to layer, the through-thickness integration is performed layerwise:

$$\mathrm{K}_e = \sum_{k=1}^{N_L} \int_{A_e} \int_{z_{k-1}}^{z_k} \left[ \left(\mathrm{B}_p^{(k)}\right)^T \overline{\mathrm{Q}}_p^{(k)} \mathrm{B}_p^{(k)} + \left(\mathrm{B}_s^{(k)}\right)^T \overline{\mathrm{Q}}_s^{(k)} \mathrm{B}_s^{(k)} \right] dz\, dA \tag{14}$$

The element load vector is assembled in the same topological framework. For a distributed transverse pressure $p$ over an element surface $A_e$, the consistent element load vector $f_e$ is

$$\mathrm{f}_e = \int_{A_e} \mathrm{N}_f{}^T\, p\, dA \tag{15}$$

with $N_f$ being load interpolation vector associated with the transverse displacement degrees of freedom.

The global stiffness matrix and the global load vector are assembled as

$$\mathrm{K} = \sum_{e=1}^{N_e} \mathrm{A}_e^T \, \mathrm{K}_e \mathrm{A}_e, \qquad \mathrm{F} = \sum_{e=1}^{N_e} \mathrm{A}_e^T \, \mathrm{f}_e \tag{16}$$

After imposing the prescribed boundary conditions, the static problem is reduced to the free degrees of freedom:

$$\mathrm{K}_{ff}\mathrm{d}_f = \mathrm{F}_f \tag{17}$$

The same imported mesh can be used for other analysis types. In free vibration analysis, the stiffness matrix is combined with the mass matrix $M$. In dynamic analysis, the time integration procedure (e. g. Newmark or Runge-Kutta) uses the same global matrices ($K$ and $M$) and load definition. In buckling analysis, the elastic stiffness matrix is combined with a geometric stiffness matrix ($K_G$) obtained from the selected preload state or prescribed in-plane load resultants. In this paper, only the static analysis is exemplary presented.

## 7. BOUNDARY CONDITIONS AND LOAD DEFINITIONS

Boundary conditions and loads are assigned using additional input data. In the present workflow, selected nodes or elements are stored in external lists which are read by the input class of the solution software. These lists may originate from physical groups in Gmsh or from selections performed in ParaView after visual inspection of the mesh.

Let $S_c$ represent the set of constrained global degrees of freedom. The imposed essential boundary conditions $d_j$ are expressed as

$$d_j = \bar{d}_j, \qquad j \in S_c \tag{18}$$

For homogeneous supports, the prescribed values are set to zero. For displacement-controlled analyses, nonzero prescribed values may be assigned to selected degrees of freedom. This is especially useful for patch loading, edge displacement control and future delamination examples where opening or closing displacements may be prescribed on selected part of the domain or laminate regions. The framework therefore avoids hard-coding supports and loads into the mesh generator. Gmsh defines the geometry and mesh; the structural input files define which displacement components are fixed and which forces, moments, pressures or prescribed displacements are applied.

## 8. POST-PROCESSING AND VISUALIZATION

The numerical results are exported in VTK format. This is convenient because ParaView software can visualize the imported geometry, the computed displacement field, layer quantities and stress components. The stress components can be exported as point- or cell-associated VTK data and visualized in ParaView. In the present paper, only the displacement field is shown for brevity, whereas stress visualization is fully supported by the developed computational framework.

At a point of the imported element and at a selected thickness coordinate z, the computed displacement field is

$$\mathrm{d}_{rec}^{(k)}(x,y,z) = [u^{(k)}(x,y,z) \quad v^{(k)}(x,y,z) \quad w(x,y,z)]^T \tag{19}$$

For flat plates, the reconstructed upper and lower surfaces remain parallel to the global plate plane. For faceted shell models, the reconstructed points are obtained by moving from the local midsurface along the local normal $e_3$:

$$\mathrm{x}_{rec} = \mathrm{x}_{mid} + z\mathrm{e}_3 + u^{(k)}\mathrm{e}_1 + v^{(k)}\mathrm{e}_2 + w\mathrm{e}_3 \tag{20}$$

This expression makes the through-thickness output consistent with the imported CAD geometry. It also provides a clear visualization path for sandwich plates and layered laminates, because the upper surface, lower surface and selected interfaces may be drawn in their true geometric position. The post-processing procedure is therefore not only a visualization tool. It is also a diagnostic tool for checking mesh orientation, local axes, support definitions, load placement and the consistency of reconstructed through-thickness fields.

## 9. NUMERICAL EXAMPLE

A simply supported four-layer laminated composite plate with an antisymmetric cross-ply stacking sequence $0°/90°/0°/90°$ is considered. All layers have the same thickness and the same orthotropic material properties:$E_1/E_2 = 25, E_2 = 1, G_{12} = G_{13} = 0.5, G_{23} = 0.2, \nu_{12} = \nu_{13} = 0.25$. The plate is subjected to a uniformly distributed transverse load of intensity $q = 1$. In this example, a square plate is considered, so that $L_x/L_y = 1$, while two span-to-thickness ratios are analyzed $L_y/h = 4$ and $L_y/h = 10$. The objective of the analysis is to investigate the influence of the finite element type and mesh topology on the maximum transverse displacement $w$, as well as on the normal stress $\sigma_x$ at the bottom fiber of the first layer at the plate center.

Four meshes are used in the comparison and are denoted as M1-M4. Mesh M1 is a structured (32 **x** 32) Q4 mesh and is used as the reference regular quadrilateral discretization. Mesh M2 is a coarse unstructured T3 mesh with 832 triangular elements, while mesh M3 is a refined unstructured T3 mesh with 6414 triangular elements. Mesh M4 is an unstructured Q4 mesh with 344 quadrilateral elements, used to assess the sensitivity of the formulation to non-regular quadrilateral discretization. Used meshes are displayed within the displacement plot in Figures 2-3.

The results obtained using the developed formulation are compared with reference numerical results obtained using SHELL181, SHELL281 and SOLID185 elements in Ansys software, as well as with the Layerwise solution reported in [6].

The results of the analysis are presented in Tables 1 and 2.

*Table 1: Transverse displacement w of the plate*

| Ly/h | SHELL 181 | SHELL 281 | SOLID 185 | Layerwise [6] | **M1** | **M2** | **M3** | **M4** |
|---|---|---|---|---|---|---|---|---|
| 4 | 1,878 | 1,832 | 1,902 | 1,874 | **2,050** | **2,036** | **2,048** | **2,048** |
| 10 | 11,248 | 11,227 | 11,891 | 11,693 | **11,951** | **11,871** | **11,960** | **11,884** |

*Table 2: Normal stress $\sigma_x(L_x/2, L_y/2, 0)$ of the plate*

| Ly/h | SHELL 181 | SHELL 281 | SOLID 185 | Layerwise [6] | **M1** | **M2** | **M3** | **M4** |
|---|---|---|---|---|---|---|---|---|
| 4 | 12,439 | 12,606 | 15,598 | 15,177 | **14,386** | **14,275** | **14,396** | **14,265** |
| 10 | 79,260 | 79,448 | 78,880 | 76,409 | **75,309** | **74,805** | **75,497** | **74,265** |

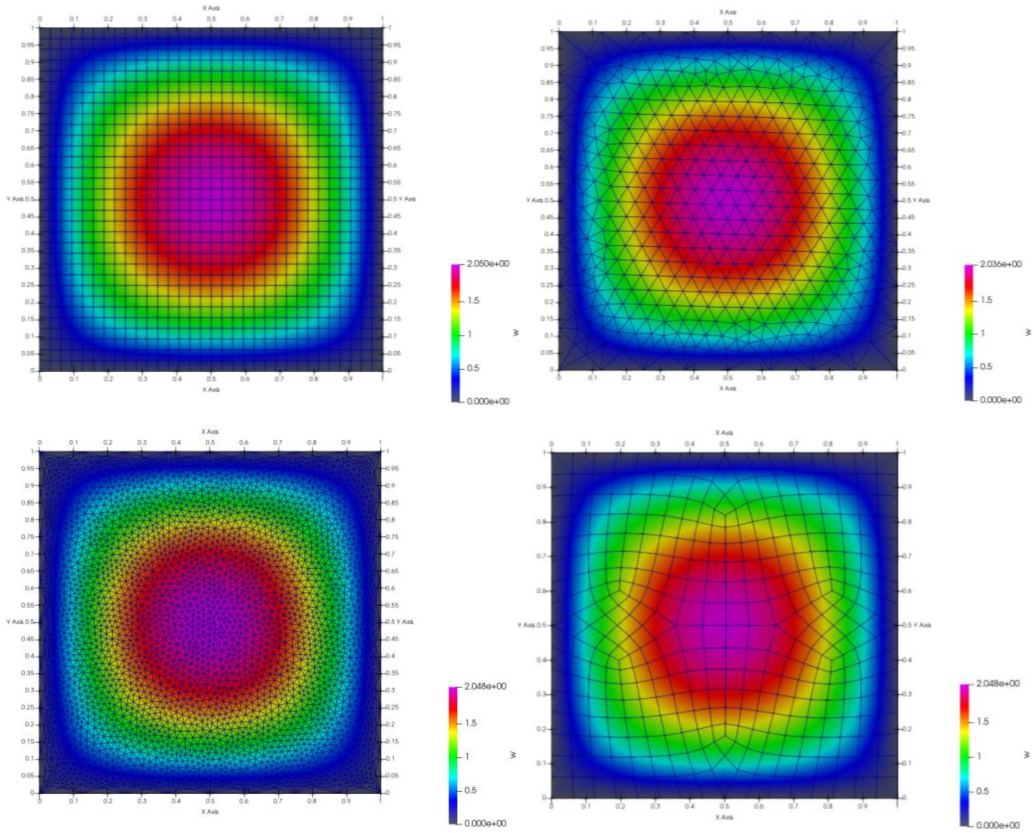


*Figure 2: Transverse displacement w; $L_y/h = 4$: M1 - left above; M2 – right above; M3 – left below; M4 – right below*

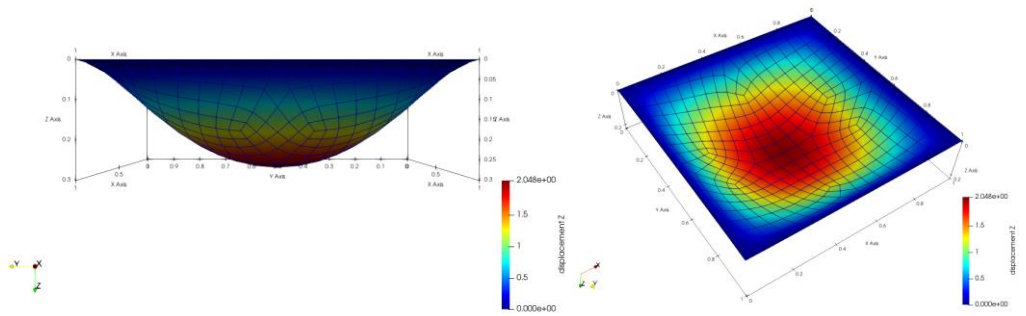


*Figure 3: Transverse displacement w; $L_y/h = 4$; M4: left YZ-view; right: isometric view with the visualized deformed shape*

The layerwise solution [6] is used as the reference value for the evaluation of the relative deviations. For $L_y/h = 4$, the transverse displacements obtained with the developed formulation range from 2.036 to 2.050, while the reference layerwise value is 1.874. The corresponding deviation ranges from approximately 8.64% to 9.39%. For the thinner plate, $L_y/h = 10$, the agreement is significantly better. The computed displacements range from 11.871 to 11.960, whereas the reference value is 11.693, leading to deviations between 1.52% and 2.28%. For the normal stress $\sigma_x$ at the bottom fiber of the first layer at the plate center, the values obtained for

$L_y/h = 4$ range from 14.265 to 14.396, while the layerwise value is 15.177. The deviation is negative and ranges from approximately -5.15 % to -6.01%. For $L_y/h = 10$, the computed stress values range from 74.265 to 75.497, compared with the reference value of 76.409, which corresponds to deviations between approximately -1.19% and -2.81%.

Although the presented CAD-based framework is fully capable of handling complex and arbitrary geometries, a benchmark square plate configuration was selected for this paper. This choice ensures a direct and transparent validation of the implemented RZT finite element formulation against well-established analytical and numerical solutions, without introducing geometry-induced boundary condition errors.

## 10. FINAL COMMENTS

A CAD-based numerical framework for Refined zig-zag finite element analysis of laminated composite plates has been presented.

The key idea is to keep the RZT mechanical formulation independent from the mesh-generation procedure.

Gmsh provides the geometry and mesh, while the structural solver performs the stiffness computation, assembly, solution procedure and the post-processing including stress analysis.

The framework is suitable for regular plates, non-rectangular plates, locally refined domains, mixed triangular-quadrilateral meshes and flat-facet shell approximations. Since the laminate definition remains part of the structural model, different material systems, stacking sequences and layer thicknesses can be analyzed on the same imported mesh.

The proposed data flow also defines the natural direction for future software development. The current input lists for boundary conditions, loads and output requests can later be generated by a graphical interface, while preserving the validated finite element solver as the computational core.

Future work will focus on exploiting the full potential of the Gmsh integration by analyzing laminates with complex cut-outs and curved boundaries, as well as incorporating shear locking remediation techniques for highly thin configurations.

## REFERENCES


[1] Tessler A., Di Sciuva M., Gherlone M., Refined Zigzag Theory for Laminated Composite and Sandwich plates, NASA/TP-2009-215561

[2] Geuzaine C., Remacle J.-F. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities, International Journal for Numerical Methods in Engineering, 79, 1309-1331, 2009.

[3] Gmsh Reference Manual, Version 4.15.2, https://gmsh.info/doc/texinfo/gmsh.html.

[4] Di Sciuva M., Sorrenti M. A Family of C0 Quadrilateral Plate Elements Based on the Refined Zigzag Theory for the Analysis of Thin and Thick Laminated Composite and Sandwich Plates. Journal of Composites Science 2019; 3:100.

[5] J.N. Reddy, Mechanics of Laminated Composite Plates and Shells: Theory and Analysis, CRC Press, 2004.

[6] Rakočević M., Popović S. Static Analysis of Laminated Composite Plates. Contemporary Issues in Theory of Structures – Monograph, Faculty of Civil Engineering, University of Belgrade