iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 14 **Issue:** III **Month of publication:** March 2026

DOI: https://doi.org/10.22214/ijraset.2026.78733

A Validated Python-Based Approach for Composite Plate Analysis with Geometric Discontinuities

Dr. Shailesh S. Kadre[1], Harsha Vardhan Sirisolla[2], Satyanand Velpukonda[3], Sivaram Ydesi[4], Rohit Gold Puspala[5]
Cyient Ltd, Hyderabad, Telangana, India

Abstract: This study presents the development and validation of an open-source finite element framework implemented in Python for the analysis of laminated composite plates using First Order Shear Deformation Theory (FSDT). The formulation employs graded quadrilateral meshing with eight-noded serendipity elements and incorporates full laminate constitutive modeling through computation of the A, B, and D stiffness matrices along with transverse shear effects. The framework enables automated geometry generation for plates with geometric discontinuities, global stiffness assembly, displacement solution, ply-level stress recovery, and contour visualization. To assess numerical accuracy, the developed solver is benchmarked against the layered shell formulation (SHELL181) in ANSYS under identical geometry, mesh density, material properties, boundary conditions, and loading. Comparisons are performed for in-plane and transverse displacements as well as normal stress components in laminated plies for a composite plate containing a central circular cutout. The Python-based results demonstrate strong agreement with the commercial solver, with displacement deviations below 1% and stress deviations within 5%. The study highlights the feasibility of using open-source computational tools for small- to medium-scale composite structural analyses, reducing dependency on high-cost licensed software for preliminary design studies and parametric investigations. Furthermore, the developed framework is extended toward the generation of high-fidelity finite element datasets to enable data-driven modeling. Specifically, the solver infrastructure is being utilized to create structured simulation databases for training Graph Neural Network (GNN)-based predictive models aimed at replacing computationally expensive finite element simulations. This integration establishes a pathway toward rapid surrogate modeling of composite structures while preserving physics-informed accuracy.

Index Terms: Finite Element Analysis (FEA), First Order Shear Deformation Theory (FSDT), ANSYS (SHELL181) Serendipity Elements, Python, Laminated Composite Plates.

I. INTRODUCTION

Laminated composite plates are extensively used in aerospace, automotive, marine, and energy applications due to their high stiffness-to-weight ratio and tailorable mechanical properties. The structural behavior of such laminates is strongly influenced by stacking sequence, fiber orientation, and geometric discontinuities such as cutouts, which often introduce significant stress concentrations. Accurate prediction of displacement and stress fields in laminated structures is therefore essential for reliable design and failure assessment. Finite element analysis (FEA) has become the standard computational tool for evaluating composite structural response. Commercial software packages provide robust implementations of layered shell elements capable of modeling laminated behavior with high accuracy. Among these, layered shell formulations based on First Order Shear Deformation Theory (FSDT) are widely adopted for moderate-thickness composite plates, as they account for transverse shear deformation effects while maintaining computational efficiency. Despite their accuracy and industrial reliability, commercial finite element platforms require high-cost licenses, which may limit accessibility for small-scale industries, educational institutions, and early-stage design exploration. For preliminary analysis, academic research, and parametric investigations, there is growing interest in open and customizable computational frameworks that can reproduce validated commercial results while remaining flexible and cost-effective. The objective of this work is to develop and validate a Python-based finite element framework for the analysis of laminated composite plates using FSDT. The proposed implementation employs eight-noded quadrilateral elements with five degrees of freedom per node and incorporates laminate constitutive modeling through evaluation of the A, B, and D stiffness matrices along with transverse shear effects. The developed solver includes geometry generation, mesh creation for plates with circular cutouts, global stiffness assembly, displacement solution, and ply-level stress recovery.

To assess numerical reliability, the results obtained from the Python framework are compared to those from a commercial finite element implementation using layered shell elements under identical modeling conditions. The comparison focuses on displacement and normal stress components, providing quantitative validation of the proposed approach. The study demonstrates the feasibility of using an open-source computational framework for laminated composite plate analysis in small to medium-scale engineering applications.

In addition to its standalone capabilities, the developed framework serves as a foundation for data-driven modeling of composite structures. The solver infrastructure is being utilized to generate high-fidelity finite element datasets across varying geometries, material configurations, and loading conditions. These datasets are intended for training Graph Neural Network (GNN)-based predictive models, with the objective of approximating finite element solutions at significantly reduced computational cost. This integration establishes a pathway toward rapid surrogate modeling and real-time structural assessment of composite systems.

II. THEORETICAL FORMULATION

A. First Order Shear Deformation Theory (FSDT)

The laminated composite plate formulation adopted in this study is based on First Order Shear Deformation Theory (FSDT), which accounts for transverse shear deformation effects and is suitable for moderately thick laminates.

According to FSDT, the displacement field at any point within the laminate is expressed as:

$$u(x,y,z) = u_0(x,y) + z\,\theta_x(x,y)$$
$$v(x,y,z) = v_0(x,y) + z\,\theta_y(x,y)$$
$$w(x,y,z) = w_0(x,y)$$

where:

- u_0, v_0 are mid-plane in-plane displacements,
- w_0 is transverse displacement,
- θ_x, θ_y are rotations of the normal about the y- and x-axes respectively,
- z is the thickness coordinate.

This formulation introduces five degrees of freedom per node.

The strain components are decomposed into membrane, bending, and transverse shear contributions:

$$\varepsilon = \varepsilon_0 + z\kappa$$

where:

- ε_0 are mid-plane membrane strains,
- κ are curvatures.

The transverse shear strains are expressed as:

$$\gamma_{xz}, \gamma_{yz}$$

A shear correction factor is introduced to compensate for the constant shear strain assumption through thickness.

B. Laminate Constitutive Modeling

Each ply is treated as an orthotropic lamina under plane stress conditions. The reduced stiffness matrix Q for a lamina is defined in its material coordinate system as:

$$Q = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix}$$

The stiffness matrix is transformed into the laminate coordinate system using standard lamination theory transformation relations to obtain $\bar{Q}$.

The global laminate stiffness matrices are obtained by integrating through thickness:

$$A = \sum \bar{Q}_k (z_k - z_{k-1})$$
$$B = \frac{1}{2} \sum \bar{Q}_k (z_k^2 - z_{k-1}^2)$$

$$D = \frac{1}{3}\sum \bar{Q}_k (z_k^3 - z_{k-1}^3)$$

where z_k represents the ply interface coordinates.
A matrix represents membrane stiffness,
B represents bending–membrane coupling,
D represents bending stiffness.
The transverse shear stiffness matrix is evaluated separately using ply-level shear moduli and assembled across the laminate thickness.

C. Finite Element Discretization

The plate domain is discretized using eight-noded serendipity quadrilateral elements. Each node possesses five degrees of freedom corresponding to:

$$\{u_0, v_0, w_0, \theta_x, \theta_y\}$$

Quadratic shape functions are employed to interpolate displacement fields within the element. Numerical integration is performed using 3×3 Gauss quadrature.
The element stiffness matrix includes contributions from:

- Membrane stiffness
- Bending stiffness
- Membrane–bending coupling
- Transverse shear stiffness

The global stiffness matrix is assembled using standard finite element procedures, and boundary conditions are enforced by partitioning the system into free and constrained degrees of freedom

III. NUMERICAL IMPLEMENTATION

The developed finite element framework is implemented in Python using numerical and visualization libraries for matrix operations and post-processing. The implementation follows a structured workflow consisting of geometry generation, mesh creation, stiffness assembly, boundary condition enforcement, system solution, and stress recovery.

A. Geometry and Mesh Generation

The problem domain consists of a rectangular laminated composite plate containing a central circular cutout. To accurately capture stress concentration effects around the hole, a structured quadrilateral mesh is generated using a block-based strategy. An inner square transition region is constructed around the circular boundary, and graded meshing is applied to ensure adequate resolution near the geometric discontinuity. An initial four-noded quadrilateral mesh is generated and subsequently upgraded to eight-noded serendipity elements by inserting midside nodes. This approach enables quadratic interpolation while maintaining structured mesh topology. The grading strategy ensures smooth element size transition from the hole boundary to the outer plate edges.

B. Global Stiffness Assembly

For each element, membrane, bending, coupling, and shear stiffness contributions are computed using 3×3 Gauss integration. The laminate constitutive matrices (A, B, D) and transverse shear stiffness matrix are incorporated directly into the element formulation.
The element stiffness matrices are assembled into a global stiffness matrix using standard finite element connectivity mapping. The implementation uses five degrees of freedom per node, resulting in a global system consistent with FSDT assumptions.

C. Boundary Conditions and Loading

Boundary conditions are applied by constraining the degrees of freedom along one edge of the plate. A uniformly distributed in-plane load is applied along the opposite edge. The system of equations is partitioned into free and constrained degrees of freedom, and the resulting linear system is solved using direct matrix inversion techniques.

D. Stress Recovery and Post-Processing

Following displacement solution, mid-plane strains and curvatures are evaluated at Gauss points and averaged at the element level. Ply-level stresses are computed using transformed stiffness matrices and laminate kinematic relations. Nodal stress values are obtained through element averaging for visualization.

Contour plots of displacement and stress fields are generated to enable direct comparison with results obtained from the commercial finite element software.

IV. VALIDATION AGAINST COMMERCIAL FINITE ELEMENT SOFTWARE

A. Problem Definition

To validate the developed Python-based finite element framework, a laminated composite plate with a central circular cutout is analyzed and benchmarked against a commercial finite element solver.

The geometry consists of a square plate of dimensions:

- Length: 100 mm
- Width: 100 mm
- Central circular hole radius: 5 mm

The laminate stacking sequence is:

$$[+45^\circ/-45^\circ/-45^\circ/+45^\circ]$$

Each ply has a thickness of 0.25 mm, resulting in a total laminate thickness of 1 mm.

Orthotropic material properties are assigned at the ply level consistent with the Python implementation. Identical elastic constants are used in both the Python framework and the commercial solver to ensure consistency.

B. Finite Element Modeling in Commercial Software

For validation purposes, the same geometry and stacking sequence are modeled using layered shell elements (SHELL181) in ANSYS. The mesh density is maintained consistently with the Python implementation to ensure comparable discretization accuracy. Boundary conditions are applied by fully constraining all degrees of freedom along one edge of the plate. A uniformly distributed in-plane tensile load of 1000 N is applied along the opposite edge. All modeling parameters, including laminate properties, thickness definition, and load magnitude, are kept identical in both implementations.

C. Comparison Criteria

The validation focuses on:

- In-plane displacements
- Transverse displacement
- Normal stress components (σxx, σyy, σxy)

Quantitative comparison is performed by extracting peak displacement and stress values from both solutions.

The percentage error is calculated as:

$$\text{Error}(\%) = \frac{|X_{Python} - X_{ANSYS}|}{X_{ANSYS}} \times 100$$

where Xrepresents the displacement or stress quantity under comparison.

V. RESULTS AND DISCUSSION

The displacement and stress distributions obtained from the Python-based framework exhibit consistent spatial patterns when compared with those from the commercial finite element model. The maximum in-plane and transverse displacements occur at locations expected from classical plate behavior under tensile loading, and the stress concentration around the circular cutout is clearly captured in both implementations.

The normal stress components (σxx, σyy, σxy) show similar magnitude and contour distribution trends, particularly in regions of high stress gradient near the hole boundary. The observed deviation, remaining below 5% for stress quantities and below 1% for displacement values, indicates strong numerical consistency between the two approaches.

Minor variations are attributed to differences in internal element formulation details, stress recovery techniques, and numerical integration procedures inherent to different software implementations.

However, these discrepancies remain within acceptable engineering tolerance for small- to medium-scale structural analysis.The results confirm that the developed open-source framework accurately reproduces the structural response of laminated composite plates modeled using a validated commercial shell element formulation. This demonstrates that reliable preliminary structural assessment can be achieved without exclusive dependence on licensed finite element software.

A. *Python FSDT Model data*

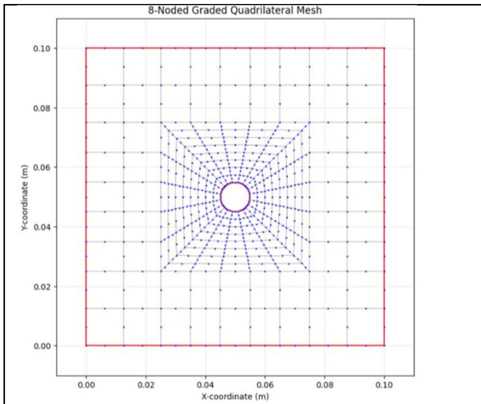


Figure1: Meshing for Plate

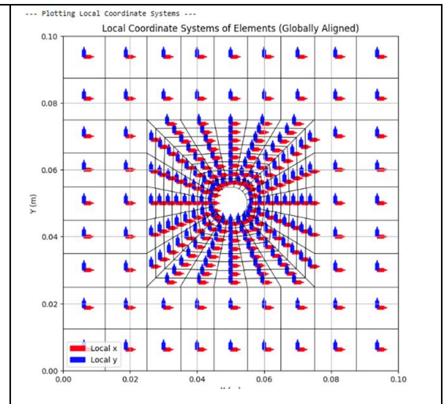


Figure2: Coordinate system

B. *Python FSDT vs Ansys Apdl results data*

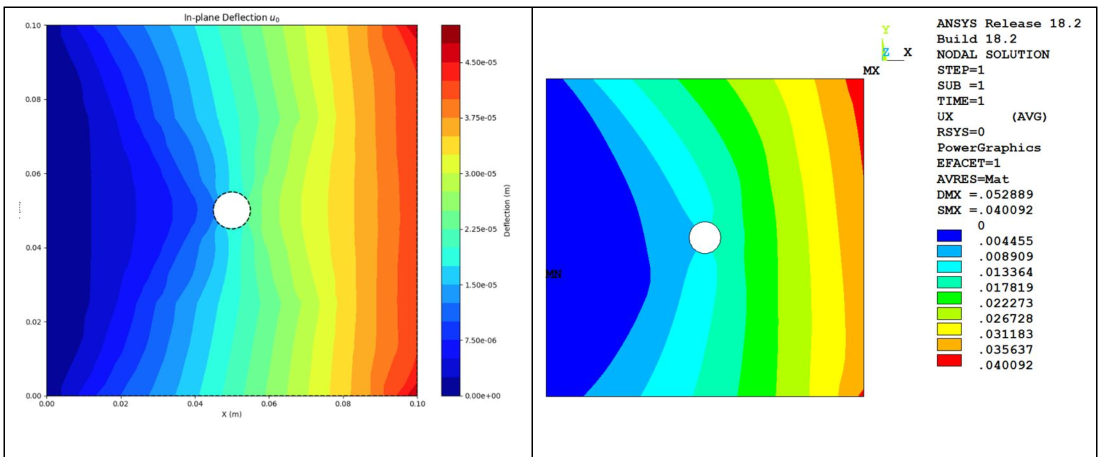


Figure 3: Comparison of X- Displacements for Python FSDT vs Ansys Apdl results data

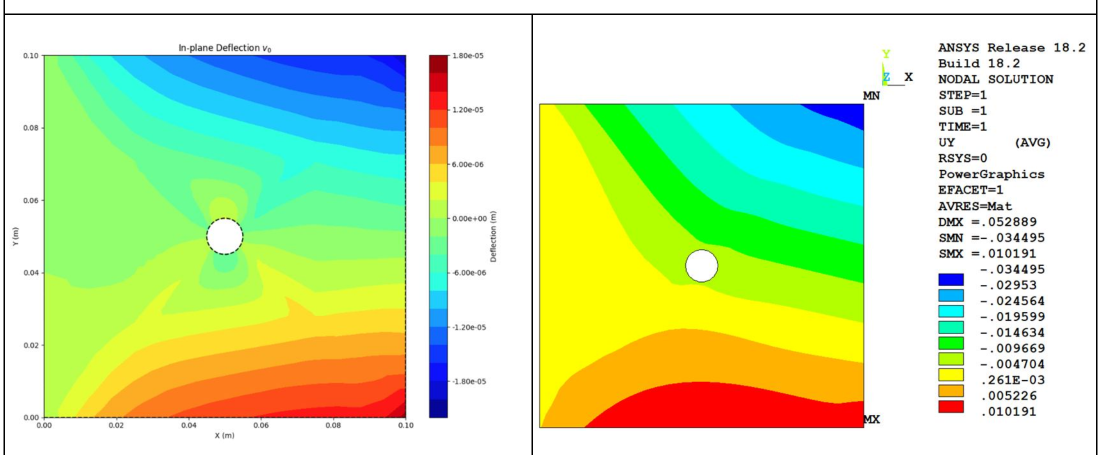


Figure 4: Comparison of Y- Displacements for Python FSDT vs Ansys Apdl results data

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 14 Issue III Mar 2026- Available at www.ijraset.com

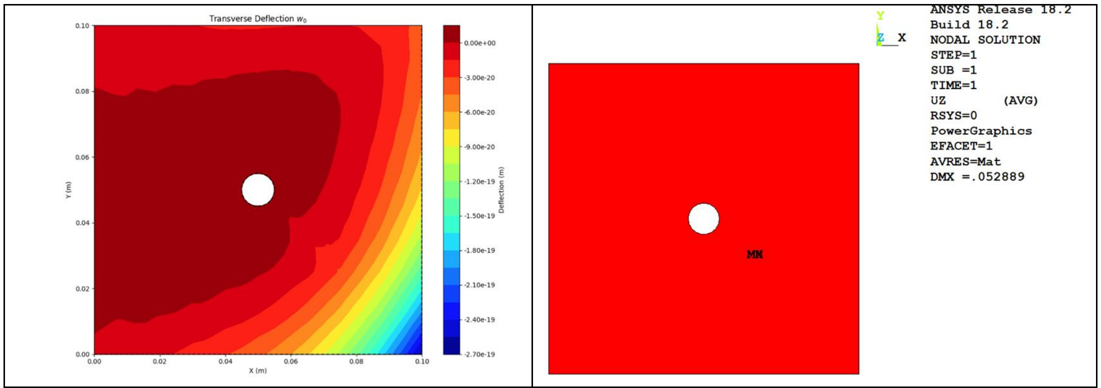


Figure 5: Comparison of Z- Displacements for Python FSDT vs Ansys Apdl results data

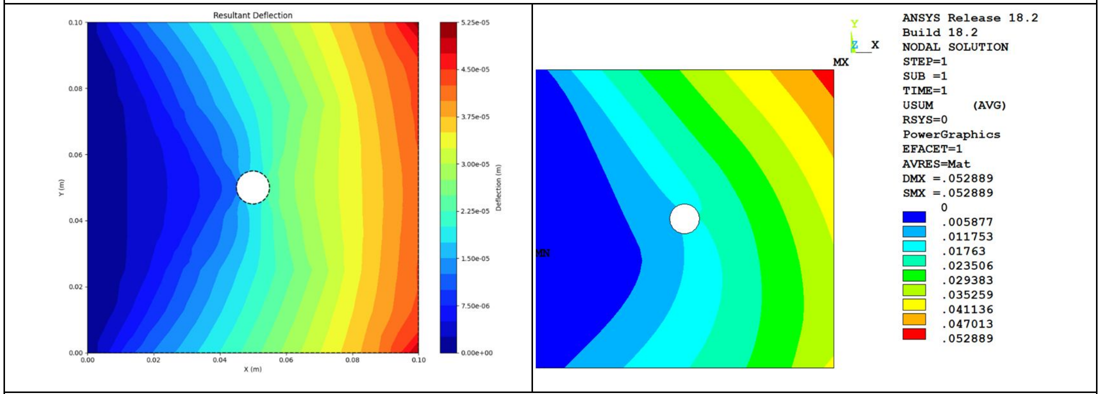


Figure 6: Comparison of USum Displacements for Python FSDT vs Ansys Apdl results data

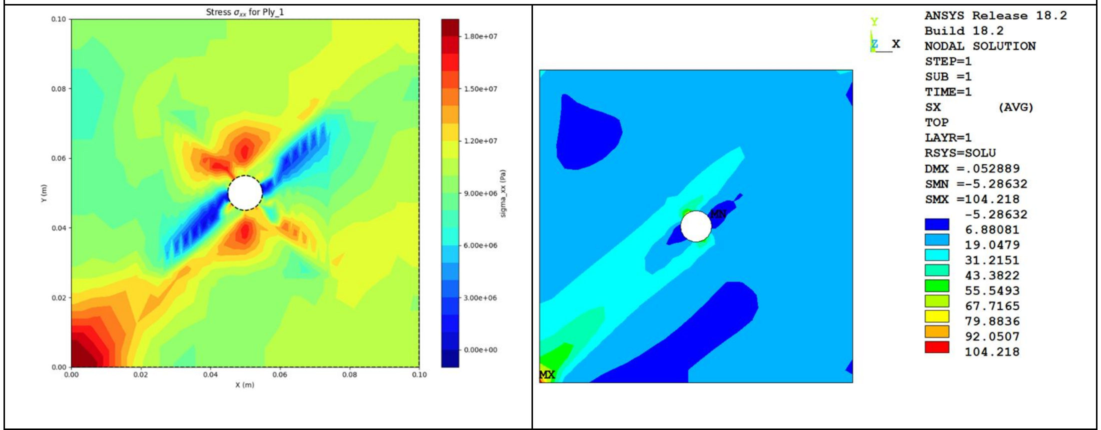


Figure 7: Comparison of Sx Stress for Python FSDT vs Ansys Apdl results data

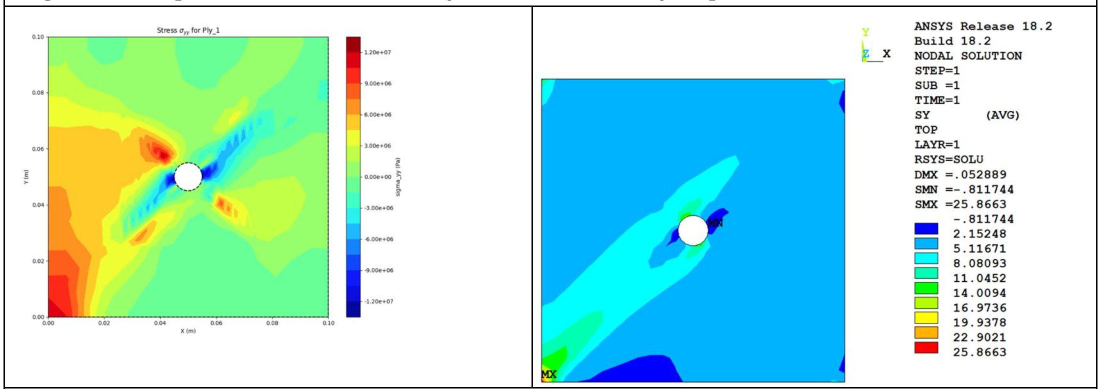


Figure 8: Comparison of Sy Stress for Python FSDT vs Ansys Apdl results data

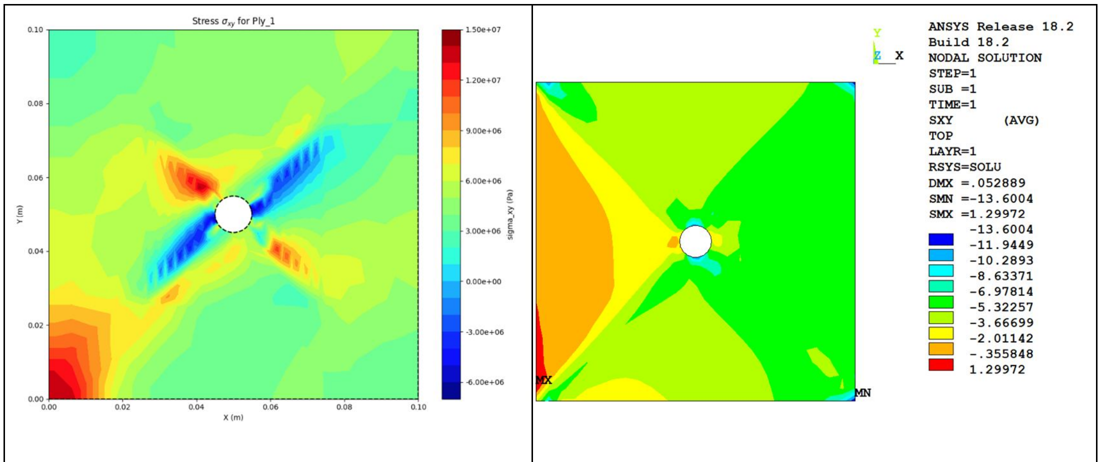


Figure 9: Comparison of Sxy Stress for Python FSDT vs Ansys Apdl results data

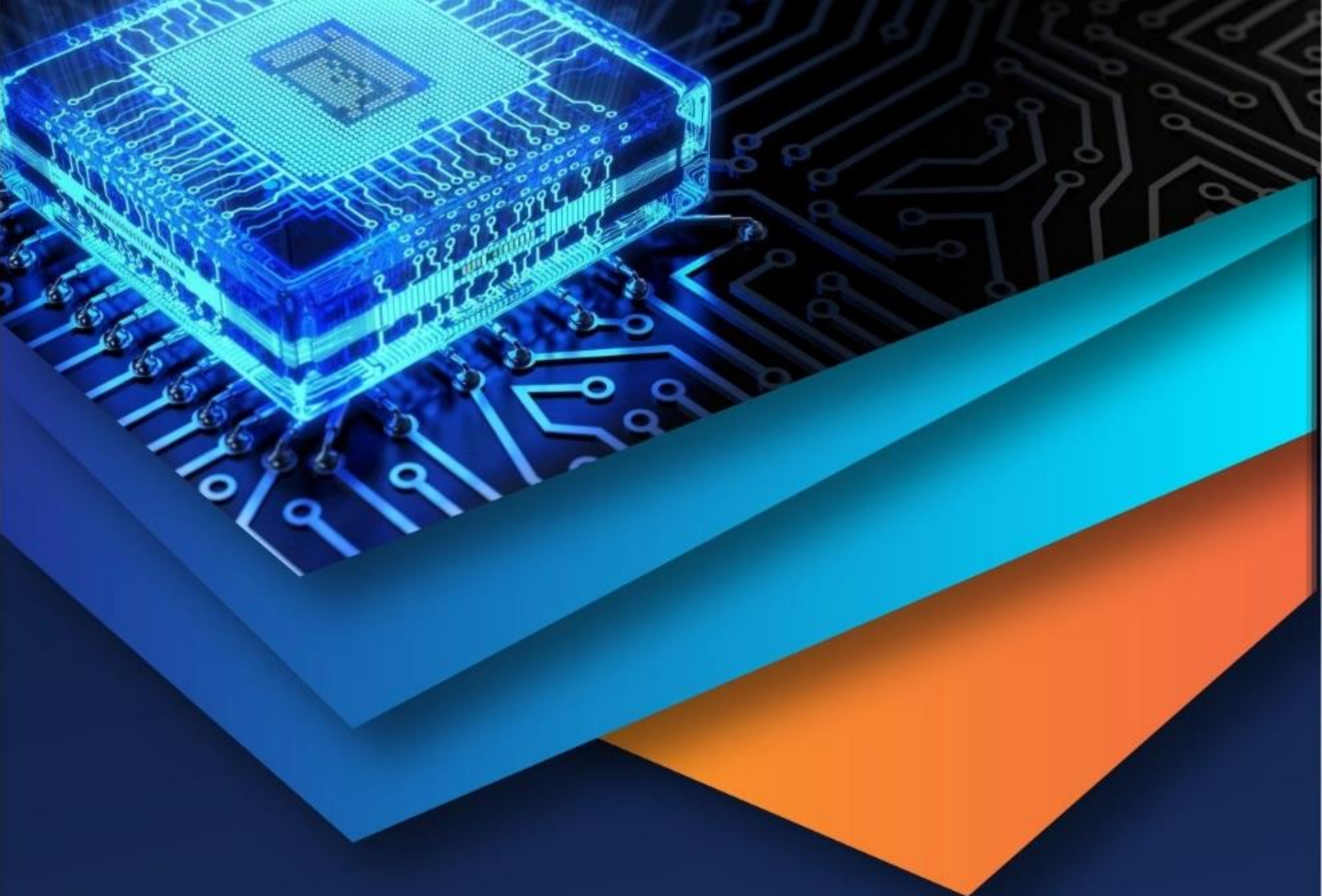
VI. CONCLUSIONS AND FUTURE PLAN

In this study, a Python-based finite element framework for laminated composite plate analysis has been developed and validated using First Order Shear Deformation Theory. The implementation includes laminate constitutive modeling, mesh generation, global stiffness assembly, and ply-level stress recovery within an open computational environment. Validation against a commercial solver shows strong agreement, with displacement deviations below 1% and stress deviations within 5%, confirming the numerical accuracy of the approach.

The framework offers a cost-effective solution for preliminary and medium-scale analyses and serves as a foundation for data-driven modeling. It enables the generation of high-fidelity simulation datasets for training Graph Neural Network (GNN)-based models aimed at approximating finite element responses with significantly reduced computational cost.

REFERENCES

[1] ANSYS Inc. (2018) ANSYS Mechanical APDL element reference – SHELL181. [online] Available at: https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_SHELL181.html
[2] Bathe, K J (1996) Finite element procedures. [online] Available at: https://web.mit.edu/kjb/www/Books/Finite_Element_Procedures_Revised_Edition.pdf
[3] Jones, R M (1999) Mechanics of composite materials. 2nd edn. [online] Available at: https://doi.org/10.1201/9781498711067
[4] Lekhnitskii, S G (1968) Anisotropic plates. [online] Available at: https://archive.org/details/AnisotropicPlates
[5] Reddy, J N (1984) A simple higher-order theory for laminated composite plates. Journal of Applied Mechanics, 51 (4), pp. 745–752. [online] Available at: https://doi.org/10.1115/1.3167719
[6] Reddy, J N (2004) Mechanics of laminated composite plates and shells: theory and analysis. 2nd edn. [online] Available at: https://doi.org/10.1201/b12409
[7] Savin, G N (1961) Stress concentration around holes. [online] Available at: https://doi.org/10.1016/C2013-0-08143-4
[8] Timoshenko, S P and Woinowsky-Krieger, S (1959) Theory of plates and shells. 2nd edn. https://archive.org/details/theoryofplatesan00timo
[9] Whitney, J M (1987) Structural analysis of laminated anisotropic plates. https://doi.org/10.1177/073168448800700108
[10] Zienkiewicz, O C and Taylor, R L (2000) The finite element method, Vol. 2: solid mechanics. 5th edn https://doi.org/10.1016/B978-075066321-2/50000-1



10.22214/IJRASET

INDEX COPERNICUS 45.98

ISRA JIF IMPACT FACTOR: 7.129

SJIFactor IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY