IAENG International Journal of Computer Science

Evaluation and Feasibility of Different Models and Methods for Composite Simulation Using Ansys

Eduardo Pires Bonhin, Sarah David Müzel, Geraldo Cesar Rosario de Oliveira, Walter Luiz Medeiros Tupinambá, Erick Siqueira Guidi, Carlos Alexis Alvarado Silva, Fernando de Azevedo Silva

Abstract— **The utilization of composite materials in various engineering sectors has gained significant prominence due to their unique characteristics. However, owing to their inherent heterogeneity, these materials often exhibit nonlinear and unpredictable behaviors. Consequently, the finite element method has seen a growing application as an invaluable tool for analyzing composites subjected to diverse scenarios. This study aims to assess the advantages and disadvantages of ANSYS APDL and Workbench modules (specifically, ACP and Static Structural) while also examining the impact of the choice of elements in simulating composite materials. The results obtained reveal that, irrespective of the chosen method and element type, the strain patterns exhibited remarkable similarity. Nonetheless, models employing shell elements demonstrated a notable advantage, requiring fewer elements and nodes. Furthermore, the recommended model is the integrated ACP model. This preference is based in its capacity to simplify layer modeling and enable the detailed analysis of strains within each layer.**

Index Terms — **Carbon Fibers, Finite Element, Materials, Polyphenylene Sulfide Composite.**

I. INTRODUCTION

Advancements in technology have spurred a growing demand for materials with extraordinary properties, surpassing the capabilities of traditional materials like metal alloys, ceramics, and polymers [1]-[3]. Composites offer a unique combination of performance attributes that are unattainable by their individual constituents [4]-[5]. These combinations enable the creation of lightweight components with remarkable strength and stiffness, along with added advantages such as resistance to high temperatures, corrosion, and impact [2], [6], [7]. This versatility makes composites more appealing than single-material alternatives [2], [6], [7]. The applications of composites span a wide array of industries, including automotive, aerospace, petrochemical, naval, electro-electronics, civil construction, energy, biomedical, and sports, among others [2], [3], [5]-[9]. Despite their extensive possibility use, composite materials pose challenges due to their cost, inherent heterogeneity, and the influence of lamination configurations on their properties. Analytical models seek to represent the various failure phenomena that impact composite performance [10] [11]-[14]. In response to these challenges and with the aim of optimizing structural designs and gaining a deeper understanding of composite behavior, numerous researchers turn to computer simulations [15], [16]. Utilizing the Finite Element Method (FEM), it becomes feasible to analyze the damage inflicted upon the matrix, fibers, and the matrix/fiber interface when composites are subjected to a diverse range of conditions, including static and dynamic loading, exposure to varying temperatures, and different pressures [17], [18].

II. COMPOSITE MODELING

In the simulation of the composite materials, the effectiveness of modeling relies on the choice of elements, with four primary types at the forefront: beam, solid, plate, and shell elements. This selection is inherently tied to the nature of composite layers, as outlined in [19]. Among these options, the beam element sees relatively limited use. According to [20], plate and shell elements are the predominant choices for modeling composite structures. The rationale behind this preference, as explained by the author, stems from their capacity to reduce the number of nodes and elements when compared to solid elements, along with their ease in modeling thick laminates. Both plate and shell elements fall under the category of two-dimensional or surface elements, given that two of their dimensions (length and width) significantly overshadow the thickness, which is determined by the laminate layers [12]-[14],[17]-[26]. Consequently, the terms "plate" and "shell" are often used interchangeably, with the plate element considered flat and the shell element curved in some contexts [17], [21], [27]. According to [20], the thickness coordinate is omitted from the general equation, effectively transforming it from a 3D problem into a 2D one. However, [19] points out that while it is possible to use 3D solid elements in composite simulation, it is generally impractical due to the high computational cost, especially for models with numerous layers or those representing real-world structures. Moreover, solid elements can lead to ill-conditioned equations when dealing with very thin laminate thicknesses. Beyond the element type, the modeling approach significantly influences the efficiency of the model and, consequently, the accuracy of the results. Presently, there are various software programs for composite simulation, each equipped with modules and routines for laminate modeling. Some of these modules simplify the manipulation of variables crucial for

Manuscript received May 9, 2023; revised April 23, 2024.

This work was supported in part by "Coordenação de Aperfeiçoamento de Pessoal de Nível Superior – Brasil (CAPES)".

Eduardo Pires Bonhin is a PhD of Universidade Estadual Paulista, Guaratinguetá, São Paulo, Brazil. (e-mail: eduardo.bonhin@unesp.br).

Sarah David Müzel is a PhD of Universidade Estadual Paulista, Guaratinguetá, São Paulo, Brazil. (e-mail: sarah.muzel@unesp.br).

Geraldo Cesar Rosario de Oliveira is a PhD candidate of Universidade Estadual Paulista, Guaratinguetá, São Paulo, Brazil. (e-mail: geraldo.cesar@unesp.br).

Walter Luiz Medeiros Tupinambá is a Professor of Technical Course in Electromechanics at Etec - Professor Alfredo de Barros Santos - CEETEPS, Guaratinguetá, São Paulo, Brazil. (e-mail: walter.tupinamba@etec.sp.gov.br).

Erick Siqueira Guidi is a professor of Mechanical Engineering Department of Universidade Estadual Paulista, Guaratinguetá, São Paulo, Brazil. (e-mail: erick.s.guidi@unesp.br).

Carlos Alexis Alvarado Silva is a professor of Mechanical Engineering Department of Technological University of Peru, Chiclayo, Perú. (e-mail: c21629@utp.edu.pe).

Fernando de Azevedo Silva is a professor of Mechanical Engineering Department of Universidade Estadual Paulista, Guaratinguetá, São Paulo, Brazil. (e-mail: fernando.azevedo@unesp.br).

composite analysis, such as the ACP module (ANSYS Composite PrepPost) by ANSYS Inc. This module streamlines the engineering of layered composites, encompassing complex definitions that involve multiple layers, materials, thicknesses, and orientations. It addresses the challenge of predicting real-world performance under various conditions, considering stresses, strains, and an array of failure criteria [28]-[37]. However, the most widely adopted module within ANSYS for composite simulation remains APDL (ANSYS Parametric Design Language). While APDL demands a deeper understanding of simulation concepts, it is more versatile than the Workbench interface, where ACP is integrated. The Workbench interface, owing to its materials database and automatic element type selection based on problem analysis, offers a more intuitive and user-friendly experience [29],[34]. This study conducts a comparative analysis between the APDL and Workbench modules of ANSYS for simulation of the composite material. The objectives are to assess ease of modeling, compare results between simulations and experiments, and evaluate the impact of element type combined with the module used on the results.

III. MATERIAL AND METHODS

A. Material Selection and Specimen Preparation

In this experimental investigation, a Polyphenylene Sulfide (PPS) composite reinforced with carbon fiber (5HS) consisting of five layers, each 0.32 mm thick, was employed. The classification of PPS-CF would be a thermoplastic matrix composite reinforced with carbon fibers. This describes the nature of the constituent materials, with the thermoplastic polymer matrix (PPS) and carbon fibers as reinforcement, as well as the shape of the constituents, with the melted matrix and carbon fibers embedded in fabric form.

The composite was created using the Compression Molding method where it was placed in a mold, subjected to controlled pressure and temperature to form the final piece.

In the PPS-CF architecture, Bidirectional Lamination was used, which is proposed when resistance is required both in the direction of the fibers and perpendicular to them, the carbon fibers can be arranged in a bidirectional configuration, generally at right angles (0° and 90°), forming a biaxial fabric pattern. The material properties are outlined in Table 1.

TABLE I
MATERIAL PROPERTIES

Ortotropic Stress Limits	
Tensile, 11	790 [MPa]
Tensile, 22=33	750 [MPa]
Compressive, 11	-644 [MPa]
Compressive, 22=33	-637 [MPa]
Shear, 12=13=23	131 [MPa]
Ortotropic Elasticity	
Young'sModulus, E11=E22=E33	40 [GPa]
Poisson'sRatio, ν12= ν13= ν23	0.25
Shear Modulus, G12=G13=G23	2,65[GPa]

Specimens were crafted from this material, measuring 216 x 39.2 mm in length and width, and featuring a centrally located 8 mm diameter hole. These specimens were subjected to tensile testing utilizing a Shimadzu Model AG-X universal testing machine equipped with a 50 kN load cell. Strain measurements were conducted using strain gauges positioned at the end of the hole and at a 33 mm distance, as illustrated in Fig. 1.

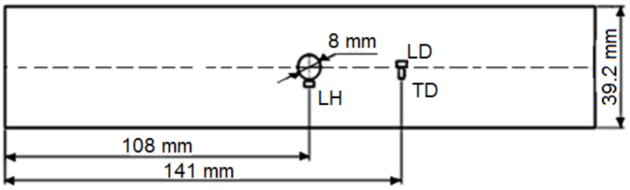


Fig. 1: Specimen and location of strain gauges, LD - longitudinal away from the hole, TD - transverse away from the hole and LH-longitudinal next from the hole.

B. Numerical Simulation

Numerical simulations were conducted employing the finite element method within the ANSYS software. The study involved a comparison between the APDL and Workbench platforms. Furthermore, within the Workbench module, a comparison was made between the Static Structural and ACP packages, which are specialized for composite materials. Regardless of the chosen method, the specimen was divided into three distinct regions, as represented in Fig. 2, aiming to replicate conditions inside the test machine's claw. In these regions, 8 kN loads were applied to each face, and strain was evaluated in the central region.

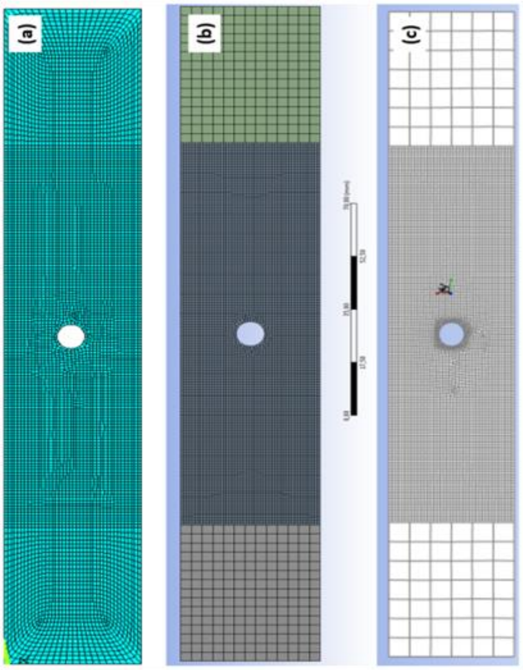


Fig. 2: Models, (a) APDL, (b) Static Structural and (c) ACP.

C. Modeling Approaches

The parameters used in the three modeling approaches were:

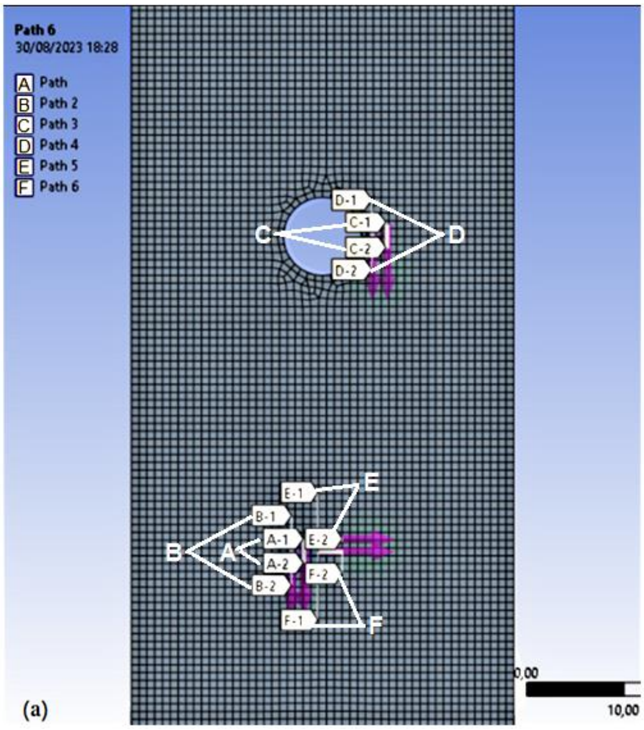
- APDL Module (Fig. 2a): The model was constructed as a two-dimensional figure using the SHELL 190 3D element, featuring 4 nodes and 6 degrees of freedom. This element is suitable for shell models, and the number and thickness of layers were specified using the multilayer tool [29,34].

- Static Structural Package (Fig. 2b): In this approach, the specimen was modeled as a homogeneous structure using elements SOLID186, CONTA174, and SURF154 for meshing. Bonded type contact was employed in the contact zones.
- ACP Package (Fig. 2c): Two models were created within the ACP package: one with solid elements and another with shell elements. Both models were generated from surface sketches, with material thickness defined by modeling five layers, each 0.32 mm thick, using woven fiber and PPS as the polymer matrix. SOLID185 and SURF154 elements were used for the solid model mesh, while SHELL 181 and SURF156 elements were used for the shell model mesh.

D. Strain Quantification

To quantify the strain in regions corresponding to the strain gauge locations:

- In the APDL module, strains were selected based on node loading variation in regions where strain gauges were placed.
- In models created in the Workbench, the Path feature was used to isolate strains in defined regions. Six Paths were constructed, two for each strain gauge, representing their lower and upper limits, as depicted in Fig. 3.



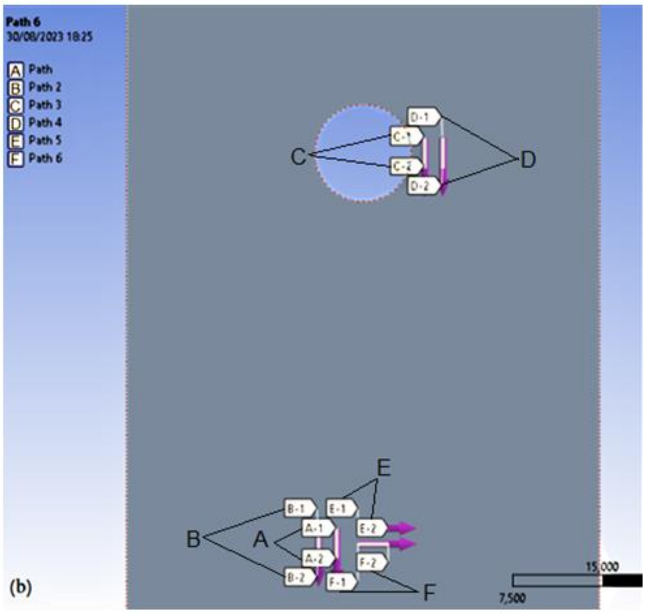


Fig.3: Path Distribution (a) Static Structural and (b) ACP.

IV. ANALYSIS AND DISCUSSION OF RESULTS

Analyzing the specimens subjected to experimental testing, a clear observation emerges: the fracture predominantly manifests in the vicinity surrounding the hole, exhibiting an X-profile, as depicted in Fig. 4. A consistent pattern is discernible across all simulation models when comparing this fractured region with the experimental findings, as illustrated in Fig. 5. Notably, the region of maximum strain aligns with expectations, closely situated near the hole, with the recorded maximum strain value approaching approximately 0.029 m/m.

Fig.4: Specimen after test.

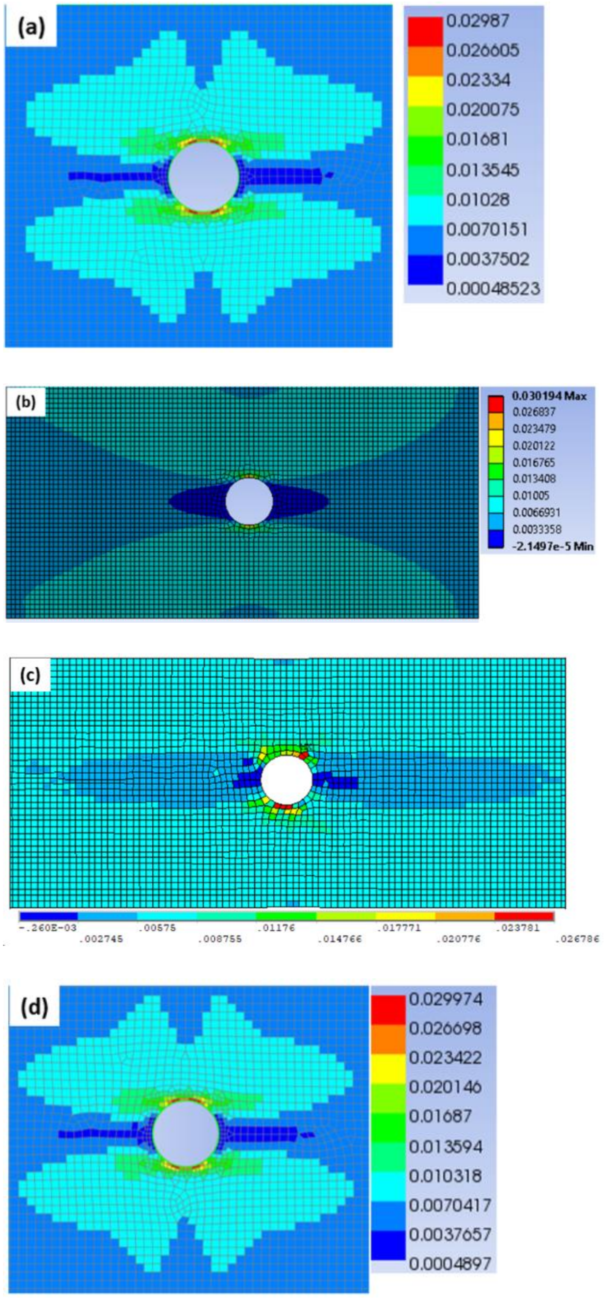


Fig.5: Normal Elastic Strain (a) Solid ACP model, first layer; (b) Static Structural Model; (c) APDL Model and (d) Shell ACP model, first layer.

The models crafted within the ACP package offer a distinctive advantage by facilitating the visualization of strain in each layer, as exemplified in Fig. 6. This capability extends to both the solid element model and the shell element model. Analyzing the strain results presented in Fig. 7, a convergence between experimental and simulated values becomes evident. As anticipated, the highest strain values occur near the hole due to elevated stress concentration in that region. In the transverse region, negative strain values are recorded, attributed to the compressive force application. A meticulous comparison of experimental and simulated values indicates nearly zero difference in the longitudinal direction near the hole (Fig. 7a). However, in the longitudinal direction away from the hole, a disparity of approximately 9% is observed (Fig. 7b), along with a 10% difference in the transverse direction (Fig. 7c). This underscores the validity of simulations, irrespective of the module used. Furthermore, an overlap ofvalues in the simulation results accentuates precision among the employed methods.

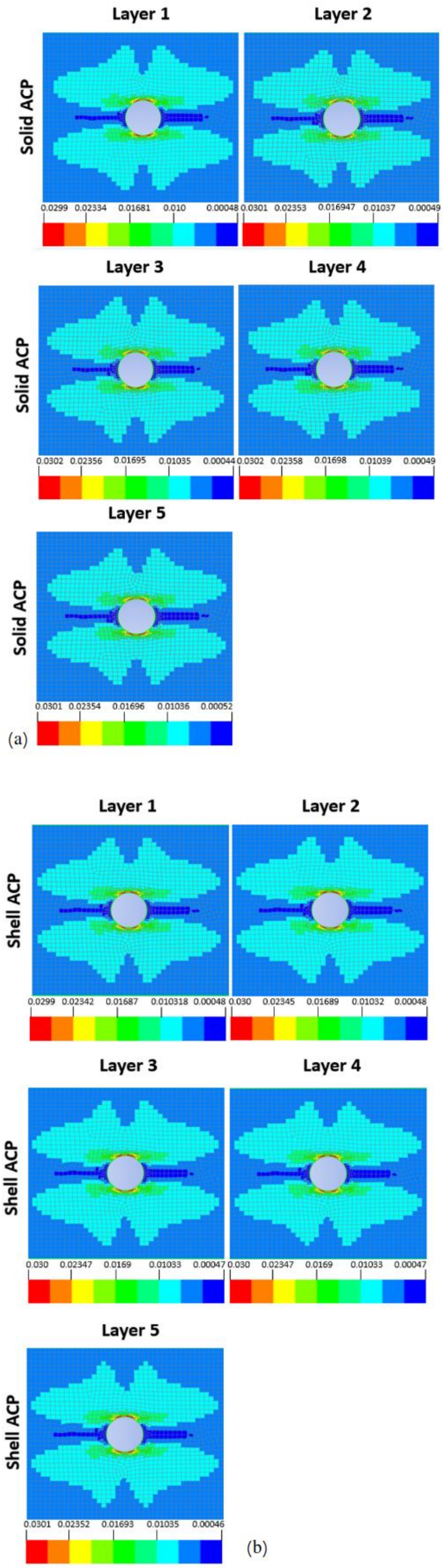
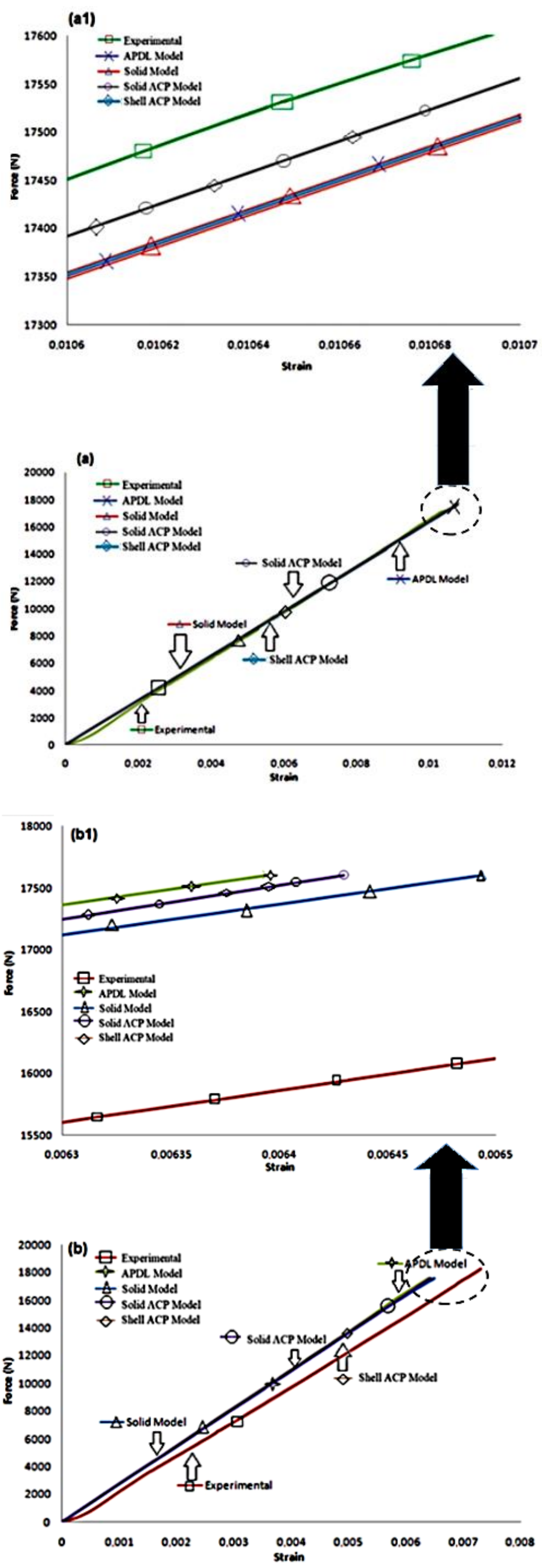


Fig.6: Normal Elastic Strain in each layer, of the ACP model: (a) Solid ACP model, (b) Shell ACP model.



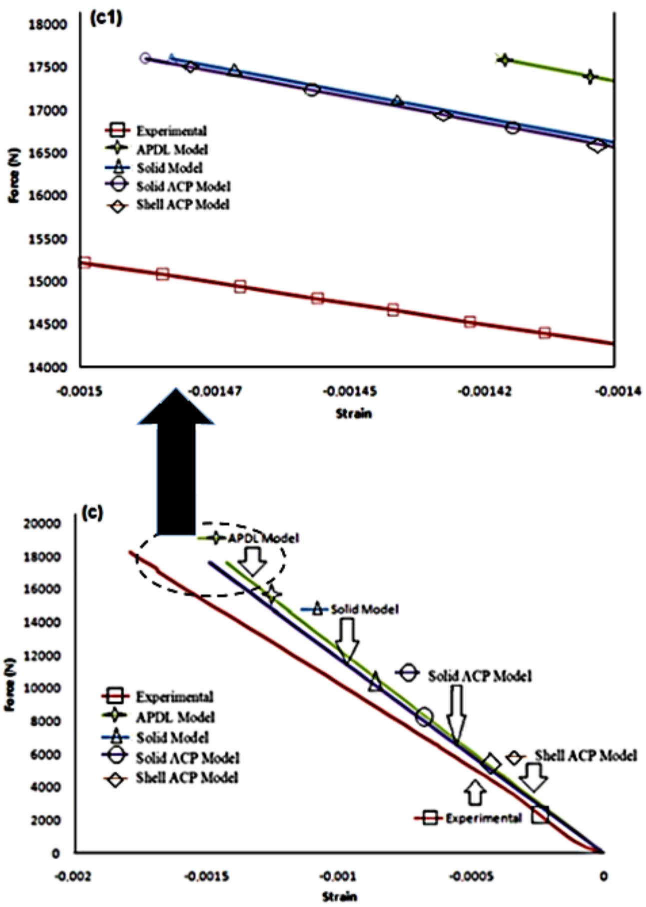


Fig.7: Force versus strain graphs (a) Longitudinal, near the hole region,(a1) Extended region, (b) Longitudinal, (b1) Extended region and (c) Transverse, away from the hole, (c1) Extended region.

Although the results are close to the experimental data, regardless of the module used, a more detailed analysis of the number of nodes and elements used in each model shows differences. Models that use solid elements required a greater number of nodes and elements compared to those built with shell elements, as illustrated in Figure 8.

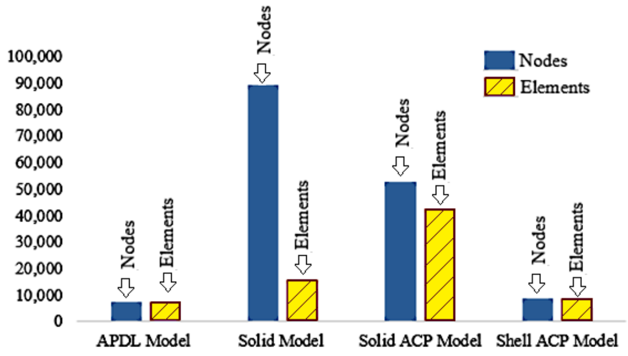


Fig.8: Comparison of the number of elements and nodes among the models.

Still according to the data presented in Figure 8, it is observed that the mesh generated in the solid model in Workbench (without using the ACP wizard) generated almost 90,00 nodes, while in the model using APDL, just over 5,000 nodes were generated, resulting in a greater demand for computational resources in the solid models. Notably, the APDL module used the fewest number of elements and nodes. However, it is essential to emphasize that the models formulated within the ACP package readily allowed the analysis of layered results, proving to be a valuable resource for simulations involving composite materials. From the point of view of using computational resources, the model made using APDL requires less computational resources, as it has a simpler graphical interface and was the model that generated the smallest number of nodes and elements.

V. CONCLUSION

In summarizing the findings derived from the comparison of the four distinct models, it is apparent that the simulation outcomes closely mirror the experimental data. The investigation has not only illuminated the merits and drawbacks inherent in each modeling approach but has also contributed valuable insights to the scientific community. The solid model, crafted within the Static Structural package, stands out for its expeditious construction, albeit accompanied by certain drawbacks. Notably, this model engenders a substantial number of nodes and elements, necessitating significant computational resources. Furthermore, its limitation lies in its capacity to furnish results solely for the entire laminate, lacking the capability to visualize layer-specific outcomes such as strain, stress distribution, and failure criteria—features seamlessly achievable in the ACP package. The APDL model, leveraging shell elements, presents a compelling alternative. While demanding fewer nodes and elements, thus minimizing computational demands, its modeling process proves more intricate due to a less user-friendly interface compared to Workbench. Models formulated through the ACP package, specifically tailored for composite materials, offer a comprehensive array of functionalities for both model creation and result evaluation. Nevertheless, this package demands a higher level of expertise owing to its intricate links between simulation parameters. Comparing the ACP models with the APDL model, albeit featuring slightly more nodes and elements than the latter, and considerably fewer than the solid model, underscores their efficacy. In essence, the choice of modeling approach hinges on a myriad of factors. The solid model excels in speed and finds effectiveness in dealing with simpler geometries and boundary conditions. Conversely, for intricate models, particularly those boasting complex layer configurations, the ACP and APDL models prove more adept. Among these, the ACP package, offering a balance of user-friendliness and extensive functionality, is preferable, while the APDL module, though more laborious, shines in computational efficiency. Ultimately, the selection of the modeling approach should be a judicious decision based on the specific simulation requirements, model complexity, and the expertise of the designer.

REFERENCES

[1] Berthelot, J.M., “Composite Materials: Mechanical Behavior and Structural Analysis”, 1 ed., New York: Springer Science & Business Media, 2012.

[2] Katsoulis, C., Kandola, B. K., Myler, P., & Kandare, E, “Post-fire flexural performance of epoxy-nanocomposite matrix glass fibre composites containing conventional flame retardants”, Composites Part A: Applied Science and Manufacturing., Elsevier Ltd, vol. 43(8), pp. 1389–1399., 2012. doi:10.1016/j.compositesa.2012.03.009.

[3] Moura, M. F. S. F., Morais, A. B. and Magalhães, A. G., “Materiais compósitos: materiais, fabrico e comportamento mecânico”,2 ed., Porto: Publindústria. 2005.

[4] Barbero, E. J., “Finite Element Analysis of Composite Materials Using Ansys”, 2 ed., CRC Press,2014.

[5] Callister Junior, W. D. and Rethwisch, D. G. “Materials Science and Engineering: An Introduction”, 8 ed., LTC,2013.

[6] Guidi, E. S., de Azevedo Silva, F.,“Study of Stress Concentration in Polymeric Composites Using Finite Element Method” (No. 2010-36-0196). SAE Technical Paper,2010.

[7] Soriano, H. L. “Método de Elementos Finitos em Análise de Estruturas”, EdUSP, 2003.

[8] Dandekar, C. R. and Shin, Y. C.,“Modeling of machining of composite materials: A review”, International Journal of Machine Tools and Manufacture. Elsevier, vol. 57, pp. 102–121,2012.doi: 10.1016/j.ijmachtools.2012.01.006.

[9] Matthews, F.L., Davies, G.A.O., Hitchings, D. and Soutis, C., “Finite Element Modeling of Composite Materials and Structures”, 2 ed, Woodhead Publishing Limited, 2003

[10] Ajeje, F. H., Penna, S. S. and Pitangueira, R. L., “Elementos finitos de casca do sistema computacional INSANE”, Revista Escola de MInas, vol. 64(4), pp. 399–405,2011

[11] Cook, R. D.,”Finite Element Modeling for Stress Analysis”, 1 ed., John Wiley & Sons, Inc,1995

[12] Vaz, L. E., “Método dos elementos finitos em análise de estruturas”, Elsevier, 2011. Available at: https://scholar.google.es/scholar?q=metodo+dos+elementos+finitos+luiz+eloy&btnG=&hl=en&as_sdt=0%2C5#0.

[13] Wan, Y., Diao, C., Yang, B., Zhang, L., & Chen, S., “GF/epoxy laminates embedded with wire nets: A way to improve the low-velocity impact resistance and energy absorption ability”, Composite Structures, vol. 202, pp 818-835,2018

[14] Wang, H., Sun, J., Li, J. and Li, W.,“Investigation on delamination morphology during drilling composite laminates”, International Journal of Advanced Manufacturing Technology, vol. 74(1–4), pp. 257–266, , 2014 doi: 10.1007/s00170-014-5973-6

[15] Gondaliya, R., Sypeck, D. and Zhu, F., “Improving Damage Tolerance of Composite Sandwich Structure Subjected to Low Velocity Impact Loading: Experimental Analysis”, Proceedings of the American Society for Composites, Thirty-First Technical Conference, (October),2016

[16] Guidi, E. S., & de Azevedo Silva, F., “Analysis of mechanical behavior in bending of polymer composites using the finite elements method” (No. 2009-36-0244), SAE Technical Paper, 2009

[17] Chung, D. D. L., “Composite Materials”, 2 ed., London: Springer London (Engineering Materials and Processes),2010 doi: 10.1007/978-1-84882-831-5.

[18] Sorrentino, L., Turchetta, S. and Bellini, C., “A new method to reduce delaminations during drilling of FRP laminates by feed rate control”, Composite Structures, Elsevier, vol. 186 (September 2017), pp. 154–164,2018. doi: 10.1016/j.compstruct.2017.12.005.

[19] Boumbimba, R.M., Froustey, C., Viot, P., Olive, J.M., Léonardi, F., Gerard, P. and Inoubli, R., “Preparation and mechanical characterization of laminate composites made of glass fibre/epoxy resin filled with tri bloc copolymers”, Composite Structures, Elsevier Ltd, vol. 116(1), pp. 414–422,2014 doi: 10.1016/j.compstruct.2014.05.028.

[20] ANSYS Inc., “ANSYS”, 2017

[21] R., & Deepak, C. C. C., “Design and Analysis of Asymmetrical Spur Gear Drive for Automobile Gear Box Application”, Applied Mechanics and Materials, 592–594, 2277–2281, 2014. https://doi.org/10.4028/www.scientific.net/amm.592-594.2277

[22] Barbero, E. J., “Finite Element Analysis of Composite Materials”, 1ed, CRC Press, 2008

[23] Rezende, M. C., Costa, M. L. and Botelho, E. C., “Compósitos Estruturais: Tecnologia e Prática”. 1 ed, Artliber, 2011

[24] Tenek, L. and Argyris, J., “Finite Element Analysis for Composite Structures”, Edited by G. M. Gladwell, Kluwer Academic Publishers, 1998 doi: 10.1007/978-94-015-9044-0.

[25] Timoshenko, S. and Woinowsky-Krieger, S., “Thoery of Plates and Shells”, 2 ed, Mc Graw-Hill, 1959

[26] Ugural, A. C., “Stresses in Plates and Shells”, McGraw Hill,1981

[27] Adams, V. and Askenazi, A., “Building Better Products with Finit Elements Analysis”, 1 ed, Onword Press, 1999

[28] Alvarado-Silva, C. A., Rosario de Oliveira, G. C., Gamarra-Rosado, V. O., & de Azevedo Silva, F., “A Polynomial Equation Model for Fatigue Crack Propagation in an Aeronautical Steel Material”, In Materials Science Forum, Vol. 1053, pp. 212-217,Trans Tech Publications Ltd.,2022

[29] Yang, B., Fu, K., & Li, Y., “Modelling damage evolution and CAI strength of composite laminates under biaxial compression”, Journal of Composite Materials, vol. 55(21), pp 2887-2898,2021

[30] Putini, E. P. G., & Silva, F. D. A., “Fatigue life estimation in a hydrogenerator rotor with cracks using the finite element method”, Journal of the Brazilian Society of Mechanical Sciences and Engineering, vol. 40(9), pp 1-15,2018

[31] A. M. Zakhezin, Y. G. Pryadko, O. P. Kolosova, S. V. Aliukov, “Diagnostics of fatigue cracks in rotor systems of machines,” Engineering Letters, vol. 28, no. 1, pp. 209-214, 2020.

[32] Song, J., Zhou, J. T., Jiang, R., Wang, M. B., Yu, X., Jia, S. L., & Chen, F., “Fatigue Crack Propagation in BFRP Reinforced RC Beams Based on Fracture Mechanics and Golden Ratio”, Engineering Letters, vol. 24, no.4, pp. 449-454, 2016.

[33] Murat Ali and Ken Mao,"Contact analysis of hip resurfacing devices under normal and edge loading conditions", Engineering Letters, 20:4, pp317-329, 2012

[34] Marques, B. M. F., Marques, T. P. Z., Silva, F. D. A., Cândido, G. M., & Rezende, M. C. “Failure criteria assessment of carbon/epoxy laminate under tensile loads using finite element method: Validation with experimental tests and fractographic analysis”, Mechanics of Advanced Materials and Structures, pp 1-10,2022

[35] Para, K., Sungnul, S., Sirisubtawee, S., &Phongthanapanich, S., “A Comparison of Numerical Solutions for Advection-Diffusion-Reaction Equations between Finite Volume and Finite Difference Methods”, Engineering Letters, vol. 30, no. 2, pp. 566–581, 2022.

[36] Chunyuan Li, "Numerical Simulation Analysis of Non-Contact Mechanical Seal Performance based on Dynamic Grid Technology," IAENG International Journal of Applied Mathematics, vol. 53, no.1, pp. 301-307, 2023

[37] Arouna Ouedraogo and Wendlassida Basile Yameogo, "Finite Volume Scheme and Renormalized Solutions for an Elliptic Operator with Discontinuous Matrix Diffusion Coefficients," IAENG International Journal of Applied Mathematics, vol. 53, no.1, pp. 308-316, 2023