Pre-processing for isogeometric analysis of laminated composities

Murilo Almeida Oliveira[1], Elias Saraiva Barroso [1]

[1]*Laboratório de Mecânica Computacional e Visualização, Departamento de Engenharia Estrutural e Construção Civil, Universidade Federal do Ceará*
Ac. Público, 728 - Pici, 60020-181, Ceará, Fortaleza
muriloao26@alu.ufc.br, elias.barroso@ufc.br

Abstract. Composites are materials made by combining multiple components to create structures that resist heavy loads and environmental conditions. Laminated shells, constructed from layers of fiber-reinforced composites, are analyzed primarily using the Finite Element Method (FEM), which approximates displacements and geometry through polynomial functions. A newer approach, Isogeometric Analysis (IGA), uses rational tasks. This work focuses on applying NURBS surfaces within IGA for laminated structures, examining both theoretical and practical aspects with a focus on the pre-processing phase. The FEMEP (Finite Element Method Educational Computer Program) software, developed in Python, is extended for modeling composite structures, with structural analysis handled by the FAST program. Examples with laminated plates demonstrate the effectiveness of the proposed pre-processing tool.

Keywords: Composites, Isogeometric Analysis, Simulation Pre-processing

1 Introduction

Laminated composites are materials made up of several layers of different materials joined together to form a single material with specific properties. These materials are used in various industries, such as automotive, aeronautics, construction, renewable energy, sports, electronics, medical, and maritime, due to their superior mechanical properties, such as strength-to-weight and stiffness-to-weight ratios, and corrosion resistance.

Due to the wide variety of characteristics of a laminated structure, the structural analysis of laminated structures becomes more complex compared to conventional materials. In this context, it is common to use numerical methods to solve the mechanical responses of the structure. The Finite Element Method is the most widely used alternative today.

CAE (Computer Aided Engineering) systems are advanced software built to solve engineering problems. The system involves 3 stages: pre-processing, processing, and post-processing. Initially, pre-processing is a phase of the system that deals with the computer modeling of the physical problem, where a mesh of the structure is generated, in addition to the inclusion of the necessary attributes, such as the type of material, thickness, support conditions, loads (which can be point or distributed), and also temperature and heat flow in the case of heat transfer problems. With all the attributes added, the analysis model can be built, and processed, evaluating the responses of the physical problem under study.

In the processing phase, isogeometric analysis (IGA), an advanced method of numerical analysis, is used. In IGA, both geometry and the approximated fields use the same mathematical functions, for instance, the Non-Uniform Rational B-Splines (NURBS). Keeping the geometry accurate during the analysis, eliminates errors in approximating the geometry of the model since rational functions are used in the meshing of the numerical model. Consequently, IGA is expected to achieve faster accuracy and convergence.

Finally, in the post-processing phase, data analysis is carried out using scientific visualization software or libraries.

This work aims to develop a pre-processing tool for the structural analysis of laminated composites. The work is developed by expanding the functionalities of an existing isogeometric analysis pre-processing software, FEMEP, developed at TECGRAF (PUC/RJ) [2]. In addition to allowing the insertion of attributes related to the laminated composites problem, a routine was also developed for writing the analysis model in the format of the FAST software, developed at the Computational Mechanics and Visualization Laboratory at the Federal University of Ceará.

The use of the system is demonstrated in an example of a composite plate with a hole, subjected to a tensile force.

2 Software FEMEP

FEMEP (Finite Element Method Educational Computer Program) is a computer program for modeling solids in two dimensions, designed specifically for creating models using isoparametric finite elements. Developed in Python, FEMEP adopts the object-oriented programming (OOP) paradigm. The user interface is built with the Qt framework, providing an interactive and visual experience for users. In addition, the software uses the OpenGL library to graphically render the model, allowing visual elements to be generated on-screen efficiently and dynamically. The models generated by FEMEP are analyzed using a tool called FEMOOLab, implemented in MATLAB under the POO architecture.

The FEMEP architecture is implemented in Python and uses JSON format. The HETOOL library provides a platform for creating two-dimensional models based on subdivisions of planes. One of the main focuses of this library is attribute management, offering a wide variety of representations for mechanical problems. With HETOOL, one can specify attributes such as loads along the edges, material properties, and support conditions. This offers great flexibility to model and analyze a variety of mechanical scenarios. The project follows the standard MVC (Model - View - Controller) model. This software architecture divides the application into three main layers: the model layer, which handles data storage; the controller, which receives user requests and processes this information in the model; and the view layer, which retrieves the data from the model to present it to the user. The Figure – shows the FEMEP interface.

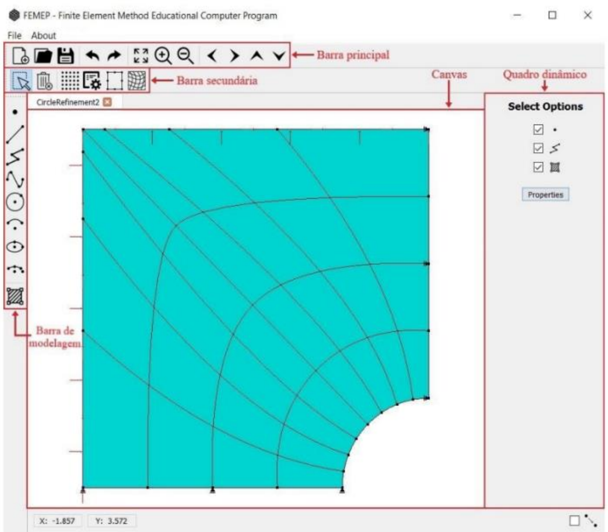

Figure 1. FEMEP's user interface

3 Pre-processing of composites

Once the geometry modeling of a problem has been completed, the user must access attribute management to configure the material properties of the structure and define the boundary conditions of the problem. A more detailed description of attribute management in the original version of FEMEP is provided in the literature [6].

Previously, in FEMEP, the attributes defined are Material, Thickness, Support Conditions, Concentrated Load, Uniform Load, Pressure, Temperature, Nodal Heat Flux, Uniform Heat Flux, Iner Heat Flux, and General Model. In this work, new attributes are introduced to enable the preprocessing of composite structure problems for the FAST program, such as Orthotropic Material and Laminated Section 2D, which is necessary to describe the stiffness of the laminate. As a result, in the system's pre-processing stage, these attributes can be applied using the attribute frame available in the FEMEP, as shown in Figure 2.

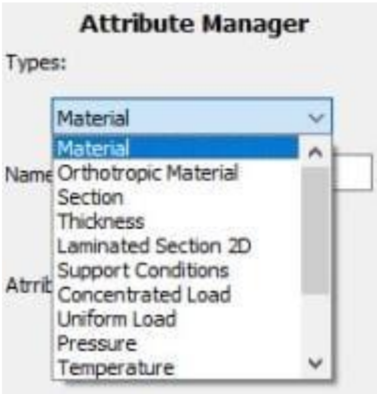

Figure 2. Attribute Manager with new attributes needed for laminated analysis.

Each attribute has its own properties. The Orthotropic Material has three elastic modulus E1, E2, and E3, three Poisson's ratios v12, v13, and v23, and three shear elastic modulus G12, G13, and G23. These attributes can be defined in the interface frame, as shown in Figure 3.

E1	54870000000.0
E2	18320000000.0
E3	0.0
v12	0.25
v13	0.0
v23	0.0
G12	89000000.0
G13	0.0
G23	0.0

Figure 3. Example of material orthotropic

In the case of the layup data, which corresponds to a matrix of the material index, thickness, and orientation of each ply, the attribute patterns available previously in the FEMEP cannot be used since it does not support matrix data type. Hence, and new attribute pattern, called matrixn3, which is stored as an array, is introduced to handle layup's data. The interface for the setup of Laminated Section 2D is presented in Figure 4.

Figure 4. Example of laminated section.

In this work, an academic program named Nlpos (nonlinear analysis post-processing tool) is adopted in the visualization of the simulation results.

4 Application in the analysis of a Laminated Plate

The implemented tool is used to study the structural responses of a laminated plate. This example deals with a symmetrical laminated with 3 plies. The outer laminae 1 and 3 are identical, with h1 = h3 = 3 mm and θ1 = θ3 = 45°. The central ply 2 has h2 = 6 mm and θ2 = 0°. The material properties of each ply are shown in Figure 3, which are modeling as an orthotropic material. The laminated section properties are shown in Figure 4.

The description of the loading condition and the plate dimensions is depicted in Figure 5 a), which consists in 2500 N/m uniform tensile. The problem is analyzed considering the symmetry of the problem, hence only a quarter part of the plate is modeled. The NURBS model used has 64 quadratic elements and is shown in the Figure 5 b).

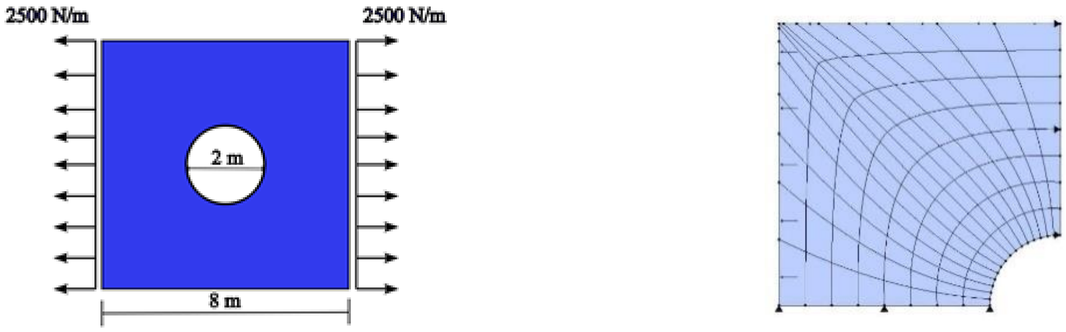

Figure 5. a) Laminated plate analysis description.

Figure 5. b) Analysis NURBS Model.

Figure 5. Description of the analysis model

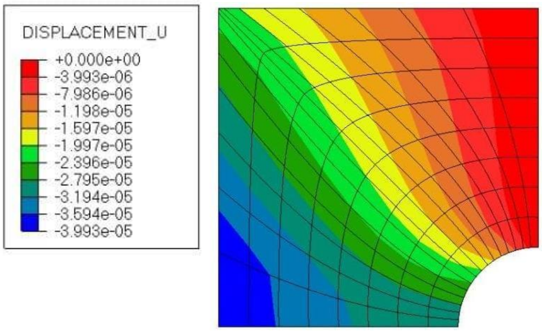

Figure 6. a) Displacement *U*

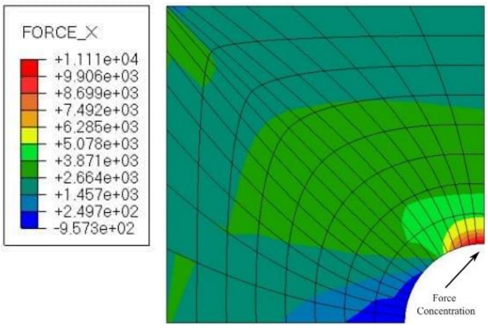

Figure 6. b) Force *X*

Figure 6. Simulation results

The results (displacement u and normal force x) obtained by the FAST program and visualized using Nlpos are shown in Figure 6 a) and b). As expected, a force concentration of 4.4 is observed close to the hole.

5 Conclusions

This paper presents a tool for pre-processing models for structural analysis of fiber-reinforced composite material structures. The implemented tool was obtained by expanding the functionality of the FEMEP program. With the program implemented, it was possible to carry out structural analysis of a laminated plate, where the IGA analysis model is exported to the FAST program format.

For future works, the pre-processing code will be extended to handle T-Splines models, exporting it to the FAST program. Moreover, the laminated module interface will be expanded to support various types of laminated, such as symmetrical and balanced laminates.

Acknowledgements. The authors acknowledge the financial support from UFC (Universidade Federal do Ceará) - PIBIC 2023/2024 - Announcement Nº 02/2023.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] A. A. L. de Freitas. Simulação numérica de um compósito de fibra de vidro e híbrido de kevlar e vidro pós ensaio de impacto de baixa velocidade. Graduating thesis, Federal University of Rio Grande do Norte, 2022.
[2] J. C. L. Peixoto. Análise isogeométrica com modelagem interativa de múltiplas regiões NURBS e T-Splines. MSc thesis, Pontifical Catholic University of Rio de Janeiro, 2024.
[3] J. C. L. Peixoto, R. L. Rangel e L. F. Martha, "Isogeometric analysis with interactive modeling of multi-patches NURBS". XLV Ibero-Latin American Congress on Computational Methods in Engineering. Porto, 2023.
[4] E. S. Barroso. Geração de malhas de alta ordem para análise isogeométrica utilizando elementos de Bézier racionais. PhD thesis, Federal University of Ceará.
[5] E. S. Barroso. Análise e otimização de estruturas laminadas utilizando a formulação isogeométrica. MSc thesis, Federal University of Ceará.
[6] D. S. Bonfim. Uma estratégia de modelagem aberta e extensível para criação de modelos de subdivisões planares para mecânica computacional. MSc thesis, Pontifical Catholic University of Rio de Janeiro, 2022.
[7] P. T. R. Mendonça, "Análise de um laminado". Materiais Compostos e Estruturas-Sanduíche, ed. 2, chapter 6, pp. 136-138, 2019.