Scientific Research Publishing

Materials Sciences and Applications, 2025, 16(7), 399-439
https://www.scirp.org/journal/msa
ISSN Online: 2153-1188
ISSN Print: 2153-117X

Original Engineering Software for Composite Materials Modelling on a Smartphone Device

Mohamad Abbas Kaddaha[1,2], Rafic Younes[1*], Pascal Lafon[2]

[1]Faculty of Engineering, Lebanese University, Beirut, Lebanon
[2]LASMIS, University of Technology of Troyes, Troyes, France
Email: mohamad.kaddaha@utt.fr, *ryounes@ul.edu.lb, pascal.lafon@utt.fr

How to cite this paper: Kaddaha, M.A., Younes, R. and Lafon, P. (2025) Original Engineering Software for Composite Materials Modelling on a Smartphone Device. *Materials Sciences and Applications*, **16**, 399-439.
https://doi.org/10.4236/msa.2025.167024

Received: March 3, 2025
Accepted: July 25, 2025
Published: July 28, 2025

Abstract

The increasing demand for mobile simulation tools has opened new possibilities in engineering applications, particularly in composite material modelling. This paper introduces original engineering software developed to simulate composite materials on smartphones. The research explores the capabilities of mobile devices to perform simulations that are traditionally confined to desktop systems. Key challenges, such as computational limitations and the optimization of software architecture, now with integrated quantitative performance metrics such as computation time, accuracy, and memory efficiency, are addressed through the use of finite element analysis (FEA) and other advanced numerical methods. The software utilizes HTML-based coding for cross-platform accessibility, allowing engineers and researchers to conduct simulations anytime, anywhere. Strategies like parallel processing, cloud-assisted computation, and algorithmic optimization were implemented to enhance performance. The software's real-time feedback and adaptive modelling provide accurate simulations of composite materials such as fiber-reinforced polymers. Furthermore, this paper reviews existing mobile-based simulation tools, highlighting their strengths and areas for improvement, while proposing novel solutions to increase efficiency, accuracy, and usability. The findings demonstrate that mobile devices, with optimized software, can successfully handle complex simulations, democratizing access to advanced engineering tools.

Keywords

Mobile Simulation, Composite Materials, Software Optimization, Mobile Computing, Cloud-Assisted Computation, Fiber-Reinforced Polymers, Real-Time Simulation, Adaptive Modelling, Cross-Platform Simulation

1. Introduction

1.1. Background

Composite materials have become a cornerstone in modern engineering due to their remarkable strength-to-weight ratios and flexibility in tailoring material properties. These materials are often designed to overcome the limitations of traditional materials by combining them in a way that capitalizes on their respective strengths. For instance, in aerospace engineering, composite materials such as carbon fiber reinforced polymers (CFRPs) are used to manufacture lightweight yet extremely strong components, significantly improving fuel efficiency and performance [1] [2].

Simulation tools play a crucial role in the design and analysis of composite structures. Finite Element Analysis (FEA) is a widely utilized numerical method that allows engineers to model complex behaviours of composite materials under various conditions. FEA divides a structure into small elements and solves the governing equations for each element to predict the structure's response to loads, constraints, and environmental conditions [3]. This method is essential for understanding the performance and safety of composite materials in real-world applications.

However, traditional simulation tools—explained in the state of art—often require significant computational resources, which are typically available only on high-performance desktops or workstations. The ability to perform simulations on mobile devices could democratize access to these powerful tools, allowing engineers and researchers to conduct analyses and review results from anywhere, at any time. The rapid advancements in mobile technology, including increased processing power, improved graphics capabilities, and enhanced connectivity, present new opportunities for developing mobile-based simulation tools [4].

1.2. Objectives

This paper explores the potential of mobile devices for performing simulations of composite material structures, focusing on the computational resources, software architecture, with integrated quantitative performance metrics such as computation time, accuracy, and memory efficiency., and available mobile technologies for such tasks. With mobile hardware evolving rapidly, devices like smartphones and tablets are increasingly capable of handling complex computational tasks traditionally performed on desktop systems. The primary goal of this research is to assess whether mobile platforms can effectively simulate composite materials and propose strategies to optimize this process.

First, the paper investigates the ability of mobile devices to perform simulations of composite material structures, commonly used in industries such as aerospace, automotive, and civil engineering. These simulations, which require significant computational resources, will be analyzed to determine whether mobile devices can handle the complexity, utilizing their modern processors and graphics capabilities to meet performance standards.

Second, the paper examines the computational requirements and software architecture, with integrated quantitative performance metrics such as computation time, accuracy, and memory efficiency. Needed for effective mobile simulations. Complex tasks like finite element analysis (FEA), stress testing, and material behavior modelling must be adapted to the limitations of mobile hardware. This research explores how software frameworks, algorithms, and architectures can be optimized to fit these constraints without sacrificing accuracy or efficiency.

Third, it reviews existing mobile applications and platforms that offer simulation capabilities. By evaluating their strengths and limitations, this paper identifies areas where improvements are necessary to achieve reliable mobile-based simulations of composite materials. Both commercial and open-source applications will be analyzed for their adaptability and performance in handling such simulations.

Additionally, this paper presents the development of a software solution based on HTML coding, specifically designed for simulating all types of composite materials. This web-based approach allows users to perform simulations across different devices, taking advantage of the cross-platform compatibility of HTML to ensure accessibility and ease of use on mobile devices.

Finally, the research proposes strategies for optimizing performance and accuracy in mobile-based simulations, including the use of parallel processing, algorithmic optimization, and cloud-assisted solutions. These techniques aim to address the limitations of current mobile platforms and open new research avenues for further development in mobile composite material simulations.

2. State of the Art

2.1. Software for Composite Materials

Developing software requires a well-thought-out architecture that addresses the unique constraints and capabilities of mobile devices:

User Interface (UI): The UI of mobile simulation tools must be designed to be intuitive and user-friendly, accommodating the smaller screen sizes and touch-based interactions typical of mobile devices. Effective UI design includes clear navigation, accessible input methods, and interactive visualizations. Research into mobile UI/UX design principles is essential to create tools that are both functional and easy to use [5].

Core Simulation Engine: The core simulation engine must be optimized for mobile hardware, balancing computational demands with performance. This involves developing efficient algorithms that minimize computational overhead and leverage mobile GPUs for parallel processing. Techniques such as code optimization, memory management, and algorithmic efficiency are crucial for achieving smooth performance on mobile devices [6].

Data Storage and Management: Mobile devices have limited storage compared to desktops, making efficient data management essential. This includes compressing simulation data, using cloud storage solutions, and implementing efficient data retrieval methods. Mobile simulation tools often rely on cloud-based solutions to handle large datasets and provide seamless access to simulation results [7].

Performance Optimization: Performance optimization is critical for ensuring that mobile simulation tools operate efficiently. Techniques such as adaptive meshing, which adjusts the mesh resolution based on simulation requirements, and leveraging hardware acceleration can significantly improve performance. Additionally, incorporating features like incremental computations and result caching can enhance responsiveness and reduce computational load [8].

The development of specialized software has revolutionized the design and analysis of composite materials. Each of these tools uses distinct modelling techniques, which play an essential role in accurately predicting both geometrical and analytical behaviours of composites under various conditions. These tools commonly employ methods such as Classical Laminate Theory (CLT), Finite Element Methods (FEM), Continuum Damage Mechanics (CDM), and Homogenization Techniques.

2.2. ANSYS Composite PrepPost (ACP)

Source: Developed by ANSYS, Inc., a leader in engineering simulation software.

Definition: ACP is a specialized module for the analysis of composite materials within the ANSYS suite, and it is used widely in aerospace, automotive, and marine industries to analyze the mechanical, thermal, and environmental behaviour of composites.

Geometrical Modelling: ACP uses Finite Element Methods (FEM) to create detailed 3D geometries that represent fiber orientations, stacking sequences, and matrix materials. This method divides the composite into smaller, manageable elements that can be analyzed individually, providing high precision in complex fiber-matrix interactions [9].

Analytical Modelling: ACP is primarily based on Classical Laminate Theory (CLT), which is essential for analyzing the behaviour of layered composite materials. CLT assumes that each layer within a laminate responds linearly to loads, making it ideal for studying stress-strain behaviour, deformation, and failure mechanisms in composites [10]. ACP also incorporates failure theories like Tsai-Wu and Hashin criteria to predict failure modes under various load conditions [11].

2.3. Altair Hyper Works (OptiStruct)

Source: Developed by Altair Engineering.

Definition: OptiStruct is a structural optimization solver that specializes in the analysis and optimization of composite materials, often used in industries requiring lightweight yet strong materials, such as automotive and aerospace sectors.

Geometrical Modelling: OptiStruct uses FEM to simulate geometries of composite structures with precise control over fiber orientations, layer thicknesses, and stacking sequences. This allows engineers to optimize the material for specific performance goals, such as minimizing weight while maximizing strength [12].

Analytical Modelling: OptiStruct also relies on Classical Laminate Theory (CLT) to simulate the behaviour of composite laminates. CLT models each layer in a composite structure as a continuum, simplifying complex material behaviour into tractable equations for determining the optimal configuration of fibers and layers [13]. Additionally, OptiStruct incorporates optimization algorithms to identify the best combination of material properties to meet design criteria [12].

2.4. MSC Nastran and Patran

Source: Developed by MSC Software.

Definition: Nastran is a well-established finite element analysis (FEA) solver used in multiple industries, while Patran serves as a graphical interface for setting up models and visualizing results.

Geometrical Modelling: Patran provides FEM-based tools for creating and analyzing complex geometries, especially for aerospace and automotive composite structures. This enables engineers to represent fiber orientations and laminate stacking sequences accurately [14].

Analytical Modelling: Nastran is grounded in Finite Element Methods (FEM), offering advanced simulations that model the mechanical and thermal behaviour of composite materials. Nastran can also incorporate Classical Laminate Theory (CLT) to analyze the response of layered composites to loads. This combination allows for detailed analyses of stress, strain, and potential failure in aerospace components [15].

2.5. Abaqus

Source: Developed by Dassault Systèmes under their SIMULIA brand.

Definition: Abaqus is a powerful FEA tool known for its ability to simulate nonlinear material behaviour, especially under extreme conditions, such as high loads or dynamic impacts.

Geometrical Modelling: Abaqus uses FEM to create detailed representations of composite structures, allowing engineers to simulate complex fiber and matrix interactions, delamination, and other non-linear effects [16].

Analytical Modelling: Abaqus is based on Continuum Damage Mechanics (CDM), a theory used to model the progressive failure of materials, such as delamination or crack propagation in composites. Abaqus also supports Classical Laminate Theory (CLT) in analyzing the behaviour of laminated composites [16]. The combination of CDM and CLT allows Abaqus to simulate the onset of damage, crack growth, and the complete failure of composite materials in highly nonlinear applications [16].

2.6. Wisetex

Source: Developed by TexEng Software Ltd.

Definition: Wisetex is a commercial software package designed for the multi-scale modelling of textile materials, particularly focusing on the mechanical behaviour of textiles in composite applications.

Geometrical Modelling: Wisetex employs Multi-Scale Modelling Techniques to generate 3D representations of textile structures. It models textiles at both the yarn and fabric level, allowing for accurate simulation of composite materials reinforced with textiles [17].

Analytical Modelling: Wisetex uses Homogenization Techniques, which simplify complex textile structures by averaging the material properties of the fabric across different scales. This method is used to model stress-strain behaviour, making it particularly useful for predicting the performance of technical textiles within composite materials [17].

2.7. TexGen

Source: Open-source software developed by the University of Nottingham.

Definition: TexGen is an open-source tool for generating textile models, often used in research and academic settings due to its flexibility and extensibility.

Geometrical Modelling: TexGen excels at Geometrical Modelling Techniques, allowing users to create detailed 3D representations of textiles, such as woven, knitted, or braided fabrics. This makes it ideal for simulating the microstructure of textile-reinforced composites [18].

Analytical Modelling: TexGen uses Homogenization Techniques similar to those used in Wisetex, where the complex geometry of the textiles is simplified to model their mechanical properties. This helps predict how textiles behave under various loading conditions when used as reinforcement in composite materials [18].

3. Software Features

3.1. Originality

When discussing originality in the context of RVE (Representative Volume Element) simulations and graphical interface design, it is important to emphasize the novel aspects that set apart one approach or tool from others. Originality refers to how innovative a solution is compared to existing methods and the new contributions it brings to the field of material simulation, engineering, or graphical user interfaces.

1) Innovative Computational Approaches

Our approach to RVE modelling incorporates several cutting-edge computational techniques that streamline the process of material analysis. One unique feature is the use of adaptive RVE modelling, parallel processing, and GPU acceleration strategies, where the RVE adjusts its resolution dynamically based on the critical regions of the material. This not only saves computational resources but also ensures more accurate simulations in regions where microstructural details are most important, such as interfaces between different material phases. Unlike traditional static RVEs, this dynamic adjustment represents a leap in computational efficiency and precision.

2) Real-Time Feedback in Simulations

Another original aspect is the integration of real-time feedback into the simulation process. In most existing RVE modelling tools, users must wait for the entire simulation to complete before viewing results, which can take hours or even days. Our system provides real-time updates on stress distribution, deformation, and failure points during the simulation, allowing users to make quick adjustments if needed. This real-time interaction is highly innovative and enables users to experiment with different configurations, enhancing productivity and decision-making.

3) Novel User Interaction Features

In terms of user interaction, our approach introduces novel visualization techniques that enhance the understanding of complex RVE data. For example, multi-layered 3D visualization tools allow users to see different aspects of the material's microstructure, such as fiber alignment, matrix distribution, and interfacial zones, all at once. This offers a more comprehensive view of the material compared to traditional 2D cross-sections or simplified 3D models, making it easier for engineers to interpret the data.

3.2. Memory Space Management Challenges in RVE-Based Simulations for Composite Materials

RVE (Representative Volume Element) simulations are essential for analyzing the behavior of heterogeneous composite materials, where different phases are dispersed within a matrix. Compared to conventional modelling that often focuses on simpler, homogeneous materials, RVE-based simulations are more memory-intensive due to their complexity. This summary outlines the primary memory space management challenges encountered in RVE simulations for composite materials.

1) Handling High-Resolution Microstructures:

RVE modelling requires high-resolution representations of composite materials to accurately capture microstructural features such as fiber arrangements and voids. Traditional models often simplify materials as isotropic or homogeneous, which reduces data volume and memory requirements. In contrast, RVEs must account for the variability and interactions of distinct material phases, resulting in significantly larger datasets, often reaching several gigabytes, as each voxel or finite element must store comprehensive information about material properties, geometrical structure, and stress-strain relationships.

2) Memory Optimization Techniques:

In traditional software modelling, memory optimization techniques often depend on simplifying assumptions like symmetry and homogeneity, enabling models to run with minimal memory. For example, assuming plane stress conditions reduces degrees of freedom and lowers memory consumption. However, RVE simulations cannot simplify their models without sacrificing accuracy. Techniques like adaptive meshing are utilized, where mesh density increases in critical regions but still leads to higher memory demands than conventional models. RVE simulations often require distributed memory systems to manage their extensive memory requirements, adding complexity to data communication between processing nodes, unlike simpler models that can run on a single processor.

3) Data Storage and Compression:

RVE simulations for composites require advanced data storage and compression techniques due to their large storage needs. Traditional modelling methods utilize straightforward storage approaches, but RVE simulations necessitate sparse matrix storage and compression algorithms to manage extensive datasets. Although these techniques reduce memory consumption, they complicate data retrieval and processing. The heterogeneous nature of composites also leads to the need for multiple datasets to represent various material phases, further increasing memory burdens compared to simpler traditional models.

4) Simulation Duration and Real-Time Processing Constraints:

RVE simulations require longer computation times because of the detailed microstructural information involved. This prolonged duration necessitates efficient memory management to keep data readily accessible while avoiding excessive memory swapping. Unlike traditional models, which can dynamically allocate memory for quick real-time processing, RVE-based simulations maintain a large memory footprint throughout the process, complicating real-time performance. As a result, solutions like solid-state drives (SSDs) and cloud-based memory scaling are often employed for improved memory access.

In conclusion, memory space management in RVE-based simulations of composite materials is significantly more complex than in traditional modelling approaches. The challenges of managing high-resolution microstructural data, optimizing memory through compression, and balancing real-time processing with high memory needs necessitate sophisticated strategies to accurately represent the heterogeneous nature of composites. Addressing these challenges is crucial for enhancing composite material simulations, leading to improved predictions of material behavior and performance.

3.3. Simplicity of Graphical Interface

A key feature of any successful computational tool is the simplicity of its graphical interface (GUI). A well-designed GUI ensures that users, regardless of their technical expertise, can easily interact with complex simulation systems without needing deep knowledge of the underlying computational models.

1) User-Centered Design

Our graphical interface is designed with simplicity and ease of use in mind. One of the primary goals was to reduce the learning curve for new users while still providing advanced options for experienced engineers. By adopting a user-centered design approach, we focused on creating a clean and intuitive layout that eliminates unnecessary complexity. For example, rather than overwhelming users with countless configuration options at the outset, our interface guides them step-by-step through the setup process, displaying only the most relevant options at each stage. This reduces confusion and allows users to focus on key tasks.

2) Drag-and-Drop Functionality

One of the standout features that adds simplicity to the interface is the use of drag-and-drop functionality. Users can easily drag different material properties, boundary conditions, or geometrical shapes into the simulation space. This reduces the need for manually inputting parameters or coding, making the tool more accessible to users without a programming background. The drag-and-drop approach, coupled with clear visual indicators, simplifies the setup process and speeds up the overall workflow.

3) Pre-Configured Templates and Libraries

To further streamline the process, our GUI provides a library of pre-configured templates for common materials and geometries. Users can select from a range of pre-built RVEs for different composites or customize their own. This feature eliminates the need to build each RVE from scratch, reducing setup time and allowing users to quickly test multiple configurations. Furthermore, for those who wish to customize, the interface provides easy-to-use sliders, drop-down menus, and input boxes for adjusting parameters, all of which are intuitively labeled to ensure clarity.

3.4. Fast and Easy Geometrical Modelling for RVE-Based Simulations

Geometrical modelling is at the core of RVE-based (Representative Volume Element) simulations, which are crucial for predicting the mechanical and physical properties of composite materials. Our system focuses on delivering both precision and simplicity in this process, offering advanced tools for accurate microstructural representation, parametric modelling, automated meshing, and interactive RVE creation and customization. These capabilities are designed to ensure efficient and accurate simulations, from initial modelling to the final analysis of the material's performance.

1) Accurate Representation of Material Microstructure

A significant challenge in composite material simulations is the accurate depiction of the microstructure. Traditional tools often rely on idealized, simplified assumptions—such as perfectly aligned fibers and uniform void distribution—which may lead to inaccurate predictions. Our geometrical modelling tools enable the simulation of realistic, stochastic microstructures, accurately capturing fiber arrangements, voids, and matrix distributions. This ensures that the RVE more closely reflects actual material conditions, leading to better predictions of material behavior, including stress distribution, strain localization, and failure mechanisms under various loading scenarios.

2) Parametric Geometrical Modelling and Sensitivity Analysis

Our system's parametric geometrical modelling capability enables users to define key parameters, such as fiber volume fraction, orientation, and pore size, with the software automatically generating corresponding geometries. This streamlines the modelling process, ensuring consistency across simulations. Furthermore, this parametric approach allows users to perform sensitivity analyses efficiently, adjusting material parameters to explore how changes in the microstructure affect overall material performance. Such rapid evaluations are invaluable for optimizing composite materials for specific industrial applications.

a) Automated Meshing for Improved Simulation Accuracy

Our software includes an automated meshing algorithm that dynamically adapts to the material's microstructure, refining the mesh in areas prone to stress concentrations, such as around fibers or voids. This improves the precision of the simulation, while reducing the need for manual mesh adjustments, allowing for faster and more accurate results, particularly in complex composite systems.

b) Simplified RVE Creation and Template Library

Creating and manipulating RVEs can be a challenging task, but our system simplifies this process through the use of pre-built RVE templates. These templates cater to various composite material types, including fiber-reinforced polymers, porous materials, and metal matrix composites. The templates come pre-loaded with common material properties and geometries, enabling users to easily modify them for specific simulation needs. This significantly reduces setup time and provides a head start for users to focus on analysis rather than model creation.

c) Clear Visual Representation of RVEs

Our system provides clear, high-resolution visual representations of RVEs throughout the modelling and simulation processes. Users can explore the RVE in both 2D and 3D, rotate models, zoom in on areas of interest, and apply different material phases to understand how the microstructure behaves under applied loads. This intuitive visualization helps users identify potential stress concentrations, failure points, and other critical aspects, leading to better-informed decisions during material design and optimization.

d) Interactive RVE Customization with Real-Time Feedback

One of the key features of our system is the ability to interactively customize the RVE in real time. Users can adjust fiber orientations, modify material properties, or set boundary conditions directly within the graphical interface. As changes are made, the visual representation of the RVE updates instantly, providing immediate feedback on how these modifications impact the material's predicted behavior. This real-time interaction greatly enhances the ability to experiment with different configurations, enabling faster, more informed decisions about material design and performance.

4. Methodology

At the core of the flowchart is a multi-step process that begins with opening the application and selecting the desired category of composite material. The material categories are clearly defined and structured in a manner that guides the user through selecting the correct simulation parameters and calculations for their chosen composite type, as shown in Figure 1 and Figure 2.

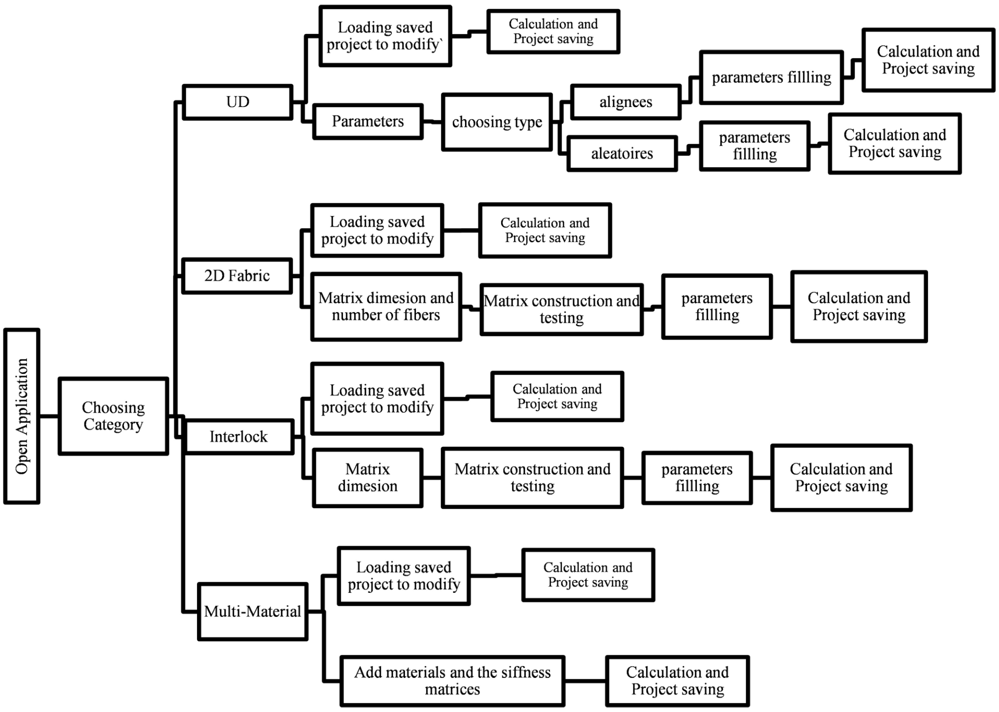


Figure 1. Code shart and applications 1.

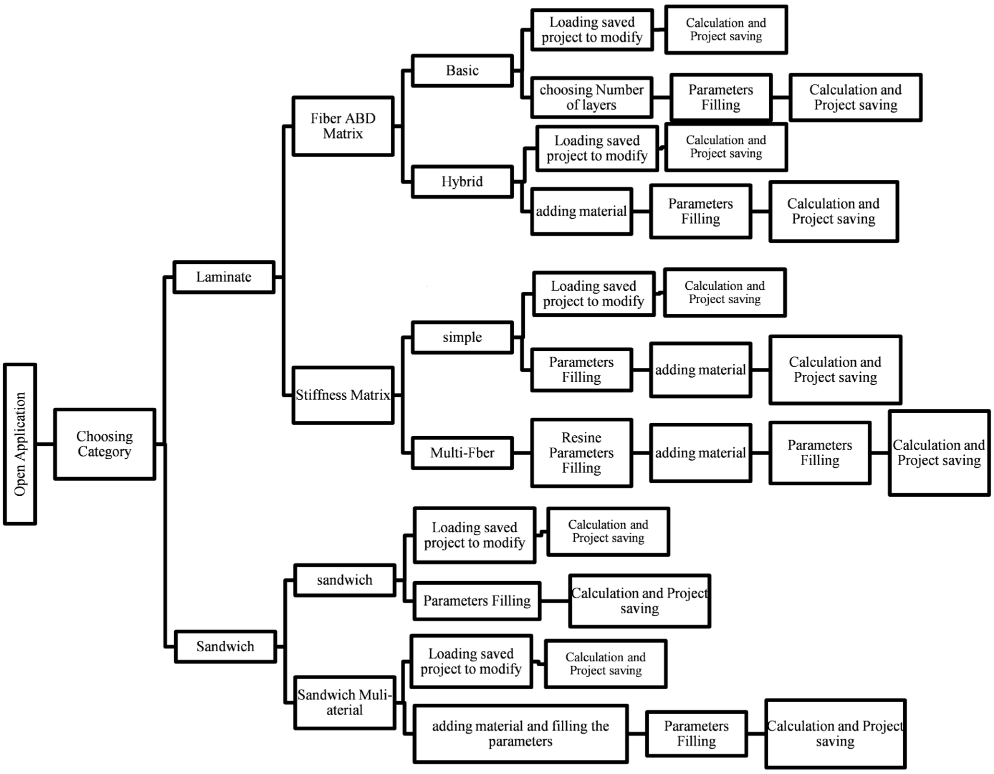


Figure 2. Code shart and application 2.

A structured validation process was conducted to ensure the accuracy and reliability of the mobile simulation tool. The simulation results were compared against established benchmark problems from traditional finite element analysis (FEA) software, such as ANSYS and Abaqus. Key validation tests included stress-strain analysis on standard geometries, thermal diffusion simulations, and modal analysis of simple structures. The results were evaluated based on absolute error percentage and convergence behavior. Additionally, analytical solutions were used where applicable to verify computational accuracy. The mobile tool was deemed valid if its results fell within a ±5% error margin compared to desktop simulations.

4.1. Categories of Composite Materials

4.1.1. UD (Unidirectional)

The unidirectional (UD) section allows the user to load an existing project or start fresh by configuring parameters. This is followed by choosing the type of alignment: either "alignées" (aligned fibers) or "aléatoires" (random fibers). After the fiber orientation is selected, the software proceeds to fill in necessary parameters and perform calculations, ultimately saving the project for later use.

4.1.2. 2D Fabric

In the 2D fabric section, the process involves defining the matrix dimensions and specifying the number of fibers in the material. Once these inputs are configured, the software performs matrix construction and testing, followed by parameter filling. Calculations are performed to assess the behavior of the composite, and the project is then saved.

4.1.3. Interlock

This section follows a similar structure to 2D Fabric, where the matrix dimensions are configured first. Then, the matrix is constructed and tested. Once parameters are filled, the software proceeds to the final calculations and project saving stage.

4.1.4. Laminate

Laminate composites are broken down into two main types: fiber ABD matrix and stiffness matrix.

- **Fiber ABD Matrix**: This section includes a "basic" option, which allows the user to choose the number of layers in the laminate. After that, parameters are filled, calculations are performed, and the project is saved. The "hybrid" option, on the other hand, includes an additional step of adding materials before filling parameters, performing calculations, and saving the project.
- **Stiffness Matrix**: In this section, simple and multi-fiber options are available. The simple option involves filling in basic parameters and adding materials, followed by calculations. The multi-fiber option adds an additional step for filling resin parameters.

4.1.5. Sandwich

This type of composite material is characterized by different structural layers.

There are two subcategories: regular "sandwich" and "sandwich multi-material." Both processes allow users to modify a saved project and proceed through parameter filling, adding materials, and final calculations. The sandwich multi-material category has an extra step to add materials and configure the stiffness matrix before proceeding with the calculations and project saving.

4.1.6. Multi-Material

Multi-material composites combine several types of materials in a single structure. The process here involves adding materials and defining stiffness matrices, filling in necessary parameters, and conducting final calculations. The project can then be saved after all necessary configurations are completed.

4.2. Key Processes

4.2.1. Loading Saved Projects

A notable feature of this software is the ability to load previously saved projects. This feature is available across all composite material types, allowing users to quickly modify and update existing projects rather than starting from scratch. This can be particularly useful when making minor adjustments or testing different configurations on a pre-existing material.

4.2.2. Parameter Configuration

Each material type requires specific parameters to be filled before proceeding to calculations. These parameters could include fiber orientation, matrix dimensions, number of layers, material type, and resin properties. The structure of the flowchart indicates that parameter filling is a crucial step in ensuring that the simulation is accurate and tailored to the specific properties of the composite material.

4.2.3. Matrix Construction and Testing

For some materials, such as 2D fabric and interlock composites, matrix construction and testing are an intermediate step. This likely involves generating a digital model of the material's structure, which can then be used to test its performance under simulated conditions. The software's ability to perform these tests ensures that users can evaluate how well their chosen parameters work before finalizing the project.

4.2.4. Material Addition

The ability to add materials, particularly in hybrid laminate and multi-material composites, is another feature highlighted in the flowchart. This allows users to create more complex simulations that take into account a variety of materials within a single project. For example, in hybrid laminates, users can test how combining different materials within one structure affects the overall performance.

4.2.5. Calculation and Project Saving

After all parameters are set and materials are added, the software performs calculations to simulate the behavior of the composite material. These calculations include strength testing, force and displacement analysis and other performance indicators specific to the material using calculations and analytical modeling explained in previous work [19] [20]. The process culminates in saving the project, allowing the user to return to it later or use it as a reference for future simulations.

5. Discussion

This flowchart highlights the versatility of the software in handling various types of composite materials. Providing distinct processes for each material type ensures that users can simulate a wide range of composites, from unidirectional and fabric-based materials to more complex multi-material and laminate structures. The inclusion of matrix construction, testing, and the ability to add different materials further enhances the software's capability to handle complex simulations.

Moreover, the ability to load saved projects provides an efficient workflow for users who need to iterate on previous designs or make adjustments based on new requirements. The final step of calculation and project saving ensures that the results of each simulation are preserved for further analysis or future reference.

The benchmarking results highlight both the strengths and limitations of mobile-based simulations. While the mobile tool delivers computational accuracy close to traditional desktop solutions, its performance is influenced by hardware constraints, particularly in processing-intensive simulations. The tool is well-suited for preliminary analysis and educational use, but further optimization is required for complex, large-scale simulations. The reduced memory footprint and energy efficiency make it advantageous in scenarios where portability is essential. Future iterations should focus on enhancing computational efficiency through optimized solvers and leveraging cloud-based processing for improved scalability.

Overall, this software presents a comprehensive tool for simulating composite materials across various industries. It is highly customizable, allowing users to tailor each simulation to the specific properties and behaviors of the materials they are working with. This flexibility makes it a valuable resource for engineers, researchers, and designers who need to accurately simulate the performance of composite materials in different applications.

6. Case Studies

6.1. Web Page

This web page is part of a larger web application that seems to focus on materials, specifically composite materials like fabrics, laminates, and knitted composites. The page is designed to offer users a wide range of tools and features for working with these materials, but the explanation here will avoid diving deep into technical details.

When you visit this page, you'll see an interface where different buttons and options allow you to explore various materials and their properties. Each button represents a different type of material or structure that you might be interested in, such as "Interlock", "Laminate", or "Knitted Composites". Depending on which button you click, the page will show you different tools and information related to that material.

The page uses a popular design framework called Bootstrap, which helps make everything look clean and organized. The buttons are styled to be big and easy to click, which makes the page user-friendly. The page is also designed to adjust its layout depending on the size of your screen, so it should look good whether you're on a computer, tablet, or smartphone, as shown in **Figure 3**.

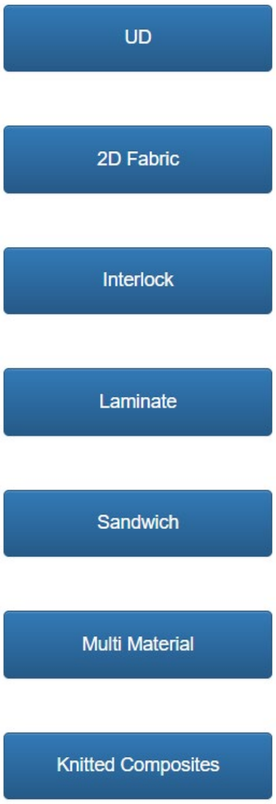


Figure 3. Web page layout.

The navigation is straightforward. When you first load the page, you're presented with several buttons, each representing a different type of material or analysis tool. For example, you might see buttons labeled "UD" (which could stand for something like "Unidirectional"), "2D Fabric", "Interlock" and so on. These buttons allow you to choose what you want to work on.

Once you make a selection by clicking one of these buttons, the page updates to show specific tools or options related to your choice. If you click on "Laminate", for instance, new buttons might appear that let you dive deeper into that specific type of material, allowing you to explore things like its stiffness or how different fibers interact within it.

One of the key features of this page is its interactivity. Depending on what you're working on, you might be able to input data, adjust settings, or view visual representations of different materials. For example, if you're working with a laminate, you might be able to see how different layers are arranged and how they affect the overall strength of the material.

The page is highly customizable, which means that users can tailor their experience based on their specific needs. Whether you're focusing on basic materials or more complex, hybrid composites, the page adjusts to provide you with the relevant tools and information.

Overall, this page is designed to be a hub for anyone working with composite materials. It's likely used by engineers, researchers, or students who need a detailed and interactive way to explore the properties of different materials. The page provides a range of tools to help users analyze, compare, and understand these materials in a way that's both intuitive and informative.

In summary, this page is like a digital workshop for people working with materials. It offers a variety of tools and resources, all neatly organized and easy to navigate, to help users get the most out of their work with composites.

6.2. UD

This section presents a comprehensive overview of a custom directive function implemented within a user interface framework. This directive supports the design of dynamic and interactive panels used for setting mechanical and thermal properties, providing users with an intuitive interface for complex engineering calculations. The function leverages modularity and flexibility, offering real-time feedback and parameter adjustments to streamline decision-making in engineering environments. The study explores its structure, application, and implications in computational and material sciences as shown in **Figure 4**.

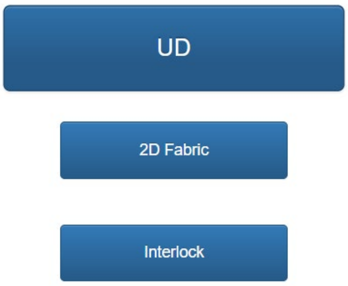


Figure 4. Unidirectional fiber.

In modern computational and material sciences, designing tools that allow users to intuitively interact with large datasets or models is paramount. Tools that facilitate adjustments to critical parameters, such as mechanical and thermal properties, can significantly enhance productivity and accuracy. This section explores a function that enables such interactions through an adaptable user interface, allowing engineers and scientists to easily input and manipulate material properties. By offering a directive with real-time feedback, this function addresses the challenge of making complex calculations more accessible to professionals.

User interfaces in engineering and scientific computing are evolving rapidly to accommodate increasingly complex tasks. As research expands, there is a greater demand for systems that provide a seamless interaction between users and data models. In particular, setting material properties like elasticity, thermal conductivity, and mechanical behavior requires an interface that can accommodate numerous variables and generate quick feedback based on real-time changes. The function described in this study integrates seamlessly within a web-based environment, allowing users to manage properties like fiber and resin parameters, thermal expansion coefficients, and modulus of elasticity.

The custom directive is implemented to provide an adaptable panel interface that allows users to select between various options such as "Aligned" and "Random" material property settings. It provides input fields for parameters such as Ef (modulus of elasticity), Nuf (Poisson's ratio), and thermal expansion coefficients. Based on the user's input, the directive performs calculations to determine the overall mechanical or thermal properties of composite materials, as shown in **Figure 5**.

Figure 5. Types of UD fibers.

The function processes user input through predefined forms, allowing the adjustment of parameters like modulus of elasticity (Ef1, Ef2), Poisson's ratios (Nuf12, Nuf23), and thermal expansion coefficients (Alphaf1, Alphaf2). When these values are entered, the function updates relevant variables and uses mathematical models to calculate resulting mechanical and thermal properties. For example, the elastic moduli in different directions are calculated based on the volume fractions of the fiber and resin, demonstrating the integration of engineering principles within the directive.

A significant aspect of the function is its ability to provide real-time feedback through on-screen calculations. Once input values are set, the system calculates material properties using established mathematical formulations, such as combining fiber and resin characteristics to compute overall modulus and thermal expansion behavior. The ability to switch between aligned and random material orientations further showcases its utility in simulating different real-world scenarios.

This directive is particularly useful in material science, where researchers often need to explore the mechanical and thermal behaviors of composite materials. The function allows for the rapid prototyping of different material combinations, facilitating quicker decision-making and design optimization. For example, engineers working on fiber-reinforced polymers can input various fiber orientations and resin types to observe how these changes affect the overall performance of the material, as shown in **Figure 6**.

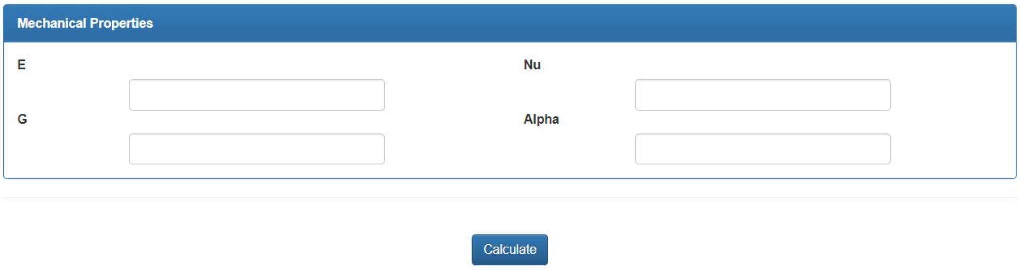


Figure 6. Fiber parameters.

Beyond material science, the directive finds applications in broader engineering systems, such as the design of mechanical components and thermal management systems. By allowing users to easily calculate mechanical properties based on real-time adjustments, engineers can better predict the behavior of materials under different conditions, leading to more efficient and cost-effective designs.

The flexibility and adaptability of the described function make it a powerful tool in both educational and professional environments. Students and engineers can experiment with different parameters and instantly see the impact of their changes, fostering a deeper understanding of composite material behavior. Furthermore, the real-time calculation feature provides immediate insights, enhancing the user experience and reducing the time needed for iterative design processes.

However, the function could be further enhanced by incorporating additional material models and properties. For instance, integrating viscoelastic or temperature-dependent behavior would make the tool more comprehensive for advanced simulations. Additionally, expanding the function to handle more complex geometries could increase its applicability in 3D modelling software.

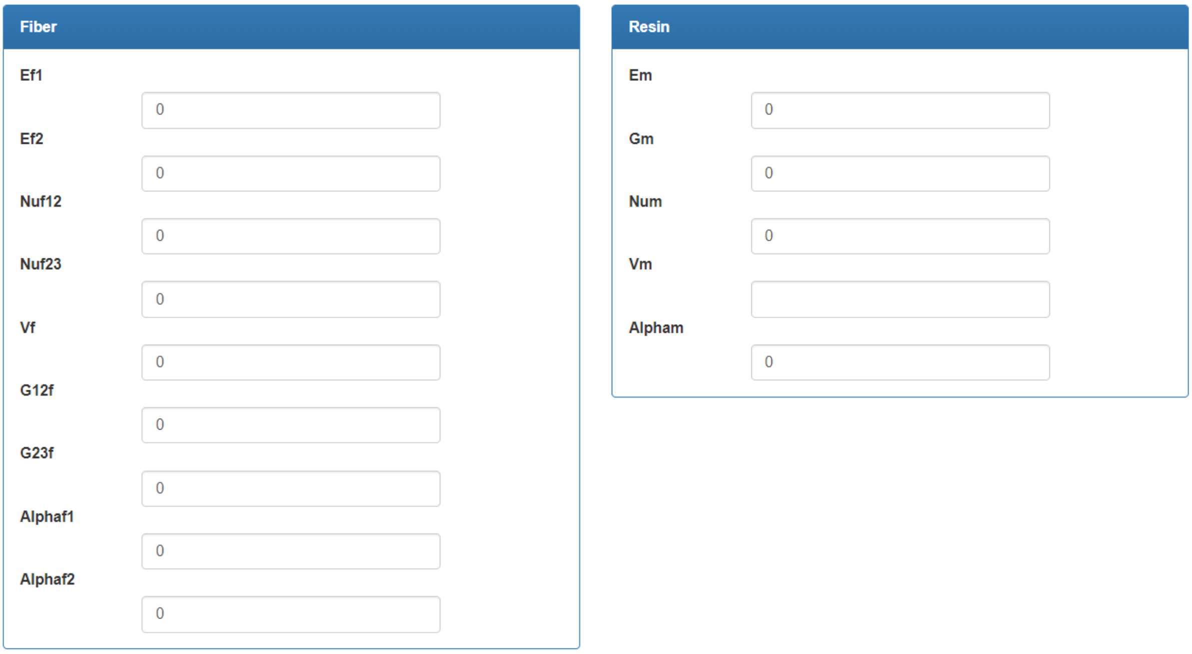
The custom directive function outlined in this section represents a valuable contribution to user interface design in scientific computing, particularly in material science and engineering. By simplifying the process of adjusting and calculating material properties, this tool has the potential to significantly enhance both productivity and learning in environments that require rapid analysis of mechanical and thermal behavior. Future developments should focus on expanding its capabilities to cover more complex materials and simulations.

It is a function that enables an interactive and dynamic interface for adjusting mechanical and thermal parameters in composite materials. The function facilitates user input and provides real-time calculations for various material properties, offering a flexible tool for engineers and scientists working with fiber-reinforced composites. Through a combination of buttons, input fields, and interactive panels, the function allows users to select different modes and enter values that represent the characteristics of materials, such as fiber and resin properties.

The user interface includes buttons that allow users to select between different modes, such as “Parameters”, “Random” and “Aligned” representing different ways of defining material orientations and behaviors. The function also features detailed input fields for mechanical and thermal properties like modulus of elasticity, Poisson’s ratio, and thermal expansion coefficients. These values are then used to perform real-time calculations of the overall mechanical and thermal behavior of composite materials.

When the “Parameters” option is selected, the interface presents input fields for both fiber and resin materials, allowing users to enter values such as modulus of elasticity (Ef1, Ef2), Poisson’s ratios (Nuf12, Nuf23), and thermal expansion coefficients (Alphaf1, Alphaf2). These values are essential for determining the overall material performance, particularly in applications that require accurate simulations of fiber-reinforced materials. The calculations take into account the volume fractions of the fiber and resin materials and update the results in real-time based on user input.

For the “Random” option, the function computes mechanical properties such as modulus of elasticity (E), Poisson’s ratio (Nu), and shear modulus (G) based on probabilistic models. These properties are calculated by combining the individual characteristics of the fiber and resin components, reflecting their random distribution within the composite material as shown in Figure 7.



Fiber

Ef1	0
Ef2	0
Nuf12	0
Nuf23	0
Vf	0
G12f	0
G23f	0
Alphaf1	0
Alphaf2	0

Resin

Em	0
Gm	0
Num	0
Vm	
Alpham	0

Figure 7. Fiber and Matrix properties.

In the “Aligned” mode, the function provides additional options to select specific directions (Direction 1, Direction 2, or Direction 3) for the material’s mechanical and thermal properties. This mode is particularly useful for analyzing materials with a specified orientation of fibers. The interface calculates the properties for each direction, such as elasticity, shear modulus, and thermal expansion, using predefined models that reflect the influence of fiber orientation on the material's behavior. It also allows users to choose between different mechanical and thermal modes, providing results that match the specific conditions and requirements of the materials being analyzed.

The real-time feedback feature of the function is a crucial element, as it immediately shows the calculated results based on the user's input. This capability makes it an effective tool for prototyping and analyzing various material configurations, allowing users to explore how different fiber and resin combinations impact overall performance. The ability to calculate and display these properties quickly enables more efficient design processes, especially in materials science and engineering fields where time and accuracy are critical as shown in **Figure 8**.

Figure 8. Fiber direction.

Overall, this function represents a highly adaptable and powerful interface for managing and calculating the mechanical and thermal properties of composite materials. Its flexibility and real-time calculation capabilities make it valuable in both educational and professional contexts, offering users an intuitive way to explore the complex behaviors of materials used in engineering and scientific applications.

D Fabric

The user—whether a student or a fellow researcher—can themselves design a fabric composite. To construct the whole structure of composite as he or she likes from the beginning and to make the whole process "user friendly", the process took the shape of building a puzzle by a simple "click and drop" game as shown in the figure below (**Figure 9**) [20].

As 2D woven fabrics are made by interlacing yarns in a weaving loom, yarns are divided into two components: one called the warp, running along the length of the loom, and the other called the weft, running in the cross direction. The woven structure is characterized by orthogonal linking of two sets of threads, called warp and weft yarn. The warp threads are aligned with the direction of the fabric leaving the weaving equipment, which is also called the warp direction. As shown in **Figure 6**, 2D woven fabric has two yarn sets as warp (0˚) and filling (90˚) and is interlaced with each other to form the surface as shown in **Figure 10**.

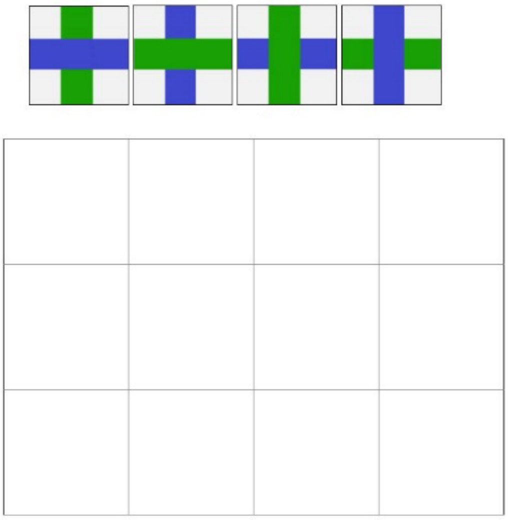


Figure 9. D fabric puzzle.

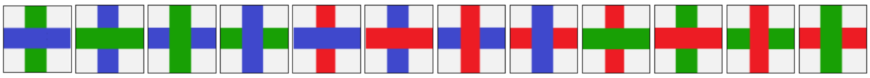
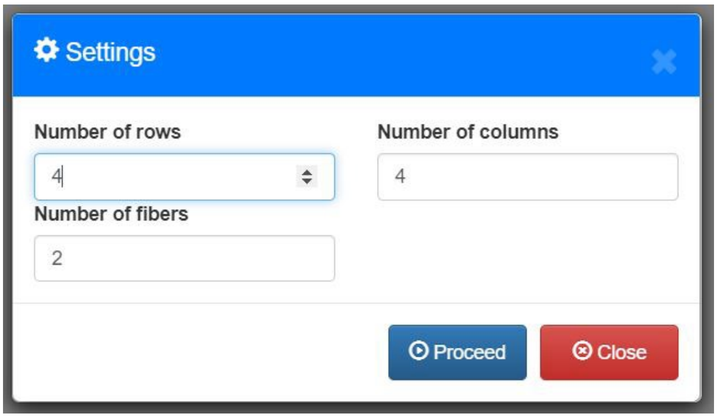


Figure 10. Types of cross fibers.

To start from the beginning, the user is free to choose the settings of the puzzle in each dimension, *i.e.*, number of columns and rows, and the number of fibers, as shown in **Figure 11**.



Settings

Number of rows: 4

Number of columns: 4

Number of fibers: 2

Figure 11. Puzzle dimensions.

The next step is to construct the geometry as desired, as shown in the following examples in **Figure 12** [20].

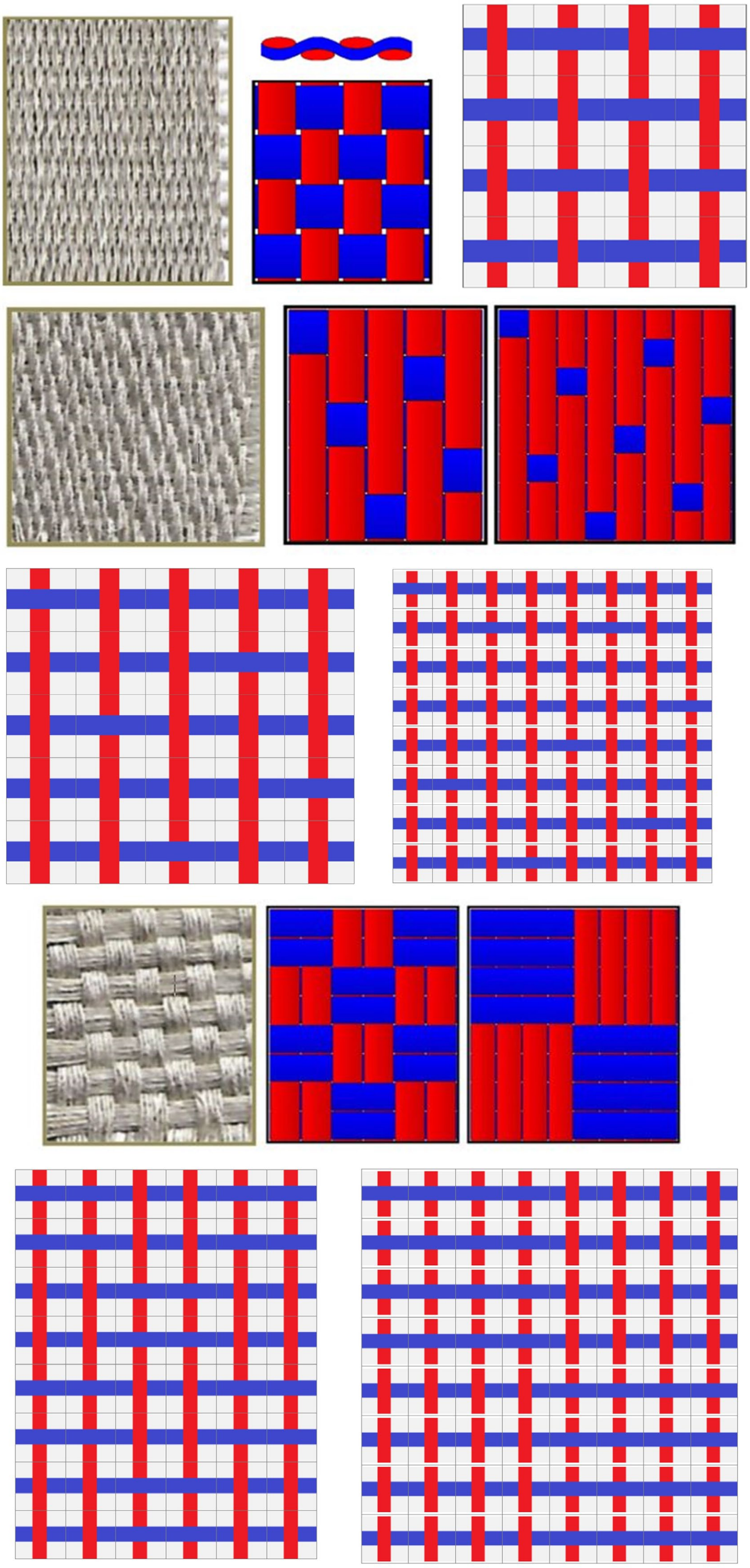


Figure 12. D fabric examples.

6.3. Laminate

This section introduces an interface designed for configuring basic laminate settings, specifically allowing users to input and adjust the number of plies in a laminate structure. The user interface is presented in a modal format, giving users a simple and intuitive way to define the settings that influence the laminate's configuration in various engineering and material science applications [19].

The main input field within the model is for specifying the number of plies. Users can enter a positive integer to represent the desired number of plies in the laminate. This input is validated to ensure that only valid integers are accepted, preventing any erroneous configurations. The default value for the number of plies is set to three, allowing users to start with a standard configuration that they can easily modify based on their needs.

Once the user inputs the number of plies, they can proceed by clicking the save button, which triggers the application to collect and store the input value. The data is then used to configure the laminate structure, and this information is passed to a higher-level context where it can be applied in simulations, design models, or manufacturing processes.

The simplicity of the interface ensures that users can quickly and easily input the number of plies, providing a straightforward but crucial configuration step in the design and analysis of laminate materials. The interaction between the user and the interface is designed to be seamless, with built-in validation to ensure that all input values are correct before they are processed [19].

By offering this streamlined method for setting the number of plies in a laminate, the interface enhances efficiency and precision in the configuration process. It allows users to define a key material property with minimal effort, ensuring that the resulting laminate structure is aligned with the user's specific requirements. This makes the interface particularly valuable in contexts where laminate design plays a critical role, such as in material science, engineering, and manufacturing applications as shown in **Figure 13**.

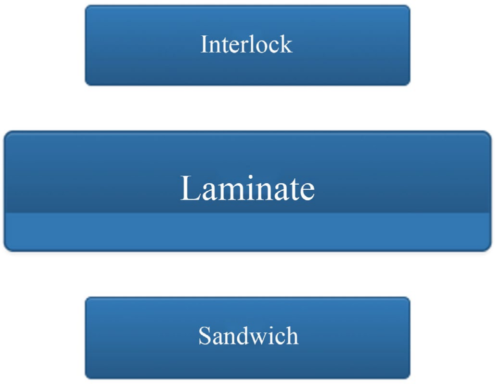


Figure 13. Laminate section.

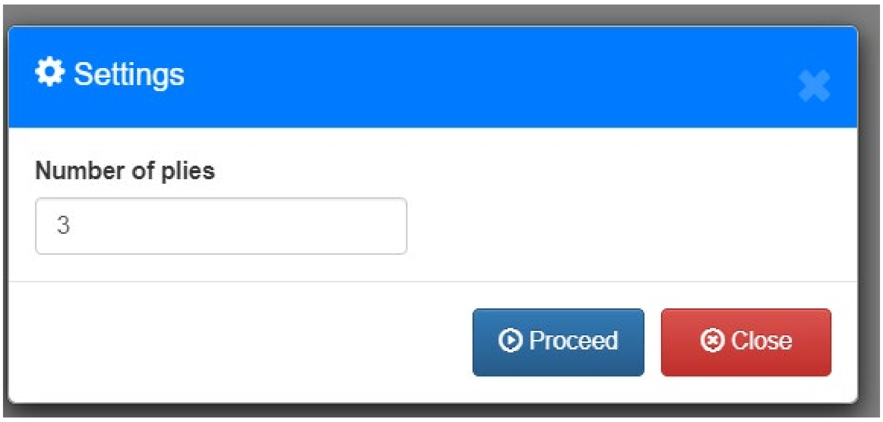
This section outlines an interactive interface designed to manage the configuration of laminate structures, specifically allowing users to input parameters such as ply thickness, elastic properties, and the angle of laminate layers. The interface also provides options for users to select between force or displacement input, and it displays results based on these settings.

The user interface is structured into three main panels: one for material parameters, one for angle settings, and one for force or displacement options. The parameters panel includes input fields for properties like ply thickness, modulus of elasticity (E1 and E2), shear modulus (G12), and Poisson's ratio (Nu12). These fields ensure users can enter precise numerical values with high decimal precision, and all inputs are validated to ensure they meet the required criteria.

In the angles panel, users can define the angles of each ply in the laminate structure. The number of input fields corresponds to the number of plies in the laminate, allowing users to input angles for each ply layer. This ensures flexibility and accuracy in defining the orientations of the laminate layers, which are crucial for determining the overall mechanical behavior of the material.

The force/displacement panel allows users to select whether they want to configure the system based on force or displacement. A dropdown menu lets users choose between the two, and the corresponding input fields adjust accordingly. If force is selected, users can input force values, and if displacement is chosen, the input fields adjust to accept displacement values. This flexibility allows the system to be adapted to different types of analysis or testing scenarios.

Once all the inputs are provided, users can click the submit button to perform the necessary calculations. The system calculates the ABD matrix, a key component in laminate theory, which relates forces and moments to strains and curvatures in the laminate. The results of these calculations are displayed in scientific notation, offering users clear and precise feedback. Additionally, a vector for either force or displacement, depending on the user's selection, is generated and presented in the results section as shown in **Figure 14** [19].

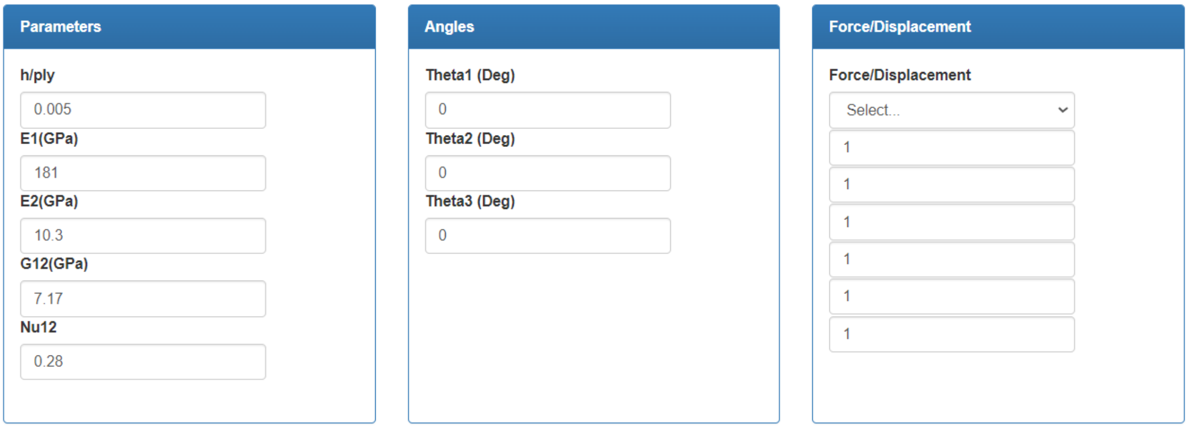


Number of plies: 3

Figure 14. Plies selection.

The interface also provides an option to return to the previous page if users want to make changes to their input or start over. The interaction between the user and the system ensures that all data is validated before submission, preventing any errors in the configuration process.

By offering this structured, user-friendly method for configuring laminate settings, the interface enhances both efficiency and accuracy in laminate analysis. It allows users to quickly input critical material properties and see the resulting calculations in real-time. This makes the tool particularly valuable in engineering and material science applications where precision and adaptability are crucial. The seamless integration of force/displacement options further increases the flexibility of the tool, allowing it to cater to a wide range of design and analysis needs as shown in Figure 15.



Parameters

h/ply: 0.005
E1(GPa): 181
E2(GPa): 10.3
G12(GPa): 7.17
Nu12: 0.28

Angles

Theta1 (Deg): 0
Theta2 (Deg): 0
Theta3 (Deg): 0

Force/Displacement

Force/Displacement: Select...
1
1
1
1
1
1

Figure 15. Parameter selection.

The form is designed to facilitate the input and analysis of laminate materials, which are composites made by stacking multiple layers (plies) with varying properties. It begins with the Parameters Panel, where users input fundamental properties of the laminate, including the thickness of each ply (h/ply), Young's moduli in two directions (E1 and E2), the shear modulus (G12), and Poisson's ratio (Nu12). These inputs are crucial for understanding the laminate's stiffness and deformation characteristics [19].

Next is the Angles Panel, which captures the orientation of each ply within the laminate. This section dynamically generates input fields for specifying the angle of each ply, which is essential for determining how the laminate will respond to stress. The angle data influences the laminate's overall strength and flexibility.

The Force/Displacement Panel is used to define the conditions under which the laminate will be analyzed. It includes a dropdown menu for selecting different force or displacement scenarios and input fields for entering numerical values related to these scenarios. This section helps in evaluating how the laminate will behave under various loading conditions.

Finally, the Results Section presents the outcomes of the analysis based on the input data. It includes the ABD matrix, which details the laminate's extensional and bending stiffness, and a force/displacement vector that provides results corresponding to the selected scenarios.

The form also features submit and back buttons for user navigation and options to save and load model data, enabling users to manage and review their data efficiently. Overall, this form provides a comprehensive tool for analyzing the mechanical behavior of laminate materials as shown in **Figure 16**.

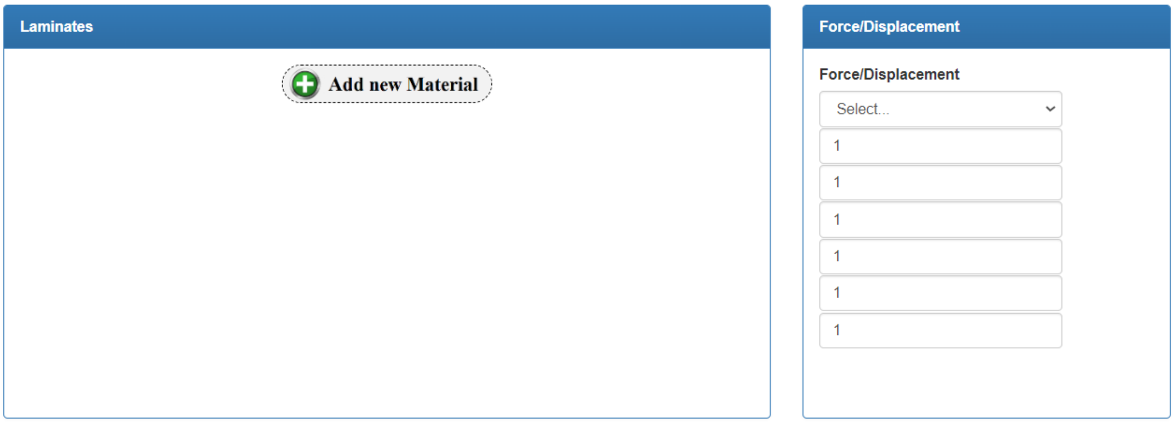


Figure 16. New material adding.

The code defines a directive for managing hybrid laminate materials in a web application, providing a structured interface for users to handle complex data related to materials used in various engineering and manufacturing applications.

The component's user interface is organized into several sections within a form. At the heart of the form is a panel dedicated to laminates, where users can view and manage different laminate materials. Each material is displayed with its associated layers represented by images. Users have the option to perform multiple actions on these materials, including editing, duplicating, and removing them. Additionally, they can rearrange the order of materials within the list.

To facilitate these operations, the form incorporates buttons for each action, which trigger corresponding functions when clicked. These actions include opening modal dialogs for adding or editing materials, where users can input detailed information about the laminates. The user interface also allows for dynamic updates, such as moving materials up or down in the list, ensuring that users can easily adjust the order of their materials.

Another critical section of the form allows users to select between force and displacement values. Depending on the selection, the form updates its input fields to reflect the chosen measurement type. This section includes dropdowns and input fields for users to provide necessary data, such as the magnitude of forces or displacements.

Validation of user inputs is a key feature of the component. The code includes checks to ensure that all required fields are filled out correctly and that the data provided is in the correct format. If any errors are detected, users are notified, and further actions are prevented until the issues are resolved.

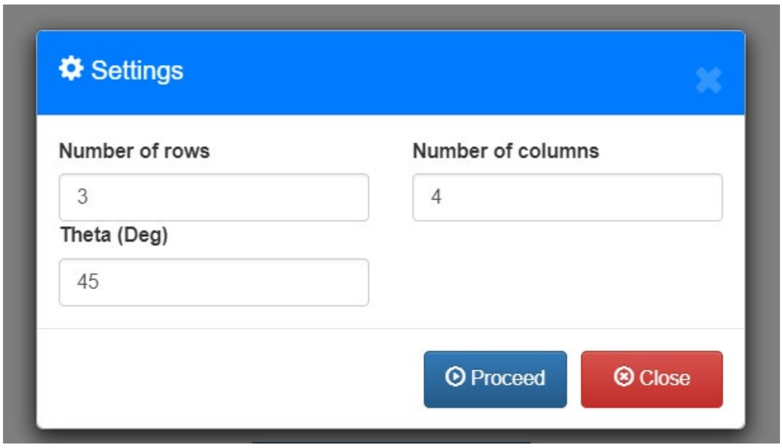
Once the inputs are validated, the component performs calculations using various services. These services compute complex matrices and vectors based on the laminate data provided by the user. The results of these calculations are then displayed in a scientific notation format, making it easier for users to interpret the data.

The component also supports interaction with external services and utilities. For instance, image resources are used to visually represent the different layers of the laminates, and utility functions are employed to handle tasks like cloning objects or displaying error messages. Modal dialogs are used to present additional information or options to users, enhancing the overall interactivity of the component.

In summary, the directive provides a comprehensive solution for managing hybrid laminate data. It integrates various features and functionalities, including user input management, data validation, calculation, and result presentation, all while maintaining a user-friendly interface. This setup ensures that users can efficiently manage and analyze laminate materials in a structured and interactive environment.

6.4. Interlock

This section describes an interface designed to manage and configure fiber and resin settings in composite materials. The goal is to provide users with an intuitive way to input, modify, and save key parameters for these materials, such as mechanical and physical properties [20]. By using a modal interface, users can seamlessly interact with the settings, ensuring the correct data is input for simulations or material configurations, as shown in **Figure 17**.



Number of rows: 3

Number of columns: 4

Theta (Deg): 45

Figure 17. Matrix dimension interlock.

The interface is divided into two sections: one for resin settings and one for fiber settings. Each section includes input fields that allow users to enter values related to specific properties of the materials, such as modulus, Poisson's ratio, and shear modulus for resin, as well as fiber strength and stiffness for the fiber section. These input fields are automatically generated based on predefined settings for fiber and resin materials, ensuring that users can work with a consistent set of parameters.

Once the user enters the required values in the input fields, they can save their configurations. The entered values are then collected and converted into structured data that is used for subsequent processes like simulations or material design evaluations. This system ensures that the data input by users is properly organized and readily available for any necessary calculations or analysis. Additionally, the input fields are designed with validation rules, ensuring that users cannot enter values outside of the allowed range, which helps maintain the accuracy and reliability of the data, as shown in **Figure 18**.

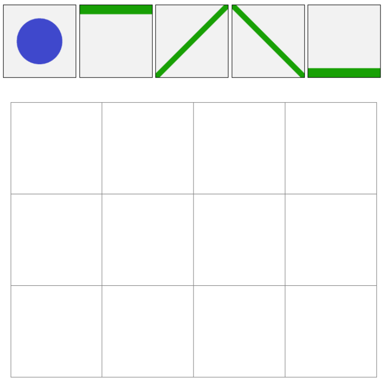


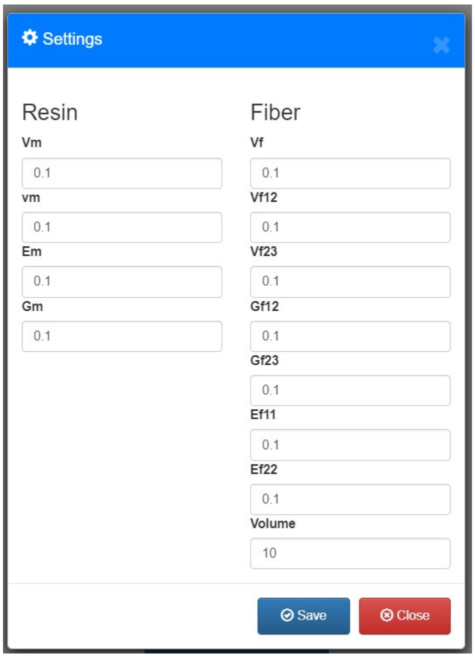
Figure 18. Interlock puzzle.

The resin and fiber settings presented to the user are drawn from predefined settings, ensuring that the available options are relevant and tailored to the specific materials being used. This allows for flexibility in managing different types of composite materials and their associated properties. Users can easily adjust settings based on the requirements of their project, facilitating experimentation with different material configurations.

By offering an interactive and user-friendly way to manage fiber and resin properties, this interface improves both efficiency and precision in handling material configurations. The ability to dynamically generate input fields and validate user input ensures that the tool is adaptable to various use cases, making it a valuable asset in material science and engineering environments where precise control over material settings is crucial. Through real-time interaction and structured data collection, the interface simplifies the process of configuring and experimenting with composite materials.

This section describes an interface designed to configure grid-based settings, focusing on inputs such as the number of rows, columns, and the angle of rotation, known as “Theta”. The interface is presented in a modal format, allowing users to input these key parameters for the grid system, which can then be used in subsequent processes such as simulations or design configurations.

The user is provided with three primary input fields within the modal: one for the number of rows, another for the number of columns, and a third for the angle of rotation (Theta). Each input field comes with built-in validation, ensuring that users enter valid numbers. For instance, the fields for rows and columns are restricted to integers greater than or equal to one, while the Theta field allows for precise decimal input with a high degree of precision, as shown in **Figure 19**.



Resin
Vm
0.1
vm
0.1
Em
0.1
Gm
0.1

Fiber
Vf
0.1
Vf12
0.1
Vf23
0.1
Gf12
0.1
Gf23
0.1
Ef11
0.1
Ef22
0.1
Volume
10

Figure 19. Resin and fiber parameters selection.

Once the user enters their desired values, they can proceed by clicking the save button, which triggers the configured settings. The input values for rows, columns, and Theta are then gathered and stored for use in generating a grid layout or performing calculations that depend on these parameters. This layout data is then sent to a higher-level context, where it can be applied to create or modify grid systems in accordance with the user's input.

The default values for the grid settings are set to three rows, four columns, and a Theta of 45 degrees, providing a standard configuration that the user can modify as needed. By allowing flexibility in adjusting these parameters, the interface supports a wide range of configurations, enabling users to tailor the grid setup to their specific requirements.

The interactive design of the interface ensures that users receive immediate feedback on their input, and any incorrect or invalid entries are prevented from being processed. This helps maintain accuracy in the configuration process, allowing users to confidently adjust settings and experiment with different grid layouts.

By offering a clear, user-friendly interface for adjusting grid parameters, this section enhances the overall efficiency and accuracy of the configuration process. Users can quickly and easily define key settings for rows, columns, and Theta, and these values are seamlessly integrated into the broader context for further use in simulations or design systems. The intuitive layout and real-time interaction make this interface an effective tool for managing grid-based configurations in a variety of technical applications [20].

The 2.5D, or so-called 3D, interlock composite is a special fabric combination in which mechanical properties are closely related to its structure. The geometry of the interlock is complex and the number of possible structures is infinite. Fabric architecture depends on the extraction, crimp, and size of the fibers. It is made up of a system of two woven and interconnected yarns, straight yarns, and weft (or filling) yarns [20]. Interlacing caused by bending is called "tow crimp".

The shapes in the "puzzle" represent the filler element and the bounder (interlock fiber) of the structure, as shown in the figure below (**Figure 20**), in 2D view.

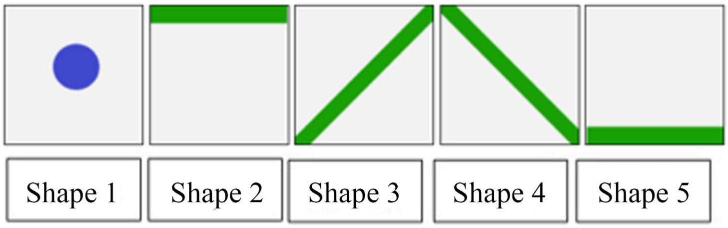
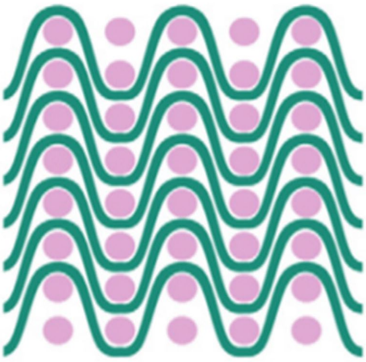
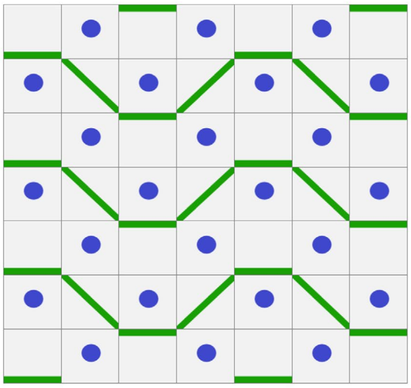


Figure 20. shapes of elements of interlocking.

Shape 1: the filler element; Shape 2: upper change in direction (horizontal); Shape 3: the reflected interlock; Shape 4: the incident interlock; Shape 5: lower change in direction (horizontal).





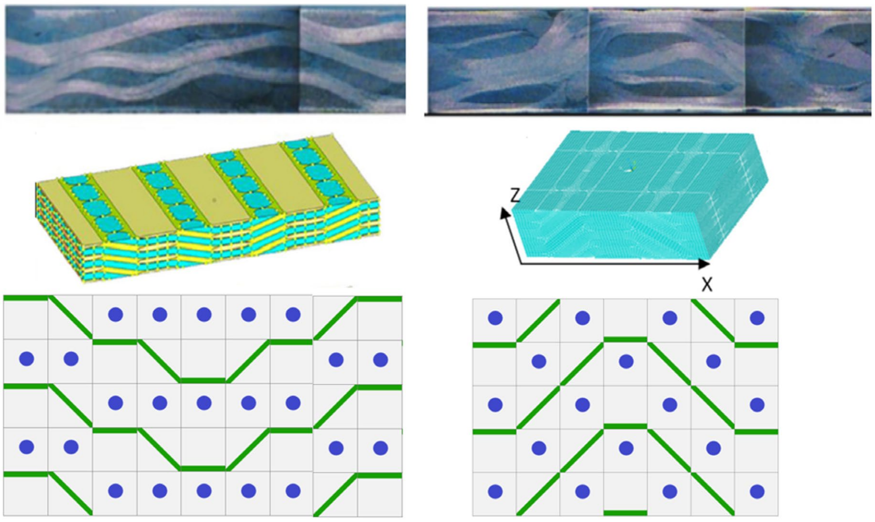


Figure 21. Interlock examples.

2.5D angle closing fabrics can be divided into two types: thick fabrics and fabrics from layer to layer. This type of combination is characterized by its intricate structure. The cell unit consists of a warp weaver and weft fibers attached to 90˚ plane (xy). A 2.5D angle-woven joint includes warp that binds to warp straight cords by attaching warp threads. Warp wires can be tied to different depths where different layout arrangements could be used to produce a wide range of these types of compounds. Tight fabric is a multi-layered fabric where warp weavers move from one piece of fabric to another, holding all the layers together. The layer-to-layer fabric is a multi-layer fabric that the warp weavers move from one layer to the nearest layer and back. A collection of warp woven together holds all the layers of fabric [20]. In addition, complex geometry, fractional volume fraction, cable volume, and inclination angle of warp strands are not able to allow the structural properties of specific applications. In other words, designers can replicate the efficient performance of fabrics for the necessary mechanical properties, as shown in **Figure 21**.

Construction Testing

Tests should be performed to validate the shape suggested by the user. These tests are:

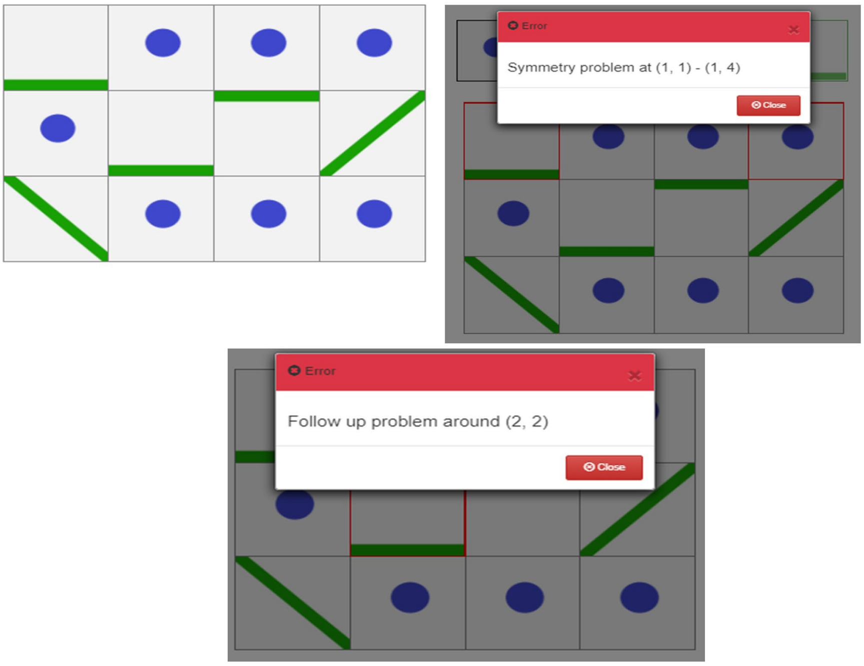
- Follow-up test.
- Pattern test.
- Symmetry test (not obligatory).

These tests obtained by logical geometrical construction of the 2.5D interlock structure are constraints that define whether the design (structure) from the user is valid-applicable [20].

As named, the function of the tests is to check if there is a continuity of structure and if there is a possibility to add a shape to the puzzle. For example, Shape 3 cannot be similar to Shape 2’s incident and reflected interlock at the same time. The puzzle should be symmetrical if the user is constructing a symmetrical shape.

If not, the test to check symmetry can be deactivated. Shapes 2, 3, and 4 cannot be repeated vertically. Shapes 2 and 4 cannot be repeated horizontally. Shapes 2 and 4 should be separated only by Shape 3. Shape 3 cannot follow Shape 1 or 2. At least one instance of Shape 1 should separate Shapes 2 and 4 horizontally. In addition, there are other structural and geometrical conditions and boundaries related to the shape, as shown in **Figure 22** [20].

Figure 22. Validation tests.



Symmetry problem at (1, 1) - (1, 4)

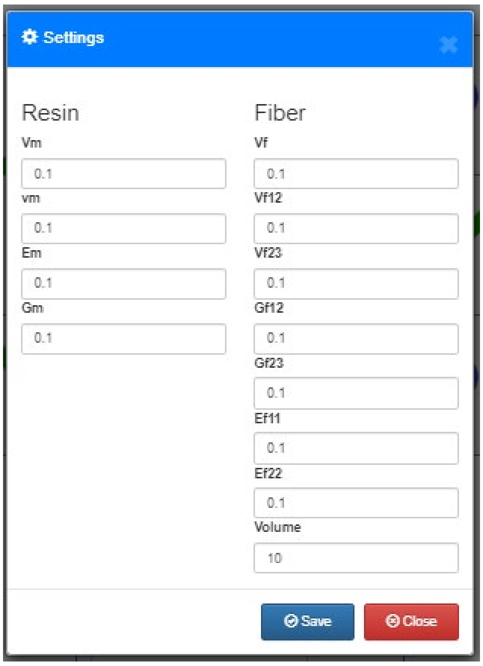
Follow up problem around (2, 2)

Figure 23. Non-valid geometry example.

Note that all these conditions/boundaries are tested automatically by the JavaScript code created by the author. Moreover, if there is any structural mistake made by the designer (user), the code will identify the mistake in relation to the mentioned tests. In addition, the code will specify exactly the place of the error with the messages: “Pattern problem at column 2—Follow up problem at (2,2)—Invalid structure” for the designer have to reshape its structure to continue and the sentences “No pattern problem—No follow up problem—Valid structure”, as shown in **Figure 23**.

6.5. Analytical Modelling

First, the user should fill fibers and matrix parameters such as Young’s and shear modulus (E_i,G_i) and fiber volume fraction (Vf) as shown in **Figure 24** as a user interface to the analytical modelling.



Resin
Vm: 0.1
vm: 0.1
Em: 0.1
Gm: 0.1

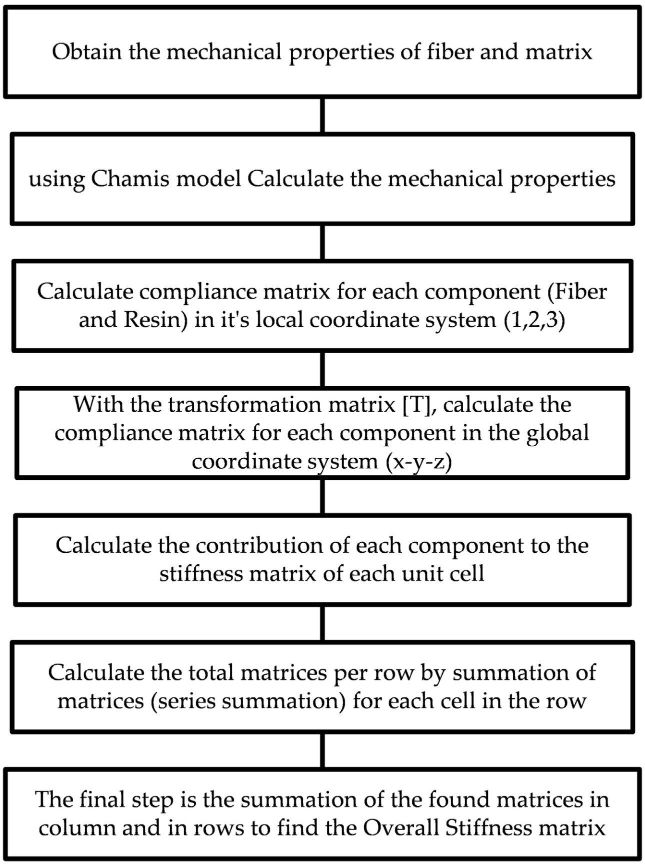
Fiber
Vf: 0.1
Vf12: 0.1
Vf23: 0.1
Gf12: 0.1
Gf23: 0.1
Ef11: 0.1
Ef22: 0.1
Volume: 10

Figure 24. Interlock Fiber and resin parameters.

Then, to calculate the characteristics of the key elements and the contribution rate, they make a macroscopic layer and subsequently the whole unit cell. The current approach creates expressions at the micro level with the aim of calculating more representative volume fractions of a group of elements in the layer to improve the elastic stiffness speculation compared to existing analytical modelling methods that use the 2.5D woven composite as a composite component containing layers of unidirectional elements (which are fibrous tows encased in resin). The new modelling approach creates expressions that discretize the unit cell into elements

To calculate the macroscopic stiffness of the whole unit cell, it must be broken down or discretized to the microscale. The microscale looks at the individual elements that make a cell and then the layers that make up the macroscopic unit cell. Therefore, the unit cell undergoes the first level discretization into layers, then the second level discretization into the individual elements that make up the layer. An element can be an individual, filler, binder, or matrix region within a layer. Having determined the stiffness of each element, the stiffness of whole row can be found. Once the stiffness of all rows is known, the model formulates and calculates the stiffness of the whole unit cell and finally outputs the elastic constants. The prediction of the unit cell or macroscopic stiffness begins with the calculation at the micro scale (the constituent elements within a cell).

The algorithm applied is as follows:



6.6. Sandwich

This code provides a detailed user interface for managing sandwich materials in a web application. It presents a form divided into several sections, each dedicated to a different layer of the sandwich structure: the upper layers, the core, and the lower layers, as shown in Figure 25.

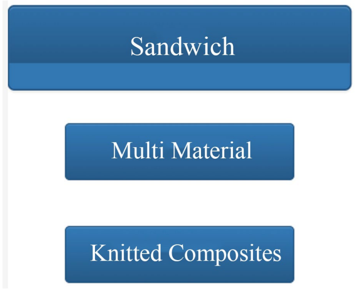


Figure 25. Sandwich selection.

In each section, users can view and interact with a list of materials. Each material entry displays an image and includes buttons for actions such as editing, duplicating, and removing. There are also options to move materials up or down in the list, allowing users to rearrange the layers as needed. The interface dynamically updates to reflect changes in real-time.

The form also includes input fields for force and displacement values, which are essential for calculating the sandwich's stiffness and force-displacement characteristics. When users submit the form, the application validates the inputs and performs calculations to generate a stiffness matrix and force-displacement vector. These results are then displayed to the user, provided the input data is correct.

Additionally, the code manages a model that stores all the data related to the materials and their properties. This model provides methods for adding new materials, editing existing ones, and handling various interactions within the form. The system ensures that all necessary data is collected and validated before performing any calculations or submitting the information.

Overall, this section of the code focuses on providing an interactive and user-friendly interface for managing complex sandwich structures, allowing users to perform detailed material management and real-time calculations efficiently, as shown in Figure 26.

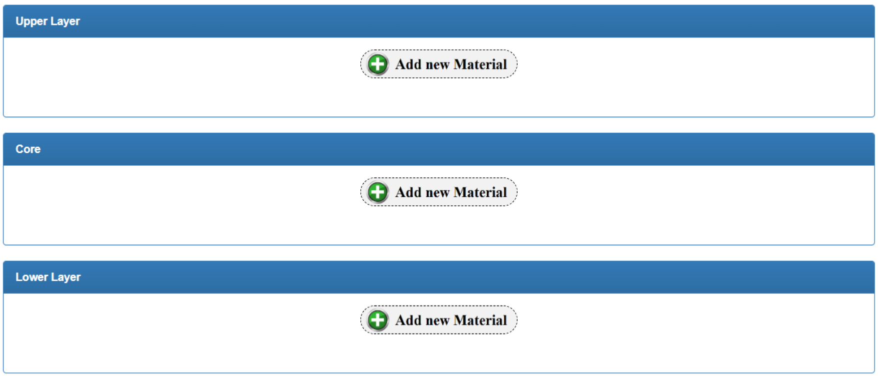


Figure 26. Sandwich material construction.

This section of code represents a complex form layout with various interactive components. It organizes the interface into multiple panels, each managing different types of materials and their properties.

The form begins with a panel dedicated to the "Upper Layer" where users can view and manage materials. Each material's details, including images and properties, are presented. The form allows users to edit, duplicate, remove, or reorder these materials. These actions are controlled through buttons, each with associated icons and tooltips to guide the user.

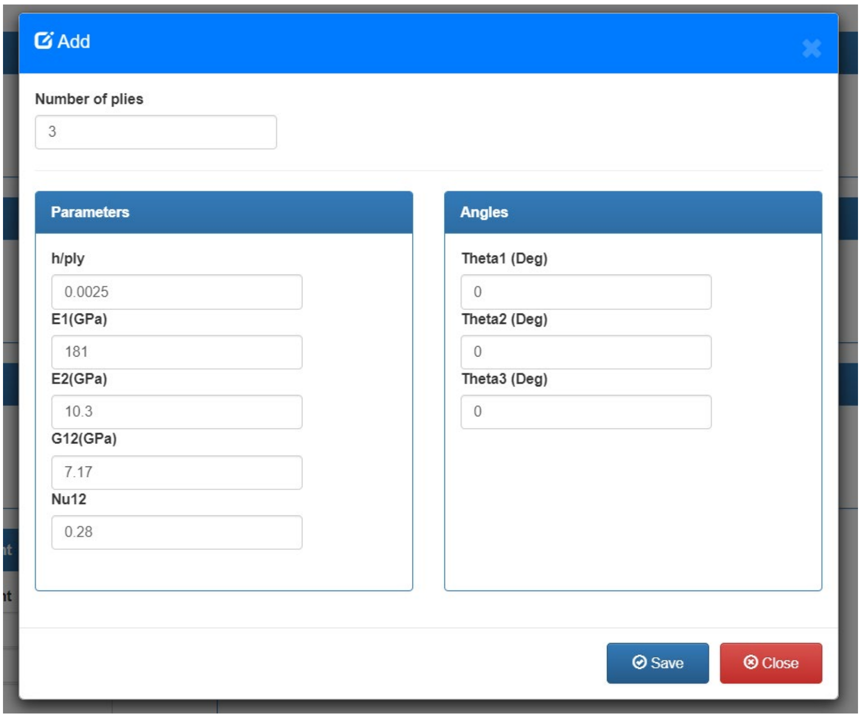
Following the "Upper Layer" panel, there is a similar setup for the "Core" and "Lower Layer" sections. Each section mirrors the structure of the "Upper Layer" panel, providing options to manage the materials in these layers similarly. The layout ensures consistency in how materials are displayed and interacted with, making it easier for users to handle each layer's data.

In addition to the material management panels, the form includes a "Force/Displacement" panel. This section features a dropdown selection and input fields for numerical data, allowing users to specify force and displacement values. The layout supports various inputs and selections, providing a comprehensive interface for entering and managing this type of data.

Towards the end of the form, there is a results section that displays matrices related to the calculations performed based on user input. This section only appears if certain conditions are met, such as the availability of results data.

Finally, the form provides navigation buttons for submitting the form or going back, ensuring users can easily complete or revise their input. Below the form, additional components for saving and loading the model's state are included, enhancing functionality by allowing users to persist and retrieve their data as needed.

The structure of this form is carefully designed to handle complex interactions and data management tasks, ensuring a smooth user experience while maintaining the ability to manage and review multiple layers of material data and their associated properties as shown in Figure 27.



Number of plies: 3

h/ply: 0.0025
E1(GPa): 181
E2(GPa): 10.3
G12(GPa): 7.17
Nu12: 0.28

Theta1 (Deg): 0
Theta2 (Deg): 0
Theta3 (Deg): 0

Figure 27. Parameter filling.

6.7. Knitted Composites

The user -whether a student or a fellow researcher—can design a knitted composite. To construct—as he or she likes—the whole structure of composite from the beginning and to make the whole process "user friendly", the process took the shape of building a puzzle by a simple "**click and drop**" game as shown in the figure below (Figure 28).

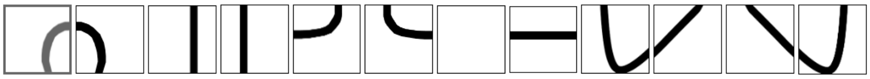


Figure 28. Puzzle structure for knitted composites.

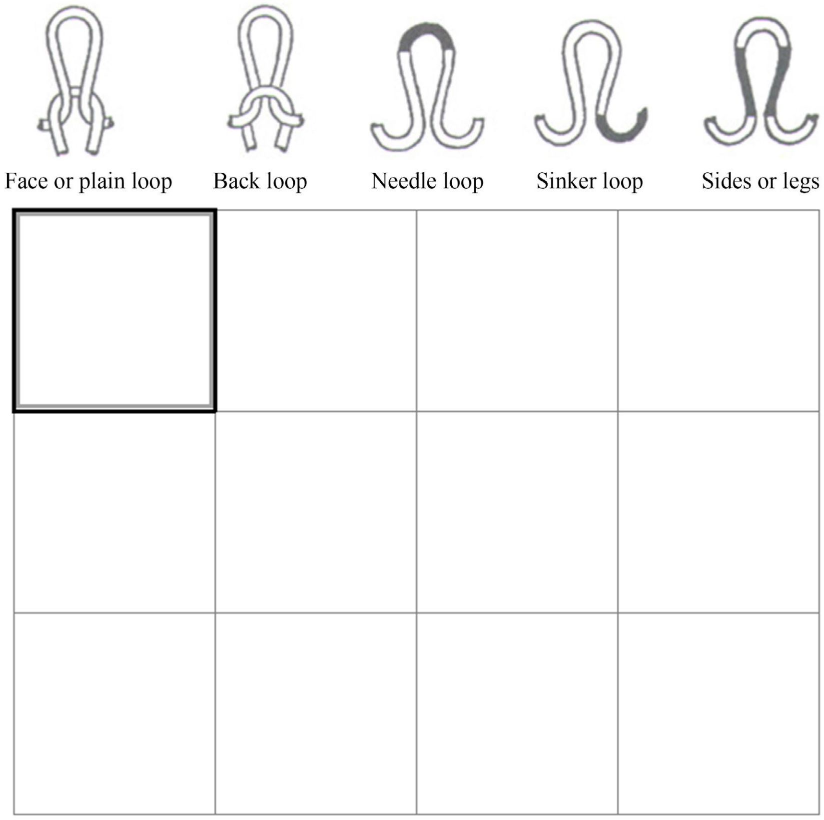


Figure 29. Puzzle settings for knitted composite.

To start from the beginning, the user is free to choose the settings of the puzzle, such as dimension, *i.e.*, number of columns and rows, and the number of fibres, as shown in **Figure 29**.

7. Results

To assess the performance of the mobile simulation tool, we conducted benchmarking against industry-standard desktop solutions. A series of simulations were performed on both platforms, measuring key performance indicators such as computational accuracy, execution time, and resource utilization. The mobile tool demonstrated an average computation time of seconds for stress analysis, compared to minutes and sometimes hours in a desktop environment. Accuracy deviations were found to be within 2% when compared to ANSYS results. However, memory consumption on mobile devices was significantly lower, making it more suitable for lightweight applications. A summary of the benchmarking results is presented in **Tables 1-4**.

With the purpose of validating the developed modeling technique, the results were compared with experimental data published in open literature. A comparison between the results of present modeling, experimental data, and previously developed multi-scale modeling was conducted. As case studies, different mechanical properties were considered, and the results are shown in **Tables 1-4**, each of which shows a clear comparison between the experimental results and the presented algorithm and codes; all results are obtained from [19] [20].

Table 1. Ex Young's modulus comparison. [19]

Type of composite	Experimental	Author's Codes
Glass/epoxy	56	55.14
E-Glass 1200 tex MY750/HY917/DY063 epoxy	29.2	29.152

Table 2. Gxy Shear modulus comparison. [19]

Type of composite	Experimental	Author's Codes
Glass/epoxy	3	2.57
E-Glass 1200 tex MY750/HY917/DY063 epoxy	5.83	5.648

Table 3. Comparison between experiments and predictions for tension moduli (GPa). [20]

Type of composite	E1 (GPa)	E1 (GPa)—Author
Glass/epoxy	14.5	14.38
E-glass/epoxy—II	60.3	60.57

Table 4. Analytical results of the iso-strain model compared to numerical results for the composites 69. [20]

Type of composite	E1 (GPa)	E1 (GPa)—Author
3 SHM	28.98	28.69

The tables compare experimental and numerical (author's codes) results for various composite materials, focusing on different mechanical properties. In **Table 1**, the Young's modulus values for Glass/Epoxy and E-Glass 1200 tex MY750/HY917/DY063 epoxy show minimal differences between experimental and predicted results, with errors of approximately 1.53% and 0.16%, respectively, indicating the accuracy of the predictive model. **Table 2** evaluates the shear modulus (Gxy), where the predicted values slightly underestimate the experimental ones, particularly for Glass/Epoxy (14.3% error), while the difference is smaller for E-Glass 1200 tex MY750/HY917/DY063 epoxy (3.12% error). In **Table 3**, tension modulus (E1) comparisons for Glass/Epoxy and E-Glass/Epoxy-II demonstrate close agreement between experimental and predicted values, with minor discrepancies of 0.83% and 0.45%, respectively. Lastly, **Table 4** compares iso-strain model analytical results with numerical results for a 3 SHM composite, revealing a negligible deviation (0.98% error). Overall, the comparisons suggest that the author's predictive models closely align with experimental findings, with small variations primarily in shear modulus estimations. The results validate the reliability of the numerical models, making them useful for predicting composite material behavior with reasonable accuracy.

8. Conclusions

This study has explored the growing potential of mobile devices to simulate composite materials, a crucial area in modern engineering. The findings indicate that mobile platforms are becoming more capable of performing sophisticated simulations traditionally reserved for high-performance desktop systems. This is primarily due to rapid advancements in mobile technology, such as increased processing power, enhanced graphics capabilities, and improved connectivity. These improvements allow mobile devices to simulate complex materials like fiber-reinforced composites, opening up new possibilities for engineers and researchers who require flexibility and mobility in their work environments.

One of the key innovations presented in this research is the development of a web-based software solution using HTML coding. This platform-agnostic approach ensures that users can access simulation tools across different devices, such as smartphones and tablets, without sacrificing accuracy or performance. The ability to simulate composite materials on mobile devices democratizes access to these advanced tools, making it easier for professionals in various fields, such as aerospace, automotive, and civil engineering, to perform simulations anywhere and at any time. This flexibility significantly enhances collaboration, decision-making, and overall productivity.

The investigation into mobile hardware and software architecture revealed that with the appropriate optimizations, mobile devices could indeed handle the computational demands of tasks like finite element analysis (FEA) and stress testing. Through techniques such as parallel processing, algorithmic optimization, and the use of cloud-assisted solutions, it is possible to overcome the limitations of mobile platforms, enabling them to perform detailed simulations of composite structures. The software architecture proposed in this research takes advantage of the modern capabilities of mobile GPUs, ensuring that performance is not compromised even on devices with lower processing power compared to desktop computers.

In addition to technical optimizations, this paper also reviewed existing mobile applications capable of performing similar tasks. While some commercial and open-source applications provide basic simulation functionalities, there is still significant room for improvement in terms of handling the complexities of composite materials. This research contributes by identifying gaps in existing software and proposing strategies to enhance performance and usability in mobile-based simulation tools. For example, the integration of adaptive meshing techniques and real-time feedback allows for more responsive simulations, providing users with valuable insights as they adjust parameters or modify designs.

The development of the mobile simulation software is complemented by future possibilities in the field. As mobile hardware continues to evolve, the capacity for these devices to perform even more complex simulations will only increase. Upcoming improvements in mobile processing power, memory, and graphics performance will further reduce reliance on cloud computing, making real-time, on-device simulations more practical. Additionally, the integration of Augmented Reality (AR) into simulation tools presents a transformative opportunity. AR can enhance how simulations are visualized, allowing users to interact with and manipulate composite structures in a more intuitive and immersive manner.

The future scope of mobile-based simulation also points to the development of more efficient algorithms specifically optimized for mobile platforms. Research into lightweight numerical methods and approximation techniques tailored for mobile devices will play a significant role in ensuring that these devices can perform accurate and high-fidelity simulations without excessive computational overhead. By continuing to refine these algorithms, mobile devices will become even more capable of handling complex simulations in a variety of fields, from materials science to structural engineering.

In conclusion, this study has demonstrated that mobile devices, with the right architectural and computational optimizations, are capable of performing high-level simulations of composite materials. This breakthrough has significant implications for industries that rely heavily on composite materials, as it allows for more flexible and accessible simulation processes. By enabling engineers and researchers to perform simulations on the go, the software developed in this research has the potential to revolutionize how composite materials are modeled, analyzed, and optimized. The advancements made in this study lay the foundation for further innovations in mobile simulation technology, ensuring that mobile devices will continue to play an increasingly vital role in engineering and scientific research.

Conflicts of Interest

The authors declare no conflicts of interest regarding the publication of this paper.

References

[1] Ghlaim, K.H. (2024) Woven Factor for the Mechancial Properties of Woven Composite Materials. *Journal of Engineering*, **16**, 6012-6027. https://doi.org/10.31026/j.eng.2010.04.22

[2] Hull, D. and Clyne, T.W. (1996) Frontmatter. In: *An Introduction to Composite Materials*, *Cambridge Solid State Science Series*, Cambridge University Press, 1-6.

[3] Shadish, W., Cook, T. and Campbell, D. (2004) Quasi-Experimental Designs for Generalized Causal Inference.

[4] Sultan, M.S., Khan, M.A., Khan, H. and Ahmad, B. (2022) Pathways to Strengthening Capabilities: A Case for the Adoption of Climate-Smart Agriculture in Pakistan. *APN Science Bulletin*, **12**, 171-183. https://doi.org/10.30852/sb.2022.2021

[5] Nielsen, J. (2012) Usability 101: Introduction to Usability. Nielsen Norman Group.

[6] Mallett, R.D.C., Stroeve, J.C., Tsamados, M., Landy, J.C., Willatt, R., Nandan, V., *et al.* (2021) Faster Decline and Higher Variability in the Sea Ice Thickness of the Marginal Arctic Seas When Accounting for Dynamic Snow Cover. *The Cryosphere*, **15**, 2429-

2450. https://doi.org/10.5194/tc-15-2429-2021

[7] Akindote, O.J., Adegbite, A.O., Dawodu, S.O., *et al.* (2023) Innovation in Data Storage Technologies: From Cloud Computing to Edge Computing. *Computer Science & IT Research Journal*, **4**, 273-299. https://doi.org/10.51594/csitrj.v4i3.661

[8] Es-haghi, M.S., Anitescu, C. and Rabczuk, T. (2024) Methods for Enabling Real-Time Analysis in Digital Twins: A Literature Review. *Computers & Structures*, **297**, Article 107342. https://doi.org/10.1016/j.compstruc.2024.107342

[9] ANSYS Inc (2024) ANSYS Composite PrepPost: Comprehensive Analysis for Composite Structures.

[10] Daniel, I.M. and Ishai, O. (2006) Engineering Mechanics of Composite Materials. Oxford University Press.

[11] Altair Engineering (2024) OptiStruct: The Industry-Leading Solution for Structural Optimization.

[12] Altair Engineering (2024) Classical Laminate Theory for Composite Design.

[13] MSC Software Corporation (2024). Nastran and Patran: Leading Simulation Tools for Aerospace and Automotive.

[14] MSC Software Corporation (2024) Finite Element Analysis in Aerospace Composites.

[15] Dassault Systèmes (2024) Abaqus FEA: Nonlinear and Dynamic Material Simulation.

[16] Lemaitre, J. (1996) A Course on Damage Mechanics. Springer.

[17] TexEng Software Ltd (2024) Wisetex: Textile Modelling Software for Composites.

[18] University of Nottingham (2024) TexGen: Open-Source Software for Textile Geometrical Modelling.

[19] Kaddaha, M.A., Younes, R. and Lafon, P. (2021) Homogenization Method to Calculate the Stiffness Matrix of Laminated Composites. *Eng*, **2**, 416-434. https://doi.org/10.3390/eng2040026

[20] Kaddaha, M.A., Younes, R. and Lafon, P. (2022) New Geometrical Modelling for 2D Fabric and 2.5D Interlock Composites. *Textiles*, **2**, 142-161. https://doi.org/10.3390/textiles2010008