Title: Efficient Simulation for Hybrid Overmolded Composite Lattice Structures: A Combined Implicit and Explicit FEA Approach

Meghana Kamble[1], Jeffrey Wollschlager[2,3,] Robert Jopson [2], Christopher Oberste[1]
[1] WEAV3D Inc, Norcross, Georgia, USA
[2] Altair Engineering, Inc, Troy, Michigan, USA
[3] University of Washington, Seattle, Washington, USA[3]

WEAV3D's composite lattice structures, referred to as "Rebar for Plastics," offer a novel solution for producing lightweight hybrid overmolded composite parts. Their unique architecture introduces modeling challenges that require tailored Finite Element Analysis (FEA) strategies. Over time, we have developed and refined a progression of FEA workflows—starting with RVE-based modeling, advancing to discrete lattice (explicit) models, and culminating in a multiscale (implicit) approach—each with trade-offs in accuracy, scalability, and setup efficiency.

RVE-based modeling, while accurate, is often slow and challenging to scale due to the need to develop and solve simplified CAD models of composite tapes and surrounding plastics for each design iteration. Discrete modeling improved accuracy by explicitly capturing tape geometry yet proved computationally expensive and impractical for large or complex parts.

To balance speed and detail, we developed a multiscale approach that uses homogenized stiffness properties via BDF (Bulk Data File) format assigned directly to lattice regions. This enables efficient full-part simulation, with dehomogenization in post-processing to extract stress and strain data in the tapes and overmolded plastic. Though less detailed than discrete models, it supports rapid design iteration and early-stage optimization. The multiscale modeling approach is validated using against three-point bending experimental data.

This paper presents a hybrid workflow: multiscale modeling enables quick stiffness evaluation and guide design iteration, while discrete modeling is applied to critical designs for detailed stress and strain analysis. The hybrid workflow is demonstrated on a heavy truck component to illustrate how the combined approach improves computational efficiency in part-level analysis of lattice-reinforced composites.

The paper will explain the principles behind the hybrid workflow and how to implement it in practice, without diving into the mathematical or coding details addressed in earlier publications. The focus is on providing engineers with a clear and usable framework for modeling hybrid overmolded composite structures using a balanced combination of implicit and explicit methods.

1. Introduction

Advancements in thermoplastic composites have significantly enabled the development of lightweight, high-performance structural components across automotive, transportation, aerospace, sporting goods, and other high-performance industries. Hybrid overmolded structures, integrating continuous fiber-reinforced unidirectional tapes with injection or compression-molded thermoplastics, offer selective reinforcement, high stiffness-to-weight ratios, and production scalability [1]. However, their heterogeneous architectures challenge traditional FEA methods, designed for isotropic or ply-based composites. These methods assume layer homogeneity and fail to capture the complex mechanical behavior of hybrid lattices with varying tape materials, spacing, and layer configurations.

WEAV3D Inc.'s Rebar for Plastics® technology exemplifies this hybrid approach, using woven composite lattices of thermoplastic prepreg tapes overmolded with polymers. These lattices act as structural reinforcements within the polymer matrix. By tailoring tape material, weave density (i.e. spacing between tapes), and layer count within a ply (Figure 1), engineers can potentially achieve up to 23–24% weight savings, 50–52% cost reductions, and closed-loop recycling of production scrap for automotive components like doors and beltline stiffeners, depending on design and baseline materials. In a case study involving a carbon fiber organosheet door component, lattices acted as structural reinforcements within the polymer matrix. By tailoring tape material, weave density (i.e., spacing between tapes), and layer count within a ply (Figure 1), the WEAV3D® equivalent design achieved up to 23–24% weight savings, 50–52% cost reductions, and enabled closed-loop recycling of production scrap, relative to the baseline organosheet design [2,3].

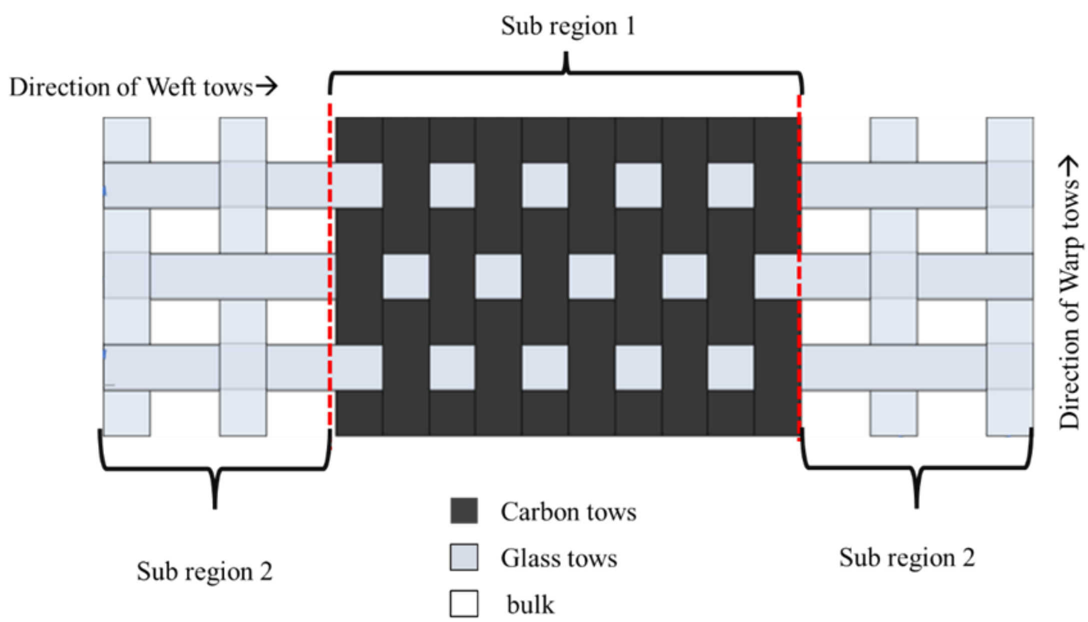


Figure 1 :Example of a Heterogeneous Lattice with Variable Tape Spacing and Tape Material

Accurately modeling the mechanical response of these heterogeneous lattices requires advanced simulation workflows capable of handling the numerous design variables involved. To address these challenges, our FEA strategy evolved through a series of progressively more capable approaches: starting with ANSYS RVE-based homogenization [4]; followed by Altair HyperWorks discrete (explicit) modeling [5]; and multiscale (implicit) modeling [6]. Each method aimed at simplifying initial FEA setup, support design parametrization, and accelerate optimization across a wide design space. The theoretical foundations of these approaches are presented in prior work [2,4-6], which readers are encouraged to consult for additional context. Understanding the progression of these methods and the limitations they aim to overcome provides important context for the development of a more robust, hybrid FEA workflow.

2. Background

Early attempts to model woven-fabric reinforced composites used the mosaic model, which idealizes the fabric as planar tiles of asymmetrical cross-ply laminates, calculating stiffness for the smallest repeat unit using classical laminated plate theory [7]. Similarly, the Fabric Geometry Model (FGM) treated fiber and matrix as composite rods within a Representative Volume Element (RVE), employing stiffness-averaging to derive a global stiffness tensor weighted by the volume fraction of each sub element. While effective, these models either oversimplify heterogeneity by ignoring crimp effects (mosaic) or requiring detailed crimp and volume fraction characterization (FGM). As composite design has expanded to include more open, engineered reinforcement architectures—such as tape-based lattice structures like WEAV3D's—more advanced modeling approaches are needed to accurately capture the geometry-driven mechanical response of these systems

We addressed these limitations via an RVE homogenization workflow developed in ANSYS [Figure 2]. An RVE CAD model of lattice geometry was created in ANSYS Material Designer to compute orthotropic properties, then assigned to the part in ANSYS ACP for stiffness optimization. Submodeling in ANSYS Mechanical extracted localized stresses in tapes and overmolded plastic. This requires precise knowledge of tape locations, which can be challenging for complex geometries. The RVE method was validated against three-point flexure tests on 6" × 6" flat panels, achieving an average modulus error of –4.4% (ranging from –14% to +6%) [4]. However, the process took ~23 minutes per design iteration due to manual CAD updates, coordination across multiple ANSYS modules, and submodeling complexity—making it impractical for rapid design exploration [4].

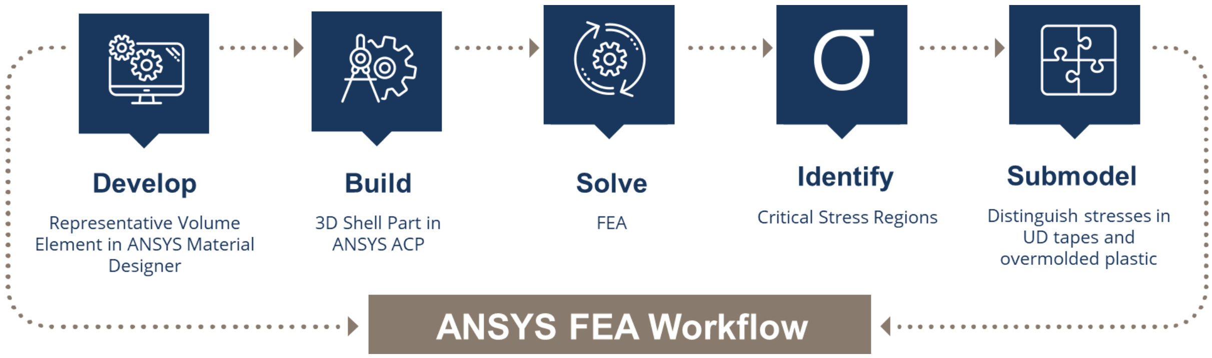


Figure 2 : ANSYS RVE Method Workflow [4]

Our next effort was an explicit modeling workflow in Altair HyperWorks (Figure 3). A Tcl script in HyperMesh parsed user-defined tow parameters (material, width, spacing, thickness, layers) from a text file, projecting tows as shell elements onto meshed geometry using a local coordinate system. The script generated a composite stack-up for static or dynamic load analysis in OptiStruct, reducing setup/solve time to ~9 minutes while achieving a +5.8% average modulus error (0.3–13.5%) in flat panel flexure tests [5]. Eliminating submodeling enhanced speed, but manual text file creation for each tape's location and properties was labor-intensive for large panels, and larger models significantly increased solve times, limiting scalability for complex parts. Furthermore, explicit modeling failed to generate tapes accurately for geometries with curved radii or sharp features [5].

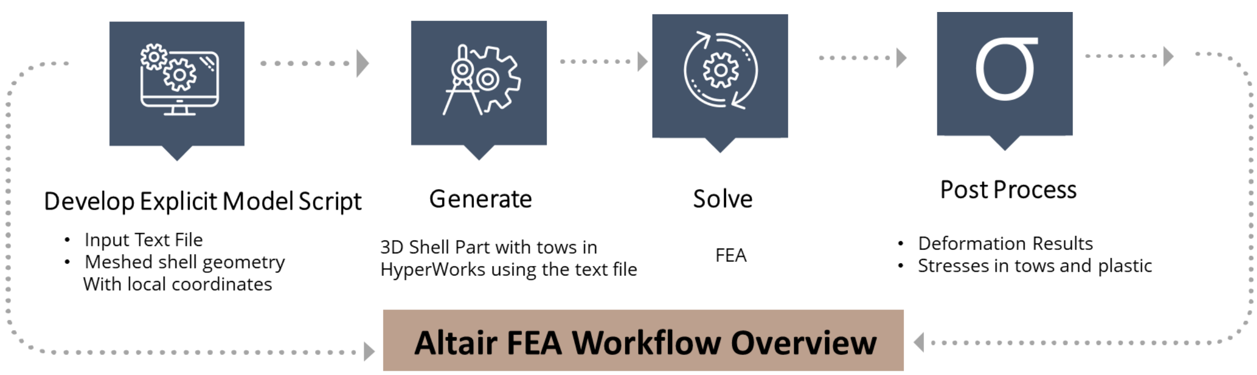


Figure 3: Altair Explicit Method Workflow [5]

To address scalability, a multiscale modeling approach based on Phase Average Modified Lamination Theory was implemented in Altair HyperWorks, enabling homogenization and an implicit de-homogenization step that maps global simulation results back to the tape scale without explicitly modeling individual tapes. This method achieved a 50% reduction in FEA time and ±15% numerical agreement across in-plane, bi-axial, and bending loads [6]. While this multiscale approach significantly improved automation and design throughput, we have identified limitations when applying it to CAD models that included a mix of element types—such as beams, and solids in addition to shell elements.

These trade-offs—such as the ANSYS RVE method's ability to support all element types and complex geometries but limited efficiency for iterative design; the Altair explicit method's fast turnaround but burdensome manual setup and geometric constraints requiring relatively flat parts without sharp curves; and the multiscale method's automation hindered by difficulties in handling mixed element types—underscore the need for a more flexible and scalable solution. The following section introduces the multiscale modeling workflow in detail, providing context for its role in enabling a hybrid modeling approach.

3. JPanel (Multiscale Modeling) Methodology

JPanel, a custom multiscale modeling tool, is a core component of the Altair HyperWorks framework, integrated with HyperWorks FEA Analysis tools (HyperMesh, OptiStruct, HyperView). Based on Phase Average Modified Lamination Theory, JPanel streamlines FEA by performing pre-processing (homogenizing lattice definitions to prepare for analysis) and post-processing (de-homogenizing results to extract tape and bulk plastic stresses). The workflow is illustrated in Figure 4.

This section provides a step-by-step guide for setting up JPanel within the Altair HyperWorks framework to perform finite element analysis (FEA) of composite lattice structures. The workflow detailed here focuses on JPanel's pre-processing and post-processing steps, including the setup of input parameters and the resulting outputs, ensuring users can efficiently evaluate composite lattice configurations for any end part.

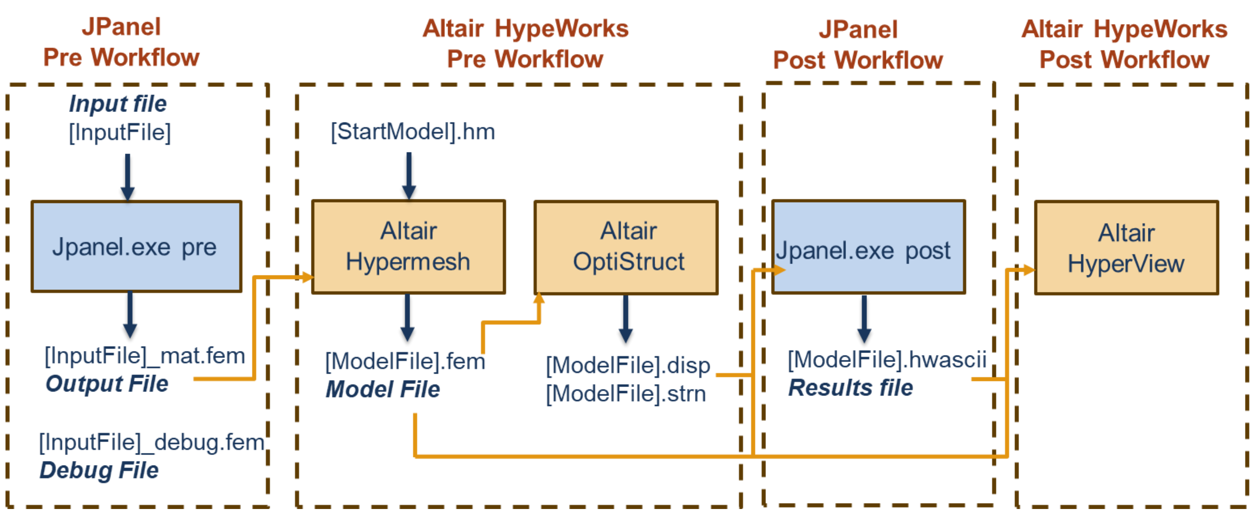
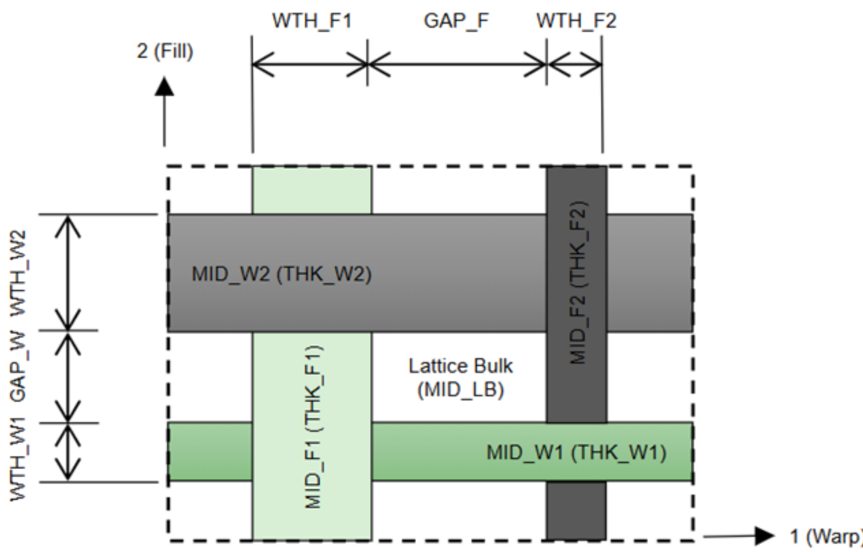


Figure 4 : JPanel (Multiscale Modeling) Workflow

Step 1: Create the Input text File for JPanel Pre Processor

Users begin by creating a input text file in bulk data format (BDF) containing LATTICE and PPANEL cards, as shown in Figure 5 and Figure 6, respectively. The LATTICE card defines tape properties, including Material (e.g., glass fiber/polypropylene, carbon fiber/polyamide), tape width, thickness, spacing, number of layers and stacking sequence. The PPANEL card specifies bulk plastic properties for the stiffened panel. These cards use parametric inputs, such as tape cover (tape width to spacing ratio), to define the lattice globally, eliminating the need to specify individual tape locations as required in the explicit model approach.



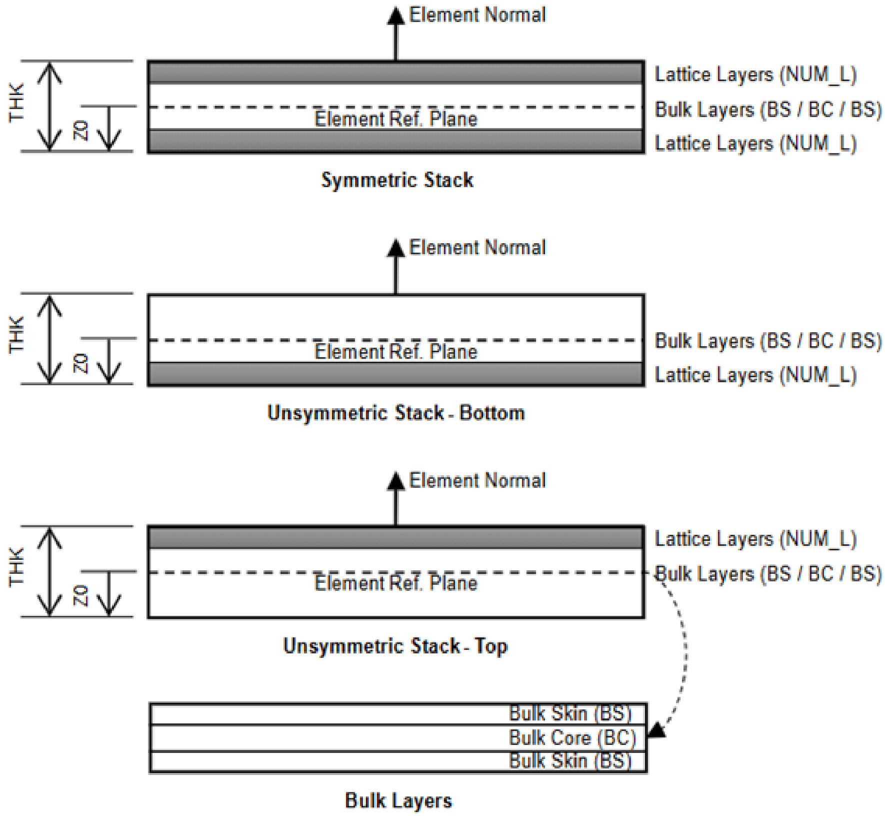
Input LATTICE CARD Format

(1)	(2) / (6)	(3) / (7)	(4) / (8)	(5) / (9)	(10)
LATTICE*	LID	LNAME	GAP_W	GAP_F	
	MID_W1	THK_W1	NUM_W1	WTH_W1	
	MID_W2	THK_W2	NUM_W2	WTH_W2	
	MID_F1	THK_F1	NUM_F1	WHT_F1	
	MID_F2	THK_F2	NUM_F2	WHT_F2	
	MID_LB				

LATTICE CARD Definitions

Field	Description
LID	Unique lattice identification number No default (Integer > 0)
LNAME	Lattice name No default (String)
GAP_W	Gap distance between weft lattice tows No default (Real)
GAP_F	Gap distance between fill lattice tows No default (Real)
MID_W1 MID_W2	Material identification number for warp1 and warp2 lattice tows respectively No default (Integer)
THK_W1 THK_W2	Thickness of warp1 and warp2 lattice tows respectively No default (Real)
NUM_W1 NUM_W2	Number of warp1 and warp2 lattice tows respectively overlayed on each other No default (Integer)
WTH_W1 WTH_W2	Width of warp1 and warp2 lattice tows respectively No default (Real)
MID_F1 MID_F2	Material identification number for fill1 and fill2 lattice tows respectively No default (Integer)
THK_F1 THK_F2	Thickness of fill1 and fill2 lattice tows respectively No default (Real)
NUM_F1	Number of fill1 and fill2 lattice tows respectively overlayed on each other No default (Integer)
WTH_F1 WTH_F2	Width of fill1 and fill2 lattice tows respectively No default (Real)
MID_LB	Material identification number for lattice bulk material No default (Integer)

Figure 5 : Pictorial Definition of Input Parameters for LATTICE Card Format and Definitions



Input PPANEL CARD Format

(1)	(2) / (6)	(3) / (7)	(4) / (8)	(5) / (9)	(10)
PPANEL*	PID	PNAME	TYPE		
	LID	NUM_L	SYM	BOT	
	THK	Z0	NSM	PHI	
	MID_BC	MID_BS	VOL_BS		

PPANEL CARD Definitions

Field	Description
PID	Unique panel identification number No default (Integer > 0)
PNAME	Panel name No default (String)
TYPE	Panel type enumeration Weav3D - Weav3D panel No default (String)
LID	Lattice identification number for the lattice used in the stiffened panel definition No default (Integer > 0)
NUM_L	Number of lattice layers used in the stiffened panel definition No default (Integer > 0)
SYM	Stiffened panel symmetry flag true - symmetric panel false - unsymmetric panel No default (bool true/false)
BOT	Stiffened panel bottom lattice flag No default (bool true/false)
THK	Thickness of stiffened panel No default (Real > 0.0)
Z0	Distance from middle surface to bottom of stiffened panel Typically set to -THK/2.0 No default (Real = -THK/2.0)
NSM	Non-structural mass of stiffened panel No default (Real)
PHI	Shear angle between warp and fill tape tows No default (Real)
MID_BC	Material identification number for bulk core material No default (Integer)
MID_BS	Material identification number for bulk skin material No default (Integer)
VOL_BS	Volume fraction of a single layer of bulk skin material to the total thickness of bulk material No default (Real >= 0)

Figure 6 : Pictorial definition of input parameters for PPANEL Card Format and Definitions

Step 2: Generate output (mat.fem) file

Users process the BDF file in JPanel Pre, which homogenizes the lattice to compute equivalent orthotropic properties. JPanel Pre outputs a mat.fem file, a solver deck in BDF format containing MAT2 cards (defining membrane, bending, transverse shear, and coupling behaviors) and PSHELL cards (specifying shell element properties), as shown in Figure 7 (right) [2]. The mat.fem file represents the lattice as a homogeneous shell, enabling efficient stiffness calculations.

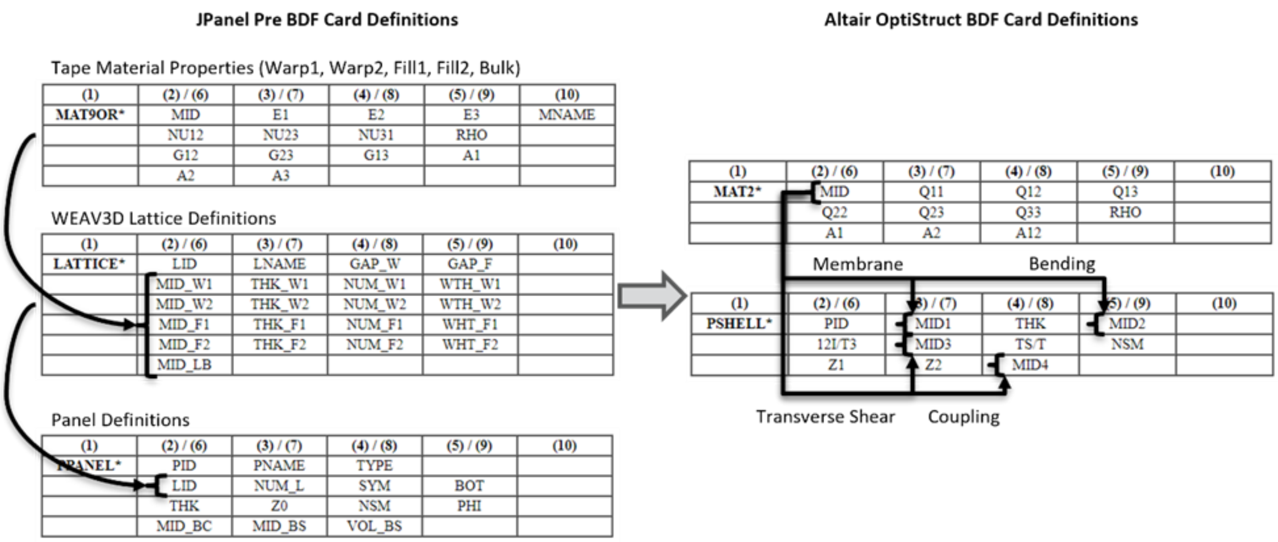


Figure 7 : Figure 7: BDF card Definitions for JPanel Pre and Altair OptiStruct [6]

Step 3: Setup and solve the FEA Model

The mat.fem file is imported into Altair HyperMesh as a solver deck, where users:

1. Mesh the part geometry
2. Assign PSHELL properties from the mat.fem file to the meshed elements.
3. Define boundary conditions and loads
4. The model is solved in Altair OptiStruct, producing an OPTI results file containing displacement and strain data, which quantifies the global stiffness of the composite lattice

Step 4: Process the results using JPanel Post

For stress analysis, users may run JPanel in post-processing mode to de-homogenize the OPTI results, calculating stresses and strains for tape and bulk plastic phases across all layers. The output is a HyperWorks ASCII results file, which can be visualized in Altair HyperView to display stress distributions in tapes and overmolded plastic.

JPanel's rapid pre-processing and ability to evaluate multiple lattice configurations make it a powerful tool for global stiffness analysis and early-stage design optimization. While it supports deflection and stiffness calculations for models with mixed beam, shell, and solid elements, its current limitation lies in post-processing—specifically, stress extraction is only available for shell-based models. Ongoing development aims to extend stress post-processing capabilities to mixed-element models, further expanding JPanel's applicability. JPanel is not constrained by design complexity, part geometry, or model size, or performance. The current version of JPanel is capable of handling sizable models, though memory management is not yet fully optimized. Ongoing development focuses on improving memory efficiency and expanding post-processing capabilities to support increasingly larger-scale simulations with higher element counts.

4. Experimental Validation (THREE-POINT BEND TEST)

4.1. Experimental Setup

To experimentally validate the proposed multiscale modeling approach in Altair HyperWorks, three-point flexural tests were conducted on flat composite plaques. The test setup is shown in Figure 8. Each plaque measured 152.4 mm × 152.4 mm with a nominal thickness of 2 mm. Four plaques were manufactured for each of the four lattice configurations summarized in Table 1.

Table 1: Lattice Design Configuration for Flexure Tests

<table>
<tr><th rowspan="2">Design No.</th><th rowspan="2">Molded Plastic Material</th><th rowspan="2">Weft Tow Material</th><th colspan="2">Weft Tow</th><th rowspan="2">No. of Lattice layers</th></tr>
<tr><th>No. of layers</th><th>Spacing (mm)</th></tr>
<tr><td>1</td><td rowspan="4">Braskem Ti4003F PP</td><td>Glass/PP (45 % Vf)</td><td>2</td><td>25.4</td><td>2</td></tr>
<tr><td>2</td><td rowspan="2">Carbon /PP (40 % Vf)</td><td>2</td><td>50.8</td><td>2</td></tr>
<tr><td>3</td><td>2</td><td>25.4</td><td>2</td></tr>
<tr><td>4</td><td>Mixed -Alternating Glass/PP (45 % Vf) & Carbon /PP (40 % Vf)</td><td>2</td><td>25.4</td><td>2</td></tr>
</table>

The lattice patterns were constructed using 25.4 mm-wide carbon and glass fiber tapes, with thicknesses of 0.16 mm and 0.25 mm, respectively. All configurations incorporated a single layer of glass fiber/polypropylene (GF/PP) warp tapes spaced at 50.8 mm center-to-center. The lattices were placed on both sides of an unfilled polypropylene core sheet. Plaques were compression molded to bond the layers and ensure structural cohesion, with weft tapes aligned along the primary loading direction during flexure.

Each specimen was loaded to failure or to the maximum deflection allowed by the fixture while crosshead force and displacement data were recorded. The flexural span was 63.5 mm. This setup enabled direct comparison between experimental results and FEA predictions, providing validation for the multiscale method.

Figure 8: Flexure Test Experimental

4.2. FEA Setup Three Point Bend Test:

The FEA setup was designed to replicate the experimental 3-point bend test, using the JPanel workflow described in Section 3. An input text file in bulk data format (BDF) containing MAT9OR, LATTICE, and PPANEL cards to define the composite lattice panel was created. The MAT9OR card specifies orthotropic material properties for the lattice tapes and bulk plastic. The LATTICE card defines tape properties, including material, width (25.4 mm), thickness (tape material dependent), spacing (50.8 mm), and layer count (single layer per side). The PPANEL card specifies the bulk plastic core's properties and thickness (2 mm).

For example, Design 1, consisting of GF/PP weft and warp tapes with a 50% tape cover (25.4 mm width, 50.8 mm spacing), a single lattice layer per side, and a polypropylene core sandwiched between, is defined in a BDF file Figure 9:

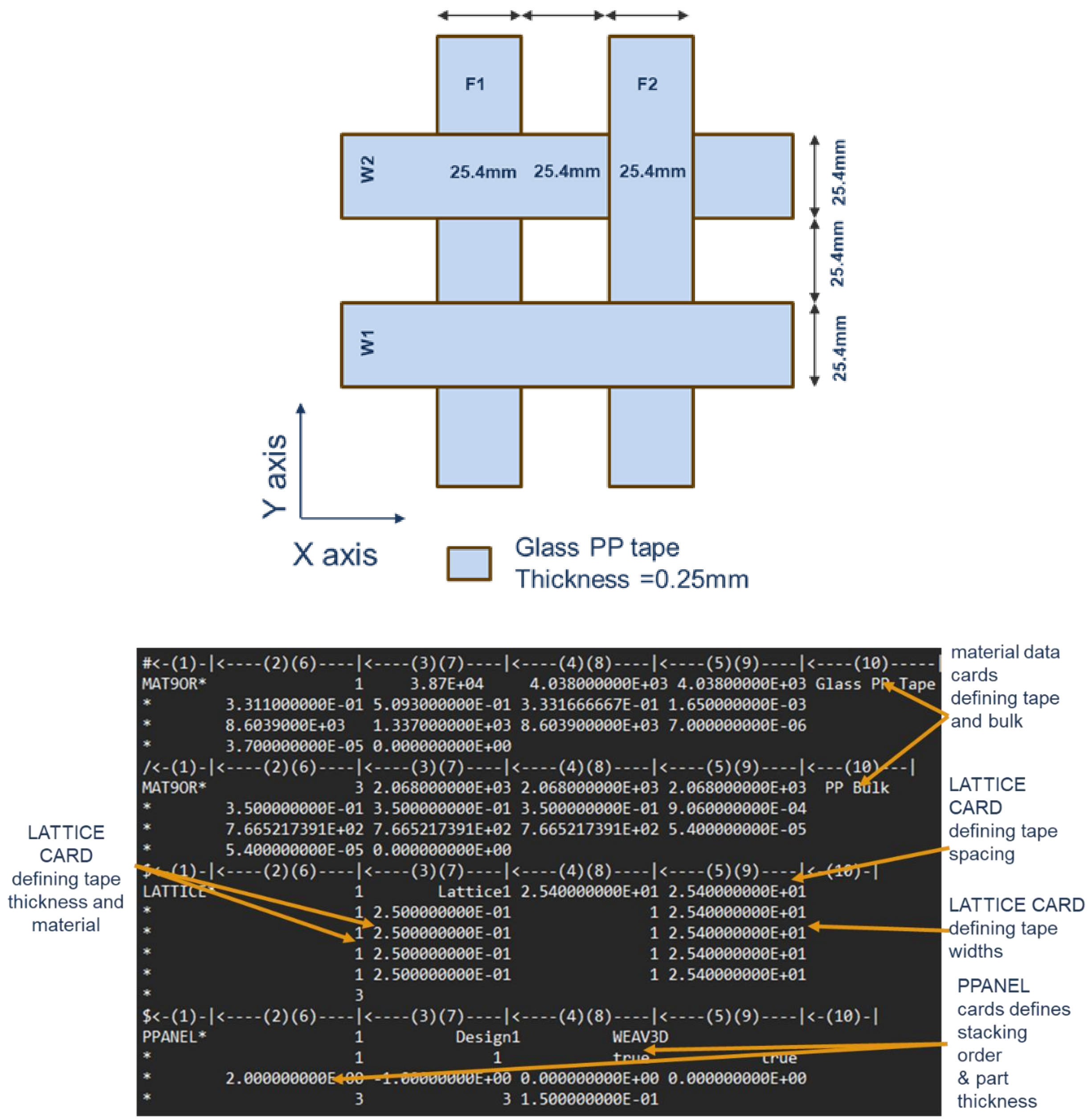


Figure 9 : Pictorial Depiction of the Lattice Configuration for Design 1 (Top) and the Corresponding BDF File Representation (Bottom)

Multiple designs (e.g., Designs 1–4) can be compiled in a single BDF file, each with unique LATTICE and PPANEL cards. JPanel pre-processing generates a mat.fem file (solver deck with MAT2 and PSHELL cards), which is imported into Altair HyperMesh. User meshed panel geometry can be assigned PSHELL properties from the imported mat.fem file. Material orientation is aligned with the experimental configuration, ensuring weft tapes (0°) correspond to the primary loading direction.

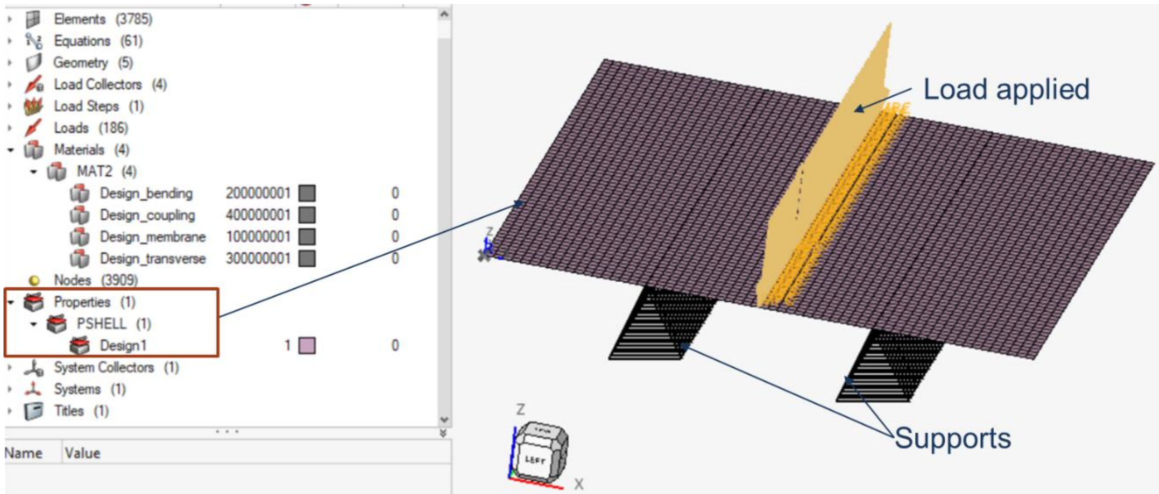


Figure 10 : FEA Setup of the Panel Flexure Test, Including PSHELL Property Assignment to the Part.

The meshed model along with the boundary conditions is then ready to be solved in Altair OptiStruct (Figure 10), producing an OPTI results file with displacement and strain data. Post-processing in JPanel de-homogenizes results to compute stresses and strains for tape and bulk plastic phases, generating a HyperWorks ASCII results file visualized in HyperView, as described in Section 3.

4.3. Discussion of Experimental Vs FEA Flexure Test Results

For each of the designs described above, the chord modulus for each sample was calculated according to a modified version of ASTM D790 that accounts for the wider specimen dimensions. The average chord modulus for each design is reported as the experimental chord modulus in Table 2. The predicted JPanel FEA modulus was calculated formulaically based on the applied load and simulated displacements, with percentage error between the experimental and Explicit method derived chord modulus calculated to determine degree of correlation. Our flexure test FEA predictions, summarized in Table 2, show that the JPanel Multiscale model exhibited good correlation with the experimental results, overpredicting the experimental modulus by an average of 0.75 % (-8 % to 3%).

Table 2: Flexure Test - Chord Modulus Comparison for Lattice Integrated Panels

Design No.	JPanel Chord Modulus (GPa)	Experimental Chord Modulus (GPa)	% Deviation
			JPanel vs. Experiment
Design 1	23.7	25.6	-8.1
Design 2	27.2	25.2	8.1
Design 3	53.2	52.9	0.4
Design 4	40.2	39	3.1

5. Hybrid FEA Approach

The following sub section outlines a decision-making framework for selecting the most effective FEA workflow based on part geometry and element types, followed by a case study that demonstrates the proposed hybrid approach.

5.1. Method Selection Framework for Hybrid FEA Workflows

To efficiently optimize composite lattice designs, engineers must evaluate key part characteristics particularly the mesh element types and the geometry of the lattice region to select a simulation strategy that balances computational efficiency with predictive accuracy. For parts composed exclusively of shell elements, the JPanel is the recommended tool for both pre-processing and post-processing. Its streamlined LATTICE and PPANEL card setup enable rapid iterative optimization, with built-in post-processing capabilities for evaluating stresses in both the tape and matrix phases (see Section 3).

However, if the CAD model includes mixed element types such as beam elements for contact modeling or solid elements combined with shells, in these scenarios, a hybrid workflow is recommended. Users can still leverage JPanel Pre for rapid homogenization and initial design iteration to identify optimized lattice configurations that meet part-level performance targets. Once a candidate design is selected, a second step is required to post-process stresses and strains in both the reinforcement and bulk phases. This step depends on the geometry of the lattice integration region:

- If the region is relatively flat, post-processing should be performed using explicit modeling in Altair HyperWorks. This method supports mixed element types and provides a scalable alternative to ANSYS RVE, especially for flat parts (Figure 3).
- If the lattice region exhibits significant curvature such as in motorcycle fenders or contoured automotive components, explicit modeling may not generate tows for these geometries. In these cases, the recommended approach is to combine JPanel Pre for initial lattice optimization with ANSYS RVE and submodeling for detailed stress analysis. Although this method has a higher setup time [5] it provides the necessary accuracy for curved geometries with mixed mesh types.

By aligning simulation strategy with both element types and lattice-region geometry, users can adopt the most efficient and accurate FEA workflow:

1. JPanel alone for shell-only parts,
2. JPanel Pre + explicit modeling for flat regions with mixed elements, or
3. JPanel Pre + ANSYS RVE approach for curved regions with mixed elements.

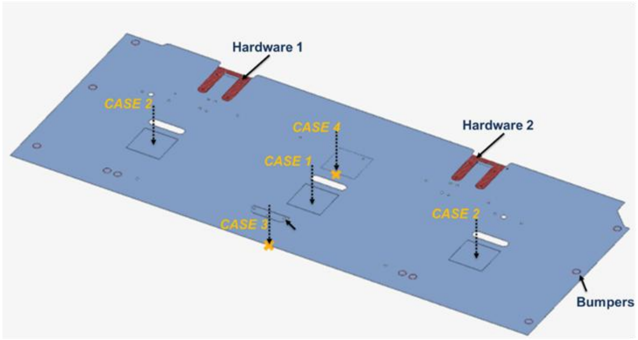
This structured methodology supports scalable, design-driven development of hybrid overmolded thermoplastic lattice composites across a wide range of geometries and use cases.

5.2. Case Study: Hybrid Simulation of an automotive part

This case study, conducted in partnership with Ascorium Industries, focuses on redesigning a bunk panel used in heavy trucks. The objective is to develop an integrated solution by combining a structurally optimized composite lattice, produced using WEAV3D® technology, with Ascorium's material system—CompoLite® HP. The goal of this design study was to develop a hybrid solution that meets or exceeds the mechanical performance of marine-grade plywood performance.

Although the bunk geometry is flat and modeled exclusively with shell elements, making it fully compatible with standalone JPanel pre- and post-processing, this case study is used to illustrate a broader hybrid workflow. In particular, it demonstrates how rapid design iteration with JPanel can be complemented with explicit modeling to validate detailed stress and strain results. This scenario also serves as a proxy for other parts with a combination of shell, beam and solid etc. elements, where JPanel post-processing is currently limited.

With this context in mind, we now turn to the FEA setup for the bunk case study. The simulation setup is illustrated in Figure 11.



$T_{X,Y,Z}$ = *Translation in X, Y, Z axis*
$R_{X,Y,Z}$ = *Rotation in X, Y, Z axis*

Bumpers : T_Z = constrained
Hardware 1,2 : R_Y = allowed ; constrained rest

Case 1:Pressure load in middle section
Case 2:Pressure load over an area both lateral sides of the panel
Case 3 :Point load at the frontal edge
Case 5 :Point load at the at the middle

Figure 11 : FEA Setup of Bunk Case Study

The users should start by setting up and solving baseline models to establish target performance benchmarks. The bunk panel was subjected to four representative load cases (Figure 11) using reference marine-grade plywood, and CompoLite HP without any lattice reinforcement. For each load case, the resulting Z-axis deflection at critical locations was measured.

As per the optimization strategy previously detailed in our SPE ACCE 2024 paper [2], begin with a first-pass design iteration by integrating a 50% cover (i.e. tape width to spacing ratio or space occupied by tape in a given area) homogeneous glass/PETG lattice with CompoLite® HP. Two configurations were evaluated: one featuring a single-layer lattice on both the top and bottom surfaces, and another with a double-layer lattice on both surfaces, sandwiching the CompoLite® HP core (Figure 12).

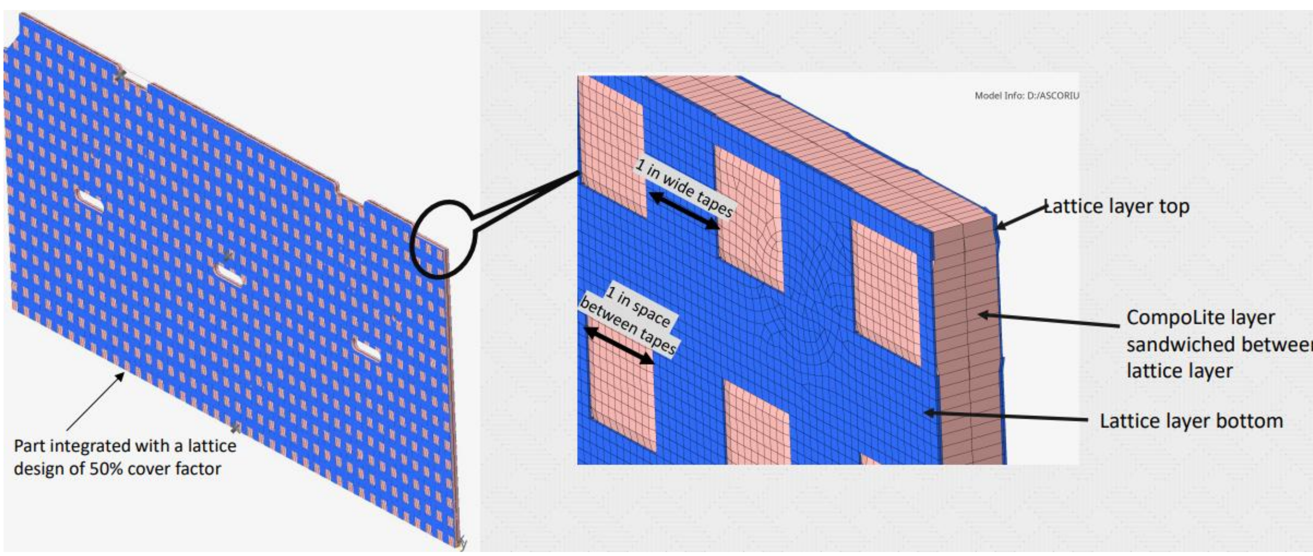


Figure 12 : Homogeneous Lattice Bunk Bed Design (Explicit FEA Model)

Table 3 summarizes the baseline model results, including deflection values for marine-grade plywood, standalone CompoLite®, and homogeneous single-layer and double-layer configurations.

Table 3 : Summary of FEA-Predicted Z-Axis Deflection for Cases 1 Through 4 (Baseline and Homogenous Designs).

Material Config.	Lattice Design	FEA Case 1 (in)	FEA Case 2 (in)	FEA Case 3 (in)	FEA Case 4 (in)
Marine Grade Plywood	N/A	0.78	0.25	2.27	0.66
CompoLite HP	N/A	1.73	0.46	3.53	1.56
PETG Glass tape + CompoLite HP	50% CF - Single Lattice Layer	1.08	0.29	2.37	0.94
PETG Glass tape + CompoLite HP	50% CF - Double Lattice Layer	0.84	0.24	1.95	0.74

This initial simulation allows the user to establish a baseline understanding of the structural contribution of a uniform reinforcement layout lattice defined by consistent tape material and spacing throughout the architecture before introducing additional lattice design complexity.

Once baseline models are solved, the next step involves generating input files for the JPanel Pre-processor. As shown in *Figure 13*, a single input file can be used to define both the homogenous lattice configurations. These lattice definitions can subsequently be imported into HyperWorks and assigned as material properties to the lower bunk geometry (section 3)

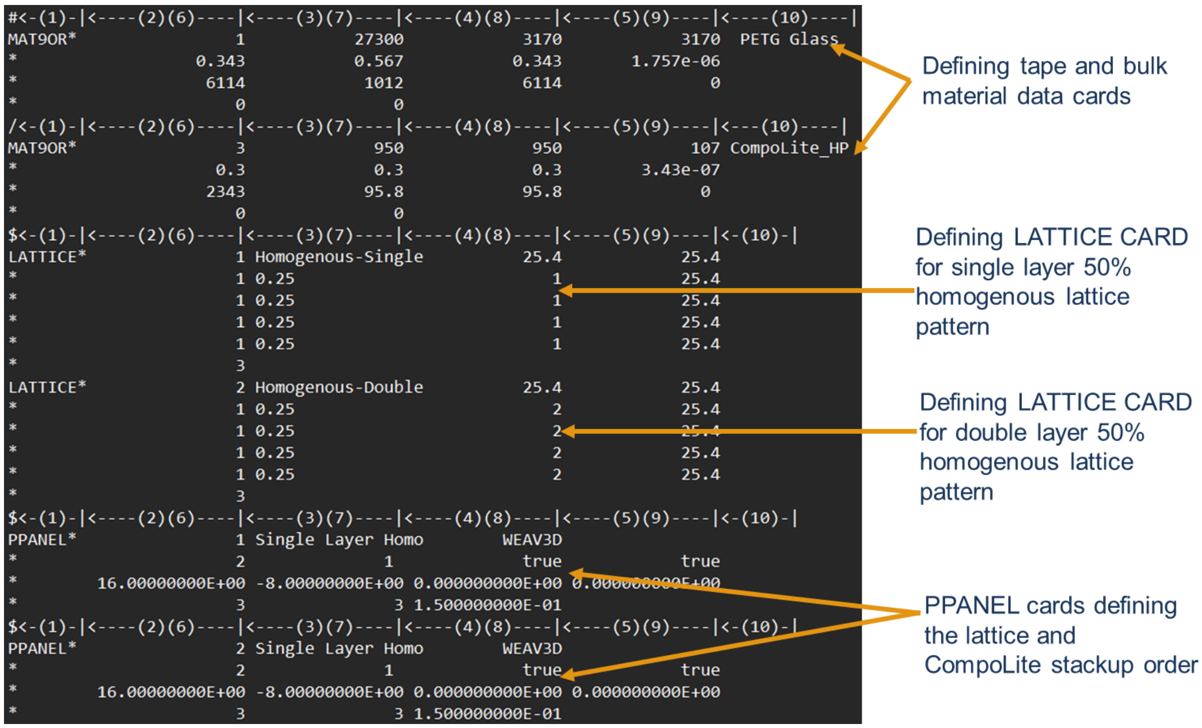


Figure 13 : Input File for JPanel Pre Processor – Homogenous lattice designs

After reviewing the baseline and first-pass simulation results, the user should begin developing a lattice placement strategy. At this stage, the focus should be on using JPanel Pre to generate and evaluate a wide range of lattice design iterations.

Although JPanel Pre supports the inclusion of multiple LATTICE and PPANEL definitions within a single BDF file, it is often necessary to update the input file, re-run JPanel Pre, and review the results after each design iteration. This iterative process helps the user understand the performance of the previous lattice configuration and refine the placement strategy accordingly. Based on the observed deflection trends, the user can adjust material selection, tow orientation, or spacing in the next input file, solve the model again, and continue narrowing toward a lattice design that meets the stiffness targets.

The goal at this stage is to quickly assess how each lattice design performs under the defined load cases and determine whether it meets or exceeds the stiffness performance targets established in the baseline analysis. Users are advised to focus solely on evaluating global deflection results during this phase, rather than stress and strain within the bulk or tape/tow materials.

In this case study, 23 lattice design iterations were explored to identify viable configurations that met the target stiffness benchmarks. While 4 candidate designs achieved a balanced trade-off between cost, weight, and performance, this paper will not detail each of those.

Instead, we present one representative candidate to illustrate the workflow and highlight key decision-making steps. HETRO 19 was integrated with CompoLite® HP as illustrated in Figure 14.

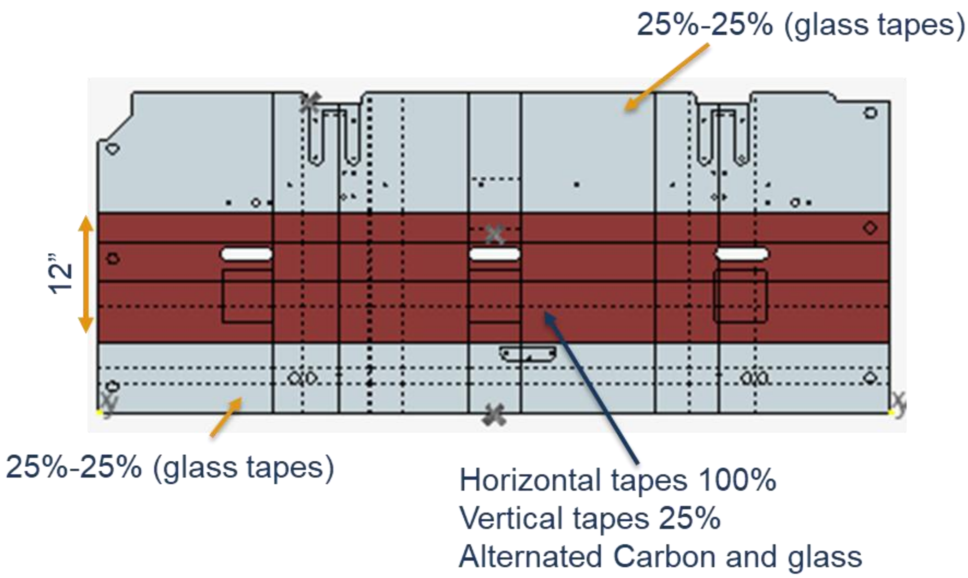


Figure 14 : HETRO Design 19 – Lattice Design Configuration

This design strategically varies lattice density—placing denser reinforcement in the central region of the panel, where FEA Cases 1, 2 and 4 are located l, while using sparser lattice regions elsewhere to optimize weight and material cost

Deflection results for HETRO Design 19, summarized in Table 4, show that this design outperforms standalone CompoLite® HP across all load cases. HETRO Design 19 demonstrated a 50.8% average improvement (measured as reduced deflection) compared to CompoLite® HP, and a 44.3% improvement over CompoLite® XP across all four FEA load cases. When compared to the marine-grade plywood benchmark, it delivers competitive performance: it performs better in FEA Cases 2 and 3, with deflection reductions of 12% and 9.3% respectively. In Cases 1 and 4, it slightly underperforms, with only a 1.3% and 6% increase in deflection. Overall, HETRO 19 demonstrates strong stiffness performance with targeted lattice placement, validating its potential as a viable and more efficient alternative

Table 4: Summary of FEA-predicted Z-axis deflection HETRO Design 19

Material Config.	Lattice Design	FEA Case 1 (in)	FEA Case 2 (in)	FEA Case 3 (in)	FEA Case 4 (in)
Marine Grade Plywood	N/A	0.78	0.25	2.27	0.66
HETRO Design 19	Varying cover Single Lattice Layer	0.79	0.22	2.06	0.70

Once target stiffness is achieved, stress validation becomes critical. Users should confirm that stress in tapes and bulk material remain below yield limits. If JPanel Post is compatible (shell-only models), this step is straightforward. If not, users must select between ANSYS RVE or explicit modeling.

In our hypothetical case, we select explicit modeling since the geometry is flat. This method is faster and more scalable than RVE for planar designs. The next step involves generating an input file for the explicit model solver that defines the exact tape layout material properties, widths, spacing, and orientation based on a local coordinate system. With explicit modeling, every tape's location is explicitly defined, allowing for a single post-processing step to extract accurate stress and strain data in both the reinforcement tapes and the surrounding bulk matrix [5]. This ensures the final design not only meets stiffness requirements but also remains within acceptable strength limits of the constituent materials under all applied load cases, confirming the structural integrity of the composite system

The heterogeneous lattice pattern for HETRO Design 19, generated using the explicit modeling method, is shown in Figure 15.

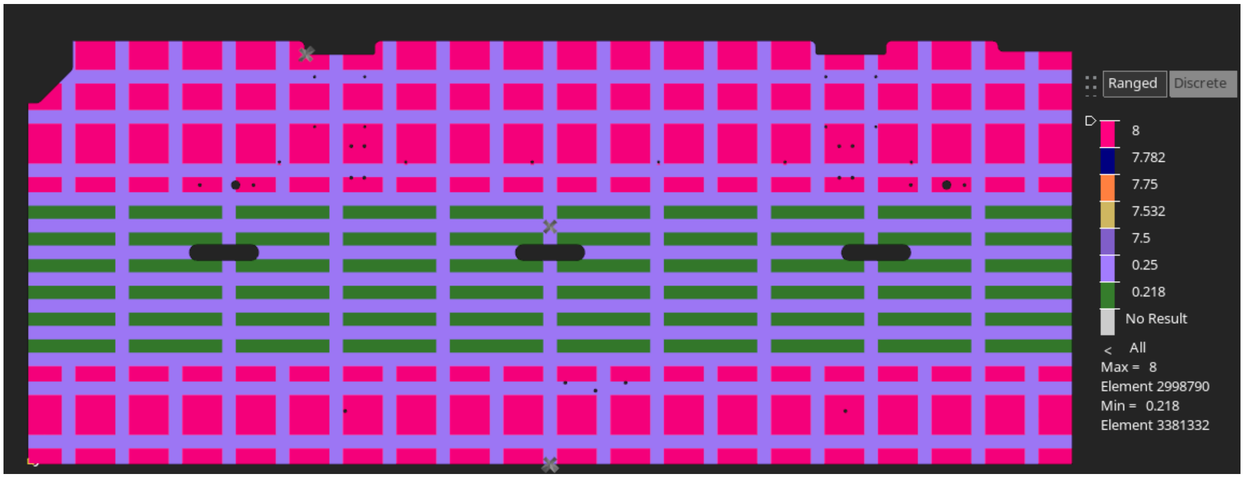


Figure 15 : Heterogeneous Design 19 – Lattice configuration output using explicit modeling.

The heterogeneous layout for HETRO Design 19, along with resulting stress fields, is shown in Figure 16. The maximum stress observed in the tapes across all load cases was ~45 MPa, while the highest stress in the bulk CompoLite® material was ~ 6 MPa. Both values remain well below the respective material strengths—885 MPa for the tapes and 16.5 MPa for the bulk, confirming the structural viability of this candidate design.

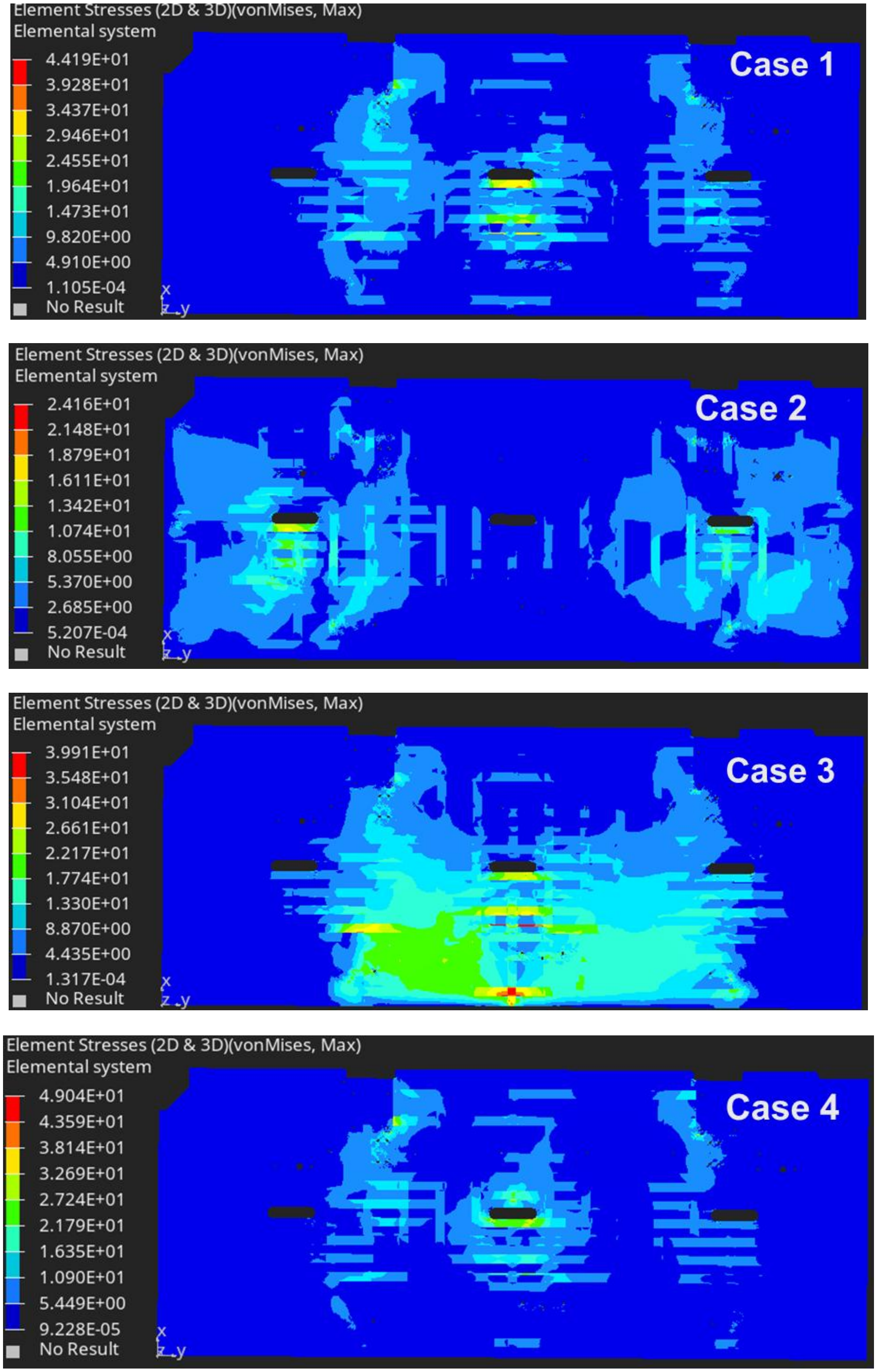


Figure 16: Post-processed Von Mises stress (MPa) in tapes and bulk, using the explicit model

In conclusion, this case study illustrates a practical hybrid workflow that balances computational efficiency and detail. Use JPanel Pre to explore a broad range of lattice design options quickly and efficiently, narrowing down to a few viable candidates based on global deflection results. Then, apply explicit modeling to those final designs to conduct detailed stress and strain analysis in both the tapes and bulk materials—ensuring yield criteria are not exceeded. This approach minimizes computational cost and manual effort by reserving the more labor-intensive explicit modeling step for only the most promising designs, while leveraging the fast iteration capabilities of JPanel for early-stage exploration

The chart in Figure 17 compares the normalized total bunk mass and cost between CompoLite HP and HETRO Design 19. While HETRO Design 19 demonstrates a modest increase in mass approximately 8% higher than CompoLite HP it also incurs a more significant cost increase of about 26%. These trade-offs are important considerations in design decision-making. The additional mass and cost reflect the inclusion of continuous tape reinforcement in HETRO 19's lattice structure, which enabled superior stiffness performance compared to standalone CompoLite HP. Ultimately, this result highlights the balance between structural performance gains and material/resource efficiency when adopting reinforced hybrid designs

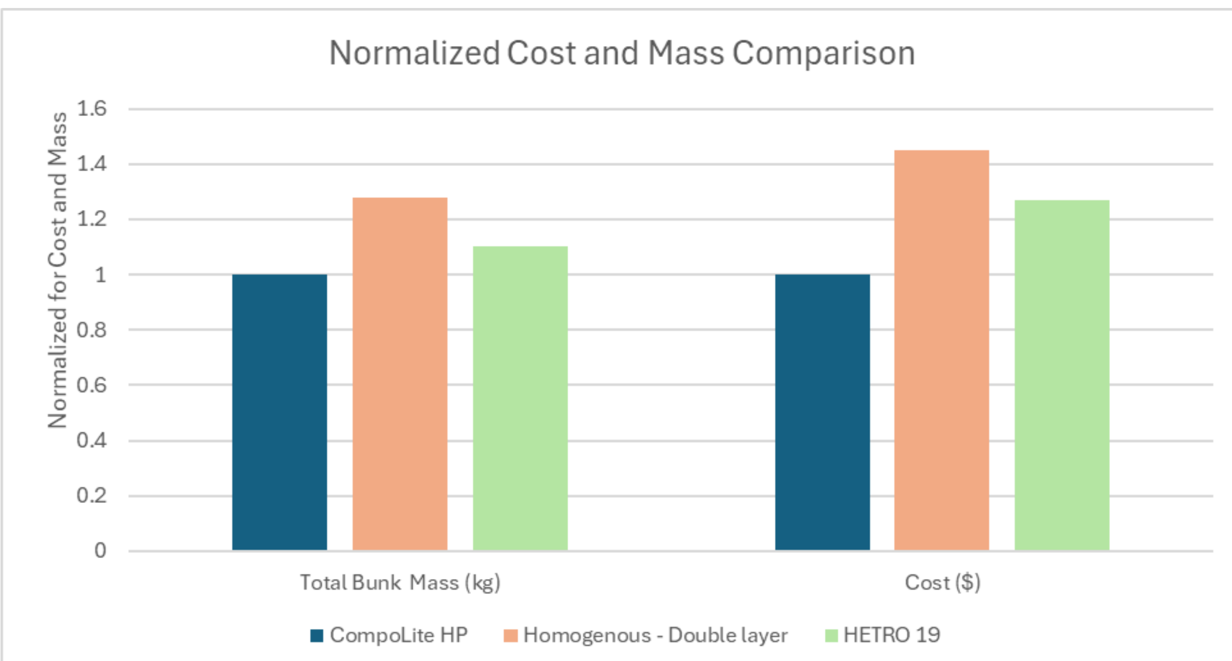


Figure 17: Normalized Cost and Mass Comparison CompoLite HP vs HETRO Design 19

6. Summary and Future work

This paper presents a hybrid FEA workflow for simulating hybrid overmolded composite lattice structures, leveraging the strengths of implicit (multiscale) and explicit modeling strategies. The multiscale approach, implemented via the custom JPanel tool, enables rapid stiffness evaluation and design iteration by homogenizing the mechanical behavior of composite lattices. Experimental validation using flexural tests on flat panels confirmed good correlation between FEA-predicted and measured modulus values, with an average error of less than 5%.

The proposed hybrid method was demonstrated on an automotive bunk panel, where a combination of early-stage deflection screening using JPanel and detailed stress analysis via explicit modeling provided both computational efficiency and high-fidelity results.

A representative heterogeneous lattice design (HETRO 19) achieved a ~50% improvement in deflection results over standalone CompoLite® HP and showed comparable or superior performance to marine-grade plywood under most load cases.

Despite these advancements, current limitations include the restriction of stress post-processing to shell-only models within JPanel. This limits the framework's applicability for parts involving mixed mesh types, such as solid and beam elements. Additionally, although the current implementation can handle large models, further improvements in memory management and solver efficiency are required.

Future work will focus on expanding JPanel's post-processing capabilities to support mixed-element models and improving the memory handling to accommodate larger-scale simulations.

7. References

[1] Rochford, H.S.; Brown, J.P. Manufacturing and Characterization of Overmolded Hybrid Fiber Reinforced Thermoplastic Composites. *Materials Science* 2020; 12: 112–120.

[2] Kamble, M., Jopson, R., Wollschlager, J., & Oberste, C. *Composite lattice reinforced part optimization with FEA: An automotive door component case study*. In *Proceedings of the SPE Automotive Composites Conference & Exhibition (ACCE) 2024*. WEAV3D Inc., Altair Engineering, Inc.

[3] Oberste, C., Kamble, M., & Tran, C. (*Structural enhancement of sustainable materials*. Presented at SPE TPO 2023, WEAV3D Inc.

[4] Kamble, M.; Oberste, C. Novel FEA Approach for Optimizing Composite Lattice Structures. In *CAMX 2022 Proceedings*; CAMX: Anaheim, CA, USA, October 17–20, 2022.

[5] Kamble, M.; Wollschlager, J.; Jopson, R.; Oberste, C. A Revised Finite Element Analysis Approach for Optimizing Composite Lattice Structures. In *CAMX Conference Proceedings*; CAMX*: Atlanta, GA, USA, October 30–November 2, 2023.

[6] Wollschlager, J. A., Oberste, C., Kamble, M., & Salviato, M. *A multiscale approach to the design and analysis of WEAV3D lattice structures*. In *Proceedings of ANTEC® 2025*. Altair Engineering, WEAV3D Inc., University of Washington

[7] T. Ishikawa and T. W. Chou, "One-dimensional micromechanical analysis of woven fabric composites," *AIAA Journal,* vol. 21, pp. 1714-21, 12/ 1983.