A Revised Finite Element Analysis Approach to Designs and Optimize Composite Lattice Reinforcements and Simulate the Mechanical Properties of Composite Lattice Reinforced Plastics

Meghana Kamble[1], Jeffery Wollschlager[2], Robert Jopson[2], Christopher Oberste[1]
[1] WEAV3D Inc, Norcross, Georgia, USA
[2] Altair Engineering, Inc, Troy, Michigan, USA

ABSTRACT

An earlier investigation, presented at CAMX 2022, identified deficiencies in the Finite Element Analysis (FEA) methods used for ply-based or isotropic materials which are unable to accurately capture the physical properties and behavior of hybrid overmolded structures based on unidirectional tape assemblies. In response, a new ANSYS FEA workflow was developed that combines the principles of Representative Volume Element (RVE) homogenization and submodeling. This workflow involves homogenization of composite lattice structures into RVEs that are then assigned to regions of a geometry to achieve part-level stiffness targets. Submodeling of critical stress regions is utilized to assess the distribution of stresses between the lattice and the molded plastic that makes up the RVE While functional, this RVE and submodeling workflow was time and labor intensive.

The current study presents a revised workflow that reduces manual intervention and the FEA setup time. This is achieved through explicit modeling using Altair HyperWorks, whereby the user identifies lattice designs to be tested for a part and inputs tape materials, tape spacing, layer count of the composite lattice into a script that is then explicitly modeled in the FEA without requiring an actual CAD model for various lattice designs to be tested. This method eliminates the need for submodeling, as stresses in tapes and overmolded material can be probed from the part level model after the FEA is solved. The benefits of this revised FEA workflow will be demonstrated through a case study of an automotive door component, educating attendees on a novel FEA workflow that can be applied to a range of hybrid overmolded composite structures.

Corresponding author: Meghana Kamble, meg.kamble@weav3d.com

Copyright 2023. Used by CAMX – The Composites and Advanced Materials Expo. CAMX Conference Proceedings. Atlanta, GA, October 30-November 2, 2023. CAMX – The Composites and Advanced Materials Expo

1. INTRODUCTION

Significant innovation in the field of thermoplastic composite manufacturing has resulted in the development of hybrid overmolded composite structures that combine continuous fiber composites with injection or compression molded compounds [1]. However, conventional FEA techniques designed for isotropic materials and ply-based composites are inadequate for accurately representing the physical properties and material behavior of these hybrids. This limitation is particularly evident in composite lattice structures constructed from woven, consolidated unidirectional tapes, which allow for variations in tape type and spacing within a specific layer (ply).

In our previous paper presented at CAMX 2022 [2], we introduced a novel ANSYS RVE method using the "homogenization" technique (Figure 1). This method involved using ANSYS Material Designer to generate Representative Volume Elements (RVEs) that accurately simulate the stiffness response of hybrid overmolded composite lattice structures. Homogenization enabled iteration through various lattice patterns to optimize part stiffness against target values.

Figure 1: ANSYS RVE Method Workflow

Homogenization assumes the uniform repetition of a Representative Volume Element (RVE) throughout a region, where the effective constitutive properties represent the entire lattice domain [3]. A lattice structure with a single tow material and spacing within a layer is termed a "homogeneous lattice," while a design with varying tow spacing and/or materials within a layer is called a "heterogeneous lattice." To characterize the material properties of a heterogeneous lattice, the designer divides the layer into smaller homogeneous sub-regions (Figure 2). An RVE is created for each sub-region using Material Designer (Figure 2). After establishing a library of RVEs, each is meshed and solved to obtain homogenized orthotropic material properties. These properties are then assigned to the respective sub-regions, resulting in a single ply with varying material properties.

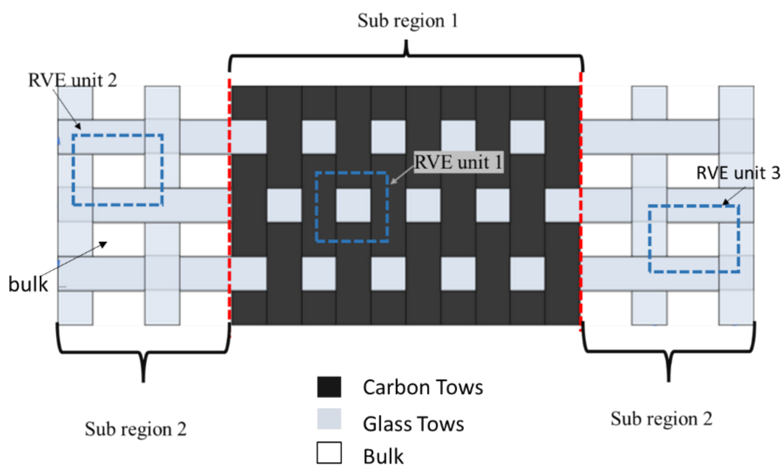


Figure 2: Example of a Heterogeneous Lattice with Variable Tape Spacing and Tape Material

The "coarse model" process, involving obtaining homogenous material properties from the RVE model, developing the part thickness in ANSYS ACP, and solving the FEA model in ANSYS Mechanical, has limitations in terms of parameterizing the spacing between UD tapes and distinguishing between stresses in the tapes and bulk plastic. It requires a new CAD model for each RVE design and further post-processing through submodeling. While it enables identification of stress-critical regions, the submodeling step involves physically drawing the tows in their exact location. The ANSYS RVE method allows for quick iterations to predict part level deformations but setting up new RVE units and analyzing stress distributions is labor-intensive.

This paper presents a new workflow using Altair HyperWorks to overcome the limitations of the previous ANSYS RVE methodology. The use of tcl scripts in HyperWorks simplifies the tow generation process and eliminates the need for submodeling to differentiate stress between the lattice and bulk plastic. Familiarity with Altair HyperWorks is required to implement this workflow.

2. METHODOLOGY

2.1. Explicit Model Script to Model Lattice Design

In Altair HyperWorks, the HyperMesh module is used to generate lattice designs through an explicit model. To streamline the process, a tcl script utilizing the HyperMesh API was developed. The script takes an Altair HyperMesh database as input, which includes the component's surface geometry and mesh, as well as a local coordinate system specifying the origin of the tow and the warp and weft tow directions relative to the component. Additionally, a text file in ASCII format is created to provide user inputs to the script (Figure 3) specifying the location, width, material, thickness, and layer count (in that order) for each lattice tow. The output of the script is an updated Altair HyperMesh database that includes a composite stack-up defining the lattice and bulk plastic.

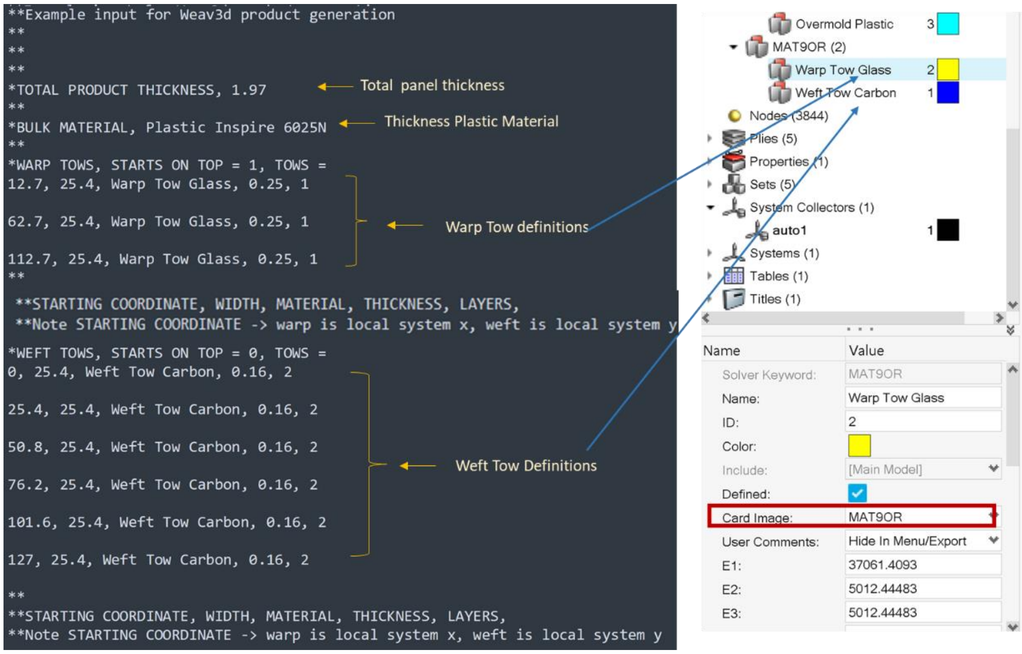


Figure 3 : Relating Input Text File with the Material Data Cards in HyperMesh

The script operates according to the following logic:

- The text file containing tow definitions is parsed, generating tow objects that encapsulate the direction, width, thickness, material, and spacing attributes of each tow.
- The geometry is meshed, a local coordinate system is created, and the material properties of tows and the bulk layer are defined in the material database. Using the local coordinate system, the tows are oriented over the part geometry, and their edges are projected into the part's mesh and geometry along the z-axis of the local coordinate system.
- Elements that fall within the projection of each tow are organized into the corresponding tow objects. Tows are employed to create ply entities in both the warp and weft directions.
- The bulk layer is defined using a final ply entity, with bulk thickness information. This thickness is calculated as the difference between a constant total part thickness and the cumulative thickness of all the tows crossing a particular element. A table is utilized to compute and assign this variable thickness to the ply entity.
- The ply entities are then stacked within a laminate entity, forming the composite structure.
- The script can generate tows for flat and slightly curved shapes. For complex geometries with significant changes in direction perpendicular to the surface, advanced projection or draping algorithms may be required but are not included in the current version of the script.

2.2. Development of a Lattice Design for a Flat Panel using Altair HyperWorks

2.2.1. Input Text File Setup

We will demonstrate the application of the script to a 152.4 x 152.4 x 1.97 mm flat panel with a lattice structure configuration of Design 1. The lattice structure consists of 6 carbon weft tows and 3 glass warp tows, with 25.4 mm center-to-center spacing (no space between tows) between the weft tows and a center-to-center spacing of 50.8 mm between the warp tows. The local coordinate system is defined at one of the vertices of the panel, and all tows in the input text file are spaced from this coordinate system. (Refer to Table 1 and Section 3 for results).

The positions of the 6 weft tows along the local Y-axis of the coordinate system are: 0 mm, 25.4 mm, 50.8 mm, 76.2 mm, 101.6 mm, and 127 mm. The positions of the 3 warp tows along the local X-axis are: 12.7 mm, 62.7 mm, 112.7 mm. It should be noted that the input file assumes the weft tows are aligned with the Y-axis and the warp tows are aligned with the X-axis of the local coordinate system. The user determines the orientation of the coordinate system relative to the geometry and creates the input file accordingly.

The user specifies the panel thickness and bulk plastic material properties in the input text file. Each tow definition in the file includes the distance from the coordinate system, width, thickness, material, and number of layers. For example, the first weft tow in this case is defined as: distance = 0, width = 25.4 mm, thickness = 0.16 mm, material = Weft Tow Carbon, layers = 2(Figure 3). Consistency in naming conventions between the input file, material cards in HyperWorks, and unit systems is important to ensure correct results.

2.2.2. CAD Setup for a Flat Panel

In Altair HyperWorks, the CAD setup involves creating a material data card and meshing the shell model of the flat panel. A local coordinate system is generated at the desired location, serving as the reference point (origin 0,0) for placing the tows in the lattice design. The input text file uses this user-defined local coordinate system (Figure 4) to generate the lattice configuration. The geometric representation of the panel includes the weft and warp tows, as well as the molded plastic component. Material data cards are defined for the tows (using MAT9OR for orthotropic materials) and for the bulk plastic (using MAT8 for isotropic materials) (Figure 3)

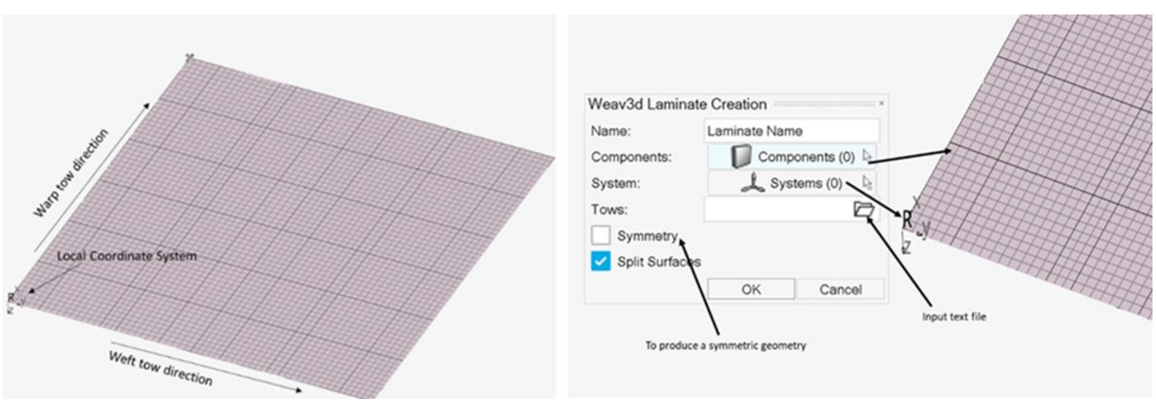


Figure 4 : (Left) Local Coordinate System; (Right) Prompts after the Primary Script is Loaded

2.2.3. Tow Generation

After generating the input file and completing the CAD setup, the user loads the primary script. The script guides the user through prompts to define the component or sub region where the tows will be incorporated. In this example, there is a single lattice region covering the entire panel surface. The user then specifies the local coordinate system and indicates if the design exhibits symmetry across the thru thickness midplane. In our study, the panel is configured as a sandwich structure, with bulk plastic positioned between top and bottom lattices. Consequently, the option for "symmetry" is enabled to reflect this characteristic However, if the user intends to implement the lattice on only one side of the component, the symmetry option should be left unchecked (Figure 4). Once all the prompts are configured, the script generates a lattice-integrated shell model (Figure 5). Users must carefully review the lattice design, verifying tow thickness, component thickness, and fiber alignment in the tows (Figure 6). This step ensures accurate and consistent simulation results.

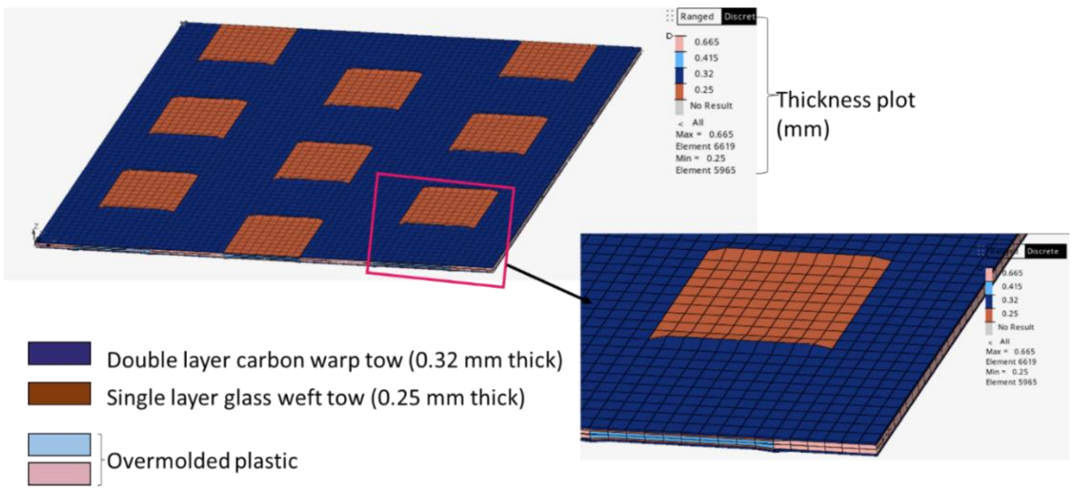


Figure 5 : Tow generated In Altair HyperWorks (Thickness plot)

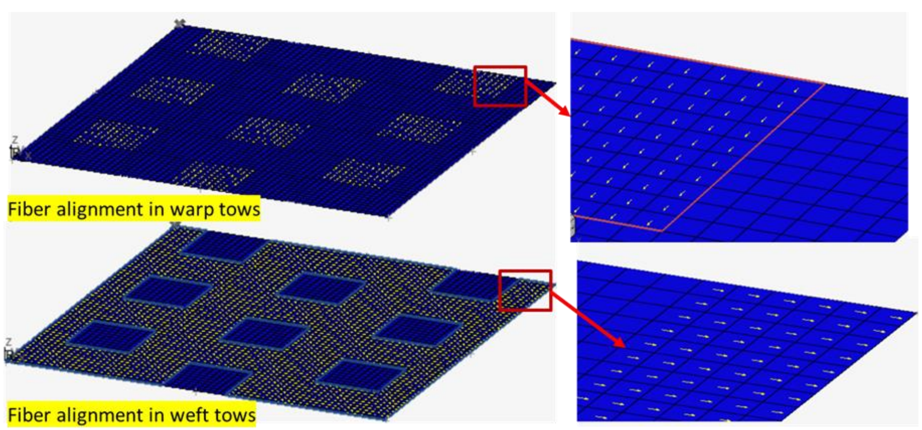


Figure 6 : (Top) Fiber alignment in warp tows; (Bottom) Fiber alignment in weft tows

2.2.4. FEA Setup for Flexure Test

FEA of a lattice reinforced plaque was conducted in Altair HyperWorks with simple boundary conditions simulating a 3-point bending test as shown in Figure 7 and the results were post processed to extract deformation and stress results.

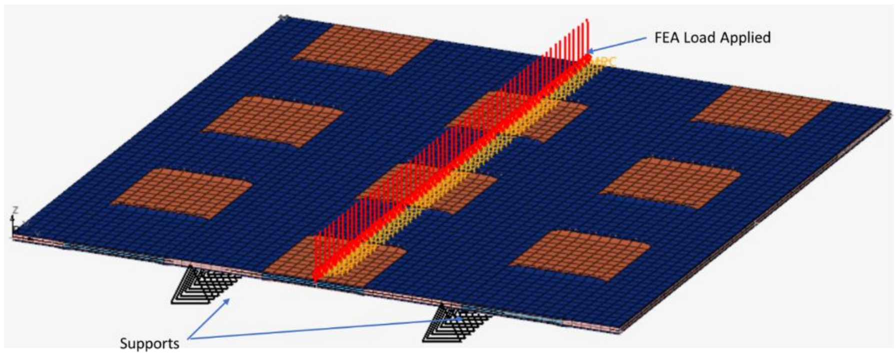
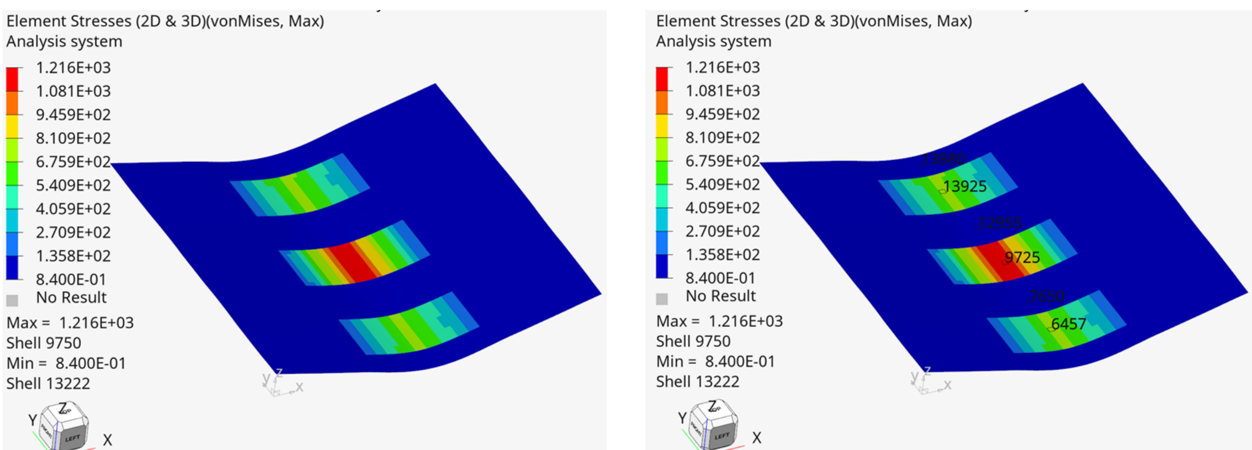


Figure 7 :Boundary Condition for Flexure Test

The lattice tows are explicitly created and meshed as shell elements, simplifying the post-processing stage. The user can easily request deformation and stress plots in a single step (Figure 8), eliminating the need for separate submodeling, which was required in the ANSYS RVE method.



Element ID	Element Configuration	Element Connectivity	Component Name	Contour(Element Stresses (2D & 3D))
6457	104	13839 15984 16013 13840	auto1	7.135E+02
7650	104	17468 17483 17485 17467	auto1	8.781E+01
9725	104	19775 19740 19741 19761	auto1	1.197E+03
12955	104	23151 22091 22090 23137	auto1	8.680E+01
13925	104	23487 24112 24141 23488	auto1	7.157E+02
13880	104	23460 24083 24099 23461	auto1	8.413E+01

Figure 8 : (Left) Deformation Result Design 3; (Right) Stresses in tows Design 3

2.3. Lattice Generation for a Complex Geometry using Altair HyperWorks

The present HyperWorks tcl script demonstrates its advanced capabilities by effectively generating tows for structures that are more complex than just flat panels, specifically an automotive door component. This door component was originally manufactured using a composite material, composed of Natural Fibers and Polypropylene (NFPP), having a fiber weight fraction of 50 %.

To enhance the performance of this NFPP door component, we conducted an optimization study using the ANSYS RVE method to generate a heterogeneous lattice structure aimed at improving the mechanical performance of the resulting component.

Altair's Explicit method was utilized to replicate the heterogeneous lattice structure developed using ANSYS RVE method. ANSYS RVE method optimization revealed that only a specific region of the part requires lattice reinforcement to meet performance targets. The part was divided into lattice and non-lattice regions in HyperWorks (Figure 9), with separate components assigned to each. Material cards are created for the glass tows and NFPP to represent material properties. The geometry is meshed, and a local coordinate system is established (Figure 9) for the lattice-reinforced region, aligning the warp tapes with the part's length (local X axis) and the weft tapes with the width (local Y axis).

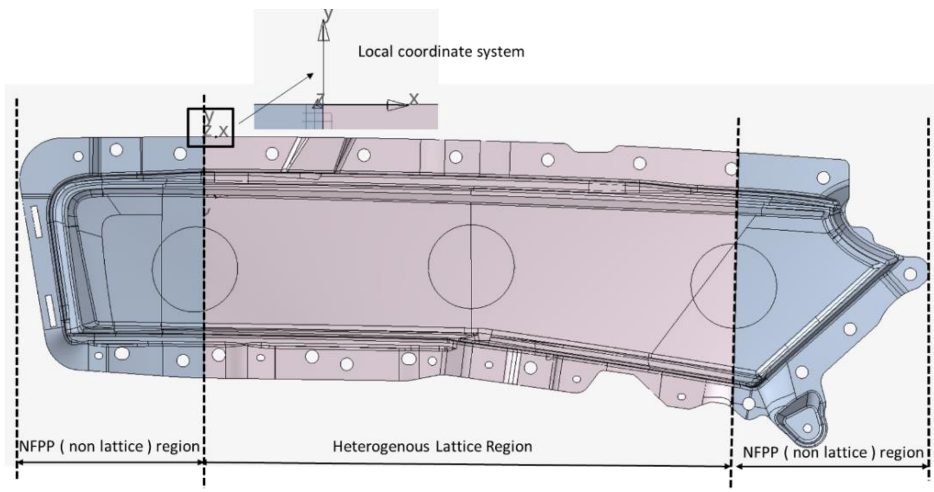


Figure 9 : Local Coordinate Systems & Door Component Partitioning into Lattice and Non-Lattice Regions (Altair Explicit methodology)

Figure 10 illustrates the optimized heterogeneous lattice design, as originally optimized by ANSYS RVE method by varying the center-to-center spacing between the tows across the width of the part. The lattice structure is divided into three regions of varying weft cover factor (CF): 50 % CF, 25 % CF, and 100 % CF, where the cover factor is the width of the tape divided by the center-to-center distance between the tapes. This indicates that along the length of the highlighted region, 50 %, 25 %, and 100 % of the area is covered with glass fiber/polypropylene (GFPP) weft tapes, respectively. These GFPP fiber tapes (0.25 mm thick) serve as the load-carrying elements, while the warp tapes that hold the lattice together are spaced at a CF of 50 %.

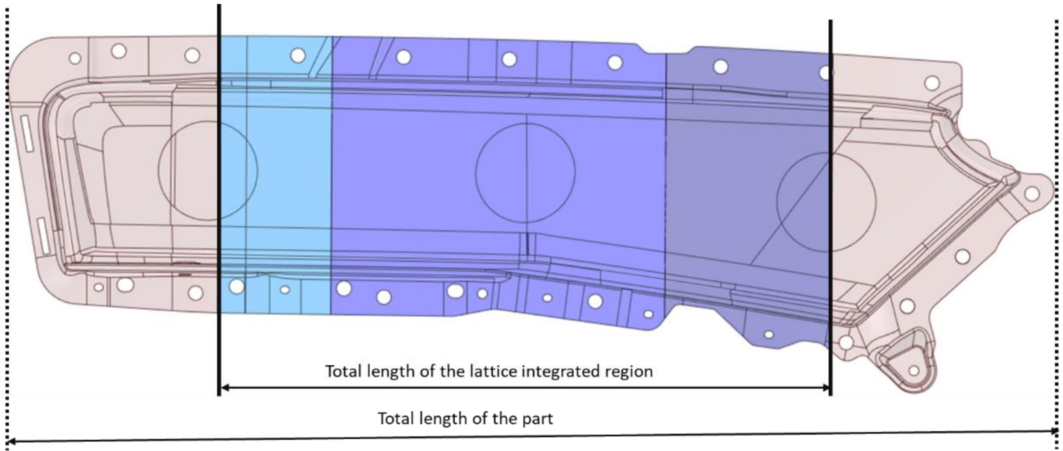


Figure 10 : Heterogeneous Pattern Optimized by ANSYS RVE methodology.

Like the flat panel example, an input text file was generated for the door part, containing information regarding the overall thickness of the component and the specific characteristics of each lattice tow. This information encompasses the tow's orientation, width, thickness, material composition, and spacing.

Upon loading the primary script, the user is prompted to select the appropriate local coordinate system, as shown in Figure 9. By utilizing the tcl script, the input text file, and the Altair Hypermesh database, the lattice design for the automotive door part is successfully replicated. (Figure 11).

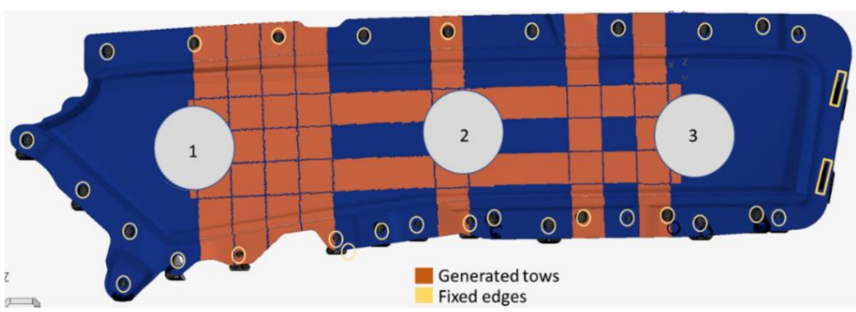


Figure 11 : Lattice Design for and the Static Load FEA Setup for the Automotive Door Component (Altair Explicit methodology)[1]

2.3.1. FEA Setup (HyperWorks) for Automotive Door Component - Static Test

A static load of 150N was selectively applied to the surface of the door component at three distinct regions: 1, 2, and 3, one region at a time in HyperWorks while the door was fixed at some locations as shown in Figure 11. The resulting deformations at each region were measured along with the

---

[1] Door component view flipped to show the lattice facing side.

maximum Von Mises stresses present in the NFPP and compared against the deformations and stresses obtained from the analysis conducted in ANSYS Mechanical.

3. EXPERIMENTATION

Plaques were manufactured for flexural testing (experimental setup shown in Figure 7 to validate the presented Explicit Model approach using Altair HyperWorks. The plaques had dimensions of 152.4 x 152.4 mm and an average thickness of 2 mm. Four plaques were produced for each of the four configurations, as summarized in

Table 1. All designs utilized GF/PP warp tapes, 1 layer, 50.8 mm spacing. Each design included a lattice on both sides of an unfilled polypropylene sheet. The lattice patterns were created using 25.4 mm wide carbon and glass tapes of respective thicknesses of 0.16 mm and 0.25 mm. All plaques were compression molded, with the weft tapes oriented in the primary flexure load direction. The plaques were loaded to failure, and flexure deflection was measured within the elastic range of the material. The flexure span was 63.5 mm.

Table 1: Lattice Design Configuration for Flexure Tests

Design No.	**Molded Plastic Material**	**Weft Tow Material**	**Weft Tow**		**No. of Lattice layers**
			No. of layers	**Spacing (mm)**	
1	Braskem Ti4003F PP	Glass/PP (45 % Vf)	2	25.4	1
2		Carbon /PP (40 % Vf)	2	50.8	1
3			2	25.4	1
4		Mixed -Alternating Glass/PP (45 % Vf) & Carbon /PP (40 % Vf)	2	25.4	1

Figure 12 : Flexure Test Experimental Setup

4. DISCUSSION

4.1. FEA Validation of the Flat Panel with Experimental and ANSYS FEA results

The FEA methodology developed in Altair HyperWorks was benchmarked against the experimental results of the flexure tests conducted using lattice reinforced compression molded plaque designs (design 1 through 4). The results obtained from the Altair Explicit model (design 1 through 3) were further compared with those derived from the ANSYS RVE model. Design 4 was developed as a result of reviewing designs 1-3 and was solely modeled using the Altair Explicit method. This comparative analysis aimed to assess the performance of the Altair Explicit method in relation to the ANSYS RVE method.

The chord modulus for each sample was calculated according to ASTM D790 and the average chord modulus for each design is reported as the experimental chord modulus. The predicted FEA modulus was calculated formulaically from the FEA deflection results. For each design a percentage error between the experimental and Altair Explicit method derived chord modulus as well as the percentage error between Altair Explicit method derived chord modulus and ANSYS RVE method derived chord modulus was calculated to determine degree of correlation. Our flexure test FEA predictions, summarized in Table 2, show that the Altair's Explicit model exhibited good correlation with the experimental results, overpredicting the experimental modulus by an average of 5.8 % (0.3 % - 13.5 %), while the ANSYS RVE method overpredicted by an average of 12.2 % (9.3 %-20 %). This indicates that the Altair Explicit method yields a more accurate and conservative prediction than the ANSYS RVE method.

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

Design No.	ANSYS Chord Modulus (GPa)	Altair Chord Modulus (GPa)	Experimental Chord Modulus (GPa)	% Deviation		
				ANSYS vs. Experiment	Altair vs. Experiment	Altair vs. ANSYS
Design 1	28.02	25.98	25.64	9.28	1.32	-7.8
Design 2	30.23	27.23	25.19	20.07	8.09	-11
Design 3	56.81	53.14	52.99	7.19	0.28	-6.4
Design 4	-	44.3	39	-	13.5	-

4.2. FEA Validation of the Automotive Door Component with ANSYS FEA Results

Furthermore, when comparing the deformations in the door component, as summarized in Table 3, the Altair Explicit model exhibits close correlation with the ANSYS predictions. The percentage deviation between the two models ranges from 0.8 % (overprediction) to -12 % (underprediction). This discrepancy is within an acceptable range and suggests consistency between the Altair Explicit and ANSYS RVE method for more complex structures.

The Altair Explicit and ANSYS RVE models both predict peak Von Mises stress values in the NFPP material below the yield strength specified in the NFPP data sheet, 38 MPa. Specifically, the Altair Explicit model predicts a stress of 20 MPa, while the ANSYS RVE model predicts a stress of 15 MPa (Figure 13), which confirms that the integration of lattices in the door component successfully achieves the target strength requirements.

Table 3 : FEA Deformation Comparison Altair Explicit Vs ANSYS RVE methodology

Point Location	**Altair FEA Deformation (mm)**	**ANSYS FEA Deformation (mm)**	**% Deviation Altair vs. ANSYS**
1	8.36	9.36	-10.6
2	7.63	8.45	-12.8
3	7.46	7.4	+0.8

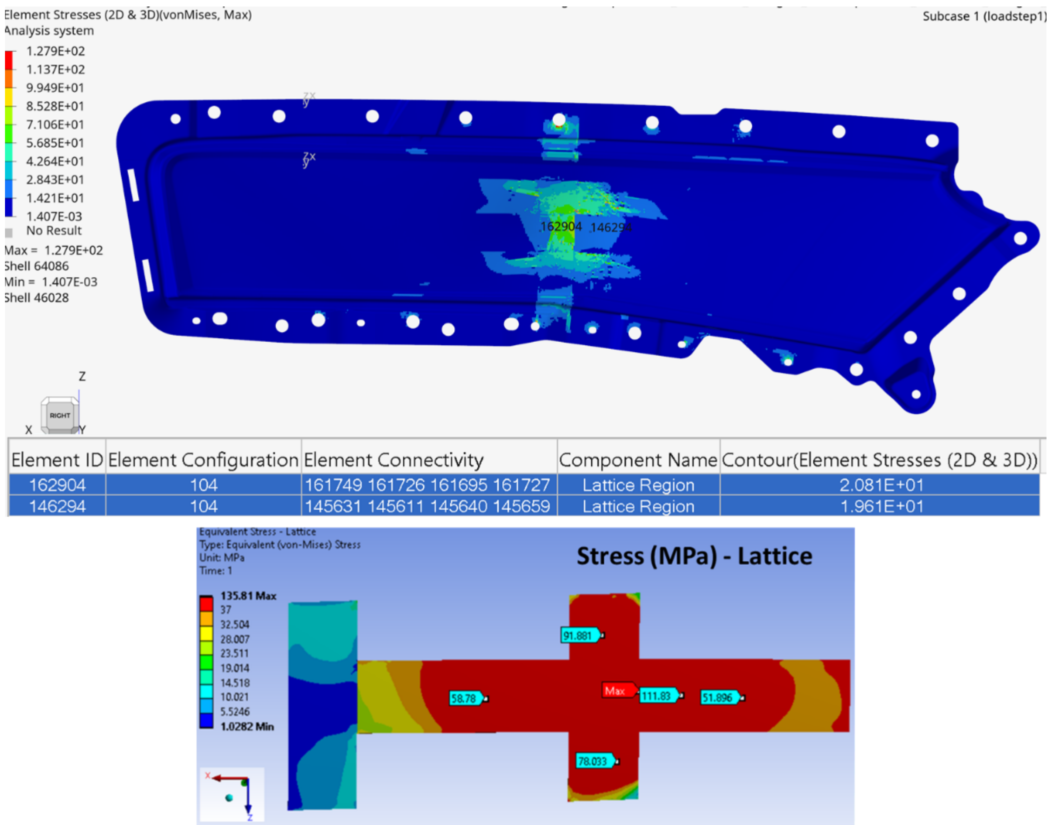


Figure 13 : (Top) VonMises Stress in NFPP – Altair FEA Model; (Bottom) VonMises Stress in NFPP- ANSYS FEA

4.3. Methodology Comparison: User Experience

The ANSYS RVE method for lattice design involves four main steps: RVE development, component thickness construction using ANSYS ACP, FEA analysis of the coarse model using ANSYS Mechanical, and submodeling. These steps utilize different modules within the ANSYS suite. RVE analysis is done in ANSYS Multiscale Designer, component thickness development in ANSYS Pre, and linear static analysis and submodeling in two different ANSYS Mechanical modules. Each step requires the creation of three separate CAD models: the RVE model, the main component, and a submodel. During preprocessing, each CAD model undergoes material data card creation, meshing, and boundary condition application. FEA solutions are obtained for each stage of the process. In the RVE stage, FEA analysis is performed to obtain homogenized material properties. In the analysis of the actual component, deformation values and stress critical regions are computed. In the submodel stage, FEA analysis is conducted to distinguish stresses within the tows and the bulk layer. Throughout each step of the optimization process, the designer is required to generate CAD, apply boundary conditions and generate an output which must be then fed into the next step. The ANSYS RVE method effectively addresses the limitations associated with traditional FEA techniques developed for ply-based composites; however, implementing this methodology can be a labor-intensive process.

The Altair Explicit method offers a simplified and streamlined workflow compared to the ANSYS RVE methodology. With Altair HyperWorks, only one CAD model is required, which is meshed and solved for FEA analysis. Preprocessing involves creating material cards and defining lattice design properties in an input text file. The FEA analysis is then solved, and deformation and stress results in the tows and bulk layers are obtained in a single post-processing step. Altair Explicit method is significantly more computationally efficient than ANSYS RVE method, as shown in Table 4. In a flexure test with integrated lattice structure and a single lattice design iteration, Altair Explicit is over 50 % faster. This speed advantage reduces analysis time and facilitates iterative lattice design optimizations, making it a time-efficient solution for structural analyses with integrated lattice structures.

Table 4: Comparison of ANSYS FEA Vs Altair Explicit methodology - Time to Setup and Solve - Flexural Load Case Applies to Flat Panel

ANSYS RVE Method			**Altair Explicit Method**		
Overview of Steps	**Setup Time**	**Solve Time**	**Overview of Steps**	**Setup Time**	**Solve Time**
RVE CAD	~ 3 minutes	~ 1 minute	*Input Text File*	~1 minute	N/A
ANSYS Pre	~5 minutes	~ 30 seconds	*FEA of the actual part*	~ 8 minutes	~ 40 seconds
FEA of Full Part	~ 5 minutes	~ 1 minute			
Submodel	~7 minutes	~ 1 minute			
Total Setup and Solve Time ~23 minutes			*Total Setup and Solve Time* ~10 minutes		

5. SUMMARY

In our previous work, we aimed to develop a novel FEA workflow using ANSYS Mechanical based on RVE homogenization for predicting deformation behavior in complex geometries with heterogeneous lattices. However, this approach required manual RVE development for each design iteration and an additional submodeling step to analyze stress distribution. Creating accurate submodels for complex geometry proved to be challenging. While an improvement over conventional techniques, this workflow was not ideal. Altair's explicit script streamlined the development of lattice-integrated models for FEA. Users input the part thickness and tow definitions, along with a meshed shell model, local coordinate system, and material cards. Explicitly generating the tows enabled clear stress differentiation. This simplified workflow reduced setup and solve time by ~50 % and eliminated the need for submodeling.

Validation of this new Altair Explicit methodology was completed for flat 3-point bend panels comprising both homogenous and heterogenous material configurations. Altair Explicit method showed 5.8 % average overprediction for the experimental modulus, while ANSYS method had 12.2 %, indicating Altair's higher accuracy.

The Altair Explicit methodology was also used to reproduce a heterogenous lattice reinforcement design for an automotive door component, originally designed using the ANSYS RVE method.

Deformations and stresses obtained from the Altair Explicit method were compared with the results from the ANSYS RVE method and the two models showed good correlation, with the predicted deformations of the two models falling between 0.8 % to -12 % and stresses within the bulk material deviated by only 2 MPa.

The Altair Explicit methodology described in this paper still has certain limitations that need to be addressed. For each lattice design iteration, the user needs to manually create an input text file with tow and bulk definitions, which can be time-consuming and labor-intensive. This manual input needs to be repeated until an optimized lattice design solution is achieved that meets the desired performance targets. Another limitation relates to the scripts ability to handle complex geometries as the script is unable to accommodate sharp radii or significant changes in the direction of the surface, resulting in situations where the script may fail to generate tows for the lattice design in these regions.

Future work will focus on expanding the capability of the script to work with more complex part surfaces, through the use of a more advanced projection or draping algorithm. In parallel, we are also working to develop a implicit model in Altair that is capable of rapid goal-seek optimization of lattice patterns. Once the implicit model identifies one or more candidate designs, the explicit model can be used to model these candidate designs with precise tow locations to verify the exact stress distribution within the part. This work will enhance the efficiency and effectiveness of the Altair Explicit methodology, making it more robust and capable of handling complex geometries with minimal manual intervention.

6. REFERENCES

[1] Rochardjo, H.S.B.; Budiyantoro, C. Manufacturing and Analysis of Overmolded Hybrid Fiber Polyamide 6 Composite. *Polymers* (2021):1-3.

[2] M. Kamble and C. Oberste, "A Novel FEA Approach to Design and Optimize Composite Lattice Reinforcements and Simulate the Mechanical properties of Composite Reinforced Plastics." *CAMX Conference Proceedings*. Anaheim, CA, October 17-20, 2022. CAMX – *The Composites and Advanced Materials Expo.*

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