A NOVEL FEA APPROACH TO DESIGN AND OPTIMIZE COMPOSITE LATTICEREINFORCEMENTS AND SIMULATE THE MECHNICAL PROPERTIES OF COMPOSITE LATTICE REINFORCED PLASTICS

Meghana Kamble, Christopher Oberste
WEAV3D Inc
Norcross, Georgia, USA

ABSTRACT

Recent advances in thermoplastic composite manufacturing have resulted in the development of hybrid overmolded composite structures, combining continuous fiber composites with injection or compression molded compounds; however, standard FEA techniques developed for isotropic materials and ply-based composites do not accurately capture the physical properties and material behavior of these hybrid materials. This problem is particularly true for composite lattice structures formed from woven, consolidated unidirectional tapes, as these materials are capable of varying tape type and spacing of tapes within a given layer of the material. This paper will provide an overview of a novel structural FEA workflow, applicable to a range of hybrid overmolded composite structures, as demonstrated through a composite lattice case study. The workflow predicts the overall performance of a plastic part with respect to changing the tape materials, tape spacing, and layer count of the composite lattice. Homogenization of representative volume elements, comprising a subunit of tapes in the composite lattice and surrounding plastic permits rapid iteration through a range of lattice patterns to solve for a stiffness optimization target. Once a suitable design is identified, submodeling techniques evaluate the relative stress distribution between the lattice and molded plastic material within the part. Additional design constraints, such as weight and cost, can be included in the optimization workflow. FEA predictions using this methodology show good correlation with experimental results collected from lattice reinforced overmolded plastic panels. The attendees will gain familiarity with a new FEA workflow that can be applied to a range of hybrid overmolded composite structures.

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

1. INTRODUCTION

Overmolding, a one-shot manufacturing process, is one of the growing number advanced technologies for fabricating lightweight composite hybrid fiber reinforced thermoplastic structures. This process combines an injection molding or a compression process with the thermoforming of continuous fiber reinforced thermoplastic laminate [1]. Continuous fiber preforms are strategically positioned in a mold cavity and then overmolded, partially or completely, by unfilled or discontinuous fiber reinforced thermoplastic. This process offers engineers the capability to produce structural parts that are selectively reinforced at high rates. Overmolding thus enables high quality parts that are more economical and are especially suitable, but not limited to, the automotive industry.

WEAV3D has developed a process to produce woven composite lattice structures from thermoplastic prepreg tapes which are then overmolded with short or long fiber-reinforced plastic to form a finished structure, thus combining the performance benefits of continuous fiber reinforced plastics with the low cost and ease of production associated with long fiber reinforced plastic. This approach enables designers to achieve the required modulus, strength, and impact properties of the final structural component by adjusting the lattice design by varying the tape material and spacing between the tapes (weave density) in each layer [2].

Most commercially available FEA models for ply-based composites focus on fiber type, fiber orientation, fiber volume fraction, and weave density in each layer as their manufacturing methods are limited to depositing a single homogenous layer at a time. However, when the weave density and fabric material is allowed to change within a given layer, it increases the design space and a workflow which can manage multiple simultaneous variables is essential.

Prior work conducted at Georgia Institute of Technology focused on solving these limitations through the development of a modelling approach in MATLAB R2017a. This approach was a hybrid between an analytical model and finite element approach. The analytical model is based on the fabric geometry model (FGM). FGM considers the fiber and matrix in the composite as a collection of composite rods. This method relies on a stiffness averaging approach to generate a global compliance tensor for a repeat unit cell of the composite by averaging the local stiffness tensor of each rod within that unit cell. These averages are weighted by the relative volume fraction of each rod. The yarn that is along the machine direction of the weave is termed the warp, and the yarn that is in cross-machine direction is referred to as the weft. Each interlace point of the weft and the warp yarn in the woven composite can be simplified into a homogenized composite tile or a representative volume element (RVE) which can be assembled in a finite element model. A RVE is defined as the smallest volume element of a material with a very accurate statistical representation of the typical material properties used in a full scale/macroscale model. This analytical-FEA hybrid approach allows programmatic generation of many composite designs and reduced computational intensity. Once the compliance tensor for each RVE has been calculated, the FEA component, utilizing MATLAB’s 2D partial differential equation (PDE) toolbox, is responsible for applying load conditions to the modelled geometry to calculate stress and strain in each volume element. Once an initial homogenous weave density (constant tape spacing over a single lattice layer) is identified, a design integrating the homogenous lattice with the part is generated. FEA is performed to observe the part level response. Based on the results of the FEA, if needed, the weave density is manually adjusted in the initial design, making this a laborious process. This MATLAB model assumes that the lattice is homogeneous with respect to both tape material and lattice density within a given region as mixing tape materials and varying lattice density requires significant manual intervention. Furthermore, as this approach relies on the 2D PDE toolbox, it can only simulate flat plates.

To overcome the limitations of the MATLAB model a RVE-based approach using ANSYS 2021 R1 - Material Designer was developed as a part of this study. The Material Designer module in ANSYS uses a technique called “homogenization.” It is assumed that the RVE is uniformly repeated over the domain of a given region and the effective constitutive properties of the unit cell characterize the entire domain as well. Homogenization permits rapid iteration through a range of lattice patterns to solve for a stiffness optimization targets. This workflow permits a quick integration of lattice structure with the part and works well even for complex part designs.

This FEA workflow described in the study can be applied to a range of hybrid overmolded structures. Prior basic knowledge of the workings of ANSYS ACP is required.

2. EXPERIMENTATION

In order to provide reference for and validation of the proposed ANSYS RVE workflow, injection overmolded plaques with dimensions of 6.25 inch by 6.25 inch by 3 mm were manufactured for flexural testing. A total of 5 plaques for each of the six different configurations were produced, as summarized in Table 1. These plaque designs were developed using the aforementioned MATLAB model to achieve specific stiffness and cost targets. The weft and warp tape spacing indicates the center-to-center distance between the weft and the warp tapes, respectively. The lattices patterns were manufactured using 1-inch-wide tapes. All plaques were edge gated and molded such that any reinforcement fibers aligned across the load path. The plaques were oriented such that the weft tapes are in the primary flexure load direction and act as an orthotropic material. Stiffness, rather than strength, was the primary focus of this study, so the samples were not loaded to failure. Flexure deflection was measured in the elastic range of the material at a load of 1080 N and the flexure span was 2.5 inches (63.5 mm).

Table 1 : Lattice Design Configuration for Flexure Tests

Design No.	Tape Material	Overmolded Plastic Material	Weft Tape Spacing	Warp Tape Spacing	Single Tape Thickness	No. of Lattice Layers
1	Carbon/PC (44%Vf)	Unfilled PC/ABS	1.75 in	2 in	0.17 mm	3
2	Carbon/PC (44%Vf)	Unfilled PC/ABS	1.5 in	2 in	0.17 mm	3
3	Carbon/PC (44%Vf)	10% Glass filled PC/ABS	1.1 in	2 in	0.17 mm	3
4	Glass/PA6 (39.5%Vf)	30% Glass filled PA6	1.38 in	2 in	0.25 mm	2
5	Carbon/PA6 (39%Vf)	30% Glass filled PA6	4.76 in	4 in	0.25 mm	1
6	Carbon /PA6 (39%Vf)	30% Glass filled PA6	1.11 in	4 in	0.25 mm	1

The experimental setup for flexure test on one of the design configurations is shown in Figure 1. Figure 2 represents an injection molded plaque specimen that exhibits slight bowing/warpage due to the CTE mismatch between the lattice on one face of the plaque and overmolded material everywhere else.

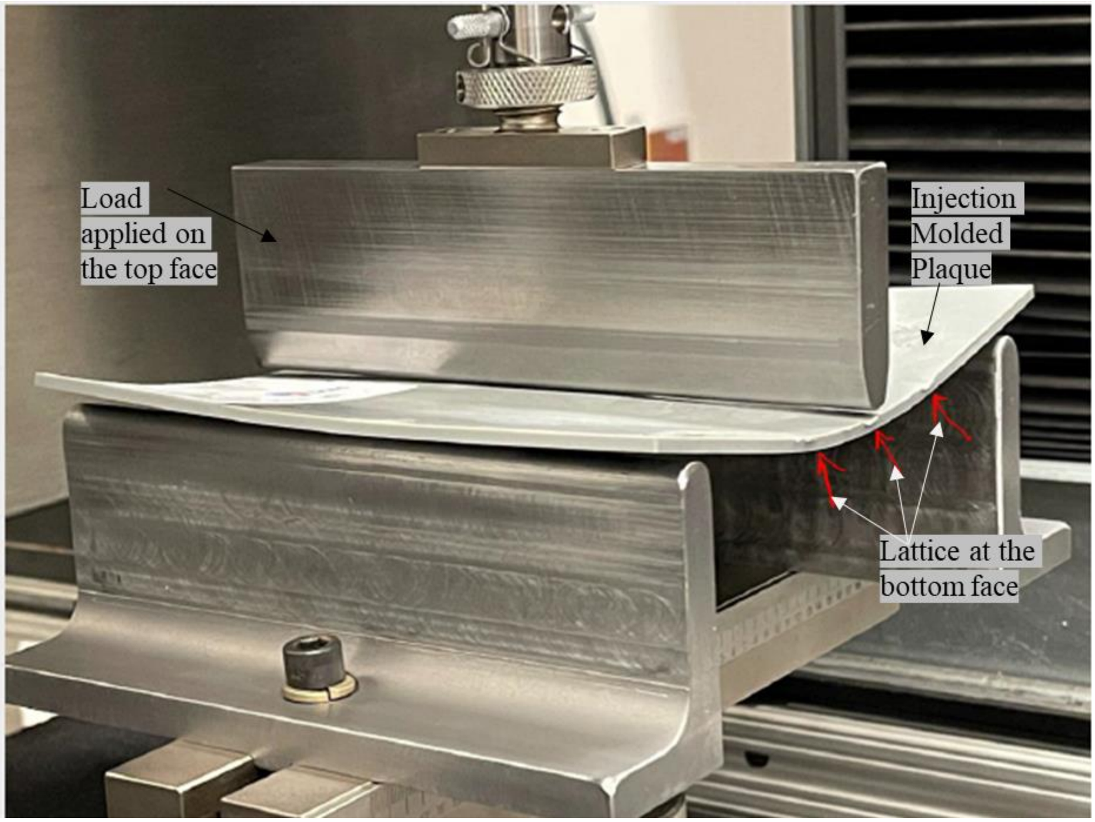


Figure 1 : Flexure Test Experimental Setup

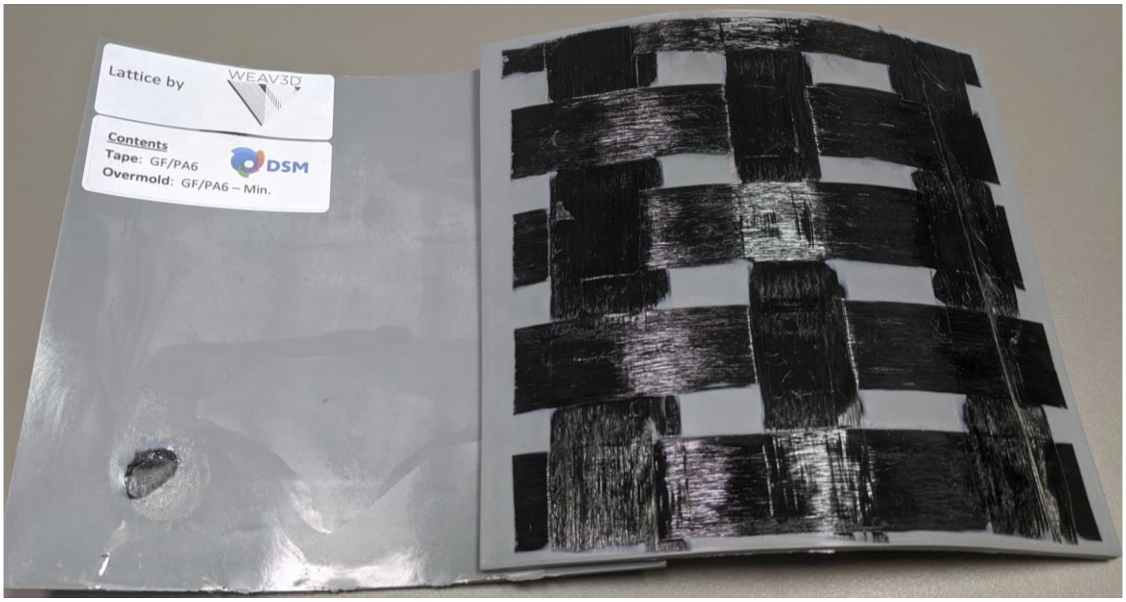


Figure 2: Lattice Reinforced Injection Molded Plaques for Flexure Test

3. METHODOLOGY

The workflow developed in ANSYS 2021 R1 was benchmarked against the experimental results of the flexure tests conducted using lattice reinforced injection molded plaque designs originally designed with the MATLAB model.

3.1 RVE Development for Lattice Design Configurations

Within ANSYS Material Designer, the homogenization process starts with modeling the RVE. In this case, RVE is the smallest subunit of half width tapes in a single layer of the lattice and the overmolded plastic surrounding the tapes that can be repeated over an entire domain of the injection molded plaque (Figure 3).

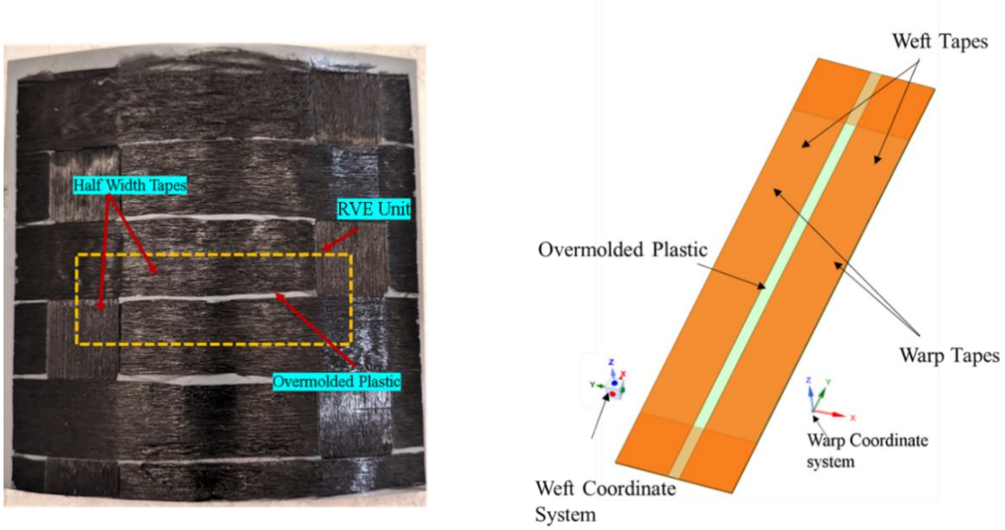
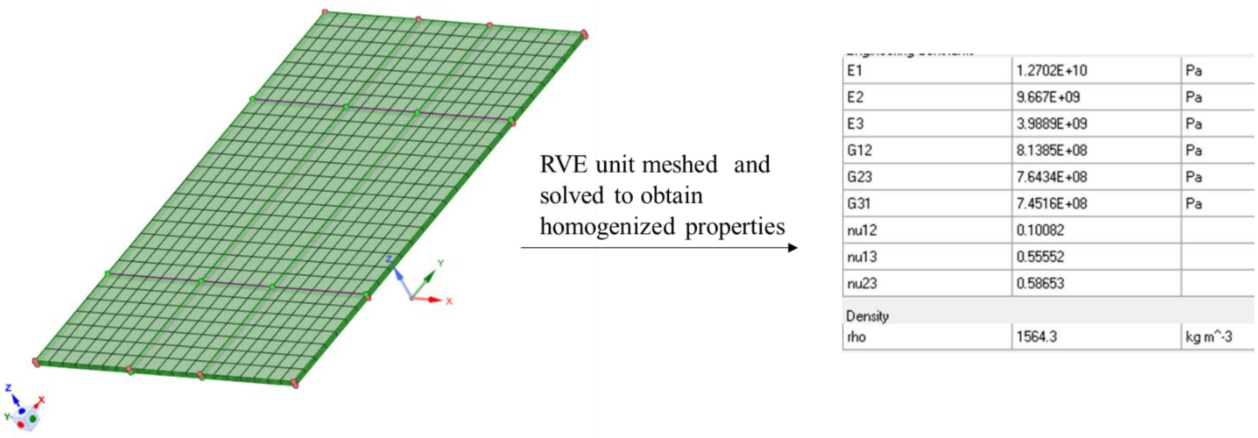


Figure 3: (Left) Example of a RVE unit for Lattice Design 3 Configuration; (Right) RVE CAD generated in ANSYS SpaceClaim

Modeling this RVE requires the creation of a simplified geometry, as well as defining the material properties of the constituent materials. The simplified RVE geometry is developed with an assumption that the weft and warp tapes sit on top of each other instead of replicating an actual woven pattern. This assumption does not have a significant effect on the final homogenized properties, as the crimp angle of these designs range from 0.8 – 2 degrees since the thickness of these prepreg tapes is very small (0.15-0.3 mm) [3]. The constituent materials are the lattice tapes (weft and warps) and the overmolded plastic that occupies the space between the tapes in a single lattice layer. Local coordinate systems are defined for the tape geometries in the weft and warp directions to capture the fiber orientation in the tapes correctly, as indicated in Figure 3

Subsequently, the geometry is meshed for finite element analysis by the user. ANSYS then exposes the RVE to several macroscopic load cases, and its response is computed. The homogenized orthotropic material data for a single layer of lattice is computed from the results of these responses (Figure 4)



E1	1.2702E+10	Pa
E2	9.667E+09	Pa
E3	3.9889E+09	Pa
G12	8.1385E+08	Pa
G23	7.6434E+08	Pa
G31	7.4516E+08	Pa
nu12	0.10082	
nu13	0.55552	
nu23	0.58653	
Density		
rho	1564.3	kg m^-3

Figure 4: RVE unit meshed to obtain Homogenized Material Properties in ANSYS Material Designer

The RVE approach explained above is similar to Ishikawa and Chou's mosaic model used to describe the elastic stiffness of woven - fabric reinforced plastics [4]. In their words, "the mosaic model is idealized as an assemblage of asymmetrical cross-ply laminates." This results in a reduction of the interlacing of the fabric into planar tiles, as shown in Figure 5 . The stiffness of these planar tiles can be calculated for the smallest repeat unit of the composite by utilizing classical laminate plate theory. The stiffness of this repeat unit is equivalent to the stiffness of the overall composite [5].

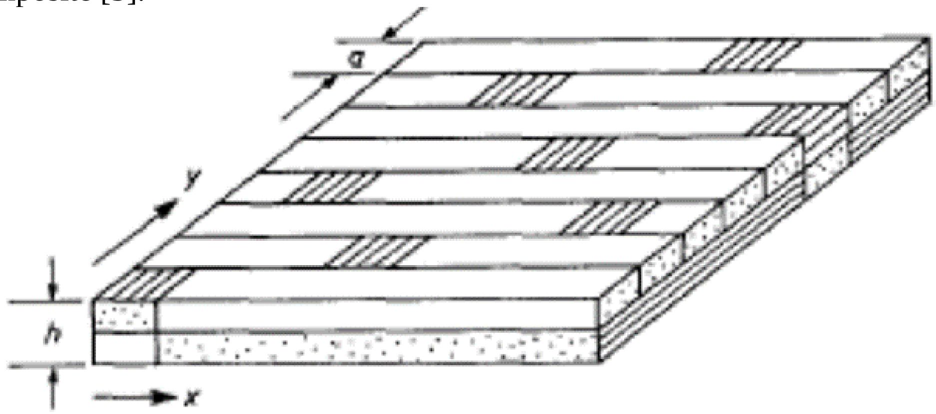


Figure 5 : Representative Element of the Mosaic Model [4]

3.2 Developing Part Thickness in ANSYS Composite Pre (ACP) 2021 R1

ACP is geared towards developing and analyzing layered composite structures commonly referred to as ply-based laminates. These laminates are often manufactured using automated tape placement or hand layup techniques. Hand layup involves manually laying down individual layers or "plies' of reinforcement known as "prepregs". This consists of thousands of fibers pre-impregnated with resin bundled into tows and arranged either in a single unidirectional ply or woven together. The human operator is replaced by a robot in the automated tape placement method. A human operator or a robot applies layers after layer of dry or impregnated fabric to the mold, consolidating each layer as applied [6]. Mechanical properties of these laminate composites at a ply level remain constant. ANSYS ACP efficiently captures parameters like fiber orientation, fiber type, fiber volume fraction and weave pattern at ply level when the material properties are constant. This is done first by generating a ply of appropriate thickness. Then the material property and desired fiber orientation is assigned to this ply. Different plies can be stacked up on top of each other to generate the required part thickness. However, if parameters like the fabric material, fiber reinforcement, and/or fabric density vary within a single ply, modeling such a ply in ANSYS ACP can become a very complex or impossible task.

In a situation where the material properties change within a ply, coupling ANSYS ACP with the RVE approach using ANSYS Material Designer helps the designer model the ply. The designer splits the ply into small sub-regions of constant material properties, then a RVE for every sub-region can be created in Material Designer (Figure 6). Once a library of such RVEs is created, each of the RVE can be meshed and solved to obtain the homogenized orthotropic material properties. Every sub-region can then be assigned with its respective homogenized material property based on the RVE allocated to it. Thus, a single ply with varying material properties can be generated in ANSYS ACP.

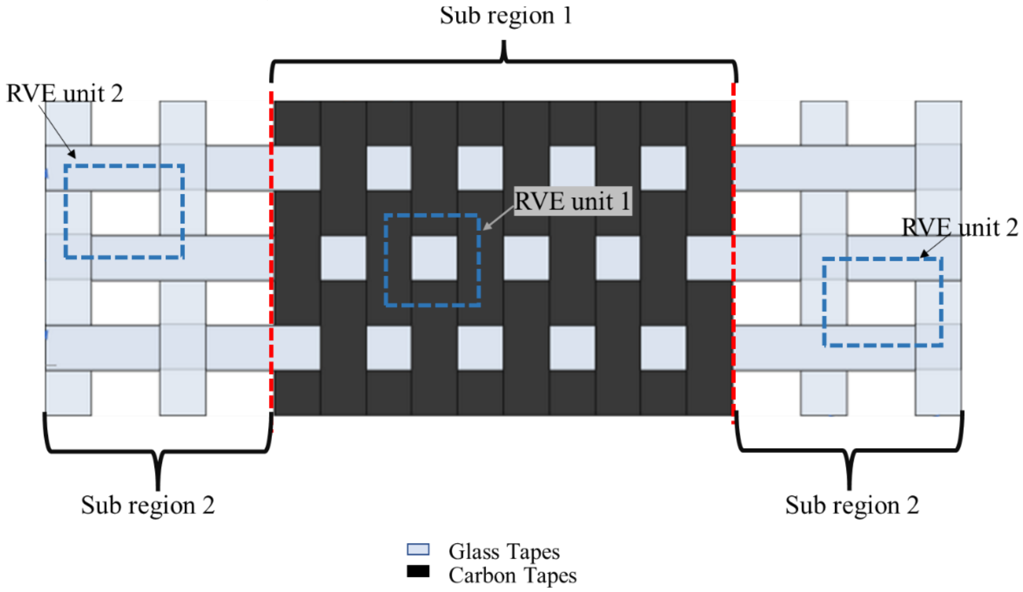


Figure 6 : Example of a Heterogeneous Lattice with Variable Tape Spacing and Tape Material

Commonly, the overmolding method is carried out in two steps. The first step is the fabrication of a continuous fibrous composite layer/ply. This ply can have constant or variable material properties. This layer can then be overmolded by molten plastic, which may or may not contain discontinuous fiber reinforcement, to cover the surface of the composite layer/ply. The resulting overmolded hybrid composite structures will have a certain thickness of just the molded plastic. The thickness of this molded plastic region can be controlled independently in ANSYS ACP, which allows the designer to add structural features like ribs to the design. The overmolded plastic region can be developed as a single thick ply or multiple thin plies. The accuracy of the final FEA analysis is mostly independent of the number of plies used to define the plastic region thickness. A stack up of multiple thin plies create multiple elements through thickness, thus giving the designer an opportunity to evaluate the stress distribution within the plastic more closely. A single thick ply to develop the plastic region reduces modeling steps and computational time

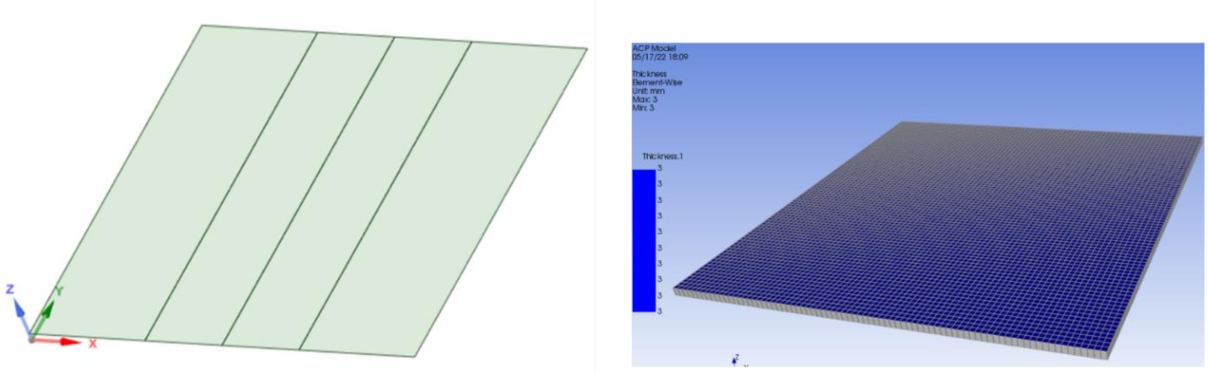


Figure 7: (Left) Ansys Spaceclaim - Surface model of the Part; (Right) ANSYS ACP - Solid Model of Part with RVE layers

The thickness of a single lattice ply is equal to the thickness of the RVE unit designed to obtain the lattice ply's homogenized material properties. The RVE unit is twice the thickness of a single tape as it comprises of weft and warp tapes on top of each other and the overmolded plastic around the tapes. ACP allows the designer to adjust the number and the placement of lattice plies within the thickness of the plaque. Once a shell surface of the part geometry is extracted, developing the thickness of the injection molded plaque reinforced with lattice is a simple process within ANSYS ACP (Figure 7). This workflow can be applied to complex part designs as the designer is no longer required to draw actual tape geometries representing the unidirectional tape reinforcements throughout the part. In the flexure experimental setup, as the weft tapes are aligned along the load path, it is essential to align the fiber direction in the RVE-based lattice plies to match the experimental setup. The approach enables the designer vary tape material and tape spacing within a ply, thus allowing part-level optimization.

3.3 Flexure Test Setup in ANSYS Mechanical

FEA of a lattice reinforced plaque is solved in ANSYS Mechanical with simple boundary conditions for the flexure test as shown in Figure 8. Overall stress response of the specimen was evaluated, and critical stress regions were identified. The FEA deformation values were compared with the experimental deflection data.

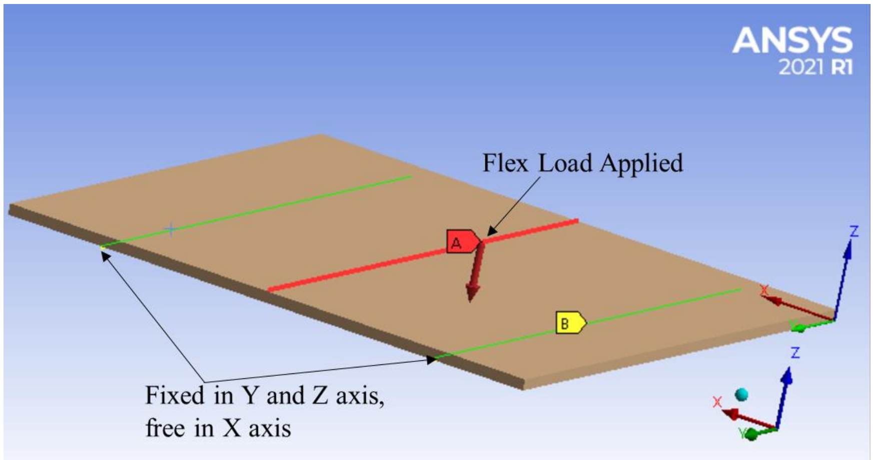


Figure 8 : Boundary Conditions for Flexure Test

3.4 Submodeling

The three-part process of obtaining homogenous material properties from the RVE model and then developing the part thickness in ANSYS ACP to later solve the FEA model in ANSYS Mechanical is referred to as Coarse Modelling. The stiffness result from the RVE model is an average stiffness of its constituent materials, i.e., the overmold plastic and the lattice. Therefore, the coarse model approximates the stresses experienced by the part. Distinguishing between the stresses seen in the tapes and the overmolded plastic is also not possible as Material Designer blends together its constituent phases to generate effective material properties of the unit cell. The coarse model thus requires further post-processing. However, the model is still helpful in indicating the relative stress-critical regions of the part defined by the von-Mises stresses and provides accurate part-level deformation information. Evaluating the results of the coarse model to identify the stress critical region or regions is the first step towards submodeling.

3.4.1 Submodeling Concept

Submodeling is a technique where a coarsely meshed model can be solved followed by a subsequent solution using only a portion of the coarse model with a more refined mesh and detailed geometrical features. This portion of coarse model, which in general is a stress critical region, is called the submodel. As shown in the Figure 9 , and explained shortly, one of the key concepts in submodeling is the designation of the “cut boundaries” defining the submodel.

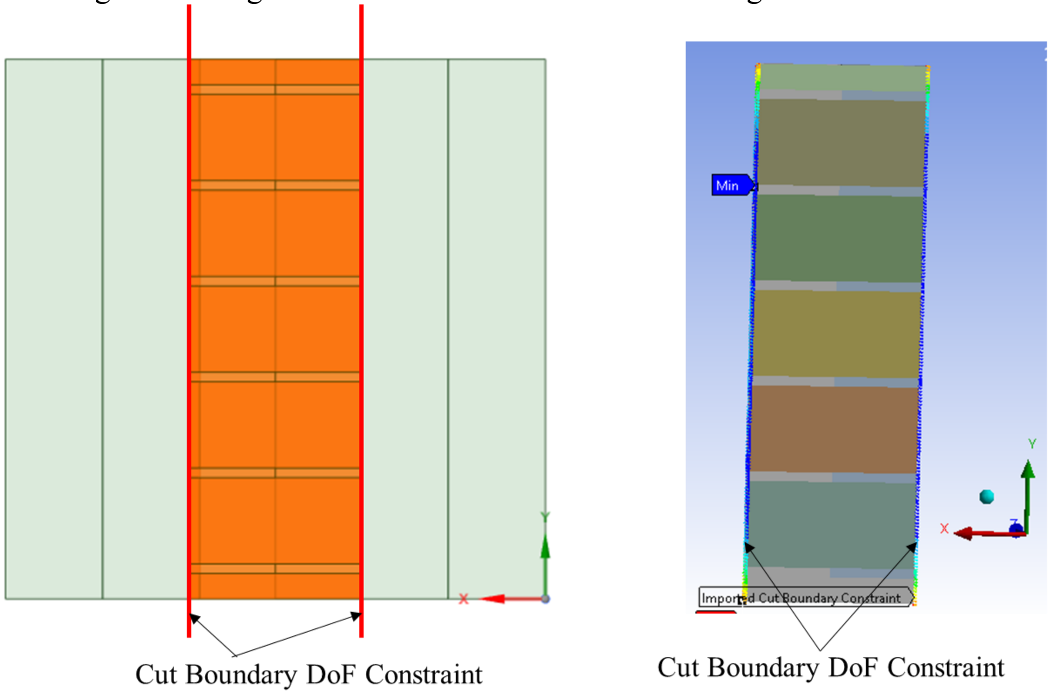


Figure 9: Cut Boundaries Highlighted on (Left) Coarse Model and (Right) Submodel

The displacements from the coarse model are mapped to the cut boundary locations on the submodel from the corresponding locations on the coarse model [7]. Both the coarse and the submodel must be in the same global coordinate system for accurate deformation data transfer. Additional boundary conditions to match those on the coarse model must be added to the submodel before solving the model. It should be noted that if the cut boundaries are too close to the stress concentrations the accuracy of the submodel can be degraded. The user may require several attempts before the location of the cut boundaries is finalized. To ensure that the cut boundary is far enough from the high stress region, a check should be performed to compare coarse and submodel results near the cut boundary. If the deformation values at the cut boundaries of the submodel are reasonably close to the values at the cut boundary locations in the coarse model, one can safely conclude that the cut boundaries are not too close to the stress concentrations.

It should be noted that the submodeling technique allows deformation data transfer from a coarse solid or a shell FEA model; however, in order to solve a submodel, the submodel must be a 3D solid design.

In this study, the von-Mises stress plot of one of the panels subjected to flexure test indicated that the central region of the panel experienced high stresses and a submodel of this central region would help identifying stresses carried by the tapes versus the stress in the bulk. Figure 10 shows the identified stress critical region that was then sub modeled.

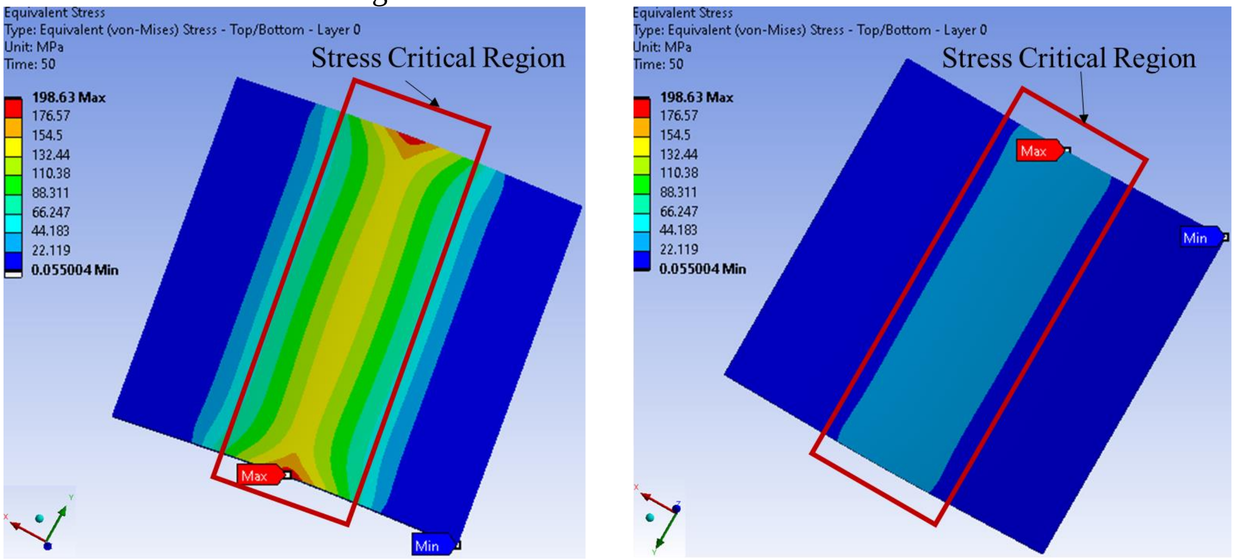


Figure 10: Von Mises Stresses Plot for Flexure Test - Coarse Model; (Left) Top View of the Coarse Model; (Right) Bottom View of the Plaque

The next step is to create a 3D solid model for the submodeling. To distinguish between the stresses in the tape and the bulk, tapes of accurate dimensions are drawn and positioned appropriately in the submodel of the injection overmolded plaque. As opposed to the RVE ply developed in ANSYS ACP, the submodel allows the designer to treat the tapes and the overmolded plastic region as separate entities. As the bulk and the tape are separate bodies (Figure 11), stresses in bulk and the tapes can be easily distinguished.

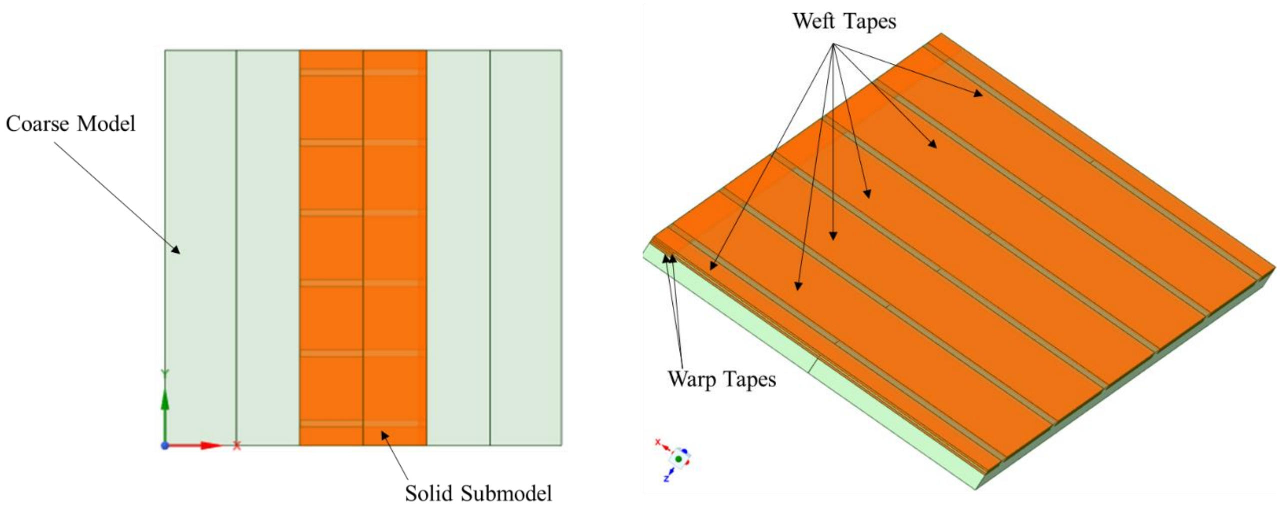


Figure 11: (Left) Location of the Submodel in the Coarse Model Geometry; (Right) Sub Model Geometry

4. DISCUSSION

4.1 Effect of Fiber Alignment in the FEA Model

Panels produced with Design 1 and 2 configurations (Table 1) used an unfilled PC/ABS, i.e., the overmold plastic did not have any fiber reinforcement. All the other panels with design configurations from 3 through 6 had some percentage of short glass fiber reinforcement in the plastic. As mentioned earlier in Section 2, all of these edge gated injection molded panels were fabricated with short fibers aligned perpendicular to the load path.

In every design configuration set, the chord modulus for each of the five samples was calculated considering the elastic range in the experimental stress strain curves. The average of the five chord moduli is reported as experimental modulus for every design configuration. The predicted FEA modulus was calculated formulaically from the FEA deflection results at 1080 N. For each design configuration a percentage error between the experimental and FEA modulus was calculated to determine degree of correlation.

Our initial predictions, summarized in Table 2, showed that the FEA model for the unfilled plastic design configurations exhibited good correlation with the experimental results, underpredicting the experimental modulus by 5 % - 10 %. However, the FEA model overpredicted the moduli significantly for all the glass filled plastic design configurations.

Table 2 : Comparison of Ansys FEA Results and Experimental Results – Flexure Test

Lattice Design Configuration	Overmolded Plastic	Experimental Modulus	ANSYS FEA Modulus	Percent Error
Design 1	Unfilled PC/ABS	6.15 GPa	5.84 GPa	-4.98 %
Design 2	Unfilled PC/ABS	6.73 GPa	6.09 GPa	-9.62 %
Design 3	10 % Glass filled PC/ABS	9.53 GPa	19.9 GPa	24.40 %
Design 4	30 % Glass filled PA6	3.57 GPa	8.06 GPa	125.6 %
Design 5	30 % Glass filled PA6	2.99 GPa	8.49 GPa	183.85 %
Design 6	30 % Glass filled PA6	5.34 GPa	12.8 GPa	139.71 %

The manufacturer's data sheet used to develop the FEA material cards for the glass filled plastics were characterized with alignment of fibers in the load direction. Further assessment indicated that the perpendicular alignment of the short fibers to the load path was not factored into the FEA material cards, which caused high overprediction of FEA moduli in design 3 through 6 (Figure 12).

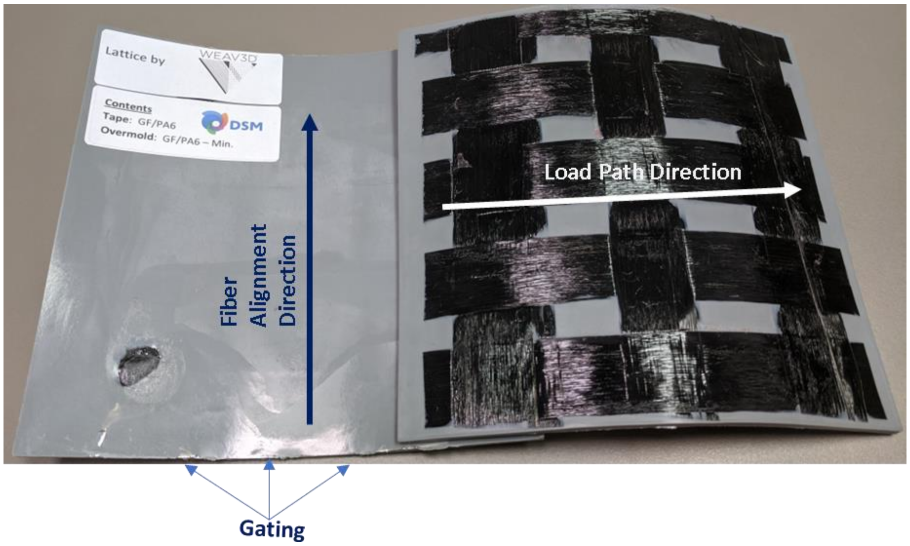


Figure 12 : Fiber Alignment Direction

To factor in the cross alignment of the fibers in the overmolded plastic, FEA material data cards of the glass filled overmold plastics were updated by rotating the properties from the manufacturer’s datasheet to align them perpendicular to the load path. This fiber reorientation reduced the Young’s Modulus values of the filled overmolded plastics (Table 3). The updated material data cards with adjusted Young’s Modulus for the filled plastics were used to re-run the FEA models. Table 4 summarizes these updated results for designs 3-6.

Table 3 : Comparison of Youngs Moduli with Fiber Alignment Parallel and Normal to Load Path

Plastic	Young Modulus Fiber Alignment Parallel to Load Path	Young Modulus Fiber Alignment Normal to Load Path
10 % Glass filled PC/ABS	5.99 GPa	1.19 GPa
30% Glass filled PA6	5.1 GPa	2.1 GPa

Table 4 : Comparison of FEA Results Adjusted for Partial Cross Alignment and Experimental Results – Flexure Test

Lattice Design Configuration	Overmolded Plastic	Experimental Modulus	ANSY FEA Modulus	Percent Error
Design 3	10% Glass filled PC/ABS	9.53 GPa	9.78 GPa	2.53%
Design 4	30 % Glass filled PA6	3.57 GPa	3.08 GPa	-13.89%
Design 5	30 %Glass filled PA6	2.99 GPa	3.15 GPa	5.33%
Design 6	30 % Glass filled PA6	5.34 GPa	5.04 GPa	-5.61%

Once the proper fiber alignment of the glass filled overmolded plastics was taken into consideration for the FEA, the percent errors reduced significantly and the Ansys FEA model showed very good correlation with the experimental results.

4.2 Stress Analysis using the Submodeling Technique

The advantage of using submodeling technique is the ability to extract and distinguish the stress values between the lattice and the overmolded plastic. The coarse model using the RVE approach accurately predicts the deflection response and helps the designer to identify critical stress areas; however, it underestimates the stresses experienced by the constituent materials within the part. Usually, prepreg tapes have very high yield strengths as opposed to the overmolded plastic (Table *5*). Therefore, it is imperative that a designer can identify the stresses in the tapes versus stresses in plastic.

Table 5 : Yield Strengths of Select Tapes and Plastic

Plastic Material	Yield Strength (MPa)
Unfilled PC/ABS	50
10 % Glass filled PC/ABS	95
30 % Glass filled PA6	110
Tape Material	Yield Strength (MPa)
Carbon/PC (44 % Vf)	1500
Glass/PA6 (39.5 % Vf)	746
Carbon/PA6 (39 % Vf)	1200

Before the stresses are evaluated, the deformations at the cut boundaries for the specimen under flexure load are compared to check if the cut boundaries are too close to the high stress regions. Figure 13 indicates that the cut boundaries are not too close to the high stress regions as the deformation values at the cut boundaries of the submodel are reasonably close to the deformation values at the cut boundary locations on the coarse model. A comparison of the total deformation plots of the coarse model versus the submodel (Figure 14) indicates that deformation data is successfully imported to the submodel from the coarse model.

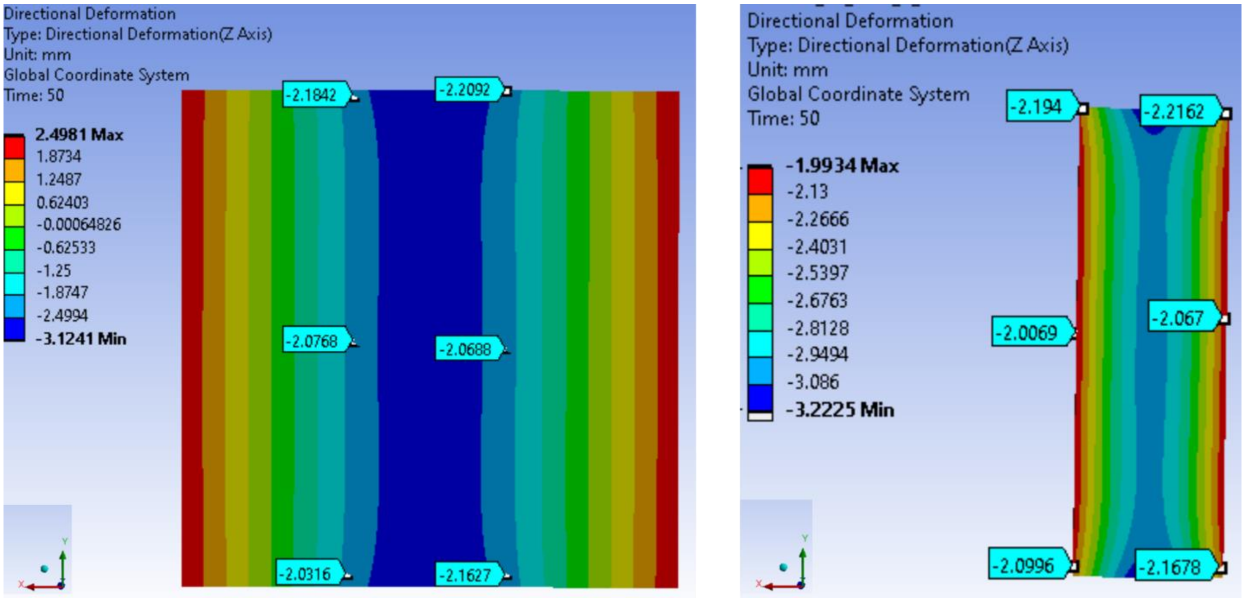


Figure 13: Directional Deformation at Cut boundaries; (Left) Coarse Model (Right) Submodel

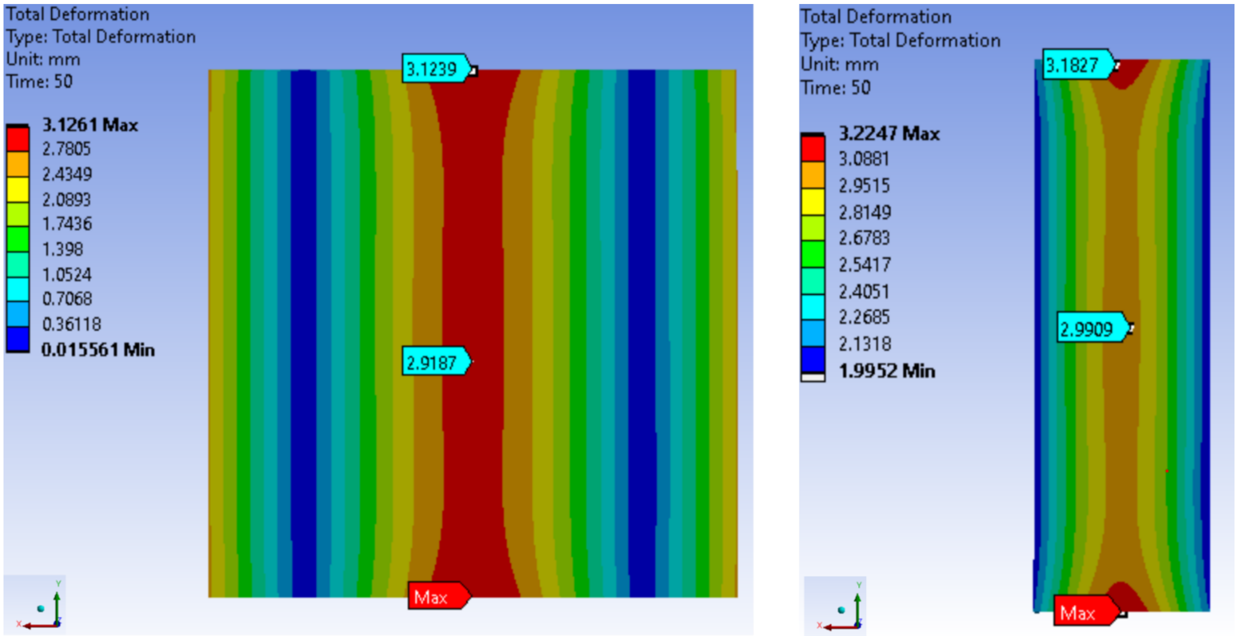


Figure 14 :Total Deformation Plot; (Left)Coarse Model; (Right) Submodel

It should be noted that FEA of this submodel with all the accurate geometrical features not only provides greater accuracy, but also saves time as the submodel regions are solved in a fraction of the time it would take to solve an entire model containing with lattice geometry.

Once the accuracy of deformation data in the submodel is confirmed, the designer can evaluate stresses in the tapes and the overmolding plastic independently (Figure 15 and Figure 16).

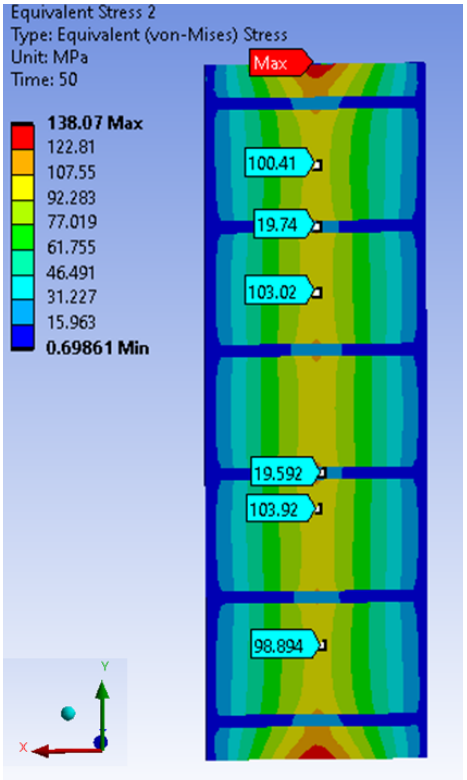


Figure 15 : Von Mises Stresses in the Tapes and the Plastic- Bottom View - Submodel.

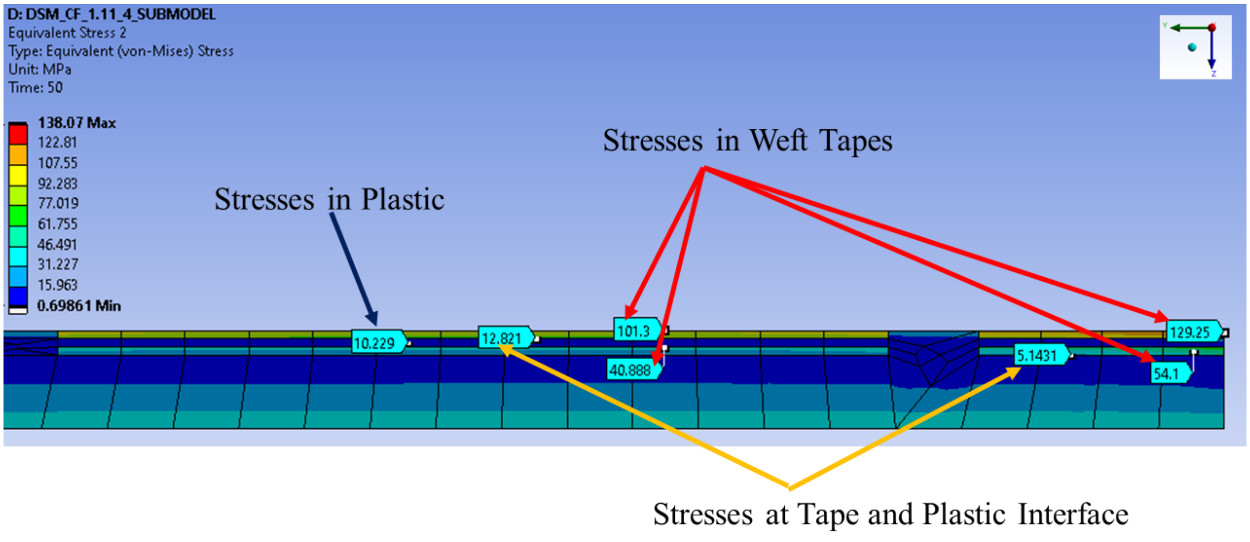


Figure 16 :Von Mises Stresses in the Tapes and the Plastic – Cross sectional View - Submodel.

Average load at flex yield for five specimens of a specific design was applied in the FEA model and a submodel was used to evaluate the stress in the molded plastic as this was observed to yield before the tapes experimentally. At the applied load, the submodel predicted a peak stress of 94 MPa in the molded plastic. While the manufacturer’s data sheet states the yield strength to be 110

MPa, the cross alignment of the fibers in the molded panels likely resulted in a reduction in strength, so we conclude that the submodel results can be used as a predictor of the failure-mode seen in the panels experimentally.

4.3 RVE based ANSYS FEA workflow for Overmolded Hybrid Structures

The RVE based FEA workflow discussed in this paper can be used for a range of overmolded hybrid composite structures utilizing automated tape laying or automated fiber placement methodologies. An RVE unit comprising of the appropriate tape spacing and tape width can be designed and solved in ANSYS Material Designer to obtain homogenized material properties for the continuous fiber region. ANSYS ACP can then be used to create the part thickness by stacking up plies of continuous fiber layer/layers and overmolded plastic regions with the desired fiber orientation. As ANSYS ACP creates the part thickness from the shell surface of the part, this process can be applied to highly complex geometries.

5. CONCLUSIONS

Overmolded thermoplastic composites combine the cost benefits of short fiber reinforcement with the performance benefits of continuous fiber reinforcement. FEA of such hybrid structures is normally incredibly complex and very laborious. Commercially available composite FEA software developed for ply-based laminate composites are often incapable of accurately capturing the hybrid nature of these overmolded structures. The FEA workflow described in this study for lattice reinforced injection molded plaques can be applied to a range of hybrid overmolded structures to accurately predict the part level response.

Simplified representative volume elements (RVEs) comprising of unidirectional tapes and overmolded plastic can be generated in any CAD software and solved in ANSYS Material Designer to obtain homogenized material properties. Once homogenized material properties are obtained from Material Designer, ANSYS ACP can used to build part thickness. The user can obtain accurate deflection data by applying the required boundary conditions to the part in ANSYS Mechanical. The design is solved using the RVE approach and a stress critical region is identified based of the Von Mises Stresses observed in the part. This stress critical region is sub modeled and solved for obtaining accurate stress data, thus saving computational time. The submodeling technique enables the user to distinguish between the stresses in the continuous fiber layer and the overmolded plastic surrounding the continuous fiber reinforcements.

Fiber alignment caused by overmolding must be captured in the FEA setup as this alignment directly affects the FEA outcome. This can be achieved by generating material data cards that model the orientation of fibers relative to the primary load path direction. Once fiber alignment was account for, the FEA workflow presented in this paper shows strong correlation with experimental results. Experimental modulus of specimens subjected to flexure load tests were compared with the predicted FEA modulus and the average error range was found to be -4.4%, with all deviations falling in the range of -14% (underpredicted) to +6% (overpredicted).

One of the crucial limitations of the current ANSYS FEA workflow developed for lattice reinforced overmolded hybrids is its inability to parametrize the spacing of unidirectional tapes within a lattice geometry for Material Designer simulations. The user needs to create new CAD model for every lattice density to obtain the homogenized material properties of the RVE. The submodeling approach also requires the designer to predict the exact location of the tapes in the submodel, which can be complicated in the case of a complex geometry. The current process requires the designer to begin the FEA with a homogenous lattice reinforcement i.e., a lattice pattern with constant center to center tape spacing and constant tape material to evaluate the part level performance. Once the performance of the part with a homogenous lattice reinforcement is gaged, the designer then must test different lattice design configurations and combinations to optimize for weight, cost, and better performance. This entails developing a RVE CAD for every weave density to be tested, assembling layers of lattices and molded plastic to develop accurate part thickness, and finally solving the FEA model. This process can be quite laborious and requires constant user intervention. Future work includes developing a script that takes into consideration current manufacturing limitations and suggests various lattice patterns for achieving the design targets. This kind of automation will help in reducing the constant 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] Oberste. (2019). Investigating the Relationship between Fiber Length, Volume Fraction, and Mechanical Properties of Fiber-Reinforced Plastics [White Paper]

[3] Stig, Fredrik & Hallström, Stefan. (2013). Influence of crimp on 3D-woven fibre reinforced composites. Composite Structures. 95. 114–122. 10.1016/j.compstruct.2012.07.022.

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

[5] T. Ishikawa and T. W. Chou, "Stiffness and strength behaviour of woven fabric composites," *Journal of Materials Science,* vol. 17, pp. 3211-20, 11/ 1982.

[6] M. Elkington, D. Bloom, C. Ward, A. Chatzimichali & K. Potter (2015). “Hand layup: understanding the manual process.” *Advanced Manufacturing: Polymer & Composites Science*, 1:3, 138-151. [DOI: 10.1080/20550340.2015.1114801]

[7] Curreli, Cristina & Di Puccio, Francesca & Mattei, Lorenza. (2018). Application of the finite element submodeling technique in a single point contact and wear problem. *International Journal for Numerical Methods in Engineering*. 116. 10.1002/nme.5940.