2025. Volume 9, Number 2 Pages 83-89

UDC: 539
DOI: https://doi.org/10.30546/09085.2025.02.310

THE STATIC ANALYSIS OF CYLINDRICAL SHELLS MADE OF COMPOSITE MATERIAL

Natig RZAYEV[1*], Mahir BASHIROV[1], Nofal NABIYEV[1]

nrzayev@beu.edu.az*

[1]Baku Engineering University, Khirdalan, Azerbaijan

ARTICLE INFO

Article history
Received:2025-09-30
Received in revised form:2025-10-05
Accepted:2025-10-15
Available online

Keywords:
Composite material;
Von Mises stress;
cylindrical shell;
layer;
deformation.
JEL Classification: L69,O31,O33, L23

ABSTRACT

In this work, a static analysis of a cylindrical shell made of glass fiber reinforced composite material was performed. The static analysis was performed using the simulation application in the Solidworks program. For this purpose, a cylindrical cover with a diameter of 50mm and a length of 100mm was designed. The surface command was used in the Solidworks program to design the part. A composite material consisting of three layers reinforced with glass fiber was selected as the material. The boundary conditions and loading cases of the cylindrical shell were applied in accordance with real operating conditions. The distribution of Von Mises stress, normal stresses along the X and Y axes and deformations were analyzed across the layers. As a result of the analysis, both intra-layer and inter-layer stress and deformation distributions were determined, and areas with particularly high stress concentrations were identified.

1. Introduction

In modern engineering, the demand for lightweight, high-strength and corrosion-resistant structures is increasing. Among them, cylindrical shells made of composite materials dominate. This type of cylindrical shells is widely used in aerospace, shipbuilding, automotive industry, energy and pipeline infrastructure. Their mechanical behavior depends on the homogeneity and heterogeneity of the material, the asymmetry of its structure and the design of the cylindrical shell, the applied loads and the placement conditions. Numerical simulation plays an important role for this. Here it is possible to evaluate the load-bearing capacity of cylindrical shells, optimize the design and identify their weak points.

The mechanical behavior of composite cylindrical shells has been studied in many scientific works. In [1], [2], the methods for calculating the elasticity constants of parts made of composite materials were given and the stability theories of cylindrical shells were improved. The effect of the directions of lifts and the sequence of layers in composite materials on the mechanical behavior was studied in [3]. In [5], stress analyses were performed on structural parts made of composite materials using the finite element method. The results of this analysis were shown to be consistent with both analytical and experimental results. The problem of crack formation in parts made of composite materials was studied in [6].

*Corresponding author.

E-mail addresses: nzayev@beu.edu.az (Rzayev Natig Samandar).

In general, the literature review shows that for an accurate assessment of the behavior of composite cylindrical shells, it is necessary to take into account the combined effects of the properties of the layers, fiber orientations, boundary conditions and loading type. The application of modern programs such as SolidWorks Simulation allows for a comprehensive analysis of these parameters and allows the results to be correlated with real engineering applications.

In this article, the static behavior of a three-layer glass fiber composite cylindrical shells is investigated using SolidWorks Simulation. The main focus is on the analysis of stress and strain distribution, differences in loading across layers and identification of potential weak areas. The aim of the article is to evaluate the behavior of composite shells from both theoretical and applied perspectives and to provide a basis for future optimization work.

2. Materials and Methods

2.1. Cylindrical shells design and material selection

The mechanical properties of the material are as follows:

E-fiber glass:

Density: 2,58g cm^{-3};

Tensile strength: 3,445Gpa;

Youngs modulus: 72,3Gpa;

Elongation: 4,8%

Poissons ratio: 0.2.

Test specimen dimensions: Outer diameter 25mm; length 100mm;

Number of layers -3. All layers are assumed to be of the same material. The angle for the first layer is 150, for the second layer 300 and for the third layer 450.

Applied force: 1000N. The sample is fixed on one side. A force of 1000N is applied across its surface.

Mesh sizes: 2.5mm and 0.125mm.

Proqram: Solidworks (simulation) .

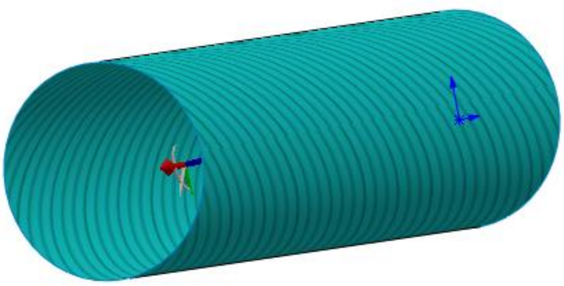


Fig. 1. *3D model of a cylindrical shell made of composite material.*

A cylindrical part made of a composite material consisting of three layers was designed. The thickness of each layer was chosen to be 0.5mm. All layers were assumed to be made of the same material. The angle for the first layer was assumed to be 150, for the second layer 300, and for the third layer 450. In total, the angles for the layers changed in 150-degree intervals (Figure 1).

2.2. Static analysis of a cylindrical shell

After applying the material properties to the 3D model in layers, boundary conditions must be accepted. For this purpose, a cylindrical part made of composite material is fixed along its two or three surfaces along the circumference (fixed geometry fastening type is selected). A force of 1000N is applied along the surface of the sample. The force is distributed at equal values along the surface of the cylindrical part (Figure 2).

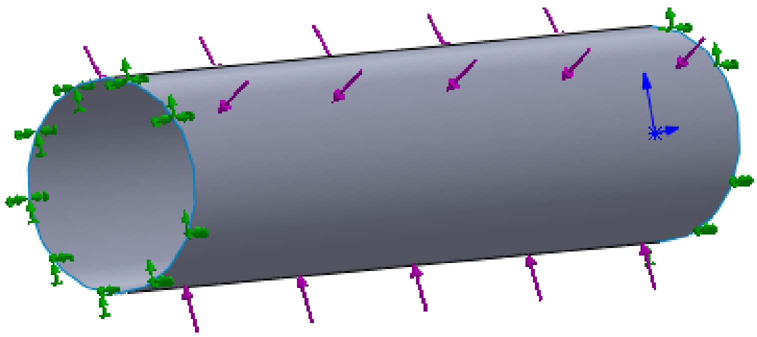


Fig. 2. *Application of boundary conditions.*

To determine the distribution of stress and strain along the layers of the part, a mesh property must be applied. The size of the mesh parameters affects the accuracy of the results. For this purpose, mesh parameters with sizes of 2.5mm and 0.125mm along the layers of the composite part were adopted (Figure 3).

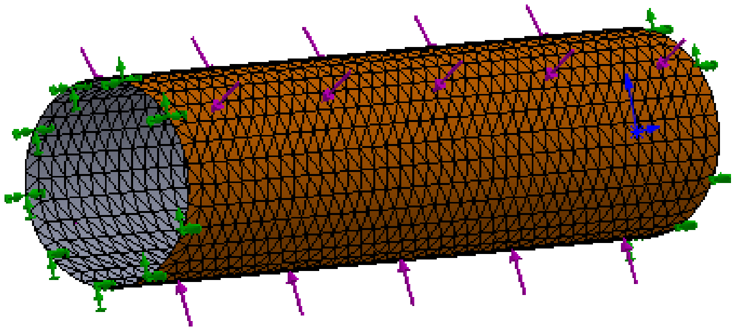


Fig. 3. *Applying mesh parameters to a composite part by layers.*

Analysis of stresses in the first layer of the composite part. First, the analysis of the Von Mises stress was carried out. The minimum value of this stress was observed at points close to the hardening zone of the sample, and the maximum value was observed in the linear direction in the middle zones. The maximum and minimum values of the stress were obtained as 1.012e+00 Mpa and 3.278e-01MPa, respectively. The results of the linear simulation analysis of the Von Mises stress show that the maximum values of the stress were distributed in the linear direction outside the hardening zone (Fig. 4 a). The maximum value of the stress along the X axis was 9.016e-03Mpa, and the minimum value was -1.116e+00 MPa. The maximum value of the stress along the Y axis was 8.581e-02Mpa, and the minimum value was -1.099e+00. The distributed values of the stresses along the X and Y axes are given in Fig. 4, b, c.

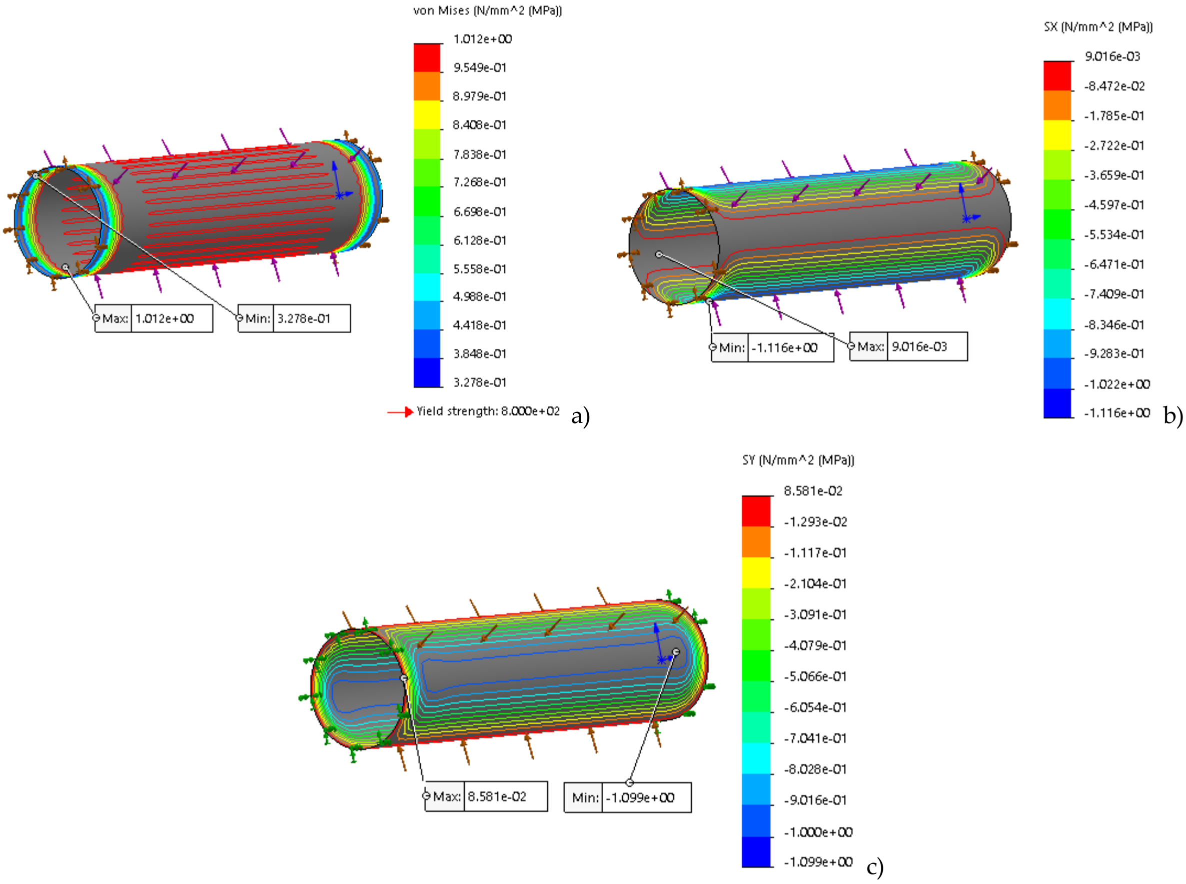
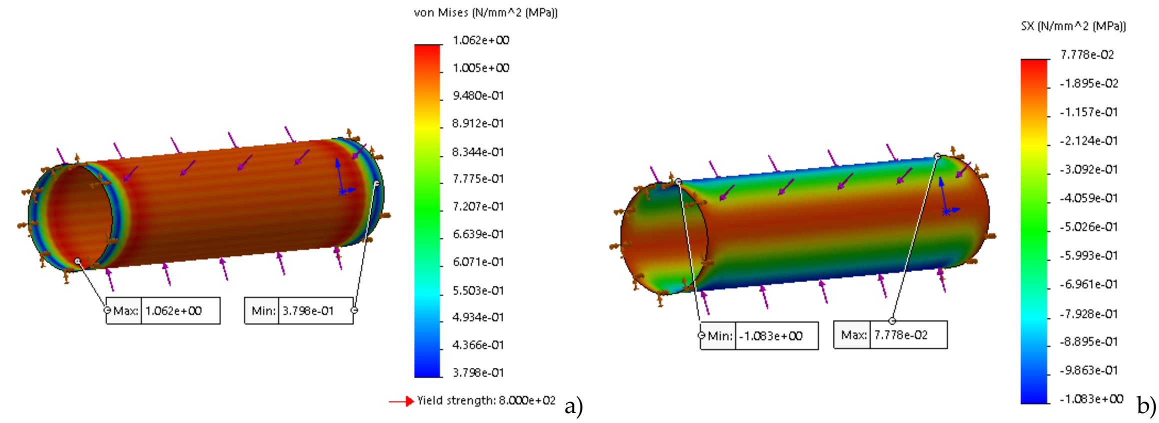
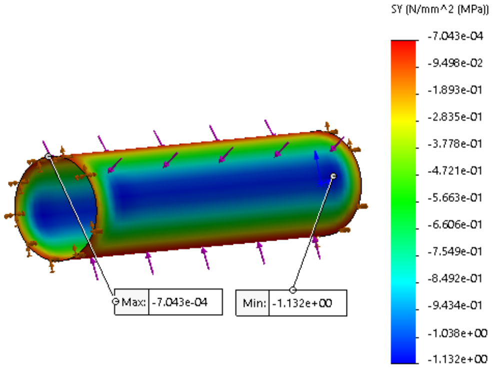


Figure 4. *Stress distribution across the first layer of the composite part.*

Stress distribution in the second layer of the composite part: The maximum value of the Von Mises stress in the second layer was 1.062e+00Mpa and the minimum value was 3.798e-01MPa. The maximum value of the stress was distributed linearly along the two middle surfaces where the layer was located, in the direction of the fibers. The minimum value of the stress was observed in the reinforcement zones. From this, a neutral zone was formed along the circumference between the stress zones (Figure 5 a).

The maximum values of the stresses along the X and Y axes in the second layer were 7.778e-02MPa and -7.043e-04MPa, respectively; the minimum values were 1.083+00 MPa and -1.132e+00 MPa. The maximum and minimum values of the stress along the X axis were formed in symmetrical sections along the length and were observed near the reinforcement zones (Figure 5 b, c).





c)

Figure 5. *Stress distribution in the second layer of the composite part.*

Stress distribution in the third layer of the composite part: The maximum value of the Von Mises stress in the third layer was observed along the circumference in the reinforcement zone. This value is 1.668e+00 MPa, respectively. The minimum value of this stress was 3.931e-01, and it was distributed in a certain width band along the circumference, partially outside the reinforcement zone (Fig. 6 a). The maximum and minimum values of the stresses along the X axis are observed to be distributed in reciprocal segments along the longitudinal section. The values of these stresses were obtained as 9.506e-03MPa and -1.153e+00MPa, respectively (Fig. 6 b).

The distribution zone of the maximum value of the stress along the Y axis is relatively small. This stress is observed in small segments along the axis of the cylinder. The distribution zone of the minimum values is relatively large. The minimum stress is observed along the axis of the cylinder. The minimum and maximum values of the stresses are -1.169e+00 MPa and -1.218e-03 MPa, respectively (Figure 6 c).

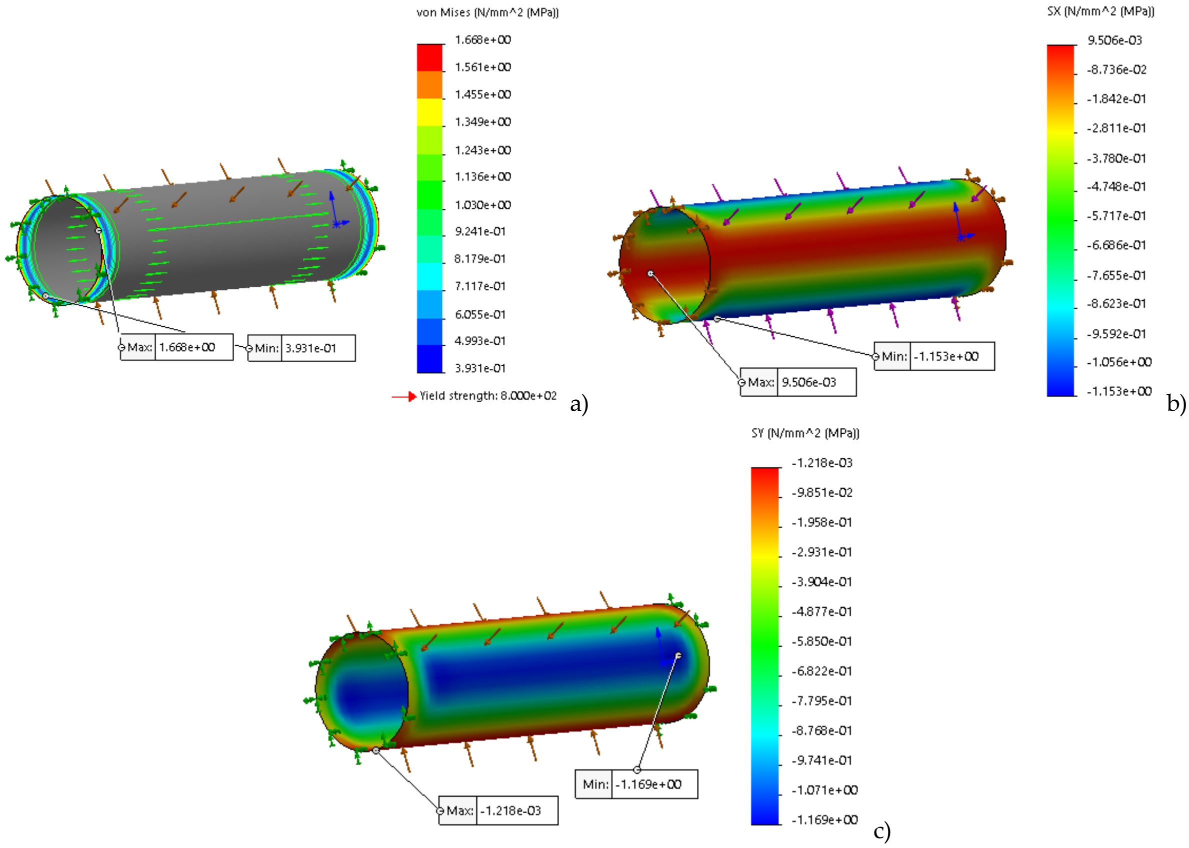


Figure 6. Stress distribution across the third layer of the composite part.

Distribution of deformation along the layers of the composite part: The minimum value of deformation for all three layers is distributed in the zones close to the fastening surface, while the maximum values are distributed along the circumference in the fastening zones, partially on the outer surfaces. The numerical value of deformation has the smallest value in the fastening zones.

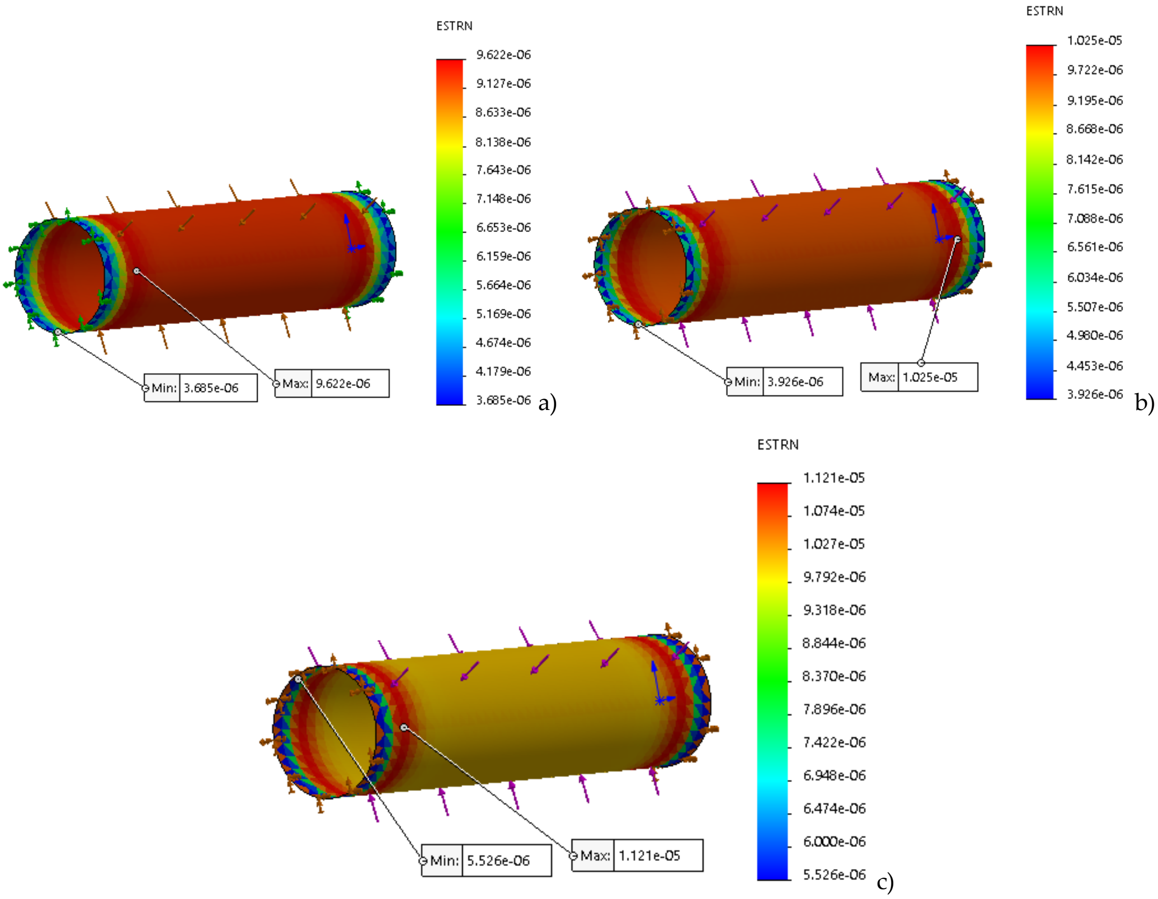


Figure 7. *Distribution of deformation in the layers of a composite part.*

As it moves away from the middle zone, the deformation increases, reaches a maximum value, and then decreases sharply. The maximum value of deformation for the first layer is 9.622e-06, the minimum value is 3.68e-06; in the second layer, 1.025e-05 and 3.926-06, respectively, and in the third layer, 1.121e-05 and 5.526-06 were obtained (Fig. 8).

3. Conclusion and discussions

In the research work conducted, the static behavior of a cylindrical shell reinforced with three layers of glass fibers was analyzed using the finite element method in the SolidWorks Simulation environment, and a number of important scientific and practical results were obtained.

1. The analyses showed that the stress distribution is not homogeneous throughout the cylindrical shell, but is concentrated more in the boundary areas and interlayer zones. This is especially noticeable when the elastic moduli of the layers and fiber directions are different. Local stress concentration was assessed as the main factor increasing the probability of damage to the cylindrical shell.
2. The ply arrangement and fiber orientation in a composite cylindrical shell have a significant impact on the overall strength and deformation behavior of the structure. While

circumferential fiber arrangement increases the resistance to internal pressure, axial fiber arrangement increases the deformation stiffness. Thus, the optimal ply selection should be determined according to the functional requirements of the structure.

3. Analysis of strain distributions showed that the three-layer composite structures exhibited a nonlinear mechanical response compared to simple isotropic cylinders. The mechanical properties and orientation angles of each layer changed the overall elastic properties, leading to the formation of both maximum displacements and local deformations.
4. Modeling based on SolidWorks Simulation software allowed to predict the static behavior of the composite cylindrical coatings with high accuracy.

The results of the study show that not only the choice of material, but also the sequence and direction of the layers should be taken into account when designing composite cylindrical shells. This approach allows to increase the reliability of the structure during operation, prevent excessive stresses and ensure long-term durability. Simulation-based approaches in this direction can contribute to achieving more optimal, reliable and economical design solutions in both production and scientific research processes in the future.

REFERENCES

[1]. Jones, R. M. Mechanics of Composite Materials. 2nd Edition, p538, 1999.

[2]. Reddy, J. N. Mechanics of Laminated Composite Plates and Shells: Theory and Analysis. CRC Pres, p838, 2003.

[3]. Daniel, I. M., & Ishai, O. Engineering Mechanics of Composite Materials. Oxford University Press, p403, 2006.

[4]. Hyer, R. E. Stress Analysis of Fiber-Reinforced Composite Materials. McGraw-Hill, p687, 1998.

[5]. Zhang, Y., Li, X., Wang, J. "Finite element analysis of laminated composite cylindrical shells under internal pressure." Composite Structures, Vol. 140, pp. 67–75, 2016.

[6]. Hashin, Z. "Failure criteria for unidirectional fiber composites." Journal of Applied Mechanics, Vol 47, no 2, pp.329–334, 1980.