EXPERIMENTAL AND NUMERICAL ANALYSIS OF STRESSES AND STRAIN IN SPECIMEN OF COMPOSITE MATERIAL

M. Guțu, PhDc
Technical University of Moldova

INTRODUCTION

There are relevant motivations for using of composite materials in wind turbine blade design. Composites offers tremendous weight savings, increased performance, and design flexibility.

The composites market is a relatively young, fast growing and technologically evolving market.

Fiber Reinforced Polymer (FRP) composites can be defined as a combination of fiber glass (or) carbon and a polymer matrix which provides reinforcement in one or more directions. FRP composites differ from the traditional structural materials such as steel or aluminum. FRP composites have anisotropic properties i.e. properties apparent only in the direction of the fibers [1].

Compared to homogenous materials like steel or plastic, composites are much more challenging to analyze. Multiple materials, complex layups, large number of plies etc. makes the job difficult to design parts with these materials [1].

Determination of the mechanical characteristics of the material obtained from different layers of woven fiber with well defined orientation can be done only by tests on the test machine. These characteristics are required as input data for finite element analysis of composite blades.

Composite modeling performed in ANSYS software allow time saving for material optimization. The intuitive implementation of composite design development in ANSYS Composite PrepPost brought out a revolution in composite simulations. We are able to realize a continuous design process from simulation results to manufacturing including design modifications within the development.

1. EXPERIMENTAL ANALYSIS

1.1. Preparation of test pieces

The pieces were obtained from a laminated plate which in turn was manufactured by Vacuum Assisted Resin Transfer Molding (VARTM) [2], figure 1.

For hand lay-up, fibre volume contents of 30–40% are typical, but the use of vacuum bagging, in which trapped air and excess volatile compounds, such as residual solvent, are extracted, consolidates the composite and allows a volume fraction of 50% or more to be achieved [2].

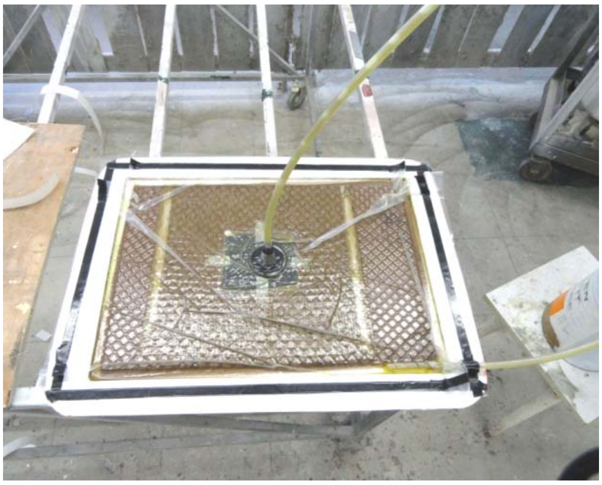


Figure 1. Vacuum Assisted Resin Transfer Molding.

Laminate specification employed in the specimen design is the following:

• 26 layers of E-Glass Woven Roving Material 300 g/m^2 (0-90$_{13}$/CSM/0-90$_{13}$);

• one layer of Chopped Strand Mat with 810 g/m^2 in the middle.

The process of vacuum resin transfer was performed at 0,7 atmospheres. So was obtained 6,6 mm thick laminate with 67% fiber volume fraction.

After process of curing of about 10 days, from plate 230 mm × 24 mm test specimens were cut, figure 2.

Figure 2. Test specimen.

1.2. Tensile test experiment

To determine the mechanical properties of the composite material used in the test pieces tensile tests were performed on Instron 8801 universal testing machine (Figure 3) in the laboratory of Mechanical Testing of the Faculty of Mechanical Engineering at the George Asachi University of Iasi.

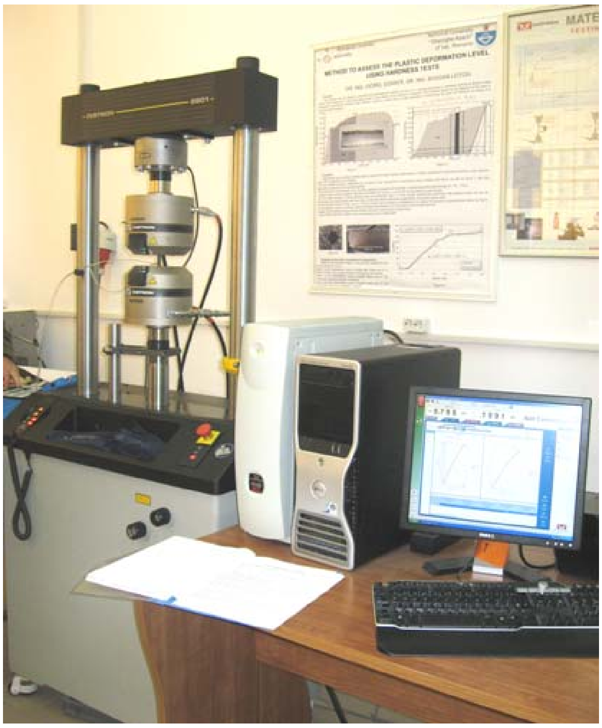


Figure 3. Instron 8801 universal testing machine.

Instron 8801 test machine has the following characteristics:

- Axial loading capacity 100 kN;
- Specimen hydraulic clamping system that prevents slipping.

Test machine is equipped with a FastTrack 8800 digital controller that records the data and transmits them to a PC unit.

Test specimen was subjected to the tensile load up to failure (aprox. 58 kN). Force-extension graph is shown in figure 7.

The stiffness characteristics of the material are the following:

E_1, E_2 - longitudinal and transverse Young Modulus;

$E_1 = E_2 = 14{,}7$ GPa - are determinate from tensile test

ν_{12} - Poisson's ratio (is allowed $\nu_{12} = 0{,}3$) [1], [2];

G_{12} - Shear Modulus

Shear Modulus can be estimated from [3]:

$$G_{12} = G_m \left[\frac{1}{(1-V_f) + \frac{G_m}{G_f} V_f} \right], \text{[GPa]} \qquad (3)$$

where G_m is the shear modulus of polyester (G_m = 1,4 GPa);

G_f is the shear modulus of E-glass (G_m = 30 GPa).

V_f is the fibre volume fraction (67%).

These characteristics are required as input data for finite element analysis of composite specimen.

2. FINITE ELEMENT ANALYSIS

Finite Element Analysis was conducted on this specimen to validate the experimental results. Simulations were performed in ANSYS Parametric Design Language [4]. A finite element model is created using Shell 181 element type (figure 4) and the material properties and loading conditions are accurately specified to simulate the actual test conditions.

The layup of first 11 layers of shell sections is presented in figure 5. The middle layer of CSM was considered as a material with linear isotropic properties.

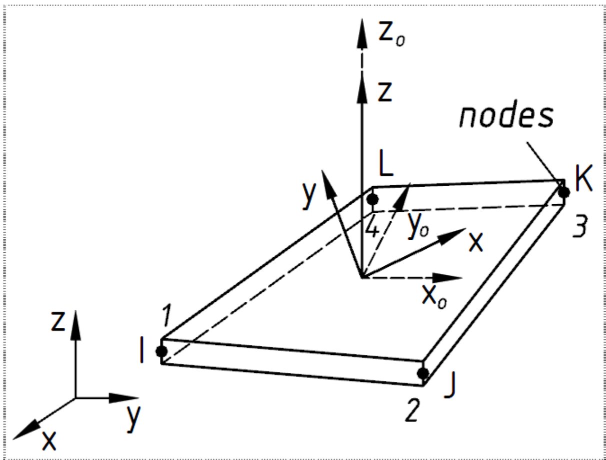
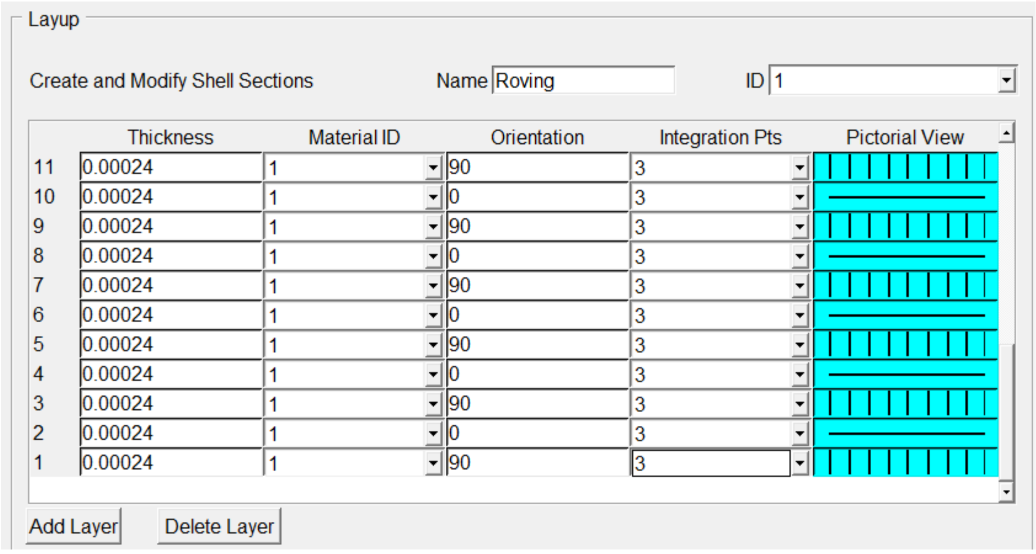


Figure 4. Geometry of SHELL181 layered finite element.

The specimens were modeled with 168 SHELL181 4-node 3-D shell element with 6 degrees of freedom at each node. (figure 6) [5]. Sample equivalent von Misses stress and displacement vector sum for 50 kN tensile load are shown in figure 7.

The results obtained from Finite Element analysis show good agreement with experimental results.



Layup

Create and Modify Shell Sections Name Roving ID 1

	Thickness	Material ID	Orientation	Integration Pts	Pictorial View
11	0.00024	1	90	3	
10	0.00024	1	0	3	
9	0.00024	1	90	3	
8	0.00024	1	0	3	
7	0.00024	1	90	3	
6	0.00024	1	0	3	
5	0.00024	1	90	3	
4	0.00024	1	0	3	
3	0.00024	1	90	3	
2	0.00024	1	0	3	
1	0.00024	1	90	3	

Add Layer Delete Layer

Figure 5. Layup of composite shell sections in ANSYS APDL.

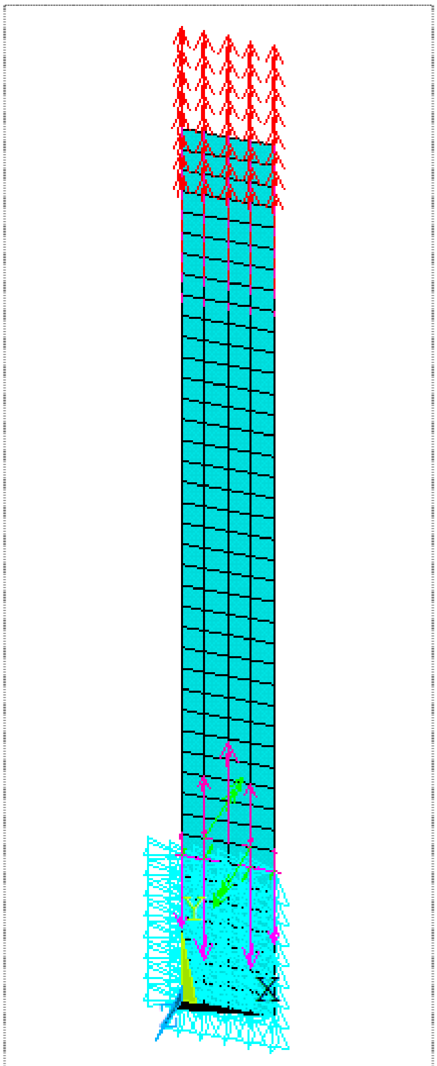


Figure 6. Finite element model of specimen and loads definition.

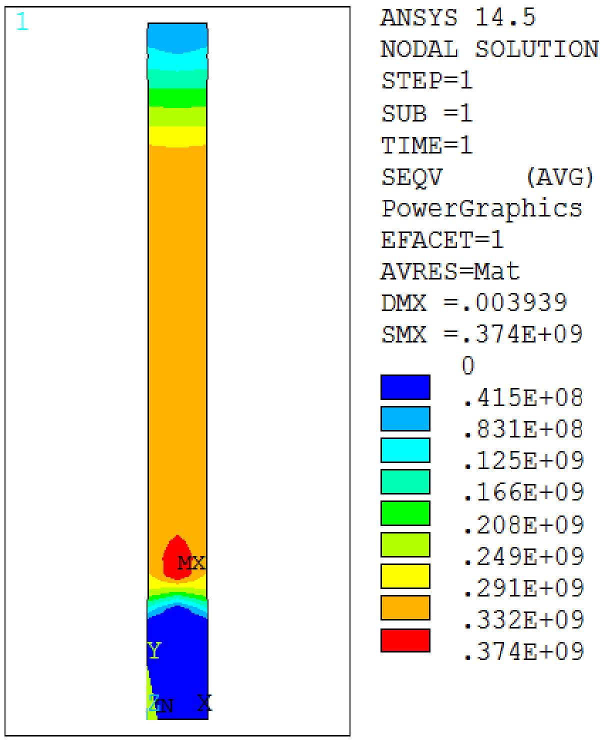


Figure 7. Sample equivalent von Misses stress and displacement vector sum for 50 kN tensile load.

3. RESULTS AND DISCUSSION

The results are showing a linear increase in extension with the increasing force. This is an expected output and it confirms with the theoretical behavior of a sample subjected to tensile stress.

Test results are shown in Figure 8.

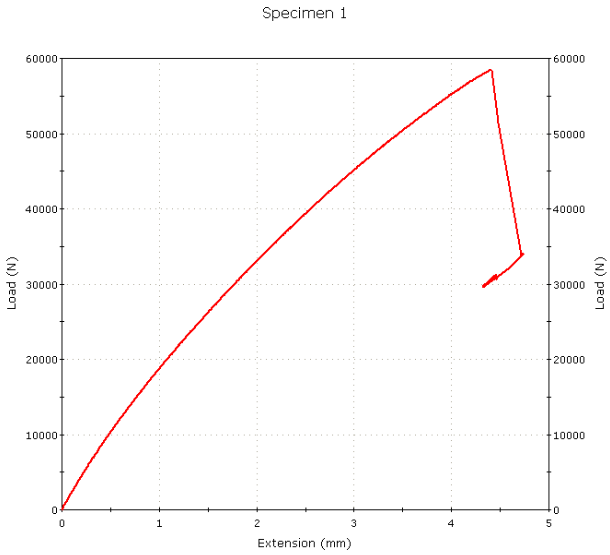


Figure 8. Force-extension graph.

Finite element results naturally involve some deviations from exact solutions due to characteristics of composites. Unlike metals, composites involve a number of uncertainties, resulting in deviations from expected results to increase [6].

In figure 9 by superimposing the results obtained from experimental study over numerical analysis, a comparison of theory and practice is given.

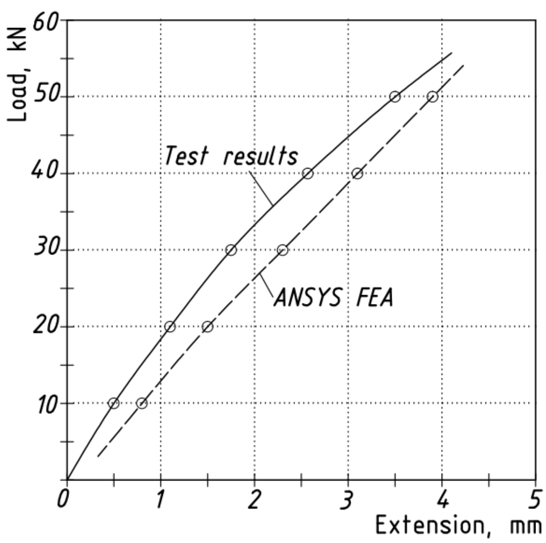


Figure 9. Force-extension graph, samples comparison of experimental and numerical analysis results.

Parameters like increasing stiffness, failed fibers and cracks in matrix material play an important role in the complex deformation mechanics of a real specimen, but they are omitted in the theoretical study, thus theoretically speaking, there exists a complete linear relationship between the applied load and the displacement as shown in the graphs.

CONCLUSIONS

The experimental results are close to the expected theoretical results.

The variations in the experimental results may occur due to several reasons:

- discontinuities in the test specimen material;
- differences between the test specimen and tensile specimen materials;
- uncertainties in the experimental setup (line noise, human error, etc.) [6].

References

1. *Rosato Donald V., Rosato Dominick V.* *Reinforced Plastics Handbook, 3rd edition. Elsevier Science 2004, pp 956-996.*

2. *Gay D., et at.* *Composite Materials Design and Applications, CRC Press 2003, p. 40-50.*

3. *Richardson D.* *The Fundamental Principles of Composite Material Stiffness Predictions, University of the West of England, presentation.*

4. *Madenci E., Guven I.* *The Finite Element Method and Applications in Engineering Using Ansys, Springer 2006, p. 397-403.*

5. *Bora B.* *Design and analysis of filament wound composite tubes* School of natural and applied sciences of Middle East Technical University, 2004, p. 36 - 46.

6. *Filiz Ç.* *Analysis of composite bars in torsion, Thesis for the degree of Master of Science, Dokuz Eylul University, IZMIR 2005.*

Recommended for publication: 26.08.2012.