Composites Part B 95 (2016) 282–292

Contents lists available at ScienceDirect

Composites Part B

journal homepage: www.elsevier.com/locate/compositesb

The precision of FEM simulation results compared with theoretical composite layup calculation

Hosein Tafaghodi Helali*, Manfred Grafinger

Technical University of Vienna (TU Wien), Institute for Engineering Design and Logistics Engineering E307, Getreidemarkt 9, 1060, Vienna, Austria

ARTICLE INFO

Article history:
Received 18 February 2016
Received in revised form
31 March 2016
Accepted 1 April 2016
Available online 9 April 2016

Keywords:
Carbon fiber (Epoxy carbon)
Mechanical properties (Mechanical constants)
Lamina (ply) and laminates
Finite element analysis (CATIA, ANSYS software)
Computational modelling (Theoretical method)

ABSTRACT

Nowadays usage of the composite materials is increasing in many fields of industry. Composite materials have better mechanical properties and heat resistance but lower weight than other materials. Analyzing the composite material in case of using it in mechanical design is inevitable. The FEM analysis is an applicable alternative for theoretical methods and it could be applied for complicated components. As the theoretical calculations for composite components are even more complicated than other materials and sometimes impossible, the comparison between the theory and FEM gives us a useful and trustable replacement for applying the FEM analysis instead of theoretical calculation. The challenge in this paper is to find an appropriate simulation for the composite component based on its load cases; the samples are a cylindrical tube under tension load, and a surface under bending moment. First step is the theoretical calculation based on the ABD matrix formulation. Second approach is the FEM modeling of the components (cylinder, surface), three types of FEM composite modeling with two FEM software (CATIA, ANSYS) are accomplished for these samples. The theoretical calculation is considered as the principal method and three FEM simulations results are compared to the principle result. The output of the first sample are strains and the second are curvatures, comparing the outputs of all methods for each sample creates an overview over the precision and applicability of each FEM method.

1. Introduction

In sample 1 (cylindrical cube) the simulation for quasi isotropic laminate layups like $[0, 45, -45, 90]_n$ [1–3]; and for symmetric orthotropic laminate layups such as $[0,\ 45,\ -45,\ 90,\ \overline{0}]_s$ are applied. The goal is to study in which case the whole layup simulation is necessary, and where calculating the average elasticity factors and considering them as input in reference surface is enough. The results for both types of layups are stablished to ascertain if FEM simulation results could cover theoretical calculation results. It is also important to know if is it possible to calculate and simulate symmetric orthotropic layups like quasi isotropic layups.

In sample 2 (surface) only the 9 layer layup $[0,\ 45,\ -45,\ 90,\ \overline{0}]_s$ is applied. The main question in this sample is which FEM simulations are precise enough to cover the theoretical results, in condition that the output is curvature and position of each ply would act its own role in the results.

The answers are discussed in this paper with these methods for both samples:

- Theoretical calculation with ABD matrix
- CATIA FEM simulation with average elastic factors
- CATIA FEM simulation with complete layups modeling
- ANSYS FEM simulation with complete layups modeling

In theoretical method parameters of ABD matrix based on material properties and layups information are calculated, then by placing the ABD matrix in Hooke's law, strain or curvature could be calculated. In second method the average elasticity factors based on the classical laminate theory formulations (CLT) are calculated and are used as an input for FEM model, these average inputs are applied instead of layup simulation in FEM software. The last two methods have the same simulation strategy, but in different FEM software. In these methods layup are simulated ply by ply on the component, and the input factors are the mechanical constant of each ply without simplification or pre-calculation. The reason for applying two FEM software is to estimate the accuracy of each one in the composite analysis. After simulation and calculation of each

* Corresponding author.
E-mail addresses: hosein.tafaghodi@tuwien.ac.at (H. Tafaghodi Helali), manfred.grafinger@tuwien.ac.at (M. Grafinger).

http://dx.doi.org/10.1016/j.compositesb.2016.04.003
1359-8368/

sample in the FEM software, the precision of outputs including strains and curvatures should be estimated by theoretical results. During the recent researches many useful theoretical calculations have been made for specific models of composite structures [4–6], and in this paper the applicability of the FEM simulation will be estimated.

2. Composite component and material properties

The composite material is regular epoxy-carbon [7] unidirectional, which is very common composite material in the industry which is used for both samples. The first layup type is $[0,\ 45,\ -45,\ 90,\ \overline{0}]_s$ and this 9 layer layup as mentioned in introduction lead to symmetric orthotropic laminate layup for the composite component. The second layup type is $[0, 45, -45, 90]_2$ and this 8 layer layup give us a quasi-isotropic laminate [8].

2.1. Sample 1-tension load: cylindrical tube under pressure

The test component is a cylindrical tube (Fig. 1), by considering that the normal and shear stresses are constant on the surface of the cylinder, the theoretical calculation for the whole surface of cylinder is not necessary. Hence considering a square element on the cylinder surface will lead to the same results and also reduce the theoretical calculation [9].

By making a section on the cylinder the balance equation for achieving the surface stresses can be written (Figs. 2 and 3):

By considering: $\sum F_x = 0$; $2[\sigma_1\ (tdy)] - p(2rdy) = 0$; Then we have: $\sigma_1 = pr/t$

By considering: $\sum F_y = 0$; $\sigma_2(2\pi rt) - p(\pi r^2) = 0$; Then we have: $\sigma_2 = pr/2t$

By importing the cylinder properties in the formulation, it is possible to calculate the stresses for both type of layups:

For 9 layer layup:$\sigma_1 = 22.22$ MPa; $\sigma_2 = 11.11$ MPa; For 8 layer layup:$\sigma_1 = 25$ MPa; $\sigma_2 = 12.5$ MPa

As in the ABD formulation inputs are forces per unit length, the obtained stresses should be converted:

For 9 layer layup: $N_{1,2} = \sigma_{1,2} \times t$; $N_1 = 50\ N/mm$; $N_2 = 25\ N/mm$

For 8 layer layup: $N_{1,2} = \sigma_{1,2} \times t$; $N_1 = 50\ N/mm$; $N_2 = 25\ N/mm$

2.2. Sample 2-Bending load: surface with bending moment

The test component of this sample is a square surface ($100 \times 100\ mm^2$) with a bending moment at the two side of it (Fig. 4). In contrast with previous sample, there will be no simplification, because the model is quite simple. The moment amount on each side of the surface is 5 N × m in the y direction.

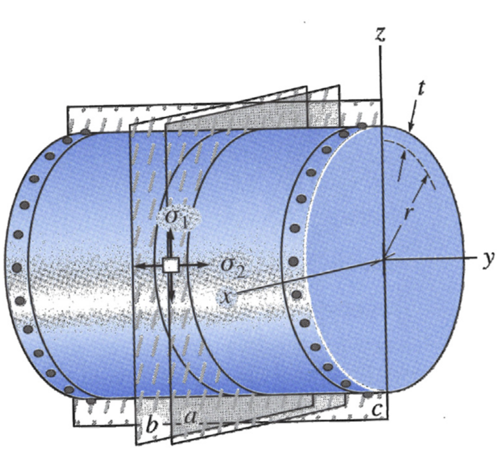

Fig. 1. Tension load on element at cylinder surface.

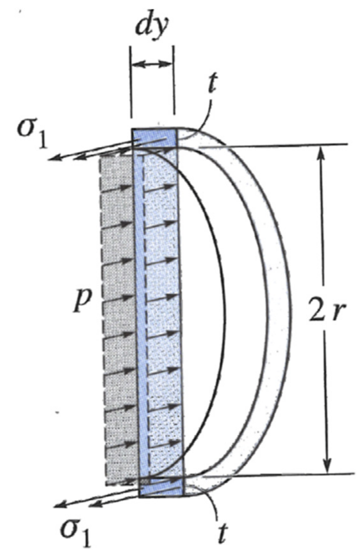

Fig. 2. Tension in X direction.

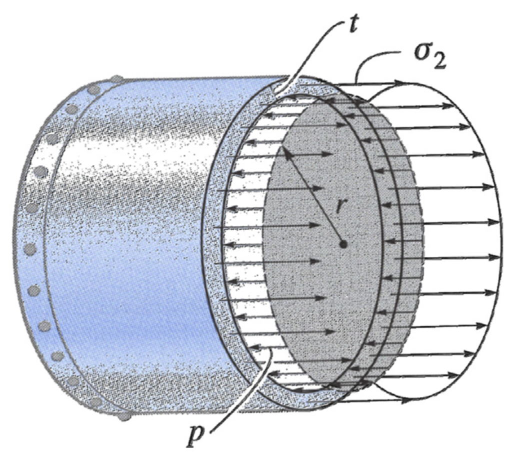

Fig. 3. Tension in Y direction.

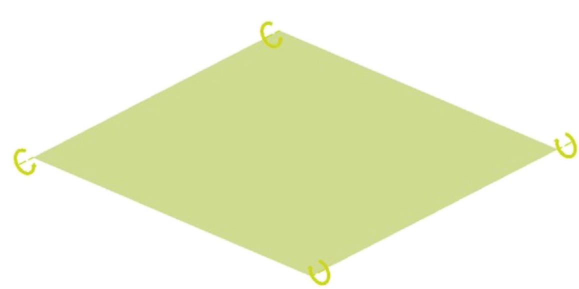

Fig. 4. Sample 2_surface under bending load.

For this sample as it mentioned in introduction one type of layup is applied. The symmetric orthotropic laminate layup (the 9 layer layup $[0,\ 45,\ -45,\ 90,\ \overline{0}]_s$) is considered for this sample. It should be noticed that in the ABD formulation the unit of the bending moments in X and Y directions are moment per unit length, therefore the moment amount dimension should be changed to moment per length and then can be used in the ABD matrix. So it could be written:

$$M_y = 5N \times m/_{0.05m} = 100^{N\times m}/_m$$

3. Theoritical calculation with ABD matrix

First the ABD matrix should be calculated for the cylinder based on its material and layup properties, then by having the ABD matrix and forces that calculated in part 2 (ε_x, ε_y, ε_{xy}) could be obtained. This theoretical calculation leads to a pattern for the FEM simulations [3,10,11].

Second the ABD matrix should be calculated for the surface based on the mechanical constants of the Table 1. Instead of forces in this sample the moment M_y is acted on the ages of surface and it should be placed into the Hooke's law equation. After solving the equation the curvatures (κ_x, κ_y, κ_{xy}) will be the output. The ABD formulation is introduced based on Hooke's law and orthotropic material properties:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \\ - \\ M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} A_{11} & A_{12} & A_{16} & | & B_{11} & B_{12} & B_{16} \\ A_{21} & A_{22} & A_{26} & | & B_{21} & B_{22} & B_{26} \\ A_{61} & A_{62} & A_{66} & | & B_{61} & B_{62} & B_{66} \\ - & - & - & + & - & - & - \\ B_{11} & B_{12} & B_{16} & | & D_{11} & D_{12} & D_{16} \\ B_{21} & B_{22} & B_{26} & | & D_{21} & D_{22} & D_{26} \\ B_{61} & B_{62} & B_{66} & | & D_{61} & D_{62} & D_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ - \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

ABD matrix is symmetric, and so the 3 × 3 submatrices A and D along main diagonal symmetric ($A_{ij} = A_{ji}$ and $D_{ij} = D_{ji}$), however the B matrix is not guarantee to be symmetric.

- N_x,N_y are vertical forces per unit length
- N_{xy} is shear force per unit length
- M_x,M_y are bending moment per unit length
- M_{xy} is twisting moment per unit length.

Table 1
Properties of composite material.

Composite material properties	
Number of the layers	9 or 8
Each layer thickness [mm]	0.25
Longitude stiffness E_1 [N/mm2]	135,000
Transverse stiffness E_2 [N/mm2]	9500
Poisson ratio ν	0.326
Shear stiffness G_{12} [N/mm2]	5270

Forces (N vectors) in the equation are related to strains (ε vectors) with A submatrix and to curvature (κ vectors) with B matrix, so moments (M vectors) are related to curvatures with D matrix and to strains by B matrix. Mechanical behavior of ABD matrix comes in following figure, symmetric layups are selected to make B submatrix equal to zero, to avoid the combination of mechanical behaviors and have more analyzable mechanical behavior [12] (Fig. 5).

3.1. Theoretical calculation for sample 1

Vertical forces (N_x, N_y) for 9 layer layup have been calculated for sample 1 in Section 2 and by having the layup information the ABD matrix is also calculate able (ABD matrix properties come from eLamX2 software[1]) [15], For acquiring the strain vector, ABD matrix should be reversed until the equation could be solved:

$$\begin{Bmatrix} 24.97 \\ 49.99 \\ 0 \\ - \\ 0 \\ 0 \\ 0 \end{Bmatrix} = \begin{bmatrix} 150026.1 & 36587.8 & 0 & | & 0 & 0 & 0 \\ 36587.8 & 118414.7 & 0 & | & 0 & 0 & 0 \\ 0 & 0 & 41424.5 & | & 0 & 0 & 0 \\ - & - & - & + & - & - & - \\ 0 & 0 & 0 & | & 8650.2 & 15127.5 & 4939.3 \\ 0 & 0 & 0 & | & 15127.5 & 27214.2 & 4939.3 \\ 0 & 0 & 0 & | & 4939.3 & 4939.3 & 17168 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \\ - \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ 0 \\ - \\ 0 \\ 0 \\ 0 \end{Bmatrix} = \begin{bmatrix} 7.209 \times 10^{-6} & -2.227 \times 10^{-6} & 0 & | & 0 & 0 & 0 \\ -2.227 \times 10^{-6} & 9.133 \times 10^{-6} & 0 & | & 0 & 0 & 0 \\ 0 & 0 & 2.414 \times 10^{-5} & | & 0 & 0 & 0 \\ - & - & - & + & - & - & - \\ 0 & 0 & 0 & | & -0.018 & 0.01 & 0,002 \\ 0 & 0 & 0 & | & 0.01 & -0.005 & -0,001 \\ 0 & 0 & 0 & | & 0.002 & -0.001 & -2,713 \times 10^{-4} \end{bmatrix} \begin{Bmatrix} 25 \\ 50 \\ 0 \\ - \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

As expected ε_x and ε_y are calculated from above equation and normal ε can be obtained. Normal ε is the result which will be compared with the FEM results and it calculated as below:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} = \sqrt{\left(6.9 \times 10^{-5}\right)^2 + \left(4.01 \times 10^{-4}\right)^2}$$
$$= 3.267 \times 10^{-4}$$

By the exact same way and inputting properties of 8 layer layup in the ABD matrix ε_x and ε_y can be obtained, the normal strain is calculated as below:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} = \sqrt{\left(9.1 \times 10^{-5}\right)^2 + \left(4.03 \times 10^{-4}\right)^2}$$
$$= 4.131 \times 10^{-4}$$

Deformation of Square element is shown in Fig. 6, deformations are calculated from the theoretical method and appear in X and Y directions.

[1] Software website link (01.06.2015): http://tudresden.de/die_tu_dresden/fakultaeten/fakultaet_maschinenwesen/ilr/aero/download/laminatetheory/index_html.

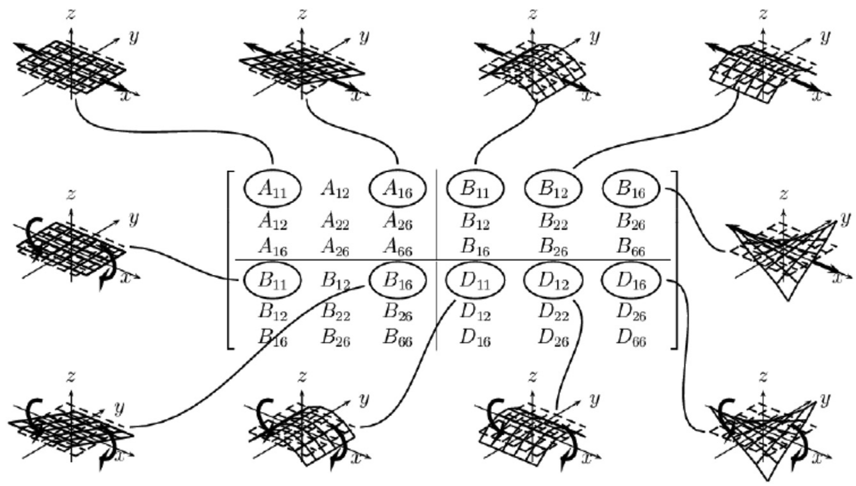

Fig. 5. ABD matrix and mechanical behaviors.

3.2. Theoretical calculation for sample 2

After calculating the ABD matrix in the ElamX2 software and having the moment M_y, the Hooke's law equation can be rewritten for this sample. By the same way of sample 1, to obtain the curvature the equation should be reversed, so it could be written as below:

$$\begin{Bmatrix} 0 \\ 0 \\ 0 \\ - \\ 0 \\ -100 \\ 0 \end{Bmatrix} = \left[\begin{array}{ccc|ccc} 150026.1 & 36587.8 & 0 & 0 & 0 & 0 \\ 36587.8 & 118414.7 & 0 & 0 & 0 & 0 \\ 0 & 0 & 41424.5 & 0 & 0 & 0 \\ \hline 0 & 0 & 0 & 8650.2 & 15127.5 & 4939.3 \\ 0 & 0 & 0 & 15127.5 & 27214.2 & 4939.3 \\ 0 & 0 & 0 & 4939.3 & 4939.3 & 17168 \end{array}\right] \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \\ - \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix}$$

$$\begin{Bmatrix} 0 \\ 0 \\ 0 \\ - \\ \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix} = \left[\begin{array}{ccc|ccc} 7.209\times10^{-6} & -2.227\times10^{-6} & 0 & 0 & 0 & 0 \\ -2.227\times10^{-6} & 9.133\times10^{-6} & 0 & 0 & 0 & 0 \\ 0 & 0 & 2.414\times10^{-5} & 0 & 0 & 0 \\ \hline 0 & 0 & 0 & -0.018 & 0.01 & 0.002 \\ 0 & 0 & 0 & 0.01 & -0.005 & -0.001 \\ 0 & 0 & 0 & 0.002 & -0.001 & -2.713\times10^{-4} \end{array}\right] \begin{Bmatrix} 0 \\ 0 \\ 0 \\ - \\ 0 \\ -100 \\ 0 \end{Bmatrix}$$

As the result for the curvature it can be written:

$$\begin{Bmatrix} \kappa_x \\ \kappa_y \\ \kappa_{xy} \end{Bmatrix} = \begin{Bmatrix} 6.82\times10^{-4} \\ -4.239\times10^{-3} \\ 1.024\times10^{-3} \end{Bmatrix}$$

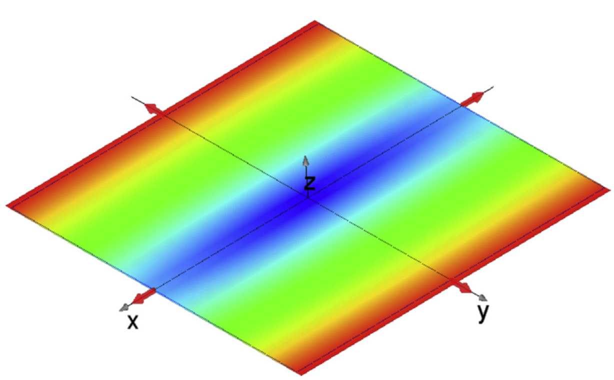

Fig. 6. Element deformation in theoretical calculation.

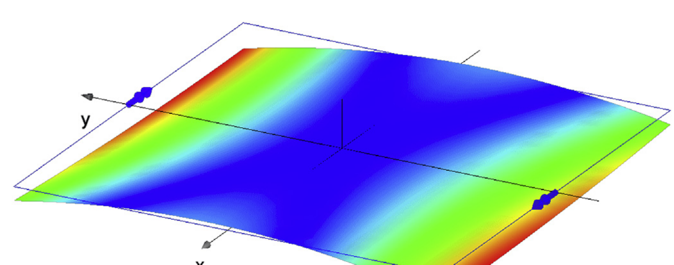

Fig. 7. Element curvature in theoretical calculation.

The curvature of the square element for this sample has been illustrated in the Fig 7:

These obtained strains and curvatures are considered as the theoretical result of the ABD method. The ABD method is considered as the reference method for the sample 1 and 2 and the accuracy of each FEM method can be obtained by comparison with the ABD method.

4. CATIA FEM simulation with average elastic factors input

The classical laminate theory (CLT) is a method for conducting calculation on multi-ply inhomogeneous laminate layups. The plies rigidities $[Q]_k$ for orthotropic plies in the ply coordinate system is determined as follows (ply and material properties used based on Tables 1 and 2 on Section 2 for 9 layer layup) [13,14]:

$$[Q]_k = \begin{bmatrix} \dfrac{E_1}{1-\nu_{12}^2\cdot\dfrac{E_2}{E_1}} & \dfrac{\nu_{12}\cdot E_2}{1-\nu_{12}^2\cdot\dfrac{E_2}{E_1}} & 0 \\ \dfrac{\nu_{12}\cdot E_2}{1-\nu_{12}^2\cdot\dfrac{E_2}{E_1}} & \dfrac{E_2}{1-\nu_{12}^2\cdot\dfrac{E_2}{E_1}} & 0 \\ 0 & 0 & G_{12} \end{bmatrix} = \begin{bmatrix} 1.36\cdot10^{11} & 3.12\cdot10^{9} & 0 \\ 3.12\cdot10^{9} & 9.572\cdot10^{9} & 0 \\ 0 & 0 & 5.27\cdot10^{9} \end{bmatrix}$$

The rigidities $[Q]_{PlyCS,k}$ of each ply are transformed from local ply coordinate system (Ply CS) to the global coordinate system of the laminate (Lam CS) in accordance with the transformation laws.

$$[Q]_{LamCS,k} = [T_\sigma]_k\cdot[Q]_{PlyCS,k}\cdot[T_\sigma]_k^T \text{ With:}$$

$$[T_\sigma]_k = \begin{bmatrix} \cos^2 & \sin^2 & 2\sin\cos \\ \sin^2 & \cos^2 & -2\sin\cos \\ -\sin\cos & \sin\cos & \cos^2-\sin^2 \end{bmatrix}$$

Based on material properties $[Q]_k$ should be calculated for all layup angels which are [0, 45, −45, 90]. The transformed rigidities of all plies, weighted according to their cross sectional ratios, are added together yield a homogeneous laminate rigidity. The result is the laminate rigidity matrix [A]:

$$[A] = \sum_k \frac{t_k}{t_{Lam}}\cdot[Q]_{LamCS,k} = \frac{3\cdot0.25}{2.25}\cdot[Q]_{lam,1} + \frac{2\cdot0.25}{2.25}\cdot[Q]_{lam,2} + \frac{2\cdot0.25}{2.25}\cdot[Q]_{lam,3} + \frac{2\cdot0.25}{2.25}\cdot[Q]_{lam,4}$$

Before distortions in the laminate can be determined, first inversion of the rigidity matrix [A] should be calculated to obtain the compliance Matrix [a]: $[a] = [A]^{-1}$

Table 2
Layup properties of cylinder.

Properties of cylinder	9 layer layup	8 layer layup
Pressure (p) MPa	5	5
Radios of cylinder (r) mm	10	10
Thickness of cylinder (t) mm	$(9 \times 0.25) = 2.25$	$(8 \times 0.25) = 2$
Thickness area of square element (A) mm^2	$(2 \times 2.25) = 4.5$	$(2 \times 2) = 4$

$$[A] = \begin{bmatrix} 6.668\cdot10^{10} & 1.626\cdot10^{10} & 0 \\ 1.626\cdot10^{10} & 5.263\cdot10^{10} & -1.907\cdot10^{-6} \\ 0 & -1.907\cdot10^{-6} & 1.841\cdot10^{10} \end{bmatrix}; \; [a] = \begin{bmatrix} 1.622\cdot10^{-11} & -5.012\cdot10^{-12} & -5.192\cdot10^{-28} \\ -5.012\cdot10^{-12} & 2.055\cdot10^{-11} & 2.129\cdot10^{-27} \\ -5.192\cdot10^{-28} & 2.129\cdot10^{-27} & 5.432\cdot10^{-11} \end{bmatrix}$$

The engineering constants for the laminate are obtained from elements of the compliance matrix:

$$E_{x,Lam} = \frac{1}{a_{11}} = 61652\left(N/mm^2\right);\; E_{y,Lam} = \frac{1}{a_{22}} = 48660\left(N/mm^2\right);$$

$$G_{xy,Lam} = \frac{1}{a_{33}} = 18409\left(N/mm^2\right);\; \nu_{xy,Lam} = \nu_{yx,Lam} = \frac{-a_{21}}{a_{11}} = 0.309;$$

By the same calculation procedure for 8 layer layup the mechanical constants of 8 layer layup can be obtained:

$$E_{x,Lam} = \frac{1}{a_{11}} = 52493\left(N/mm^2\right);\; E_{y,Lam} = \frac{1}{a_{22}} = 52493\left(N/mm^2\right);$$

$$G_{xy,Lam} = \frac{1}{a_{33}} = 20052\left(N/mm^2\right);\; \nu_{xy,Lam} = \nu_{yx,Lam} = \frac{-a_{21}}{a_{11}} = 0.309;$$

4.1. Simulation for sample 1

For quasi-isotropic laminate layups (8 layer layup)$E_x = E_y$, but when this method is used for symmetric orthotropic laminate layups (9 layer layup)$E_x \neq E_y$, as in average method just one input is possible for E modulus, E_x is used as the input elasticity modulus in 9 layup method. These engineering constants now can be used as material properties in the CATIA model.

The cylinder is modeled (Fig. 8 left side) according to the geometry properties of Table 2 in the CATIA software. The boundary conditions are placed at the two sides of the cylinder (two circles marked by red color) that make a uniform deformation on the cylinder surface. The meshing on the surface of the cylinder should be square and with specific size (2 mm), in this case the strains of one square element on the cylinder surface (2×2 mm^2) will be appropriate for comparison with the theoretical result (Fig. 8 right side). Inside pressure and thickness (Table 2) of the cylinder are also considered for the model in software. After modeling and calculating the results in the CATIA software, the strain of the cylinder will be available:

Although all of strains on the surface of cylinder have the same amount, one element on the surface is selected to establish the strain amounts:

For 9 layer layup (using E_x as input):

$\varepsilon_x \approx 7.75 \times 10^{-5}$ And $\varepsilon_y \approx 3.23 \times 10^{-4}$ so we have:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} \approx 3.32 \times 10^{-4}$$

For 8 layer layup:

$\varepsilon_x \approx 4.33 \times 10^{-4}$ And $\varepsilon_y \approx 1.02 \times 10^{-4}$ so we have:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} \approx 4.446 \times 10^{-4}$$

4.2. Simulation for sample 2

The whole calculation process of 9 layer layup for sample 2 is similar to the sample 1. The material properties are also similar to the sample 1 properties. Therefore the calculated mechanical constants can be used for sample 2.

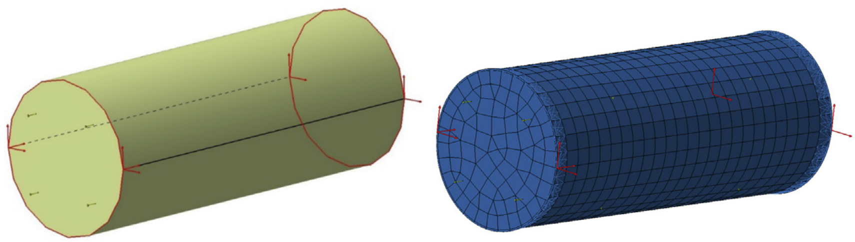

Fig. 8. Cylinder with boundary conditions and with 2 mm mesh.

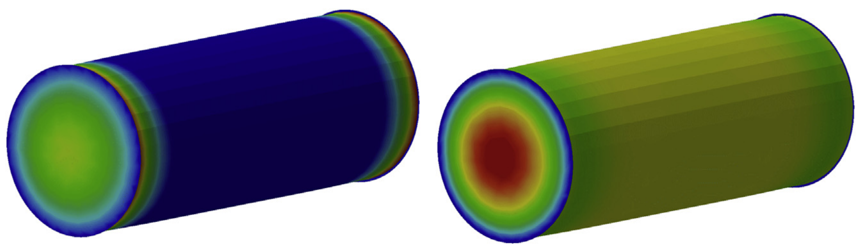

Fig. 9. Strain of cylinder (ε_x in left side and ε_y in right side for 9 layer layup).

The square surface of sample 2 is modeled in Fig. 10 (right side). The moments are positioned on two edge of the surface (5 N × m). It should be noticed that, for the boundary condition first the surface should have a pure bending and second the surface should be fixed in the way that it cannot be moved in any direction. Therefore three fix points are considered for the boundary condition on the middle of the surface. The mesh type is the normal quadrat mesh but the mesh size is important in this sample. As this surface model simulates the theoretical square element, the mesh size should be as large as possible. Therefore the mesh size is considered 25 mm, which divides the whole surface to 16 elements and the quantity of curvature will be more realistic. According to the laminate layup definition in Table 1, the thickness of each ply is considered 0.25 mm and the number of layup for sample 2 is 9, therefore the thickness of the surface will be 2.25 mm.

After modeling and calculation the results in the CATIA software, the deformation of the surface will be available as it is illustrated in Fig. 11. The maximum displacement amount is 1.05 mm on the red edges of the surface. In fact the factor that should be extracted in this sample is the curvature in X and Y directions. There is an option in the CATIA software which shows the curvature quantity of composite part in all directions as it is illustrated in Fig. 12. Otherwise the radius of deformation should be calculated and by reversing the radius, the curvature will be obtained. Therefore the curvature of surface will be:

$$\begin{Bmatrix} \kappa_x \\ \kappa_y \end{Bmatrix} = \begin{Bmatrix} 4.5 \times 10^{-4} \\ -1.33 \times 10^{-3} \end{Bmatrix}$$

5. FEM simulation with complete layups modeling

We talked about two FEM simulation with complete layups in introduction. In this method theoretical calculation is not necessary, because the laminate is modeled ply by ply in the FEM software. The samples are modeled with two different FEM software (CATIA, ANSYS) in this simulation; therefore there will be two types of result for one sample model. The question that comes to mind here is: why should this simulation be done with two FEM software? This is done because of two reasons. First if the results of both FEM software were nearly the same, it would prove that this method is acceptable. Second the comparison of the results of the two software will lead to a better overview over the precision of each FEM software. As these two methods have the same modeling procedure, they are studied together in this section. The samples of the simulation are the same as Figs. 8 and 10 with the same

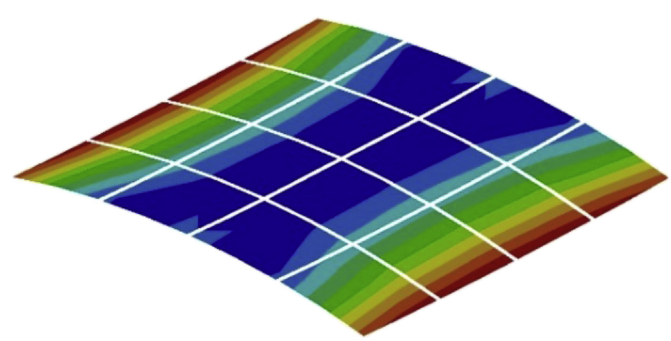

Fig. 11. Sample 2 deformation.

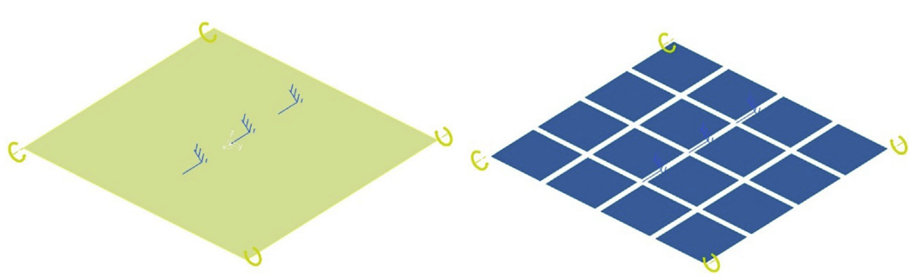

Fig. 10. Surface with boundary conditions and square quadrat mesh.

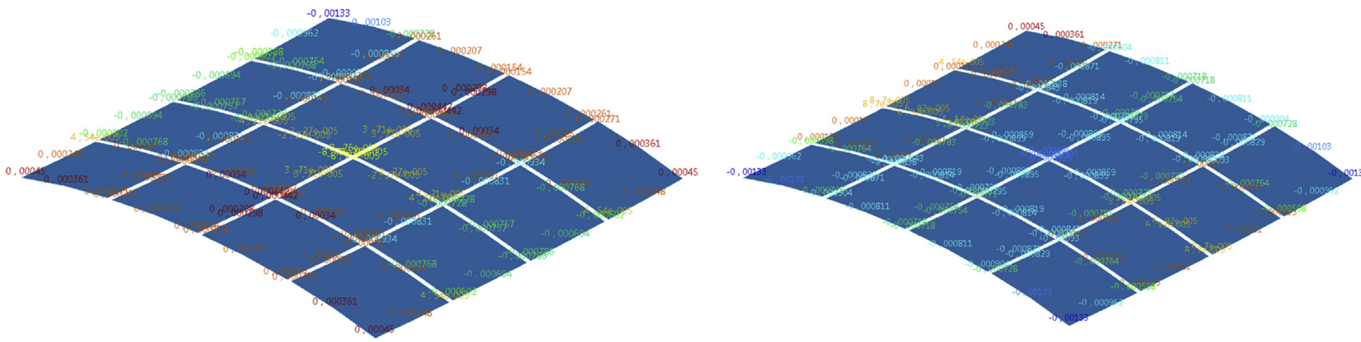

Fig. 12. Curvature in X (left side) and Y (right side) directions.

Table 3
Properties of composite layup.

Composite layup properties for orthotropic	
Number of the layers	9 or 8
Each layer thickness	0.25
Young's Modulus in X direction E_1 [N/mm2]	135,000
Young's Modulus in Y direction E_2 [N/mm2]	9500
Young's Modulus in Z direction E_{12}	9500
Poisson ratio ν_{xy}	0.326
Poisson ratio ν_{yz}	0.4
Poisson ratio ν_{xz}	0.4
Shear modulus G_{xy} [N/mm2]	5270
Shear modulus G_{yz} [N/mm2]	3100
Shear modulus G_{xz} [N/mm2]	3100

properties. in this method all layers are modeled ply by ply on the sample surface, and the orthotropic composite properties of regular epoxy carbon is considered for each layer, which are illustrated in Table 3.

As the 2D properties of orthotropic material are used for the surface layup of the samples, some properties of Table 3 (ν_{yz}, ν_{xz}, G_{yz} and G_{xz}) are not used for the layup.

5.1. Simulation for sample 1

In CATIA each layer angle is represented by separate color as shown in Fig. 13 (left-side), and as it is clear all 9 layers modeled on cylinder surface (for 9 layer layup). ANSYS PrePost (the composite area of ANSYS for preprocessing and post processing of composite layups) shows each layer by its own angle on surface, as example layer with 45° is shown in Fig. 13 (right-side). In this method one square element (2 × 2) on the surface of cylinder is also considered to compare with theoretical results. Hence the mesh size is 2 mm and mesh type is 2nd order square (Mesh size and pressure are based on Table 2). After calculating the model with the described layups and meshes, the results will be ready. Fig. 13 shows the strain factor in X and Y directions in CATIA software.

In the ANSYS software the process of getting to the strain is somehow easier, because there is an option in the software called equivalent elastic strain. By using this option the equivalent strain ε will be illustrated (Fig. 15). The strain is uniform on the cylinder surface (Figs. 14 and 15) both in CATIA and ANSYS. Note that like the average method, in this method the sample (2 × 2 square element) should also be selected from the middle of the cylinder surface and not near the edges to avoid the influence of boundary conditions and have a correct sample.

For 9 layer layup:

$\varepsilon_x \approx 9.2 \times 10^{-5}$ And $\varepsilon_y \approx 3.4 \times 10^{-4}$ so we have:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} \approx 3.52 \times 10^{-4}.$$

For 8 layer layup:

$\varepsilon_x \approx 7.19 \times 10^{-5}$ And $\varepsilon_y \approx 4.41 \times 10^{-4}$ so we have:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} \approx 4.46 \times 10^{-4}.$$

The results for ANSYS software:

For 9 layer layup: $\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} \approx 3.70 \times 10^{-4}$; For 8 layer layup:

$$\varepsilon = \sqrt{\varepsilon_x^2 + \varepsilon_y^2} \approx 4.54 \times 10^{-4}.$$

In the composite post processing of the FEM software, it is also possible to have a plot of the strain and stress for each ply separately. With this possibility the stress distribution at each ply of the 9 layer layup are established and compared with theoretical results to make sure about the validity of the calculations. The left side Fig. 16 are theoretical stress distribution values in each layer for the

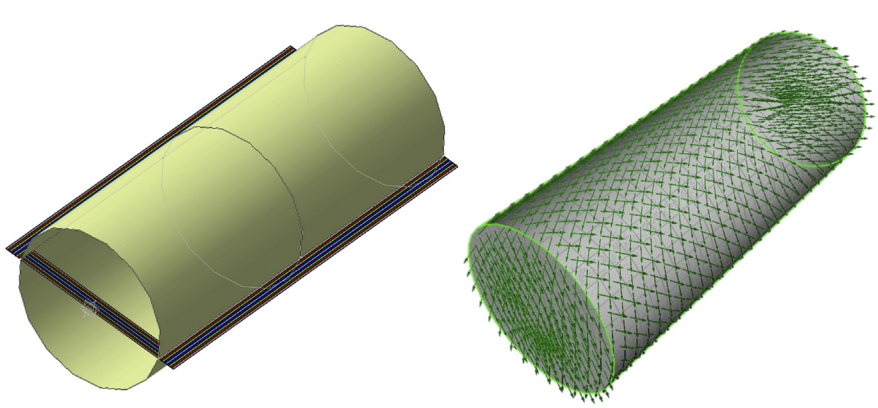

Fig. 13. Composite layups of CATIA (left-side) and ANSYS (right-side).

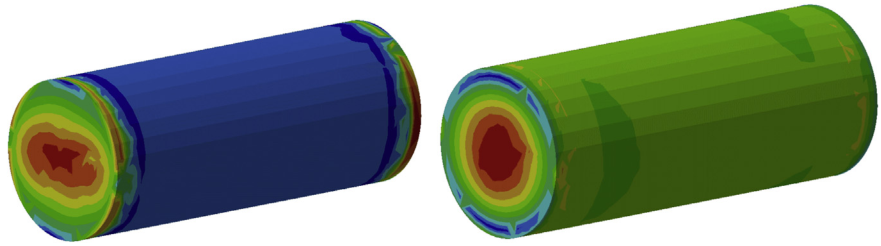

Fig. 14. Cylinder strain in CATIA (ε_x in left side and ε_y in right side for 9 layer layup).

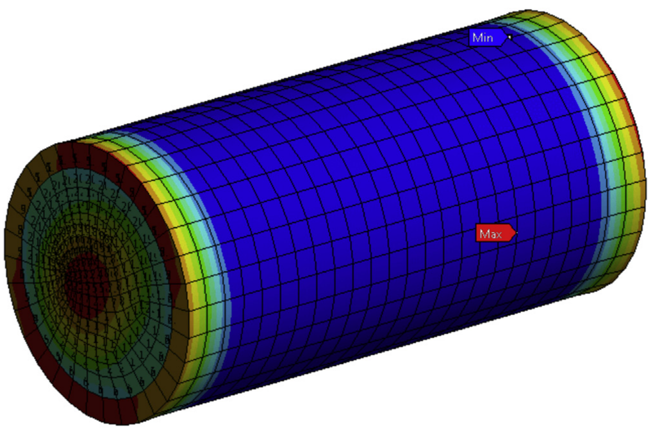

Fig. 15. Cylinder equivalent elastic strain in ANSYS for 9 layer layup.

9 layer in X and Y directions (established by eXlam2 software). On the right side, there are the values for two FEM software is also in the X and Y directions. As it can be seen from the figures, both FEM software have a good adaptation with each other and with the theoretical stress distributions.

5.2. Simulation for sample 2

In case that all method could be compared to each other, they should have a same model and same mesh size. Therefore the sample model is similar to the average method model (Fig. 10) with the same properties; and the mesh size is considered 25 mm similar to the average method (Fig. 10). The layup procedure on the surface is ply by ply, the orthotropic composite properties of regular epoxy carbon are considered for each ply according to the Table 3. The 9 layer layup should be modeled in the CATIA and ANSYS. The layup simulation procedure for surface is similar to the procedure described for cylinder in Section 5.1. The moments and boundary conditions are similar to the average method (Fig. 9). The surface thickness after layup will be 2.25 mm (0.25 mm for each ply). The outer edge of the red element in the Fig. 17 is considered for extracting the curvature of the surface. In fact the maximum curvature occurs at this place. After calculating the model with the following laminate layup and mesh properties, the result will be accessible. The deformation of the surface in the CATIA software is illustrated in Fig. 18 left side. The maximum deformation amount is 2.9 mm on the red edge of the surface. And similar to the average method curvatures can be extracted from CATIA software as it shown in the Fig. 18 right side.

Therefore the curvature vector of the CATIA model can be written as below:

$$\begin{Bmatrix} \kappa_x \\ \kappa_y \end{Bmatrix} = \begin{Bmatrix} 4.69 \times 10^{-4} \\ -3.4 \times 10^{-3} \end{Bmatrix}$$

In the ANSYS software the procedure of calculating the curvature factors is somehow harder, because there is no option for showing the curvature factors. Therefore in this case some calculations should be done first deformation should be calculated, second radius should be measured from the deformation in each direction, third the radius should be reversed until the curvature can be obtained. The maximum deformation as it illustrated in Fig. 19 is 2.95 mm which also occurs on the outer edge of the element. It should be noticed that this amount expresses the total deformation. Hence the deformation which should be used for calculating the curvature has different amount in each direction.

After calculating the curvature based on the deformation in each direction, the curvature vector of ANSYS software can be written:

$$\begin{Bmatrix} \kappa_x \\ \kappa_y \end{Bmatrix} = \begin{Bmatrix} 5.75 \times 10^{-4} \\ -3.51 \times 10^{-3} \end{Bmatrix}$$

6. Comparison

Based on what has been calculated and modeled in previous sections, it is possible now to compare the results. In these samples (sample 1 and 2) the theoretical method is the reference method, and the three other methods are compared to this method in case the precision of each be evaluated.

6.1. Sample 1-tension load

The strain results for 8 and 9 layer layup are illustrated in Table 4. By considering the loading on sample 1 is tension load, the average method for both type of layups is applicable. In this sample both complete methods have also acceptable results in compare of the theoretical results. Although in sample 1 there is no difference between using the average method or the complete method, the average method is simpler and less time consuming. Therefore instead of simulating the whole laminate layup (complete method), the average method can be used and the similar results could be obtained. This method has even been tested and used in industry for different type of layups even with honeycomb, and the results were quite satisfactory and passed the industry test results.

6.2. Sample 2-bending load

In contrast with tension load in sample 1 where the average method had good results for both layups, the average method outcomes for bending load in sample 2 are really weak and non-acceptable. In this method the maximum deformation occurs in the middle of surface edges as it is illustrated in Fig. 11. But as it displayed in Fig. 20 the maximum deformation should be at the end of the edges. That means the shape of deformation and also its quantity in the average method is not correct. According to the Table 5, the major curvature occurs in Y direction. By the simple comparison between the average method amount and the

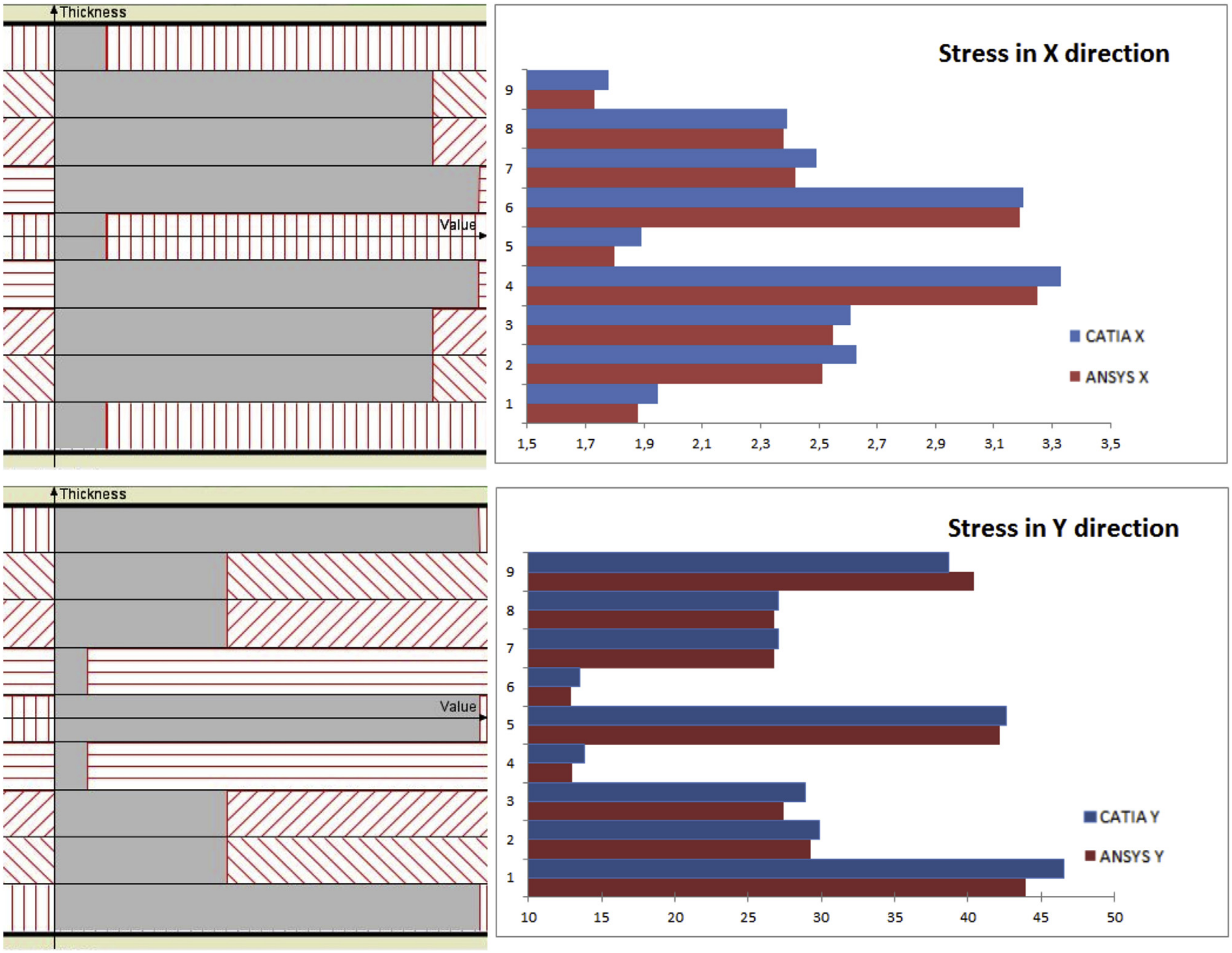

Fig. 16. Cylinder stress distribution in each layer in X and Y directions (9 layer layup).

theoretical amount, it would be realized that the average method curvature is lower than even half of the theoretical method amount. Therefore when the force and moments have a bending and buckling effect, the position of plies will play an effective role on the precision of the result. Since in the average method the position of the ply is not considered (according to CLT formulation [10]), its results are not precise enough.

According to the Table 5 both complete methods (with CATIA and ANSYS) have a good coverage of the results in compare to the theoretical results. After comparing the sections of Fig. 20, it would be clear that the complete method has a correct distribution of deformation over the surface of sample. Hence in case of bending or buckling loadings acting on the composite structure, the complete layup simulation would be a good choice for analyzing the structure and the result will be more realistic. There is a difference between the theoretical and the complete methods results visible in sample 2, and that could be related to the assumption which has been made for the theoretical calculation in ABD matrix. These assumptions simplify the procedure of the theoretical calculation and also neglect some boundaries of calculation precision. For instance in ABD formulation [1,3] it has been mentioned that a line straight and perpendicular to the middle surface remains straight and perpendicular to the middle surface during deformation. The assumption is not necessarily considered by the FEM calculation; therefore the complete method has more realistic results. This is the reason the average method shows more conformity with theoretical results in sample 1.

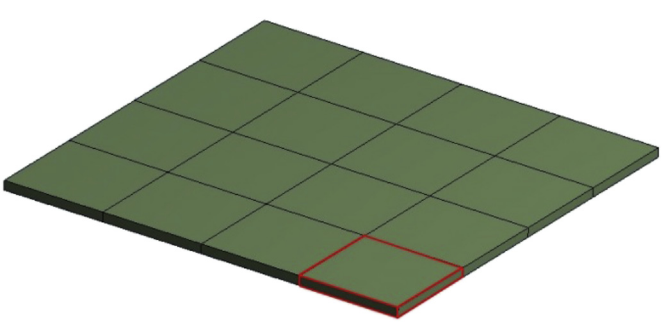

Fig. 17. Surface after layup.

With all of these benefits reaching the complete simulation is not always simple, imagine the layer thicknesses are less than 0.1 mm and the number of layers exceed to 100 with the orthotropic orientation (for example$[45, -45]_n$). In these cases, simulation with the CATIA takes lots of time and is sometimes even impossible, but there are some options in ANSYS that help the user simulate the model easier and faster. That is why the complete simulation is constructed with two different FEM software, to show

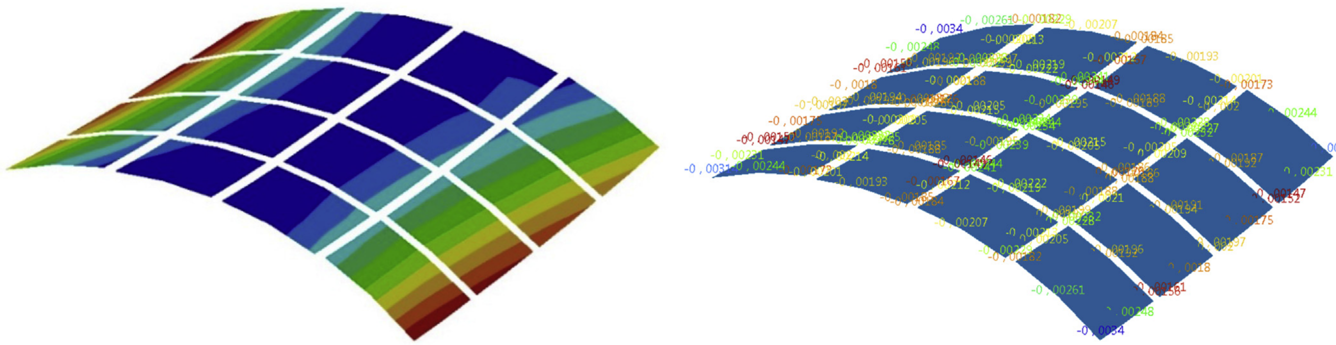

Fig. 18. Surface deformation in CATIA (left side), surface curvature in Y direction (right side).

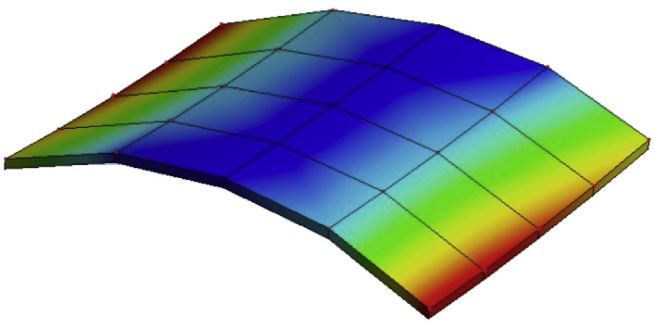

Fig. 19. Surface deformation in ANSYS.

the similarity of results in one method using different FEM software. However the possibility of simulation depends on the software facilities.

7. Conclusion

The theoretical calculation is precise and trustable because it is based on proven formulations. Yet some questions still continue to arise. For example, how far the theoretical calculation can be applied? What should be done for complicated composite parts? How much time and energy should be used for calculation of even a simple part like these two samples? And the most important question is whether the theoretical calculation is even possible in complicated cases? The answer is clear and that's why in recent years the FEM simulation is improved and applied much more than before, and because in composite material analysis calculations are even more complicated, the FEM simulation is more intransitive.

Table 4
Sample 1_strain results of different methods for 8 and 9 layer layup.

Sample1- cylinder under pressure		
Comparison of strain Results	ε for 9 layer layup	ε for 8 layer layup
Theoretical calculation	3.267×10^{-4}	4.131×10^{-4}
CATIA simulation with average factors	3.32×10^{-4}	4.44×10^{-4}
CATIA simulation with complete layups modeling	3.52×10^{-4}	4.46×10^{-4}
ANSYS simulation with complete layups modeling	3.70×10^{-4}	4.54×10^{-4}

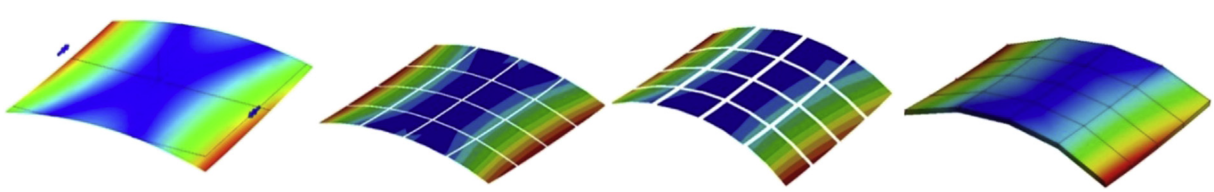

Fig. 20. Surface deformation of all four methods.

Table 5
Sample 2_curvature results of different methods for 9 layer layup.

Sample 2- surface under bending moment		
Comparison of curvature results	κ in X direction	κ in Y direction
Theoretical calculation	6.82×10^{-4}	-4.239×10^{-3}
CATIA simulation with average factors	4.5×10^{-4}	-1.33×10^{-3}
CATIA simulation with complete layups modeling	4.69×10^{-4}	-3.4×10^{-3}
ANSYS simulation with complete layups modeling	5.75×10^{-4}	-3.51×10^{-3}

The average method is a good compromise between time consumption and quality of results for composite layups under tension load and even for the simulations where a honeycomb exists between the layup. However when the main loading have a bending and buckling effect, the analysis of this method is not precise anymore. In fact, in the bending analysis the position of each layer orientation is quiet important but the average method does not consider the position of the oriented layer. The complete layup simulation as it comes from its name is of course more precise and also time consuming process for the orthotropic laminates. But if the main load causes bending or twisting moment, there will be no choice but the complete simulation method. Having the stress, strain and failure data of each layer separately is another advantage of this method. In the post processing section of FEM software it is possible to check out the deformation and also stress distribution of each ply separately. This option is really useful for finding the failure plies, after identifying these plies their position could be changed or the number of plies could be increased to avoid the failure of the laminate.

Studying these two very simple samples creates an overview on how the FEM simulations are acting and what are their advantage and disadvantages. Hence before getting involved in the simulation process, first the composite layup should be studied and an appropriate method should be selected.

Software

1. ANSYS wokbench16, Static structure, Composite PrePost modules
2. CATIA, Part design, Surface design, Composite design modules
3. ElamX2,2

References

[1] Altenbach H, Altenbach J, Kissing W. Mechanics of composite structural elements. Springer; 2004. p. 27–33.
[2] Nettles AT. Basic mechanics of laminated composite plates. NASA reference publication; 1994. p. 20–7. 1351.
[3] Kaw Autur K. Mechanics of composite materials. 2nd ed. Taylor & Francis Group; 2006. p. 326–32. P. 79–99.
[4] Barretta R, Feo L, Luciano R, Marotti de Sciarra F. An Eringen-like model for Timoshenko nanobeams. Compos Struct 2016;139:104–10.
[5] Barretta R, Feo L, Luciano R. Some closed-form solutions of functionally graded beams undergoing nonuniform torsion. Compos Struct 2015;123:132–6.
[6] Barretta R, Feo L, Luciano R. Torsion of functionally graded nonlocal viscoelastic circular nanobeams. Compos Part B 2015;72:217–22.
[7] Choi Bok-Lok, Choi Byoung. Numerical method for optimizing variables of carbon-fiber-reinforced epoxy composite coil springs. Compos Part B Eng 2015;82:42–9.
[8] Wang HW, Zhou HW, Gui LL, Ji HW, Zhang XC. Analysis of effect of fiber orientation on young modulus for unidirectional fiber reinforced composites. Compos Part B Eng 2014;56:733–9.
[9] Hibbeler Russell C. Engineering mechanics:statics (German version). Pearson; 2005. p. 694.
[10] Kueh ABH, Pellegrino S. ABD matrix of single-ply triaxial weave fabric composites. Cambridge UK: University of Cambridge; 2007.
[11] Ersoy Hakan. Optimum laminate design by using singular value decomposition. Compos Part B Eng 2013;52:144–54.
[12] Calard Vincent. Formulas and equations for the classical laminate theory. 2011. p. 11.
[13] R&G faserverbundwerkstoffe GmbH. Handbook composite materials. 20031.20–4.
[14] Brunbauer Julia, Pinter Gerald. Fatigue life prediction of carbon fiber reinforced laminates by using cycle-dependent classical laminate theory. Compos Part B Eng 2015;70:167–74.

Web Reference

[15] ELAMX 2,2. Author: Andreas Hauffe, Technical University of Dresden. Last modified: 12.05.2015. Contact person: andreas.hauffe@tu-dresden.de