Research Article

Jerzy Marszałek*, Jacek Stadnicki, and Piotr Danielczyk

Finite element model of laminate construction element with multi-phase microstructure

https://doi.org/10.1515/secm-2020-0044
Received Jul 27, 2020; accepted Sep 28, 2020

Abstract: The article describes a method of creating a mesoscale finite element model of a fabric reinforced laminate that replicates the smallest repetitive fragment of its microstructure – RUC (Repetitive Unit Cell). The model takes into account the influence of the number and orientation of layers, the weave of the reinforcement fabric as well as manufacturing technology on the strength and stiffness of the laminate. The constants of the finite elements forming RUC (equivalent cross-sectional parameters, limit values of forces ensuring layer integrity) are determined experimentally by performing uncomplicated tests of specimens of a particular laminate. A special preprocessor was developed to generate the finite element model of the construction element from laminate, which automatically creates the so-called batch file defining the model. The usefulness of the preprocessor was checked by simulating a three-point bending test of a laminate door beam of a passenger car. The obtained calculation results were verified experimentally.

Keywords: layered composites; delamination; mesoscale finite element model; preprocessing

1 Introduction

Fabric reinforced layer composites – laminates are more and more often used in the construction of planes, boats, wind turbines, vehicles and other devices and machines [1–3]. Compared to other traditional construction materials, such as steel or aluminum alloys, they are light and they have good strength and corrosion resistance [4]. An important feature of laminates is the possibility of shaping their microstructure during manufacture by selecting the type of components, *i.e.* reinforcement and matrix, number and orientation of layers, and component shares in the laminate. This allows to adapt their properties to the nature and direction of the transferred loads, mainly through the appropriate reinforcement arrangement. Laminates are most often used in making thin-walled constructions, which are the equivalents of shell constructions made of traditional materials. A laminate construction should have appropriate functional and mechanical properties, not worse than a construction made of traditional materials, while retaining all the other advantages resulting from using laminates. Designing laminate construction elements is commonly carried out with the support of the finite element method (in short FEM). Compared to analytical methods, such an approach allows you to analyze the strength and stiffness of irregular-shaped elements and take into account many factors determining the mechanical properties of laminates, resulting from their microstructure. Using the finite element method, a laminate can be modeled at one of three levels of observation [5], which differ in the details of microstructure reflection (Figure 1).

At the macroscale level (Figure 1a), laminate is treated as a continuous, homogeneous anisotropic material. For the construction of finite element model (in short FE models), two-dimensional or three-dimensional finite elements of sandwich type are used, due to which macroscale models are characterized by good numerical efficiency. The influence of the number of reinforcement layers, their thickness and location as well as the direction of arrangement are taken into account using the reinforcement theory [6] and the classical lamination theory [7]. Many commercial FEM calculation programs have such analysis methods implemented in which a finite element of the sandwich type is used and the properties of particular layers and, consequently, of the whole laminate are determined using the matrix of material constants, with the quality of the analysis results depending on the values of these constants. As the laminate has a distinct microstructure resulting from maintaining the separation of matrix and reinforcement phases, its resultant properties are not

***Corresponding Author: Jerzy Marszałek:** University of Bielsko-Biala, Bielsko-Biala, Poland; Email: jmarszalek@ath.bielsko.pl
Jacek Stadnicki: University of Bielsko-Biala, Bielsko-Biala, Poland; Email: jstadnicki@ath.bielsko.pl
Piotr Danielczyk: University of Bielsko-Biala, Bielsko-Biala, Poland; Email: pdanielczyk@ath.bielsko.pl

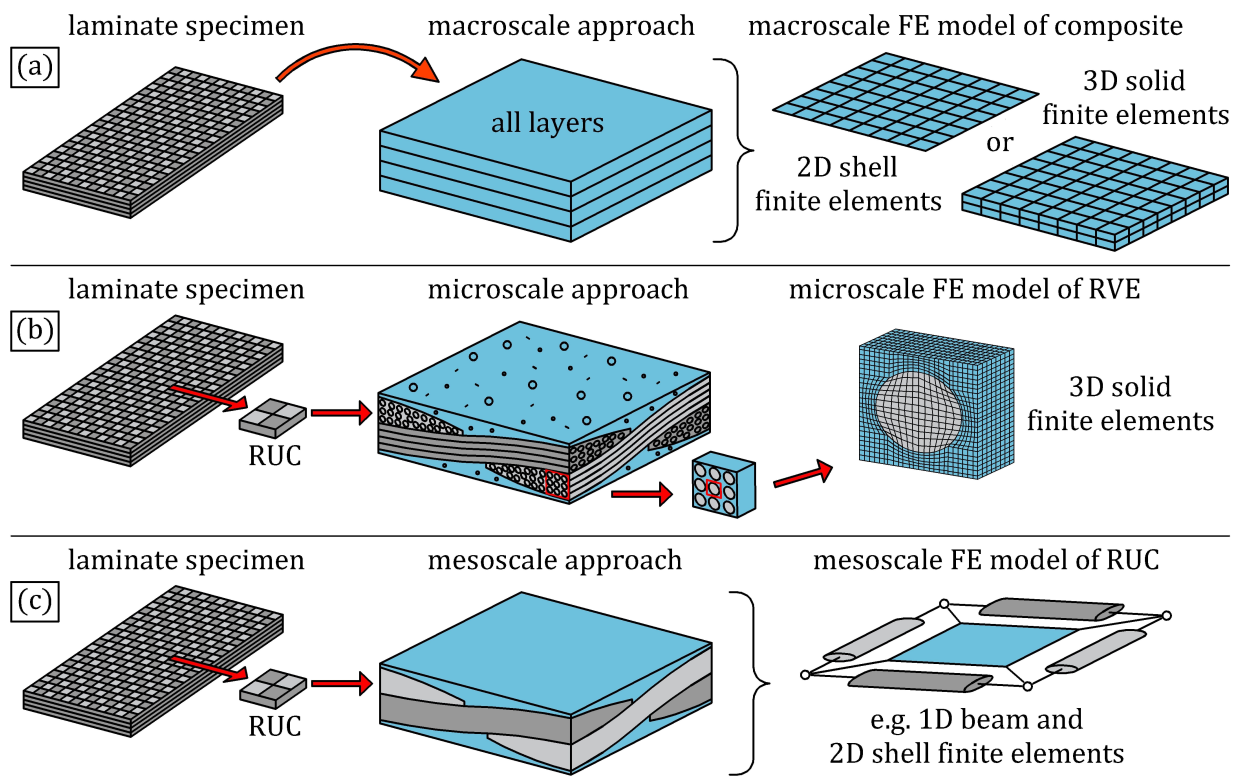

Figure 1: Levels of modeling of layered composite. (a) macroscale level, (b) microscale level, (c) mesoscale level.

the sum of the properties of the components. The labor consumption of creating macroscale FE models of laminates does not differ significantly from the labor consumption of creating FE models of structure elements from traditional construction materials (*e.g.* sheet metal). The analysis of the macroscale model strength is then simply the analysis of strength of particular layers using yield criteria [8]. However, it should be noted that omitting the laminate microstructure and using simplifying assumptions (*i.e.* the theory of reinforcement and the classical lamination theory) may lead to analysis results that deviate from reality [9, 10]. The most common form of failure of laminate constructions is a crack between layers – delamination. Simulation of crack initiation and propagation in the macroscale model is possible through the use of special computational methods, *e.g.* the VCCT technique [11] or the cohesive zone model [12]. With this purpose in mind, special node bonds are defined between layers or additional cohesive elements are introduced at the site of a potential crack, which is equivalent to the need to predict the site and size of the crack.

Auxiliary microscale models are often used to determine the averaged material characteristics for macroscale models [13]. They take into account the smallest individual elements of the laminate microstructure in the form of a representative volume element RVE (Figure 1b), which requires precise discretization of the model with finite elements of very small sizes. For this reason, microscale models have a very large number of degrees of freedom, thus they are characterized by high labor consumption of creation and low numerical efficiency and that is why they are not used in engineering calculations for modeling construction elements, especially those of complex shape and large sizes.

A reasonable compromise between the accuracy of microstructure reflection and the mechanical properties of the laminate and its numerical efficiency are mesoscale models, which simplify the microstructure of the laminate at the level of components – reinforcement and matrix (Figure 1c). In this model, the so-called repetitive unit cell RUC is extracted, which represents the smallest element of laminate microstructure.

In work [9], a mesoscale laminate model is presented, in which carbon fiber rovings and matrix forming RUC were modeled only with the help of one-dimensional finite elements of the beam type, and the model was used to analyze the stiffness of a plane wing. High numerical efficiency and good compliance of simulation results with experimental test results were obtained. In turn, work [14] presents a mesoscale model composed of RUCs made of shell and beam finite elements, which was used in the bending simulation of a composite beam of the reinforcement of a passenger car door. In the simulation of door beam failure as a result of delamination, the procedure of removing finite elements modeling interlayer bonding was used, which eliminated the need to predict the potential crack sites [15]. The analysis of the simulation results showed very good compliance of the model with the results of experimental tests. In the construction of repetitive unit cells at the mesoscale level, two-dimensional finite elements are also used. These types of models were also used in simulation of deformation and strength of non-matrix-impregnated fabric [16, 17]. In works [18, 19], shell elements were used to reflect unit cells, taking into account the separation of reinforcement and matrix phases. In turn, an example of using various types of finite elements in modeling a RUCs at the mesoscale level was presented in work [20]. The reinforcement in the form of mutually interwoven yarns was modeled with four-node shell elements and the surrounding matrix with eight-node solid elements.

Despite the advantages of mesoscale laminate models, modeling of real construction elements is too labor-consuming for engineering practice due to the necessity of multiple replication of the RUC model (Figure 1c). The solution to this problem could be a special generator (preprocessor) that would allow to build mesoscale models for any construction elements in an automated way, especially for elements of complex and irregular shape. This paper presents a preprocessor that generates a mesoscale laminate model using the proposed RUC structure for the surface geometric model of a construction element created in a CAD program.

2 Idea of RUC to mesoscale model of laminate

The mesoscale model presented in Figure 2 takes into account, in a simplified way, the laminate microstructure using a group of finite elements that create RUC. The idea of the RUC structure was explained on the example of a laminate reinforced with a roving fabric with a plain weave. The proposed RUC structure replicates the weave of the reinforcement fabric, so it can be easily modified and adapted to reinforcement fabrics of other weaves [14]. Due to the replacement of the interlacing site of the weft and warp with one node, the unit cell for a single lamina is defined by four nodes lying in one plane (Figure 2, layer I – nodes 1, 2, 3, 4, layer II – nodes 5, 6, 7, 8). The roving sections between the nodes are modeled with one-dimensional finite elements R of the beam type. The cross section A_R of the beam element R corresponds to the cross section of a single roving.

Beam element R has a second cross-section parameter, *i.e.* the moment of inertia of the cross-section I_R, because in fact the matrix-impregnated roving is not flaccid and has bending stiffness. The matrix filling the space between roving sections is modeled by means of a four-node finite shell element S of thickness T_S. The connection of the unit cells in the direction of z axis (Figure 2) is realized by means of vertical beam elements P with material properties of matrix.

The mesoscale model made of RUCs simplifies the laminate microstructure while a single RUC is not an elementary volume of the laminate treated as a continuous medium. The finite elements forming RUC only connect with each other in nodes, thus the adhesion forces between the components are not taken into account directly but their influence is taken into account in the structure

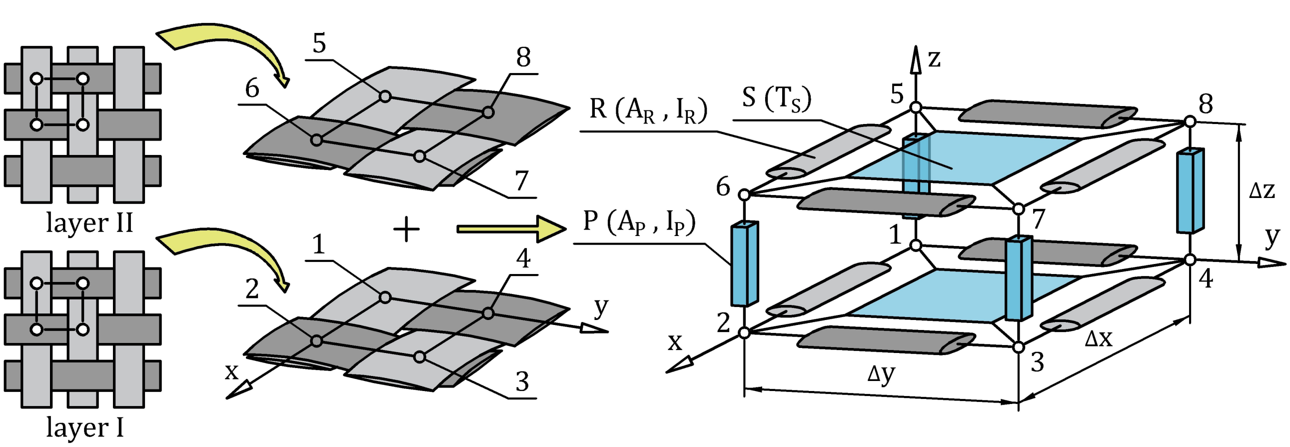

Figure 2: Mesoscale model of two joined RUCs.

and in the equivalent cross-sectional parameters of RUC. The problem of selection of equivalent cross-sectional parameters of finite elements forming RUC was solved by performing experimental calibration of the FE model. It consisted in comparing the deflection of a rectangular laminate specimen subjected to three-point bending, measured and calculated as a result of FE model analysis. A parametric optimization task was formulated in which cross-sectional parameters of beam elements A_P, I_P, I_R and thickness of shell element T_S (Figure 2) were searched due to the best compliance of the calculated and experimentally measured deflection of the laminate specimen while maintaining the volume shares of components in the laminate. The model calibration procedure was discussed in detail on the example of another mesoscale model in work [9]. It is worth adding that this method of determining the equivalent cross-sectional parameters of RUC finite elements allows to take into account the influence of lamination technology on the strength and stiffness of laminate construction elements and, at the same time, it can be carried out using available laboratory equipment. The proposed RUC structure means that the mesoscale models of laminate construction elements made of laminate created by its replication can be used in simulating the most common form of laminate construction failure – delamination. Initiation and propagation of a crack between the layers of reinforcement can be simulated by removing vertical beam elements P (Figure 2), *i.e.* the elements modeling bonds between the layers of fabric in which the force exceeded the limit value (cohesive separation of reinforcement layers) or by reducing their cross-sectional parameters. The description of the method of determining the limit values of forces in the beam elements binding the laminate layers for three basic crack modes, *i.e.* opening, sliding shear and scissoring shear, and the description of the method of simulating delamination were described in detail in [15].

3 Mesoscale model generator for laminated structures

As it was mentioned in the introduction, the FE model of a laminate construction element made of replicated RUCs reflecting in a simplified way the laminate microstructure (*e.g.* as shown in Figure 2) should be generated in an automated way. The proposed authors' original method of preparing such a model takes place in three stages, which are described in detail below on the example of a real laminate construction element. To generate the model, the Ansys Mechanical program environment was used, equipped with the internal command language APDL (Ansys Parametric Design Language) to create the so-called batch files. It is worth noting that the choice of this particular command language does not change the generality of the procedure described below in any way, and its entire course can be easily adapted to other FEM programs.

3.1 Stage 1 – preparing of 2D surface FE model

As laminate constructions are most often thin-walled constructions, *i.e.* whose thickness is negligibly small compared to other dimensions, the basis for building their model is the outer surface of the geometric model of the construction element. Such a surface can be automatically discretized with four-node surface elements of type Shell181 using the tools available in the FEM program. Using the Ansys for this purpose, we receive a file with the definition of a discretized surface, which is a list of the codes of the nodes of finite element mesh written in format $N, \; i, \; x_i, \; y_i, \; z_i$ where N is the node with number i and $x_i, \; y_i, \; z_i$ are its Cartesian coordinates and the list of finite element codes E, i, j, k, l where E is the finite element and i, j, k, l are the numbers of the nodes on which the element is stretched. It is worth noting that the surface is discretized correctly if vectors normal to all shell finite elements are oriented in the same direction and the orientation of the normal vector defines the order of nodes in the sequence i, j, k, l. The number of nodes of the mesh obtained in this way composed of shell elements that will reflect the matrix in the first layer of the laminate (elements S in Figure 2) will be denoted as $n_{(1)}$.

3.2 Stage 2 – procedure of nodes generation for adjoining layers of laminated structure

It consists in building up subsequent layers of the laminate by generating nodes in its adjoining layers according to the diagram shown in Figure 3 for two exemplary finite elements for which node l is common.

Due to the need to maintain the order of sequence of numbers of nodes defining particular finite elements, it is not possible to use the *Offset* command commonly available in FEM programs. For this reason, a special procedure for replicating layer nodes was developed, consisting in determining the vector being the direction of shift of each node, which is the resultant of vectors normal to the finite elements for which the moved node is common. The

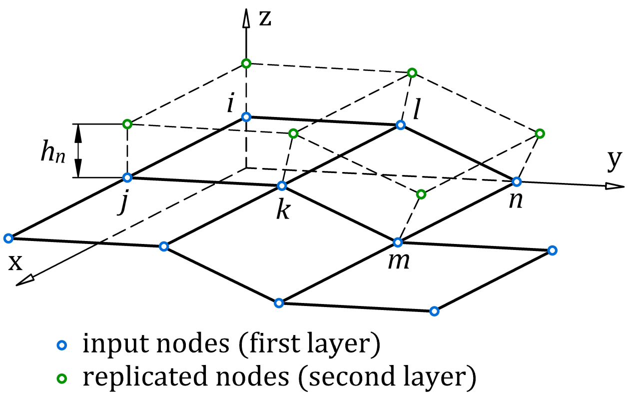

Figure 3: Surface model consisting of five finite elements.

method of defining a vector normal to an element in the selected node l is explained in Figure 4a.

For the first layer and node l selected for replication as well as the selected element with code E, i, j, k, l the program creates two vectors $\boldsymbol{u}_{li}$ and $\boldsymbol{u}_{lj}$. Vector $\boldsymbol{u}_{li}$ has its beginning at the replicated node l, which appears as the fourth in the element code, and its end at node i, which appears first in the element code. In turn, vector $\boldsymbol{u}_{lj}$ is defined by the fourth and second node in the element code, respectively. Vector $\boldsymbol{v}_l^{(ijk)}$, with the beginning at node l, normal to the plane formed by vectors $\boldsymbol{u}_{li}$ and $\boldsymbol{u}_{lj}$ is the vector product of these vectors $\boldsymbol{v}_l^{(ijk)} = \boldsymbol{u}_{li} \times \boldsymbol{u}_{lj}$, and its orientation results from the definition of the vector product. This procedure is repeated for subsequent nodes along the perimeter of the element, maintaining the order of sequence of nodes in the element code that defines orientation of the normal vector. After generating and normalizing all possible normal vectors in nodes to unit length of, a resultant vector is determined for a node with a specific number, which is the geometric sum of all normal vectors for the elements for which the node is common.

An exemplary vector $\boldsymbol{w}_l$ indicating the direction of replication of node number l for two adjacent shell elements is the resultant of two unit vectors normal to these elements, *i.e.* $\boldsymbol{w}_l = \boldsymbol{v}_l^{(ijk)} + \boldsymbol{v}_l^{(kmn)}$ (Figure 4b). Vectors indicating directions of replication of surface model nodes are also converted to unit vectors.

After determining the vectors indicating the directions of replication of all nodes, the generation of nodes belonging to subsequent layers of model q (q = 1,2,...,t) begins – formula (1). At this point what should be given is the number of layers of laminate t as well as the distances between adjacent layers h_n. The generated nodes are assigned numbers according to formula (2). For the purpose of further considerations, it was assumed that the node numbers in the first layer of the model will be supplemented with the index (1), *i.e.* $i \rightarrow i_{(1)}$.

$$\begin{bmatrix} x_{l(q)} \\ y_{l(q)} \\ z_{l(q)} \end{bmatrix} = \begin{bmatrix} x_{l(1)} \\ y_{l(1)} \\ z_{l(1)} \end{bmatrix} + h_n \cdot (q-1) \cdot \boldsymbol{w}_i \qquad (1)$$
$$for\ q = 1, 2, \ldots, t$$

$$l_{(q)} = l_{(1)} + n_{(1)} \cdot (q-1)\ for\ q = 1, 2, \ldots, t \qquad (2)$$

The use of formula (2) to generate node numbers allows to keep the numbering system unchanged, *i.e.* the node numbers in subsequent layers $l_{(q)}$ are the numbers of the nodes from the first layer $l_{(1)}$ increased by a value that depends on the number of the layer q and the number of all nodes in the first layer $n_{(1)}$. Keeping this numbering is necessary in proper generation of the laminate FE model, because vectors normal to the elements of each layer are oriented in

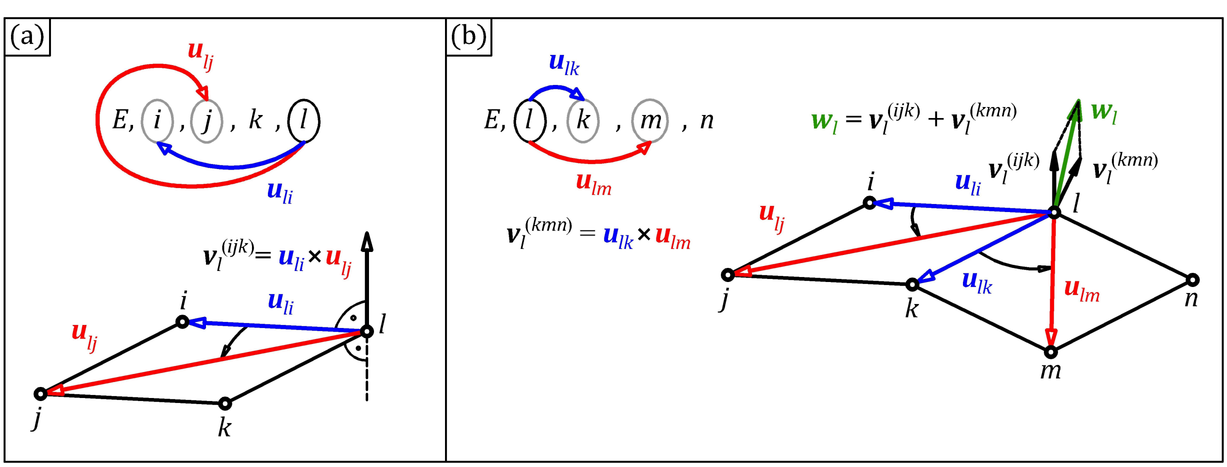

Figure 4: The method of determining vectors. (a) the normal vector for a four-node surface element E, i, j, k, l in node l, (b) the vector indicating replication direction of node l for two adjacent elements.

the same direction. The diagram of the way of generating nodes in subsequent layers of the model is presented in Figure 5 for the replicated node with number $l_{(1)}$.

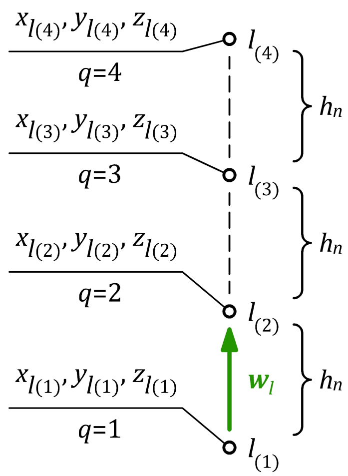

Figure 5: Diagram of a fourfold replication of node number $l_{(1)}$.

The above procedure for generating mesoscale model nodes, formulated as a pseudo-code, is presented below.

Data:
File *nodes.txt* with the definition of nodes in format
N, i, x_i, y_i, z_i
File *elements.txt* with the list of codes of finite elements
E, i, j, k, l

Result:
File *new_nodes.txt* with the definition of nodes for a model created of *n* layers

Pseudo code:
Import nodes from file *nodes.txt* to table *nodes*
Import elements from file *elements.txt* to table *elements*
Create an empty table *curElements*
For each node in table *nodes*

- find all elements in table *elements* containing node and enter them into table *curElements*

If table *curElements* is not empty **Then**
For each element from table *curElements*

- calculate normal vectors for all elements whose code contains the node
- save the obtained normal vectors in file *normal.txt*

End For
End If
End For
Copy normal vectors from file *normal.txt* to table *vectors*
Create empty table *vectorsSum*
For each element in table *vectors*
If in table *vectorsSum* there is no normal vector of element
Then

- enter the vector in table *vectorSum*

Else

- add vectors of tables *vectors* and *vectorsSum* in the node
- change resultant vector into unit vector

End If
End For
For each element of table *vectorSum*

- save unit vector in file *unit_vectors.txt*

End For
Import nodes from file *nodes.txt* to table *nodes*
Import unit vectors from file *unit_vectors.txt* to table *vectors*
Create empty table *curNodes*
For each element from table *vectors*
For *curNodes* = 1 is *n + 1* (*n* – number of layers)

- calculate the coordinates of nodes in subsequent layers of laminate and save them in file *new_nodes.txt*

End For
End For

3.3 Stage 3 – superstructure of laminate microstructure

After completing this stage, a finite element mesh is obtained for all *n* layers of the model and then the finite elements forming subsequent RUCs are generated, which is described in detail in Stage 3. After creating the mesoscale model nodes of all laminate layers, the procedure of generating RUCs begins. Generating shell elements S modeling the matrix in subsequent layers is carried out by increasing the number value of nodes defining shell elements of the first layer. For example, an element stretched on nodes with numbers $i_{(1)}, j_{(1)}, k_{(1)}, l_{(1)}$ will be replicated in layer q according to formula (3).

$$E,\ i_{(1)} + n_{(1)} \cdot (q-1),\ j_{(1)} + n_{(1)} \cdot (q-1),\ k_{(1)} + n_{(1)} \cdot (q-1),\ l_{(1)} + n_{(1)} \cdot (q-1) \quad for\ q = 1, 2, \ldots, t \tag{3}$$

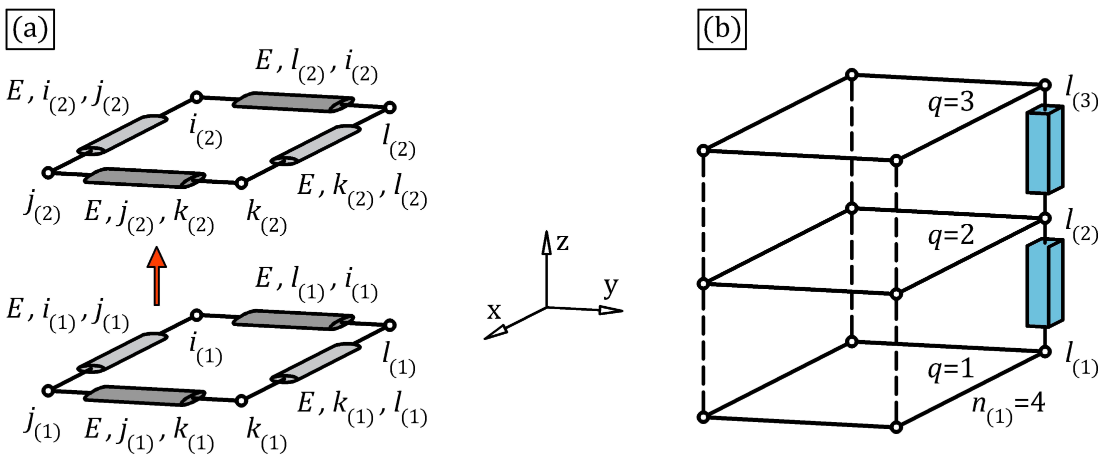

Figure 6: Diagram of generating beam elements. (a) R modeling roving sections, (b) P modelling the matrix between the layers.

The next step is the beginning of the procedure of generating one-dimensional beam elements reflecting the roving sections of the reinforcement fabric and bonding between layers. Beam elements R modeling the roving sections (Figure 6a) are generated along the edges of shell elements and replicated in subsequent layers. As in the APDL language the beam element is defined by code E, i, j, where i and j are node numbers, the beam element stretched between nodes $i_{(1)}, j_{(1)}$ after replication will receive the code according to formula (4).

$$E,\ i_{(1)} + n_{(1)} \cdot (q-1)\,,\ j_{(1)} + n_{(1)} \cdot (q-1)\,, \quad for\ q = 1, 2, \ldots, t \tag{4}$$

The next step after generating all elements R is generating beam elements P (Figure 6b), which model the laminate layer bonds by means of a matrix (Figure 2). In this case, the element is generated between node $l_{(1)}$ belonging to the first layer of the model ($q = 1$) and its equivalent $l_{(2)}$ on the second layer ($q = 2$) and similarly between subsequent layers. Generating codes of subsequent elements is carried out according to formula (5), where the number of generated elements R is $p_r = t - 1$.

$$E,\ l_{(1)} + n_{(1)} \cdot (p_r - 1)\,,\ l_{(1)} + n_{(1)} \cdot p_r, \quad for\ p_r = 1, 2, \ldots, t-1 \tag{5}$$

In this way, you can automatically generate a complete mesoscale model of a laminate construction element, reflecting its microstructure in a simplified way. It can be successfully used in modeling any laminate construction element, including an element of complex and irregular shape whose surface geometric model was designed in a CAD program. It is worth adding that the labor consumption and time needed to prepare the mesoscale FE model of a laminate construction element using the described 3-stage procedure and the labor consumption and time needed to prepare a similar model from finite elements of sandwich type dedicated to modeling layer structures are similar.

4 Verification of the model on the example of a real construction element

The developed preprocessor was used to generate a mesoscale model for a laminate beam reinforcing a passenger car door, and then to simulate its failure as a result of delamination in the three-point bending test. The beam was made of eight layers of carbon fabric 3k CF200 style450 which were impregnated by epoxy resin Biresin® CR120 during wet/hand lay-up process. For the purpose of preparing a computational model, a substitutive elastic modulus E_R was determined for a single carbon fiber roving in a static tensile test ($E_R \cong$ 105 GPa). The material properties of the resin were read from the product data sheet [21].

In accordance with the procedure described in point 3, first a discrete surface model of the beam was created, consisting of four-node finite elements of the surface type (Figure 7). Two text files were generated for this model – *nodes.txt* file, containing the numbers and coordinates of all nodes of finite element mesh of the model, and *elements.txt* file, in which the codes defining all four-node surface elements of the model were stored (Stage 1). After importing the files to the preprocessor, the nodes of finite element mesh of successive layers of the beam model were automatically generated (Stage 2), between which RUCs

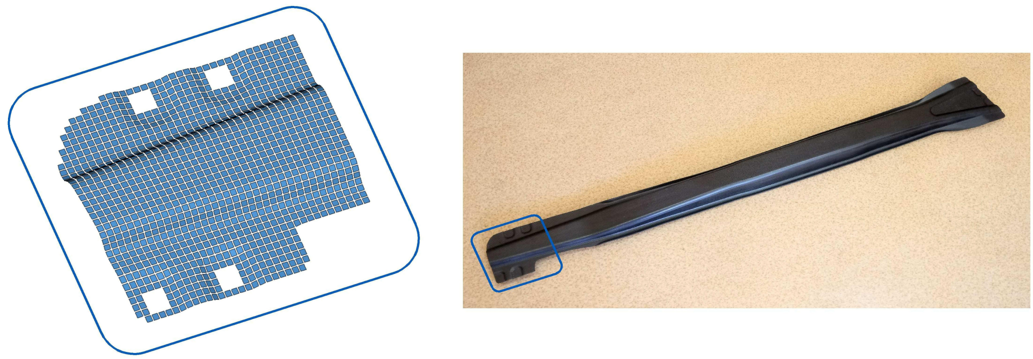

Figure 7: Door beam and a fragment of its surface model with visible four-node finite elements.

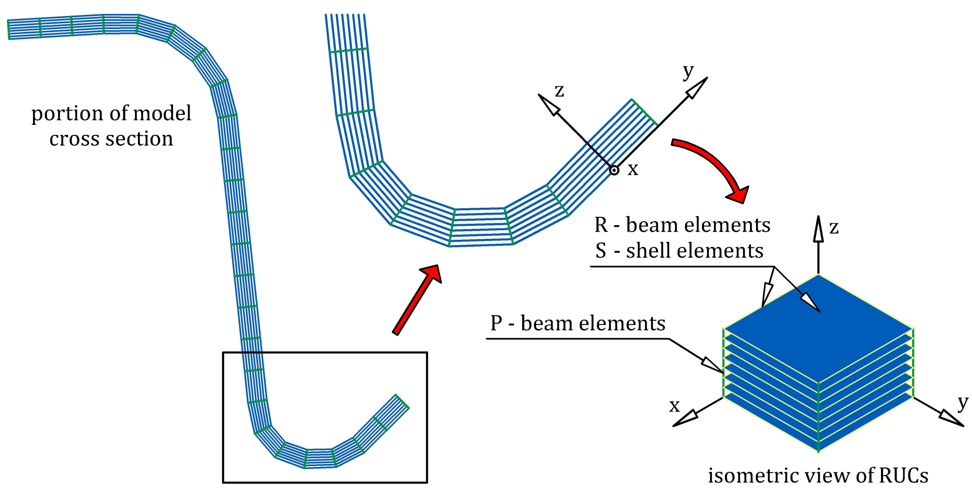

Figure 8: The layered structure of the mesoscale model generated using the developed preprocessor.

were built in according to the definition of a single RUC (Figure 2), reflecting the beam laminate microstructure in a simplified way (Stage 3).

Figure 8 shows an exemplary fragment of the beam model cross-section obtained as a result of using preprocessor.

The three-point bending test of the beam was carried out on a test stand meeting the requirements of standard [22]. During the test, vertical displacement of bending pin f_S was applied, acting in the middle of the beam span, and the vertical force F_S, with which the pin acted on the beam, was measured. A detailed description of the stand and method of conducting the test is provided in [14]. The result of the test was the experimental curve of force as a function of deflection $F_S(f_S)$. Figure 9 shows a photo of a fragment of the beam destroyed during the test with marked sites of cracks caused by delamination.

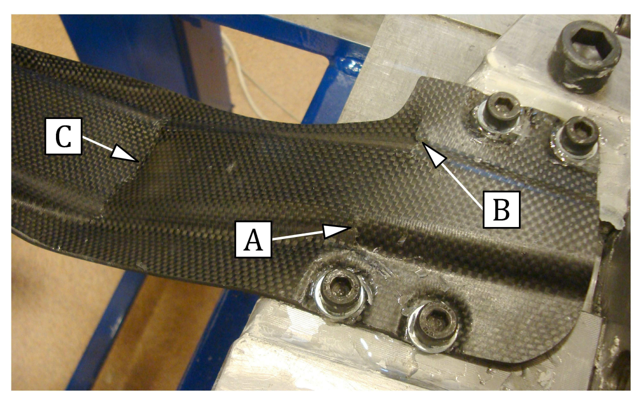

Figure 9: A fragment of the beam destroyed during the test with marked crack sites.

Then a three-point bending test of the beam was simulated using its mesoscale model and obtaining computational curve of force F_S as a function of deflection f_S. During the simulation, in each step, the values of forces in beam elements P (Figure 2) binding the laminate layers were checked and compared with the limit values. If they were exceeded, the cross-sections of beam elements P were reduced to finally remove them completely [15]. In this way a simulation was carried out of initiation and propagation of interlayer cracks leading to delamination. A detailed de-

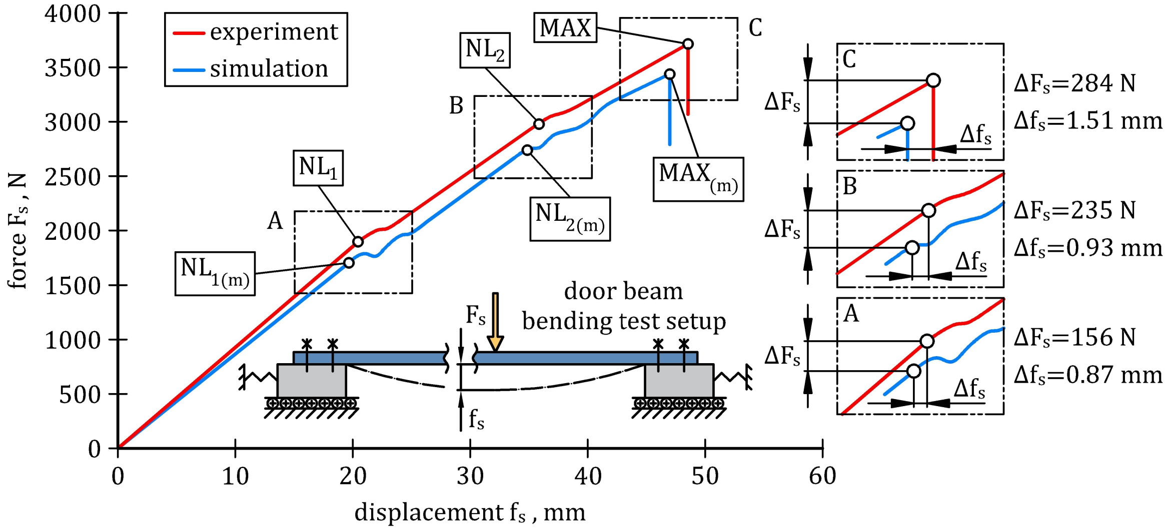

Figure 10: Comparison of the experimental curve of door beam bending with the computational curve for the mesoscale model.

scription of the beam model and the way of performing the simulation is provided in [14]. To verify the correctness of the behavior of the mesoscale model, the experimental and computational curves $F_S(f_S)$ were compared, as shown in Figure 10. Three points were marked on the computational curve: point (A) – $NL_{1(m)}$, in which the first crack was initiated, point (B) – $NL_{2(m)}$, in which the second crack was created and point (C) – $MAX_{(m)}$, beyond which the beam was broken, which resulted in its total failure. Similar points were observed during the experimental test (Figure 9). At point NL_1, the curve changed from linear to nonlinear for the first time and then returned to linear again after a while, which is connected with the formation of the first crack (A, Figure 9) in the beam. At point NL_2, a similar change in characteristic occurred, which is connected with the beginning of the propagation of the second crack (B, Figure 9) and after exceeding the MAX point, the beam was broken at point (C, Figure 9). Figure 10 also shows the differences (absolute errors of deflections Δf_S and forces ΔF_S in specific points between the experimental and computational curves, which were then calculated into relative values, referring them to the experimentally measured values. The largest relative error of force F_S was 8% and of deflection f_S of 4%, which can be considered a very good compliance of the compared results. The bending stiffness of the mesoscale FE model of the beam corresponded to the bending stiffness of the real door beam. Moreover, the failure of the model occurred in the same sites as the failure of the real beam during experimental tests [14].

5 Conclusions

The preprocessor described in the article generates a mesoscale FE model of a fabric reinforced laminate that reflects in a simplified way its microstructure, *i.e.* the number and orientation of reinforcement layers and the type of reinforcement fabric weave. The constants of this model (equivalent cross-sectional parameters of finite elements forming RUC) can be determined experimentally by conducting easy-to-perform experimental tests of specimens of a particular laminate, which also takes into account the influence of technology of manufacturing (wet lamination, laminating from prepregs in an autoclave, etc.) laminate construction elements on their mechanical properties. Conducting complementary tests destroying specimens of a particular laminate for three basic crack modes, *i.e.* opening, sliding shear and scissoring shear, allows to determine the limit values of forces in beam elements binding the layers in the laminate model, thanks to which the developed model can be additionally used to simulate delamination of laminate construction elements. It is worth stressing that this way of simulating delamination does not require defining the potential crack sites between the layers or introducing to the model additional special finite elements serving the purpose of simulating crack propagation.

The mesoscale FE model of the laminate was experimentally verified and the usefulness of the preprocessor in its generation was confirmed on the example of a laminate door beam of a passenger car subjected to a three-point bending test. The time and labour consumption of developing a FE model of a laminate construction element

– a door beam using the preprocessor were similar to the time and labour consumption of developing a FE model of a beam made of finite elements of sandwich type, which are dedicated to modeling layered structures in commercial FEM programs. The described preprocessor uses the internal command language of Ansys Mechanical and generates the so-called batch files, which define the FE model, but other programs used for FEM analysis have similar facilities for automating model creation, and a similar preprocessor can be created for them. The preprocessor can be easily adapted by modifying the RUC structure to generating FE models of laminates in which the reinforcement is the fabric with non-plain weave and even adapted to modeling laminates which are reinforced with layers of knitted fabric.

References

[1] Mehrabadi FA, Khoshravan M. Mode III interlaminar fracture and damage characterization in woven fabric-reinforced glass/epoxy composite laminates. J Compos Mater. 2012;47(13):1583-92.

[2] Willett HG. Characterisation of composites for wind turbine blades. Reinf Plast. 2012;56(5):34-6.

[3] Kelly J, Mohammadi M. Uniaxial tensile behavior of sheet molded composite car hoods with different fibre contents under quasi-static strain rates. Mech Res Commun. 2018;87:42-52.

[4] Karpat Y, Değer B, Bahtiyar O. Drilling thick fabric woven CFRP laminates with double point angle drills. J Mater Process Technol. 2012;212(10):2117-27.

[5] Bednarcyk BA, Stier B, Simon JW, Reese S, Pineda EJ. Meso- and micro-scale modeling of damage in plain weave composites. Compos Struct. 2015;121:258-70.

[6] Jones RM. Mechanics of composite materials. 2nd ed. Boca Raton: CRC Press; 1999.

[7] Barbero EJ. Introduction to composite materials design. 2nd ed. Boca Raton: CRC Press; 2011.

[8] Gudmundson P. Fracture and damage of composite laminates. In: Hult J, Rammerstorfer FG, editors. Engineering mechanics of fibre reinforced polymers and composite structures. Wien-New York: Springer; 1994. p. 111-39.

[9] Stadnicki J, Tokarz Z. Mesoscale finite element model for calculating deformations of laminate composite constructions. Adv Mech Eng. 2016;8(2):1-9.

[10] Vinšová L, Urban T. Testing of mechanical properties of thick-walled carbon fiber composite for FEM simulations. Mater Today Proc. 2017;4(5):5989-94.

[11] Rybicki EF, Kanninen MF. A finite element calculation of stress intensity factors by a modified crack closure integral. Eng Fract Mech. 1977;9(4):931-38.

[12] Lindgaard E, Bak BLV, Glud JA, Sjølund J, Christensen ET. A user programmed cohesive zone finite element for ANSYS Mechanical. Eng Fract Mech. 2017;180:229-39.

[13] Naghdinasab M, Farrokhabadi A, Madadi H. A numerical method to evaluate the material properties degradation in composite RVEs due to fiber-matrix debonding and induced matrix cracking. Finite Elem Anal Des. 2018;146:84-95.

[14] Marszałek J, Stadnicki J. Durability assessment of composite structural element reinforced with fabric due to delamination. Autex Res J. 2020;doi: 10.2478/aut-2020-0022. Epub 2020 Jul 6.

[15] Marszałek J, Stadnicki J. Simulation of delamination in a composite using a shell-beam mesoscale finite element model. Fibres Text East Eur. 2018;4(130):97-103.

[16] Gatouillat S, Bareggi A, Vidal-Sallé E, Boisse P. Meso modelling for composite preform shaping – simulation of the loss of cohesion of the woven fibre network. Compos Part A Appl Sci Manuf. 2013;54:135-44.

[17] Giannaros E, Kotzakolios A, Sotiriadis G, Tsantzalis S, Kostopoulos V. On fabric materials response subjected to ballistic impact using meso-scale modeling. Numerical simulation and experimental validation. Compos Struct. 2018;204:745-54.

[18] García-Carpintero A, Van Den Beuken BNG, Haldar S, Herráez M, González C, Lopes CS. Fracture behavior of triaxial braided composites and its simulation using a multi-material shell modelling approach. Eng Fract Mech. 2018;188:268-86.

[19] Zhang C, Binienda WK. A meso-scale finite element model for simulating free-edge effect in carbon/epoxy textile composite. Mech Mater. 2014;76:1-19.

[20] Vašíček M. Computational prediction of the mechanical properties of a 2D triaxially braided composite. Journal of Middle European Construction and Design of Cars. 2015;12(2):10-16.

[21] Product data sheet. Biresin® CR120. Composite resin system. 2014.

[22] GMW16418. Door beam bending strength test. GMW standard. 2016.