Polish Society of Composite Materials
Polskie Towarzystwo Materiałów Kompozytowych

21: 1-2 (2021) 29-39

COMPOSITES THEORY AND PRACTICE
ISSN: 2084-6096
ISSN (online): 2299-128X

Katarzyna Gojny*, Adam Dacko

Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Institute of Aeronautics and Applied Mechanics Division of Strength of Materials and Structures, ul. Nowowiejska 24, 00-665 Warsaw, Poland
**Corresponding author. E-mail: kgojny@meil.pw.edu.pl*

Received (Otrzymano) 6.12.2020

INVESTIGATION OF FINITE ELEMENT (FE) MODELLING OF COMPOSITE MATERIALS: SHELL, SOLID AND SOLID LAYERED COMPOSITE MODELLING – COMPARISON OF IMPACT ON SIMULATION RESULTS

Composites materials have attracted a great deal of interest for use in various fields e.g. the transport industry, construction, sport equipment and even home applications. This is due to the excellent properties of composite materials such as their high strength/stiffness to mass ratio. Nowadays, finite element (FE) modelling allows investigation of the behaviour of composite materials subjected to loading before the production stage. This paper compares the available ways of FE modelling of composite materials: shell, solid shell and solid layered composite modelling in MSC Patran/Nastran software. The aim of this research work was a comparative analysis and determination of the influence of the applied modelling type on the simulation results. Numerous finite element analyses (FEAs) (tensile and bending) of different narrow plate (resembling a slender, beam-like structural member) structure models were performed, i.e. laminated beam and sandwich beam models, with different layups of layers. The obtained numerical results allow the authors to conclude that shell composite finite element modelling can be considered a practically optimal choice due to reduction of the modelling effort and time, as well as due to obtaining consistent simulation results, especially when having only the basic manufacturer's set of material constants.

Keywords: **composite materials, finite element (FE) modelling, finite element analysis (FEA), shell composite modelling, solid shell composite modelling, solid layered composite modelling**

BADANIE RODZAJÓW MODELOWANIA MES MATERIAŁÓW KOMPOZYTOWYCH: MODELOWANIE POWŁOKOWE, BRYŁOWE I BRYŁOWO-WARSTWOWE – PORÓWNANIE WPŁYWU NA WYNIKI SYMULACJI

Materiały kompozytowe cieszą się dużym zainteresowaniem w różnych dziedzinach, np. przemysł transportowy, budownictwo, sprzęt sportowy i nawet zastosowania domowe. Wynika to z doskonałych właściwości materiałów kompozytowych, takich jak wysoki stosunek wytrzymałości/sztywności do masy. Obecnie modelowanie metodą elementów skończonych (MES) pozwala na badanie zachowania się materiałów kompozytowych poddanych obciążeniu przed etapem produkcji. W artykule porównano dostępne sposoby modelowania MES materiałów kompozytowych: modelowanie powłokowe, bryłowe i bryłowo-warstwowe w oprogramowaniu MSC Patran/Nastran. Celem pracy badawczej była analiza porównawcza i określenie wpływu zastosowanego modelowania na wyniki symulacji. Przeprowadzono liczne analizy MES (zginanie i rozciąganie) dla różnych modeli wąskiej płyty (przypominającej smukły, belkowy element konstrukcyjny), tj. dla modelu materiału laminatu warstwowego oraz struktury przekładkowej, z różnymi układami warstw. Uzyskane wyniki numeryczne pozwalają stwierdzić, że modelowanie powłokowe materiałów kompozytowych jest praktycznie optymalnym wyborem ze względu na zmniejszenie nakładu pracy i czasu oraz uzyskanie spójnych wyników symulacji, zwłaszcza gdy dysponuje się tylko podstawowym zestawem stałych materiałowych producenta.

Słowa kluczowe: **materiały kompozytowe, modelowanie MES, analiza metodą elementów skończonych (MES), modelowanie powłokowe kompozytów, modelowanie bryłowe kompozytów, modelowanie bryłowo-warstwowe kompozytów**

NUMERICAL MODELLING OF COMPOSITE AERONAUTICAL STRUCTURES

Composites are becoming ever more common materials in numerous applications in various fields like the transport industry, construction, sport equipment and even home applications. This is due to the excellent properties of composite materials such as the high strength/stiffness to mass ratio. The application areas of glass fiber reinforced plastics (GFRP) are steadily increasing. In 2017 in Europe, more than two-thirds of

GFRP applications came from transport and construction/infrastructure [1]. Composite materials are widely used in air transport. They may be utilized for the primary structure components (e.g. wings, empennage, fuselage), control components (e.g. ailerons, rudders, spoilers), interior components (floors, doors), in addition to exterior components (fairings, landing gear trap doors) [2].

Numerical modelling is very often used in the aviation and aerospace industry. It allows investigation of the behaviour of composite parts in the design phase and saves expensive physical testing of real structures. In the creation of composite numerical models, different types of finite element (FE) modelling of composite materials may be used (Fig. 1) [3, 4]. Depending on the composite material structure and part thickness or proportions, various finite elements may be used. It can be noticed that shell elements are a dominating choice for the numerical modelling of aeronautical structures and components, especially for the wing structure, i.e. skin, spar and rib models made of layered composite materials [5-7]. Furthermore, fuselage sections are often modelled with shell elements as well [8]. However, in the case of composite sandwich structures, the cores may be modelled as shell or solid elements [9, 10]. Shell composite FE modelling is based on shell topology whereas solid shell and solid layered FE composite modelling types are based on solid topology. These types of FE modelling of composite materials are available in MSC Patran/Nastran software. The composite layup is assigned to the shell or the solid finite elements. The underlying theoretical difference between solid shell and solid layered composite modelling is the use of the assumed strain formulation [11].

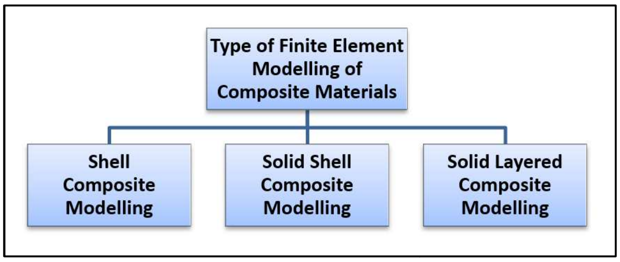


Fig. 1. Types of finite element (FE) modelling of composite materials in MSC Patran/Nastran software

Rys. 1. Typy modelowania metodą elementów skończonych (MES) materiałów kompozytowych w oprogramowaniu MSC Patran/Nastran

MSC Patran/Nastran software provides different FE modelling approaches. Research articles on comparing different modelling methods of composite materials are rare. Due to this fact, this paper compares the available ways of FE modelling of composite materials: shell, solid shell and solid layered composite modelling in MSC Patran/Nastran software (2018 version). The aim of this research work was a comparative analysis and determination of the influence of the applied modelling type on the simulation results.

SHELL COMPOSITE MODELLING

Thin laminated composites traditionally are modelled as shell elements in MSC Patran/Nastran software (Fig. 2). The typical topology (or in FE – connectivity) of a 2-dimensional (2D) shell finite element is the element called “QUAD4” (4-node quadrilateral). A convenient feature of shell composite modelling is the relative ease of changing the thickness of the layup – at every stage of the modelling process. Shell composite modelling consists in creating a geometric surface and assigning the property to the model – the user defines the structure (layup definition of the shell) and thus defines the total thickness of the shell structure [3].

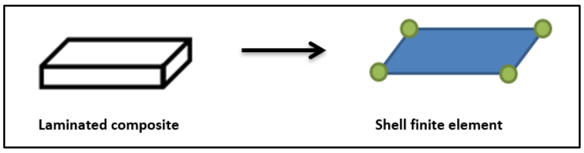


Fig. 2. Shell composite modelling

Rys. 2. Powłokowe modelowanie kompozytów

SOLID LAYERED COMPOSITE MODELLING

For thick laminates or when the state of stress in the structure is three dimensional (Fig. 3), the appropriate choice is solid layered composite modelling [4]. It is available in nonlinear implicit analysis called “SOL 400” in MSC Patran/Nastran software [3, 4]. The ply stresses and interlaminar stresses can also be obtained. What can be a great asset is that this modelling approach supports progressive ply failure analysis.

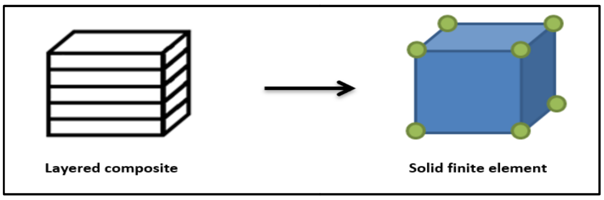


Fig. 3. Solid layered composite modelling

Rys. 3. Bryłowo-warstwowe modelowanie kompozytów

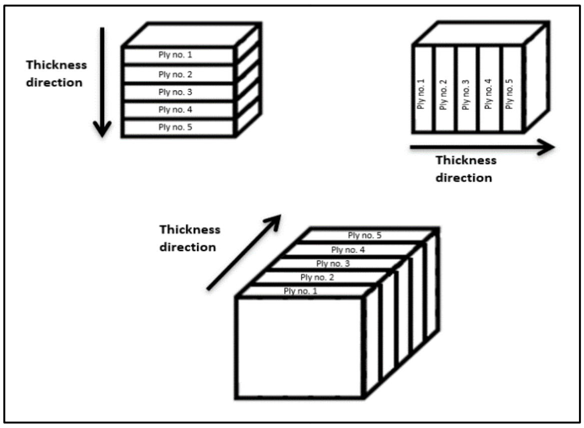


Fig. 4. Definition of orientation of plies in solid layered composite modelling

Rys. 4. Definicja układu warstw w bryłowo-warstwowym modelowaniu kompozytów

The topology of this FE modelling type is 3-dimensional (3D). A typical example of a solid finite element is a 3D hexagonal finite element called "HEX8" (linear 8-node topology). Solid layered composite modelling typically involves the creation of a solid object (during creation of the model geometry) and the composite part thickness is then defined by the solid dimensions. In the case of solid layered composite modelling, it is important to define the orientation of the plies that are components of the solid elements (Fig. 4). It is done during solid laminate property assignment.

Generally, in FEM solid elements tend to be overly stiff in bending. In classical solutions, in order to overcome this drawback and correctly simulate the model structural member behaviour in bending, several layers of solid elements through the thickness are required. Solid composite elements inherit the same behaviour. Therefore, the solid composite elements with this default formulation are called solid layered composite (or regular solid composite) elements. The solution to this problem of "over-stiffening" is the to use another modelling approach called solid shell composite modelling.

SOLID SHELL COMPOSITE MODELLING

Basically, too high bending stiffness causes the numerical issue called in FE terminology "shear locking". First order elements like solid elements may exhibit this behaviour. Then, the numerical results for bending are not satisfying / true. A real structure subjected to pure bending exhibits a curved shape (Fig. 5, left). The behaviour of standard finite elements may be different from the real one (Fig. 5, right) – the horizontal edges of the finite element do not bend to curves.

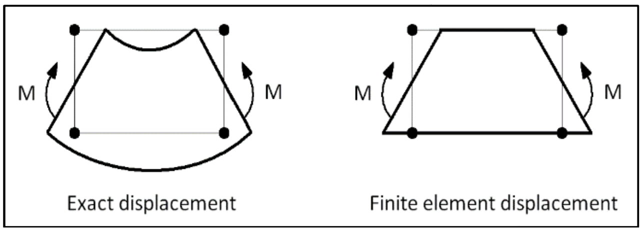


Fig. 5. Exact and finite element displacement in bending – diagram (authors' own elaboration)

Rys. 5. Rzeczywiste i numeryczne odkształcenie podczas zginania – szkic (własne opracowanie)

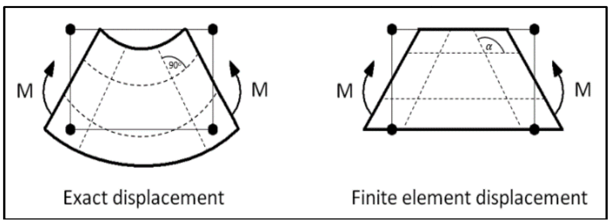


Fig. 6. Isoparametric lines for bending – diagram (authors' own elaboration)

Rys. 6. Izoparametryczne linie dla zginania – szkic (własne opracowanie)

In Figure 6 the upper edge experiences compressive stresses and the bottom edge experiences tensile stresses. Due to the fact that the isoparametric lines (dotted lines) of the finite element remain straight (Fig. 6, right), angle α between the deformed isoparametric lines is not equal to 90°, as it should be (Fig. 6, left). Angle α in FE is changed, which introduces some artificial shear stress. This shear stress makes the finite element overly stiff – the energy of deformation is cumulated in this shear phenomenon, instead of working for bending deflection. Therefore, significant effort (strain energy) goes into shearing the element rather than bending it. This results in generally too low displacements and stresses. The solution to this problem is to use solid shell composite modelling.

Solid shell composite modelling is used when the bending is dominant and the model does not have several layers of solid elements through the thickness [4]. In this formulation, the assumed strain functions are added to the finite element formulation to make the element behave more like a shell element when loaded in bending. Solid composite elements with this assumed strain formulation are called solid shell composite elements [4]. The numerical results for both solid shell and solid layered composite modelling types are available in the nonlinear implicit analysis called "SOL 400" [3, 4].

TYPE OF FINITE ELEMENT ANALYSIS "SOL 400"

In the products of MSC Software Corporation the format "SOL n" means the solution sequence and is used to select the type of analysis where "n" is a positive integer identifying the solution type or the character name of the solution procedure [12]. For example, "SOL 101" means the solution type for linear static analysis. The general static and dynamic implicit formulation for nonlinear analyses is labelled as "SOL 400".

In a nonlinear problem, the stiffness of the structure changes during the application of load, and therefore such a nonlinear problem requires an incremental solution scheme. The results from the previous step are used as a starting point for the next step, i.e. the displacements are calculated for each step and the stiffness is updated during the analysis.

Advanced composite materials require nonlinear simulations to be performed in order to better predict the real behaviour of the structure and increase the quality of the stress results. Therefore, "SOL 400" was used because it allows advanced composite modelling.

FE MODELLING METHODOLOGY

The modelling methodology was based on the selection of finite elements depending on the availability of the material data. For shell elements, the number of material data (6 material constants) required for finite element analysis (FEA) is smaller than for solid elements (9 material constants). The assumption which was used for all FEAs was that for the composite models the adhesive (glue) layer was not modelled because its effect has no significant influence on the final numerical results [9].

In the case of shell composite modelling, 2D orthotropic material (in MSC software nomenclature called "MAT 8") was used for the shell elements. "MAT 8" allows the following 6 material constants to be inserted: E_{11}, E_{22}, ν_{12}, G_{12}, G_{23}, G_{31}. In the case of solid layered and solid shell composite modelling types, 3D orthotropic material (in MSC software nomenclature called "MATORT") was used for solid elements. "MATORT" allows the following 9 material constants to be entered: E_{11}, E_{22}, E_{33}, ν_{12}, ν_{23}, ν_{31}, G_{12}, G_{23}, G_{31}. The equations used for the stress and strain calculations based on the material constants for shell elements (using "MAT 8") are the following [13]:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \\ \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_{yx}}{E_y} & 0 & 0 & 0 \\ \frac{-\nu_{xy}}{E_x} & \frac{1}{E_y} & 0 & 0 & 0 \\ 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xz}} & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \\ \tau_{xz} \\ \tau_{yz} \end{Bmatrix} \qquad (1)$$

where: x, y, z – are the material coordinate axes, E – Young's modulus, ν – Poisson's ratio, G – shear modulus, σ – stress, ε – strain, τ – shear stress, γ – shear strain.

The equations used for the stress and strain calculations based on the material constants for solid elements (using "MATORT") are the following [13]:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_{yx}}{E_y} & \frac{-\nu_{zx}}{E_z} & 0 \\ \frac{-\nu_{xy}}{E_x} & \frac{1}{E_y} & \frac{-\nu_{zy}}{E_z} & 0 \\ \frac{-\nu_{xz}}{E_x} & \frac{-\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 \\ 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} \qquad (2)$$

The composite laminate is a build-up of various materials into plies that differ in thickness and direction of the fibers. MSC software allows a composite laminate to be created using property types corresponding to thin shell formulation or solid shell formulation ("PCOMP" or "PCOMPLS", respectively) [13]. The "PCOMP" property type was used to create the composite laminate using shell elements, whereas the "PCOMPLS" property type was applied to the composite laminate made of solid elements. Both property types allow the definition of typical details of a layered composite: the number of layers, their thicknesses, the orientation angle of the fibers in a given layer and the material from which a given layer is made. Figure 7 presents the modelling methodology.

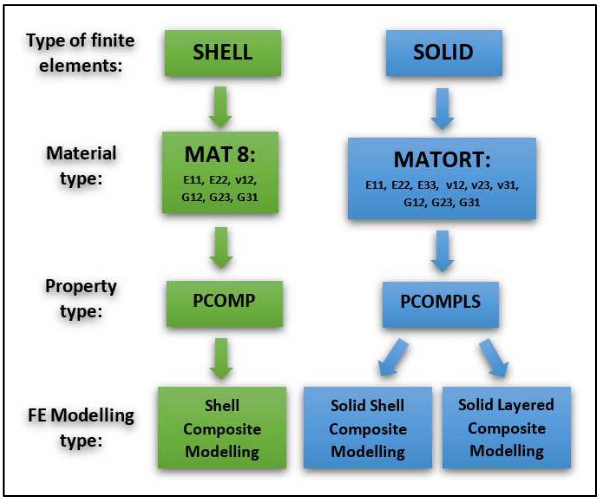


Fig. 7. Modelling methodology

Rys. 7. Metodologia modelowania

FINITE ELEMENT ANALYSES (FEAS)

The objective of the presented investigation was to compare and explore the influence of the applied modelling on the simulation results for bending and tensile finite element analyses (FEAs) of different types of finite element (FE) modelling of composite materials (Fig. 8):

- shell composite modelling,
- solid shell composite modelling,
- solid layered composite modelling.

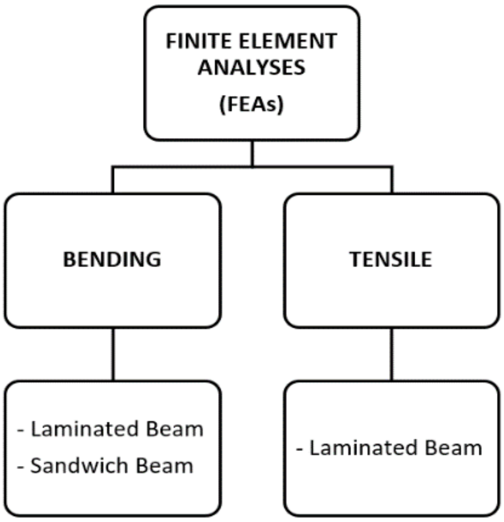


Fig. 8. Types of conducted finite element analyses (FEAs)

Rys. 8. Rodzaje przeprowadzonych analiz metodą elementów skończonych (MES)

Laminated beam and sandwich beam models were used for the numerical analyses. In the case of bending FEAs, both the laminated and sandwich beam models were investigated, whereas only laminated beam models were studied in the tensile FEAs. Tables 1 and 2 present the material constants used for two (2D) and three (3D) directions (dimensions), which, since these analyses were for demonstration purposes only, were taken from the literature. The material used for the laminated beams was:

- unidirectional prepreg [14].

The materials used for the sandwich beams were the following:

- unidirectional prepreg [14],
- Nomex™ honeycomb core [15].

Depending on the modelling requirements, the data were used for 2D orthotropic or 3D orthotropic type of material.

TABLE 1. **Material data for unidirectional prepreg**
TABELA 1. **Dane materiałowe dla jednokierunkowego prepreg**

UNIDIRECTIONAL PREPREG material data		
Quantity:	**2D Orthotropic:**	**3D Orthotropic:**
Material name:	**Unidirectional prepreg**	**Unidirectional prepreg**
E_{11} [MPa]	139 000	139 000
E_{22} [MPa]	9500	9500
E_{33} [MPa]	–	9500
ν_{12} [–]	0.32	0.32
ν_{23} [–]	–	0.5
ν_{31} [–]	–	0.02
G_{12} [MPa]	5400	5400
G_{23} [MPa]	3700	3700
G_{31} [MPa]	5400	5400

TABLE 2. **Material data for honeycomb core**
TABELA 2. **Dane materiałowe dla rdzenia struktury przekładkowej**

HONEYCOMB CORE material data		
Quantity:	**2D Orthotropic:**	**3D Orthotropic:**
Material name:	**Nomex™ core**	**Nomex™ core**
E_{11} [MPa]	4597	4597
E_{22} [MPa]	3536	3536
E_{33} [MPa]	–	3250
ν_{12} [–]	0.212	0.212
ν_{23} [–]	–	0.212
ν_{31} [–]	–	0.15*
G_{12} [MPa]	1678	1678
G_{23} [MPa]	1400	1400
G_{31} [MPa]	1619	1619

* Value of ν_{31} was calculated using the following formula:

$$\nu_{31} = \nu_{13} \cdot \frac{E_{33}}{E_{11}} = 0.15$$

FINITE ELEMENT (FE) MODELS

In the following sections, the beam model term is used. It refers to the global proportions of the structure, resembling a slender, beam-like structural member. In fact, the structure is treated more like a plate or shell structure, based on 2D (or 3D) geometry, not on 1D (line) modelling. The following beam models were created based on the model structure (Fig. 9):

- laminated beam – consisting of thin layers of composite laminate,
- sandwich beam – made of composite sandwich structure, i.e. core and face sheets (treated as layers).

In the case of models named laminated beam and sandwich beam, the modelling of the whole structure was investigated. For the laminated beam, three different layups were analysed depending on the number of layers:

- model no. 1 with one layer
- model no. 2 with two layers,
- model no. 3 with five layers.

Sandwich beam, model no. 4 consisted of five layers: the core and each skin (face sheet) is made of two layers. For each model labelled from 1 to 4, all three types of composite finite element (FE) modelling approaches were applied (Table 3). In Table 3, the tick mark means that this type of composite FE modelling was used for the given model.

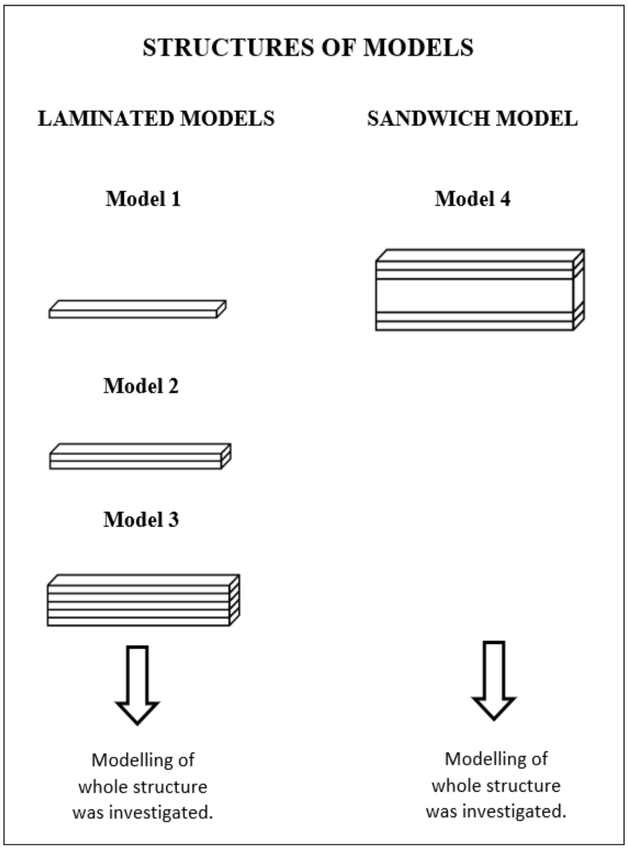
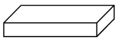
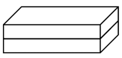
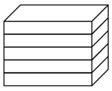
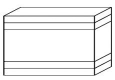


Fig. 9. Structures of models – general overview
Rys. 9. Struktury modeli – przegląd ogólny

TABLE 3. **Types of composite FE modelling used for FEAs**
TABELA 3. **Typy modelowania elementami skończonymi kompozytów zastosowane dla analiz MES**

MODEL NO.	TYPE OF MODELLING used for whole model for FINITE ELEMENT ANALYSES (FEAs)		
	SHELL Composite Modelling	**SOLID SHELL Composite Modelling**	**SOLID LAYERED Composite Modelling**
LAMINATED BEAM			
MODEL 1 	✓	✓	✓
MODEL 2 	✓	✓	✓
MODEL 3 	✓	✓	✓
SANDWICH BEAM			
MODEL 4 	✓	✓	✓

The beam dimensions were the same for all the models: length 100 mm, width 10 mm, and the thickness varied depending on the number of composite layers. The meshing parameters are presented in Table 4. The chosen mesh was relatively coarse to eliminate the effect of "convergence of solution" for finer meshes. For denser meshes, the distinction of the influence of a given modelling method on the simulation results is less visible. The beam was loaded at the free end: the vertical load of 0.1 N was applied for bending and the horizontal load of 10 000 N was applied for tension.

TABLE 4. **Mesh parameters of beam models**
TABELA 4. **Parametry siatki dla modeli belkowych**

MESH PARAMETERS of LAMINATED and SANDWICH BEAM MODELS			
Quantity:	**SHELL Composite Modelling**	**SOLID SHELL Composite Modelling**	**SOLID LAYERED Composite Modelling**
Mesh type:	QUAD4	HEX8	HEX8
Number of finite elements FEs):	20 FEs	20 FEs	20 FEs
	Note: 10 FEs along beam length 2 FEs along beam width 1 FE along beam height		

The support end was fully built-in to the wall and therefore all 6 DOFs (degrees of freedom) were constrained (Figs. 10 and 11). All the FEAs were set as nonlinear static, although the range of behaviour was totally linear as the shell and solid composite modelling types are implemented in the nonlinear static solution ("SOL 400"). The finite element (FE) software used in the research work was MSC Patran for pre/post-processing processes and MSC Nastran was used as the finite element method (FEM) solver [3]. In this method, the approximations of the solution over each finite element in terms of nodal values are sought [16].

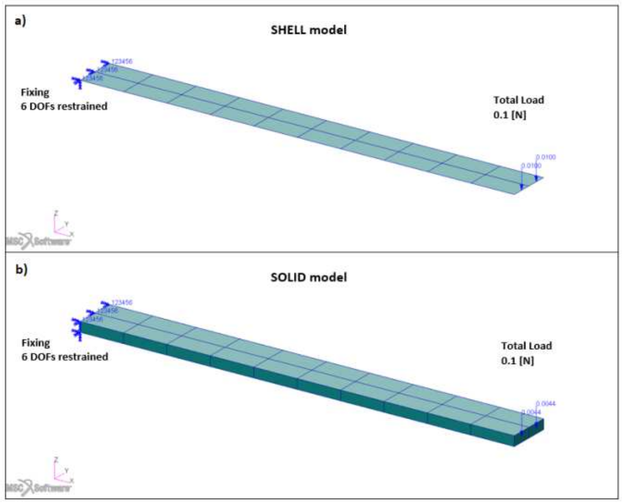


Fig. 10. Beam shell (a) and solid (b) FE model in bending

Rys. 10. Powłokowe (a) i bryłowe (b) modele elementów skończonych dla zginania

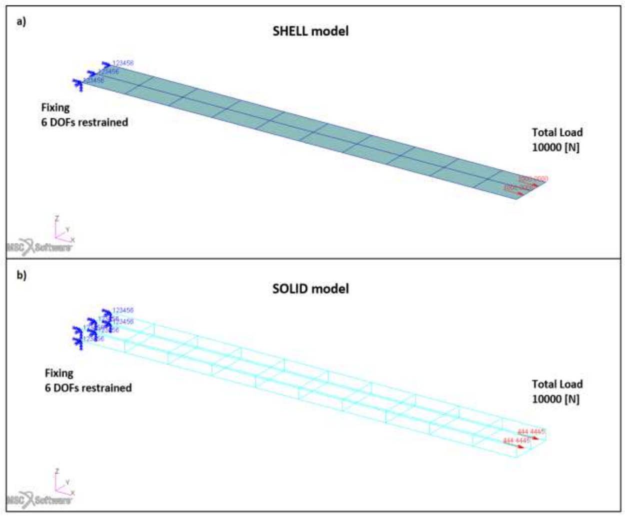


Fig. 11. Beam shell (a) and solid (wireframe view) (b) FE model in tension

Rys. 11. Powłokowe (a) i bryłowe (widok krawędziowy) (b) modele elementów skończonych dla rozciągania

Three models of laminated beam were used, consisting of different numbers of layers. Figure 12 presents the number of layers and the total thickness for each laminated beam model. Each layer of model no. 1-3 is made of a unidirectional prepreg. The fiber orientation angle is 0°, which means that the fibers are aligned with the beam length. Figure 13 presents the sandwich beam model (model no. 4) consisting of 5 layers: 4 thin layers (top and bottom skin, 2 layers each) made from a unidirectional prepreg separated by the honeycomb core. For the sandwich beam model, the fiber orientation angle is 0°, which means that the fibers are aligned with the beam length.

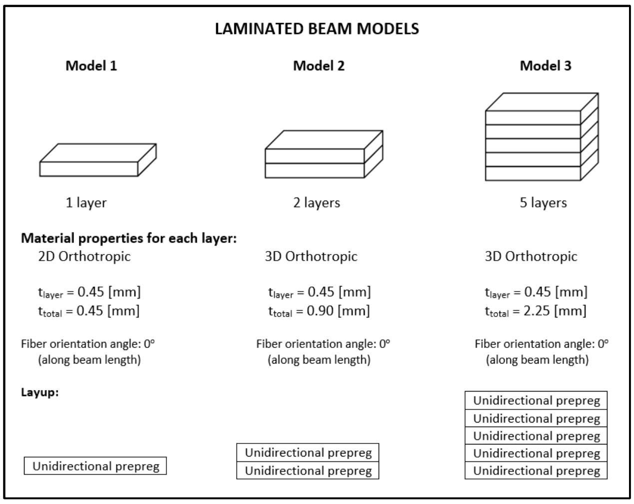


Fig. 12. Parameters of laminated beam models

Rys. 12. Parametry modeli belkowych laminatu warstwowego

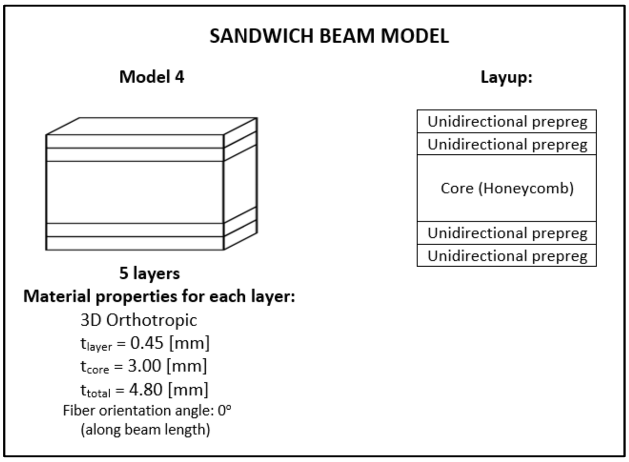


Fig. 13. Parameters of sandwich beam model

Rys. 13. Parametry modelu belkowego o strukturze przekładkowej

RESULTS

21 different FEAs were conducted (three modelling types x four models = 12 bending FEAs and three modelling types x three models = 9 tensile FEAs). The numerical results for all four models, no. 1-4, have been compiled in Tables 5 and 6. Four different quantities were chosen for comparison of the results: the total displacements (consisting of vertical deflection only), the X stress component (along the beam), the Y stress component (across the beam), and the XY stress. The stress components are shown for the first layer and in the case of XY stress, the absolute values are presented in Tables 5 and 6. In the tables the “e” notation is used, which corresponds to the results on the plots. The displacements are in millimetres and the stresses are in megapascals. Due to the large number of obtained images with the simulation results, only some selected images are presented.

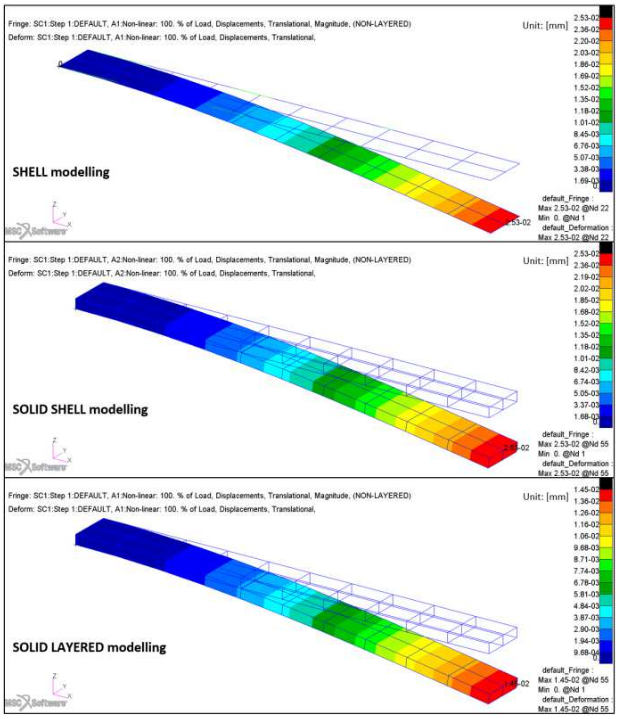


Fig. 14. Displacements for Model no. 3 in bending FEAs: shell, solid shell, solid layered FE modelling

Rys. 14. Odkształcenia dla Modelu nr 3 dla analiz MES dla zginania: modelowanie powłokowe, bryłowo-powłokowe, bryłowo-warstwowe

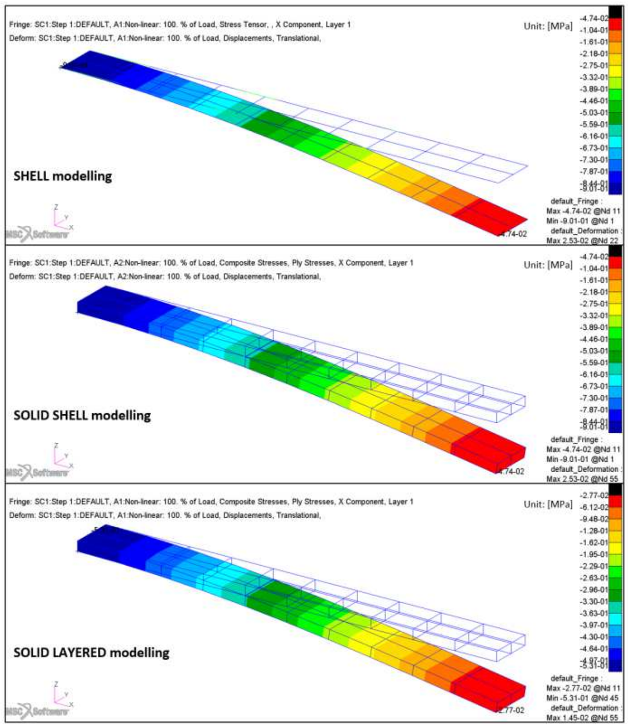


Fig. 15. X stress component for Model no. 3 in bending FEAs: shell, solid shell, solid layered FE modelling

Rys. 15. Składowa naprężenia X dla Modelu nr 3 dla analiz MES dla zginania: modelowanie powłokowe, bryłowo-powłokowe, bryłowo-warstwowe

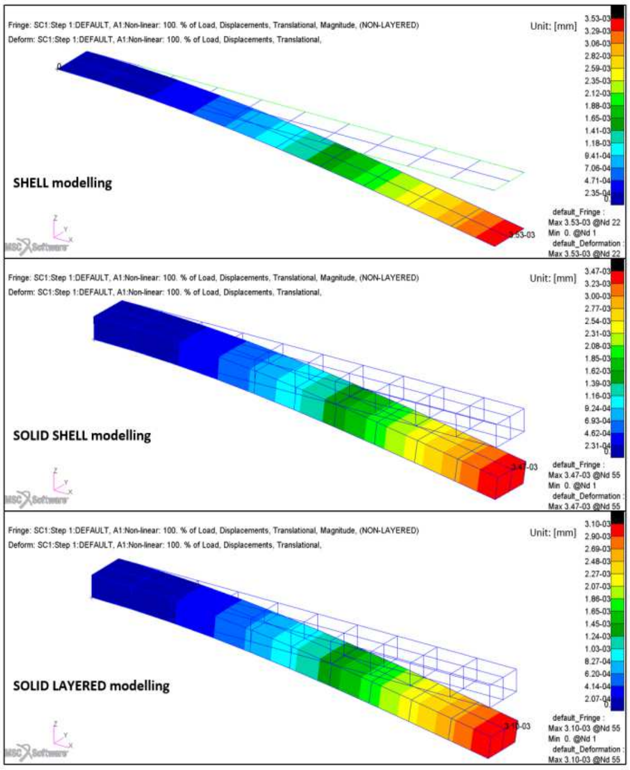


Fig. 16. Displacements for Model no. 4 (sandwich structure) in bending FEAs: shell, solid shell, solid layered FE modelling

Rys. 16. Odkształcenia dla Modelu nr 4 (struktura przekładkowa) dla analiz MES dla zginania: modelowanie powłokowe, bryłowo-powłokowe, bryłowo-warstwowe

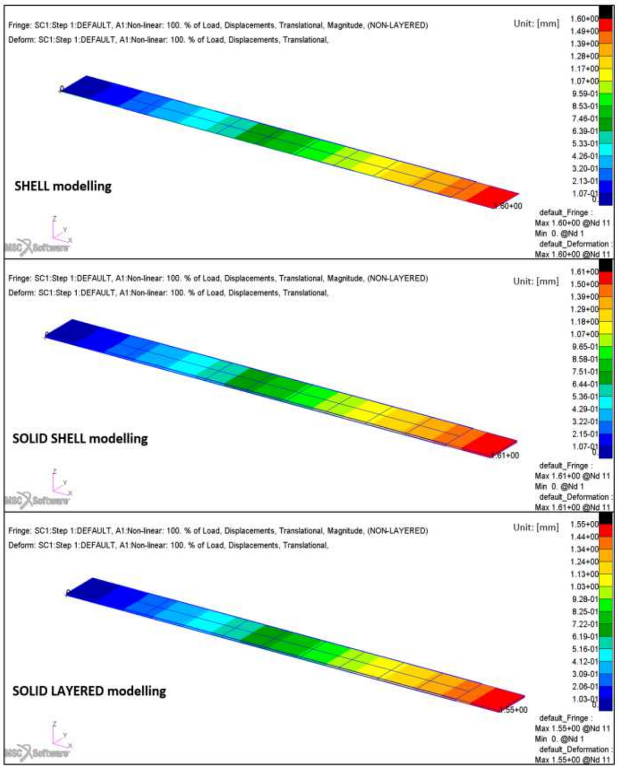


Fig. 18. Displacements for Model no. 1 in tensile FEAs: shell, solid shell, solid layered FE modelling

Rys. 18. Odkształcenia dla Modelu nr 1 dla analiz MES dla rozciągania: modelowanie powłokowe, bryłowo-powłokowe, bryłowo-warstwowe

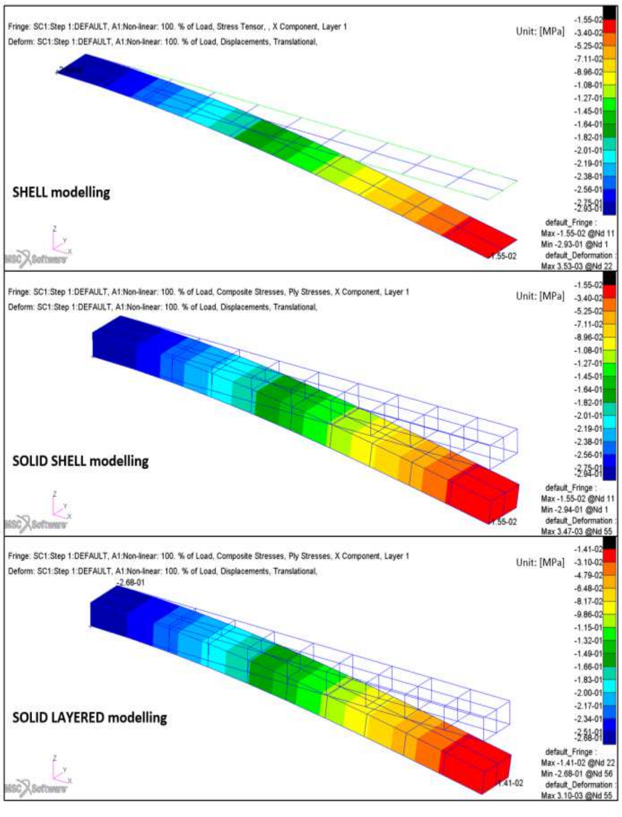


Fig. 17. X stress component for Model no. 4 (sandwich structure) in bending FEAs: shell, solid shell, solid layered FE modelling

Rys. 17. Składowa naprężenia X dla Modelu nr 4 (struktura przekładkowa) dla analiz MES dla zginania: modelowanie powłokowe, bryłowo-powłokowe, bryłowo-warstwowe

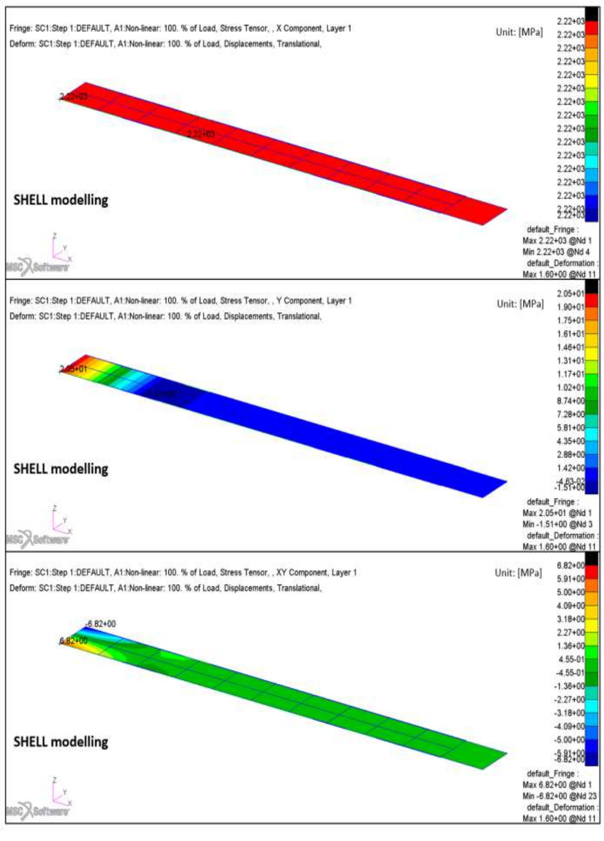


Fig. 19. X, Y stress components and XY stress for Model no. 1 in tensile FEAs: shell modelling

Rys. 19. Składowa naprężenia X, Y i naprężenie XY dla Modelu nr 1 dla analiz MES dla rozciągania: modelowanie powłokowe

TABLE 5. **Simulation results for bending FEAs**
TABELA 5. **Wyniki symulacji dla analiz MES dla zginania**

MODEL	Quantity	Unit	SHELL Composite Modelling	SOLID SHELL Composite Modelling	SOLID LAYERED Composite Modelling
BENDING FINITE ELEMENT ANALYSES (FEAs) OF LAMINATE BEAM MODELS					
MODEL 1	Max. displacement	[mm]	3.15	3.14	0.165
	X stress for 1st layer: at free end	[MPa]	1.05e-03	6.65e-02	2.34e-04
	at fixed end		1.01e-04	1.81e-03	-1.47e-04
	Y stress for 1st layer: at free end	[MPa]	1.65e-04	5.80e-03	2.05e-05
	at fixed end		–5.63e-04	–3.22e-02	–1.22e-04
	Max. XY stress for 1st layer	[MPa]	6.32e-03	4.45e-03	1.56e-05
MODEL 2	Max. displacement	[mm]	0.395	0.393	0.071
	X stress for 1st layer: at free end	[MPa]	–0.184	–0.184	-0.034
	at fixed end		–3.52	–3.52	-0.65
	Y stress for 1st layer: at free end	[MPa]	1.87e-03	2.44e-03	–1.38e-03
	at fixed end		–4.77e-02	–4.92e-02	–2.76e-02
	Max. XY stress for 1st layer	[MPa]	1.36e-02	1.43e-02	4.89e-04
MODEL 3	Max. displacement	[mm]	0.0253	0.0253	0.0145
	X stress for 1st layer: at free end	[MPa]	–0.0474	–0.0474	–0.0277
	at fixed end		–0.901	–0.901	–0.531
	Y stress for 1st layer: at free end	[MPa]	4.32e-04	5.71e-04	–8.55e-04
	at fixed end		–1.02e-02	–1.11e-02	–2.26e-02
	Max. XY stress for 1st layer	[MPa]	3.25e-03	3.27e-03	2.07e-03
BENDING FINITE ELEMENT ANALYSES (FEAs) OF SANDWICH BEAM MODEL					
MODEL 4 Sandwich structure	Max. displacement	[mm]	3.53e-03	3.47e-03	3.10e-03
	X stress for 1st layer: at free end	[MPa]	–1.55e-02	–1.55e-02	–1.41e-02
	at fixed end		–2,93e-01	–2.94e-01	–2.68e-01
	Y stress for 1st layer: at free end	[MPa]	8.07e-05	9.14e-05	–2.60e-04
	at fixed end		–2.97e-03	–3.17e-03	–1.14e-02
	Max. XY stress for 1st layer	[MPa]	9.21e-04	9.16e-04	2.14e-03

TABLE 6. **Simulation results for tensile FEAs**
TABELA 6. **Wyniki symulacji dla analiz MES dla rozciągania**

MODEL	Quantity	Unit	SHELL Composite Modelling	SOLID SHELL Composite Modelling	SOLID LAYERED Composite Modelling
TENSILE FINITE ELEMENT ANALYSES (FEAs) OF LAMINATE BEAM MODELS					
MODEL 1	Max. displacement	[mm]	1.60	1.61	1.55
	X stress for 1st layer: at free end	[MPa]	2.22e+03	2.22e+03	2.16e+03
	at fixed end		2.22e+03	2.21e+03	2.22e+03
	Y stress for 1st layer: at free end	[MPa]	–1.51	–2.32	–3.46
	at fixed end		20.5	41.7	94.1
	Max. XY stress for 1st layer	[MPa]	6.82	5.22	15.2
MODEL 2	Max. displacement	[mm]	0.799	0.802	0.786
	X stress for 1st layer: at free end	[MPa]	1.11e+03	1.11e+03	1.09e+03
	at fixed end		1.11e+03	1.11e+03	1.12e+03
	Y stress for 1st layer: at free end	[MPa]	–0.789	–1.21	–2.08
	at fixed end		10.1	20.8	47.6
	Max. XY stress for 1st layer	[MPa]	3.48	2.68	7.77
MODEL 3	Max. displacement	[mm]	0.32	0.32	0.317
	X stress for 1st layer: at free end	[MPa]	4.44e+02	4.44e+02	4.42e+02
	at fixed end		4.44e+02	4.44e+02	4.52e+02
	Y stress for 1st layer: at free end	[MPa]	–0.323	–0.494	–0.914
	at fixed end		4.00	8.33	1.92
	Max. XY stress for 1st layer	[MPa]	1.41	1.09	3.16

CONCLUSIONS

Based on the numerical results for the laminated beam and sandwich beam models (Tables 5 and 6), the following conclusions can be stated:

- The simulation results for both the shell and solid shell composite FE modelling types are very close to each other; the numerical values and distributions are almost identical.
- The displacements for both the shell and solid shell composite FE modelling types are practically identical, whereas the stresses are slightly different and tend to be similar for models with more layers. It can be

concluded that solid shell composite FE modelling is more accurate for thicker structures.

- The distributions of displacements and stresses are almost the same for the three types of composite FE modelling.
- The displacements and stresses obtained for solid layered composite FE modelling are smaller than the displacements and stresses for the shell and solid shell composite FE modelling types. Therefore, it can be concluded that solid layered composite FE modelling "over-stiffens" the structures.
- The assumed strain formulation is an effective tool when using solid elements because then the results for solid shell elements are very close to the results for the shell elements. This applies especially to the displacements.
- It should be emphasized that material data is often not widely available and may not be disclosed. Therefore, conclusions can be drawn from the bending and tensile FEAs of models made from composite materials depending on the number of available material constants (Fig. 20).

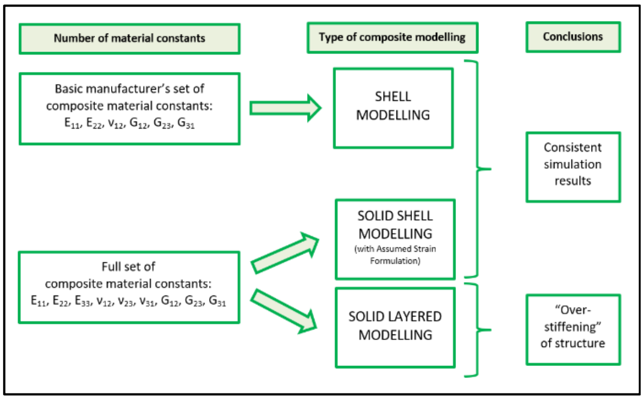


Fig. 20. Choice of composite finite element modelling type based on number of composite material constants

Rys. 20. Wybór modelowania elementami skończonymi kompozytów na podstawie liczby kompozytowych stałych materiałowych

- For the basic manufacturer's set of material constants, it is advisable to use shell composite modelling instead of solid composite modelling, especially due to the fact that both types of modelling generate very similar results (number-wise and distribution-wise). Shell composite modelling allows one to change the thickness in the material properties assignment step in the program without changing the geometry, while solid elements require a change in the geometry in order to change the thickness of the structure. From the practical point of view, the change in the thickness can be done much quicker for shell elements and thus reduce modelling effort and time.
- For a full set of material constants when solid elements are used, it is advisable to use solid shell composite modelling, which incorporates the assumed strain formulation. The presented simulation results for solid shell composite modelling and for shell composite modelling are very similar. Therefore, solid shell composite modelling allows structures to behave more like shells, whereas in solid layered composite modelling the "over-stiffening phenomenon" occurs, especially visible in bending.

SUMMARY

From the point of view of the costs of finite element modelling, it is much more efficient to use shell composite FE modelling. This modelling type:

- allows one to change the thickness of the composite without changing the geometry of the model,
- does not require one to define the orientation of the plies (in solid finite elements it is more cumbersome),
- reduces modelling effort,
- reduces modelling time,
- can be used for the basic manufacturer's set of composite material constants, which is a very important issue when not all material data are available,
- allows one to obtain consistent simulation results.

The presented numerical studies showed the development of different FE modelling approaches to composite materials. As a result of the numerical simulations, displacement and stress fields were obtained for the composite shell and solid finite element models. A comparison of the obtained numerical results allowed the authors to conclude that the developed numerical models allow one to study the impact of different FE modelling types of composite materials on the simulation results. This is especially important in the case where research studies on comparing different methods of FE modelling of composite materials are rare.

The simulation results for solid layered composite FE modelling showed some numerical stiffening of the structure. On the other hand, the numerical results for solid shell composite FE modelling (with the assumed strain formulation) had very good agreement with the results for shell composite FE modelling. However, one of the main limitations of this modelling type is the need for more material data, which is not always available or published. Therefore, shell composite FE modelling is a good choice as it requires fewer material data, it reduces the modelling effort and time, as well as allows one to obtain consistent simulation results, especially when having only the basic manufacturer's set of material constants.

REFERENCES

[1] Sauer M., Kühnel M., Witten E., Composites Market Report 2017: Market Developments, Trends, Outlook and Challenges, Federation of Reinforced Plastics, 2017.

[2] Gay D., Hoa S.V., Tsai S.W., Composite materials design and Applications, CRC Press, Boca Raton 2003.

[3] MSC Software Corporation, MD Nastran & MSC Nastran Quick Reference Guide, 2018.

[4] MSC Software Corporation, Section 2 Solid Composites, Composites Technology Day, MSC Users Conference 2012.

[5] Barkanov E., Ozoliņš O., Eglītis E., Almeida F., Bowering M.C., Watson G., Optimal design of composite lateral wing upper covers. Part I: Linear buckling analysis, Aerospace Science and Technology 2014, 38, 1-8.

[6] Rumayshah K.K., Prayoga A., Agoes Moelyadi M., Design of High Altitude Long Endurance UAV: Structural Analysis of Composite Wing using Finite Element Method, Journal of Physics: Conference Series 2018, 1005(1), 5th International Seminar of Aerospace Science and Technology, 1-11.

[7] Shetty B.P., Reddy S., Mishra R.K., Finite element analysis of an aircraft wing leading edge made of GLARE material for structural integrity, Journal of Failure Analysis and Prevention, 2017, 17(5), 948-954.

[8] Perfetto D., Lamanna G., Manzo M., Chiariello A., Di Caprio F., Di Palma L., Numerical and experimental investigation on the energy absorption capability of a full-scale composite fuselage section, Key Engineering Materials 2019, 827, 19-24.

[9] Giglio M., Manes A., Gilioli A., Investigations on sandwich core properties through an experimental – numerical approach, Composites Part B 2012, 43(2), 361-374.

[10] Anoshkin A.N., Zuiko V.Yu., Alikin M.A., Tchugaynova A.V., Numerical simulation of mechanical behaviour of composite sandwich panels with defects, ECCOMAS Congress 2016, VII European Congress on Computational Methods in Applied Sciences and Engineering, Eccomas Proceedia ID: 2376, Conference Proceeding ID: 9724, 7800-7809.

[11] MSC Software Corporation, MSC Nastran Nonlinear (SOL 400) User's Guide, 2018.

[12] MSC Software Corporation, Getting Started with MD Nastran User's Guide, 2010.

[13] MSC Software Corporation, Linear Static Analysis, User's Guide, 2011.

[14] Chen J., Hallett S., Wisnom M. R., Modelling complex geometry using solid finite element meshes with correct composite material orientations, Computers and Structures 2010, 88(9), 602-609.

[15] Roy R., Kweon J.H., Choi J.H., Meso-scale finite element modeling of Nomex^{TM} honeycomb cores, Advanced Composite Materials 2014, 23, 1, 17-29.

[16] Reddy J.N., An Introduction to the Finite Element Method, 2nd Edition, McGraw-Hill, Inc., 1993, 5-6.

