PROBLEMY MECHATRONIKI
UZBROJENIE, LOTNICTWO, INŻYNIERIA BEZPIECZEŃSTWA

ISSN 2081-5891 **9, 4 (34), 2018, 101-112**

PROBLEMS OF MECHATRONICS
ARMAMENT, AVIATION, SAFETY ENGINEERING

Numerical Analysis of a DMA Epoxy-Carbon Composite Study

Paweł KOWALECZKO[1*], Andrzej PANAS[2],
Mirosław NOWAKOWSKI[1]

[1]Air Force Institute of Technology
6 Księcia Bolesława Str., 01-494 Warsaw, Poland
[2]Military University of Technology, Faculty of Mechatronics and Aerospace,
2 gen. Witolda Urbanowicza Str., 00-908 Warsaw, Poland
Corresponding author's e-mail address: pawel.kowaleczko@itwl.pl

Received by the editorial staff on 14 November 2017
The reviewed and verified version was received on 20 December 2018

DOI 10.5604/01.3001.0012.7336

Abstract. The results of numerical simulations performed for Dynamic Mechanical Analysis (DMA) measurements of thermal and mechanical (or thermomechanical) properties performed on a model composite structure are presented herein. The simulated elastic response of an epoxy-carbon fibre composite specimen was analysed for a case by which the model specimen was subjected to three-point bending with a free support. The epoxy-carbon fibre composite studied as explained herein exhibited extreme differences between the elastic properties of the epoxy resin matrix and the carbon fibre reinforcement. In addition, the carbon fibre reinforcement was both internally and structurally anisotropic. The numerical simulations were performed to demonstrate a qualitative dependence of the DMA measurement results on a certain structure of the investigated specimen and to determine if the DMA results could be qualified as effective or apparent.

A macro-mechanical model of the specimen was developed and the numerical calculations were performed by applying a COMSOL/M FEM (Finite Element Method) modelling software. The carbon fibre reinforcement was modelled with an orthotropic composite structure of planar laminar inclusions or as a disperse composite with circular inclusions. While modelling different characteristic dimensions of inclusions were taken into account. Representative material properties were assumed from the results of the appropriate experimental investigations and form certain literature reference data. The effect of the composite layer configuration and their characteristic dimensions on the evaluated model elastic modulus value was also studied. The numerical modelling results are in a qualitative agreement with the results of the DMA investigations performed on real composite. They also proved the effectiveness of the developed numerical simulation methodology in modelling of micro- and macromechanical phenomena occurring during the DMA study.
Keywords: composite material, DMA, numerical simulation, COMSOL

1. BACKGROUND

DMA (Dynamic Mechanical Analysis [1, 2], a.k.a. dynamic thermal mechanical analysis, dynamic thermomechanical analysis) is currently among the most popular tools used to determine viscoelastic properties of materials. DMA has proven itself especially well in the determination of the viscoelastic properties of heterogeneous composites. DMA comprises testing with application of oscillating force and observation of a test specimen oscillating response. The oscillating response is, essentially, the displacement of the test specimen. By analysing the response, it is possible to determine the elastic modulus, E' (the so-called ‘storage modulus’), and the loss modulus, E'', which are, respectively, the real component and the imaginary component of a generalized complex Young’s modulus determined with DMA. The real component complies with the strain phase. The imaginary component is offset by $\pi/2$. The real component of Young’s modulus is zero for perfectly viscous bodies. The imaginary component of Young’s modulus is zero for perfectly resilient bodies [3]. Composites which feature viscoelastic characteristics have both moduli not equal to zero, whereas $E'' < E'$.

One of the most practical methods of determining the properties of glass (or graphite) fibre reinforced composite materials is to test their flat specimens with the edges freely supported and the oscillating variable force input applied along the centreline of the specimen (see [1]). This method has been developed for testing the properties of homogeneous viscoelastic materials. It is highly sensitive to changes in the dimensions and geometry of the specimens. It is then necessary to determine the reliability of experimental results obtained with the method. For heterogeneous materials, it is problematic to determine the modulus of elasticity, E', and the relation of the calculated value with Young’s modulus values determined with other test methods, e.g. tensile tests.

The problem here is to qualify the calculated values of E' either as effective properties or apparent properties, as the case may be.

The concept of 'effective properties' is a category of parameters which can be homogenised; note that macroscopic test results do not facilitate a determination of whether the tested structure is heterogeneous or not. 'Apparent properties' are the calculated material parameters of heterogeneous structures that can be characterised as substitutive homogeneous materials. The potential contributors to apparent properties include physical phenomena which exceed the confines of the theoretical description of the primary phenomenon. For the DMA discussed herein, it may include adhesion at the matrix/fibre interface. Macroscopic test results usually facilitate the effective ones among the properties being determined. Mathematical modelling is applied for testing the sensitivity of methods, i.e. to determine its metrological preconditions and to qualify the values determined with the method as effective or apparent ones. Unfortunately, given the sheer complexity of the theoretical description, analytical modelling is largely restricted and poorly effective for the purposes contemplated in the previous sentence.

Numerical modelling suffers from less problematic restrictions in the same context. The primary challenge in numerical simulations run with commercial software products is the missing facilities to model the properties of viscoelastic materials. It is however possible to run calculations and determine the displacement within test specimens in response to an input of mechanical loads. A facility for those calculations is provided in the environment of COMSOL Multiphysics software [4, 5, 6]. The results of numerical simulations can provide inputs for an analysis of method sensitivity regarding the determination of the modulus of elasticity. It is also feasible to do a qualitative determination of how the layout of graphite fibres in an epoxy resin matrix composite and the geometrical features of test specimens of the material affect the calculated viscoelastic effective or apparent properties.

This paper describes operations involving an application of numerical modelling in testing the metrological preconditions of DMA-based measurements carried out on specimens of a carbon (or graphite) fibre reinforced epoxy matrix composite. All tests were limited to an analysis of the elastic response of the specimens, which condensed the problem at hand to a statistical problem of a loaded beam. The development of the methodology of numerical simulations and the verification of efficiency of modelling for the numerical simulations aside, the numerical simulations were performed to demonstrate a qualitative dependence of the DMA measurement results on a certain structure of the investigated specimen and to determine if the DMA results could be qualified as effective or apparent. Positive results would enable expansion of the scope of future theoretical and numerical analysis.

2. FORMULATION OF THE COMPUTATIONAL PROBLEM

The authors assumed that a module of COMSOL Multiphysics would be used to run numerical calculations to produce the response of a test specimen to a mechanical load input. The numerical simulations were limited to a DMA test with a three-point support of the specimen by which the test specimen was placed on two supports with the contact points of the supports on the specimen's surface prevented from moving along the axis *Oy* (Fig. 1). The surface of the test specimen opposite to the side contacting the supports was loaded with a force sufficient to cause the test specimen material to bend.

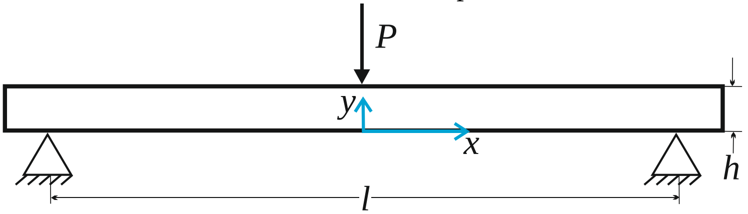


Fig. 1. Diagram of loading the test specimen (a beam) with a force input applied in between the supports

A problem of static bending loads was formulated for a test specimen with a constant rectangular cross-section, a height of *h* and a width of *b* (in the direction perpendicular to plane *Oxy*). The spacing of the supports under the test specimen was *l*. An analytical solution of the problem provided a dependence by which the beam bend, *f*, could be determined: [7]

$$f = \frac{l^3}{48} \frac{P}{E\, J_a} \tag{1}$$

with: E – Young's modulus; J_a – moment of inertia of the cross-section relative to the horizontal neutral line; P – pressure input applied to the middle of the beam (the test specimen).
For the rectangular cross-section, the moment of inertia was:

$$J_a = \frac{h^3 b}{12} \tag{2}$$

Substitution of (2) in (1) provided this:

$$f = \frac{l^3}{4bh^3} \frac{P}{E} \tag{3}$$

A transformation of the foregoing dependence allowed a determination of E:

$$E = \frac{l^3}{4bh^3} \frac{P}{f} \tag{4}$$

3. THE NUMERICAL MODEL

The test specimen had a two-dimensional numerical model developed for the numerical simulation. However, the applied software suite adopted the model as a three-dimensional structure with a specific declared dimension of b along the axis Oz (Fig. 1, see the perpendicular to the plane of view). The authors assumed b = 10 mm, according to the data from standard specimens.

The inner structure of the composite material was modelled by applying rectangular or round inclusions representing the modelled fibres. The areas declared for the numerical calculations were divided into finite triangular elements (Fig. 2). The numerical calculations were run for three different types of the specimen models.

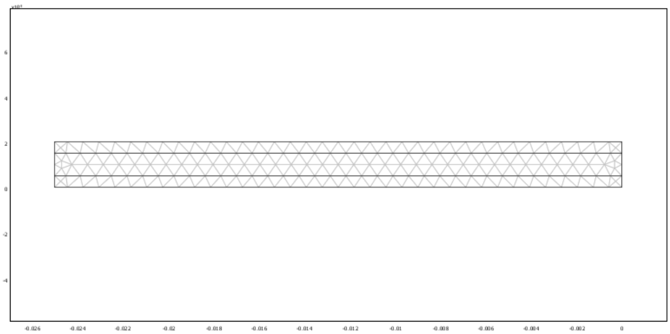
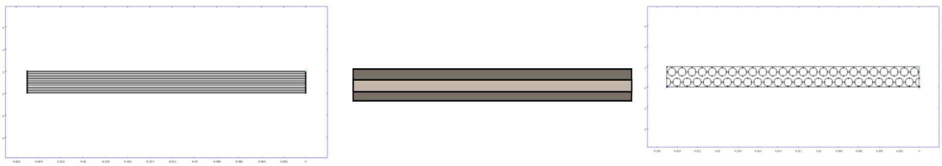


Fig. 2. Diagram of the three-layer laminate specimen breakdown into finite triangular elements

For each type, the fibre content was the same and 50% by volume. Three basic types of the test specimens were investigated:

a) a multi-layer laminate with thin layers of the modelled rowing and the carbon fibres aligned along the axis Ox. Here, for the sake of retaining symmetry, an 11-layer fibre reinforcement was adopted with the first and last layer on the exterior of the test sample, and with all 11 layers alternating with 10 layers of the matrix (Fig. 3(a));



(a) Multi-layer laminate (b) Three-layer laminate (c) Rowing cross-section model

Fig. 3. Models of the specimen internal structure types

b) a three-layer laminate constructed with the same principle as above (Fig. 3(b));
c) a sample with regular, dispersed circular inclusions which provided a rough model of the rowing carbon fibres aligned with the axis *Oz*, i.e. in perpendicular to the side surface of the specimen model (and essentially being the rowing cross-section, Fig. 3(c)).

Only one half of the test specimen was numerically modelled for the numerical calculations by applying the symmetry of load (a flip around the axis *Oy*). Hence, Figures 3(a), (b) and (c) show only one half of the test specimen subject to loading. In each version of the numerical calculations, the support point was at the coordinates (–25.0 mm; 0.0 mm) with an analogical (not shown) support point at the coordinates (0.025 m; 0 m). A force input of 10 N was applied to the point at the coordinates (0 m; 0.002 m). However, given the applied flip of the test specimen around the axis *Oy*, the applied force input was double the value. Thus, the actual force input studied in this paper was 20 N. The characteristic dimension of the specimen models are shown in Table 1.

Table 1. External dimensions of the specimen models

Characteristic dimensions of the specimen models		
Support spacing, *l*, mm	Height, *h*, mm	Width, *b*, mm
50	2	10

For the numerical modelling, it was assumed that the carbon fibre reinforcement could be flat inclusions of a homogeneous anisotropic material. The material properties of the resin and the carbon fibres from the reference literature data [2, 6, 8, 9, 10] were used to determine the representative material properties of the modelled composite structures. The list of model data is shown in Tables 2-4. The purpose of the numerical simulations was first to determine qualitative results; specific physical parameter values for the resin and the graphite were irrelevant. Of importance was to retain a disproportion between the properties of the matrix and the properties of the fibres, to represent the anisotropy of the fibres, and to reproduce a generally proper internal structure of the test specimens which (given their constant geometrical features) was the primary contributor to the variations in the calculated values of *E’*.

Note that a numerical simulation check was run for each of the test specimen models with an assumption that the matrix and the fibres featured the physical characteristics of the resin. The checks were intended to verify whether the modulus of elasticity calculated from dependence (4) in the parametric identification procedure complied with Young’s modulus value assumed when building the numerical model. Further numerical simulations were run on the laminate specimens in two different configurations:

- *zgz* – where the outer layer was the resin;
- *gzg* – where the outer layer was the carbon fibre rowing.

Table 2. Selected material properties of the graphite fibres in the multi-layer laminate and the three-layer laminate

Multi-layer laminate / Three-layer laminate – graphite fibres							
Young's modulus, GPa			Poisson's ratio ν_{xy}, ν_{yz}, ν_{xz}			Modulus G_{xy}, GPa	Density ρ, kg/m^3
E_x	E_y	E_z					
385	6.3	6.3	0.20	0.20	0.20	7.7	1940

Table 3. Selected material properties of the graphite fibres in the rowing model

Rowing model – graphite fibres							
Young's modulus, GPa			Poisson's ratio ν_{xy}, ν_{yz}, ν_{xz}			Modulus G_{xy}, GPa	Density ρ, kg/m^3
E_x	E_y	E_z					
6.3	6.3	**385**	0.20	0.20	0.20	7.7	1940

Table 4. Selected material properties of the resin in all three specimen types

Multi-layer laminate / Three-layer laminate / Rowing model		
Young's modulus, E, GPa	Poisson's ratio	Density ρ, kg/m^3
3	0.49	1200

4. NUMERICAL SIMULATION AND CALCULATION RESULTS

The result of the numerical calculations was the determination of the modelled beam bend, f, at the point of force input in each of the analysed cases. Figure 4 shows examples of the calculations for the three-layer laminate model. For the determined bends, dependence (4) was applied to calculate the values of E', as was the case in the processing of the results of recording the specimen's response in the DMA tests (see [1]). Table 5 and Figs. 5–8 show the calculation results for the different versions of this study.

As already said, the mesh was built with triangular elements (Fig. 2). Test calculations were also run to analyse the sensitivity of the numerical model to the numerical mesh number. The calculations were run for nominal and dense versions of the numerical mesh. The nominal mesh had 308 elements, while the dense numerical mesh had 19,712 elements. The maximum deflection values were, respectively: f_1 = 2.617 mm for the nominal numerical mesh and f_2 = 2.619 mm for the dense numerical mesh.

Table 5. Specimen bend values and the modulus of elasticity value for different models and configurations

	Case no.	Configuration	Bend, *f*, m	Modulus of elasticity, Pa
Multi-layer laminate	1	*gzg*	$4.864 \cdot 10^{-5}$	$1.61 \cdot 10^{11}$
	2	*zgz*	$5.270 \cdot 10^{-5}$	$1.48 \cdot 10^{11}$
	3	Resin	$2.621 \cdot 10^{-3}$	$2.98 \cdot 10^{9}$
Three-layer laminate	4	*gzg*	$3.417 \cdot 10^{-5}$	$2.29 \cdot 10^{11}$
	5	*zgz*	$1.600 \cdot 10^{-4}$	$4.88 \cdot 10^{10}$
	6	Resin	$2.619 \cdot 10^{-3}$	$2.98 \cdot 10^{9}$
Rowing cross-section model	7	Resin	$2.620 \cdot 10^{-3}$	$2.98 \cdot 10^{9}$
	8	Fibres	$1.029 \cdot 10^{-3}$	$7.59 \cdot 10^{9}$

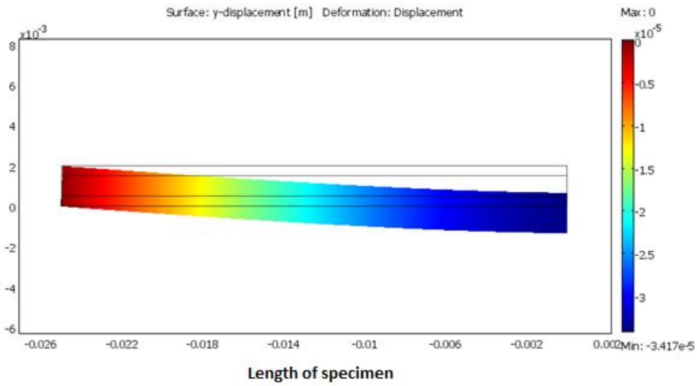


Fig. 4. Examples of the numerical simulation results produced for the three-layer laminate specimen in the *gzg* configuration: distribution of displacements along the axis *Oy*, with a presentation of the outline of the deformed specimen

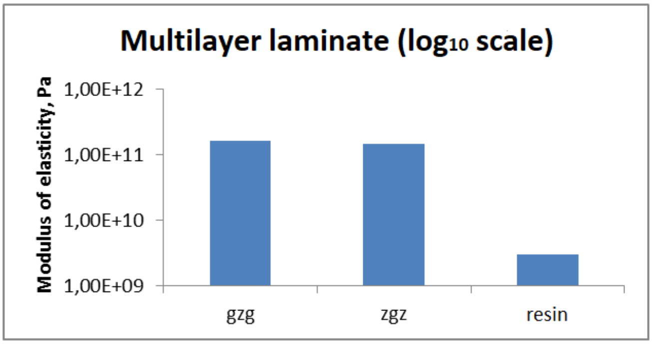


Fig. 5. Calculated values of the modulus of elasticity for the multi-layer laminate specimen models

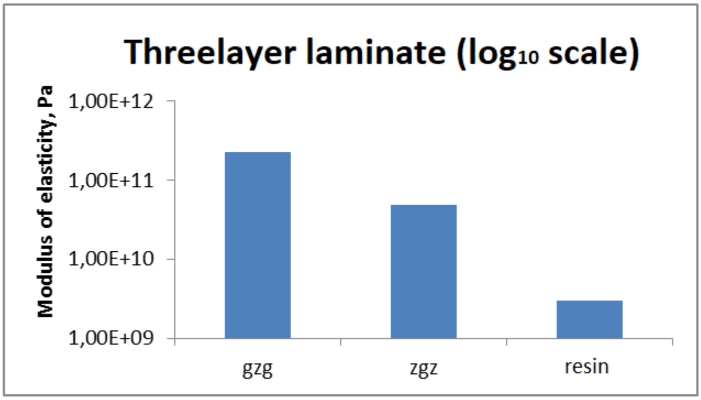


Fig. 6. Calculated values of the modulus of elasticity for the three-layer laminate specimen models

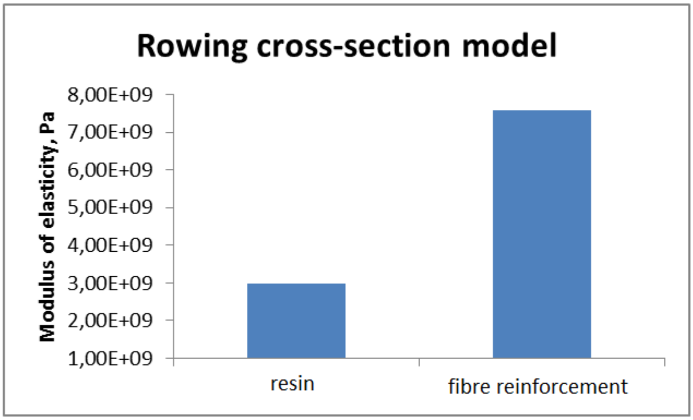


Fig. 7. Calculated values of the modulus of elasticity for the rowing cross-section model in the transverse configuration

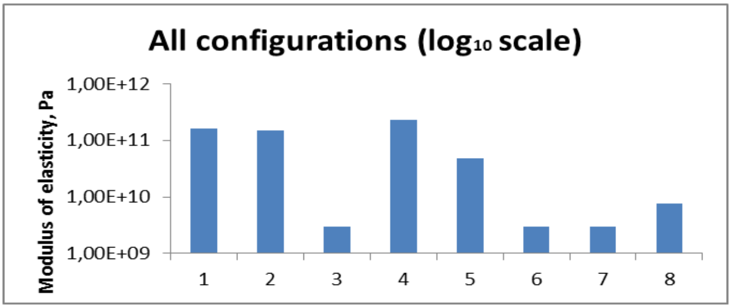


Fig. 8. Calculated values of the modulus of elasticity for all specimen models, all configurations studied (the numbering is applied from Table 5)

A conclusion was valid that given that the difference between both values was below 0.1%, the density of the mesh elements of the specimen model had no significant impact on the results of the numerical simulation. Similar results were produced for other numerical calculation versions.

5. ANALYSIS OF RESULTS

The analysis of the results listed in Table 5 and shown in Figures 4–7 validated a conclusion that the values of the identified modulus of elasticity, *E'*, for the specimens made from the resin only generally complied with the values declared for the model in COMSOL. It was reasonable to conclude that all other results of the numerical simulations were reliable. Among the analysed specimen types, the rowing cross-section model featured the lowest modulus of elasticity, equal to 7.59 GPa. Conversely, the highest modulus of elasticity, equal to 229 GPa, was produced for the three-layer laminate model with the *gzg* configuration. Significant differences were found when the results were compared between the multi-layer laminate models and the three-layer laminate models. The three-layer laminate models had higher absolute and relative differences between the calculated modulus of elasticity (180.2 GPa, respectively, which provided a higher to lower value ratio of 4.7/1) than the multi-layer laminate models (13 GPa, respectively, and 1.08/1). For the three-layer laminate models, a major contributor was the orientation of the layers: the *gzg* configuration specimen was 4.7 times more rigid than the *zgz* configuration specimen. This was an effect of the transfer of stress within the specimens.

The stresses would become smaller as they approached the plane of symmetry of the specimen. The plane of symmetry was perpendicular to plane *Oxz* and crossed the centre of the test specimen. When a more rigid material was located in the outer layers, the entire test specimen became more rigid. These conclusions seem obvious from a strength analysis perspective. However, the tests completed, as discussed herein, helped visualise the problem of homogenising the properties of heterogeneous composite structures. The multi-layer laminate did not reveal differences as big as above. Given the assumed number of 11 layers of inclusions, which meant that the transverse dimension of an inclusion was approximately 20 times less than the transverse dimension of the whole structure, the difference between the calculated values of stiffness was 8% between the two cases. The calculated values of the modulus of elasticity, *E'*, for the modelled multi-layer structure were between the limit values (229 GPa to 48.8 GPa) achieved for the three-layer model.

6. CONCLUSION

The general conclusion of the tests discussed here should focus on the effectiveness of the developed method of numerical analysis.

The results provided by the author pointed to a qualitative agreement with the results of testing real-life composite structures [12]. The results also proved the effectiveness of the method of DMA modelling for structurally homogeneous and heterogeneous specimens both on macrostructural and microstructural levels. The macrostructural heterogeneousness were represented by the three-layer laminate models. Here, the dimension of the inclusions was comparable to the characteristic dimension of the investigated specimen. The multi-layer laminate models could be generally qualified as microstructural models. The dimension of the inclusions was small enough that, provided a uniform distribution of the inclusions, an 8-percent difference resulted in the calculated values of the modulus of elasticity between two special cases, *zgz* and *gzg* (Table 5). This condition permitted characterisation of the structure with volumetric composition fractions (with the structure homogenised, see [11]). An important practical conclusion was the proof of a strong directional dependence of the modulus of elasticity, *E'*, determined in these tests. The bend, or deflection, or the response of the modelled system configuration to the inputs depended highly on the orientation of fibres in the modelled specimens, as shown in the lengthwise and transverse configuration tests done on the models. This was confirmed by the differences in the modelling results between the laminates and the transverse rowing cross-section model (Table 5). The natural course of further research would be operations related to three-dimensional modelling with consideration of the large-scale nature of modelled composite structures. Application of other calculation models in COMSOL/M, or the addition of proprietary subroutines, could facilitate the viscous component, represented by the loss modulus, *E''*, in the models.

REFERENCES

[1] Menard P. Kevin. 1999. *Dynamic Mechanical Analysis. A Practical Introduction*. CRC Press, Boca Raton.

[2] Panas J. Andrzej, Janusz Błaszczyk, Andrzej Dudziński, Krzysztof Figur, Anna Foltyńska, Anna Krupińska, Mirosław Nowakowski. 2016. „Badania wpływu temperatury na zmiany właściwości cieplnych i mechanicznych osnowy lotniczego konstrukcyjnego materiału kompozytowego". *Mechanika w lotnictwie ML-XVII* II : 147-158.

[3] Sobczak Renata, Zygmunt Nitkiewicz, Józef Koszkul. 2002. „Badania dynamicznych właściwości mechanicznych kompozytów na osnowie polipropylenu wzmocnionych włóknem szklanym". *Kompozyty* 2 : 78-80.

[4] COMSOL Multiphysics User's Guide, 1998-2008, COMSOL AB.

[5] Structural Mechanics Module User's Guide, 1998-2008, COMSOL AB.

[6] Structural Mechanics Module Model Library, 1998-2008, COMSOL AB.

[7] Jakubowicz Antoni, Zbigniew Orłoś. 1978. *Wytrzymałość materiałów*. Warszawa: Wydawnictwo Naukowo-Techniczne.
[8] Miwa Minoru, Akiyosi Takeno, Y. Mori, Takeichiro Yokoi, Akira Watanabe. 1998. "Effects of Young's Modulus of Epoxy Resin on Axial Compressive Strength of Carbon Fibre". *Journal of Materials Science* 33(15) : 3885-3890.
[9] [dataset] *Material Property Data Base MPDB v. 8.37*, JAHAM Software, Inc., (accessed July 2017).
[10] http://www.mse.mtu.edu/~drjohn/my4150/props.html (*Properties of selected fibres*), (accessed November 2017).
[11] Ostoja-Starzewski Martin. 2017. *Mechanics of Random Media*. Lecture materials - Program Katedra Ad Hoc. Military University of Technology, Faculty of Mechanical Engineering, Warsaw, 26-29 June 2017.
[12] Panas J. Andrzej, M. Białecki. 2017. *Raport z badań LBC/04/2017*. Warszawa: Wydawnictwo ITWL.

Analiza numeryczna metodyki badań DMA kompozytu epoksydowo-węglowego

Paweł KOWALECZKO[1], Andrzej PANAS[2], Mirosław NOWAKOWSKI[1]

[1] Instytut Techniczny Wojsk Lotniczych
ul. Księcia Bolesława 6, 01-494 Warszawa
[2] Wojskowa Akademia Techniczna, Wydział Mechatroniki i Lotnictwa
ul. gen. Witolda Urbanowicza 2, 00-908 Warszawa

Streszczenie. W pracy przedstawiono wyniki numerycznych badań symulacyjnych pomiaru właściwości cieplnomechanicznych (termomechanicznych) struktury kompozytowej. Uwagę skupiono na symulacji sprężystej odpowiedzi próbki kompozytu epoksydowo-węglowego poddanego analizie termomechanicznej DMA (Dynamic Mechanical Analysis) w trybie zginania z trójpunktowym swobodnym podparciem. Kompozyt epoksydowo-węglowy jako obiekt analizy charakteryzuje się bardzo dużą dysproporcją właściwości sprężystych materiału osnowy i wypełnienia. Dodatkowo cechy anizotropii wykazuje już sam materiał wypełnienia w postaci włókien węglowych. Celem wykonanych badań numerycznych było jakościowe określenie wpływu struktury na wyniki badań DMA i określenie możliwości zaklasyfikowania uzyskiwanych wyników jako właściwości efektywnych bądź pozornych. Do opracowania modelu makromechanicznego badanej próbki i przeprowadzenia obliczeń wykorzystano program modelowania MES COMSOL. Wypełnienie włóknami węglowymi modelowano za pomocą ortotropowych wtrąceń warstwowych lub kołowych o różnych wymiarach charakterystycznych. Reprezentatywne dane materiałowe przyjęto na podstawie wyników badań własnych i danych literaturowych. Zbadany został wpływ rozmieszczenia poszczególnych warstw oraz ich wymiaru charakterystycznego na określany z modelowej zależności moduł sprężystości *E*'.

Słowa kluczowe: materiał kompozytowy, DMA, symulacja numeryczna, COMSOL