TRANSACTIONS ON AEROSPACE RESEARCH
4(257) 2019, pp.23-31
DOI: 10.2478/tar-2019-0020
eISSN 2545-2835

VALIDATION OF NUMERICAL MODELS USED FOR DESIGNING THE COMPOSITE BLADE FOR ILX-27 ROTORCRAFT

Jakub Wilk
Łukasiewicz Research Network – Institute of Aviation,
Al. Krakowska 110/114, 02-256 Warsaw, Poland
jakub.wilk@ilot.edu.pl

Radosław Guzikowski
Łukasiewicz Research Network – Institute of Aviation,
Al. Krakowska 110/114, 02-256 Warsaw, Poland
Radoslaw.guzikowski@ilot.edu.pl

Abstract

The paper presents the validation procedure of the model used in the analysis of the composite blade for the rotor of the ILX-27 rotorcraft, designed and manufactured in the Institute of Aviation, by means of numerical analyses and tests of composite elements. Numerical analysis using finite element method and experimental studies of three research objects made of basic materials comprising the blade structure – carbon-epoxy laminate, glass-epoxy composite made of roving and foam filler – were carried out. The elements were in the form of four-point bent beams, and for comparison of the results the deflection arrow values in the middle of the beam and axial deformations on the upper and lower surfaces were selected. The procedure allowed to adjust the discrete model to real objects and to verify and correct the material data used in the strength analysis of the designed blade.

Keywords: composites, FEM modelling, mechanical tests.

1. INTRODUCTION AND LITERATURE REVIEW

From the point of view of strength and rigidity, a typical rotorcraft rotor blade consists of three basic elements: a spar, a casing and a blade grip for mounting to the rotor [1]. Aerodynamic loads are applied to the strength elements of the blade by covering it [1]. Due to the material of blade production, it is possible to divide the blade into metal, composite and mixed ones, in which the metal beam is combined with the composite coating [1]. In the case of composite blades, the spars may have one of the basic forms [2]:

a) a pipe made of one- or two-way fabrics impregnated with resin;
b) roving bundles formed into a compact spar;
c) a structure combining the two above methods.

Strength analyses with the finite element method become an inseparable stage of designing composite aircraft structures as a part of the so-called block approach, examples of which are shown in [3÷5]. The designing of rotorcraft rotor blades is no different – numerical analyses are vital part of the processes described in [6,7] They also played an important role in the development of the composite structure of the ILX-27 rotor blade for the rotorcraft main rotor, carried out at the Institute of Aviation [8]. The blade structure consisted of 3 basic materials – laminar carbon epoxy composite used to cover the structure, glass-epoxide composite made of roving, used as a spar material, and foam filler. Numerical analyses were used to determine optimal composite parameters in terms of the number of layers and directions of reinforcement, as well as to determine the entire structure strength.

The key element of numerical strength analysis is model validation, which confirms the correct representation of a real object by a discrete model [9]. The common approach is to compare the results of numerical analyses with the experimental investigation of an element representing the designed structure, as in [10,11]. The paper presents the validation procedure of the model used in the analysis of the composite blade by means of analyses and tests of composite elements. It presents the procedure in detail and is thus valuable as a source of guidelines for future design activities. Validation of the model allows to confirm the solution correctness for the whole structure, as well as to verify the material data used, which is particularly important when using the data taken from the literature or provided by the material manufacturer. In addition, validation analyses can be a significant help in finding the differences between calculations and experimental results of the designed blade.

2. TEST OBJECTS AND LOADING CONDITIONS

Three types of objects in the form of beams made of materials used in the construction of the blade were subjected to numerical analyses and strength tests. Element 1 was a rectangular cross section composite beam made entirely of S2-Glass R-310 449 glass rowing and RenLam LY120 resin whose nominal dimensions are shown in Figure 1a. The direction of the reinforcement was in line with the beam axis. The construction of the element corresponded to the spar construction of the designed blade. The element's length and width were adjusted to the capabilities of the test fixture. Such test object was used to verify the applicability of the material data determined in specimen testing in the analysis of the structure with bending as the dominant component of the loading, as well as to validate the numerical calculations for a simple case.

The other two elements were created by combining the materials used in the construction of the blade. Element 2 was a beam with a specific sandwich structure. Skins were made of 12 layers of MTM44-1/EHTA40(6k) carbon-epoxy 2/2 twill fabric preimpregnate with reinforcement orientation [60°/15°/-15°/15°/-60°/-15°/-15°/-15°/-60°/15°/-15°/15°/60°]. The core consisted of a composite made of S2-Glass R-310 449 glass rowing and RenLam LY120 resin with reinforcement orientation coincident with the axis of the element. The covers were bonded to the core using AF 164-2K film adhesive. The layup corresponded to the fragment of the blade where the glass composite spar joins the casing. The nominal dimensions used in the model are shown in Figure 1b.

Element 3 had the form of a beam with a typical sandwich structure. The skins were made of three layers of the same fabric preimpregnate as element 2 with reinforcement orientation [0°/45°/0°] in relation to the beam axis. They were bonded to the ROHACELL 51WF foam core using AF 164-2K film adhesive. This was similar to a fragment of a composite blade in which the foam core is directly connected to the casing. A significantly reduced skin thickness, with respect to the actual design, was used to take into account the effect of a much less rigid core on the component's behavior. The thickness of the core is the maximum thickness used in a blade that can be made from a single foam sheet. The nominal dimensions of element 3 are shown in Figure 1c.

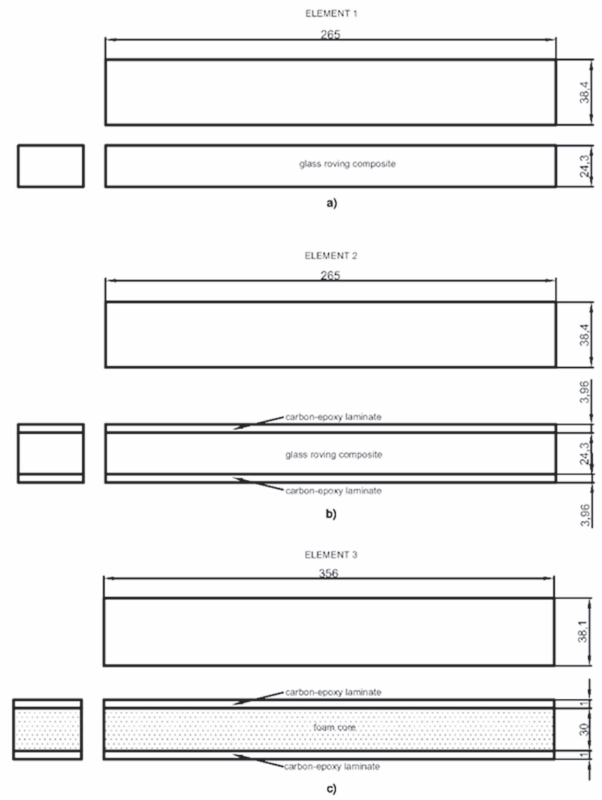


Figure 1. Dimensions of the examined objects: a – Element 1, b – Element 2, c – Element 3 [own elaboration, 2017]

The composite blade designed in the Institute of Aviation was made in Out Of Autoclave technology, therefore the manufacturing process used to produce the test objects was consistent with the manufacturing process specification for the blade.

The rotorcraft rotor blade is subjected to a complex state of loading. It can be divided into axial tension, bending in planes of higher and lower stiffness and torsion. For the purpose of the validation of numerical calculations, bending cases have been chosen. The axial loading is too easy a case to verify the material data and the model due to the negligible shear and the small interactions between the materials in the structure. On the other hand, the available testing equipment did not allow for imposing torsional loads.

For all objects, the four-point bending setup in accordance with ASTM D7249 [12] was used, schematically shown in Figure 2. The elements were loaded to a specified maximum force with a constant actuator speed of 0.5 mm/min and then unloaded. The test parameters for each element are summarized in Table 1.

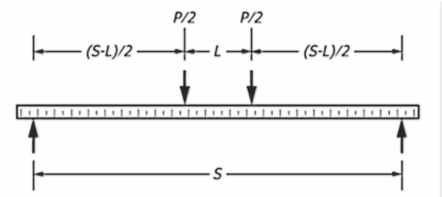
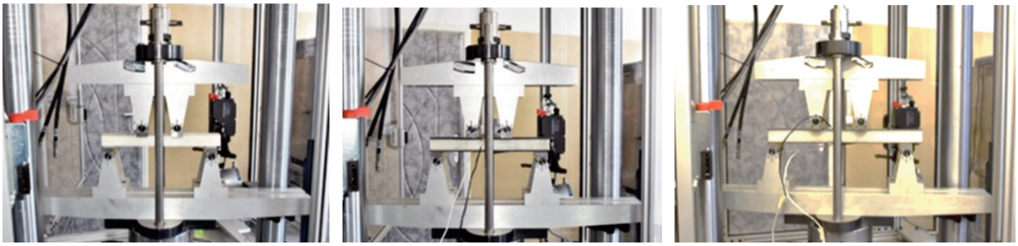


Figure 2. Load diagram of the test piece [1]

Table 1. Parameters of the conducted tests

Element	Lower support spacing, S, mm	Upper support spacing, L, mm	Maximum load force, P, N
1	228.6	76.2	15 000
2	304.8	101.6	800
3	228.6	76.2	15 000

In order to compare the results of the experiment with the results of numerical analyses, the loading force, displacement of the piston of the testing machine, deflection in the middle of the beam by means of a deflectometer and deformation in an axial direction on the lower and upper surface of the specimen by means of strain gauges were recorded. Test stands used in the experiments are shown in Figure 3.



a b c

Figure 3. Four-point bending: a – Element 1, b – Element 2, c – Element 3 [own elaboration, 2017]

3. NUMERICAL MODELS

The numerical models of the tested elements corresponded exactly to the numerical model used in the analysis of the complete rotor blade of the ILX-27 rotorcraft main rotor [13,14]. The analyses were carried out in the MSC.Nastran software, with the use of SOL 106 solver which accounts for large deformations.

The material data used in the analyses are presented in Table 2. For glass-epoxy and carbon-epoxy composites the material properties were determined experimentally in tests at room temperature [15÷17]. Carbon-epoxy laminate is used only for thin-walled structures, therefore the material properties related to the direction transverse to the surface were not determined. In the case of foam core, the material data provided by the supplier were assumed [18].

Table 2. Material data used in the numerical analyses

Composite made of S2-Glass roving (fibre volume 52 %)		Composite made of MTM44 preimpregnate (normalized data)	
Young's modulus along the fibers, E_1, MPa	46 340	Young's modulus in direction 1, E_1, MPa	51 948
Young's modulus across fibers, E_2 / E_3, MPa	12 267	Young's modulus in direction 2, E_2, MPa	51 297
Shear modulus, G_{12} / G_{13}, MPa	3 563	Shear modulus, G_{12}, MPa	3 615
Shear modulus, G_{23}, MPa	3 597	Poisson's ratio, ν_{12}.	0.06
Poisson's ratio, ν_{12}	0.30	ROHACELL foam core	
Poisson's ratio, ν_{13}	0.30	Young's modulus E, MPa	51 948
Poisson's ratio, ν_{23}	0.42	Shear modulus G, MPa	3 615
		Poisson's ratio, ν	0.06

The mechanical tests were reproduced in numerical analyses with the necessary simplifications, the same as in rotorcraft blade modeling. In order to reduce the complexity of the model and increase the efficiency of calculations, the film adhesive used in sandwich structure to bond the skins to the core was not modelled. The assumption was made of a rigid connection between the elements with possible modification of the material data after comparison of the results from the numerical model and experimental studies, compensating for the lack of the film adhesive [2, 9]. The dimensions of the samples and the support conditions corresponded to those used in the experiments.

The model consisted of two types of elements. Skins made of laminate, with a small thickness compared to other dimensions, were modeled using two-dimensional SHELL elements with the material properties of a layered composite. Composite made of glass roving and foam core were modeled using cubic SOLID type elements as orthotropic and isotropic material [19], respectively. The coordinate systems were oriented in such a way that the main direction (direction 0) coincides with the axis of the tested elements.

4. ANALYSIS OF THE RESULTS

Numerical calculations were carried out for all elements considered and then changes were introduced to numerical models in order to improve their correlation with the real objects. Material data were chosen as parameters to be changed because some of them were assumed based on literature data not supported by own mechanical testing. The value of the deflection at the midpoint of the beam was used as a measure of the coincidence between the test results and the calculations.

4.1. Analysis of element 1

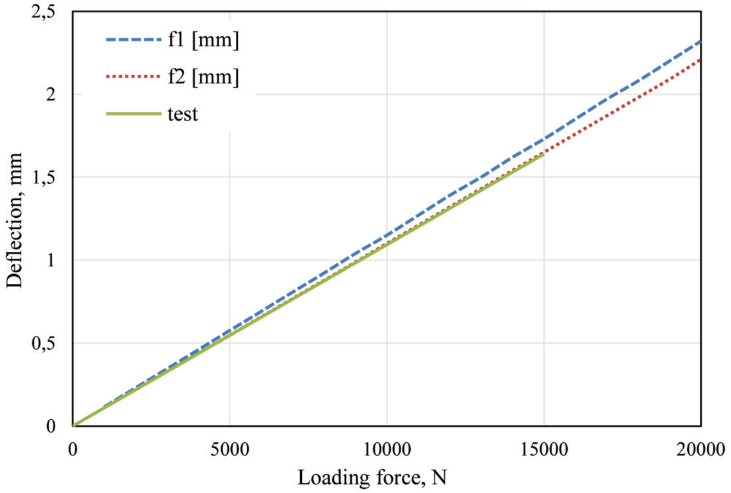


Figure 4. Specimen deflection as a function of applied force for ILX-27-E1 specimen [own elaboration, 2017]: f1 – FEM analysis using input material data; f2 – FEM analysis using modified material data; test - specimen deflection in a three-point bending test

The first one to be analyzed was the simplest of the objects under investigation – a beam made of glass rowing. For the input parameters, the difference in deflection at the center of the beam for the maximum force of 15,000 N was 0.09 mm, which accounted for 5.5% of the total deflection. The discrete model was less rigid. In order to reduce the discrepancy between the calculations and the experimental study, Young's modulus was modified and introduced into the discrete model. The initial value was taken from the strength tests, but the stiffness of the composite sample depends on the fiber volume content. This was not known for the tested specimens as well as for the composite from which element 1 was cut, so Young's modulus could differ. Additionally, the element was bent and the applied stiffness was determined in the tensile test. In order to correlate the results, the modulus of elasticity of the composite was increased from 46 340 MPa to 49 000 MPa. This value brought the difference in deflection to 0.01 mm, i.e. 0.61% of the total deflection. Figure 4 shows a comparison of the applied force vs. deflection curves obtained during the experimental test and the calculation.

4.2. Analysis of element 2

In the next step, element 2 was analyzed due to the combination of two materials in its structure – carbon-epoxy laminate and glass roving composite. The latter has already been checked and modified, so in this case the MTM 44 material was under investigation. The material data for this material was determined in an experimental way. The numerical calculations showed that the discrete model is more rigid than the actual object. For a loading force of 15 000 N, the numerically calculated deflection was 6.84 % lower than measured during the experiment. For this type of element, the above result was considered acceptable. Figure 5 shows a comparison of the load vs. deflection curves obtained during the experimental test and from the calculations. The difference observed is caused by the representation of the real object in the numerical calculations. In the tested specimens the layers of different materials were bonded with a film adhesive, while in a discrete model it was modelled as a rigid connection. This is accepted accuracy in a complete blade model taking into account the design efficiency, therefore no modifications were made to the model structure or to the material properties used.

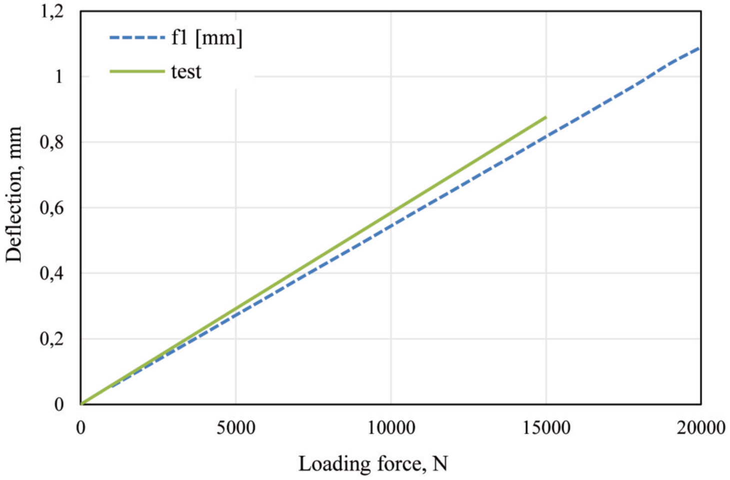


Figure 5. Specimen deflection as a function of applied force for ILX-27-E2 specimen [own elaboration, 2017]: f1 – FEM analysis using input material data; test - specimen deflection in a three-point bending test

4.3. Analysis of element 3

The last analyzed object was element 3 in the form of a sandwich beam, which consisted of carbon-epoxy laminate skins and a foam core. Numerical calculations for the initial parameters gave, for the force of 800 N, a deflection higher by 23.7% than measured experimentally. The material properties of the laminate made of MTM 44 preimpregnate were verified in previous analyses, while of the data on Rohacell 51WF came entirely from the material datasheet issued by the manufacturer. Due to large discrepancies between calculations and the experiment, both Young's modulus and shear modulus, were changed, from 75 MPa to 130 MPa and from 24 MPa to 30 MPa respectively. The analysis carried out for the new parameters resulted in a 2.8 % difference in the deflection with a lower stiffness of the discrete model, which was considered an acceptable result. A comparison of the load vs. deflection curves obtained during the experimental test and the calculations for the two sets of parameters is shown in Figure 6.

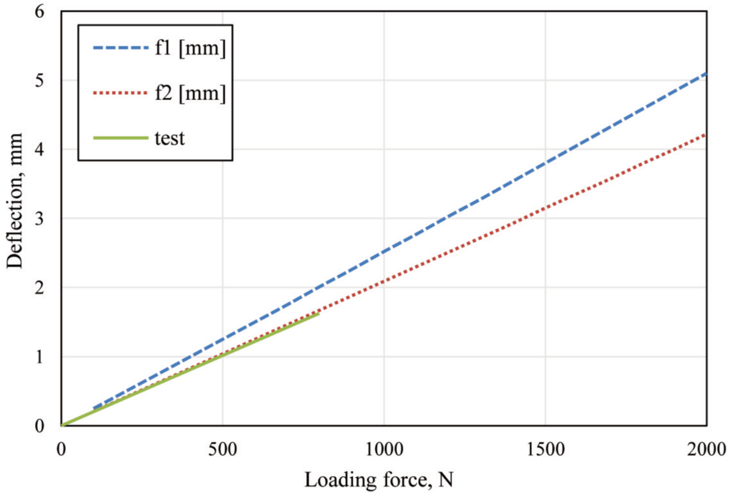


Figure 6. Specimen deflection as a function of applied force for ILX-27-E3 specimen [own elaboration, 2017]: f1 – FEM analysis using input material data; f2 – FEM analysis using modified material data; test - specimen deflection in a three-point bending test

4.4. Modifications of material properties

The comparison of the results of numerical calculations and experimental tests has led to modifications of the properties of some materials used to manufacture the test objects. The deflection at the specimen center and the strain in the direction of the specimen axis was selected for final comparison of the experimental tests and the modified numerical analyses The results are shown in Tables 4 and 5.

Table 3. Modifications of the material properties of the roving composite and the foam core

Material	Modified property	Initial value	Final value	Percent change
Glass Roving Composite S2-Glass	Young's modulus, MPa	46 350	49 000	5.7 %
Rohacell® foam core	Young's modulus, MPa	75	130	73.3 %
	Shear modulus, MPa	24	30	25.0 %

Table 4. Comparison of specimen deflection in the experiment and FEM analysis

Element	Force, N	Experiment	FEM	Difference in deflection	
		Deflection, mm	Deflection, mm	Absolute, mm	Relative, %
Element 1	15000	1.640	1.65	0.010	0.61
Element 2	800	1.625	1.67	0.045	2.77
Element 3	15000	0.877	0.817	-0.060	-6.84

Table 5. Comparison of the upper and lower skin strain in the experiment and FEM analysis

Element	Force, N	Experiment		FEM		Difference			
		Upper skin	Lower skin	Upper skin	Lower skin	Upper skin		Lower skin	
		με	με	με	με	με	%	με	%
Element 1	15000	2 942	3 048	3 220	3 380	-278	-9.45	-332	-10.89
Element 2	800	739	696	807	697	-68	-9.20	-1	-0.14
Element 3	15000	1 801	1 874	1 610	1 640	191	10.61	234	12.49

5. CONCLUSIONS

The conducted work has led to the following conclusions:

1. Deflection obtained from the analyses with the use of initial material data were 5.5 % to 23.7 % higher than those resulting from experimental studies – the calculated stiffness of the beam models was lower than the stiffness of real objects by the same amount.
2. Young's modulus for the composite made of glass rowing was modified by 5.7 % from 46 340 MPa to 49 000 MPa, which decreased the difference in deflection to 0.61 %.
3. The material data for the carbon-epoxy laminate made of prepreg MTM44-1/EHTA40(6k) were determined correctly, which was confirmed by the consistency of the calculations with the experiment amounting to 7 %.
4. The Rohacell foam core material data provided by the manufacturer did not correspond to the actual material properties. The Young modulus shall be increased by 73.3 % from 75 MPa to 120 MPa and the Kirchhoff module by 25.0 % from 24 MPa to 30 MPa.
5. The validity of the discrete models and proper representation of the tested objects have been confirmed. The differences in the deflection obtained with calculations and with experimental tests were 0.6 %, 2.8 % and 6.8 % for elements 1÷3, respectively. The maximum difference in the measured deformations was 12.5 %.
6. For strength analyses of the ILX-27 design blade it is recommended to adopt the material data after the modifications presented.

BIBLIOGRAPHY

[1] Szablewski K., Jancelewicz B., Łucjanek W., 1995, Introduction to rotorcraft design, Wydawnictwa Komunikacji i Łączności, Warszawa, (in Polish).

[2] Kiraly R., Head R.E., 1983, Manufacturing Methods and Technology (MANTECH) Program: Manufacturing Techniques for Composite Main Rotor Blade for the Advanced Attack Helicopter, United States Army Aviation Research and Development Command, St. Louis, MO.

[3] Composite Materials Handbook – 17, ed., 2012, Composite Materials Handbook, Volume 3 – Polymer Matrix Composites – Materials Usage, Design, and Analysis, Chapter 4, SAE International.

[4] Feraboli P., 2009, Composite Materials Strength Determination Within the Current Certification Methodology for Aircraft Structures, Journal of Aircraft, Vol. 46, No. 4.

[5] Fawcett, A., Trostle, J., and Ward, S., 1977, 777 Empennage Certification Approach, 11th International Conference for Composite Materials, Melbourne, Australia.

[6] Hodges D. H., 1990, Review of composite rotor blade modeling, AIAA Journal, 28 (3).

[7] Bull J.W., Ed., 1996, Numerical Analysis and Modelling of Composite Materials, Springer Netherlands, Chapter 1: Analysis of composite rotor blades.

[8] Konik R, 2016, Numerical analysis of composite tail rotor blade, Transactions of the Institute of Aviation, No. 3(244), pp. 175-188, (in Polish).

[9] Awrejcewicz j., Ed., 2011, Numerical Analysis – Theory and Application, InTech, Chapter 8: Numerical Validation Methods.

[10] Pyrzowski Ł., Sobczyk B., Witkowski W., Chróścielewski J, 2016, Three-point bending test of sandwich beams supporting the GFRP footbridge design process - validation, 3rd Polish Congress of Mechanics (PCM) / 21st International Conference on Computer Methods in Mechanics (CMM), Taylor & Francis Group.

[11] Sears A. T., 1999, Experimental validation of finite element techniques for buckling and postbuckling of composite sandwich shells, MSc. Thesis, Montana State University, Bozeman.

[12] ASTM International, 2016, Standard Test Method for Facing Properties of Sandwich Constructions by Long Beam Flexure, ASTM D7249/D7249M – 16.

[13] Kaddour A.S, Hinton M.J., 2012, Input data for for test cases used in benchmarking triaxial failure theories of composites, Journal of Composite Materials, No. 19-20/vol. 46; 2295-2312.

[14] Allen H.G., 1969, Analysis and Design of Structural Sandwich Panels, Pergamon Press, Oxford, UK.

[15] Zalewska M., 2017, Allowable values for the analysis of the spar of the main rotor blade for ILX rotorcraft: Roving AGY S2 Glass R-310 449, TEX 675, resin Araldite LY 1564/Aradur 2954, 4S/LK/2016 (in Polish), Institute of Aviation, Warsaw.

[16] Karny M., 2016, Mechanical testing of tensile strength in 90° direction of the specimens made of glass roving in RTA conditions accoridng to ASTM D3039, 63/LK/2016, Institute of Aviation, Warsaw, (in Polish).

[17] Zalewska M., 2016, Allowable values for the analysis of the casing of the main rotor blade for ILX rotorcraft for material MTM44-1/EHTA40(6k)-2x2T-284-40%, 5S/LK/2016, Institute of Aviation, Warsaw, (in Polish).

[18] Evonik Industries, 2017, ROHACELL® WF Technical Information.

[19] Quick Reference Guide: MSC Nastran, 2013.

WALIDACJA MODELI NUMERYCZNYCH UŻYTYCH DO PROJEKTOWANIA KOMPOZYTOWEJ ŁOPATY DO ŚMIGŁOWCA ILX-27

Streszczenie

W artykule zaprezentowano procedurę walidacji modelu wykorzystywanego w analizie łopaty kompozytowej do wirnika nośnego śmigłowca ILX-27, projektowanej i wytwarzanej w Instytucie Lotnictwa, za pomocą analiz numerycznych oraz badań elementów kompozytowych. Przeprowadzono analizę numeryczną metodą elementów skończonych oraz badania eksperymentalne trzech obiektów badawczych wykonanych z podsta-wowych materiałów składających się na strukturę łopaty – laminatu węglowo-epoksydowego, kompozytu szklano-epoksydowego wykonanego z rowingu oraz wypełniacza piankowego. Elementy miały postać belek zginanych czteropunktowo, a do porównania wyników wybrano wartości strzałki ugięcia na środku belki oraz odkształceń osiowych na powierzchniach górnej i dolnej. Przeprowadzona procedura pozwoliła na dopasowanie modelu dyskretnego do rzeczywistych obiektów oraz weryfikację i korekcję danych materiałowych użytych w analizie wytrzymałościowej projektowanej łopaty.

Słowa kluczowe: kompozyty, modelowanie MES, badania mechaniczne.