Experimentální Analýza Napětí 2011
Experimental Stress Analysis

Investigation of response of composite plate subjected to low-velocity impact

Tomáš Mandys, Tomáš Kroupa, Vladislav Laš, Robert Zemčík & Jan Bartošek[1]

Abstract: This paper deals with numerical simulations of an impact event on unidirectional longfiber composite plate. Tensile and bending tests are used for determination of material properties. Simulations are performed and executed in Ansys LS-Dyna solver. Implicit static and explicit dynamic solvers are used for investigation of composite plate response by different loadings.

Keywords: Drop tester, Impact, LS-Dyna, UD Composite

1. Introduction

The composite materials are widely used in the industry and become the preferable alternative to conventional materials. The main advantages of composite materials are high stiffness to weight ratio, corrosion resistance and mainly product design variability. The disadvantage of composite materials is predisposition to failure that cannot be visible by naked eye. In practice the failure can be detected using X-ray or CT-scan but it is very time-consuming and expensive. The failure or damage is irreversible and it causes the stiffness and strength reduction of the composite material. Therefore, it is very necessary to consider this fact in industrial applications and ensure the safety of the composite construction in case of all expected loads. It is also important to consider the random impact loads which can cause the damage too.

This paper is focused on validation of material properties of composite material for crash simulations performed in LS-Dyna. The material properties are usually obtained from tensile tests. The obtained properties can be partially different depending on test specimen size. Therefore, a lot of tests must be performed and obtained parameters must be statistically evaluated. The material properties may change in dependency on the production (for example by changing of technological process or material base supplier).

The material properties were verified for composite material made from prepreg EHKF420-UD24K-40 with Toray T600SC carbon fibers and epoxy resin. This unidirectional long fiber composite material is produced by LA-Composite Company in Prague.

2. Material model

The finite element model of the layers used within all analyses described below was transversely isotropic material model [3]

[1] *Ing. Tomáš Mandys; Ing. Tomáš Kroupa, PhD.; Prof. Ing. Vladislav Laš, CSc.; Ing. Robert Zemčík, PhD.; Ing. Jan Bartošek; Department of Mechanics, Faculty of pplied Sciences, University of West Bohemia; Univerzitní 22, 306 14 Pilsen, Czech Republic; tmandys@kme.zcu.cz*

$$\begin{bmatrix} \varepsilon_L \\ \varepsilon_T \\ \gamma_{LT} \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} & 0 \\ S_{12} & S_{22} & 0 \\ 0 & 0 & S_{66} \end{bmatrix} \cdot \begin{bmatrix} \sigma_L \\ \sigma_T \\ \tau_{LT} \end{bmatrix}, \tag{1}$$

where $S_{11} = \frac{1}{E_L}$, $S_{22} = \frac{1}{E_T}$, $S_{12} = \frac{-\nu_{LT}}{E_L}$ and $S_{66} = \frac{1}{G_{LT}}$.

Four layered thin shell elements were used for analyses. In the cases when the specimens were supported by metal parts, the analysis was considered as a contact analysis.

3. Experiments and numerical simulations

3.1. Static tensile test of composite beam

Specimens made of four layers of longfiber unidirectional carbon-epoxy composite material were subjected to tensile test. Dimensions of specimens were 270 mm × 15 mm and thickness 1.15 mm. The velocity of the loading was 10 mm·s^{-1}. Fig. 1 and 2 show the force-displacement dependence for specimens which fiber direction forms angle 0° and 90° with direction of loading force. Figures show results which was obtained experimentally and by means of numerical simulation. The simulation was performed as a quasi-static analysis considering the finite strains theory. These tests were used for determination of Young's moduli in longitudinal (*L* – direction parallel to the fibers) and transverse (*T* – direction perpendicular to the fibers in the composite plane) directions. The letter elasticity parameters were obtained from manufacturer. The material properties are summarized in Table 1.

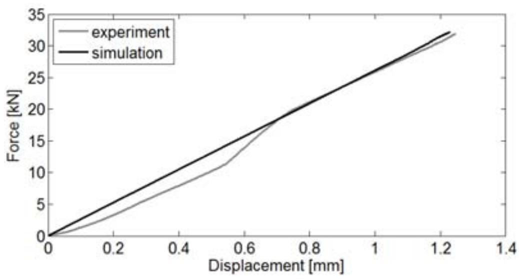

Fig. 1. Force-displacement diagram for specimen with 0° fiber angle.

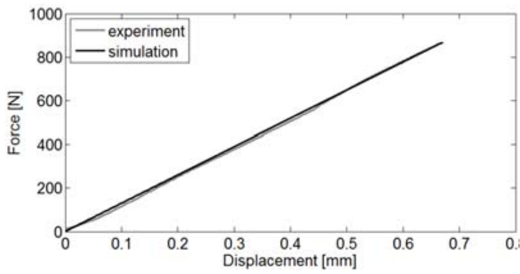

Fig. 2. Force-displacement diagram for specimen with 90° fiber angle.

Table 1. Elastic constants and strengths of composite material

E_L [GPa]	E_T [GPa]	G_{LT} [GPa]	ν_{LT} [-]	ρ $[kg \cdot m^{-3}]$
153.4	7.8	4.5	0.28	1510

3.2. Static bending test of composite beam

Three point bending was performed in order to validate data obtained from manufacturer and from static tensile tests. Used specimens were made of the same material mentioned in previous paragraph. Dimensions of specimens were 270 mm × 30 mm with the same thickness as the specimens mentioned above. The fiber direction was parallel to supports shown in the Fig. 3. Distance and radius of the supports was 150 mm and 10 mm, respectively. The velocity of the loading was 10 mm·s^{-1} again. Fig. 4 shows the dependences of the loading force on the deflection of the beam, which were obtained from experiment and numerical simulation.

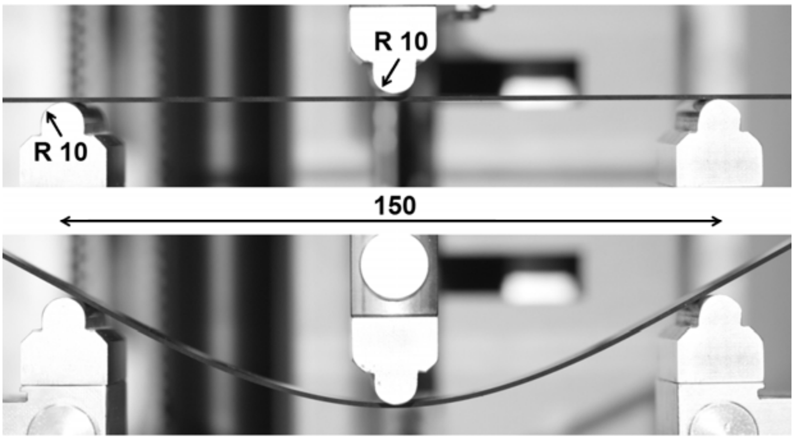

Fig. 3. The way of loading specimens and geometry of supports for three point bending

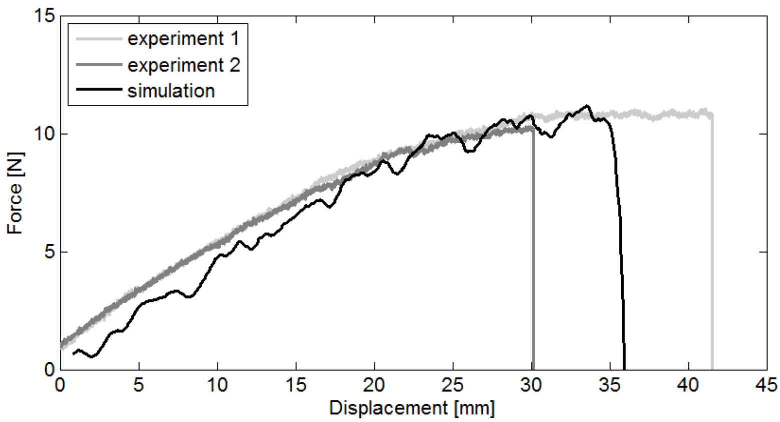

Fig. 4. Force-deflection diagram for three point bending

3.3. Plate static bending

Validation of material properties of unidirectional long fiber composite was performed for a larger specimen, namely a plate with the size of 270 mm × 270 mm and thickness was 1.15 mm. The composite plate was simply supported along two opposite edges on a steel stand that was developed for fixing and laying the plates. The edges of the plate overlapped by 10 mm – see Fig. 5. The testing device enables to set the starting height of the impactor and the impact place directly via movable vertical and horizontal linear guides. The impactor is accelerated only by gravity. The impactor is equipped by the accelerometer that enables recording the time-force response (contact force) between the impactor and the tested body. Laser sensor was used for the measurement of the deflection in four selected locations (see Fig. 5). Fig. 6 shows the drop tester with the measure laser sensor. The impact points were always in the middle of the plate.

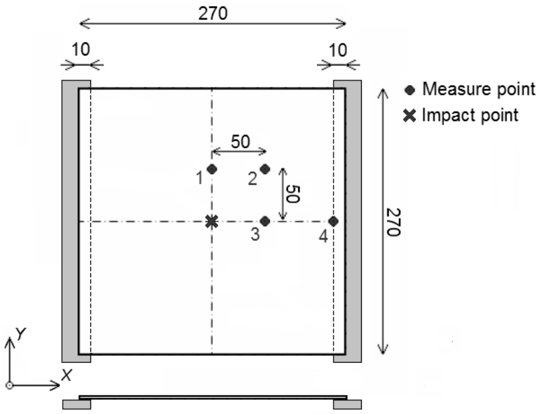

Fig. 5. Steel stand to enabling laying the composite plate with the measure points

Fig. 6. Drop tester with the laser sensor and the steel stand

Firstly, static deflection of composite plate loaded by the weight of the impactor was measured. The total mass of impactor was 0.217 kg and the deflections were measured in points 2 and 3 (see Fig. 5). The deflections were identified for two cases of the fiber direction of the plate in x (90°) and y (0°). Table 2 shows the confrontation of static response of the plate between experimental and simulation results. Finite element model consisted of composite plate with 54 × 54 shell elements and again four integration points through the thickness of the plate. The numerical simulation was solved as static analysis. The static load was simulated via the point force.

Table 2. Static deflections of the composite plate

Fiber angle [°]	Measurement point	Experiment [mm]			Exp. average [mm]	Simulation [mm]	Difference [%]
0	2	1.888	1.914	1.893	1.898	2.075	9.3
	3	2.071	1.999	2.140	2.070	2.140	3.4
90	2	0.321	0.323	0.320	0.321	0.332	3.4
	3	0.488	0.480	0.486	0.485	0.471	2.9

3.4. Dynamical response of composite plate

Using of the drop tester eliminates most of the possible inaccuracies, which can occur if balls or steel tubes for deducing the impact load are used [5].

The composite plate response for the dynamic load was detected by transverse low velocity impact. The impact velocity of the impactor was 0.56 $m \cdot s^{-1}$. The fiber direction is perpendicular to the supports. The time-deflection dependence in particular points (see Fig. 5) and the time-impact force dependence in impact point was recorded.

The explicit solver was chosen for simulation. The solver is based on central differences scheme for time integration (calculated stable timestep was 0.353 μs). The finite element model was created as a full contact problem. Impactor made a several contacts with the plate during one experiment. Because of complex shape of impactor only the part consisting from the accelerometer was meshed using solid elements. The total mass of the impactor was then added to this part. Figs. 7-10 show the comparison of deflections in particular points between experiment and numerical simulation. The results are compared for the range 0 - 40 ms from the start of impact event. The comparison of the contact forces can be seen in Fig. 11.

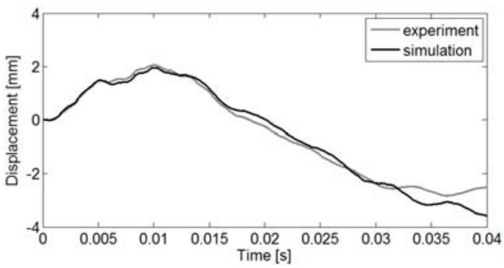

Fig. 7. Deflection comparison for the measurement point 1

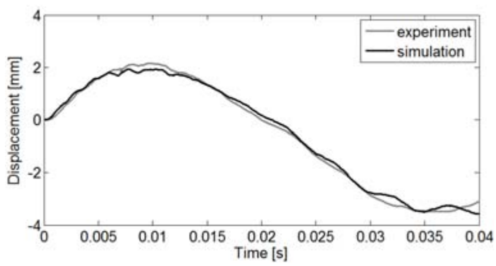

Fig. 8. Deflection comparison for the measurement point 3

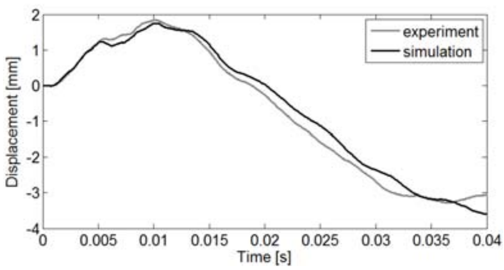

Fig. 9. Deflection comparison for the measurement point 2

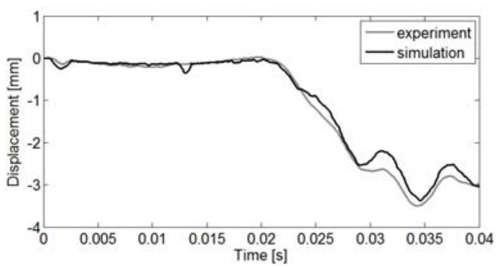

Fig. 10. Deflection comparison for the measurement point 4

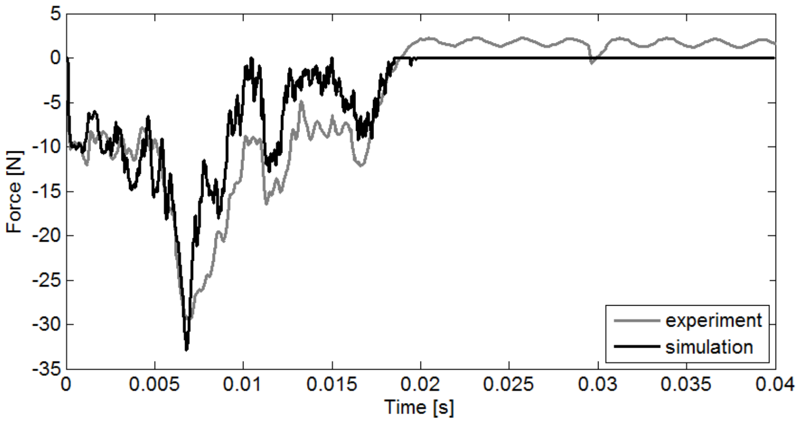

Fig. 11. Contact force comparison in the impact point

4. Conclusion

The aim of this paper was to evaluate the response of unidirectional longfiber composite material. The validation was realized via tensile test, three point bending test and transverse low velocity impact. The numerical simulations were solved in the finite element program LS-Dyna, where the static, quasi-static and dynamic analyses were performed using implicit and explicit solvers. Results from this paper will be used for more complex analyses such as low velocity impact on composite structures with progressive failure or identification of impact location.

Acknowledgement

This work has been supported by the research project of GAČR GD101/08/H068 and GAP101/11/0288 and the research project of the Ministry of Education of Czech Republic no. SGS-2010-046.

References

[1] Hallquist J.: *LS-Dyna Theory Manual*. Livermore Software Technology Corporation, March 2006.

[2] Kroupa T., *Damage of composites subjected to impact*. Doctoral thesis, University of West Bohemia in Pilsen 2006.

[3] Laš V., *Mechanika kompozitních materiálů*, University of West Bohemia, Plzeň 2008.

[4] *LS-Dyna Keyword User's Manual, Version 971*. Livermore Software Technology Corporation, 2007.

[5] Mandys T., *Numerická simulace rázu tělesa na kompozitovou desku*, Diploma work, University of West Bohemia, Plzeň 2010.