COMPUTATIONAL MECHANICS 2019
35th conference with international participation
Srní
November 4 - 6, 2019

Finite element analysis of composite tubes with integrated loop connections

A. Malá[a], V. Kulíšek[b], N. Schmidová[a], T. Zámečníková[a], T. Ponížil[b], T. Mareš[a]

[a] *Faculty of Mechanical Engineering, Czech Technical University in Prague, Technická 4, 160 07 Prague 6, Czech Republic*
[b] *Compo Tech PLUS, spol. s r.o., Nová 1316, 342 01 Sušice, Czech Republic*

The paper deals with the modelling of integrated loop connections of composite tubes by finite element (FE) analysis. Composite beams have been employed in many structural applications during the last decades as the composite materials benefit in their light-weight properties with a great strength and stiffness. However, their application is sometimes limited due to difficult designing of connection interfaces, which would enable to transfer loads into the composites. The currently used connection interfaces of composite tubes (bonded joints, mechanical joints) are demanding in terms of manufacturing and often offer only limited load transfer in comparison with the strength of the composite tube. That is the reason for development of a less problematic connection, where the stress transfer will be continuous and also the connection will be sufficiently strength and stiff. Therefore, a project dealing with connections of tubes has been started with aim to develop integrated loop connections, which are made directly from the composite tube endings. One of the goals of the project is to build a reliable numerical model of the tube and its connection interface with aim to predict the joint strength and to obtain a tool for the joint design.

Coupons with integrated loops were manufactured by fibre winding and placement technology. Simultaneously, their FE models were created. The first type of the investigated coupon was designed, manufactured and tested (see Fig. 1 below left). The basic test of the coupon was in loading by the tension force. In the FE model, the loading of the tube by tensional force was applied by a steel cylinder inserted into a pair of the loops on one side of the coupon. In the second ending, an identical cylinder was inserted as the fixed support (see Fig. 1).

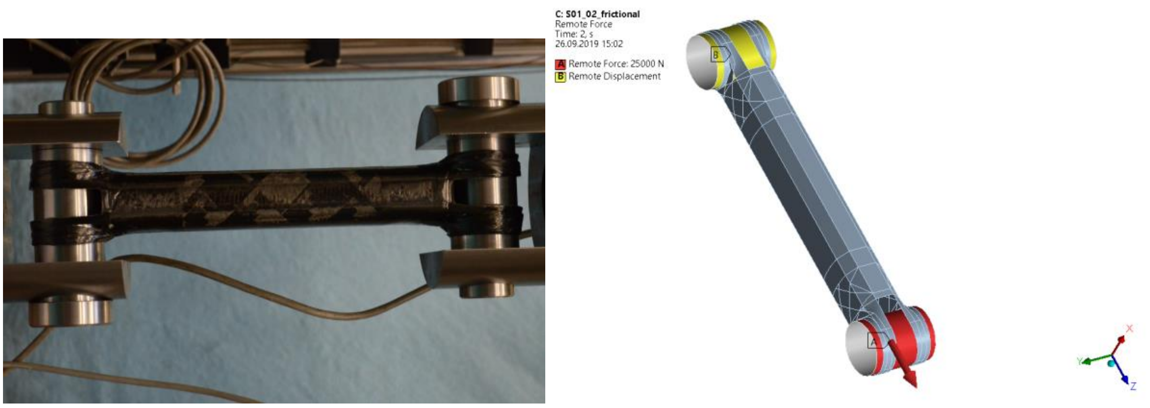

Fig. 1. Composite coupon with integrated loops and its model

The FE model of coupon was created to develop the connection interface and the tube lay-up with the required mechanical properties of tube near the integrated loop joints The FE model was built up in ANSYS version 19.2 and the definition of laminate was set up using the ANSYS Composite Prep/Post (ACP) package. During preparation of the model, it was important to specify the lay-up stacking sequence and connections between tube and cylinders in the best accordance with reality. The basic stacking sequence and material properties were provided by the coupon supplier. The aim was to build the model as close to reality as possible, to test the model response and compare it with the experimental results. And if the predicted strength would match the experimental results, to optimize the composite lay-up of the tube and integrated loops to maximize the coupon strength and stiffness.

The specification of loops layer thicknesses was problematic, because in the real coupons, the fibre layers tend to change thickness together with their width, while keeping the constant fibre amount in the layer. To deal with this problem, the specification of layers thicknesses for the most critical parts of the model was made using linear functions, because there was a linear dependence between the thickness and the width of layer.

The basic FE model was made from shell elements; to input all the layers, the model was divided into sections and all the layers of fibre tows were input as close as they were made in the reality (see Fig. 2 for the demonstration of one fibre tow loop specification in the model).

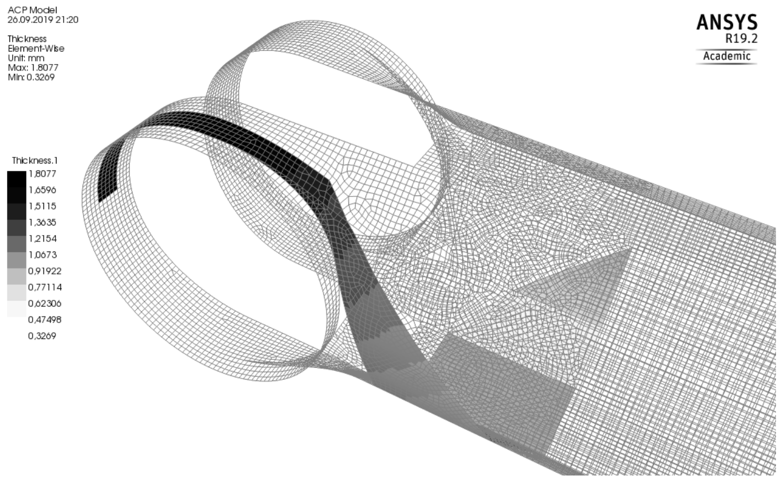

Fig. 2. Specification of 1 layer in the tube and loop

An experimental testing was carried out performing tensile tests of the coupons and evaluating the force-displacement behaviour of the coupon together with a digital image correlation in the critical parts of the coupons. The similar type of loading was performed using the FE model with aim to perform a comparison between the experimental and numerical results. Damage has been represented by the Inverse Reverse Factor (IRF) in the numerical model and by the Digital image correlation results from the experimental tests. The strength failure in the FEA was presented by three failure criteria: Tsai-Wu, Tsai-Hill and Maximum Stress. All three criteria were used to evaluate the failure parameter of the model given the full lay-up and nominal strength values of the composite layers.

The critical places of the construction were found by the FEA. The first deformations are initialized on the roving which sets up the loops. Next deformations are initialized on the square, placed near to the loops. Subsequently the critical areas are getting bigger with the increasing force value. The analysis is linear, which is the reason why the deformed places still retains the same mechanical properties (a progressive damage model has not been used) even after the ply-failure is detected. Damage of the composite coupon, interpreted by a failure index, during the increasing loading is shown in Fig. 3.

In the experiment, the first cracking was visible in the tube body and this was captured with FE precisely. This is corresponding to the tube damage, which is visible for load 0.4*Fmax. The developed FE model is consistent with the experimental testing in finding of the initial failure. However, the final strength from the FE prediction was much lower than the final strength from the experimental specimen. There, the maximal failure from experiment exceed the predicted critical load almost five times, which clearly show the limitation of the first-ply failure criteria for the final strength prediction.

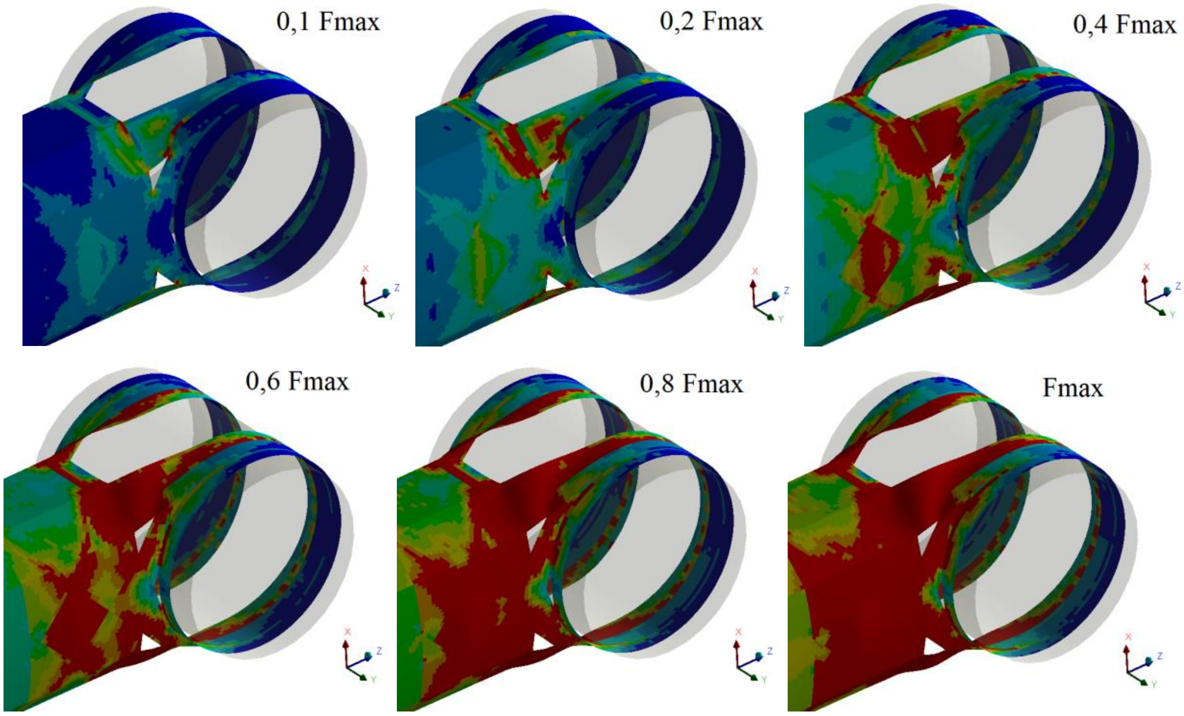

Fig. 3. Damage spreading (rupture in red)

The critical places of design were found by the proposed FE model and modelling approach. The current model is able to provide a tool in improving the tube strength and increasing the initial failure, as the body failure happened in the first coupons at load five times smaller than the critical one. On the other hand, failure criteria must be developed to predict the critical load in the fibre tow loops, which will be aim of the future work.

Acknowledgements

This work has been supported by project No. SGS18/175/OHK2/3T/12 of the Grant Agency of the Czech Technical University in Prague and by project No. TJ02000252 of the Technology Agency of the Czech Republic. Support of the Compo Tech PLUS company in development, manufacturing and testing of composite parts is also greatly acknowledged.

References

[1] ANSYS 19.2 Help.
[2] Quintelier, J., Samyn, P., de Baets, P., Ost, W., van Paepegem, W, Wear of steel against carbon fibre reinforced PPS, Tribology in Industry 27 (2005) 29-35.
[3] Yi, X., et al., The effective friction coefficient of a laminate composite, and analysis of pin-loaded plates, Journal of Composite Materials 34 (1) (2000) 69-87.