Optimization of composite airframe rib using tailoring blank technology

J. Šedek [1,a]

[1] *VZLÚ – Aerospace Research and Test Establishment, Czech Republic*
[a] *sedek@vzlu.cz*

Abstract: The contribution deals with work concerning the design and optimization of composite rib intended for use in airliner door construction. New progressive tailoring blank technology of manufacturing thermoplastic composites requires specific demands on rib design. Replacing the metal machined rib by the composite one brings advantages of lower weight and effectively utilized material with minimal waste. The optimization process involved cut-out design and lamina placement. Numerical tools were utilized using finite element software ABAQUS and Hashin and LaRC failure theories were incorporated. Based on the optimization, the real part was manufactured and the static strength was evaluated. The experimental deformation and strain field measured by non-contact optical measuring system (DIC) was in good accordance with numerical analysis. Finally, the test of static strength was performed and strength according to strength theories was evaluated.

Keywords: FEM; optimization; rib; thermoplastic composite; tailored blank.

1 Introduction

Continuous fibre-reinforced thermoplastics belong to the group of high-performance composites and are increasingly utilized in design of aerospace structures. Due to their properties, thermoplastic composites are able to meet specific aerospace requirements. Thermoplastics can be manufactured by thermoforming, which is in some aspects similar to the process utilized for non-reinforced plastics. The thermo-forming process is normally conducted at a processing temperature that is higher than the melting temperature of the thermoplastic resin to allow the textile prepreg to deform. The composite part can vary in thickness, fibre orientation, material composition and shape. It can be achieved through the stacking arrangement of the tailored prepreg blanks. However, the thermoforming process of a part with variable thickness is not elementary and takes advanced demands on moulds construction and process timing, especially during heating before co-consolidation. In the work package in the project “Research and development of modern technology processes for new application of high-tech reinforced thermoplastic” supported by Technology Agency of the Czech Republic, the composite demonstrator representing the rib of the aircraft door structure was designed and manufactured.

2 Composite rib design and optimization

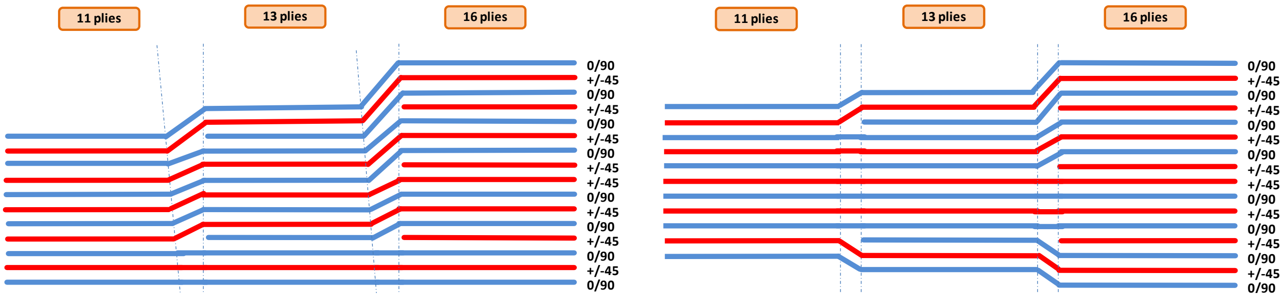
The rib is located opposite the console of the door lock and transfers the load resulting from the cabin pressure acting on the door to the fuselage construction. The rib is reinforced at the lower flange by fuselage skin and fixed with auxiliary and edge frame at vertical margins. Basic dimensions of the rib are determined by original construction; almost triangular shape with the height of 153 mm and the length of 333 mm. The upper and lower parts contain reinforcing flange and the lower part is in construction joined with the skin. The rib is loaded in shear through the longer vertical edge, where the moment of load is applied. The moment of load results from the stop fitting connected to the longer vertical edge and loaded by 36 kN. The upper flange transfers tension stress and the lower flange transfers pressure stress.

2.1 Lamina stacking pattern

Prepregs are stacked in the rib construction in a sequence with varying weave orientation and different blanks shape, which result in variable rib thickness. The location of the thickness change is the point of eventual difficulties in manufacturing process and interlaminar issues in service. In this point the prepreg stacking can be realized in different ways.

The simplest method to ensure a joint of adjacent layers is to add the reinforcing layers of lamina directly on the base plies and thermoform them together. By this process a simple bond joint can be formed. All plies remain straight but the interlaminar stresses in the bond joint can cause peeling of reinforcing plies. The significance of the bond joint shape made by thermoforming technology was investigated in recent work [1]. Continuous change of the thickness in tapered bond joint is preferable due to its lower interlaminar stresses in contrast to the simple lap bond joint with common end. It is also preferable that the material carries the load in entire joint when tapered.

Preferable way to realize connection of continuous and ending prepreg layers is the implementation of ending ply between continuous plies. Tests of simple and spliced joints show resistance of spliced joint against interlaminar damage under static and cyclic loading. The ending plies are covered by continuous plies which act against the increase of peel stress and prevent delamination. The construction of splice joints can also be realized in different manners. The thickness can vary in steps with straight base ply or the thickness can vary from the center plane of the laminate as shown in Fig. 1. The stacking according to the straight base ply is useful when designing the mold because one part of the negative mold can be constructed as flat basis.



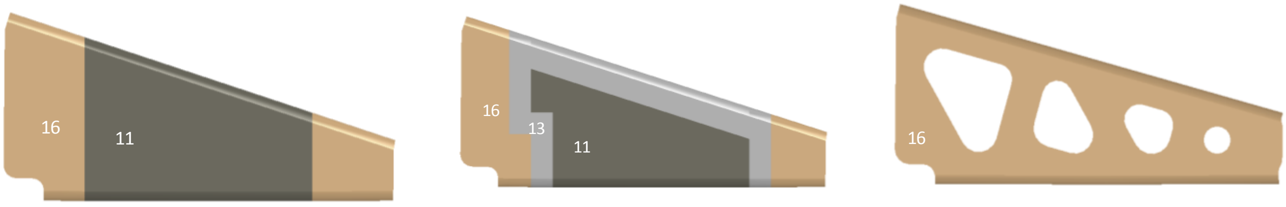
a) straight base ply b) straight plies near the center plane

Fig. 1: Spliced joint of laminate

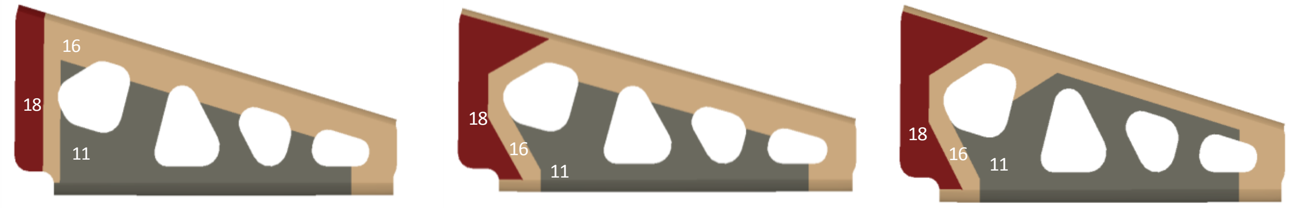
Four possible arrangements were specified a priori to optimization with following thicknesses: 18 plies (5,58 mm), 16 plies (4,96 mm), 13 plies (4,03 mm) and 11 plies (3,41 mm). Plies consist of carbon fabric reinforced thermoplastic prepreg with mass per unit area of 285 g/m^2 and matrix weight content of 43 %. The carbon fabric has satin weave 5HS and the matrix is polyphenylene sulphide (PPS). Stacking sequence of the demonstrator rib was realized according to the concept of straight base ply shown in Fig. 1a). The 18 plies arrangement was made in the fixture area at the higher vertical margin.

2.2 Prepreg blank tailoring

Tailoring of the prepreg blanks was conducted in order to decrease the weight and optimize load transfer through embedding cut-outs and through design of the prepregs shape. Series of finite-element (FE) analyses were carried out in order to optimize the shape and position of prepreg blanks and cut-outs. Some optimization steps are shown in Fig. 2. In basic design, the reinforcement of attaching section was applied and then, the effect of thickness variation was studied. In following steps, the cut-out effect was analysed and in pre-final steps the additive reinforcement of the margin was made where the moment of load is applied. Finally, the reinforcement was expanded up to the upper and lower flange and little correction of the thinnest area was applied. The cut-outs were designed in accordance with fibres direction in the prepreg so that the continuity of fibres was kept. It is contrast to cut-outs in metal webs, which are almost always designed circle shaped.



a) basic design b) prepreg blanks variation c) cut-outs



e) reinforcement of attach section f) expansion of attach section g) final design

Fig. 2: Steps of optimization design; numbers in figures indicate the number of plies.

2.3 Details of finite element model and evaluation

Finite element analyses were carried out in ABAQUS 6.13.2 software. Different lamina sections were specified according to stacking sequence. Nominal thickness of the lamina is 0,31 mm and the material constants as the elastic moduli and the strengths are specified in Tab 1. The composite was represented by shell elements of S4R a S3R type with characteristic element size of 5 mm (see Fig. 3a). Lamina stacking was prescriebed according to Fig. 1a) so only details of thickness change was not modeled properly.

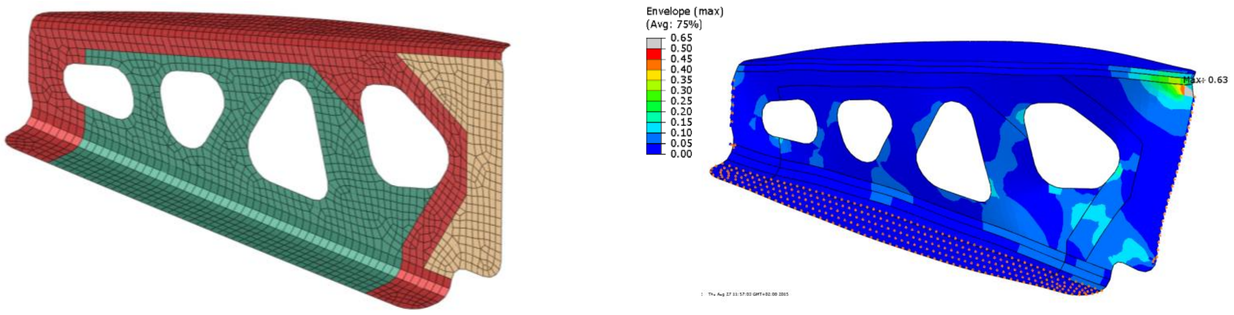
Tab. 1: Material constants of the lamina t = 0,31 mm.

Material constant	E_{11} [N/mm^2]	E_{22} [N/mm^2]	E_{33} [N/mm^2]	ν_{12} [-]	ν_{13} [-]	ν_{23} [-]
Value	58000	58000	10500	0.05	0.41	0.41
Material constant	G_{12} [N/mm^2]	G_{13} [N/mm^2]	G_{23} [N/mm^2]			
Value	4100	3480	3480			
Material constant	Tensile strength [MPa]	Compressive strength [MPa]	Tensile strength [MPa]	Compressive strength [MPa]	In-plane shear strength [MPa]	
Value	750	590	750	590	105	

Throughout the optimization, the stress fields and damage variables were evaluated. The aim was to decrease stress concentration in the corners of cut-outs and to regularize the stress field across the rib. Especially principal stresses across all plies were evaluated and the damage indications according to Hashin [2] and LaRC03 [3] failure criteria were evaluated as well. After several optimization steps, the excessive stress concentrations in the corners of cut-outs were eliminated.

Critical point was indicated in the upper part near the longer vertical edge, where the moment of load is applied (see Fig. 3b). Moreover, in this point, the bold joint is located. According to the prescribed load, the bold joint transfers the load of 1.6 kN. To ensure no damage due to bearing pressure the minimal bolt diameter is 8 mm with safety factor of 2.5. So this point is weakened in addition by the hole for the bolt.

Failure criteria indicated the critical point consistently. LaRC03 indicated higher value of failure index than Hashin's criteria (0,63 versus 0,23 at the load of 36 kN). Due to the location of the critical point, it is not possible to evaluate the criteria each other experimentally. In order to do so, the critical point should be located at the joint-free area.



a) FE mesh – reduced shell elements

b) Index of LaRC03 failure criteria, Load 36 kN, deformation scale factor 25x

Fig. 3: FE model of the rib demonstrator.

Buckling was evaluated during optimization steps in addition. The lower flange is loaded in pressure and eventual loss of stability could occur. The load at the loss of stability considerably depends on boundary conditions which in optimizing steps do not follow the real conditions fully, however some simplification was accepted. The loss of stability of optimized alternatives was determined at the load that is 5 times greater than the specified load. Appropriate eigenshape shows buckling of the web under the cut-out near the loaded edge.

Conclusion

The rib demonstrator from high performance thermoplastic composite was designed and through FEM analysis optimized. The step-by-step optimization process was applied resulting in optimum prepreg blank tailoring and cut-outs shape design. Utilizing failure criteria the critical point was determined. The LaRC03's failure index in the critical point was higher than the Hashin's failure index.

Acknowledgement

This work was funded by the support of applied research and experimental development provided by the Technology Agency of the Czech Republic in the project ALFA TA03010209.

References

[1] J. Šedek, R. Hron, M. Kadlec, Bond Joint Analysis of Thermoplastic Composite Made from Stacked Tailored Blanks. Applied Mechanics and Materials, Vol. 827, 2015. pp.161-168, ISSN 1662-7482

[2] Z. Hashin and A. Rotem, A Fatigue Criterion for Fiber-reinforced Materials, Journal of Composite Materials, vol. 7 (1973), pp. 448–464.

[3] G.C. Davila. et al., Failure Criteria for FRP Laminates, Journal of Composite Materials, Vol. 39 (2005), pp. 323-345.

[4] J. Křena, P. Roškanin, Vývoj a zkoušky optimalizovaného žebra z vyztuženého termoplastu C/PPS, Transfer, Praha, VZLÚ, Vol. 27 (2016), pp. 24-27