About numerical analysis of mechanical joints in laminate structure

Elżbieta Szymczyk [1,a)], Krzysztof Puchała [b)], Jerzy Jachimowicz [c)]

[1] *Department of Mechanics and Applied Computer Science,*
Faculty of Mechanical Engineering, Military University of Technology
Urbanowicza Street 2, 00-908 Warsaw, Poland
e-mail: kmiis@wat.edu.pl

[a)] elzbieta.szymczyk@wat.edu.pl
[b)] krzysztof.puchala@wat.edu.pl
[c)] jerzy.jachimowicz@wat.edu.pl

Abstract. Composites, in general, are brittle material and more notch sensitive than metal alloys. Therefore, mechanical joints of composite parts require a special attention during both the designing and manufacturing process. The simulation of a pin loaded laminate was carried out. The specimen was made of quasi-isotropic CFRP laminate. Nonlinear analysis was performed using Newton-Raphson method with MSC.Marc. The specimen load capacity was estimated using the maximum stress and Hashin failure criteria in quasi-linear as well as in progressive failure analyses. In the paper, failure indices and bearing curves for various contact descriptions (node to segment analytical and discrete, segment to segment and using simplified kinematic ties) and two models of a laminate structure (single layer model and eight orthotropic plies) are compared. The assumed criterion and properties of failure progressive analysis strongly influence the results.

INTRODUCTION

Nowadays, large aircrafts such as Boeing 787 and Airbus A350 are built in fifty percent of composites (e.g. CFRP, GFRP or Glare). In general, composites are brittle material and more notch sensitive than metal alloys. Therefore, mechanical joints of composite parts require a special attention during both the designing and manufacturing process.

The aim of the paper is to present some aspects connected with Finite Element analysis of mechanical joints in composite structures. Numerical simulation depends on mechanical parameters of laminates, fasteners and contact interfaces. Additionally, simulation results can be strongly influenced by parameters of numerical models, including material models. Comparison of selected numerical models in the aspect of bearing performance of CFRP laminate is presented in the paper. Two models/methods of a laminate modelling and various contact descriptions have been taken into account.

OBJECT DESCRIPTION AND NUMERICAL ANALYSIS

There are several ways of modelling the laminate with FEM:

- the simplest method is the usage of homogenous material (without defining separate layers) (Irisarri et al. 2012);
- whole laminate thickness is described with one layer of composite finite elements which consists of orthotropic plies (cases c1 in the paper); stiffness of composite elements is calculated using the classical laminate theory in a brick (3D) or a shell (2D) model (Irisarri et al. 2012, Klasztorny et al. 2012);
- each ply is described as one layer of finite elements made of orthotropic material and the elements of two layers

are connected in common nodes (cases p8 in the paper) (Aktas et al. 2009, Ireman 1998, Tserpes et al. 2002);

- layers of orthotropic finite elements (describing neighbouring plies) are connected using cohesive elements (or simple contact/glue) elements (describing interaction between plies) (Irisarri et al. 2012, Jachimowicz et al. 2015);
- layers of orthotropic elements (plies) are connected with elasto-plastic finite elements (describing an adhesive layer); in this type of model, mesh density should be adapted to adhesive layer thickness and decrease gradually over ply thickness (Szymczyk et al. 2014).

The simulation of pin loaded laminate was performed. The specimen was made of quasi-isotropic CFRP laminate. The specimen is 85 mm long, 36 mm wide and 3 mm thick according to ASTM standards. The hole diameter is 6 mm (with bolt-hole clearance of 0.02 mm).

A solid element was used to model a composite part and a bolt. It is an eight-node element with linear interpolation functions and three translational degrees of freedom per node. Due to laminate lay-up symmetry, only a half of the laminate thickness was modelled. Two models of a laminate structure (single layer model - case c1 and eight ply model - case p8) were taken into account.

The specimen was subjected to displacement (kinematical) or force loading. Nonlinear analysis was performed using Newton-Raphson method with an appropriate convergence criterion (e.g. residual force and/or displacement) using Marc code. Although, failure indices give insights into the behaviour of a particular layer, they are valid only to the first failure occurrence. The specimen load capacity was estimated using both the maximum stress and Hashin failure criteria in quasi-linear as well as in progressive failure analyses. The indices of a failure criterion together with the degradation procedure were used to gradually reduce stiffness of the composite part.

CONCLUSIONS

Various models of a laminate structure and a contact description are used in numerical simulations of mechanical joints since selection of numerical model depends on the aim of analysis.

The numerical results are based on lamina determined properties, therefore, the behaviour of each lamina is considered separately. Higher stiffness of the single layer model (case c1a) and its higher strength, in comparison to an eight-ply model (case p8a), result from different ways of integration, and consequently, significant layer deformations for the latter case. Gradual stiffness degradation can be described more appropriately using an eight-ply model since the model behaviour in this case is more realistic.

The assumed criterion and properties of failure progressive analysis strongly influence the results. Quasi-linear analysis with Hashin failure results in a lower load capacity estimation. The gradual stiffness reduction analysis with displacement loading in a single layer model results in an upper estimation of the specimen load capacity. Residual stiffness in the bearing area significantly influences the bearing performance. In this area, the material failure is a result of compression. An increase in residual stiffness causes an increase in the specimen load capacity and result in a larger hole deformation.

Convergence of nonlinear simulations depends on the loading method. Displacement loading, in general, is more stable and faster, especially if various failure criteria and degradation rules are taken into account.

ACKNOWLEDGMENTS

The research has been funded from the Polish-Norwegian Research Programme under the Norwegian Financial Mechanism 2009-2014 within Project Contract Pol-Nor/210974/44/2013 and Research Project funded from Ministry of Science and Higher Education 2018.

REFERENCES

1. Irisarri F.X., Laurin F., Carrere N., Maire J.-F. ”Progressive damage and failure of mechanically fastened joints in CFRP laminates – Part I: Refined Finite Element modelling of single-fastener joints”, Composite Structures **94**, 2269–2277 (2012).
2. Jachimowicz J., Szymczyk E., Puchała K. ”Study of material mass efficiency and numerical analysis of modified CFRP laminate in bearing conditions”, Composite Structures **134**, 114–123 (2015).