Journal of Achievements in Materials and Manufacturing Engineering
VOLUME 20
ISSUES 1-2
January-February
2007

Optimisation of the rivet joints of the CFRP composite material and aluminium alloy

W. Hufenbach [a], L.A. Dobrzański [b], M. Gude [a], J. Konieczny [b], A. Czulak [a,]*
[a] Institut für Leichtbau und Kunststofftechnik (ILK), Technische Universität Dresden, Dürerstrasse 26, D-01062 Dresden, Germany
[b] Division of Materials Processing Technology, Management and Computer Techniques in Materials Science, Institute of Engineering Materials and Biomaterials, Silesian University of Technology, ul. Konarskiego 18a, 44-100 Gliwice, Poland
* Corresponding author: E-mail address: acz@ilk.mw.tu-dresden.de

Received 04.11.2006; accepted in revised form 15.11.2006

Materials

ABSTRACT

Purpose: The project included analysis of strain, cracking, and failure of riveted joints of plate elements made from the carbon-fibre-reinforced plastics (CFRP) and from the 6061 aluminium alloy.
Design/methodology/approach: The modelled static tensile strength test carried out for the plates from CFRP and from the 6061 aluminium alloy joined with the steel rivet. Computer simulation was carried out with IDEAS software package employing the FEM.
Findings: Simulations using the mesh with a bigger number of FEM elements do not yield better accuracy of calculations and do not improve convergence with the results of laboratory experiments. Only the calculation time gets longer. Computer simulation has also show that the type of contacts employed between elements affects the results significantly.
Research limitations/implications: For the composite materials, joints between materials and computer simulation examinations are planed.
Practical implications: Results obtained for the mesh with 4 and 5 FEM elements are the closest to the results of laboratory experiments, which is confirmed by the strain plot. Simulations using the mesh with a bigger number of FEM elements do not yield better accuracy of calculations and do not improve convergence with the results of laboratory experiments. Only the calculation time gets longer. Computer simulation has show that the type of contacts employed between elements affects the results significantly.
Originality/value: The paper presents influence of fibre mesh closeness on convergence of the results with laboratory tests. Simulation results were collected and compared with the laboratory static tensile strength tests results.
Keywords: Composites; Mechanical properties; Computer simulation; FEM

1. Introduction

Nowadays, the composite materials, like plastics reinforced with carbon fibres (CFRP), glass (GFRP) or aramid ones (AFRP), are widely used in various industry branches: in the armaments industry, automotive-, aircraft- [1], chemical industry, electrical engineering, building engineering [2], and even in cryogenics [3, 4].

The widespread applications of these materials result from the intensive research carried out by many scientific and research centres all over the world [5-7] whose goals are their optimization and potential applications in many industry branches, and also employment of FRPs as the starting material for development of the new intelligent composite materials – smart materials [8].

Operation of the entire structures and also of their elements is analysed from the point of view of their limiting states which -

when exceeded - result in change of material's behaviour being considered by their designer as hazardous for the designed structure. Loss of load carrying capacity (e.g., crack), and loss of the elastic- or elastic-plastic stability, like – for example – buckling, may be examples of the limiting states [9].

It is assumed that there are four reasons for origination and development of damage processes in the fibre composites. There are as follows: cracking of matrix, breaking of fibres, loss of connection of fibres with the matrix or drawing out fibres from the matrix [10]. One has to mention the following theories among the effort hypotheses deciding the possibility of appearing any material damage form in any stress case:

- maximum stress theory,
- maximum strain theory,
- Tsai-Hill theory,
- Tsai-Wu tensor theory,
- Hashin-Puck theory.

The above mentioned theories deal only with forecasting the beginning material failure, except the Hashin-Puck's theory, thanks to which one can recognize in addition the defect type (breaking of fibres or matrix) [11].

On the other hand, the finite element method yields solution approximated by division of the analysed model into sub-areas characteristic of: shape, number and type of nodes, nodal values, approximating functions.

The approximate solution in the finite element method is assumed to take the following form:.

$$u_p(x_j) = \sum_{l}^{k} N_i(x_j) v_i \qquad (1)$$

where: $N_i(x_j)$ – approximating functions, called shape functions, v_i – unknown nodal values of the searched quantity, playing roles of coefficients to be determined [12].

Currently, the finite element methods and the boundary element method are the most often used methods (including the computer based ones) employed for modeling, analysis, and simulation of the behaviour of structures [13].

Calculation results are confronted with the laboratory tests results. The differences noticed are the consequences of the measurement and calculation errors, resulting from the computational model, which is differing from the real structures. The reason may be also a lack of the exact data about the parameters of the analysed structure and the limited precision of numerical calculations (numerical errors).

The numerical errors result from the limited precision of calculations carried out by computers and may influence the accuracy of calculations significantly [14, 15].

An interesting and important issue is development and optimisation of riveted joints of the CFRP composite material with the classic aluminium alloys, whose use is unavoidable sometimes in the lightweight constructions.

2. Experimental procedure

Two plates were subjected to the static tensile test: aluminium one from the 6061 alloy and the CFRP (*carbon-fibre-reinforced plastic)* joined with the steel rivet. The aluminium plate was fixed and the tensile load was applied to the plate made from CFRP. Contacts were defined between the elements (Fig.1). Geometry of plates (dimensions for the half used in the calculations):

- width: 22.5 mm,
- length: 100 mm,
- CFRP thickness: 1.8 mm,
- arrangement of fibres in the CFRP plate 0°
- thickness of the aluminium plate: 2 mm,
- plate overlap: 20 mm,
- diameter of the holes in plates: 4.8 mm,
- diameter of the steel rivet: 4.74 mm,
- diameter of the rivet head: 12.6 mm,
- height of the rivet head: 2.32 mm,
- diameter of the headed rivet part: 4.07 mm,
- height of the headed rivet part: 1.48 mm,
- total rivet height: 11.63 mm.

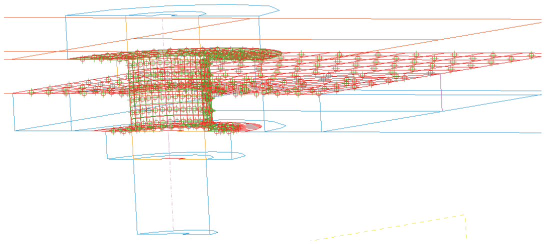

Fig. 1. Contacts defined between the joined elements

In addition verification according to Tai-Wu criterion was carried out, to determine the test pieces' cracking mechanism. Symmetry of the investigated joint was taken into account to ease the analysis, so only a half of the assembly was modeled (Fig. 2), which made it possible to reduce the model complexity and cut the analysis time.

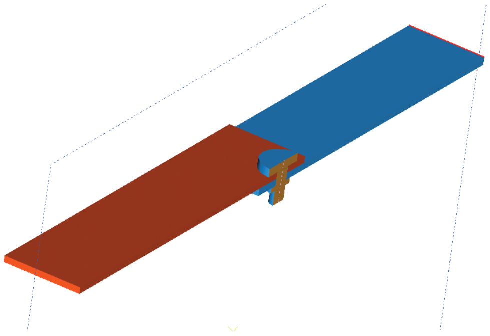

Fig. 2. Geometry of the riveted joint

Boundary conditions:

- tensile load: 1770 [MPa],
- removing all degrees of freedom in the joint location from the 6061 alloy,
- symmetry,
- preventing the dislocation in the Y axis direction at the load application place,

- definition of contacts between the CFRP plate and the rivet, rivet and the aluminium plate, and between the plates.

Four FEM mesh variants were analysed, differing with closeness of elements in each plate's thickness. 3, 4, 5, and 6 mesh elements were assumed for 1.8 mm thick CFRP plate, and for 2 mm thick aluminium alloy plate. Arrangement of the elements in the analysed joint's FEM mesh is presented in Fig. 3.

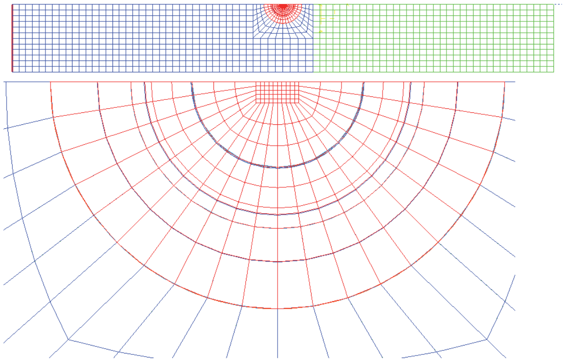

Fig. 3. Arrangement of elements in the FEM mesh

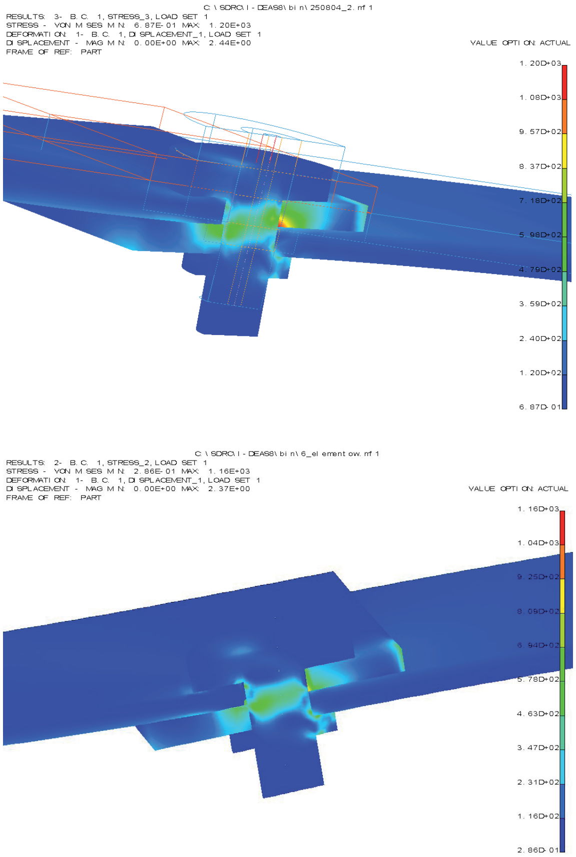

Fig. 4. Stresses induced in the riveted joint location for the mesh with a) 3, b), and 6 FEM mesh elements on the plate cross section

3. Results

The significant differences in results (location of the biggest stresses and their values) were observed in the modelled static tensile strength test carried out for the plates from CFRP and from the 6061 aluminium alloy joined with the steel rivet – depending on the contact type between the elements. In case of contacts generated automatically the results differed significantly from those obtained in the laboratory tests; however in case of the contacts defined manually the results were consistent with the laboratory ones. Moreover, it was noticed that the biggest differences in stress values and locations in which they occurred, and also strains and areas of the critical stresses took place for the FEM mesh defined with 3 and 6 elements on the plate transverse section (Fig. 4, 5). Differences in the obtained results depending on the method of defining the contacts between the elements are listed in Table 1 and 2.

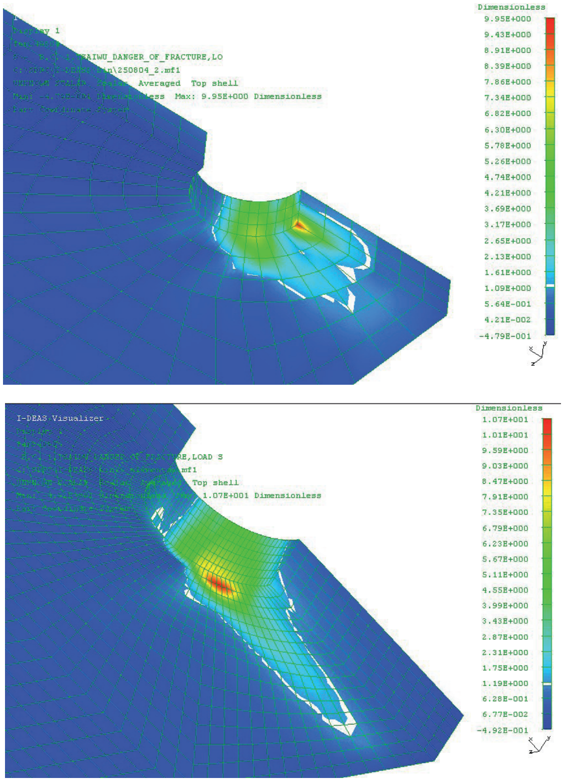

Fig. 5. Critical stresses according to Tai Wu criterion in the CFRP composite material for the mesh with a) 3, b), and 6 FEM mesh elements on the plate cross section

4. Conclusions

In case of defining the FEM mesh with 3 and 6 elements on the cross section the results obtained in computer simulation differ most from the laboratory results. The conclusion arising from this is that the FEM mesh is unsuitable in this case. One can

Table 1.
Comparison of the elongation of the CFRP test piece and generated stresses depending on the contact type

Number of mesh elements	Contact generated automatically		Contact defined manually	
	Elongation, mm	Stress, MPa	Elongation, mm	Stress, MPa
3	2.44	1200	2.38	876
4	2.40	1130	2.4	1010
5	2.40	1140	2.39	1040
6	2.37	1160	2.37	1160

Table 2.
Analysis results – risk of fracturing the CFRP element according to Tsai Wu criterion

Number of mesh elements	Tsai Wu Risk of fracture	Tsai Wu Mode of fracture
3	9.95	1
4	1.14	1
5	1.10	1
6	1.02	2

notice, using the maximum strain plot that the biggest shift has occurred in systems with 4 and 5 mesh elements.

Results obtained for the mesh with 4 and 5 FEM elements are the closest to the results of laboratory experiments, which is confirmed by the strain plot. Simulations using the mesh with a bigger number of FEM elements (e.g., 8, 10 or more) do not yield better accuracy of calculations and do not improve convergence with the results of laboratory experiments. Only the calculation time gets longer (Fig. 6).

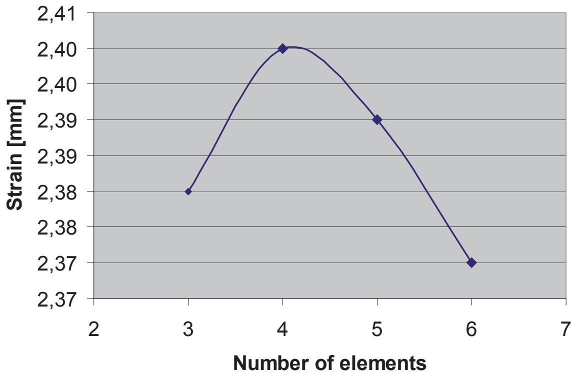

Fig. 6. Criterion of strain plot

Computer simulation has also show that the type of contacts employed between elements affects the results significantly. In case of contacts generated automatically the location of the biggest stresses changes its place, and none of them is correct, and their locations are random; whereas in case of the contacts defined manually this location is constant.

References

[1] R. Zimmermann, R. Rolfes, CFRP fuselage structures – postbuckling permitted, Air & Space Europe, 3, No. 3/4, 2001.

[2] S. Swiatecki, Building better bridges with CFRP, Reinforced plastics, March (1998) 44-47.

[3] R. Heydenreich, Cryotanks in future vehicles, Cryogenics 38 (1998) 125–130.

[4] T. Horiuchi, T. Ooi, Criogenic properties of composite materials, Criogenics 35 (1995) 677-679.

[5] W. Hufenbach, W. Błażejewski, L. Kroll, R. Böhm, M. Gudeb, A. Czulak, Manufacture and multiaxial test of composite tube specimens with braided glass fiber reinforcement, Journal of Materials Processing Technology, 162-163 (2005) 65-70.

[6] W. Hufenbach, L. Kroll, R. B¨ohm, A. Langkamp, A. Czulak, Piping elements from textile reinforced composite materials for apparatus construction, 12th International Scientific Conference AMME, (2003) 391-398.

[7] W. Hufenbach, L. Kroll, M. Gude, A. Czulak, R. Bohm, M. Denczak, Novel tests and inspection methods for textile reinforced composite tubes, 11th International Scientific Conference of Contemporary Achievements in Mechanics, Manufacturing nad Materials Science, (2005) 488-491.

[8] W. Hufenbach, M. Gude, L. Kroll, Composites Science and Technology (2002) 2201–2207.

[9] R.G. Cuntze, A. Freund, The predictive capability of failure mode concept-based strength criteria for multidirectional laminates, Composites Science and Technology 64 (2004) 343-377.

[10] W. Hufenbach, M. Gude, A. Czulak, Actor-initiated snap-through of unsymmetric composites with multiple deformation states, Journal of Materials Processing Technology 175 (2006) 225–230.

[11] W. Hufenbach, L. Kroll, Laminated cylindrical shells under mechanical and hygro-thermal loads, Advances In Engineering Software 23 (1995) 83-88.

[12] M. Dacko, W. Borkowski, T. Niezgoda, M. Wieczorek The Finite element methods in mechanical design, Arkady 1994.

[13] A.P. Wilczyński, Polymer fibrous composite" WNT Warszawa 1996.

[14] G. Rakowski, Z. Kacprzyk, The Finite element methods in mechanical design, Warsaw Univercity Press, Warszawa 2005.

[15] O.C. Zienkiewicz, The Finite element methods, Arkady, Warszawa 1972.