Agnieszka Derewońko

Failure Simulation of Metal-Composite Joints

**Department of Mechanics
and Applied Computer Science,
Military University of Technology,**
ul. Kaliskiego 2, 00-908 Warsaw, Poland;
E-mail: aderewonko@wat.edu.pl;

Abstract
The paper presents a numerical analysis method allowing the determination of the degree of failure of an adhesive joint with the use of CAE software. The results of an experimental test are used to describe properties of the joint parts. Described herein is the behaviour of a multi-stepping lap joint of metal with a multilayered fibre composite that is often used in the aircraft industry. Modelling of the contact problem between the metal surface and layer of the composite enables determination of the value of the external tensile load at which the process of failure of the joint begins. The author also proposes a simple method to shape solid finite elements which allows the elimination of the singularity around finite element vertices. The method has been applied to numerical analysis of the joint, in which the layers were stacked at various angle orientations. The influence of the step length of the metal part on the joint strength was also considered. The results of numerical analysis were compared with those of experimental investigations of the joint strength, giving satisfactory agreement.

Key words: *failure, metal-composite bonded joint, finite element method (FEM).*

Introduction

Adhesive bonding and co-curing are widespread methods used to join two materials of different stiffness such as metal and fibre-reinforced composites. This method is widely used in aircraft constructions and the repair of their skins [1, 2].

Numerical analysis is a comfortable method to determine the structure strength and to characterise the stress and strain fields in the metal and composite.
The design of adhesively bonded and co-cured metal-composite structures requires accurate material property data [3, 4] due to the stiffness mismatch. It is important for the designers to model the laminated composite accurately due to many uncertainties and stochastic finite element approaches which are discussed in plenty of research [5, 6].

Methods of defining a model of different types of co-cured joints are discussed in many papers [7, 8]. The effective design and analysis of the stepped lap require a methodology to obtain stress distributions through each part of the joint. A series of studies on different types of lap joints, e.g. the stepped lap, were investigated using continuum mechanics and a numerical approach [9, 10].

The main objective of this study was to determine the configuration of an adhesive joint with composite metal for which a large tensile load will not cause the failure of the joint and its strength will remain high. To achieve this goal, it was necessary to develop a method that enables estimation of the external load causing the beginning of the unsticking of the composite layer from the metal and determination of the initial location of failure.

An engineering approach for the modelling of full-scale co-curing multi-stepped lap joints is presented. Professional engineering software codes were used to generate accurate models and perform analyses. Laboratory tests of co-curing multilayer laminated composite specimens were applied to find the material elastic constants. Uniaxial tensile tests were carried out to verify the numerical analysis method.

Three-dimensional finite elements were utilised to model the multi-stepped lap joint. To design an optimal composite lay-up composite structure, 3D finite elements were used. The interface failure was considered due to the frictional contact problem between the metal and laminated composite. Those regions are expected to have high stress gradients caused by different material stiffness and material discontinuity. The description of the failure mechanism is limited to an indication of the external load which causes the failure initiation and determination of the regions where it occurs. The effect of the step length and shape of the finite elements located on an edge of the step on the values of the stresses in this area is considered.

Numerical results are confirmed by the laboratory tests.

Materials and methods

The co-cured CE 8201-245-45/120 carbon/epoxy woven fabric prepreg [11] was utilised for joining two metal parts (***Figure 1***).

Two types of multi-stepped lap joints were considered. In both cases, the largest thickness of the part is equal to 2.43 mm. In the first one, called „Uniform”, in the metal part, three steps of equal length were cut. In the other , „Nonuniform”, two outer steps of equal length were much shorter than the central one. Depending on the depth of the step, one or two layers of the composite were bonded to the metal. The thickness of the composite layer was equal to approximately 0.27 mm. Each layer was characterised by a different orientation of fibres. ***Table 1*** shows the combinations of the composite layer for „uniform” (a) and „nonuniform” (b) joints. To avoid confusion, the layer number is preceded by the letter „w”. The layer configurations shown in ***Table 1*** are commonly used in the construction of the aircraft wing skins and the plane tails.

Table 1. *Composite layer orientation.*

Joint label	a (uniform)							b (nonuniform)						
Layer No.	w1	w2	w3	w4	w5	w6	w7	w1	w2	w3	w4	w5	w6	w7
Orientation	0°°	45˚	0˚	45˚	45˚	0˚	45˚	0°°	45˚	0˚	45˚	0˚	45˚	0˚

Derewońko A. Failure Simulation of Metal-Composite Joints.
FIBRES & TEXTILES in Eastern Europe 2013; 21, 5(101): 131-134.

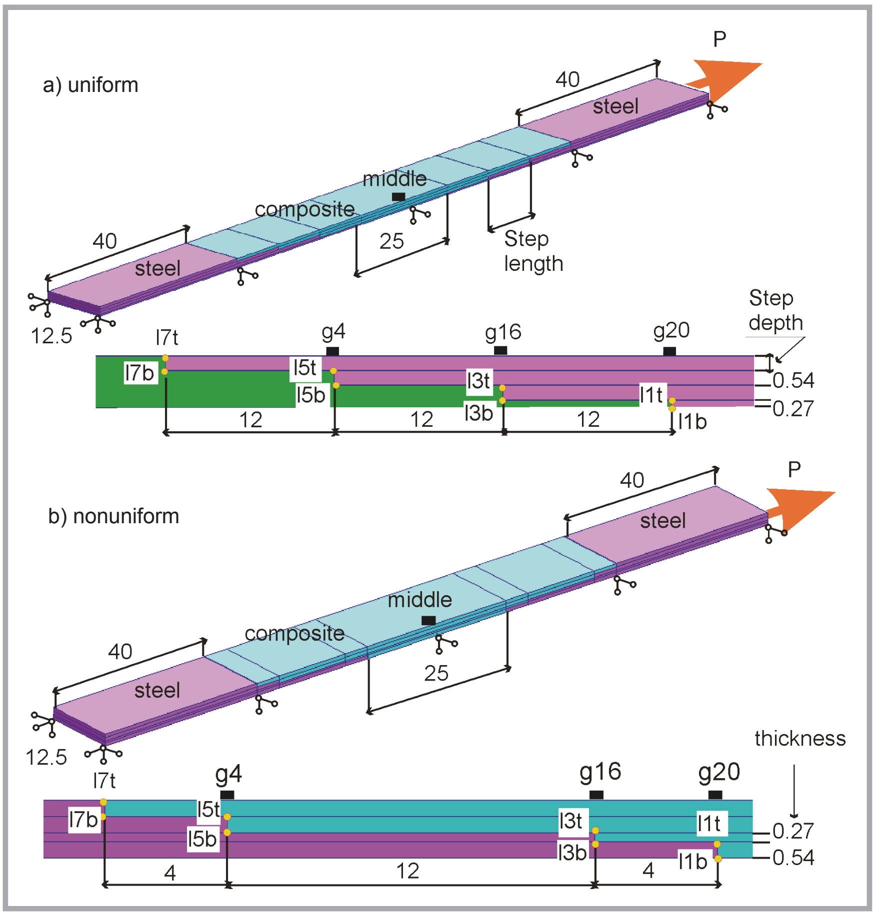

Figure 1. *Overall dimension for uniform (a) and nonuniform (b) joint.*

Table 2. *Orthotropic elastic constants for the composite.*

E_{11}, MPa	E_{22}, MPa	E_{33}, MPa	G_{12}, MPa	G_{23}, MPa	G_{31}, MPa	ν_{12}	ν_{23}	ν_{31}
58093	58093	9759	3545	2564	2564	0.015	0.54	0.16

Table 3. *Failure material parameters for the composite.*

X_t, MPa	X_C, MPa	Y_t, MPa	Y_C, MPa	Z_t, MPa	Z_C, MPa	S_{12}	S_{23}	S_{31}
650	555	650	555	10	500	181	132	132

The overall numerical model dimensions for each joint are shown in ***Figure 1***. Due to the double symmetry, only a quarter of the joints are modelled. Adequate boundary conditions were applied (***Figure 1***). The degrees of freedom were removed in the symmetry planes in directions perpendicular to them. In the area of the slide holder, all degrees of freedom, except the direction of the load, were removed. However, on the opposite side of the sample, in the area of the permanent holder, all degrees of freedom were removed.

The value of the tensile load amounted to 25988 N and was the same for each joint. 3-D deformable bodies were modelled by means of three-dimensional, lower-order HEX elements, with the use of the MSC.Marc programme [12]. Eight-node, isoparametric composite brick elements were used to model the laminated composite [12]. These elements allow to define layer-by-layer material identifications, layer thicknesses, and orientation angles for the laminated composite material. The element was integrated with the use of a numerical scheme based on the Gauss quadrature. Each layer contained four integration points. The mass matrix of the element was formed using eight-point Gaussian integration. Second Piola-Kirchhoff stresses and Green strains were the output at each integration point in the case of large deformations.

Material parameters for 30HGSA steel were obtained during the laboratory tests. Based on the experimental tests, it was assumed that the composite deforms only in the elastic range. Young modulus E = 197 GPa, Poisson's ratio ν = 0.3 and a stress-strain curve with a yield stress of 331 MPa, were implemented as the material model of metal. The laboratory tests were prepared in accordance with ASTM standards and literature references for composite materials [13 - 15]. This guide summarises the application of ASTM standard test methods (and other supporting standards) to continuous-fibre reinforced polymer matrix composite materials, like tension and compression for specimens cut in two directions and the shear test. The ASTM standards most commonly used or most applicable are included, emphasising the use of standards of Committee D30 for Composite Materials. The method of testing is described in the report on the implementation of research [16].

Orthotropic elastic constants for the composite are shown in ***Table 2***. Failure mechanism-based progressive damage analysis was postulated for evaluating the tension failure of the metal - composite joints. "At each integration point, Marc calculates six failure indices FI and six strength ratios SR. For the Maximum Stress Failure Criterion, the strength ratios SR are the reciprocals of the corresponding failure indices FI" [12]. The maximum stress criterion is used to estimate the joint failure. The failure material parameters are shown in ***Table 3***. X_t, X_c are the maximum stresses allowable in the 1-direction in tension and compression; Y_t, Y_c are the maximum stresses allowable in the 2-direction in tension and compression; Z_t, Z_c are the maximum stresses allowed in the 3-direction in tension and compression; S_{12} is the maximum in-plane shear stress allowable; S_{23} is the maximum σ_{23} shear stress allowable, and S_{23} is the maximum σ_{31} shear stress allowable.

The contact problem is modelled between the metal and composite stuck surfaces [17]. The metal/composite interface is modelled as two sets of separated nodes. According to the MSC. Marc manual [12], when contact is found a multipoint constraint (called tying) is automatically imposed. When the exterior faces (3-D) of the other deformable bodies are known, a constraint expression is formed. The constraint equation is such that the contacting node should be able to slide on the contacted segment, subject to current friction conditions.

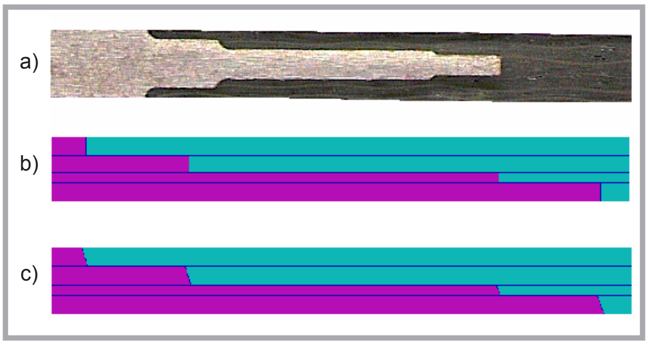

Figure 2. *Nonuniform triple stepped lap joint and its models.*

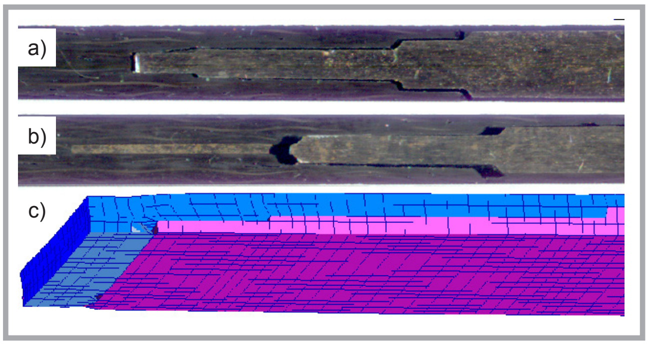

Figure 3. *Specimen failure: a) nonuniform, b) uniform, c) numerical model.*

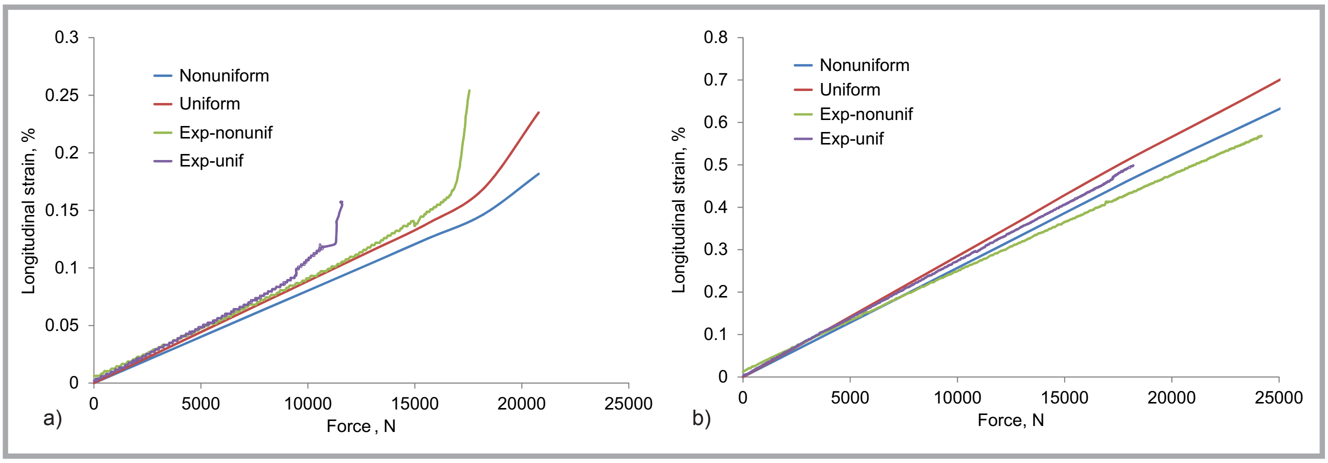

Figure 4. *Comparison of the results of the laboratory tests and numerical analysesł a) g4, b) middle, indicate positions of the gauges.*

When Coulomb friction is used with the stress-based model, the integration point stresses are first extrapolated to the nodal points and then transformed, so that a direct component is normal to the contacted surface. Having applied this normal stress and relative sliding velocity, the tangential stress is then evaluated and a consistent nodal force is calculated. The penalty method was used for the purpose of performing the contact analysis [18].

While preparing the co-cured interface model, it was assumed that the relative contact surface traction was zero in the current configuration. This modelling method allows to simulate the joint failure and to determine the stress/strain for the metal and composite separately.

A node on the patch (face of 3-D deformable body) slides freely until it reaches the intersection between the segments (faces). If it is concave, the node first tries to slide along the line of intersection before moving to the next segment. This is the natural (lower energy) state of motion. Since the bodies are continuously changing in shape, the corner conditions (sharp convex, smooth or sharp concave) are continuously being re-evaluated. The default value of the smooth angle (α_{smooth}) is 20° for the 3-D model. The value of α_{smooth} is important in controlling the computational costs. A larger value of α_{smooth} reduces the computational costs, but it might lead to inaccuracies as well [12].

Two different methods were applied for modelling the free step edge of the stepped lap joint, which is shown in ***Figure 2.a***. The joint model, with a standard finite element shaped with a right angle between edges as in Case 1, is presented in ***Figure 2.b***. Case 2 is shown in ***Figure 2.c***. The small rounding of the concave corner is modelled. In ***Figure 2.b*** and ***Figure 2.c***, the blue colour indicates the layer composite, while purple shows metal.

Results and discussion

Numerical analyses and laboratory tests show that the nonuniform triple stepped lap joint is a better structure than the uniform one. The yield in the third step of the metal part appears for the uniform joint due to its insufficient thickness (***Figure 3***). This problem does not occur for the nonuniform joint, where the step thickness is doubled. Laminated composite failure occurs only in the nonuniform joint in the step ends.

Experimental verifications of the numerical analyses for each of the joints are presented in ***Figure 4***. The results for the triple stepped lap joint with equal step lengths (“Uniform”) are less accurate than in the case of the joint with various step lengths (“Nonuniform”). Symbols “g4” and “Middle” indicate positions of the gauges, as presented in ***Figure 1***.

The structural strength of the joint of unequal step size (nonuniform) increases by about 9% more that than that of the uniform joint.

The metal/composite interface edges are critical regions where failure starts in the joints. Numerical analyses and the laboratory tests also show that the nonuniform triple stepped lap joint is a more recommended structure as no yielding occurred in the metal parts. It is known that the end steps define the stepped-lap joint load capacity, which depends signif-

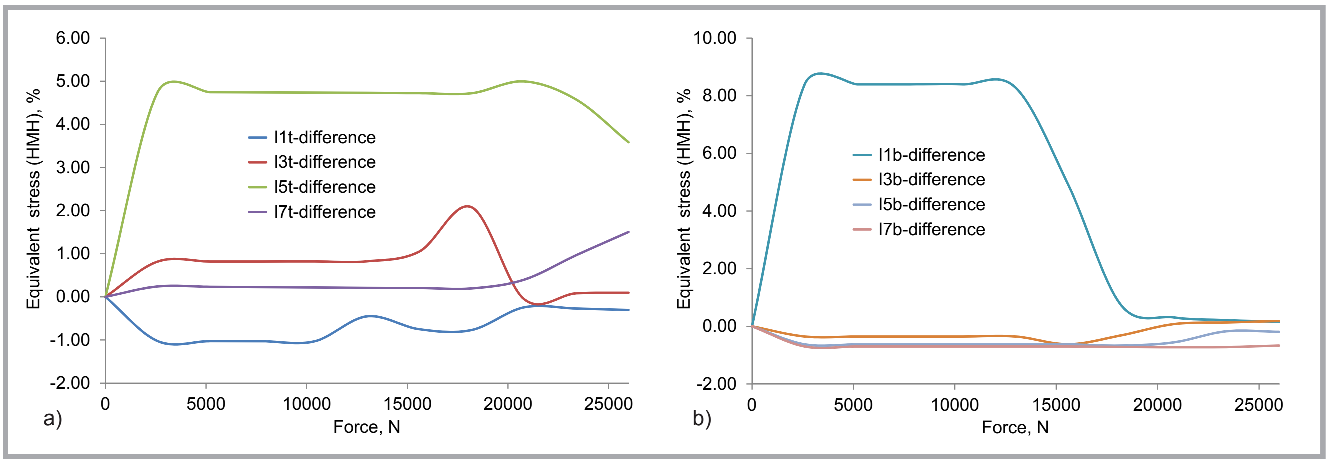

Figure 5. *Difference in von Mises equivalent stress; a) convex corner, b) concave corner.*

icantly on the step length. The results are shown as a diagram. Differences in the von Mises equivalent stress with regard to the two nodes in the metal\composite interface corners on the symmetry surface as a function of the load value are presented in ***Figure 5***.

The HMH symbol in the description in ***Figure 5*** is an abbreviation of Huber-Mises-Hencky strength theory.

The biggest disparities (up to 8%) occur in nodes l1b and l5t, which are symbolised in ***Figure 1***. An edge slope model allows to avoid right corners, normals discontinuity and to make the surfaces both physically continuous and topologically contiguous.

Conclusions

Failure of the uniform joint is not caused by the unsticking of the composite and metal layer but by the damage of the metal parts, characterised by the large ratio of the length to thickness of the step at the end of the joint. Therefore in aircraft constructions, it is preferable to use a combination of the uneven steps because no yield in the metal parts occurs. It is known that the stepped-lap joint load capacity is defined by the end steps [9] and depends significantly on their length.

The application of standard software tools and obtaining stress discontinuity on the interface between the metal and composite are the main advantages of the methods presented due to the frictional contact modelling. The edge slope model presented allows to avoid right corners, normal's discontinuity and to make the surfaces both physically continuous and topologically contiguous. 3-D finite elements and numerical analysis methods applied for composite modelling are useful for determining the strength of triple stepped-lap metal-composite joints on a global scale.

Acknowledgements

This work has been made possible through the financial support of National Centre for Research and Development (NCBR) under research grant INNOTECH-K2/IN2/56/182840/NCBR/13

References

1. Ahn SH, Springer GS. Repair of Composite Laminates, DOT/FAA/AR-00/46, www.tc.faa.gov, 2000.
2. Mollenhauer DH, Fredrickson B, Schoeppner G, Iarve EV, Palazotto A. Analysis and Measurement of Scraft-Lap and Step-Lap Joint Repair in Composite Laminates. In: *16th International Conference on Composite Materials*, Japan, 2007.
3. Duncan B, Crocker L. *Failure of flexible adhesive joints.* Report MATC(A)36, 2001.
4. Godzimirski J. *Wytrzymałość doraźna konstrukcyjnych połączeń klejowych*. Warszawa WNT, 2002.
5. Onkar AK, Upadhyay CS, Yadav D. Probabilistic failure of laminated composite plates using the stochastic finite element method. *Composite Structures* 2007; 77: 79-91.
6. Da Silva LFM, das Neves PJC, Adams RD, Spelt JK. Analytical models of adhesively bonded joints - Part I: Literature survey. *International Journal of Adhesion & Adhesives* 2009; 29: 319–330.
7. Kweon JH, Jung JW, Kim TH, Choi JH, Kim DT. Failure of carbon composite-to-aluminium joints with combined mechanical fastening and adhesive bonding, *Composite Structures* 2006; 75: 192-198.
8. Da Silva LFM, das Neves PJC, Adams RD, Wang A, Spelt JK. Analytical models of adhesively bonded joints—Part II: Comparative study. *International Journal of Adhesion & Adhesives* 2009; 29: 331–341.
9. Barroso A, Paris F, Mantic V. Representativity of the singular stress state in the failure of adhesively bonded joints between metals and composites. *Composites Science and Technology* 2009; 69: 1746–1755.
10. Anyfantis KN. Finite element predictions of composite-to-metal bonded joints with ductile adhesive materials. *Composite Structures* 2012; 94: 2632–2639.
11. SGL CARBON GROUP, Carbon Composites, www.sglcarbon.com, 2000.
12. MSC. Marc Volume A: Theory and User Information, Version 2005, The MacNeal-Schwendler Corporation.
13. ASTM D4762 - 11a Standard Guide for Testing Polymer Matrix Composite Materials, ASTM International.
14. ASTM D3039/D3039M-07 Standard Test Method for Tensile Properties of Polymer Matrix Composite Materials, ASTM International.
15. ASTM D5379/D5379M-98 Standard Test Method for Shear Properties of Composite Materials by the V-Notched Beam Method, ASTM International.
16. Piekarski & others, Sprawozdanie końcowe z realizacji pracy badawczej No 4 T12C 010 27, Warszawa, 2007.
17. Derewońko A, Godzimirski J, Kosiuczenko K, Niezgoda T, Kiczko A. Strength assessment of adhesive-bonded joints. *Computational Materials Science* 2008; 43: 157–164.
18. Peric DJ, Owen DRJ. Computational Model for 3-D contact problems with friction based on the Penalty Method. *Int. J. of Meth. Eng*. 1992; 35.

Received 14.08.2012 Reviewed 05.11.2012