

NUMERICAL ANALYSIS OF THE METAL-COMPOSITE JOINTS

Agnieszka Derewońko

*Military University of Technology, Department of Mechanics and Applied Computer Science
Kaliskiego Street 2, 00-908 Warsaw, Poland
tel.: +48 22 6837906, fax: +48 22 6839461
e-mail: a.derewonko@wme.wat.edu.pl*

Roman Gieleta

*Military University of Technology, Department of Mechanics and Applied Computer Science
Kaliskiego Street 2, 00-908 Warsaw, Poland
tel.: +48 22 6839226, fax: +48 22 6839461
e-mail: r.gieleta@wme.wat.edu.pl*

Abstract

The development and implementation of the new technologies and materials are one of the conditions allowing improving the aircraft structures. The composite materials enable to reduce the aircraft weight and keep appropriate strength and fatigue properties. Application of the composite materials in the wing elements, and metal alloys in assemblies of the aircraft fuselage requires elaboration of the appropriate method for technology joining metal and composite. Bonding eliminates breakages in the composite plies which are initiated during assembly process of the structure with mechanically fastened joints e.g. bolts [1]. The strength and stiffness of the composite depend on shape and direction of the fibre which carry the tensile and compression stresses. Shear stress and strength of the composite depend on the resin system. Therefore, the use of composites brings more complicated design approach compare to metal.

The aim of the study is to find appropriate method to determine the failure loading of the metal-composite aircraft wing pivot fitting. Numerical and laboratory investigations for metal-composite joints are presented. Interaction between the composite plies is modelled as the contact problem. It allows determining the strain and stress at the contact surfaces and stress concentration fields. Contact modelling technique is also applied to model interface between metal and composite.

Engineering approach to the model full-scale co-curing triple stepped lap joint is presented. The professional engineering software tools are used to create accurate models. Tensile tests of the metal-composite joint were carried out to verify the numerical analysis methods.

Keywords: *metal-composite joint, FEM, contact problem, nonlinearity, aircraft structure*

1. Introduction

The development and implementation of the new technologies and materials are one of the conditions allowing improving the aircraft structures. The composite materials enables to reduce the aircraft weight and keep (maintain) appropriate strength and fatigue properties. Weight decrease of the aircraft allows to extend subassemblies size e.g. wing span as in the Airbus A380 structure. About 22% of the A380 is made of composite. Application of the composite materials in the wing elements, and metal alloys in assemblies of the aircraft fuselage requires elaboration of the appropriate method for technology joining metal and composite. Bonding eliminates breakages in the composite plies which are initiated during assembly process of the structure with mechanically fastened joints e.g. bolts [1].

The strength and stiffness of the composite depend on shape and direction of the fibre which carry the tensile and compression stresses. Resin matrix is a material that holds reinforcement (fibres) together. Shear stress and strength of the composite depend on the resin system. Therefore,

the use of composites brings more complicated design approach compare to metal. In order to get the required strength and stiffness, several plies must be used with various thicknesses and fibre (woven fabric) orientations. Usually, in the finite element model, composite plies are converted into one layer with equivalent stiffness [3, 6]. Only one integration point is needed across the one ply thickness. It is particularly attractive when the number of composite layers is large and all the layers of the laminated composite have the same length, because the analysis of these smeared shell structures uses less computer time and storage space. When the structure consists of laminated composite layers with different length other modelling techniques are necessary.

The aim of the study is to find appropriate method to determine the failure loading of the metal-composite aircraft wing pivot fitting. Numerical and laboratory investigations for metal-composite joints are presented. Laboratory tests of the metal-composite joints are modelled. Eight-node 3D finite elements with the composite's attributes are used in the discussed method to develop the composite specimen's numerical model. Each ply of the composite has orthotropic property.

Interaction between the composite plies is modelled as the contact problem. It allows determining the strain and stress at the contact surfaces and stress concentration fields. Contact modelling technique is also applied to model interface between metal and composite.

Engineering approach to the model full-scale co-curing triple stepped lap joint is presented. The professional engineering software tools MSC.Patran and MSC.Marc are used to create accurate models. Laboratory tests of the co-curing multilayer laminated composite specimens are applied to find the material elastic constants. Tensile tests of the metal-composite joint were carried out to verify the numerical analysis methods.

2. Theoretical background

Eight-node, isoparametric composite brick element are used to model the laminated composite [5]. The element allows defining the layer-by-layer material identifications, layer thicknesses, and orientation angles for a laminated composite material. The element is integrated using a numerical scheme based on Gauss quadrature. The mass matrix of this element is formed using eight-point Gaussian integration. Second Piola-Kirchhoff stresses and Green strains are output at each integration point for the case of large deformations. For each individual layer, various constitutive laws can be used. The constitutive law for an orthotropic elastic compressible material, used to model prepreg, is given by the following more general form of Hooke's Law:

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_{xx}} & -\frac{\gamma_{yx}}{E_{yy}} & -\frac{\gamma_{zx}}{E_{zz}} & 0 & 0 & 0 \\ -\frac{\gamma_{xy}}{E_{xx}} & \frac{1}{E_{yy}} & -\frac{\gamma_{zy}}{E_{zz}} & 0 & 0 & 0 \\ -\frac{\gamma_{xz}}{E_{xx}} & -\frac{\gamma_{yxz}}{E_{yy}} & \frac{1}{E_{zz}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{xy}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{zx}} \end{bmatrix} \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix}. \quad (1)$$

Contact between deformable bodies in MSC.Marc is automatically solved. This procedure is based on constrain minimization problem. The augmented Lagrangian method is a procedure of numerical implementation of the contact constraints. When contact is automatically detected, the degrees of freedom are transformed to a local system and displacement constraints are imposed such as:

$$\Delta \mathbf{u}_{normal} = v \cdot \mathbf{n}, \quad (2)$$

$$\Delta \mathbf{u}_{tangential} = v \cdot \mathbf{t}, \quad (3)$$

where:

v - the prescribed relative velocity,

\mathbf{n} - normal vector,

\mathbf{t} - tangential vector in the direction of relative velocity.

Along a contact surface, the material flows in one direction in part of the surface and in the opposite direction in another part of the surface. This flow is defined as a relative sliding velocity vector. Therefore the tangential vector is described as:

$$\mathbf{t} = \frac{v}{\|v\|}. \quad (4)$$

The penalty method is a procedure used to numerically implement the contact constrains [4]. The governing FEM equation is solved by the Newton-Raphson method:

$$\mathbf{K}\mathbf{u} = \mathbf{F}, \quad (5)$$

where:

\mathbf{u} - nodal-displacement,

\mathbf{F} - external nodal-load vector,

\mathbf{K} - tangent-stiffness matrix.

A kinematics' boundary condition is continuously changing because the normal and tangent vectors can change from iteration to iteration. This leads to change of the global stiffness matrix by the stiffness matrix contribution follows from:

$$K_{ij} = -\frac{\partial f_{t_i}}{\partial v_k} \frac{\partial v_k}{\partial \Delta u_{t_i}}, \quad (6)$$

where f_{t_i} is friction force in the iteration i .

Therefore, the contact normal and internal forces and stresses are obtained for each load increment [5]. The normal contact stress must be positive inside of the contact area and zero outside [2, 4]. Therefore, a node should separate when the tensile force or the normal stress exceeds the surface tension. Such an approach allowed to determine and compare the forces and stress distribution at the adhesive and adherend surfaces.

A failure mechanism-based progressive damage analysis was postulated for analysing the tension failure of the composite specimen. To estimate a failure of the specimen and metalcomposite joint the maximum stress criterion is used:

$$F = \max \left(\frac{\sigma_1}{X_t}; -\frac{\sigma_1}{X_C}; \frac{\sigma_2}{Y_t}; -\frac{\sigma_2}{Y_C}; \frac{\sigma_3}{Z_t}; -\frac{\sigma_3}{Z_C}; \left| \frac{\sigma_{12}}{S_{12}} \right|; \left| \frac{\sigma_{23}}{S_{23}} \right|; \left| \frac{\sigma_{31}}{S_{31}} \right| \right). \quad (7)$$

F is the failure index ($F = 1.0$). Other failure material parameters are contained in Tab. 1.

Tab. 1. Failure material parameters for the composite

Parameter	Symbol	Value [MPa]
Maximum tensile stress in x-direction	X_t	650
Maximum absolute value of compressive stress in x-direction	X_C	555
Maximum tensile stress in y-direction	Y_t	650
Maximum absolute value of compressive stress in y-direction	Y_C	555
Maximum tensile stress in z-direction	Z_t	10
Maximum absolute value of compressive stress in z-direction	Z_C	500
Maximum absolute value of shear stress in xy-plane	S_{12}	181
Maximum absolute value of shear stress in yz-plane	S_{23}	132
Maximum absolute value of shear stress in zx-plane	S_{31}	132

Two methods of composite modelling are considered. In the first one „14w” only one ply is modelled as 3D finite element layer. In the second one „7w” two plies are modelled as one 3D element layer. The dimension of the last model is twice as small as „14w” therefore less computer time consuming.

3. Laminated composite

3.1. Laboratory tests

The composite specimens for static and fatigue tests are made of CE 8201-245-45/120 prepreg (KDK8042 carbon fabric impregnated with E201 epoxy resin) shown in Fig. 1 which is characterized by the same volume fraction and mechanical properties of fibres in the warp and weft directions (an orthotropic material). The mechanical properties of the composite were determined in accordance with ASTM standards.



Fig. 1. a) piece of the prepreg, b) specimen

The equipment and the production method of the semi-finished product to make test specimens are shown in Fig. 2. After four prepreg layers had been laid up in the space of gripping area ten prepreg layers were laid up shaping the gauge and gripping areas of the specimens. The curing process was carried out in the period of two hours at the temperature of 130 C and under the required pressure. Next, the semi-finished product was cut and ground in order to obtain the required width of the specimens.

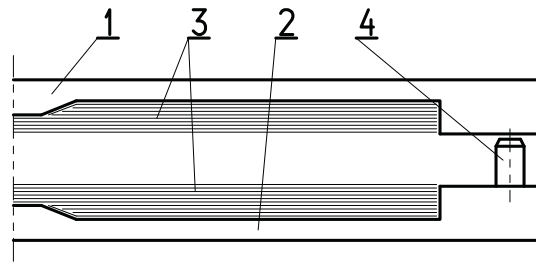


Fig 2. Method of production: 1 – top mould, 2 – bottom mould, 3 – prepreg layers, 4 – dowel pin

The INSTRON 8802 universal testing machine was used for determining the material properties of the composite. The composite material characteristics are contained in Tab. 2.

Tab. 2. Orthotropic elastic constants of 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}
58 093	58 093	9759	3545	2564	2564	0.0154	0.5356	0.1575

3.2. Numerical analyses

Numerical analysis for specimen of the laminated composite loaded with tensile force equal 45 000 N is performed. Two models of the joint are prepared. Only a half of the specimen is modelled due to symmetry. The shape, overall dimensions and coordinate system as local coordinate system in the each ply for model of the tensile specimen with length equals 100 mm are shown in the Fig. 3a. Thickness of the element layers for model „7w” is presented in Fig. 3b and for „14w” one in the Fig. 3c. Orientation of each ply is aligned with the direction of the tensile load.

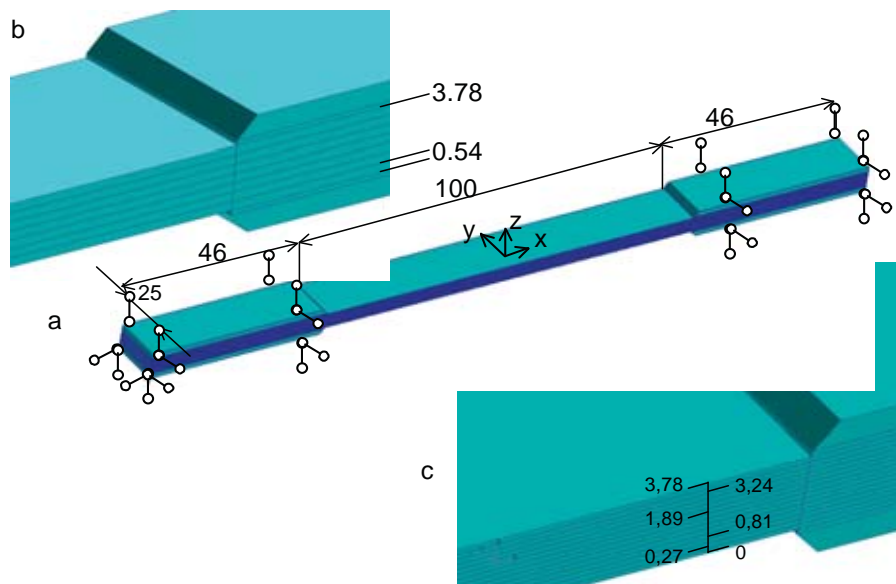


Fig. 3. Dimensions of the specimen: a) overall model, b) model „7w”, c) model „14w”

Numerical analyses results are compared with laboratory tests. The longitudinal strains versus load value for two models are shown in Fig. 4. The graph in this figure indicates that modelling technique and iteration procedure which were applied in the numerical analyses were appropriate to describe laminated composite behaviour.

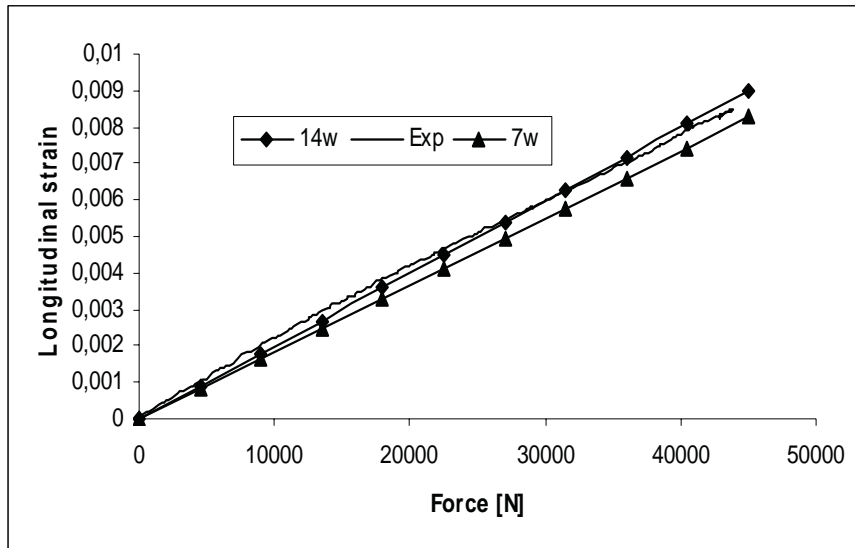


Fig. 4. Comparison of the results of the laboratory tests and numerical analyses for models

The distribution of equivalent HMH (Huber-Mises-Hencky) stress on the symmetry plane for „7w” and „14w” models are shown in Fig. 5. This figure presents the stress change through the composite thickness and stress concentrations in the grip end.

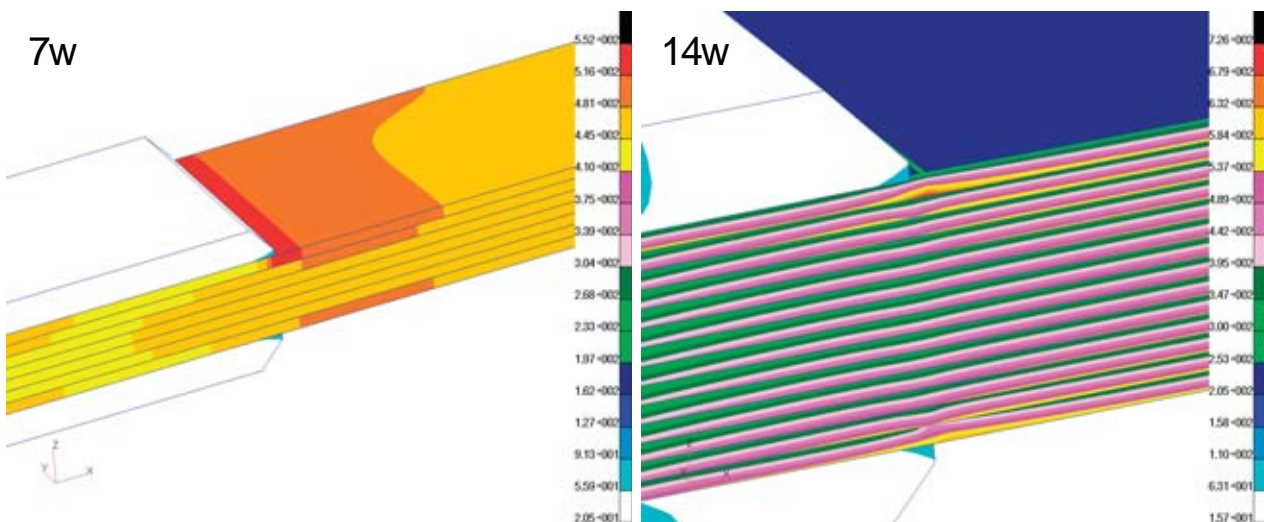


Fig. 5. Equivalent HMH stress on the symmetry plane for model „7w” and „14w”

The highest equivalent HMH stress is inside laminated composite, between first and second ply for „14w” model (Fig. 5b). For „7w” model this area is at the end of the grip region, outside the composite part. Modelling method applied at the „14w” caused non-physical twice stress growth through one ply.

4. Metal-composite joint

4.1. Laboratory tests

A metal-composite joint is consisted of two triple stepped metal parts which are made of St3S steel and the fourteen prepreg plies. The prepreg is co-cured to the metal. The step width is 12 mm, it height is 0.54 mm and corresponds to the thickness of two prepreg plies. The photograph of the specimen is shown in Fig. 6.



Fig. 6. Tensile of the metal-composite specimen

Depending on the step depth one or two prepreg layers with different individual orientations are attached to the metal. The layers configurations in the symmetry plane shown in Tab. 3, is very often applied to the aircraft wing skin and tail-plane. The angle between fibre orientation and global axis x is presented in the line „orientation”. Layer numbering starts from the neutral axis of the specimen (Fig. 7).

Tab. 3. Composite layers orientation

Layer number	w1	w2	w3	w4	w5	w6	w7
Orientation	0°	45°	0°	45°	0°	45°	0°

4.2. Numerical analyses

Tensile test of the metal-composite joint is modelled and numerical analysis of it is performed. The value of the tensile load is 25 988 N. Two methods are used to model composite part. Two prepreg plies are modelled as one layer of the three-dimensional lower-order finite element HEX (Fig. 7b) just us in the model „7w” in the model of the laminated composite. One 3D element layer is corresponded to one prepreg ply in the model „14w” (Fig. 7c) in the second one. Composite property is attributed to brick elements which are used in the models. Three-dimensional finite elements HEX are used to create model of the metal parts. Isotropic elastic plastic material is applied to this case. Due to symmetry only a half of the joint is modelled. The adequate boundary conditions are applied (Fig. 7). The shape, overall dimensions and boundary conditions are shown in Fig. 7. The names „g12”, „g24”, „g36” and „middle” indicate positions of the gauges presented by orange lines. The universal testing machine grips are modelled by degrees of freedom in the plane perpendicular to the surface of the metal lap (Fig. 7).

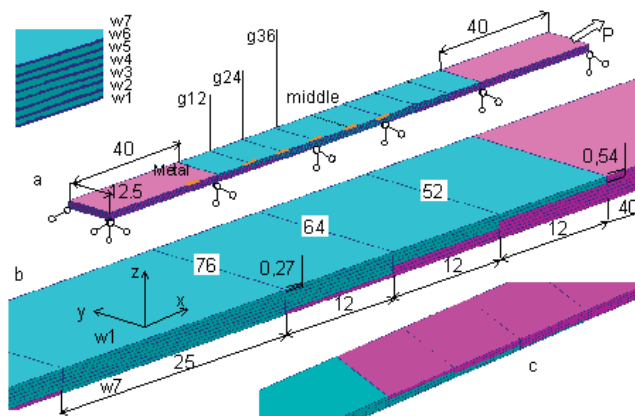


Fig. 7. a) overall dimension, boundary condition, b) model „14w”, c) model „7w” for metal-composite specimen

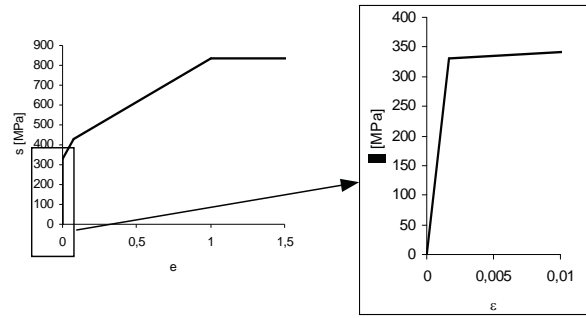


Fig. 8. The stress-strain relationship for steel

Material parameters for metal – St3S steel – and composite were obtained during laboratory tests. Young modulus $E = 197$ GPa and Poisson’s ratio $\nu = 0.3$ and stress-strain curve (Fig. 8), with the yield stress 331 MPa, are applied to describe the material model of the metal.

The orthotropic elastic constants for the composite are shown in Tab.2. The failure mechanism-based on progressive damage analysis is postulated for analysing the tension failure metal-composite joints. To estimate the joint failure the maximum stress criterion is used. The failure material parameters are contained in Tab. 1.

Values of the longitudinal strains versus load from laboratory test are compared with results of the numerical analyses (Fig. 9). The graphs for model „14w” are presented in the left column and for „7w” one in the right. The names of the graphs are corresponded to gauges in Fig. 7.

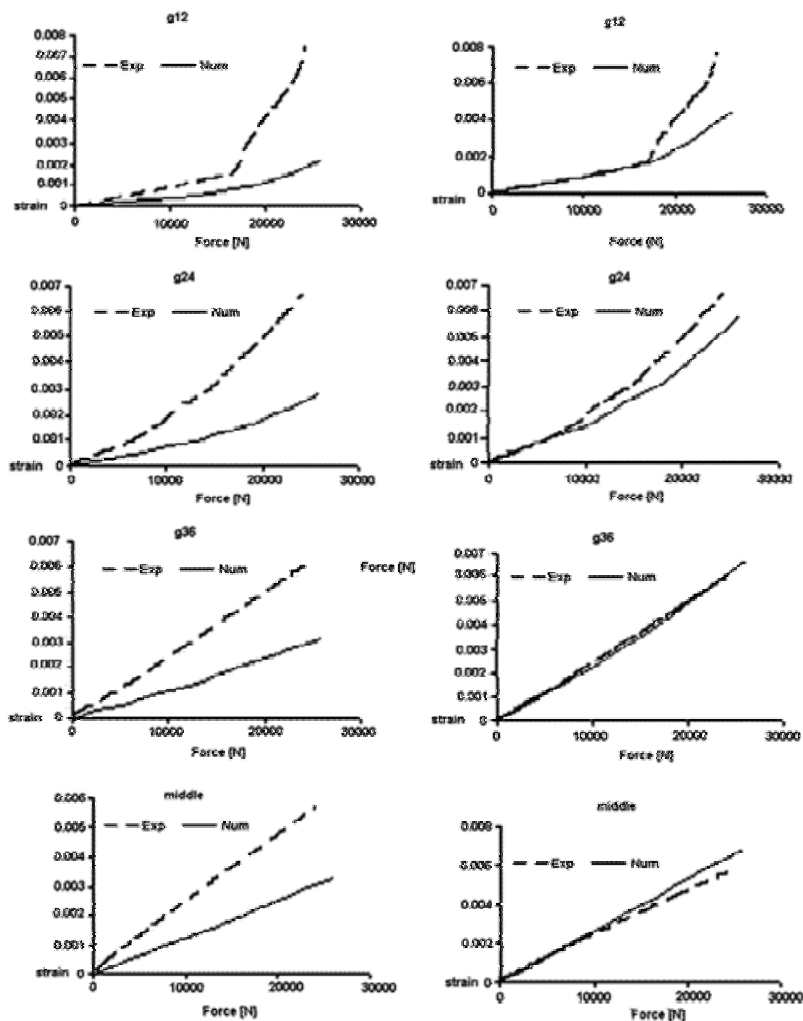


Fig. 9. Comparison of the results of the laboratory tests and numerical analyses

Distributions of the magnitude displacement and deformation caused by maximum load for both models are presented in Fig. 10. The same scale for each model indicates that displacement value does not depend on the modelling method of the laminated composite. Debonding between composite and steel is shown in the enlarged fragment of the specimen.

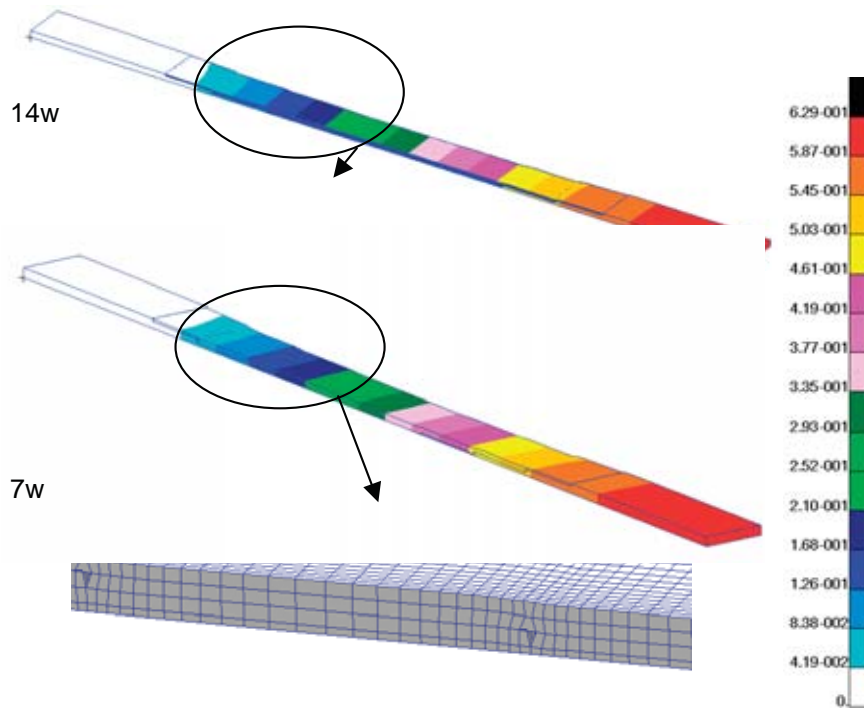


Fig. 10. Distribution of the magnitude displacement and joint deformation

Distributions of the magnitude strain (calculated as equivalent HMH strain) are presented individually for steel and composite parts due to presentation clearance (Fig. 11, 12). Distributions for metal parts are named „steel” and for composite one – „composite”.

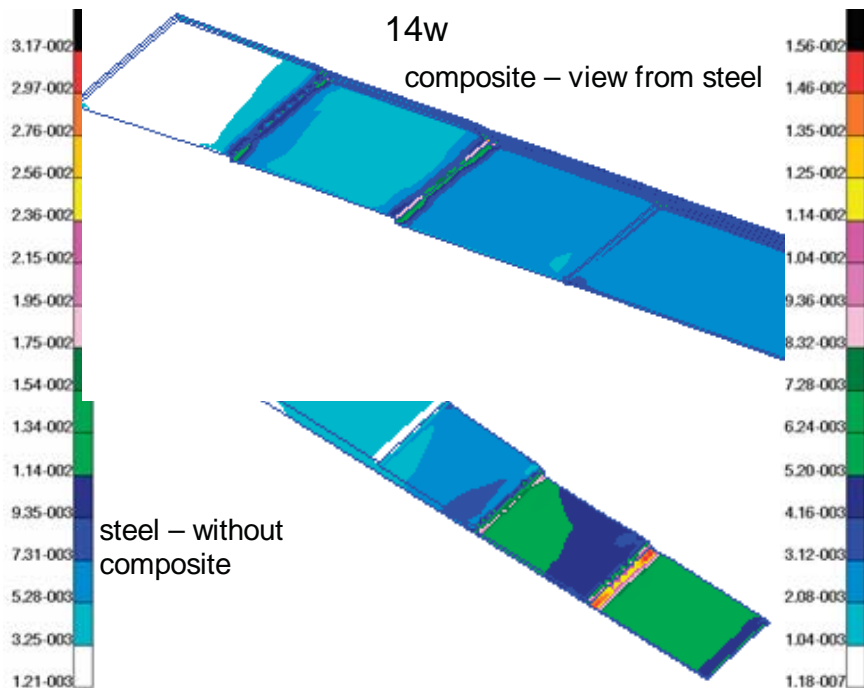


Fig. 11. Distribution of the magnitude strain for model „14w”

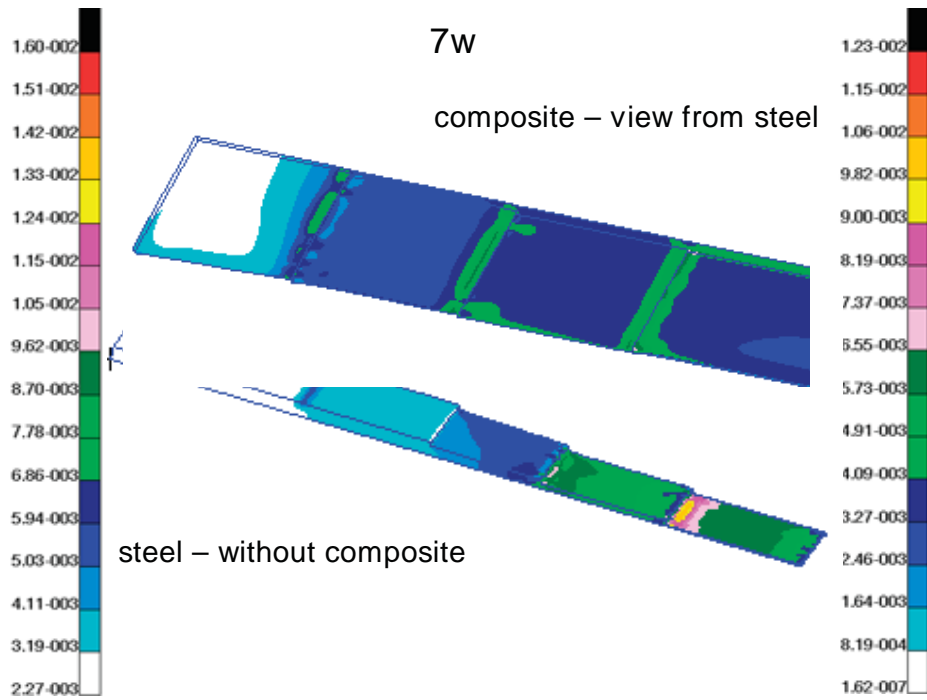


Fig. 12. Distribution of the magnitude strain for model „7w”

Distributions of the equivalent HMM stress for metal and composite parts separately for „14w” and „7w” models are presented in Fig. 13 and Fig. 14, respectively. The stress and strain concentrations occur at the step edge independently on model. Solving procedure of the contact problem in the MSC.Marc induces points stress concentrations. The values change of the equivalent HMM stress at the vertex nodes of the laminated composite in the symmetry plane versus load for „14w” and „7w” models are presented in Fig. 15a and Fig. 15b, respectively. The marks in the legends of the graphs correspond to x coordinate of the nodes on the step edge which are shown in Fig. 7.

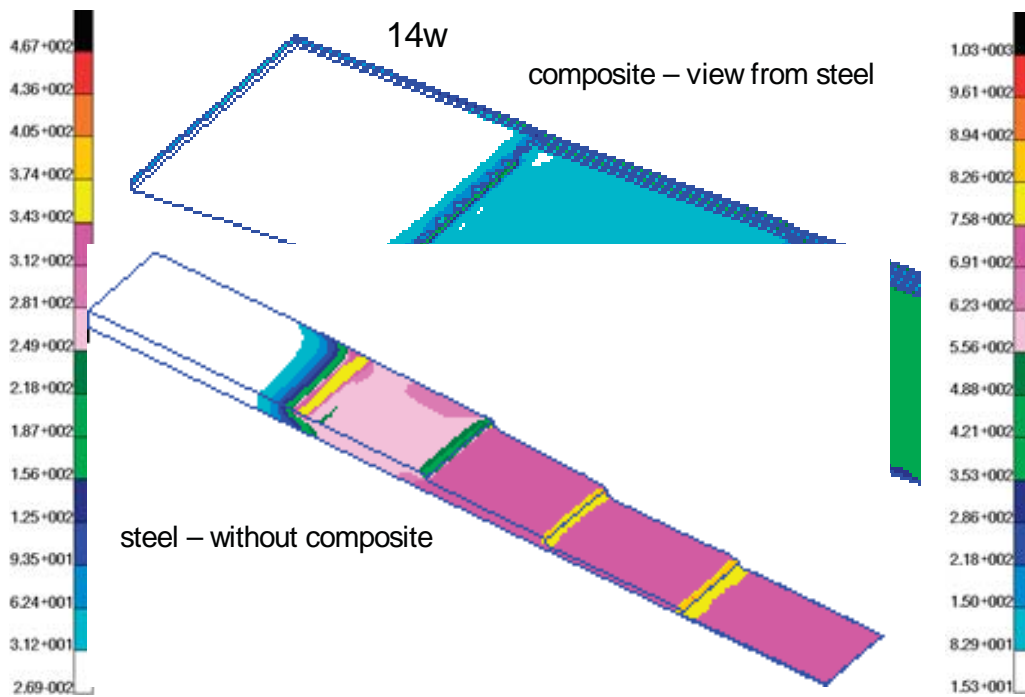


Fig. 13. Distribution of equivalent HMM stress for composite and step parts for model „14w”

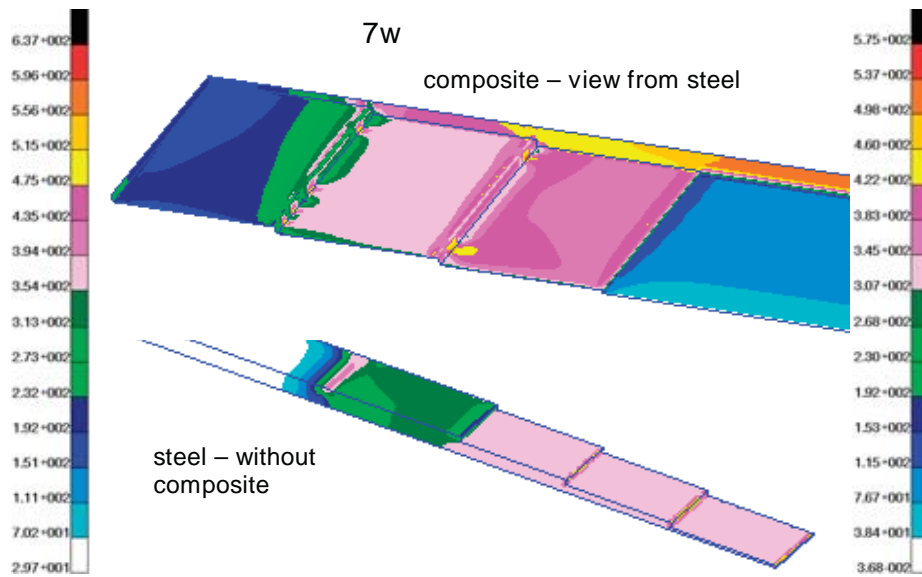


Fig. 14. Distribution of equivalent HMH stress for composite and step parts for model „7w”

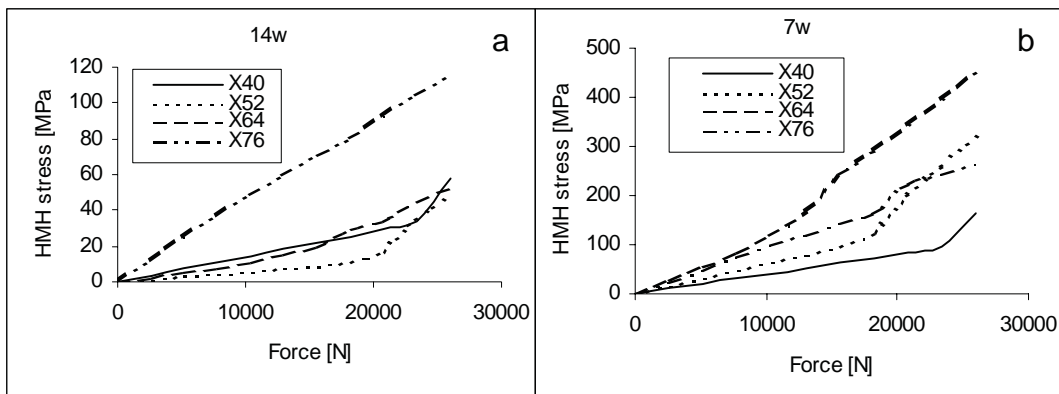


Fig. 15. Values of equivalent HMH stress in the composite vertex for „14w” (a) and „7w” (b) models

The highest equivalent HMH stress occurs at the vertex of the composite second step for „7w” model.

The yield in the third step of the metal part appears at the joint due to its too small thickness (Fig. 16).

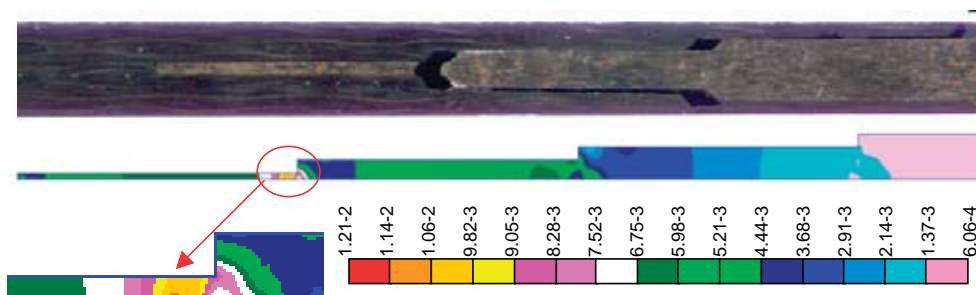


Fig. 16. Equivalent HMH strain of the metal-composite joint

5. Conclusions

Numerical analyses enable to describe the local and global failure form of the composite and metal-composite specimens (mainly on the step edges).

Taking into account displacement it is difficult to observe any particular difference in the numerical results for discussed models (Fig. 4). However, magnitude strain values are higher for

„14w” model than for „7w” one. Maximum values occur at the points of the local stress risers. They are equal 49.5% for steel and 21% for laminated composite. The equivalent HMM stresses at steel are lower about 18% at „14w” model than at „7w” one. For composite those values are higher about 40% at „7w” model than „14w”.

The step edges, where two materials are stucked, are the places with the local stress and strain risers. „7w” model is better to future calculation according to Fig. 9. Two prepreg plies modelled as one 3D element layer which is attributed to composite property enable to obtain accurate results with decreasing model size and saving computer time.

The advantages of the presented method are application of standard software tools and obtaining stress discontinuity on the interface between metal and composite due to the frictional contact modelling. 3D finite elements and composite modelling method applied to numerical analysis are useful to determine the strength of the triple stepped-lap metal-composite joint in the global scale.

References

- [1] Broughton, W. R., Crocker, L. E., Gower, M. R. L., *Design Requirements for Bonded and Bolted Composite Structures*, NPL Report MATC(A)65, United Kingdom 2002.
- [2] Derewońko, A., Niezgoda, T., Godzimirski, J., *3D Numerical Investigation of Tensile Loaded Lap Bonded Joint of Aircraft Structure*, Journal of KONES, Vol. 13, 2006.
- [3] German, J., *Podstawy mechaniki kompozytów włóknistych*, Cracow University of Technology, Cracow 1996.
- [4] Jaeger, J., *New Solutions in Contact Mechanics*, WIT Press, 2005.
- [5] MSC.Marc ver. 2005, Vol. A, *Theory and User Information*, The MacNeal-Schwendler Corporation.
- [6] Muc, A., *Mechanika kompozytów włóknistych*, Księgarnia Akademicka, Cracow 2003.

"The author's reward was sponsored by Society of Collective Management of Copyrights of Creators of Scientific and Technical Works KOPIPOL with registered office in Kielce with duties obtained on the ground of the art. 20 and art. 20¹ of law on copyrights and related rights."