International Journal of Mechanical and Production Engineering, ISSN(p): 2320-2092, ISSN(e): 2321-2071
Volume- 6, Issue-5, May-2018, http://iraj.in

PRELIMINARY STUDIES ON THREE DIMENSIONAL STRESS ANALYSIS OF FASTENER JOINT IN COMPOSITE LAMINATE

[1]SHABBIR AHMED R.M, [2]MOHAMED HANEEF

[1]Nizwa College of Technology Nizwa, Oman
[2]Ghousia College of Engineering, Ramanagaram, India

Abstract - In this work, we have conducted a preliminary study on three-dimensional stress analysis of the composite laminate using ANSYS Parametric Design Language (APDL) scripting. So here we will concentrate only on a simple problem of a hole field with a rigid pin in a finite composite plate under the plate load and pin load systems and then extend the technique for three dimensional cases. A composite laminate having circular hole with circular pin has been used to understand the mechanics of load transfer in a pin joint. The commercial software ANSYS has been used for the contact stress analysis problem as the software is capable of handling problems of pin-joints successfully.The interface is assumed to be smooth and the absence of friction simplifies the problem by making the shear stress uniformly zero on the interface whether the contact is full or partial.

Keywords - Composite materials; Finite element analysis; Contact stresses.

I.INTRODUCTION

Composite structures are finding lot of applications in the field of automotive and aerospace applications. But the most important aspect of the composite structure is decided by the joints as most of the structural properties are depended on it. The joints are essential to connect different components of a structure or sub-assemblies of similar or dissimilar materials. The most commonly used joints used to join the composite structures are either adhesively bonded joints or mechanically fastened [1]. The presence of load carrying joints makes the characterization of stiffness and strength of the composite structures very complicated. The behaviour of the load carrying joints depends upon various factors like materials, type of joint, design parameters and the geometry associated with it. Most of the properties of composite material are decided by the properties of individual materials used like type of the fibre and matrix used. The material parameters like type of fibre, resin type, surface treatment of fibres, volume fraction of fibres and stacking sequence of laminates. Most commonly used composites are either carbon or glass fibre reinforced epoxy composites. The type of joint may be single lap, single cover butt or bolted joint. The design parameters are load direction, loading rate, type of load; static or dynamic, laminate thickness, clearance between hole and pin.It can be seen that there are so many variables the characterization of load carrying joints become impossible. So the behaviour of basic joints in limited number of composite structures and influence of most important parameters are studied in detail and from this the behaviour of other type of joints can be predicted [2-4].

Although there are numbers of experimental studies on mechanically fastened joints, but the analytical study on the two dimensional stress analyses, especially three dimensional stress analyses are few. Most models use linear or non-linear two dimensional finite element methods for the stress analysis in composite structures. Dano et al. [5]used two dimensional model developed by ABAQUS to study the deformation behaviour of pin loaded joint in a composite plate assuming fastener to be rigid. To predict bearing stress-pin displacement curve, progressive failure analysis of pin loaded composite plated were performed until joint failure occurred. The model proposed predicted the failure strength values of pin loaded composite plates were well within the 15% of experimental data. Gao and Noda [6] reported stress analysis of bolt loaded hole in piezoelectric composite laminates using complex potential method. It was assumed that the infinite composite plate with free circular whole was subjected to mechanical and electric loading uniformly. A general solution for an arbitrarily distributed mechanical and electric load on hole surface in the form of series by superposition of Green function for a generalized point load acting at the hole rim. Though the work presented a simple approach for stress analysis of bolt loaded hole in infinite composite plate but no attempt were made to compute analytical or experimental work was done in order to validate it.The research work on three dimensional stress analysis of pin or bolt loaded composite laminates have also been carried out. For example, Tserpes et al. [7] investigated the effect of failure criteria of bolted joints using parametric finite element analysis based on three dimensional progressive damage models. The predicted load-displacement curves and failure loads were compared with the available experimental results of single-lap single-bolt joints. The stiffness of the joint predicted by the model was found to be in utmost accurate agreement with all the configurations of different joint geometries and laminate stacking sequence. The joint strength analysis comprising of Maximum stress and Hashin-type failure criteria showed an agreement of 7% of the experimental results. Yang et al. [8] presented the three dimensional stress analyses along with effects of friction and bearing force for composite laminates with an elastically pinned hole using multilayer boundary element method.The work with the help of contact maps and stress states around the pinned hole showed the importance of three dimensional approaches to study the underlying physics. Wang et al. [9] reported the progressive failure analysis of single lap bolted joint based on extended finite element method using linear elastic properties. MATLAB code written by the author was used to calculate the three dimensional laminate equivalent material properties. Subsequently experimentation was also carried out using HTA/6376 carbon fibre epoxy composite based laminates. The variation in failure load of joint by the analytical model and experiment was about 12.7%. Further increase in bearing strength of joint was observed with the increase in width-to-hole diameter ratio.

It is proposed to study the three dimensional analysis of pin-plate problem of composite laminate could be very fruitful. The proposed work is an effort in three dimensional stress analysis of pin joints with round pin in a hole in a composite plate using commercial software ANSYS as a tool for finite element analysis.

II. PROBLEM DESCRIPTION

A circular hole in a composite plate with a circular pin is the basic configuration considered for study to understand the mechanics of load transfer in a pin joint. The modes of load transfer are pin bearing and plate loads. The pin in the plate could be of interference, push or clearance fit, depending on the pin diameter being greater than, equal to or less than the hole diameter. The pin-plate interface exhibits a state of partial contact in all the fits above certain load level on the joints. The extent of contact and separation change with load level leading to a non-linear problem in misfit (interference or clearance) pins. In push fit the problem is linear with invariant (but unknown) extents of contact/separation.

The contact stress problem at the pin – plate interface can be analysed by iterative or inverse methods of approach. In iterative approach, the extents of contact/separation are determined iteratively for a given load level. Inverse technique determines the load level required for various extents of contact/separation. This technique has an extensive potential to generate vast parametric data with limited computational effort, but is applicable only for problems where the initiation of contact/separation and the nature of their variation are known. For anisotropic composites such an approach could be too complicated. The major hurdle in the use of iterative approach is that it could be expensive for parametric study. In most of the earlier works special software is developed for the analysis of two dimensional pin-joints using either iterative or inverse technique, this software’s serve specific purpose of the requirements of the problem. Almost all the software’s develop for the analysis of pin-joints that are available in literature for two dimensional analyses. And also they do not take in to account the effect of friction at pin plate interface. In the present study it is proposed to use commercial software ANSYS for the first time for contact stress analysis problem in general and pin-joints in particular. The ANSYS software is capable of handling problems of pin-joints successfully. This software has capability of considering effect of friction on pin plate interface using friction elements, in cases where ANSYS does support it is proposed to develop a code (software) which is capable of handling three dimensional problems of pin joints[10,11].

As a necessary first step to validate the ANSYS software, several problems of pin-joints in isotropic and orthotropic plates, for which load contact relations and stress distributions are available in literature will be analysed and the results will be compared. We introduce our three dimensional analysis of pin joints in composites first by studying the load transfer through a basic configuration of a round pin in a round hole in a finite orthotropic plate. The interface is assumed to be smooth. The absence of friction simplifies the problem by making the shear stress uniformly zero on the interface whether the contact is full or partial. Also in the absence of friction, partial contact problems for clearance and interference fits are mathematically identical. The effect of geometry on load contact behaviour, radial and hoop stresses will be studied. We start with a simple problem of a hole field with a rigid pin in a finite composite plate under the plate load and pin load systems and then extend the technique for three dimensional cases.

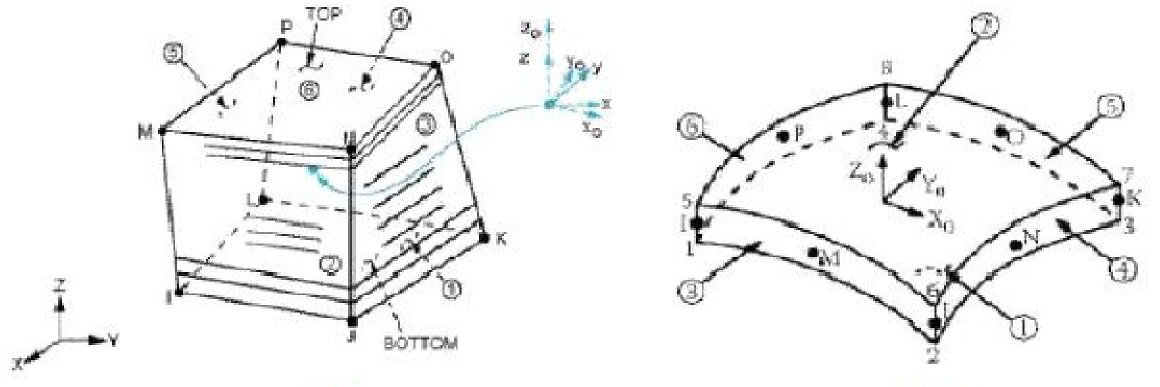
The next stage is to use the ANSYS software for finite element analysis to predict the strength of fastener joints in laminated composite plates. Experimental results available in literature survey on strength of fasteners joints in composites will be utilized for this study and stress analysis will be carried out on the same configurations used in those experimental programs using three dimensional analysis. The strength prediction will be based on the lowest of the tension, bearing and shear out modes of failure.The three dimensional analysis of pin joint considering the effect of friction on pin plate interface could lead to better understanding of free edge delamination. The effect of friction on load contact behaviour and maximum stresses will be demonstrated using ANSYS software for finite element analysis.

III. NUMERICAL MODELLING AND ANALYSIS

The basic configuration of three-dimensional model of the composite laminate has been generated in ANSYS using ANSYS Parametric Design Language (APDL) scripting. The APDL program is written such as the following important parameters can be varied at any point of time in the study:

a) Number of layers
b) Lay-up sequence
c) Size of the laminate
d) Thickness of each layers
e) Material property of each of the layers present in the model

For finite element modelling of solid structures SOLID185 & SHELL 281 element as shown in Fig. 1 a & b is used. This model is defined by 8 nodal points. Each of these nodes has 3 DOF: translations in the nodal x, y, and z directions. The element input data includes the anisotropic material properties. The layered configuration is one of the most important characteristic of a layered laminated composite material. Each layer could be made of a different orthotropic material and at the same time can also have different orientations of principal directions.



(a) (b)

Fig. 1. SOLID 185 – Layered solid element available in ANSYS element library, a) Schematic showing the layered nature of finite elements b) SHELL 281 element

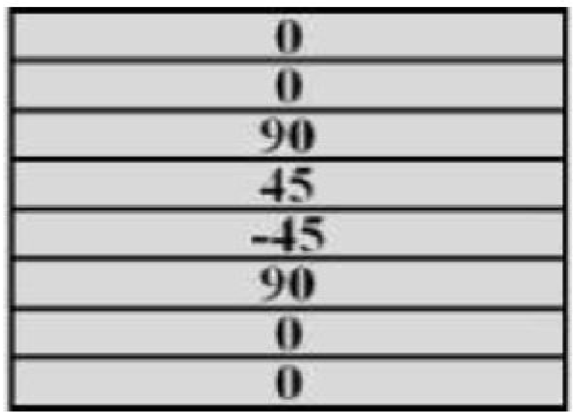


Fig.2. Lay-ups of three laminates considered for study - Anti-symmetric laminate (Note: All angles are in degrees)

A finite element model of the composite is built with the following parameters:

[1] Length = 0.1 m (100 mm)
[2] Breadth = 0.1 m (100 mm)
[3] Ply thickness = 0.000125 m (0.125 mm)
[4] Number of lamina = 8
[5] Laminate thickness = 8 * Ply thickness = 0.001 m (1 mm)
[6] Layup = [0/0/90/45/-45/90/0/0]
[7] The material properties considered are for glass-epoxy composite with the following data:
[8] Ex = 126.6 GPa, Ey = 8.7 GPa,PRxy = 0.32, PRyz = 0.47, PRxz = 0.32
[9] Gxy = 3.7 GPa, Gyz = 2.9 GPa, Gxz = 3.7 GPa
[10] Hole radius: 0.005 m (5 mm)
[11] Applied displacement: 0.0005 m (0.5 mm)

An initial analysis is carried to study the effects of contact pressure on the stress distribution in the laminate plate. A general asymmetrical lamina is considered for initial study.

IV. RESULTS AND DISCUSSION

The near-hole stresses in the composite laminate give an idea of mechanical behaviour of the composite laminate. Fig.3 shows an asymmetrical laminate [0/0/90/45/-45/90/0/0] lay-up is considered for study. Fig. 4 shows the geometry of the laminated composite plate considered for analysis. The figure shows the individual line numbers on which the boundary conditions will be applied during the analysis.Fig.5 shows the meshed model of the laminated composite plate. The elements considered for study is SHELL 281.

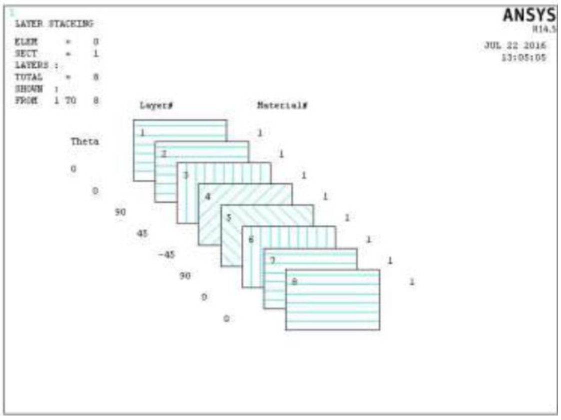


Fig. 3. Lay-up of the composite considered for study

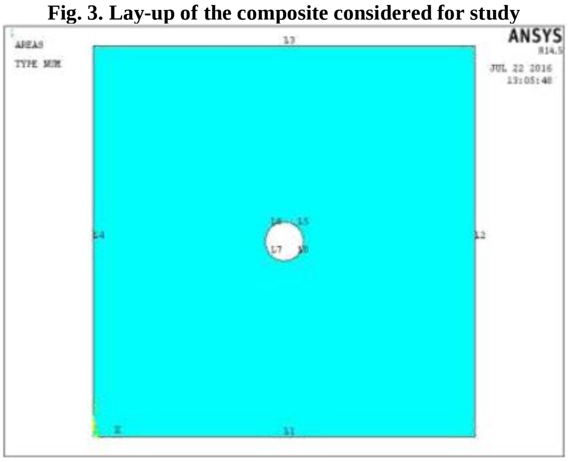


Fig. 4. Geometry of the composite laminate considered for study.

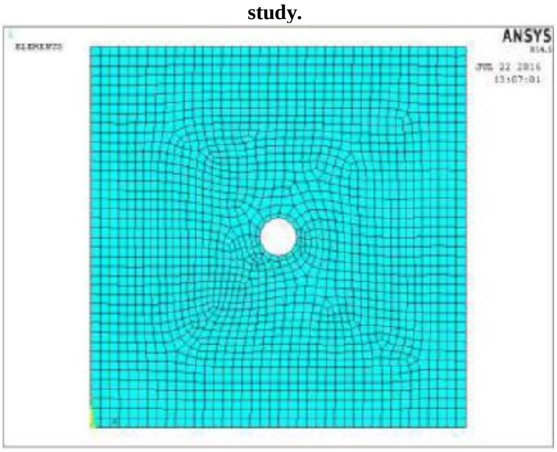


Fig. 5.Finite element model of the composite.

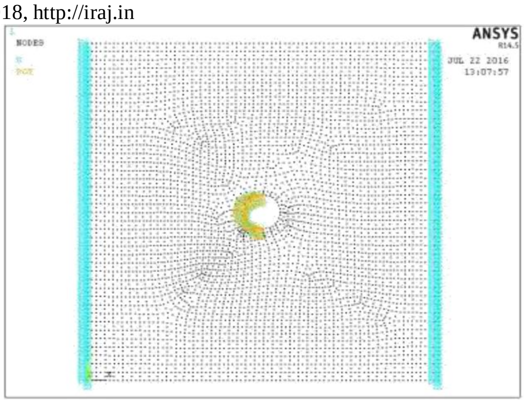


Fig. 6.Applied boundary conditions on the model.

Fig. 6 shows the boundary conditions applied on the finite element model. The nodes on the left edge of the model are applied with all the DOF constraints of zero. The part of the circle highlighted represents the no-slip region. And the other represents the no-contact region. The nodes on the right edge of the model are constrained from motion along y and z however, are applied with a displacement of 0.0005 m (0.5 mm) so as to produce tensile deformation of the member. Fig. 7 shows the definition of path on the model, along the length of which, variation of various properties such as nodal deformation and stresses can be studied in the form of a graph. The path is considered along the laminate-pin interface for push fit analysis under plate loading.

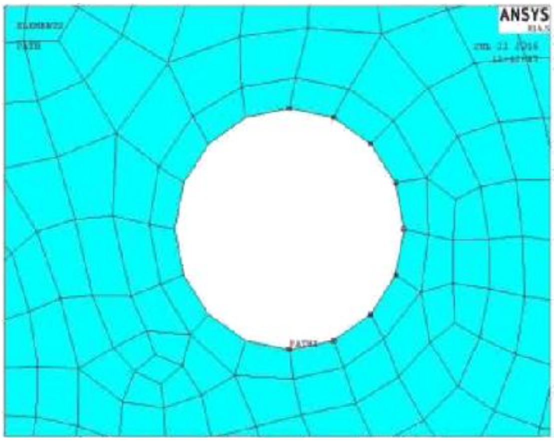
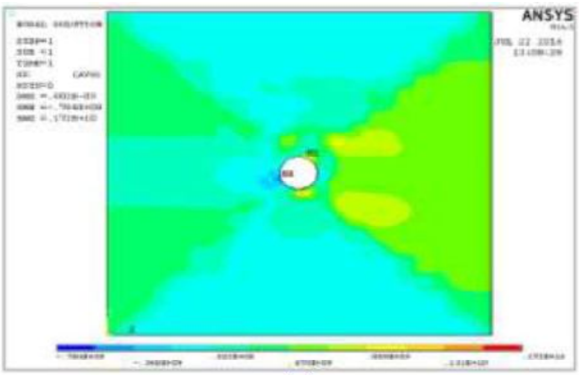
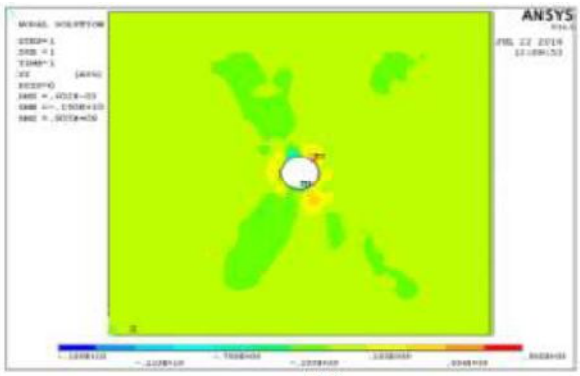
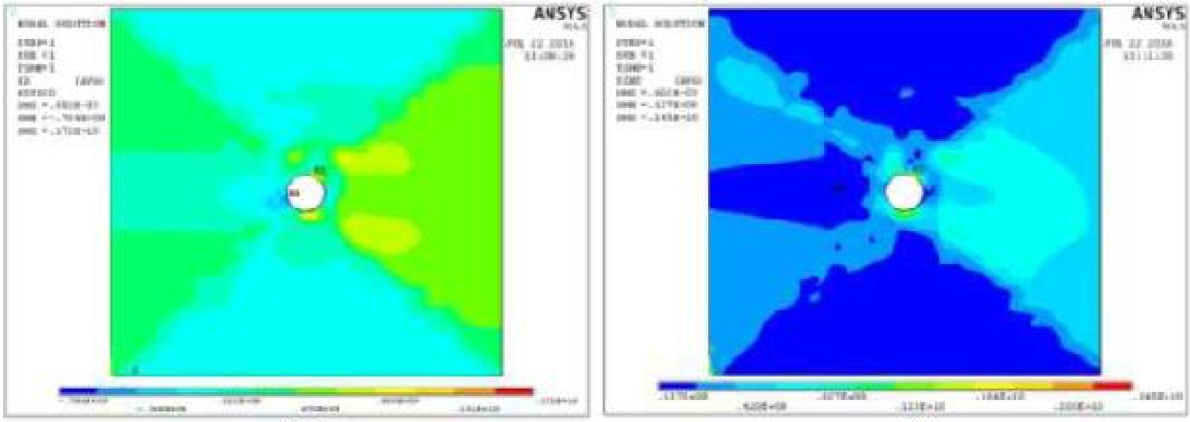


Fig. 7. Definition of path along which data is measured.







(c) (d)

Fig. 8. Distribution of stresses in the model. (a) óxx, (b) óyy, (c) óxyand (d) Stress intensity in (all in N/m2)

Fig. 8 shows the variation of stresses in the model. The stresses are found to be maximum at the interface of pin and laminated composite. A contact analysis is not performed in this analysis. However, the analysis is carried out to get an insight of the stress distributions.

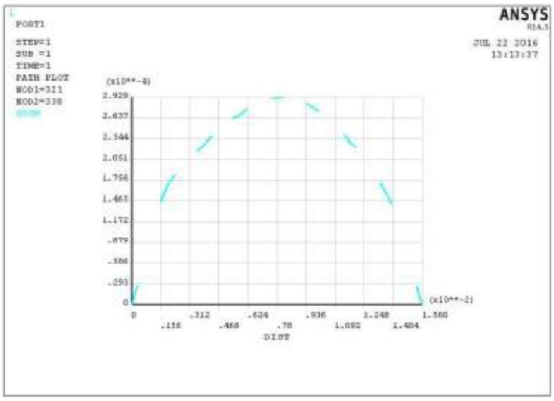


Fig. 9. Variation of displacement vector sum along the defined path.

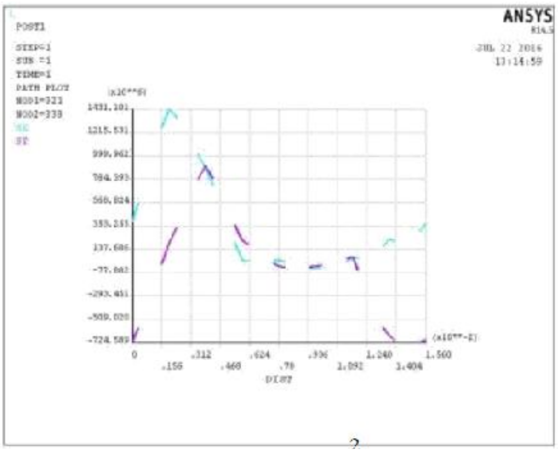


Fig. 10. Variation of óxx and óyy(in N/m2) along the defined path.

On the path defined which can be seen in Fig. 7, the variations of total displacements and normal and shear stresses have been mapped onto. The Fig. 9 and 10 shows the variation of the displacement vector sum and the stresses along the circumference of the hole starting from topmost point to the bottom point in the clock-wise direction. Thus in the figure (b), the lowest value of the x-axis represents zero degrees while the maximum value of the x-axis represents 90 degrees. As expected, stresses do show a variation with angle. A plot of radial stress and hoop stress would show the exact variation of the stresses with the angle.

CONCLUSIONS

A simple and effective method to predict the contact stresses in the composite laminate consisting of circular hole with a rigid pin has been carried out. The analysis results indicate the stresses to be dependent on the angle of contact.The results indicate that the simple analysis will not suffice and that the contact analysis has to be performed in order to get accurate results which describe the stress distribution accurately.

REFERENCES

[1] McCarthy, M.A., “BOJCAS: bolted joints in composite aircraft structures”, Air & Space Europe 3 (2001), 139-142.

[2] Waszczak, J.P., Cruse, T.A., “Failure mode and strength predictions of anisotropic bolt bearing specimens”, Journal of Composite Materials 5 (1971), 421-425.

[3] Jong, T de., “Stresses around pin-loaded holes in elastically orthotropic or isotropic plates”, Journal of Composite Materials 11 (1977), 313-331.

[4] Godwin, E.W., Matthews, F.L., “A review of the strength of joints infibre-reinforced plastics: Part 1. Mechanically fastened joints”, Composites (1980), 155-160.

[5] Dano, M.L., Gendron, G., Picard, A., “Stress and failure analysis of mechanically fastened joints in composite laminates”, Composite Structures 50 (2000), 287-296.

[6] Gao, C.F., Noda, N., “A general solution for a bolt loaded-hole in piezoelectric composite laminates”, Composite Structures 69 (2005), 442-448.

[7] Tserpes, K.I., Labeas, G., Papanikos, P., Kermanidis, T., “Strength prediction of bolted joints in graphite/epoxy composite laminates”, Composites Part B 33 (2002), 521-529.

[8] Yang, B., Pan, E., Yuan, FG., “Three-dimensional stress analyses in composite laminates with an elastically pinned hole”, International Journal of Solids and Structures 40 (2003), 2017-2035.

[9] Wang, Z., Zhou, S., Zhang, J., Wu, X., Zhou, L., "Progressive failure analysis of bolted single-lap composite joint based on extended finite element method", Materials and Design 37 (2012), 582-588.

[10] Aluko, O., Whitworth, H.A., "Analysis of stress distribution around pin loaded holes in orthotropic plates", Composite Structures 86 (2008), 308-313.

[11] Kapti, S., Sayman, O., Ozen, M., Benli, S., "Experimental and numerical failure analysis of carbon/epoxy laminated composite joints under different conditions", Materials and Design 31 (2010), 4933-4942.

★★★