Stress Analysis in an Unidirectional Carbon/Epoxy Composite Material

Dragan D. Kreculj
PhD student
University of Belgrade
Faculty of Mechanical Engineering

In this paper the stresses in the models from unidirectional carbon/epoxy composite material are studied. Software packages, based on the Finite Element Method (FEM), can be used in order to predict stress distribution on the examinated model. In that way, characteristic places with maximal stresses on the same model are determined. Structure damages in the model and eventually fracture will be created at first on these locations. In a report, stress tests in the corresponding models from composite material at strain on traction, shear and pressure are presented. For that purpose, specific commercial program packages MSC Patran and Nastran are used. By comparison the supposed and the obtained stress values it is concluded that there are insignificant deviations between them. The results of simulations show that the application of these programs is justified for this kind of problem.

Keywords: *composite materials, stress analysis, finite element method.*

1. INTRODUCTION

An introduction of new technology follows the development and use of modern materials. Today, composite materials are the subject of an intensive development and use. These materials have significantly better mechanical and other characteristics than their constituent elements.

Properties of composite materials that make them more specific than other materials are: large strength, high stiffness, small density and mass, resistance to corrosion and high temperatures, the ability to create complex shapes. Most of composites are created in order to improve the combination of mechanical characteristics of materials, such as stiffness, toughness and strength in conditions of environment influences, or at higher temperatures. These materials also have a considerable potential for absorbing kinetic energy during crash [1]. The ability of these materials to meet the specific needs for different structures makes them highly desirable. Improvement in design, materials and manufacturing technology enhance the application of composite structures.

Because of their extraordinary mechanical properties composite materials have an important application in aircraft constructions. The technology has been explored extensively for aerospace applications, which require high strength and stiffness to weight ratio [2]. Carbon/Epoxy composites are the most used composite materials in primary structures of the aircraft. They are usually used in a form of multilayer composites (laminates). For the implementation of composite materials in aviation, the most important feature is their behaviour on the dynamic loads and resistance to fatigue [3]. Nowadays the amount of composite materials in modern aircraft constructions is increasing. For example, fifty percent of new Boeing 787 structure is made from composites.

Received: October 2008, Accepted: December 2008
Correspondence to: Dragan D. Kreculj
Faculty of Mechanical Engineering,
Kraljice Marije 16, 11120 Belgrade 35, Serbia
E-mail: kreculj@ptt.rs

Quantity and degree of damages created in aircraft structures may be different. Such damage may have little influence, but may be critical for the construction integrity and service life. For that reason, it is essential to determine the actual construction state because of damages.

Preventing failure of composite material systems has been an important issue in engineering design. The two types of physical failures that occur in laminated composite structures and interact in complex manner are intralaminar and interlaminar failures. Intralaminar failure is manifested in micro-mechanical components of the lamina, as fiber breakage, matrix cracking, and separating of the fiber-matrix interface. Generally, structures made from fiber reinforced composite materials are designed that the fibers carry the bulk of the applied load. Interlaminar failure, such as delamination, refers to separating of adjacent lamina. The possibility that intralaminar and interlaminar failures occur in structural components is considered a design limit, and establishes restrictions on the usage of full potential of composites. Due to the lack of through-the-thickness reinforcement, structures made from laminated composite materials and adhesively bonded joints are highly susceptible to failure caused by interfacial crack initiation and growth. The delamination phenomenon in a laminated composite structure may reduce the structural stiffness and strength, redistribute the load in a way that the structural failure is delayed, or may lead to structural collapse. Therefore, delamination is not necessarily the ultimate structural failure, but rather it is the part of the failure process which may ultimately lead to loss of a structural integrity [2].

The basic design characteristics of modern aircraft are the use of complex software packages, in both design and service. Most of these softwares are based on the Finite Element Method (FEM). With the help of these programs, it is possible to determine the accurate number of measurable places on the structure, with increasing stress values. This enables optimization of necessary measure equipment, which is integrated on aircraft and directly contributes to shorten the time of testing and project price reduction. The same kind of procedure provides an opportunity for tracking defects in structures during their service life within maintenance procedures.

2. STRUCTURAL ANALYSIS OF CONSTRUCTION

2.1 Basics

Structural analysis is the process by which the relevant data for the structure are obtained. These data can be the following: stresses, deformations, displacements, oscillations, etc [5]. It can be said for structural analysis that it presents an integral and important part of the design.

Methods for structural analysis are divided into: analytical, numerical and experimental. In modern design a wide range of software packages are used, such as CATIA, PRO/Engineer, Solid Works, Inventor, and others that enable fast work on different projects. For stress/strain analysis, a combination of programs MSC Patran and Nastran is often used. With adequate accuracy, these programs provide the opportunity to be back in the early stage of the project and get reliable information about the validity suggested dimension of anticipated constructive solutions. Advantages of the work with this software are numerous. First of all, they enable easy creation of models and options for their corrections. The perceived deficiencies of constructions can be easily removed and in this way the project costs become smaller. By the way, the time required for the accomplishment of some analysis is quite reduced. Software packages that have the possibility of structural analysis are based on the FEM [4].

The main task of stress analysis on aircraft structure consists of three parts:

- finite element modeling (MSC/PATRAN),
- finite element analysis (MSC/NASTRAN) and
- computer aided detail stress analysis (MSC/PATRAN).

They are presented in detail in chapter 3.

Stress analysis is a complicated and highly repeated routine work. Even an experienced engineer can not avoid taking much time in routine calculating. The finite element analysis processed by MSC/NASTRAN will produce a large amount of output (including displacement, internal load and stress), but the use of MSC/PATRAN can display and inspect the analysis results rapidly. The purpose of inspection is to evaluate the accuracy and reliability of the finite element model and the analysis results [15].

Stress calculation in a structure model is an important and not easy task. The situation becomes even more complex through a large number of models and procedures available today. In some cases even the same model is proposed, with a different outer appearance only [11].

Verification of damages in constructions represents the raw use of structural analysis. Correlation between the FEM data and existing conditions needs careful testing of boundary conditions, material characteristics, geometry, operating environment and damaged parts. Assumptions and approximations must be adjusted to a minimum. By comparison of obtained data from the models and the real structures, appropriate conclusions are derived [14].

2.2 Finite Element Method (FEM)

Finite Element Method (FEM) is a numerical method of structural analysis [6]. The basic idea of this method is a physical discretization of a continuum. This implies a dividing accounted domain (some structures) on the finite number of small dimensions and simple shapes, which represent the basis for all considerations. This makes a mesh of so-called "'finite elements"'. Over discretization of a continuum, one type of finite elements or combination of several types may be used. The finite elements are connected by common nodes, so that they make the original structure. Mesh generation is the division of a certain area on nodes and finite elements. Commercial software packages have a built-automatic division of the areas for the purpose of obtaining one faster as well as qualitative solutions. This is of big importance in large or very complex engineering tasks [8].

Theoretically viewed, the discussed domain has infinite degrees of freedom. With this method, such a real system is replaced by the model, which has a finite number of degrees of freedom. With a mathematical perspective, instead of differential equation system, which defines equilibrium state of the entire model, the FEM use provides a common system of algebraic equations. Move from a mathematical to physical domain, the entire process comes down to complicated (but solved) system equations. At certain conditions the loads act only in certain points of finite element, which are called nodes. On the basis of well-known displacements in nodes, determination of stresses in nodes can be done, as well as in other points of final elements, which enabled stress-strain analysis of structures. The basic task is to choose the model which best approximates the appropriate boundary conditions. In case of lack of exact criterion, which was largely a matter of engineering and intuitions, quality knowledge of the nature of discussed problems, the theory of finite element allows to get response to this very important question [7].

The FEM is used to find out: stresses and deformations in the complex and unusually shaped components; conditions of fluid flow around buildings; heat transfer through gases and in other applications. A complete model takes into account geometry components, used materials, load conditions, boundary conditions and other significant factors. Appropriate use of FEM permits that component is tested before it is made. Consecutive iterations of that part would be modified, in order to attain the minimum weight with supply of an adequate strength. To view the basic equations in the FEM, various methods are used [7].

The main advantage in structure projects by computer use is the possibility of simulations. In that way, the behaviour of structures in real working conditions is examined. The investigated model replaces the real construction with a certain accuracy [9]. Sometimes it was necessary to create a physical model to examine its properties. Today, most of the work on the design is done in virtual environment [6].

The finite element method is the dominant discretization technique in structural mechanics. It was originally an extension of matrix structural analysis, developed by structural engineers. The FEM has been used in every field, where differential equations define the problem [8]. The process implementation of the FEM, based on solving differential equations, is leading to the residue method (Galerkin method) or to the variation methods (principle of virtual work, the principle of minimum potential energy). In a deformable bodies mechanics various methods are usually used, since their use is relatively simple.

It can be said that the FEM solution process consists of the following steps:

- Divide structure into piece elements with nodes (discretization/meshing);
- Connect (assemble) the elements at the nodes to form an approximate system of equations for the whole structure (forming element matrices);
- Solve the system of equations involving unknown quantities at the nodes;
- Calculate desired quantities (e.g., strains and stresses) at selected elements.

The FEM is used in deformable bodies mechanics to solve various static and dynamic problems. There are linear and nonlinear FEM analyses. Linear FEM analysis is based on a few basic assumptions: theory of small deformation, material is linear elastic. Nonlinear MKE analysis takes into account the material nonlinearity and geometrical nonlinearity (large deformations) of a considered system.

3. SIMULATION TESTING AND RESULTS

In this paper, for the linear static stress analysis, software packages MSC Patran and Nastran are used. Program Patran has a role of pre-processor and post-processor too. In Patran (pre-processor) to test the model are defined: form, mesh elements, support conditions, materials and loads. Then, on the basis of these data, a stress calculation is accomplished by Nastran software. After that, program Patran (post-processor) shows the distribution of the appropriate stresses.

Simulation stress testings on the models from unidirectional (UD) composite material T300/976 Carbon/Epoxy are done and results are displayed. This material is commonly used in the aircraft constructions. Mechanical properties of that composite material are shown in Table 1 [10].

In the tests the following loads are applied:

- tension,
- shear and
- pressure.

As a consequence, there appear appropriate strains and stresses in the models. Stresses can cause certain damages in the model constructions.

Test specimens are typical plates of a rectangular cross-section with recommended dimensions. Dimensions for all three specimens are presented in Table 2.

Table 1. Mechanical properties of UD Carbon/Epoxy T300/976 composite materials

E_1 [GPa]	135.2
E_2 [GPa]	9.24
G_{12} [GPa]	6.28
ν_{12}	0.32
G_{IC} [J/m^2]	193
G_{IIC} [J/m^2]	455
Tension strength [MPa]	1475
Shear strength [MPa]	76.6
Pressure strength [MPa]	1307

Table 2. Specimen dimensions

	Tension	Shear	Pressure
Length [mm]	155	80	560
Width [mm]	25	20	30
Thickness [mm]	0.13	0.13	0.13

For a failure it is used Tsai Wu failure criterion. This is a tensor criterion, which is one of the most used failure criteria for composite materials. Mathematical formulation for this criterion is complex [12]. For only one layer and plane strain state the Tsai Wu criterion is simplified.

Force (load) calculation for all three tests is done according to the well-known mechanics equations [13], from knowledge of stress and surface cross-section. The numbers of nodes in the models are arbitrarily chosen. Limit stress are given in Table 1.

3.1 Tension test

Test model is loaded on tension. Dimensions of a specimen are outlined in Table 2. Mesh of elements on Figure 1 is shown and it consists of individual elements (5 mm × 5 mm). Displacement limitations in Figure 2 are presented. The following is introduction of material properties, creating composite with stacking sequence of laminate (only one layer for a simple analysis) and allotment feature elements to the model.

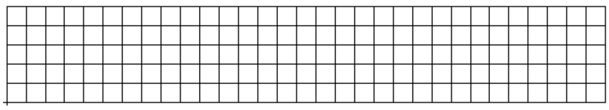

Figure 1. Mesh of elements

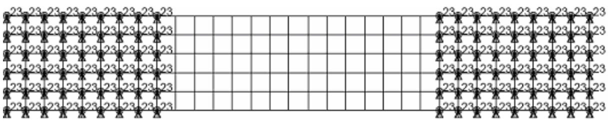

Figure 2. Displacement limitations

On the basis of Tsai Wu failure criterion, or stress at which it comes to break in the model, the value of the stress limit is equal 1475 MPa (Table 1). With this data and by calculated surface cross-section model (from data in Table 2), it can be concluded that the model will fail when force is 4794 N. When that force is scheduled to 54 nodes (chosen), it follows that force per node is 89 N. Forces on the model are shown in Figure 3.

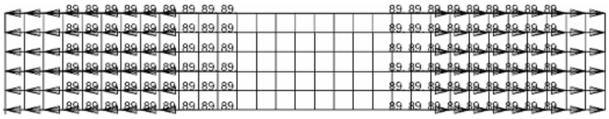

Figure 3. Forces in the model

Stress distribution on the model is shown in Figure 4. From that figure the place with maximal stress (1561 MPa) is clearly seen. The difference between this and the supposed value (1475 MPa) is slight.

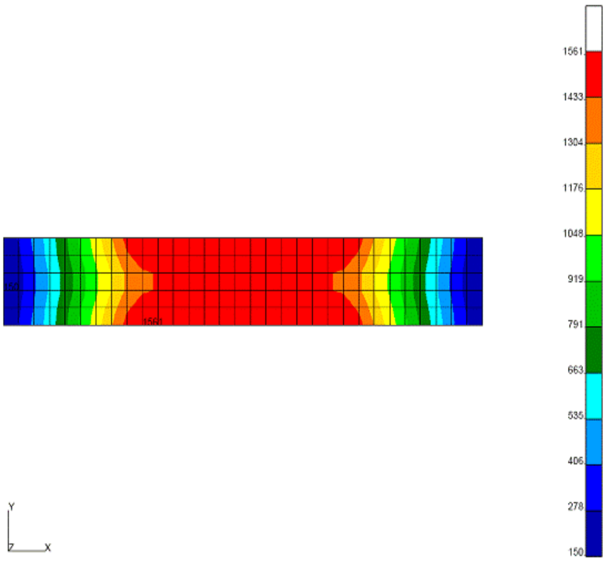

Figure 4. Stress distribution in the model

3.2 Shear test

Chosen test model is loaded on shear. Dimensions of a specimen are shown in Table 2. With an individual element (4 mm × 4 mm) is created mesh of elements (Fig. 5). Displacement limitations in Figure 6 are shown.

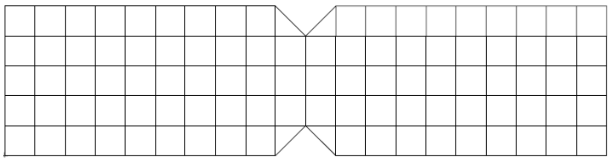

Figure 5. Mesh of elements

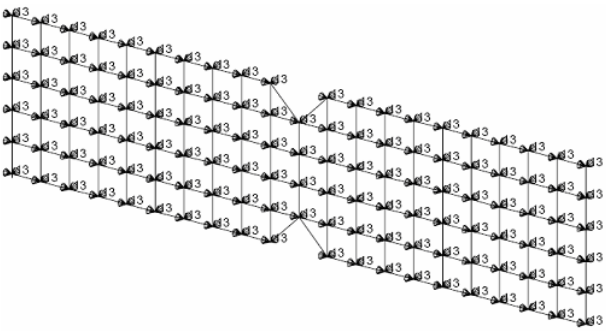

Figure 6. Displacement limitations

Then enter material properties, forming composite with stacking sequences of laminates (one layer) and association features to the model elements is made.

By Tsai Wu failure criterion, stress at which the model will break is 76.6 MPa (Table 1). Taking the above mentioned and surface cross-section model into account (calculated from data in Table 2), it follows that force for failure is 162 N. Since the number of nodes is chosen to be 60, therefore force per node is 2.7 N. Forces on the model in Figure 7 are presented.

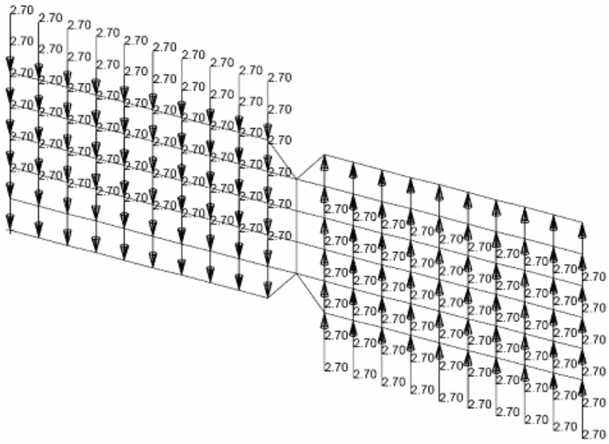

Figure 7. Forces in the model

In Figure 8 stress distribution on the model is shown. This figure shows the largest stress location (80.3 MPa). A discrepancy of this and the supposed value (76.6 MPa) is relatively small.

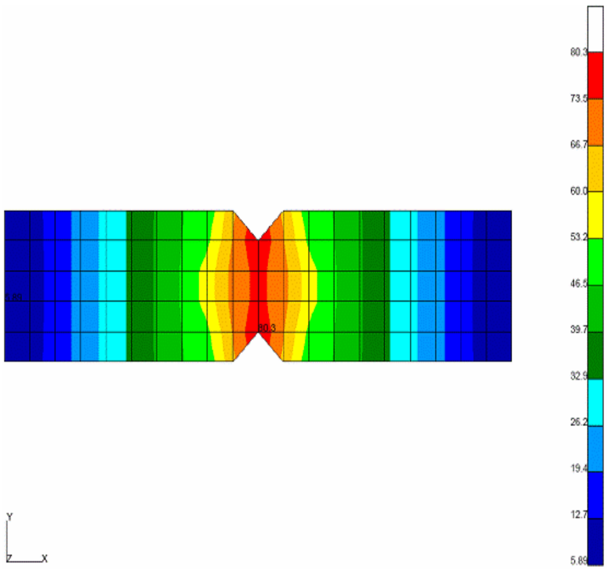

Figure 8. Stress distribution in the model

3.3 Pressure test

Appropriate test specimen is a typical plate of certain dimensions (see Table 2). This specimen is loaded on pressure. The model consists of singular elements (10 mm × 10 mm), and mesh elements are displayed in Figure 9.

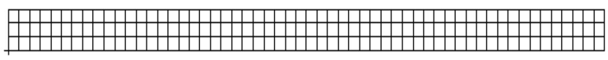

Figure 9. Mesh of elements

After that input material properties forming composite with stacking sequences of laminates (core and two layers) and adding features to the model elements is done.

On the basis of the Tsai Wu failure criterion, stress at which model will break is equal to 1307 MPa. With the cross-section surface model (calculated from data in Table 2), the model will fail when the force is 2040 N. It is chosen to be 8 nodes in the model. Force of 2040 N is distributed on 8 nodes and because of that the force per node is 255 N (Fig. 10).

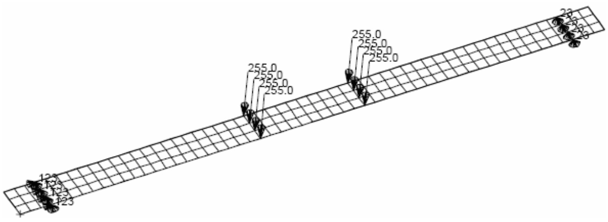

Figure 10. Forces in the model

In Figure 11 stress distribution on the model is shown. In this figure the most loaded point (1305 MPa) is precisely marked. It is clear that a deviation from the supposed value (1307 MPa) is low.

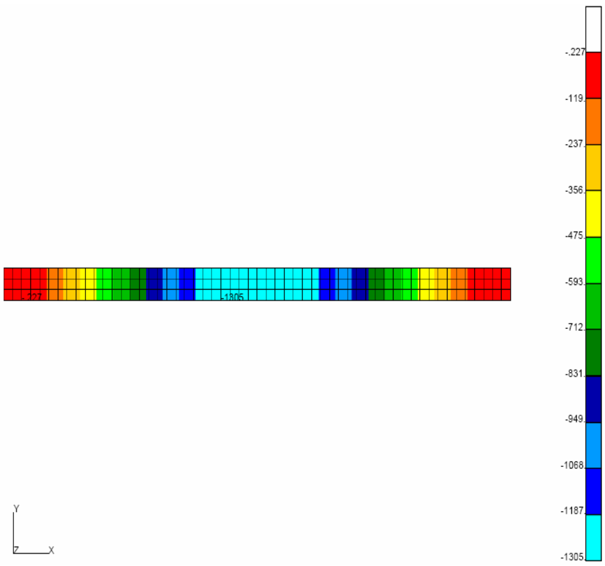

Figure 11. Stress distribution in the model

4. CONCLUSION

Damages in constructions from composite materials have different causes. They can occur in the production process or during exploitation of these structures. Such damages may have negligible influence, but they may be critical for the integrity and service life of constructions from composites. Therefore, it is extremely important to assess the state of constructions because of damages. It is necessary to predict accurately where damages will appear, how they will spread and when they will eventually produce a fracture and failure of constructions.

The development of modern military and civilian aircraft in the world and in our country requires a great amount of finance. As a result, there is a tendency to develop software packages, which could predict conditions on the aircraft constructions. These programs for structural analysis are mainly based on the finite element method (FEM). With these program packages on a formed model realistic conditions that construction was subjected to during service life can be simulated. The results of simulations offer the possibility of tracking and analysis of critical positions in construction during the design process, but also during its exploitation.

In order to determine stresses in the composite materials, software packages MSC Patran and MSC Nastran are used. One composite material is chosen and appropriate models are formed, which are, because of simplicity, composed of only a single layer. Simulation testings of the models on tension, shear and pressure are done. By using appropriate program, stress distributions on the model were achieved.

For all three tests (tension, shear, pressure) deviations from the obtained values and supposed values are inconsiderable. On the basis of the results from the tests of these programs, potentially critical locations in model structures are determined, due to the existence of maximum stress, and accordingly possible occurrence of damage and fracture. Obtained data can be used for further testing and analysis of damage present in these locations on the real constructions. With a particular probability, the model behaves almost the same as the actual construction. However, these facts have to be proved and confirmed by experiments. In that way, we can get accurate insight into the structure state because of damage, with the maximum optimization in terms of time and cost of testing the same.

REFERENCES

[1] Baker, A., Dutton, S. and Kelly, D.: *Composite Materials for Aircraft Structures*, American Institute of Aeronautics and Astronautics, Reston, 2004.

[2] Henkhaus, K.: Overview of research on composite material impact behavior, in: *Electronic Proceedings of the 16th ASCE Engineering Mechanics Conference*, 16-18.07.2003, Seattle, Paper 86.

[3] Rasuo, B.: *Aircraft Production Technology*, Faculty of mechanical engineering, Belgrade, 1995, (in Serbian).

[4] Zhao, H: *Stress Analysis of Tapered Sandwich Panels with Isotropic or Laminated Composite Facings*, MSc thesis, The Graduate School, University of Maine, Maine, 2002.

[5] Niu, M.C.Y.: *Airframe Structural Design*, Conmilit Press Ltd, Hong Kong, 1988.

[6] Jovanovic, M. and Filipovic, Z.: Applying of Software Packages CATIA in Prediction of Possible Damages to Real Aircraft Construction, Flight test center, Belgrade, 2005 (in Serbian).

[7] Bathe, K.-J.: *Finite Element Procedures in Engineering Analysis*, Prentice-Hall, Englewood Cliffs, 1982.

[8] Montemurro, M.P., Houde, M.J.L. and Hansen, J.S.: *Finite Element Analysis of the Impact Response of Composite Structures*, Institute for

aerospace studies, University of Toronto, Toronto, 1993.

[9] Ritter, W.J.: *Application of Energy Methods to Modeling Failures in Composite Materials and Structures*, MSc thesis, Montana State University, Montana, 2004.

[10] Dash, P.K. and Chatterjee, A.K.: Effects of environment on fracture toughness of woven carbon/epoxy composite, Journal of the Institution of Engineers (India), Vol. 85, No. 5, pp. 1-9, 2004

[11] Rohwer, K., Friedrichs, S. and Wehmeyer, C.: Analyzing laminated structures from fiber-reinforced composite material – an assessment, Technische Mechanik, Vol. 25, No. 1, pp. 59-79, 2005.

[12] Smojver, I.: *Mechanics of composite materials*, Faculty of Mechanical and Naval Engineering, Zagreb, 2007, (in Croatian).

[13] Jones, R.M.: *Mechanics of Composite Material*, Taylor & Francis, Philadelphia, 1999.

[14] Emery, T.R.: *Identification of Damage in Composite Materials using Thermoelastic Stress Analysis*, PhD thesis, School of Engineering Sciences, University of Southampton, Southampton, 2007.

[15] Ming, C., Wu, T.-S., Lee, C.-H. and Lee, N.-H.: *Integrating MSC Software and CADSA Program for the Aircraft Detail Stress Analysis*, Aerospace Industrial Development Corporation, Taichung (Taiwan), 1999.

АНАЛИЗА НАПОНА У УНИДИРЕКЦИОНАЛНОМ *CARBON/EPOXY* КОМПОЗИТНОМ МАТЕРИЈАЛУ

Драган Д. Крецуљ

У овом раду проучава се напон на моделима од унидирекционалног *carbon/epoxy* композитног материјала. Софтверски пакети, базирани на Методи Коначних Елемената (МКЕ), користе се како би се предвидела расподела напона на испитиваном моделу. На тај начин одређују се на истом моделу карактеристична места са максималним напонима. Структурна оштећења на моделу и евентуално лом прво ће бити створени на тим локацијама. У извештају су представљени тестови напона на одговарајућим моделима од композитног материјала при напрезању на затезање, смицање и притисак. За ту намену користе се специфични комерцијални програмски пакети *MSC Patran* и *Nastran*. Поређењем претпостављених и добијених вредности напона закључује се да постоје безначајна одступања међу њима. Резултати симулација показују да је примена ових програма оправдана за ову врсту проблема.