PERIODICA POLYTECHNICA SER. MECH. ENG. VOL. 46, NO. 2, PP. 139–158 (2002)

FINITE ELEMENT MODELLING OF THE DAMAGE AND FAILURE IN FIBER REINFORCED COMPOSITES (OVERVIEW)

András SZEKRÉNYES* and József UJ**

*Research Group for Dynamics of Machines and Vehicles
Hungarian Academy of Sciences
H–1051 Budapest, Hungary
Phone: +36 1 463 1170
e-mail: szeki@mm.bme.hu
**Department of Applied Mechanics
Budapest University of Technology and Economics
H–1512 Budapest, Hungary
Phone: +36 1 463 2228
e-mail: uj@mm.bme.h

Received: Mai 2, 2001

Abstract

The failure and the analysis of composite structures can be various. The failure modes can be investigated separately, but these modes generally arise together (e.g.: matrix cracking and delamination). Due to this experimental observation the main problem is to construct models that can follow more than one of the failure modes. To solve this problem the most proper method is the finite element method (FEM), because of its simplicity. The aim of this paper is to summarize a few publications about the damage and failure of composite materials and structures. The publications were grouped according to the mode of failure, the aim is to give a short description about the finite element modelling of the given failure mode. The models were reconstructed using the finite element code COSMOS/M.

Keywords: composite, micromodel, macromodel, unit cell, specimen, FE analysis.

1. Introduction

Watching the structure of composites we can notice its heterogenous nature. The composite material consists of fibers and matrix. The fibers are embedded into the matrix. During the manufacturing process the matrix is warmed up to high temperature, then the fibers are added. Due to the high temperature and the mismatch between the thermal properties of the components, there is interaction between the fibers and the matrix. The formation of interface is a consequence of this operation. Hence the composite is a three-phased material. The length and the arrangement of the fibers are various. The most frequent ones are the long unidirectional continuous and the short unidirectional arrangements. Another important feature of the composites is the periodicity. This feature can make the FE analysis of the composite materials simple. In literature there are three basic types of analysis [11, 19].

Microscale analysis:

The modelling of the local constituents (fiber, matrix, interface) using the periodicity and the symmetry.

Macroscale analysis:

The modelling of the global composite structure using the homogenized material properties of the composite.

Mesoscale analysis:

A scale between the macro- and microscale, which can be a micromodel that is not periodic in all directions or, considering laminated structures, this may mean the analysis of one elementary ply.

In fact, there is another scale, named multiscale analysis, coupling the three ones above. In this case we need parallel computers with big capacity. Multiscale approach was presented by FEYEL and CHABOCHE in [4] for composites.

2. Material Models for the Finite Element Analysis

There are significant differences between the material properties of the fiber and matrix. The stiffness of fiber is much higher than the matrix stiffness. Major part of the loading is transferred onto the fiber. Experimental data prove that the global composite structure behaves as a linear elastic material [10]. If local analysis is applied, then the considering of heterogeneity is necessary.

2.1. Materials of the Reinforcing Fibers

The material of fibers has a wide range of type (e.g.: glass, carbon...etc.). It is assumed that the glass fiber is isotropic, the carbon fiber is transversely isotropic linear elastic material. The mechanical and other properties of the fibers are available in the literature [10].

2.2. Material Modelling of the Matrix

The modelling of the matrix is a far complex problem. In general the matrix is a nonlinear elastic isotropic and hysteretic material. In case of polymer matrix (e.g.: epoxy, vinylester, polyester...etc.) the applying of linear elastic material model can give wrong results during the FE analysis. Due to this experience it is useful to apply rheological material models for the matrix. These are the viscoelastic models that consist of the well-known Maxwell and Kelvin–Voight elements and the combinations of them [2, 6]. Metal matrix composites (MMC) require using an elastoplastic, viscoplastic and elastoviscoplastic material model [3, 4]. Inelastic material models are reachable in [14]. Inelastic behaviour can be simulated by stiffness or strength degradation. Previous investigations describe that using a linear material model damage occurs in higher loading than in the case of the nonlinear model [19]. Because of that the nonlinear material law describes more accurately the stress-strain relationship than the assumed linear. Commercial FE programs usually contain nonlinear material models, but there is a chance to build in another [19]. For that a user defined subroutine and the modification of it is needed [15].

2.3. Properties of the Interface

The modelling of interface also can cause problems. The interface has been widely acknowledged, but the interface properties are not yet available [1]. The properties of the interface play critical role because the interface transfers the load onto the fibers. Experiments prove that the thickness of interface is not negligible. Interface can be modelled as elastic isotropic material, but there are also other approximations. For example an elastic/perfectly plastic spring model can describe the interface properties [3]. The spring model has different stiffness in the radial, tangential and longitudinal directions. ANTIFANTIS described an interface model, which assumes that the properties of interface (E, ν) vary along the thickness of interface [1]. Varying the adhesion parameters the perfect and imperfect interface can be simulated. Other authors provided a cohesive interface model for metal matrix, short fiber composites [21]. Experiments show that stress peaks can occur in the interface region [1]. In most cases the initiation of the damage depends on the interface properties. Several papers neglect the interface; these models assume perfect contact between the fiber and the matrix [3, 19]. The interface plays significant role in the damage process of composites.

3. Macro- and Micromodels, Unit Cells

3.1. Macroscale Finite Element Models

In the case of macroscale analysis we consider the composite structure as a homogeneous orthotropic continuum. Knowing the material properties of the constituents and the fiber-volume fraction we can compute the homogenized properties with the law of mixture [10]. Most of the commercial FEM programs contain two- and three-dimensional elements that are suitable for constructing global models. These are shell, solid and multilayer shell elements. For the using of these elements the fiber orientation is required. Applying these global models local microprocesses, including local stress-strain distribution cannot be followed. These processes are very important in the failure of composites (e.g.: interface damage, fiber fracture...etc.).

3.2. Microscale Finite Element Models

Unit cells and RVE (Representative Volume Element) models are the most convenient forms in the micromechanical analysis. Making unit cells depends on the geometry and the applied load. Depending on the structure and fiber orientation these models can be two- and three-dimensional ones. Due to the heterogeneity different material models must be applied to describe the three-phased composite. The construction and the mesh size of the finite element model have a great effect on the results of the analysis. Constructing the FE model the prescription of proper boundary conditions is required.

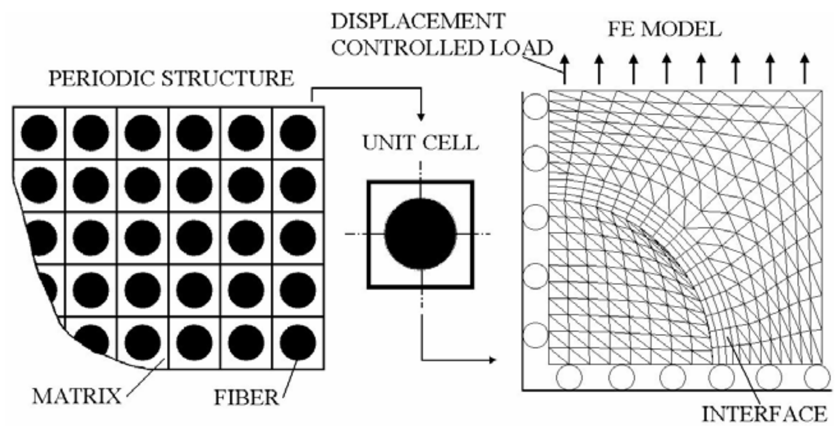

Fig. 1. 2D finite element model, square fiber arrangement

Generally, the load of micromechanical models is displacement or strain controlled. This means that one of the boundary curve or surface of the model is loaded with displacement in a given direction [1, 19]. *Fig. 1* shows the making of 2D model, the cut is perpendicular to the longitudinal direction of the fibers. Assuming the periodicity of composite a double symmetrical unit cell can be made. Using symmetrical conditions the FE model is only quarter of the former one. *Fig. 1* also shows the proper boundary conditions. In *Fig. 1* the fiber and the matrix are covered by triangular, the interface is covered by rectangular plane elements in plane stress state. Depending on the aim of the analysis, there are several cases when the cut must be longitudinal.

If a composite structure is built by plys with different orientation a 3D FE model is required [3, 19]. In *Fig. 2* a three-dimensional unit cell and finite element model are shown. Using the periodicity a 3D multi-cell can be made, the finite element model is just one eighth of the multi-cell. Proper boundary conditions can be seen also in *Fig. 2*. Between fiber and matrix a perfect bonding is assumed and no debonding occurs. Three of the surfaces cannot move at the normal direction. Remained surfaces can be loaded with given displacements.

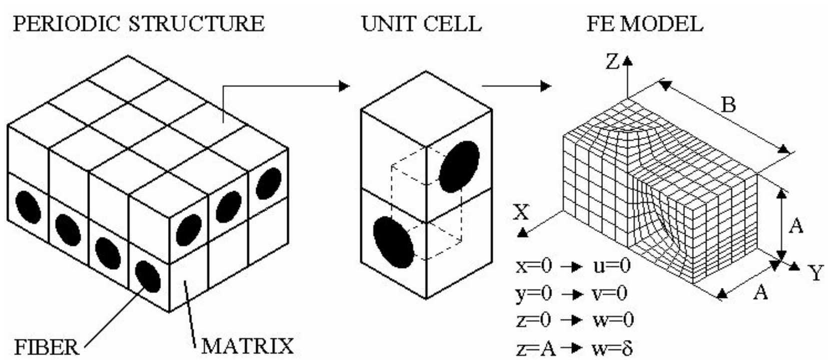

Fig. 2. 3D finite element model

The fiber arrangement is various. Most of the FEM programs have not got a well-built in CAD system. These CAD systems do not guarantee smoothness and accuracy that is always required. Because of that a proper CAD system is applicable [16]. Most of the FE codes can handle models drawn in CAD. Complicated fiber arrangements (e.g.: woven fabric, basket weave and triaxial braided unit cells) raise the claim to use CAD system. More information can be found in [16]. The former paper presents modelling of composite unit cells using CAD, and also describes the construction of unit cells with logical (Boolean) operations.

4. Failure Analysis of Composites

Failure means that one of the components reaches the yield stress, then damage occurrence and progression can be observed. Damage can progress on different directions. Usually matrix cracking is the first damage process to take place since the matrix has the lowest stress to failure. A failure criterion is needed to establish initial damage. This can be the maximum stress or strain criterion [19]. Damaged zones can propagate and reach the border of each other. Finite element tests have a wide application range. In most of the cases a simplified, loaded model is investigated by FEM, the distribution of stress and strain fields gives enough information about the damage process. Models can be analysed with holes and notches, considering stress concentration effect. The following sections give a short review about the finite element analysis of damage and failure processes in composites using previous publications.

4.1. Modelling of Interface Damage

Experimental observations show that the damage process can be initiated by the fiber-matrix interface. The damage of the interface causes further problems (e.g.: debonding, fiber pullout). The interface strength plays an important role in the damage process. Depending on the loading the interface failure be can caused by the mismatch of the moduli and Poisson's ratios. SCHÜLLER et al. presented a very useful method for modelling the interface debonding [12]. A single fiber is embedded in a necked matrix block, as shown in *Fig. 3*. Mechanical loading can be uniaxial compression or tension. Compression test has two disadvantages. First the fiber often breaks or buckles before interfacial debonding can occur. Another drawback is the additional transverse stress components, which makes the stress field non-axisymmetric one. Under compression the transverse component is a tensile one between the fiber and the matrix.

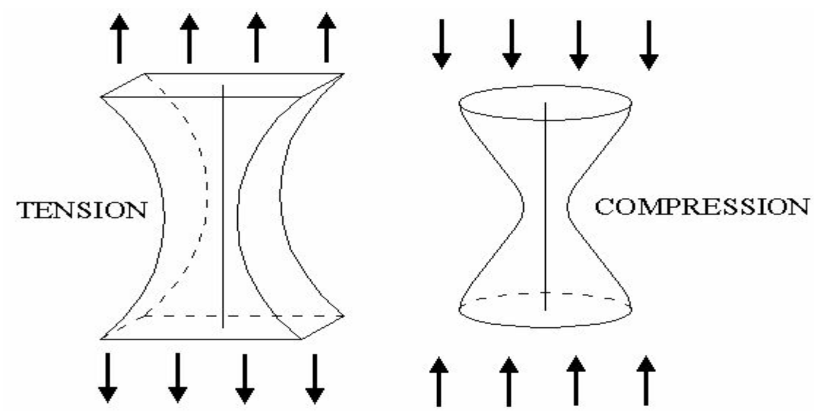

Fig. 3. Necked single-fiber composite specimens

The problem of fiber buckling can be avoided by applying the tension test. The transverse component is compressive; a sufficiently small neck can compensate it (see *Fig. 3*). The limit stress is determined by the matrix yield stress. The single-fiber specimen can be various, an axisymmetrical model is the most convenient because the stress field will be even axisymmetrical. The homogeneous tensile loading of the interface allows the measurement of the real interface strength. For the verifying of FE results real specimens are needed. The preparation of axisymmetrical specimens is very difficult. This method cannot follow the effect of adjacent fibers. Interface damage plays an important role in most of the failure modes.

4.2. The Effects of Thermal Residual Stresses

Opinions differ about thermal residual stresses. Ones say that major part of the thermal residual stresses relax with the passage of time [19], others say that residual stresses have significant effects on the micromechanical behaviour of composites [5, 7].

Manufacturing process induces thermal residual stresses in composites. After curing and cooling the composite, the matrix is subjected to a triaxial stress state [5]. Shrinkage during cooling and the mismatch between thermal properties of the constituents are the main reasons of residual stresses. The thermal residual stress is a compressive one in the fiber and tensile one in the matrix. If the composite structure is loaded the triaxial stress state in matrix will increase. FIEDLER et al. investigated thermal residual stresses using three different models [5]. A concentric cylinder model, a hexagonal model and a special composite model were used with long continuous fibers. All models consist of two-dimensional plane strain elements. The models are depicted in *Fig. 4*. Thermal residual stresses are constants along the circumference of fiber in the cylinder model. The composite model contains the hexagonal unit cell that is only 10% of the whole model. This local area is surrounded by elements with homogeneous properties. The hexagonal arrangement makes possible to investigate the effect of adjacent fibers. Thermal residual stresses were examined along the circumference of the central fiber in radial, tangential and longitudinal directions. The local fiber-volume fraction was varied from 20% to 80%. Looking at the hexagonal and composite model residual stresses in all the three directions show sinus-like distribution along the circumference. Increasing fiber-volume fraction the maximum and minimum values also increase.

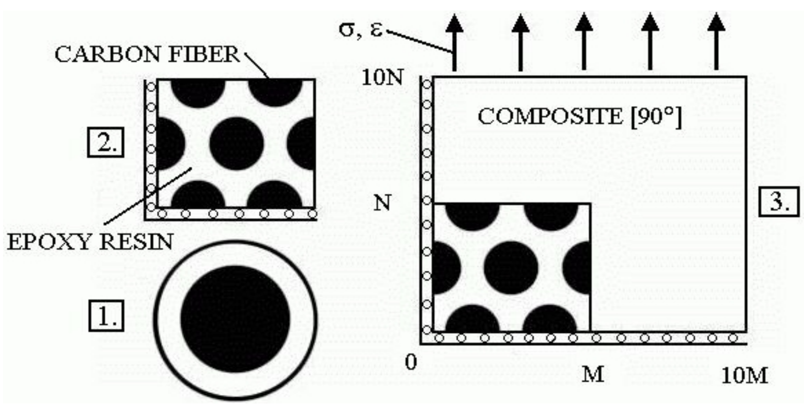

Fig. 4. Cylinder (1.), hexagonal (2.) and special composite (3.) models

The hexagonal unit cell shows the results that neighbouring fibers more or less influence the residual stress components. Results obtained by the composite model show a decrease of thermal residual stresses with increasing local fiber-volume fraction. The location and the value of the applied stress at which failure occurs are influenced by the local fiber-volume fraction. Depending on the applied matrix criterion the damaged area can be different. The three used criterions are the parabolic, the strain energy density and the von Mises criterion. Varying the fiber-volume fraction these criterions give very distinct results. The von Mises criterion predicts over the maximum load, so it is not suitable to predict initial matrix failure. In fact thermal residual stresses reduce the maximum bearable load, but they do not affect the location of initial failure.

The effect of thermal residual stresses in short-fiber-reinforced composites was investigated by ZHOU et al. [21] with Al matrix and SiC fiber. The applied FE model and the boundary conditions are shown in *Fig.'5*.

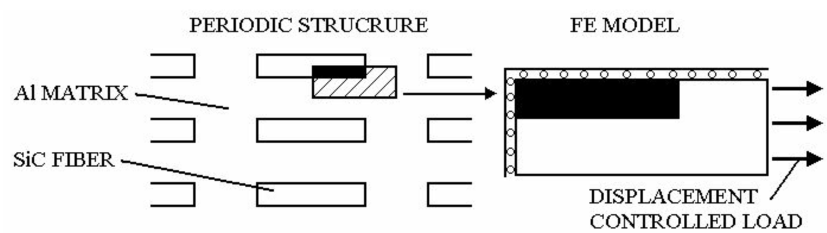

Fig. 5. FE model for short-fiber-reinforced Al/SiC composite

Cooling from high to room temperature the residual shear stress concentrates near the corner of the fiber end and induces initial damage along the interface. Considering thermal cycling the radial, longitudinal and shear stress distributions were investigated. The results show considerable stresses ahead of the fiber end and around the corner of the fiber. Applying thermal cycling interface damage is more dominant than matrix damage. Thermal cycling decreases the strength and the toughness of metal matrix composites (MMC). The paper shows that the thermal cycle range rather than the number of cycles dominate damage process. Cooling and thermal cycling change the mode of failure. Considering only tensile load failure arises as matrix damage around the fiber end and no debonding occurs. Considering cooling with tensile load debonding along the fiber end and transverse damage across the matrix occur. Finally applying thermal cycling and mechanical load the MMC fails by debonding along the fiber ends and fiber sides, followed by necking instability of the matrix ligament.

Microscopic behaviour of MMC is strongly influenced by the inelastic deformation of the matrix. ISMAR and co-workers presented a study about MMCs reinforced by different types of fiber [7]. The damage of matrix is affected by the properties of the fibers. A non-symmetrical [0°/90°] fiber arrangement model was investigated, *Fig. 6* depicts the 3D model. Boundary conditions are identical with those shown in *Fig. 2*. The three different fiber types: $\alpha - Al_2O_3$, SiC and carbon fiber. All fibers behave elastically, $\alpha - Al_2O_3$ and SiC are isotropic, the carbon fiber is described as transversely isotropic material. Interface between fiber and matrix is assumed to be very strong and no debonding occurs. The $\alpha - Al_2O_3$ has the highest tensile modulus, SiC has the lower, and modulus of carbon fiber is between the formers. During fabrication process, the composite passes through several temperature regions. Thermal residual stresses cause local inelastic deformation.

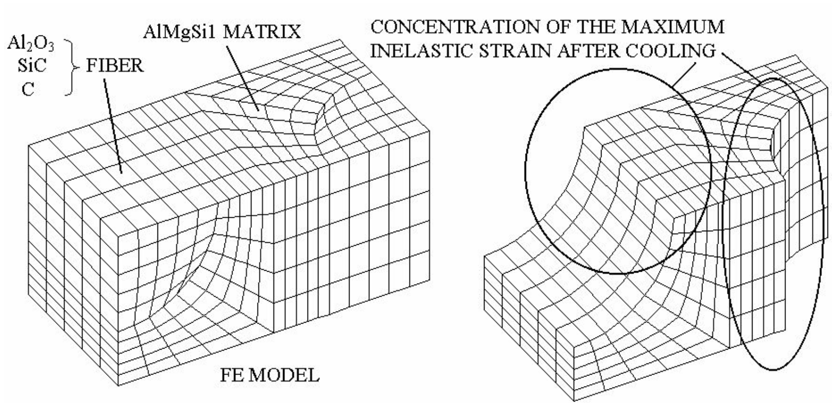

Fig. 6. 3D FE model for metal matrix composite

A cooling process is firstly examined. Seeing the three fiber types the inelastic strain shows highest values in the case of carbon fiber. After cooling the equivalent inelastic strain concentrates areas shown in *Fig. 6.* Inelastic strain distribution reaches the maximum values where the fibers are the closest to each other. Varying the fiber-volume fraction does not significantly alter the relative distribution of equivalent inelastic strain but strongly affects the extent of it. Because of that the effects of thermal cooling is considered in the following investigations of the mechanical behaviour. Mechanical loading is displacement controlled with given strain rate. Under monotonic loading Al_2O_3-reinforced composite shows maximum inelastic strain owing to the larger difference between the tensile modulus of the fiber and matrix. The stress-strain curve of the three fiber reinforced composites shows that the slowest decline of the tangent modulus can be observed in the case of carbon fiber. On the other hand Al_2O_3 fiber has the fastest decline. Cyclic loading was considered with given strain rate and temperature. Metal matrix behaves inelastically applying the given loading amplitude. The density of inelastic performance after forty cycles decreases depending on the fiber type. Increasing loading amplitude inelastic behaviour strongly increases. Cyclic loading indicates damage progression and damage accumulation. Damage is distinctly concentrated in two matrix areas; the Al_2O_3-reinforced composite exhibits the highest damage values. Heightening the fiber-volume fraction, the inelastic deformation slightly increases. Fiber with large tensile modulus increases the strength of composite. On the other hand the danger of premature failure during cyclic loading has to be considered. Finally, the fiber-volume fraction decisively affects the mechanical behaviour of metal matrix composites.

4.3. *Matrix Damage and Matrix Cracking*

Matrix crack occurs when the equivalent stress reaches the yield stress, after fiber end can be debonded and a local plastic zone in the matrix is the result of further loading. In the case of short-fiber reinforced composites around the fiber end a penny-shaped crack can arise. SIRIVEDIN et al. presented a study about matrix cracking in short-fiber composite [13]. Depending on the strength of the interface this crack can progress on various ways. In the case of relatively weak interface, a cylindrical interface crack propagates from the debonded fiber end. If the interface is strong then a conical matrix crack can be observed propagating from the debonded fiber end. This can be described by the angle of the conical crack. The applied finite element model is depicted in *Fig. 7.* Using the symmetry of the unit cell a 2D model can be presented.

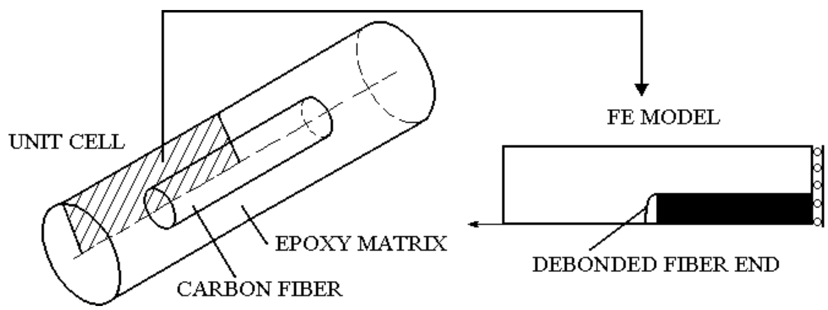

Fig. 7. Unit cell and axisymmetrical FE model

A penny-shaped crack resulting from fiber-end debonding is assumed to exist. The mechanical loading is tension. The matrix is described as an elastic-plastic material. Under loading micro-void nucleation, growth and coalescence are observed. Near the corner of fiber a local plastic zone is formed. If the plastic strain reaches the critical fracture strain value crack will be propagated. The value of the critical strain can be determined approximately. The angle of crack affects the direction of crack propagation; the angle is described by the location of the maximum stress. The paper presents mechanical loading, thermal loading and the combination of the two above. Results show those thermal residual stresses and strains strongly affect the condition of the interface and change the location of the maximum equivalent strain.

Laminated structures also require investigating the reason of matrix damage. A $[0°, 90°_3, 0°]_T$ cross-ply structure is investigated in [19], in the case of epoxy matrix and glass fiber. The cross-ply contains long continuous fibers. The FE model is shown in *Fig. 8.* It is assumed that fibers are uniformly distributed in the matrix and have the same radii. The loading is strain controlled; the boundary conditions are also depicted in *Fig. 8.* Attention is focused on the interior domain; no free edge effect is included. Fibers are linear elastic; matrix behaves as nonlinear viscoelastic material. Both constituents are homogeneous and isotropic.

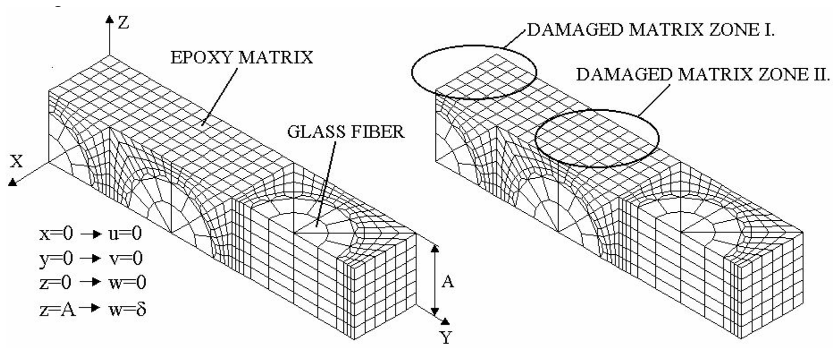

Fig. 8. 3D FE model and location of the damage, $[0°, 90°_3, 0°]_T$

Criterion for the epoxy matrix describes if the maximum principal strain reaches the critical strain value, damage progression will occur in matrix. FE results indicate that local damage zones arise in two areas, shown in *Fig. 8.* Increasing loading both damage areas grow and coalesce. Applying linear analysis the damage occurs at higher loading. So that the nonlinear model describe more accurately the real process. Experimental data support this fact. Nonlinearity plays an important role in the analysis of composites.

4.4. Modelling of Composite Structures with Holes and Notches

Numerous applications of composites require the presence of holes and cutouts. These can cause inhomogeneous stress distribution in the structure. Cracks and flaws act important role, because these affect the strength of the composite. Crack can be caused by the manufacture process or loading. Concerning failure criteria, three basic types of approaches can be found in literature [8, 10].

Fracture Mechanics Models:

These models assume a localised damage emanating from the hole, represented by an equivalent crack, which causes failure when it reaches a critical size. The strength of the laminate is related to the fracture toughness or the strain energy release rate. The fracture toughness and the critical size seemed to be hole-size and material dependent. This is the basic idea of LEFM (Linear Elastic Fracture Mechanics).

Point and Average Stress Criterion:

It is assumed that failure occurs when the stress in one point at a certain characteristic distance or the average stress along the characteristic distance from the hole attains the unnotched strength of the laminate. Both criterions can be expressed in closed-form equations. Experimental data show: those characteristic distances are also hole-size and material dependent.

Progressive Damage Models:

Throughout the loading process damage and resulting changes occur in the stress distribution. The appropriate combinations of failure modes, failure criterion and property degradation law are required.

Open-hole tensile strength of quasi-isotropic laminates is investigated by MORAIS in [8]. Stress analysis near edges and discontinuities can only be accurately performed using 3D models. 2D models give non-negligible errors in laminates. The central hole has the effect of stress concentration, the stress concentrates around the hole. Considering this fact MORAIS presented a mixed 2D/3D model. The model is shown in *Fig. 9*. The specimen was modelled with 2D elements, except for a narrow zone around the hole, where 3D brick elements were used. The connection between the 3D and 2D regions was done imposing identical in-plane displacements on the boundary nodes of identical in-plane coordinates and using interface elements. The longitudinal strength of laminated structures depends on the size. Failure criterion does not follow this fact. First a $[0°/45°/90°/-45°]_s$ laminate was investigated. The loading and boundary conditions are depicted in *Fig. 9*. The laminate is transversely isotropic; properties can be obtained using classical laminate theory. The inhomogeneous stress distribution can be obtained along a central ligament. The 3D stresses considered are layer average stresses. The stress concentration depends on the hole-size. Comparing the 2D and the mixed models the mixed model shows higher stress concentration values near the hole than the 2D model. 2D models underpredict stress concentrations, further approximately identical values were obtained by both models from the hole. The strength of structure with central hole depends on the longitudinal strength of the laminate. In situ fiber strength can be estimated using experimental data and the mixed model.

Laminates with triple parallel-arranged and equally spaced notches can be found in [20]. Glass- and carbon-fiber/epoxy composites containing through or surface notch were investigated. *Fig. 10* shows the laminate and the FE model. The FE model is only a quarter of the structure, the stacking sequence is $[0°/90°/0°/90°/0°]_s$. The laminated composite behaves as orthotropic linear elastic continuum, the interlaminar matrix is isotropic. The paper presents results taking account of the interlaminar matrix and not, respectively. Notch can be surface or through notch. The mesh was gradually refined to the notches. Stress concentration factors (SCF) were introduced for the central and the deviate notch varying the distance between

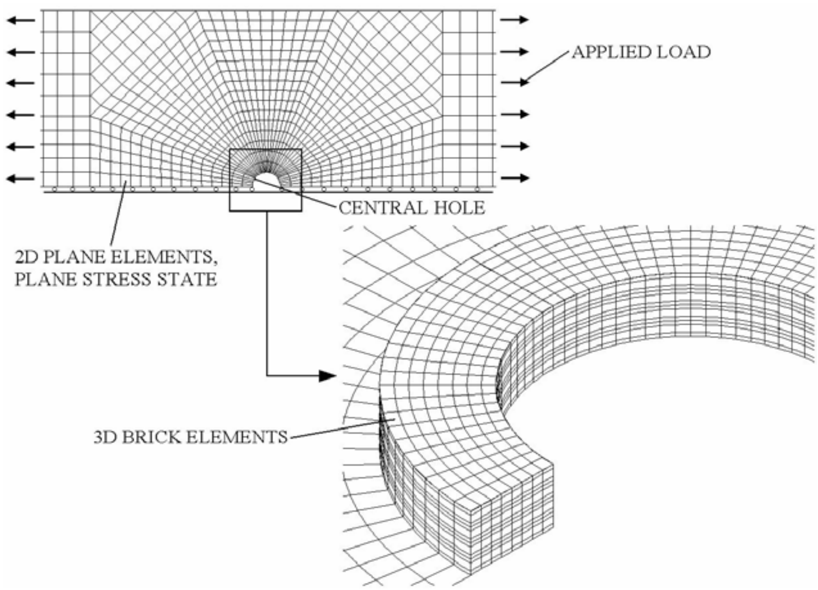

Fig. 9. Mixed 2D/3D model with central hole

the two notches. The depth of layer describes the surface notch. Results show that stress concentration factors near the notch tips change from high to low then revert to high with spacing between both neighbouring notches.

Stress concentration factor is always larger at the deviate notch. Depth of the notch increases the SCF. In order to validate the FE predictions a total of twenty specimens with triple parallel notches were examined. Failure tests show the change of the failure load exhibits an opposite trend with the notch stress concentration factors. Tests also show that failure always occurs at one of the two deviate notches.

A notched laminated structure, including splitting and delamination was investigated by WISNOM and CHANG [18]. A cross-ply laminate of carbon/epoxy with lay-up [90°/0°] with a center crack has shown that splitting occurs in the 0° plies perpendicular to the notch together with narrow areas of delamination. The [90°/0°] laminate was modelled with four noded plane elements. The elements are plane stress ones. Duplicate nodes were used on either side of any ply interfaces where delamination is expected. The same interface element was also used to model splitting in the 0° ply. The schematic illustration and the FE model are depicted in *Fig. 11*. Separate elements were used to represent the 0° and 90° plies. Duplicate coincident nodes were used for the two plies to allow relative displacement to occur during delamination. These nodes were connected with interface elements. Initially linear elasticity was assumed, but it became apparent that the nonlinear in-plane shear response was very important. The interface was modelled with nonlinear springs between the coincident nodes. The springs are initially assumed to be

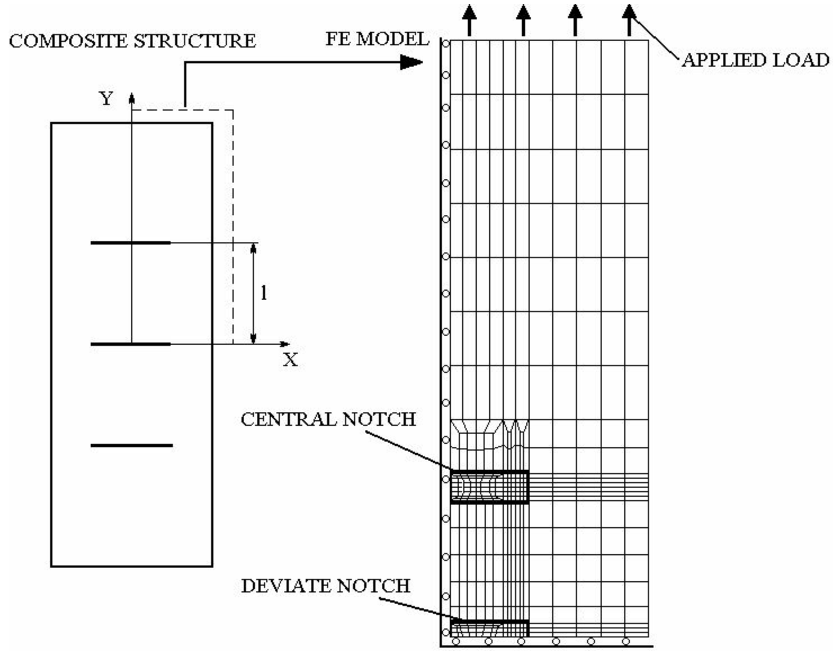

Fig. 10. Laminated structure and FE model with triple parallel notches

elastic. When a critical relative displacement between plies is reached the interface is assumed to fail. The tensile loading was applied in increments. Initial results show splitting initiated from the end of the notch, and then progressively increased in length.

It was accompanied by the formation of a narrow zone of delamination. The extent of delamination is slightly less than the split length. It is the splitting that drives the delamination. The analysis was repeated with refined FE mesh. Results show that the analysis is not sensitive to the mesh size. The difference between linear and nonlinear analysis was also investigated. The use of linear analysis gives unreal results; hence nonlinearity acts an important role. The analysis was also repeated with the spring stiffness halving to examine the effects of interface parameters. In this case the damage progression is similar to the original case, but the size of the yield zone has greatly increased.

4.5. Interlaminar Delamination and Transverse Cracking

Damage models can be constructed using proper combination of the global element types. NISHIWAKI et al. developed quasi-three-dimensional model for the modelling of interlaminar delamination and transverse cracking independently [9]. The

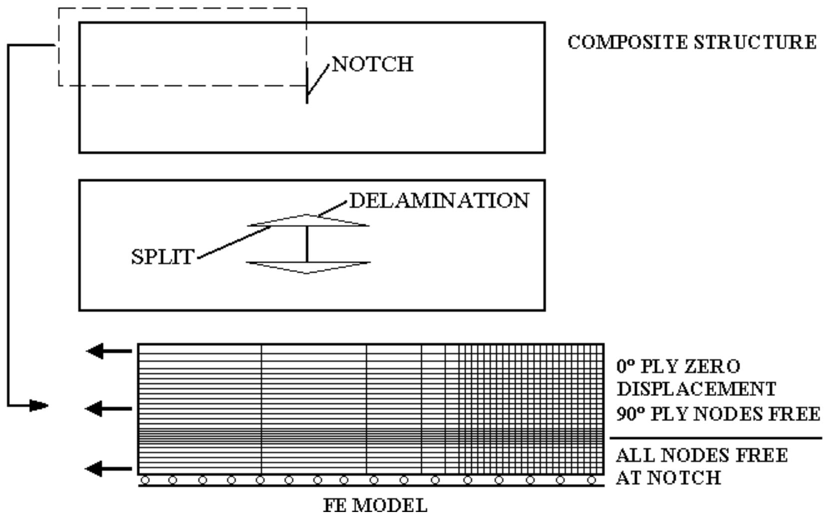

Fig. 11. Schematic illustration of damage and FE model

FE model is depicted in *Fig. 12* and consists of shell and beam elements, which represents fiber and resin respectively. The fiber reinforcement is concentrated in the shell elements with the maximum fiber-volume fraction of 90.7%. Beam elements represent the interlaminar matrix. The loading is lateral compression.

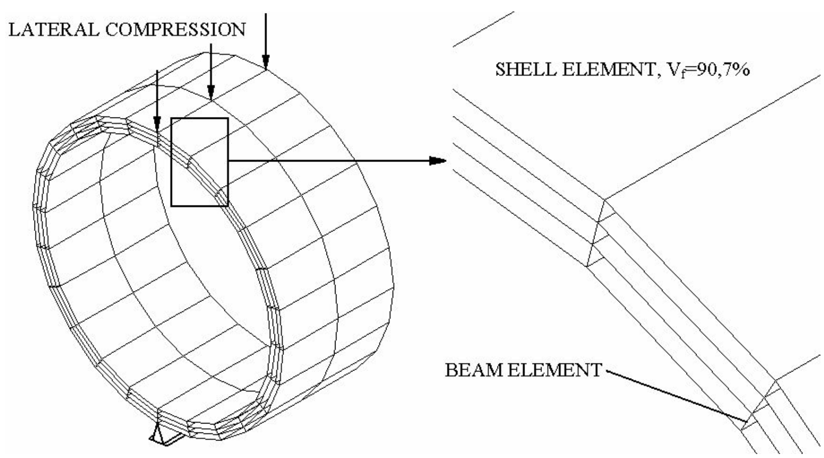

Fig. 12. Four layered quasi-three-dimensional cylinder model

The cylinder model was investigated with different stacking sequences. Results show different loading limits, transverse cracking occurs if a given shell element reaches the yield stress and delamination can be observed if one of the beam elements yields. Increasing compressive load more and more shell and beam elements reach the yield stress. Interlaminar delamination is caused by shear stresses. Finally, the estimation of the critical strength was performed. The critical strength was the load level corresponding to initial yielding of beam elements along the axial direction.

The interlaminar fracture is one of the major problems for fiber composites. Its occurrence greatly reduces the stiffness of a structure. Modelling of interlaminar crack can be performed using end-notched flexure specimen (ENF) under in-plane shear deformation mode [17]. Applying the ENF the fracture toughness can be measured. The aim is to provide quantitative information on interlaminar crack growth in fiber composites and understand the role of fibers on the stress distribution in the surrounding matrix. The used FE model and the meshed structure are shown in *Fig. 13*. The model contains only a single fiber on each side of the starting defect. The main concern is the role of the blunt starting defect, introduced by the notch. The in-plane shear loading was introduced by assigning a fixed displacement to the top and bottom surfaces. Boundary conditions are also shown in *Fig. 13*. Experimental ENF tests were conducted to verify FE results using two different specimens. The maximum principal stress shows that the highest stress occurs around the corners of the starting defect, extending towards the interface. The stress distribution indicates two mechanisms that are responsible for the fracture initiation. *Fig. 14* depicts that the fracture can be initiated either from the corner of the starting defect or at the interface. The latter occurs when the interfacial strength is lower than a critical percentage of the matrix strength. This percentage can be obtained from the stress distribution around the notch.

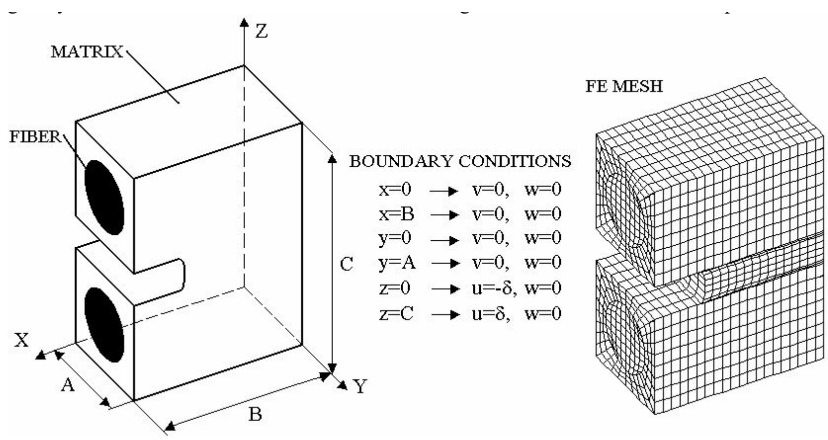

Fig. 13. End notched flexure specimen and FE mesh

When the crack is initiated from the interface, its growth is expected in both directions. The further crack can grow towards the starting defect and interfacial debonding occurs. If the interface strength is sufficiently high, the crack initiation is caused by tensile failure of the matrix around the starting defect. The crack

Fig. 14. Local damage process

propagates towards the interface because of the shear failure. It is expected that the interface strength can significantly influence the measured strain energy release rate.

4.6. *Modelling of Fiber-Fragmentation*

A finite element analysis was carried out by DAVIS and QU to simulate the fiber-fragmentation process in a single-fiber composite (SFC) specimen [3]. A single fiber is embedded in a uniform matrix material. Under monotonic axial loading the fiber will fracture. As the load increases further, more fiber fracture takes place as a result of load transfer from the matrix to the fiber. The fiber-fragmentation process will stop because either interfacial debonding or the fragments are so short that not enough loads can be transferred to the fiber for further fracture. This saturated length is related to the interfacial strength. The SFC fragmentation test is suitable to characterize the in situ interfacial strength and the understanding of the micromechanics in SFCs can be transferred to real composites with higher fiber-volume fractions. The Al matrix is assumed to be elastic/plastic, the interface is modelled by elastic/perfectly plastic spring layer. *Fig. 15* illustrates the SFC and the applied undamaged and damaged FE models. As the applied load is increased, the fiber will fracture when the fracture strain is reached. The fiber fracture creates a penny-shaped crack, creating a localized plastic zone in the matrix. The number of fracture per unit length will increase; i.e. the fiber-fragment length will get shorter and shorter as load increases. The shear stress on interface will keep increasing till the interface fails. There is a saturation length for the fiber fragments, which depends on the interface and the fiber strength. The SFC specimen containing many fractures can be simplified into a unit cell model containing one fragment. Only one quarter of the unit cell needs to be modelled for finite element analysis. A series of finite element models of the unit cells with decreasing length was used to analyse the fragmentation process. To simulate the successive fiber fracture, a series of 21 SFC unit cells with decreasing length were used, applying solid elements. The procedure was the following. The fiber strength was calculated and the load was increased until the axial stress in the middle of the fiber fragment exceeded the fiber strength. With the next specimen the calculation was repeated. This was done without and with fiber fracture on the model. It is found that although the radial and tangential stresses in the matrix vary radially, the magnitude of the longitudinal stress is much higher. Therefore the latter dominates the von Mises stress. One notable difference between the unfractured and fractured models is that the longitudinal shear stress was not zero in the latter case.

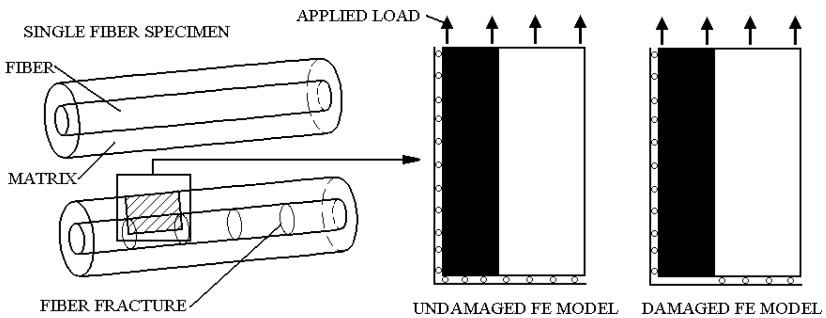

Fig. 15. Finite element modelling of fiber fracture

It seems that the stress-strain curve is not a good indicator of the interfacial shear strength. The applied effective strain/normalized fiber-fragment length is much suitable to describe the effect of the interface strength. Finally, it is possible to estimate the interface strength by inspecting the fiber-fragment lengths from a SFC test at a given effective strain.

5. Conclusions

The process of the finite element modelling of failure and damage is summarized in *Fig. 16*. Due to the heterogeneous structure, the modelling of composites is a quite complex problem. The macro-, meso- and micromechanical analyses are useful ways to approach the failure and damage process of composite materials. The type of the model depends on the failure mode and the aim of the FE analysis. A given failure mode can be represented by more than one type of models. The composite specimens play an important role; the results can be transferred into real composites. Considering previous papers the damage process is always related to the interface strength. The 3D models presented here consider perfect interface; the further purpose can be the effects of varied interface parameters. Thermal residual stresses strongly affect the strength of the composite. Increasing fiber-volume fraction, thermal residual stresses can increase. Holes and notches have the effect of stress concentration. Fracture mechanic problems can be handled applying LEFM, point- or average stress criterion and progressive damage models. Initial damage occurs when considering the used criterion, stress or strain reaches a given critical value.

Using the finite element method the damage and failure can be predicted with more or less accuracy, but to validate the results experimental tests and observations are always required.

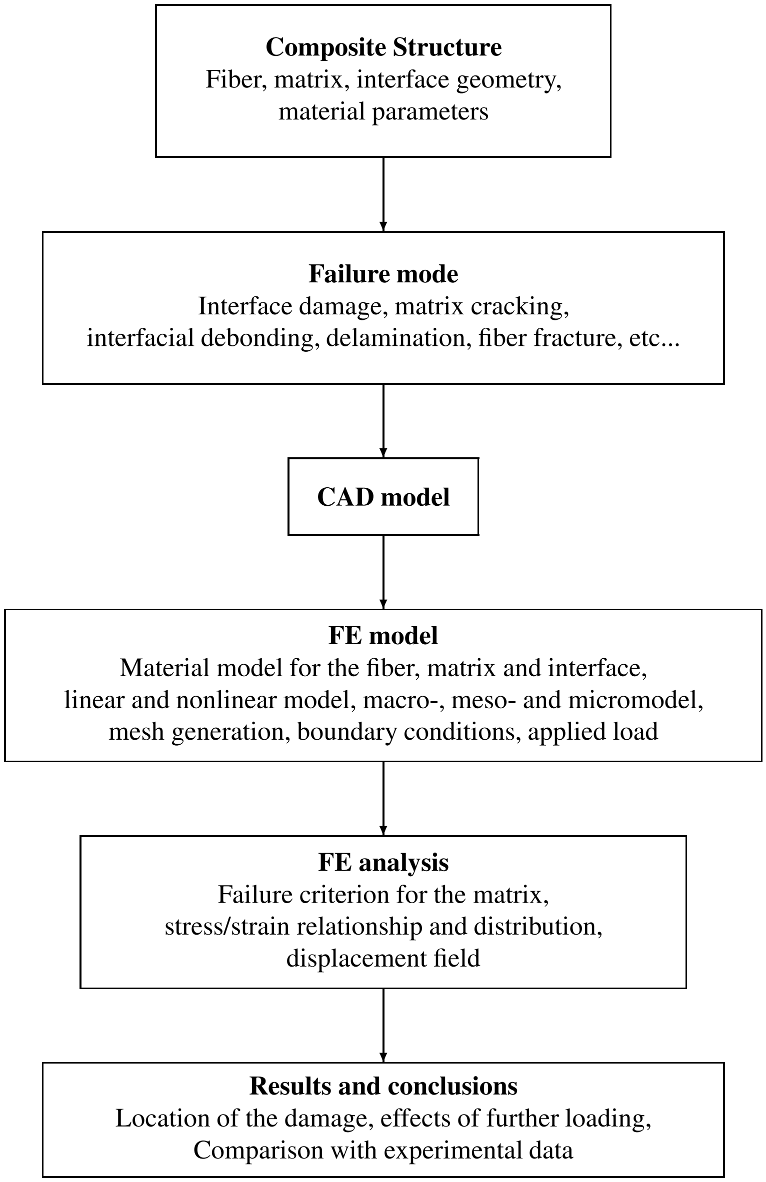

Fig. 16. Process of the finite element modelling of composites

References

[1] ANTIFANTIS, N. K., Micromechanical Stress Analysis of Closely Packed Fibrous Composites. *Composites Science and Technology* **60** (2000), pp. 1241–1248.
[2] BODOR, G. – VAS, L. M., *Polimer anyagszerkezettan*, Műegyetemi Kiadó; 1995.
[3] DAVIES, J. E. – QU, J., Numerical Analysis of Fiber Fragmentation in a SiC/Al Single-Fiber Composite Specimen, *Composites Science and Technology*, **60** (2000), pp. 2297–2307.
[4] FEYEL, F. – CHABOCHE, J.-L., FE^2 Multiscale Approach for Modelling the Elastoviscoplastic Behaviour of Long Fibre SiC/Ti Composite Materials, *Computer Methods in Applied Mechanics and Engineering*, **183** (2000), pp. 309–330.
[5] FIEDLER, B. – HOJO, M. – OCHIAI, S. – SCHULTE, K. – OCHI, M., Finite-Element Modelling of Initial Matrix Failure in CFRP Under Static Transverse Tensile Load, *Composites Science and Technology*, **61** (2001), pp. 95–105.
[6] HOLMES, M. – JUST, D. J., *GRP in Structural Engineering*, Applied Science Publishers, 1983.
[7] ISMAR, H. – SCHRÖTER, F. – STREICHER, F., Inelastic Behaviour of Metal-Matrix Composites Reinforced with Fiber of Silicon Carbide, Alumina or Carbon: a Finite-Element Analysis, *Composites Science and Technology*, **60** (2000), pp. 2129–2136.
[8] MORAIS, de A. B., Open-Hole Tensile Strength of Quasi-Isotropic Laminates, *Composites Science and Technology*, **60** (2000), pp. 1997–2004.
[9] NISHIWAKI, T. – YOKOYAMA, A. – MAEKAWA, Z. – HAMADA, H. – MORI, S., Quasi-Three-Dimensional Analysis for Composite Cylinder under Lateral Compressive Loading, *Composites for the Pressure Vessels Industry*, **302** (1995), pp. 87–92.
[10] PHILLIPS, L. N., *Design with Advanced Composite Materials*, The Design Council, Springer-Verlag 1989.
[11] ROSPARS, C. – DANTEC, E. LE, – LECUYER, F., A Micromechanical Model for Thermostructural Composites, *Composites Science and Technology*, **60** (2000), pp. 1095–1102.
[12] SCHÜLLER, T, – BECKERT, – LAUKE, B. – FRIEDRICH, K., Single-Fibre Transverse Debonding: Tensile Test of a Necked Specimen, *Composites Science and Technology*, **60** (2000), pp. 2077–2082.
[13] SIRIVEDIN, S. – FENNER, D. N. – NATH, R. B. – GALIOTIS, C., Matrix Crack Propagation Criteria for Model Short-Carbon Fibre/Epoxy Composites, *Composites Science and Technology*, **60** (2000), pp. 2835–2847.
[14] SIVASELVEN, M. V. – REINHORN, A. M., Hysteretic Models for Deteriorating Inelastic Structures, *Journal of Engineering Mechanics*, **126** (2000), pp. 633–640.
[15] Structural Research and Analysis Corp. Cosmos/M 2.0 Complete Finite Element Analysis System Documentation 1996.
[16] SUN, W. – LIN, F. – HU, X., Computer-Aided Design and Modelling of Composite Unit Cells, *Composites Science and Technology*, **60** (2001), pp. 289–299.
[17] TODO, M. – JAR, P.-Y.B. – TAKAHASHI, K., Initiation of Mode-II Interlaminar Crack from an Insert Film in the End-Notched Flexure Composite Specimen, *Composites Science and Technology* **60** (2000), pp. 263–272.
[18] WISNOM, M. R. – CHANG, F., Modelling of Splitting and Delamination in Notched Cross-Ply Laminates, *Composites Science and Technology*, **60** (2000), pp. 2849–2856.
[19] XIA, Z. – CHEN, Y. – ELLYIN, F., A Meso/Micro-Mechanical Model for Damage Progression in Glass-Fiber/Epoxy Cross-Ply Laminates by Finite-Element Analysis, *Composites Science and Technology*, **60** (2000), pp. 1171–1179.
[20] ZHAO, J. H. – WANG, X. F. – DHARANI, L. R. – WEI, J., Notch Stress Concentrations and Failure Characteristics in Laminates with Triple Parallel Notches, *Composites Science and Technology*, **60** (2000), pp. 2865–2872.
[21] ZHOU, C. – YANG, W. – FANG, D., Damage of Short-Fiber-Reinforced Composites Considering Cooling and Thermal Cycling, *Journal of Engineering Materials and Technology*, **122** (2000), pp. 203–208.