UDK 66.017:519.61/.64
Professional article/Strokovni članek

ISSN 1580-2949
MTAEC9, 48(4)549(2014)

IDENTIFICATION OF THE INITIAL FAILURE AND DAMAGE OF SUBSTITUENTS OF A UNIDIRECTIONAL FIBER-REINFORCED COMPOSITE USING A MICROMODEL

UPORABA MIKROMODELA ZA UGOTAVLJANJE ZAČETNIH NAPAK IN POŠKODB SESTAVIN KOMPOZITA, ENOSMERNO OJAČANEGA Z VLAKNI

Hana Srbová, Tomáš Kroupa, Robert Zemčík

University of West Bohemia in Pilsen, Department of Mechanics, Univerzitní 22, 306 14 Plzeň, Czech Republic
hsrbova@kme.zcu.cz

Prejem rokopisa – received: 2012-08-31; sprejem za objavo – accepted for publication: 2013-10-11

The main goal of this investigation was to test the capabilities of simple failure and damage criteria for the material phases of a unidirectional long-fiber carbon-epoxy composite material. This goal requires the identification of the material parameters in order to simulate simple tensile tests performed on thin coupons with various fiber orientations. Furthermore, a failure criterion for the fibers and a damage criterion for the matrix are proposed and the material parameters are identified in order to capture the moment of the first failure or damage. Finite-element analyses are performed on a unit cell with applied periodical boundary conditions. The identifications were performed using Optislang 3.2.0 and Python 2.4. The finite-element analyses were performed using Abaqus 6.11-1.

Keywords: unidirectional fiber composite, identification, matrix work-hardening function, unit cell, micromodel, non-linear

Glavni namen predstavljenega dela je preizkus sposobnosti enostavnih meril za napake in poškodbe pri sestavinah epoksikompozita, enosmerno ojačanega z dolgimi ogljikovimi vlakni. Ta cilj zahteva poznanje parametrov materiala za simulacijo enostavnega nateznega preizkusa, izvedenega na tankih ploščicah z različno orientacijo vlaken. Predlagana so merila za porušitev vlaken, za poškodbe osnove in ugotovljeni so parametri materiala za ugotovitev trenutka prve porušitve ali poškodbe. Analiza končnih elementov na osnovni celici je bila opravljena z uporabo periodičnih robnih pogojev. Identifikacija je bila izvršena z uporabo Optislang 3.2.0 in Python 2.4. Analize končnih elementov so bile izvršene z Abaqus 6.11-1.

Ključne besede: kompozit, enosmerno ojačan z vlakni, identifikacija, funkcija deformacijskega utrjevanja osnove, osnovna celica, mikromodel, nelinearen

1 INTRODUCTION

Composite materials are widely used due to their advantageous stiffness-to-weight and strength-to-weight ratios. The composites' material properties depend on the volume ratio of the constituents and their properties.

There are several approaches to modeling a composite material's behavior. The material model can be modeled on the micro-, meso- and macro-scale levels. The macromechanical approach requires, compared to the micromechanical, less detailed and time-consuming modeling. In[1] the non-linear behavior of composites was modeled using a macroscopic approach. Nevertheless, the micromechanical approach gives more detailed information about the constituents,[2] for example, the damage and failure of the matrix and fiber. It is also possible to capture the nonlinear behavior of a composite by assuming the constituents to be nonlinear materials. A composite material may vary its properties due to matrix cracking and fiber breaking during loading.

A knowledge of damage mechanisms on the micro level of the material is important during the design processes of structures. This fact is more important when it comes to a structure made of heterogeneous materials, such as composites or when the task is to avoid the damage or failure of the structure.

2 EXPERIMENT

Tensile tests of thin coupons made of unidirectional longfiber carbon-epoxy composite SE84LV-HSC-450-400-35 were performed on a ZWICK/ROELL Z050 test machine. The coupons were cut from one large plate using a water jet.[3]

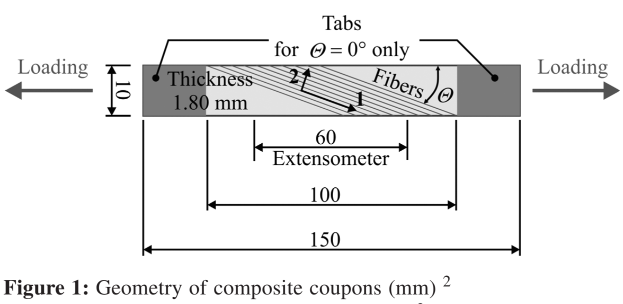

Figure 1: Geometry of composite coupons (mm) [2]
Slika 1: Geometrija ploščice kompozita (mm) [2]

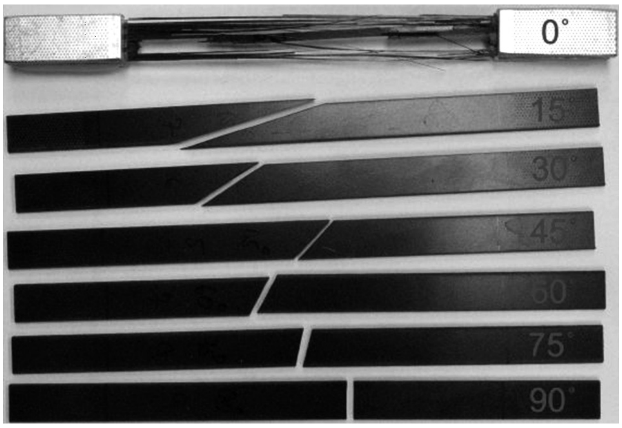

Figure 2: Cracked specimens with aluminum tabs on a specimen with a 0° orientation[2]
Slika 2: Razpokani vzorci z aluminijevimi zavihki na vzorcu z orientacijo 0° [2]

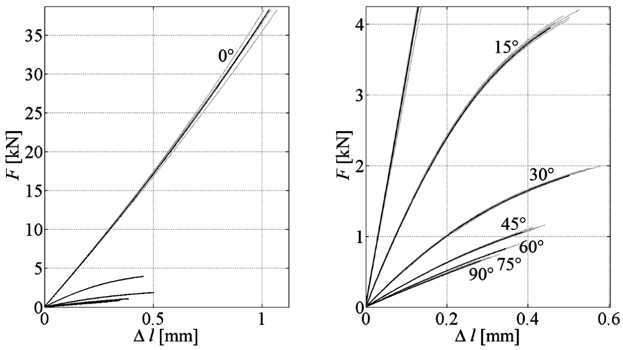

Figure 3: Measured force-displacement diagrams (grey) and averaged values (black)[2]
Slika 3: Diagrami sila – raztezek (sivo) in povprečne vrednosti (črno)[2]

The fiber direction forms angles of 0°, 15°, 30°, 45°, 60°, 75° and 90° with the direction of the loading force (**Figure 1**). There were 10 specimens tested for each angle θ. The cracked specimens are shown in **Figure 2**. The specimens loaded along the fiber direction were fractured by fiber failure. All the specimens loaded at a different angle are fractured by matrix failure. The resulting force-displacement diagrams are shown in **Figure 3**.

2.1 Micromodel

The geometry of the finite-element model (micromodel) with a periodically repeated volume (unit cell) of the unidirectional composite material was created in the CAD program Siemens NX. The unit cell is meshed using ten-node tetrahedral elements. A perfect honeycomb distribution of fibers and a fiber volume ratio of 55 % were assumed (**Table 1**). Furthermore, the finite strain theory was used. The geometry of the unit cell was exported to Abaqus/CAE 6.11-1 for the finite-element analysis.

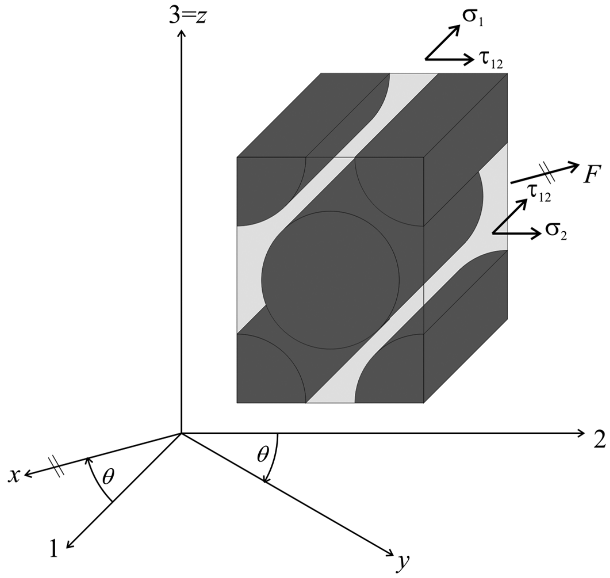

Figure 4: Rotated coordinate systems and loading of the unit cell
Slika 4: Rotirani koordinatni sistemi in obremenitve osnovne celice

Table 1: Geometry ratios of the unit cell
Tabela 1: Geometrijska razmerja osnovne celice

Fiber radius	r
Short side length	1.28 r
Long side length	2.22 r

The global coordinate system (*xyz*) is given by the force direction (*x*) and the direction perpendicular to the composite surface (*z*). The local coordinate system (123) is defined by the unit-cell edges, where the axis directions correspond to the fiber direction (1) and the directions perpendicular to it (**Figure 4**).

Assuming uniaxial stress across the whole specimen, the behavior of the material can be simulated by loading the unit cell with a normal stress corresponding to an external force F:

$$\sigma_x = \frac{F}{A} \tag{1}$$

where A is the cross-section of the specimen.

The effect of the loading force is transformed to the local coordinate system using the transformation:

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{bmatrix} = \begin{bmatrix} \cos^2\theta & \sin^2\theta & 2\sin\theta\cos\theta \\ \sin^2\theta & \cos^2\theta & -2\sin\theta\cos\theta \\ -\sin\theta\cos\theta & \sin\theta\cos\theta & \cos^2\theta - \sin^2\theta \end{bmatrix} \begin{bmatrix} \sigma_x \\ 0 \\ 0 \end{bmatrix} \tag{2}$$

where θ is the angle of rotation between the local and global coordinate systems.[3]

The results from the finite-element analysis (strains) are transformed back to the global coordinate system using the transformation:

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} = \begin{bmatrix} \cos^2\theta & \sin^2\theta & -\sin\theta\cos\theta \\ \sin^2\theta & \cos^2\theta & \sin\theta\cos\theta \\ 2\sin\theta\cos\theta & -2\sin\theta\cos\theta & \cos^2\theta - \sin^2\theta \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{bmatrix} \tag{3}$$

The unit cell must also respect the periodical boundary conditions:

$$\begin{aligned} \Delta u &= u_{\mathrm{B}} - u_{\mathrm{A}} \\ \Delta v &= v_{\mathrm{B}} - v_{\mathrm{A}} \\ \Delta w &= w_{\mathrm{B}} - w_{\mathrm{A}} \end{aligned} \tag{4}$$

where Δu, Δv and Δw are the translation differences of a pair of opposing nodes in directions 1, 2 and 3, respectively. These differences must remain constant for all the pairs of corresponding nodes on opposite sides. The periodical boundary conditions were implemented in Abaqus/CAE.

2.2 Material model of fiber

The fibers were modeled as transversely isotropic elastic material respecting the non-linear behavior in the axis direction. The dependence of the longitudinal Young's modulus of the fibers on the strain in the axial direction of the fibers is[4]:

$$E_{11}(\varepsilon_{11}) = E_1^0 (1 + g\varepsilon_{11}) \tag{5}$$

where g is the coefficient describing the measure of non-linearity and E_1^0 is the initial Young's modulus of the fiber in the longitudinal direction.

It is assumed that fiber failure occurs when the failure index in the axial direction, defined as:

$$f_1 = \frac{\sigma_1^{\mathrm{f}}}{X_{\mathrm{T}}^{\mathrm{f}}} \tag{6}$$

where σ_1^{f} is stress in direction 1 in the fibers and $X_{\mathrm{T}}^{\mathrm{f}}$ is the tensile strength of the fibers, reaches the value of 1.

2.3 Material model of matrix

The matrix was modeled as an isotropic elasto-plastic material. The elastic parameters were the Young's modulus E_{m} and the Poisson's ratio μ_{m}. The Von Mises plasticity was used with the isotropic hardening and work-hardening function proposed in the form:

$$\sigma^y = \sigma_0^y + \frac{T_0 \bar{\varepsilon}_{\mathrm{p}}}{\left[1 + \left(\frac{T_0 \bar{\varepsilon}_{\mathrm{p}}}{\sigma_{\mathrm{A}}^y}\right)^n\right]^{\frac{1}{n}}} \tag{7}$$

where the yield stress σ^y depends on the equivalent plastic strain $\bar{\varepsilon}_{\mathrm{p}}$ and on additional material parameters where σ_0^y is the initial yield stress, T_0 is the tangent of the hardening curve for $\bar{\varepsilon}^{\mathrm{p}} = 0$, σ_{A}^y is asymptotic stress and n is a shape parameter.

The damage initiation criterion inspired by the criterion for the ductile damage of metals was used, where the fracture is assumed to be caused by the coalescence of voids in a matrix and the damage initiates when the following condition (damage factor) is fulfilled:

$$\omega_{\mathrm{D}} = \int \frac{\mathrm{d}\bar{\varepsilon}^{\mathrm{p}}}{\bar{\varepsilon}_{\mathrm{D}}^{\mathrm{p}}(\eta)} = 1 \tag{8}$$

where function:

$$\bar{\varepsilon}_{\mathrm{D}}^{\mathrm{p}} = \bar{\varepsilon}_{\mathrm{D}}^{\mathrm{p}}(\eta) \tag{9}$$

is the equivalent plastic strain at the onset of the damage and it is a function of the stress triaxiality factor:

$$\eta = \frac{\sigma_{\mathrm{h}}}{\sigma_{\mathrm{eqv}}} \tag{10}$$

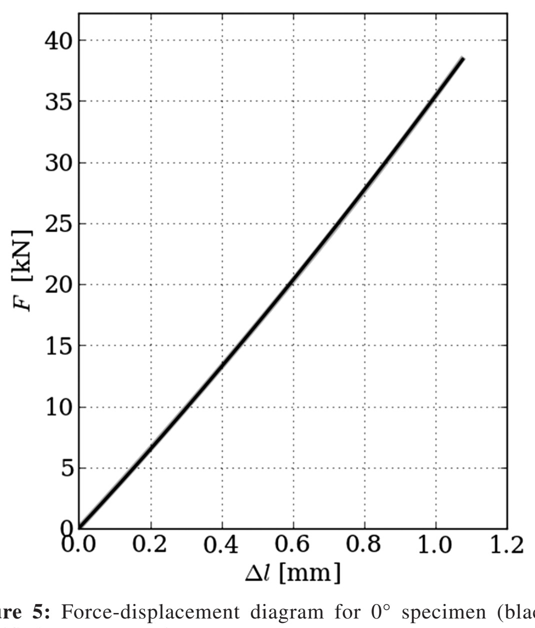

Figure 5: Force-displacement diagram for 0° specimen (black – identified, grey – experiments)

Slika 5: Diagram sila – raztezek za vzorec 0° (črno – identificirano, sivo – eksperimenti)

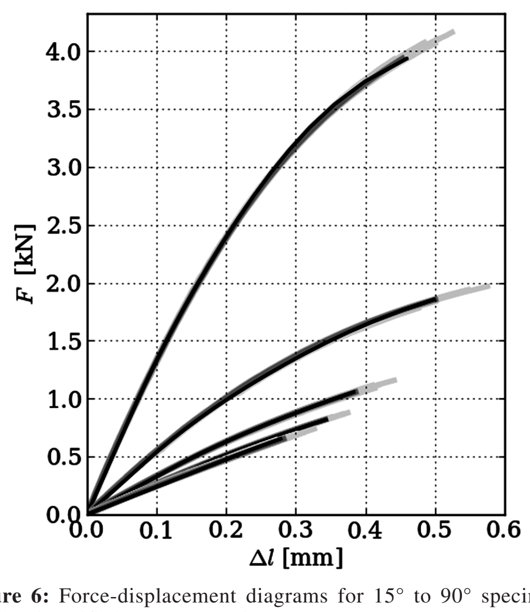

Figure 6: Force-displacement diagrams for 15° to 90° specimens (black – identified, grey – experiments)

Slika 6: Diagrami sila – raztezek za vzorce od 15° do 90° (črno – identificirano, sivo – eksperimenti) where σ_h is hydrostatic stress and σ_{eqv} is the Von Mises stress.[5]

2.4 Identification process

An optimization process, similar to the one used in[2], was performed using Python scripts, the optimization software OptiSlang and the finite-element system Abaqus. The identification consisted of two steps. The first step was to find the best combination of all the elastic and plastic material parameters by minimizing the difference between the experimental and numerical force-displacement diagrams. Moreover, the parameter $\Delta\theta$ which represents the inaccuracy when cutting the samples from the plate was identified. The second step was to identify the tensile strength of fibers and the function $\bar{\varepsilon}_D^p$ by minimizing the difference between the value of ω_D and 1 at the moment of fracture in the experiment. The dependency of the equivalent plastic strain on the onset of damage $\bar{\varepsilon}_D^p$ was proposed as a piecewise linear function with five control vertices.

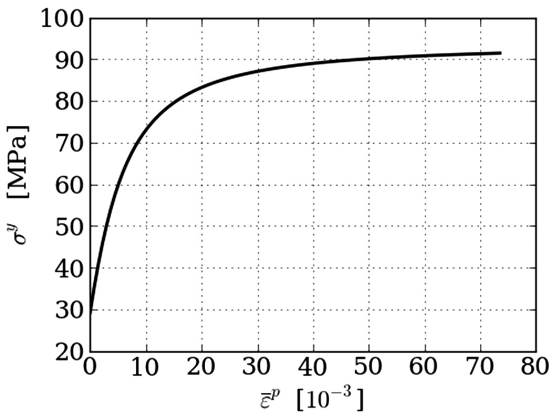

Figure 7: Work-hardening curve
Slika 7: Krivulja deformacijskega utrjevanja

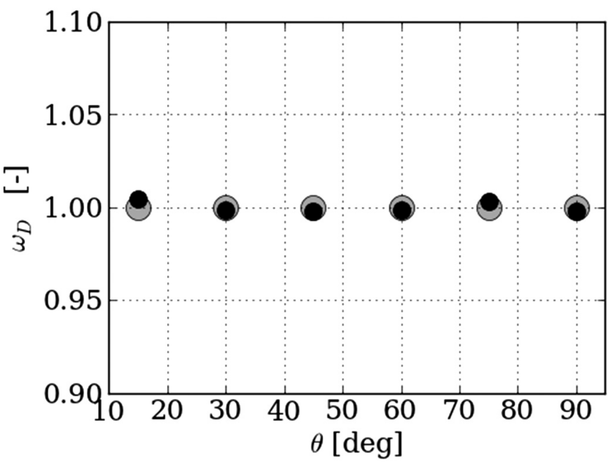

Figure 8: Maximum damage factors for specimens from 15° to 90° (black – identified, grey – experiments)
Slika 8: Faktorji maksimalne poškodbe pri vzorcih od 15° do 90° (črno – identificirano, sivo – eksperimenti)

3 DISCUSSION OF RESULTS

The identified parameters and the parameters given by the manufacturer are summarized in **Table 2**. The identified parameters correspond well with the ones identified in[2] where the analysis was performed in MSC.Marc software using eight-node brick elements.

The force-displacement diagram for the specimen 0° shows a convex shape caused by stiffening of the fibers (**Figure 5**). The force-displacement diagrams of the rest of the specimens have concave shapes caused by the matrix non-linearity (**Figure 6**). The identified shape of the work-hardening function is shown in **Figure 7**. The ability of the model to predict the initial damage of the matrix is visualized in **Figure 8**. The identified dependency of $\bar{\varepsilon}_D^p$ on the triaxiality stress factor is shown in **Figure 9**. The distribution of the damage factor for the matrix in the 45° specimen in the moment when the first damage to the matrix occurs is shown in **Figure 10**.

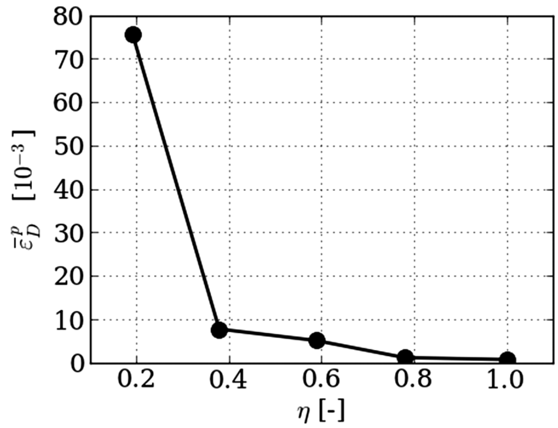

Figure 9: Dependency of the equivalent plastic strain at the onset of damage on the triaxiality stress factor
Slika 9: Odvisnost ekvivalenta plastičnega raztezka na začetku poškodbe od triosnega faktorja napetosti

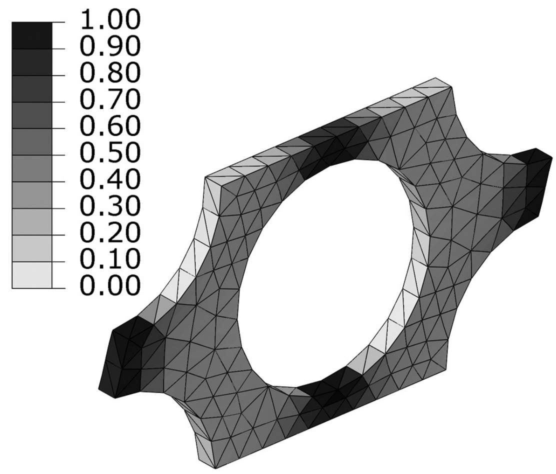

Figure 10: Distribution of damage factor (factor is not calculated for fibers, deformation is magnified 10 times)
Slika 10: Razporeditev faktorja poškodbe (faktor ni izračunan za vlakna, deformacija je povečana za 10-krat)

Table 2: Summarized parameters (i = identified, m = manufacturer)
Tabela 2: Povzeti parametri (i = ugotovljeni, m = proizvajalec)

Parameter	Units	Value	note
E_1^0	GPa	188.73	i
g	–	18.76	i
$E_2 = E_3$	GPa	15.00	m
$\mu_{12} = \mu_{13}$	–	0.30	m
μ_{23}	–	0.40	m
$G_{12} = G_{13}$	GPa	5.00	m
G_{23}	GPa	4.00	m
$X_T{}^f$	GPa	3.88	i
E_m	GPa	3.61	i
μ_m	–	0.40	m
σ_0^y	MPa	29.24	i
T_0	GPa	8.49	i
σ_A^y	MPa	93.54	i
n	–	1.35	i
$\Delta\theta$	°	–0.98	i

4 CONCLUSION

Finite-element analyses of simple tensile tests on unidirectional composite specimens with various fiber orientations were performed. The material parameters of the orthotropic non-linear elastic material of the fibers and of the non-linear elasto-plastic material of the matrix were identified. The simple failure (fiber) and the damage (matrix) criteria were successfully tested and good agreement with the experiments was achieved.

Future work will focus on an approximation of the dependency of the equivalent plastic strain at the onset of the damage on the triaxiality stress factor using a smooth function and modeling of the subsequent damage process using a cohesive-theory-based approach.

Acknowledgement

The work has been supported by the projects GA P101/11/0288 and European project NTIS – New Technologies for Information Society No. CZ.1.05/1.1.00/02.0090.

5 REFERENCES

[1] T. Kroupa, V. Laš, R. Zemčík, Improved Non-Linear Stress–Strain Relation for Carbon–Epoxy Composites and Identification of Material Parameters, Journal of Composite Materials, 45 (**2011**) 9, 1045–1057

[2] H. Srbová, T. Kroupa, R. Zemčík, Identification of the Material Parameters of a Unidirectional Fiber Composite Using a Micromodel, Mater. Tehnol., 46 (**2012**) 5, 431–434

[3] D. Roylance, Transformation of Stresses and Strains, Department of Materials Science and Engineering, Massachusetts Institute of Technology, Cambridge, 2001

[4] I. M. Djordjević, D. R. Sekulić, M. M. Stevanović, Non-Linear Elastic Behavior of Carbon Fibers of Different Structural and Mechanical Characteristic, Journal of the Serbian Chemical Society, 72 (**2007**) 5, 513–521

[5] S. S. Bhadauria, M. S. Hora, K. K. Pathak, Effect of Stress Triaxiality on Yielding of Anisotropic Materials under Plane Stress Condition, Journal of Solid Mechanics, 1 (**2009**) 3, 226–232