LOCAL EXPERIMENTAL DETERMINATION OF MATERIAL CHARACTERISTICS FOR POLYMERS RELATED TO FEM ANALYSIS

Martin Dobeš*, Jan Vrbka*

Polymer materials exhibit a high ductility. Determination of the yield strain as well as the break (ultimate) strain is usually done on the basis of tensile tests performed on standard samples and evaluated for normalized measured length, supposing the homogeneous material deformation along the sample axis. This experimental approach does not take into account the plastic strain concentration in small neck area during the final deformation phase before the sample rupture, what is typical for the plastics behavior. Application of so defined material characteristic in the case of Finite element method (FEM) analyses of real constructions made of TSCP plastics (typical semi-crystal polymer) led to significantly conservative (smaller) values of ultimate loads compared to the measured ones. A special experimental method making use of high-speed camera has been developed to determine the strain in defined small area of local strain concentration being also in correlation with the FEM element size. Application of the more realistic (higher local) break strain value in the case of FEM analyses of real TSCP constructions led to much better agreement between the calculated and measured stiffness and ultimate load values.

Keywords: *polymer, experiment, break limit, FEM analysis, high-speed camera*

1. Introduction

To obtain basic information about the constitutive relation of TSCP polymer material the standard tensile tests on flat samples for the measured active length of 50 mm were performed, see Fig. 3. The slow (quasi-static) monotone force loading has been applied. Measured stresses σ depended only on strains ε. No time dependence during the sample loading has been observed, therefore the elastic-plastic material model was utilized. Because of the almost linear character of the stress-strain curve at the first loading part the Young elasticity modulus E, yield stress σ_{y} and yield strain ε_{y} can be defined. The break (ultimate) strain ε_{u} corresponding to the sample rupture has been also determined for the measured active length of 50 mm, see Fig. 3. It means that we gained the average value of break strain. But near the loading end the polymer TSCP deformation becomes very inhomogeneous with the intensive strain concentration at the small neck area. This problem is not respected in the norm.

The computational model of material should be in good correlation with computational method applied for the stress, strain a safety analysis. In the case of the FEM nonlinear analysis taking into account the large strains and large displacements, the stress and strain

* Ing. M. Dobeš, prof. RNDr. Ing. J. Vrbka, DrSc., dr.h.c., Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, 616 69 Brno, CZ

values inside of each FE element are calculated, including also elements in neck area and material characteristics (esp. break strain) should respect this local necking phenomenon. But material data delivered by TSCP material producer [1] are determined as average values for already mentioned standard sample length and therefore are not proper for correct FEM analysis in the neck area exhibiting a high concentration of plastic deformation. To avoid this discrepancy a method of experimental detection of real break strain in the neck area was developed based on the extension measurement of defined sample part during the tensile test, making use of sample grid deformation. The complete test is performed with the help of high-speed camera, and the strain and stress are calculated from real records. The details of the method are explained below.

2. Experimental detection of the real TSCP ultimate strain used for FEM analyses

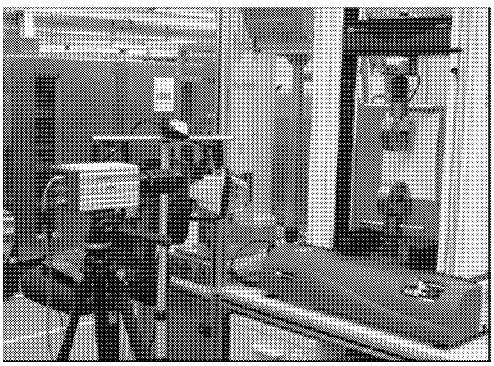
The material producer gives the break (ultimate) strain limit $\varepsilon_\mathrm{u} = 30\,\%$ for TSCP polymer utilized. The FEM analysis of selected constructions performed with this value led to very conservative results (much smaller) for construction ultimate load and ultimate displacement. It is evident that break strain values delivered by TSCP material producer do not correspond to the real break strain which occurs in the neck area.

The normalized tensile samples ISO 527-1/2 have been used for the experimental determination of the ultimate strain in the neck area. On each sample the horizontal grid with distance 5 mm was drown, see Fig. 1b. The sample deformation during the loading process has been recorded making use of the high-speed camera.

Coming from the known sample loading and measured extension of defined sample part the following material characteristics can be obtained for the stress, strain and strength analysis of constructions made of plastic material TSCP, for example

- Engineering stress – strain relation from extension of standard sample length 50 mm,
- ‘True stress’ – ‘true strain’ relation from extension of standard sample length 50 mm,
- Engineering stress – strain relation from extension of necked area
- ‘True stress’ - ‘true strain’ relation from extension of necked area
- Real break (ultimate) strain at the failure place (neck area)

This approach can be also used for any measured sample length corresponding for example to the finite elements size.



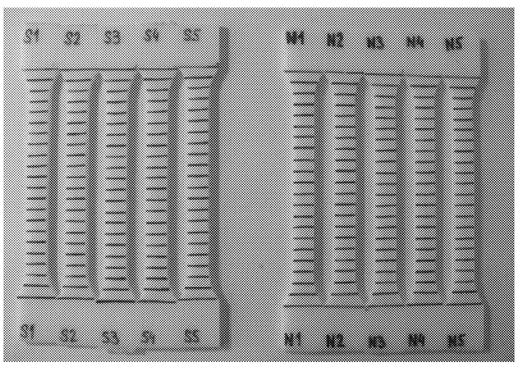


Fig.1: a) The measure system with high-speed camera, b) Tensile samples with horizontal grid (S1 . . . S5 – stabilized material, N1 . . . N5 – non stabilized material)

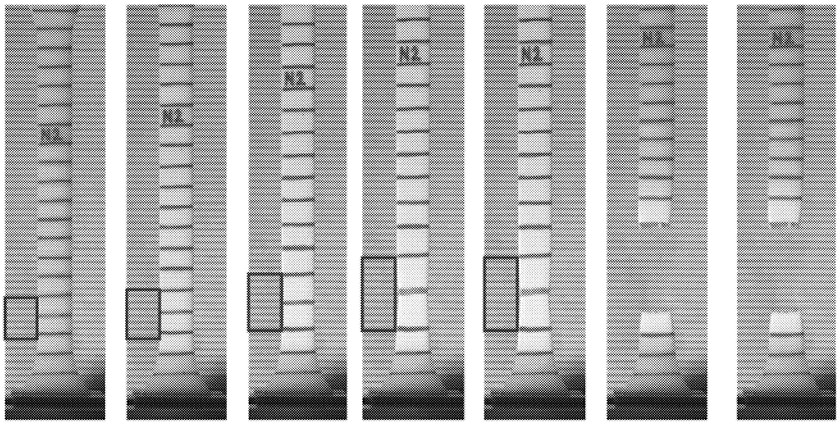


Fig.2: Deformation of the sample during tensile test (deformation loading velocity 50 mm/min)

3. Experimental results and calculation of the stress and strain values

Time course of the sample deformation during the loading is illustrated in following Fig. 2

During the sample loading the loading force F as well as extension Δl_i of defined sample length l_i are measured and engineering stress σ_{eng} and engineering strain $\varepsilon_{\mathrm{eng}}$ are calculated as follows

$$\sigma_{\mathrm{eng}} = \frac{F}{S_0} \;, \tag{1}$$

$$\varepsilon_{\mathrm{eng}} = \frac{\Delta l_i}{l_i} \;, \tag{2}$$

where S_0 denotes the initial cross section area.

'True stresses' σ_{true} and 'true strains' $\varepsilon_{\mathrm{true}}$ are then determined in standard way for the 1D stress state corresponding to the tensile test, assuming an incompressible material

$$\sigma_{\mathrm{true}} = \sigma_{\mathrm{eng}}\,(1+\varepsilon_{\mathrm{eng}}) \;, \tag{3}$$

$$\varepsilon_{\mathrm{true}} = \ln(1+\varepsilon_{\mathrm{eng}}) \;. \tag{4}$$

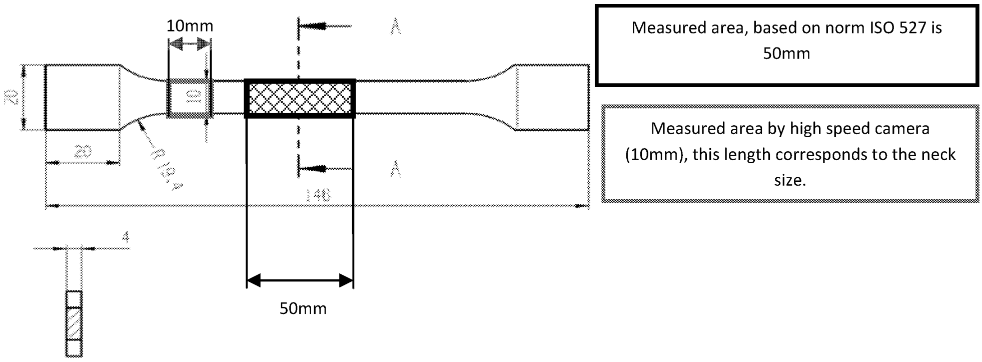


Fig.3: Tensile test sample ISO 527-2

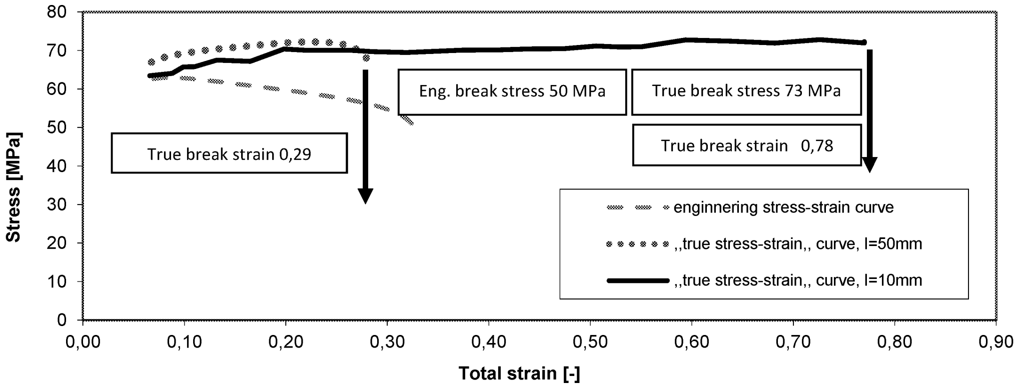


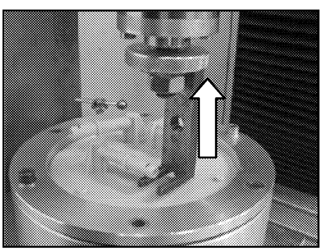
Fig.4: Comparison of stress-strain curves obtained from tensile test

In the paper two measured sample lengths were applied for the strain evaluation, namely standard length 50 mm and length of 10 mm, corresponding to the neck size, see Fig. 2. The both measured sample parts are shown in following Fig. 3.

The calculated engineering as well as 'true stress' – 'true strain' curves $\sigma_{\rm eng}(\varepsilon_{\rm eng})$ and $\sigma_{\rm true}(\varepsilon_{\rm true})$ are presented in Fig. 4 for measured sample part of length 50 mm (short curves) and length of 10 mm (long curve).

4. Verification of the experimental method

The proposed method of local determination of 'true stress' – 'true-strain' relation was verified by comparison of experimental results on real construction made of TSCP polymer material with the corresponding FEM computational simulation. The hydraulic connector (see Fig. 5) of the car fuel system was loaded by increasing loading force during the prescribed bending test till the rupture occurred. The dependence of the loading force on the displacement at the place of the outer force application was measured (see Fig. 5a) and ultimate (break) displacement $w_{\rm u}$ and ultimate (break) force $F_{\rm u}$ were determined, corresponding to the connector rupture. The outer load was realized by displacement increments in prescribed direction. The loading force was measured with the help of the strain gauge force transducer during the loading process. The fuel connector has to withstand the defined loading force. The loading velocity at the load place was small – $v = 20\,$mm/min. The corresponding maximal strain velocity at the sample necked area of length 10 mm by tensile test was $d\varepsilon/dt = 0.040\,s^{-1}$ and at the most deformed connector area $d\varepsilon/dt = 0.033\,s^{-1}$. The difference is not significant and moreover the both values are small enough so that the mechanical problem solved can be considered as a quasi-statically one.


(a)

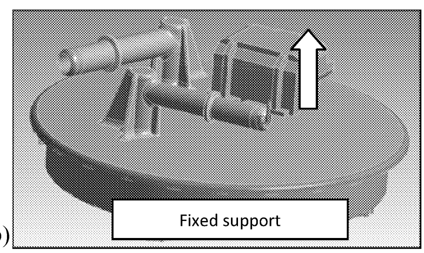
(b)


Fig.5: Loading (a) and boundary (b) conditions

The deformation variant of Finite Element Method (FEM) was utilized for computational stress and strain modeling at the hydraulic connector. The mechanical problem to be solved exhibits a strong nonlinear character from geometrical as well as material point of view, because of large displacements and large strains and nonlinear elastic-plastic material behavior. A principle of virtual work has been employed for FEM mathematical formulation and the total Lagrangian formulation has been used. The virtual work of internal forces is here expressed with the help of 2nd Piola-Kirchhoff stress tensor S_{ij} and Green-Lagrange strain tensor E_{ij} going out of the following relation [2]

$$\int \sigma_{ij}\,\delta\varepsilon_{ij}\,\mathrm{d}V^{\mathrm{t}} = \int S_{ij}\,\delta E_{ij}\,\mathrm{d}V^0 \ , \tag{5}$$

where σ_{ij} means Cauchy (true) stress tensor and ε_{ij} is Cauchy strain tensor. The element volume $\mathrm{d}V^0$ is related to the reference (initial) geometrical configuration and $\mathrm{d}V^{\mathrm{t}}$ corresponds to the instantaneous (deformed) one.

The nonlinear mechanical problem is solved in incremental way for defined outer loading increments $\Delta\mathbf{r}$. For each loading step the iteration procedure is applied, governed by following FEM equilibrium equation

$$\mathbf{K}^{(i)}\,\Delta\mathbf{U}^{(i)} = \Delta\mathbf{r} - \Delta\mathbf{f}^{(i)} \ , \tag{6}$$

where $\mathbf{K}^{(i)}$ means tangential stiffness matrix, $\mathbf{U}^{(i)}$ is vector of displacement parameters increments and $\Delta\mathbf{r}$ and $\Delta\mathbf{f}^{(i)}$ are increments of external loading resp. internal forces vectors. The iteration procedure is stopped in the moment, when the right side (unbalanced forces) in equation (6) becomes negligible. The Newton-Raphson method was utilized for numerical solution.

ANSYS FEM code has been adopted for the problem solution [3]. The outer loading was realized through the prescribed small displacement increments, with respect to the material as well as geometrical nonlinearity.

The FEM mesh of the plastics connector body is illustrated at Fig. 6. The connector tube internal diameter was $d = 8\,\mathrm{mm}$ with wall thickness $t = 2\,\mathrm{mm}$. 3D quadratic tetrahedral elements SOLID 187 and hexahedral elements SOLID 186 with the element size of 0.3 mm in the most stressed connector parts have been used to create the FEM geometrical model. The FEM geometrical model consists of 88 333 elements and 144 463 nodes.

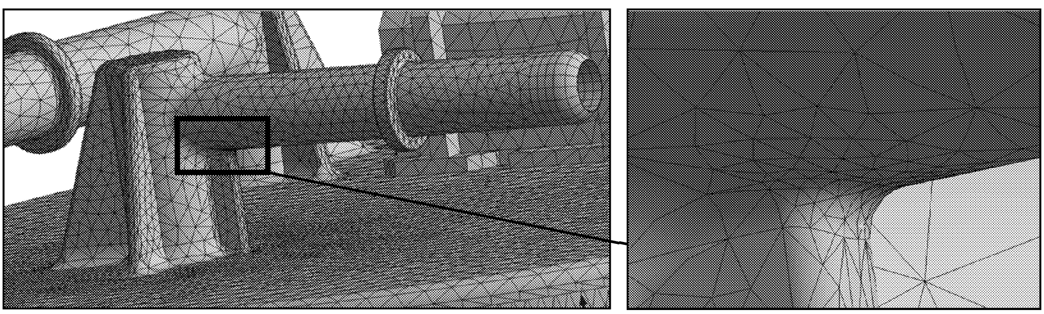


Fig.6: FEM mesh of the hydraulic connector

FEM calculations have been performed with two material models. First the 'true stress' – 'true strain' relation (Fig. 4) experimentally obtained from tensile tests was used to describe TSCP polymer material behavior. In the elastic part linear material behavior was assumed with elasticity modulus E = 2650 MPa and Poisson's ratio $\mu = 0.4$. Because of the TSCP connector material differs little bit from the tested material the modified material characteristics have been taken into account. Following the material producer data the elasticity modulus and Poisson's ratio were here $E = 2850$ MPa and $\mu = 0.4$ and the break (ultimate) strain ε_u exhibits about 20 % bigger value compared to the material used for the tensile test, what leads to $\varepsilon_u = 0.35$ calculated for measured sample length $l = 50$ mm and to $\varepsilon_u = 0.94$ for the local neck area. The measured 'true stress' – 'true strain' curve from Fig. 4 has been extrapolated to this higher value.

Boundary conditions correspond to the real test. The connector flange is placed in the fixing device, see Fig. 5b. The loading place is defined by the company norm.

5. Comparison of experimental results and computational analysis

The measured relation of loading forces versus displacements at loading place of the hydraulic connector is presented in Fig. 7 by dotted curve together with results achieved by FEM computational simulations – full curve marked as *FEM_eng* corresponds to the engineering stress-strain relation at Fig. 4, dashed curve marked as *FEM_true* is relates to the 'true stress' – 'true strain' relation at Fig. 4 both for standard TSCP material and dot-and-dash curve marked as *FEM_con_true* corresponds to the 'true stress' – 'true strain' relation at Fig. 4 modified for the connector TSCP material with higher break strain. The FEM calculation was performed for nonlinear mechanical problem from the geometrical point of view, (large strains, large displacements) as well as material one.

Lower picture in Fig. 7 shows relation between vertical connector displacement and average von Mises equivalent total strain calculated by FEM in the most deformed finite elements in the strain concentration area defined by company experience. For the break

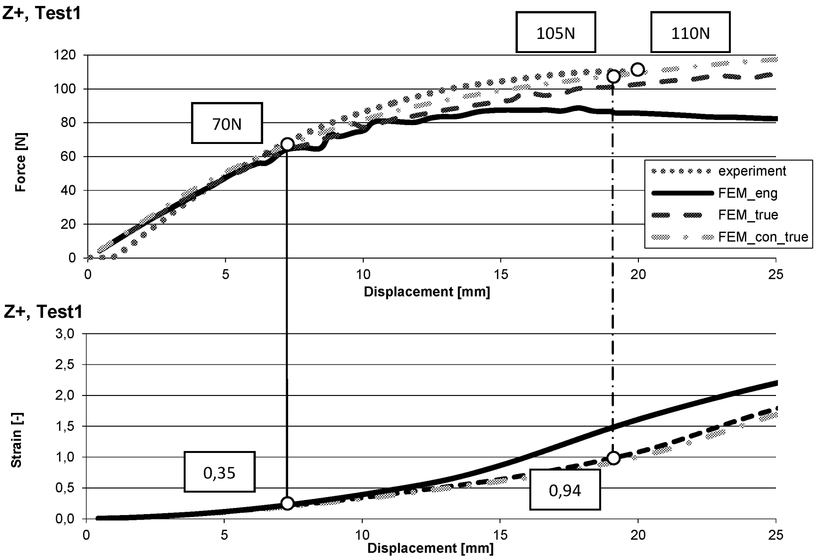


Fig.7: Dependences of the loading force and equivalent strain on the displacement

(ultimate) displacement w_u – this parameter can be taken as an ultimate (break) strain – $\varepsilon_u = 0.95$. The value agree relatively well with the value determined with the help of proposed experimental method for standard TSCP material and for neck area of length 10 mm – $\varepsilon_u = 0.78$, see Fig. 4. As it was already mentioned, the hydraulic connector material differs little bit from standard TSCP material S2320 used for tensile tests and corresponding ultimate (break) strain $\varepsilon_u = 0.94$ is in very good agreement with $\varepsilon_u = 0.95$ obtained from the FEM hydraulic connector stress and strain analysis (corresponding curve in Fig. 7 marked as *FEM_con_true*). It is evident from Fig. 7, that application of engineering material data (full curves) is not acceptable for investigated problem, exhibiting large displacements and large strains.

Application of the break strain value $\varepsilon_u = 0.35$ from material producer data, defined for whole measured length $l = 50$ mm in the FEM computational stress and strain simulation of hydraulic connector led to much smaller break (ultimate) displacement w_u as well as break (ultimate) load F_u – $w_{up} = 7$ mm, $F_{up} = 70$ N (see Fig. 7) compared with values obtained on the basis of proposed experimental method, taking into account the local concentration of plastic deformation in neck area and – $w_u = 19$ mm, $F_u = 105$ N (see curve *FEM_con_true* in Fig. 7) and from experiment performed with hydraulic connector – $w_u = 20$ mm, $F_u = 110$ N (difference in forces F_u about 5 %) – see Fig. 7.

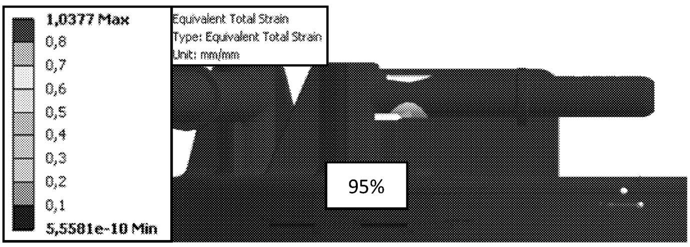


Fig.8: Distribution of the von Mises equivalent total strain

In the loading test the hydraulic connector was destroyed in the place of the maximum principal stress (tensile stress), which is identical with the place of the maximal von Mises equivalent total strain. The strain distribution in the connector body is presented in Fig. 8. The average value from most deformed finite elements can be taken as the break (ultimate) strain $\varepsilon_u = 0.95$ of the material used.

6. Conclusion

In the article a new experimental method specially proposed for plastics materials has been presented to determine the strain in defined small area of local strain concentration (neck area) during the specimen tensile test, making use of the speed camera record. The 'true stresses' and 'true strains' are calculated here during the sample loading process and corresponding 'true stress' – 'true strain' curve $\sigma_{true}(\varepsilon_{true})$ till the sample rupture is determined. For the investigated strain concentration area of length 10 mm on the plastics material TSCP sample the calculated break (ultimate) strain ε_u was about 2.7 times higher than the data from material producer. The proposed experimental method has been verified with the help of the loading experiment on the real hydraulic connector and following comparison of the measured displacements and loading forces with those obtained on the basis of FEM computational experimental method proposed. Application of this more realistic material values for the FEM analysis of the real hydraulic connector made of TSCP material led to the much better agreement between calculated and measured break (ultimate) loads and displacements values.

Acknowledgement

This study was supported by the grant project FSI-S-11-11/1190.

References

[1] Material characteristics of the TSCP, http://www.basf.com/group/corporate/en/
[2] Holzapfel G.A.: Nonlinear solid mechanics, John Wiley & Sons, 2001, Chichester, New York
[3] ANSYS 14.5-help (2012), http://www.ansys.com/Support/Documentation

Received in editor's office: July 30, 2013
Approved for publishing: February 25, 2014