NUMERICAL SIMULATION OF TENSILE TESTING OF PE 80 POLYMER SPECIMENS

by

***Simon A. SEDMAK*[a*], *Zorana Z. GOLUBOVIĆ*[b], *Alin Constantin MURARIU*[c], and *Aleksandar S. SEDMAK*[b]**

[a] Innovation Centre of Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia
[b] Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia
[c] National R&D Institute for Welding and Material Testing, Timisoara, Romania

Original scientific paper
https://doi.org/10.2298/TSCI170219203S

The aim of this paper is to present the behaviour of specimens made of polyethylene material PE 80, subjected to tensile load until failure. Measurements of the temperature distribution have been done using the infrared thermography during specimens loading. Finite element analysis was performed in ABAQUS software, where numerical models were made based on the thermograms and force-displacement diagrams obtained from these experiments. Afterwards, results from the simulation were compared with the experimental results and it was determined in which way the model can be optimized so that these results comply at an acceptable level. Numerical model has shown that the highest values of plastic strain were located near the notch. Value of this plastic strain is several times greater than the values in the remaining parts of the specimen. The numerical analysis also determined that defining the load in displacement form was a much better solution than defining it using the force, since the results have shown much better compliance, and the calculation time was much shorter in this case.

Key words: *PE 80 polymers, infrared thermography, finite element method, visco-plasticity, thermogram*

Introduction

Polyethylene has found its application in the manufacturing of pipe systems, primarily for the transport of water and gas, wherein it has been used for over 40 years [1-3]. The main advantages over other materials used for pipelines include good weldability, corrosion resistance, and light weight [4]. One of the more commonly used polymers for this purpose is the PE 80, due to its exceptional performances in terms of failure behaviour. The PE 80 polyethylene is characterized by minimum hoop stress of 8 MPA, for up to 50 years at the temperature of 20 °C [5]. The aim of this research was to develop a model for describing the behaviour of such pipe systems under tensile load, to which they are typically subjected to during exploitation.

One of the important factors to take into account when it comes to PE 80 pipelines is the ageing behaviour of this material. Ageing may cause changes in the material properties which could in turn significantly affect the behaviour and integrity of such pipelines. In this paper, special focus will be on the influence of temperature on the plastic behaviour of specimens

* Corresponding author, e-mail: ssedmak@mas.bg.ac.rs

made of PE 80 polyethylene. For this purpose, numerous experiments with different loading rates of PE 80 specimens have been performed, and a numerical model was developed using finite element method [6], in order to simulate these experiments as realistically as possible.

The experiments involved the tensile tests of specimens until failure, wherein measuring of the temperature distribution has been done using infrared thermography [7] during the loading. Based on the thermograms and force-displacement diagrams obtained from these experiments, 3-D numerical models were made in ABAQUS software, for the purpose of finite element analysis. Subsequently, the results obtained from this simulation were compared with the experimentally obtained ones, which enabled determining of the ways in which the model can be optimized so that these results comply at an acceptable level.

Materials and method

The experiments were performed for varying load rates of 100, 250, and 500 mm/min, at room temperature, since the loading rate can have a significant influence on the tensile strength of polyethylene materials [8]. All test specimens were notched in their central section. In this research, the specimen with the loading rate of 250 mm/min was selected for the numerical simulation. The experimental part of this research was carried out within the laboratories of ISIM, Timisoara, Romania. The experiment involved subjecting the specimens to tensile load and the stress-strain diagram, as well as the temperature were measured. For the purpose of measuring the temperature field, the infrared thermography method was used. This method has numerous applications, one of which is the quality assessment of high-density polyethilene (HDPE) pipes [3, 7]. Infrared thermography involves the use of special thermographic cameras which detect radiation within the infrared spectrum, emitted by the specimen, and based on its levels, determine the temperature distribution. The temperature magnitude increases with the stress levels in the specimens, and thus these two quantities can be correlated. As a result of these experiments, thermograms and videos showing the behaviour of specimens during tensile testing in real time were obtained and used as a base for the numerical simulation.

For the purpose of developing a numerical model, the mechanical properties of PE 80 polyethylene needed to be determined, including Young modulus, Poisson's ratio (for the elastic analysis), the stress-strain curves (for plasticity), and the visco-plastic power law parameters [9] for the viscous behaviour analysis. The behaviour of the numerical model was defined in three stages: the elastic stage, which occurs at the beginning of the loading, the plastic stage, which follows the elastic one, and finally the visco-plastic stage, whose initiation can be identified on the force-displacement diagram as the moment when the displacement starts to increase with time, while the force remains constant. In this case, visco-plastic behaviour was defined as time dependent, as opposed to strain dependent, since the time dependent model is typically used for modelling of significant plastic strain caused by thermo-mechanical loading, whereas the strain dependent models are better suited for fractures caused by shear stresses [10, 11]. For this reason, strain hardening was not included in the simulation. Regardless of the nature of visco-plastic behaviour (time or strain dependent), in ABAQUS it is based on the creep power law [11], according to which creep (visco-elastic) strain is defined by:

$$\varepsilon^c = A\sigma^n t^m \tag{1}$$

where A, n, and m are the temperature dependent power law material constants, and are used as parameters in the material defining stage of the numerical calculation.

Each of these stages was simulated using the appropriate behaviour in the material defining step of the analysis in ABAQUS. Elastic parameters were determined from literature (Poisson's ratio) [12] and from the force-displacement (Young modulus) diagram for the P250 specimen, fig. 1, whereas the same diagram was used in order to obtain the true stresses and strains used for simulating the plastic stage of specimen deformation. These parameters are shown in tab. 1. The experimental set-up which was used for infrared thermography can be seen in fig. 2.

True stresses and strains are obtained based on the stress and strain values (σ_{nom} and ε_{nom}) from the σ-ε diagram, using eqs. (1) and (2):

$$\sigma_{tru} = \sigma_{nom}(1 + \varepsilon_{nom}) \tag{2}$$

$$\varepsilon_{tru} = \ln(1 + \varepsilon_{nom}) \tag{3}$$

In this case, the nominal stress and strain values were determined based on the available force and displacement diagrams.

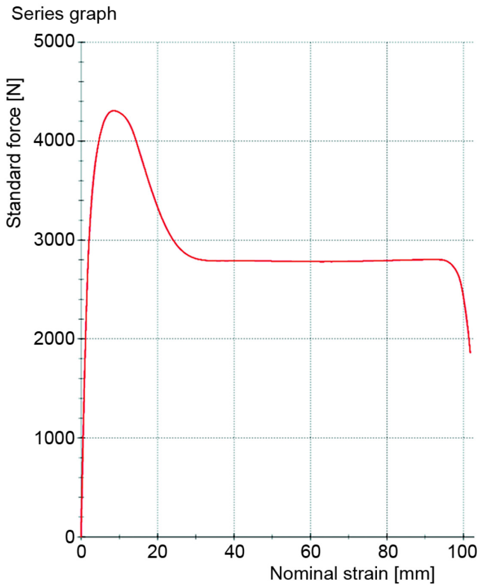

Figure 1. The experimentally determined force-displacement diagram for P250 specimen

Table 1. Mechanical properties of the PE 80

Material	Young modulus [MPa]	Poisson's ratio [–]	True stress [MPa]	True strain [–]
PE 80	1340	0.48	12.608 30.787	0 0.0686

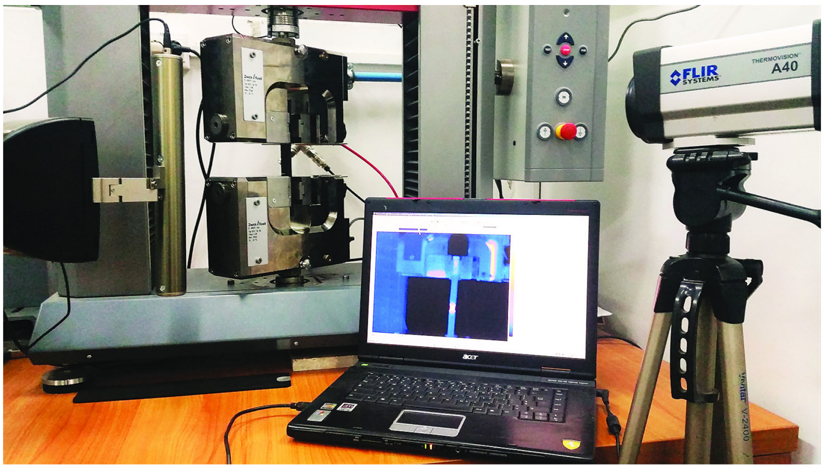

Figure 2. Experimental set-up used for infrared thermography

The temperature distribution diagrams (thermograms) were used in order to determine the relevant temperatures for each behaviour stage (elastic, plastic, and visco-plastic), combined with the force-displacement curve. Elastic behaviour included the loading interval which corresponds to the linear part of the force-displacement, whereas the plastic behaviour takes place between the elastic and visco-plastic parts of the diagram (visco-plastic behaviour starting point was described earlier). Examples of such thermograms can be seen in fig. 3, wherein the results are given in [°C]. The lines in the diagrams represent the changes of temperature over time for the axial and lateral directions, denoted as L01 and L02 in each of the images. These temperatures were then used as parameters during the material defining stage in ABAQUS. The parameters used for visco-plastic behaviour are shown in fig. 4, defined for both the beginning of the visco-plastic stage and the maximum temperature achieved at the end of the loading. It should be noted that all temperatures were defined in Kelvins. The initial temperature for the elastic behaviour was adopted as 295 K, corresponding to the room temperature, whereas the final temperature for this stage was 309.8 K. Temperatures used for the simulation of plastic behaviour ranged from 309.8 to 345.7 K. The parameters used for visco-plastic behaviour are shown in fig. 4, defined for both the beginning of the visco-plastic stage and the maximum temperature achieved at the end of the loading, with temperature values of 345.7 and 364.8 K at the beginning and the end of this stage, respectively. Figure 5 shows the geometry of the specimen used, along with the central notch whose dimensions can also be seen.

In the following section of the paper the 3-D model is shown, including its mesh and the load, which was applied as displacement corresponding to the experimentally determined strain at failure (102.5 mm). The load started at 0 and was gradually increased in accordance with predefined amplitude. The amplitude was defined in a way that ensured that the displacement (strain) initially increased at a slower rate, whereas after reaching the plasticity and later the visco-plastic deformation, the rate at which it increased was significantly accelerated. Boundary conditions were defined as fixed on the side of the specimen opposite to the loaded side. In addition, the lower surface of the specimen was constrained in the vertical, y-direction, in order to prevent significant deformation in that direction, so that the model would more accurately resemble the real testing conditions.

The numerical model mesh was made using C3D8R hex elements. The mesh around the notch area was made finer, since this was the location where highest plastic strain was expected to occur. The model contains a total of 97.550 elements and 105.300 nodes. The numerical model is shown in fig. 6. Also, shown in this figure are the boundary conditions, and the applied load (in form of a 102.5 mm displacement).

Results and discussion

The results obtained for plastic strain in the numerical model have shown, as expected, that the highest values were located near the notch, like in the case of the physical experiment. The highest plastic strain was 90,91%, in the corners of the notch area, as can be seen in fig. 7. It can also be seen that this value is several times greater than the plastic strain levels in the remaining parts of the specimen. The slightly increased plastic strain at the fixed end of the model was a consequence of boundary conditions, and as such, was not considered relevant for this calculation. The specimen in the numerical simulation also deformed in the similar way as the experimental one, due to way in which the boundary conditions were defined and applied. It should also be noted that the numerical specimen did not break, as opposed to the real specimen used in the experiment.

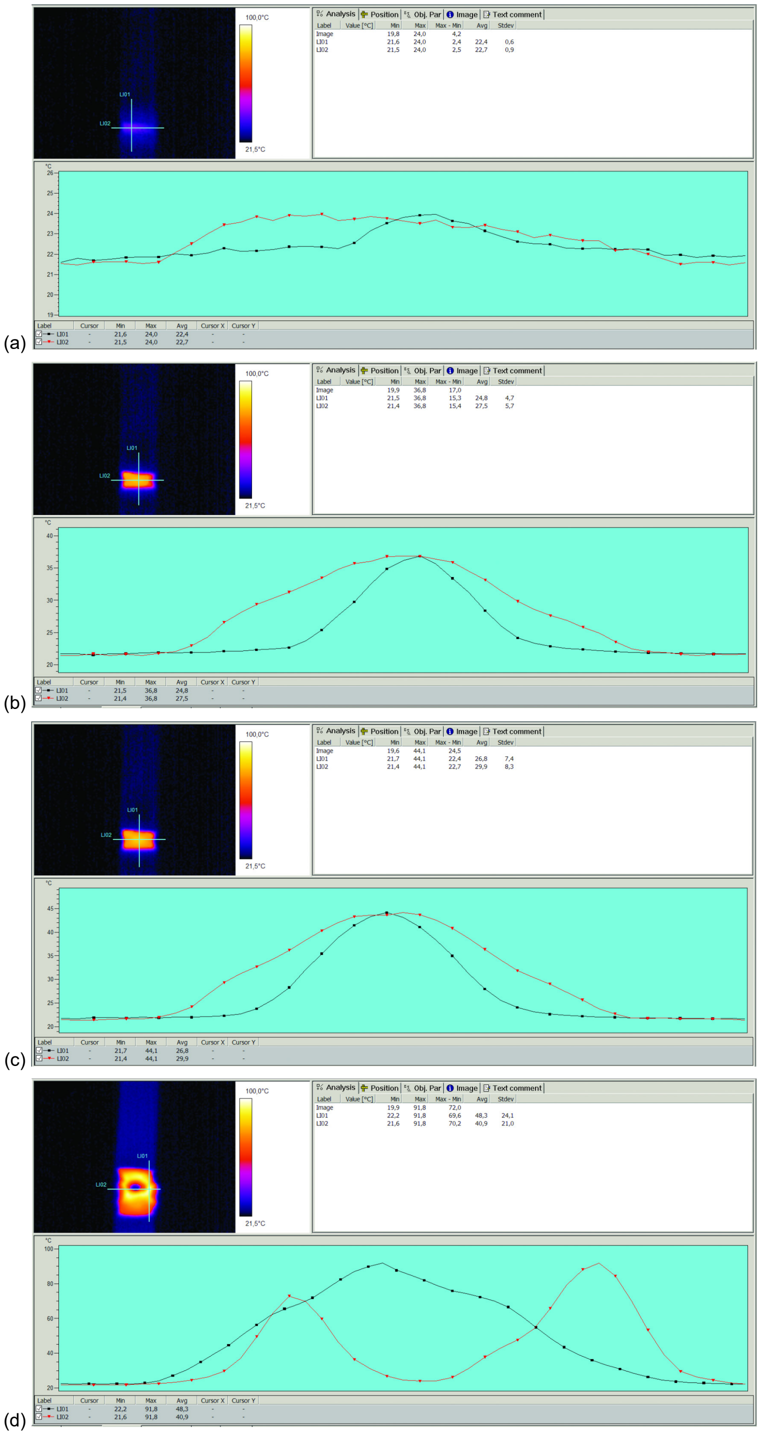

Figure 3. Thermograms for specimen P250; (a) elastic stage, (b) plastic stage, (c) visco-plastic stage, (d) specimen failure

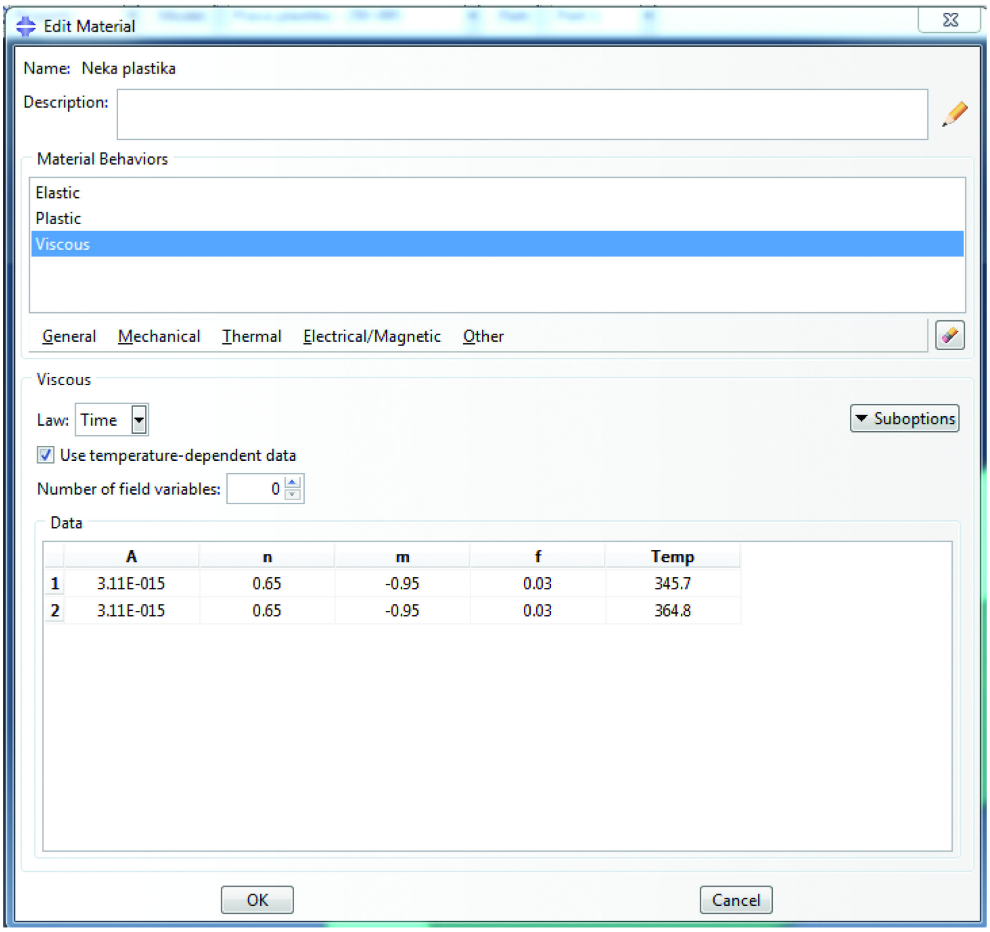

Figure 4. Visco-plastic analysis parameters [8]

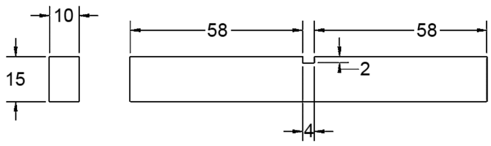

Figure 5. Geometry of the specimen including the notch

The stress concentration, which resulted in considerable plastic strain in the notch, in comparison to the specimen as a whole, corresponds to the temperature distribution at failure point in the tensile test, as can be seen in fig. 8, which was taken from the specimen P250 video. It can be seen here that the temperature at the central part of the specimen (with the notch) is significantly greater than the temperature in the remaining parts of the specimen.

It should be taken into account that the temperature concentration in the plastic strain area was the consequence of deformation that occurred due to the tensile load applied during the experiment, *i. e.* that the high stresses and strain that have occurred during the plastic and visco-plastic stages of the experiment, which were then used as parameters for the finite element model, resulted in the significant increase in temperature of the specimen around the notch.

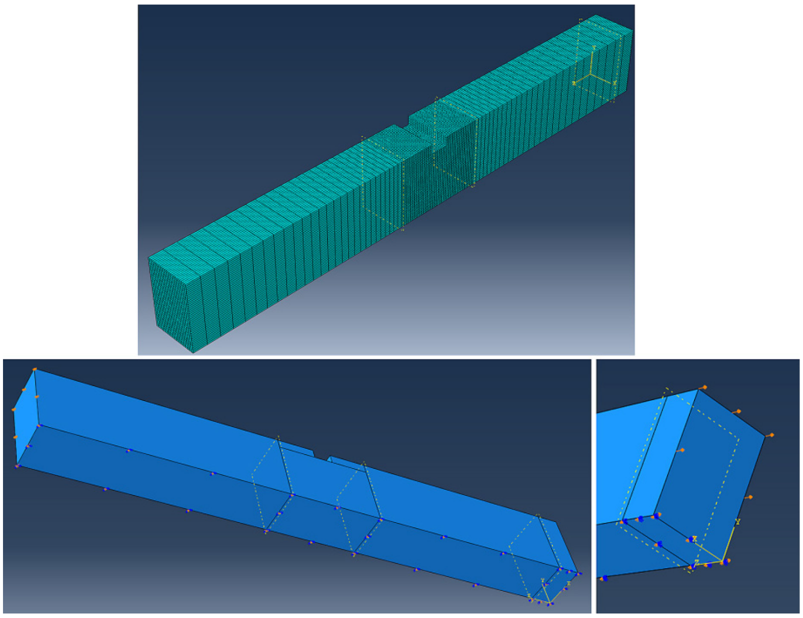

Figure 6. The 3-D numerical model and its boundary conditions and load

The strain obtained by the experiment was around 85%, as can be seen from the force-displacement diagram (determined as the maximum displacement – 102.5 mm, divided by the initial specimen gauge length of 120 mm), whereas the numerically obtained value was 90.91%, thus the results from the experiment and the numerical simulation have shown very good compliance. In addition, the plastic strain distribution corresponds to the real conditions, as in both cases the highest strain occurred in the notch area, where failure occurred in the case of the tensile test.

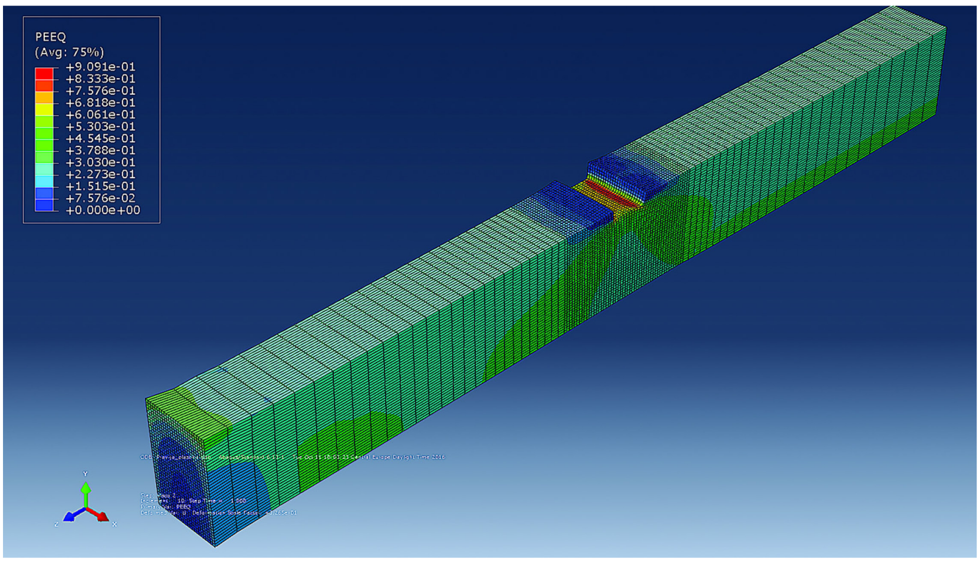

Figure 7. Results for plastic strain (PEEQ) obtained by the numerical simulation

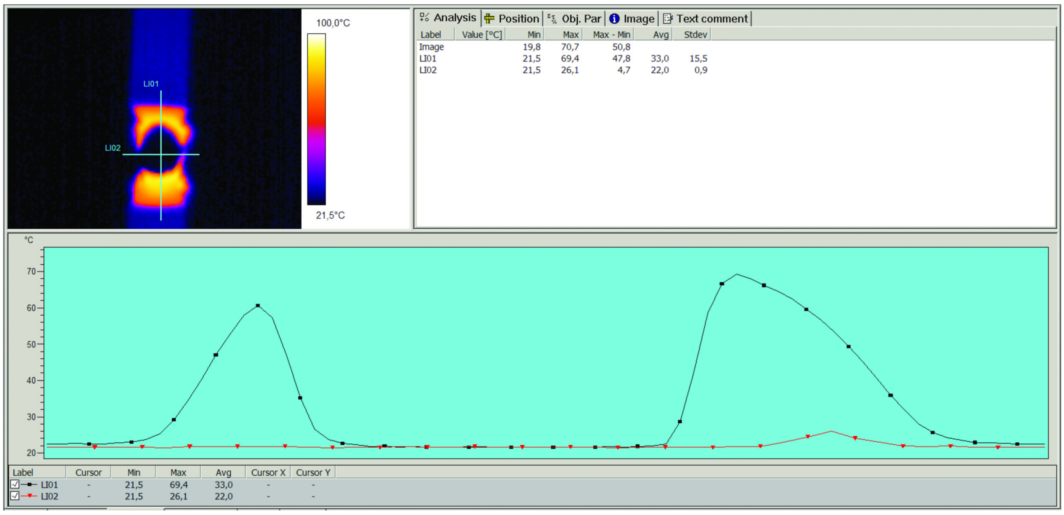

Figure 8. Temperature distribution during the failure of the real P250 specimen

The values obtained from the numerical model were also influenced by the way in which the loading amplitude was defined, as it needed to be adjusted in order to better relate to the behaviour of the real specimen (in terms of displacement). In addition, in order to define the amplitude so that the load increases at a slower rate in the beginning, and in order to correspond to the experiment, this was also done in order to prevent early occurrence of considerable plastic strain, which would result in very long calculation times, and in most cases, in the calculation being aborted due to these extreme values. Defining the amplitude in the most realistic manner possible should be taken into account as one of the factors when models like this one are created, in order to find the compromise between the real conditions and the load used in the numerical calculation.

The numerical analysis also determined that in this particular case, defining the load in displacement form was a much better solution than defining it using the force (as pressure applied to the surface where the specimen was held by the test machine grips), since the results have shown much better compliance, and the calculation time was much shorter in this case.

Conclusions

The 3-D numerical simulation of the PE 80 specimen behaviour, subjected to tensile load has been presented in this paper, taking into account the effects of temperature on the material properties. The simulation was based on thermograms and force-displacement diagrams experimentally obtained, and it involved three deformation stages with elastic, plastic, and visco-plastic behaviour. The results obtained from the numerical model have shown a good compliance with the experimental ones, in terms of equivalent plastic strain values and its distribution in the specimen.

This analysis has shown that there is a possibility of developing representative 3-D numerical models in order to simulate the visco-plastic behaviour of polymer specimens subjected to tension. Further research should be focused on creating of additional models of other similar specimens in order to verify the accuracy of the simulation, based on the experimental results obtained by tensile testing of PE 80 specimens under different load conditions, in terms of load rate (including the rates of 100 and 500 mm/min). Visco-plastic parameters and loading conditions should also be taken into account, as the way in which they are defined affects the obtained results. Hence, further work should involve the optimisation of such numerical models, which should include obtaining of accurate data about the power law parameters for the polyethylene material used in this research (PE 80), since there is still significant room for improvement.

References

[1] Murariu, A., *et al.,* Long-Term Behaviour of Polyethylene PE 80 Pressurized Pipes, in Presence of Longitudinal Simulated Imperfections, *Materiale Plastice*, *47* (2010), 3, pp. 263-266
[2] Frank, A, *et al.,* Numerical Assessment of PE 80 and PE 100 Pipe Lifetime Based on Paris-Erdogan Equation, *Macromolecular Symposia*, *311* (2012), Jan., pp. 112-121
[3] Murariu, A., Lozanovic-Sajic, J., Temperature and Heat Effects on Polyethylene Behaviour in the Presence of Imperfections, *Thermal Science, 20* (2016), 5, pp. 1703-1712
[4] DesLauriers, P. J., *et al.*: A Comparative Study of Multimodal *vs.* Bimodal Polyethylene Pipe Resins for PE-100 Applications, *Polymer Engineering and Science*, *45* (2005), 9, pp. 1203-1213
[5] ***, https://www.ineos.com/globalassets/ineos-group/businesses/ineos-olefins-and-polymers-europe/brochures/ ineos_flyer_hdpe_pipes.pdf
[6] Doncheva, E., *et al.,* Finite Element Analysis of Fracture Resistance Parameters for Stationary Semi-elliptical Surface Cracks in High Strength Steel, *Structural Integrity and Life*, *15* (2015), 3, pp. 131-134
[7] Murariu, A., *et al.*, Application of Thermography in Materials Science and Engineering, in: *Infrared Thermography* (Ed. R. V. Prakash), InTech, 2012
[8] Murariu, A., *et al.,* Tensile Behaviour of Polyethylene Under Different Loading Rates in the Presence of Imperfections, *Structural Integrity and Life*, *16* (2016), 1, pp. 15-18
[9] Zare-Shahabadi, A., Hajimoradi, A., Finite Element Modeling of Creep in Polymer/CNT Nanocomposites, *Indian J. Sci. Res*., *1* (2014), 2, pp. 929-933
[10] Ghoneim, H., Chen, Y., A Viscoelastic-Viscoplastic Constitutive Equation and its Finite Element Implementation, *Computers & Structures, 17* (1983) 4, pp. 499-509
[11] ***, http://www.ewp.rpi.edu/hartford/~ernesto/F2005/CINVESTAV/Notes/ch6.pdf
[12] ***, Web: http://www.gpsuk.com/uploads/docs/1636.pdf

Paper submitted: February 19, 2017
Paper revised: September 18, 2017
Paper accepted: September 22, 2017