



Hybrid thermographic-FEA method for evaluation of the plastic strains in the vicinity of the crack in loaded specimen

M. Žďárský, J. Valach

Czech Technical University in Prague, Faculty of Transportation Sciences, Department of Mechanics and Materials, Na Florenci 25, 110 00 Prague 1

xzdarsky@fd.cvut.cz; valach@fd.cvut.cz

KEYWORDS. Thermoplasticity; Finite element method; CT specimen.

INTRODUCTION

The development of reliable method to measure deformations belongs to the most important tasks in study of constructions and materials nowadays. The availability of these methods is a prerequisite for effective application of materials in all technical areas because they the reliable knowledge of true state of constructions allows us to use them safely and extend their life span. The one of way how to reach these goals is an analysis of behavior of metals during large plastic deformations. If we will know the processes which are responsible for cumulations and growth of defects in materials we can identify the undesirable symptoms prior catastrophic failure. The aim of the presented research is to find out more about the plastic strain processes by using thermo-camera and FEM.

DESCRIPTION OF METHOD

Thermoplasticity is a phenomenon which is connected to an irreversible processes characterizing plastic deformation. There is a growth of temperature in the deformed places due to dissipated energy from the plastic work - more than 90% plastic energy is converted into heat and released from the specimen. Therefore if we know the temperature field evolution during the deformation, we have an opportunity to find out the distribution of plastic deformation in the specimen.

EXPERIMENT

The presented hybrid method consists of the two main parts – the experimental and the numerical. The aim of experiment was record input data which reflected the real response of the material for the known loading. The CT specimen was specially treated on its surface to enhance the thermal emissivity by polishing and subsequent painting. Then the specimen was fixed in a loading device and after that a dynamometr and an extensometer were installed. The last it was installed a thermo-camera FLIR and then whole system was synchronized to be able to simultaneously record the data signals from the dynamometr - forces, from the extensometer – displacements and from the thermo-camera the temperature field).

At the beginning of the test the CT specimen started to deform elastically. Then it deformed plastically in the vicinity of crack because of the biggest concentrations of stresses reaching the plasticity criterion. The thermoplastic phenomenon was also observed and easily detected in places where the growth of temperature was the highest. As the experiment

continued the deformations extended from the tip of notch together and created the temperature field with a noticeable gradient. Later the situation become more difficult to describe, because the places with the temperature increase on the surface, but due to the conduction of heat from the inner parts of the specimen. But the most exposed places were always easily detected.

The result of experiment was a table with the values of time, forces, displacements and the pictures their temperature response. These data were used in the numerical model as the inputs (loading values) or for verification (the pictures).



Figure 1: Thermo-camera with CT specimen.

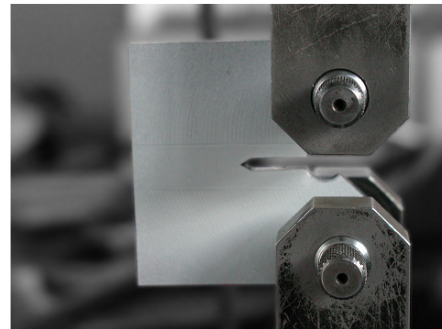


Figure 2: Fixed CT specimen.

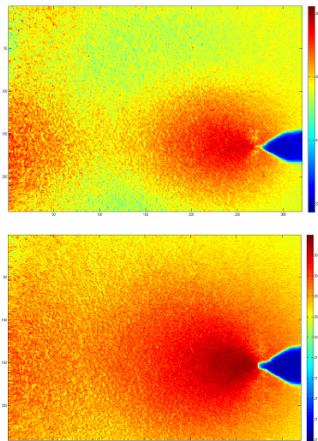


Figure 3: The development of heat by loading.

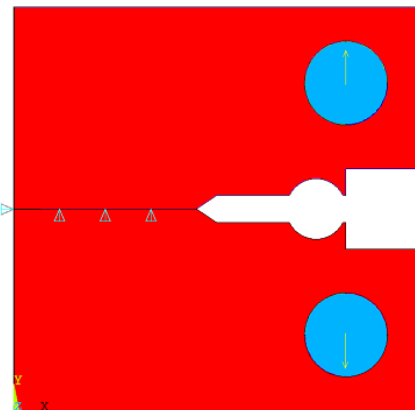


Figure 4: The boundary conditions of computation.

COMPUTATION

The FEA model was created (geometrically according to sizes of real specimen; material model was chosen as multilinear) with plane elements (PLANE82 – 8 nodes, 2 degrees of freedom in each node) up to plastic limit (the process of solving was canceled if this limit was overrun) in ANSYS. The forces were placed to the centroids of stems which were inserted to the two holes of CT specimen and from here was the loading transmitted by contact elements to the specimen. The axis of symmetry of the specimen was horizontally and vertically fixed as boundary condition. The direction of forces was chosen to simulate the real loading in vertical direction. The values of forces (or displacements) were chosen according to the table of measured forces from the experiment. The result – density of plastic work - of chosen loading images picture (Fig. 5). The similar picture but obtained from thermo-camera is in the picture (Fig. 6).

RESULTS

The result from computation (loading by forces) – the density of plastic work [$J \cdot m^{-3}$] – were compared with experimental data – the density of heat [$J \cdot m^{-3}$] – in ANSYS. The comparison is in previous page below. We can see the order of magnitude correspondence between the pictures in the wide area around the notch. But there is a



considerable difference in the small area around the tip. This difference is caused by simplified model which does not reflect the heat transport time dependent phenomena, which are directly recorded by the thermo-camera. The explication results from finite velocity of conduction heat which does not included in the model: The heat energy was not dispersed from the places with the biggest deformations to the places without plastic deformation. In general terms it can be said that the results were confirmed the original idea to use the thermographical record for observation of the development of plastic deformation.

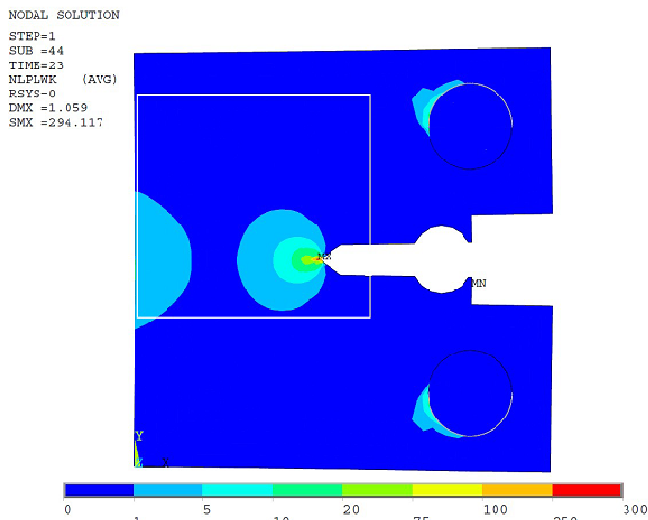


Figure 5: Density of plastic work from computation [$\text{MJ}\cdot\text{m}^{-3}$]

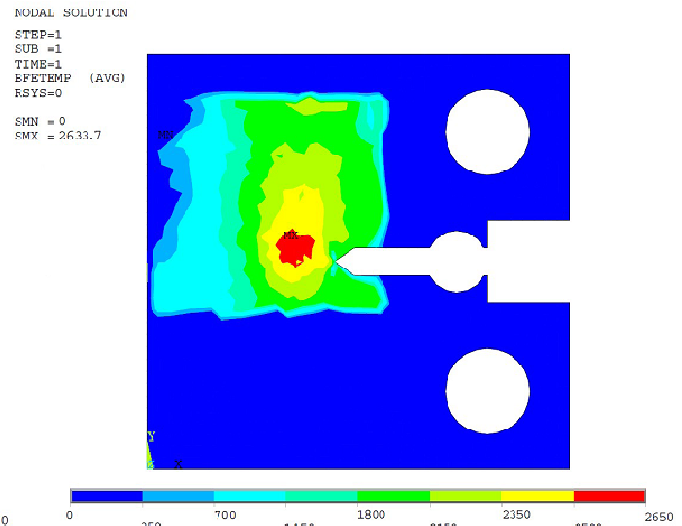


Figure 6: Density of heat from thermo-camera [$\text{KJ}\cdot\text{m}^{-3}$]

CONCLUSION

Comparing heat fields (during loading) in a vicinity of the notch (until initialization of the crack), we found out, the FEA model and the experiment yield similar results. The description and modeling of regions far away from the notch involves consideration of more complex physics (transition phenomena) therefore the differences of maximum values of heat between experiment and simplified model are bigger.

ACKNOWLEDGEMENT

The research has been supported by Grant Agency of the Czech Technical University in Prague (grant No. SGS10/218/OHK2/2T/16), by the Grant Agency of the Czech Republic (grant No. P105/10/2305), by research plan of the Ministry of Education, Youth and Sports MSM6840770043 and by research plan of the Academy of Sciences of the Czech Republic AV0Z20710524.

REFERENCES

- [1] X. W. Sun, et al. Quantitative plastic deformation analysis by means of thermoplastic effects, Beijing. Elsevier Science Publishers. (1993) 213.
- [2] K. S. Bhalla, A. T. Zehner, X. Han, Engineering Fracture Mechanics, (2003) 2439.
- [3] Springer Handbook of Experimental Solid Mechanics, ed. By W. N. Sharpe, (2008) 1098.
- [4] M. Žďárský, Comparison of Temperature Field of the Mechanically Loaded Structural Part Measured by a Thermo-camera to FEA Model, Bachelor Thesis, Czech Technical University in Prague, Faculty of Transportation Sciences, CZ (2009).