STATIC STRESS MODELLING IN RUBBER SAMPLE WITH AND WITHOUT ARTIFICIALLY PUNCTURED CRACK

Moskova Z.[1], Mokrysova M.[1], Kopal I[1]
[1]Faculty of industrial technologies, Institute of material technological research, I. Krasku 491/30, 020 01 Puchov, Slovak Republic, svecova@fpt.tnuni.sk

ABSTRACT
Fracture and fatigue of rubber-like materials is still an open problem. In engineering applications elastomers are frequently exposed to complex combination of cyclic loads. Hence, a better understanding of the material's resistance to crack initiation and propagation under fatigue loading is gaining increasing practical importance.
The present study focuses on the static loading modelling of rubber sample with and without artificially punctured crack [1]. For modelling needs were used Mooney-Rivlin constants [2] evaluated in Abaqus from experimental data. Both simulated results were compared with experimental results.
Keywords: fracture, rubber-like materials, static loading modeling

1. INTRODUCTION
Fatigue, crack initiation and crack propagation in rubber-like materials, generality, simulation of non-linear materials is world-wide solved the problem [3-6]. Thereby, that every rubber mixture is in a matter of fact unique in its composition, in industry it is required always newly define behavior hyperelastic material. Seeing that application of rubber-like materials in service is highly expanded and in the most cases the safety in service of products depend from their lifetime, it is very important to be able to predict their behavior at given loading. And more than, if in given product has found some breach, that might start up at service.
In the present days we are able to simulate fatigue, crack initiation and crack propagation in hyperelastic materials by various FEM software. We have used Abaqus 6.5 as FEM software with university license. It was shown, that finite element method (FEM) is most suitable advance for analyse and optimalisation of fatigue processes.
Evaluated results can predict stress intensity, strains as well as point of crack initiation or break material.
This advance makes it possible to reduce time-consuming experiments. Result is dependent from incoming data, as geometry samples, material properties, boundary conditions etc.

2. EXPERIMENTAL PART
Important incoming data for static simulaton were geometry dimensions of the sample Fig,1 material density $\rho_{E2} = 1{,}235.10^{-9}\, g.mm^{-3}$, MR constants C10= 0,2421 MPa, C01= 0,2237 MPa obtained in Abaqus for rubber mixture E2. Rubber mixture used in this experiment is commonly used in practice. It is tread rubber mixture, so it has big sense for this simulation.

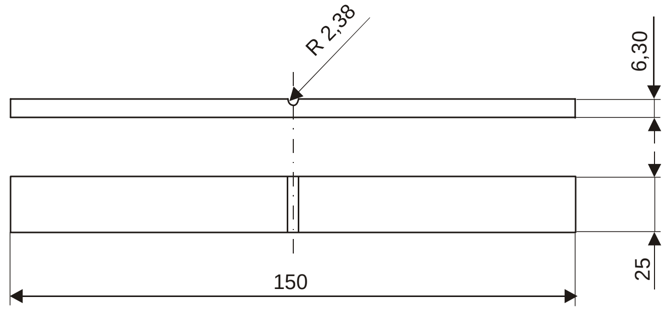

Figure 1. Geometry dimensions of the sample

Boundary condidtions were set with no degree of freedom on the left side, 1 degree of freedom on the right side, that simulate experiemnt. As we can see on this Figure 2, the sample is in maximal loading possition, that is equivalent to the experiment. Experiment was performed according to standard [7].

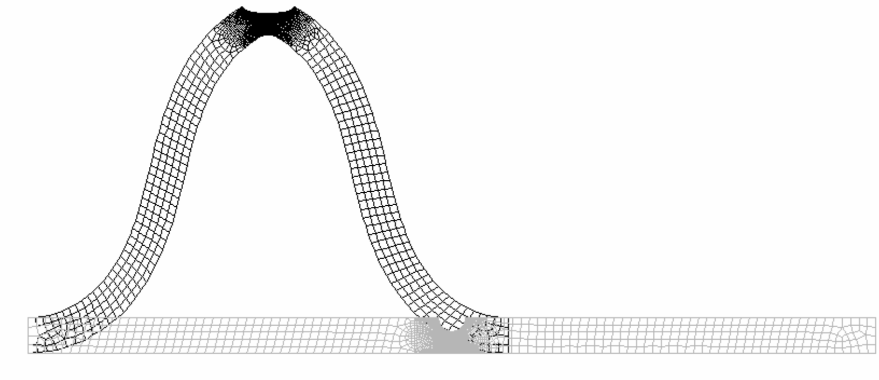

Figure 2: Sample in zero postition and in maximal position

On the next figure (Fig3) is presented simulated sample withou artificially punctured crak. As we can see, the place with the highest stress concentracion is equal to the place where during experiment after certain number of cycles starts crack initiation.

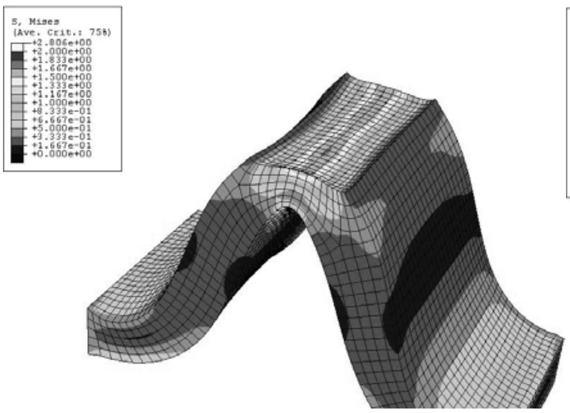

Figure 3: Stress concentration in sample without crack

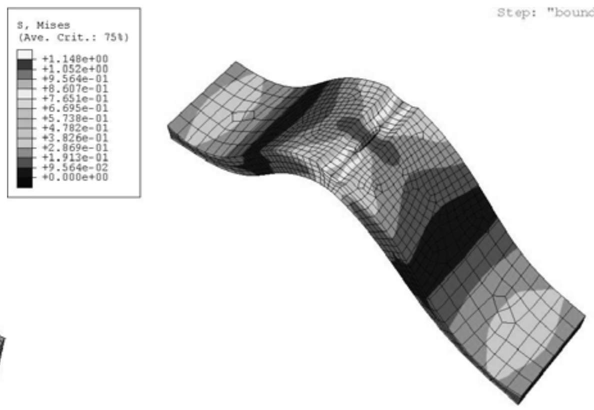

Figure 4: Stress concentration in sample with crack

Figure 4 presents stress concentration in sample with artificially punctured crack. In comparisom with experiment we can state, that in the same place starts crack growth.
The values of MR constants were estimated from experimental data in software MS Excel, Abaqus and Ansys. In the simulation we took into account MR constants acquired in Abaqus, which were shown as the most exact.

Straining and crack propagation simulation in given mixture was performed in FEM software Abaqus version 6.5 (university license). We have been concerning on static determination of given problems. From the simulation results for the sample without artificial made cut implies, that on the places with maximal stress concentration during trial we can observe crack creation.

3. CONCLUSION

In this work are presented results of static loading impact on crack initiation and crack propagation for rubber compound in Abaqus version 6.5 (university license). These results were in good agreement with experimental tests. Dynamic simulations are in-process.

4. REFERENCES

[1] Švecová, Z, Cúth, V, Slabeycius, J, Kopecký, M.: Evaluating of hyperelastic material behavior ,In: Sborník vědeckých prací Vysoké školy báňské - Technické univerzity Ostrava Řada hutnická. - Ostrava: Technická univerzita, 2005, ISBN 80-248-0860-9. - 2005, roč. XLVIII, č.1,s.245-250

[2] Cúth, v, Kopecký, m, Švecová, Z, The cause of dynamic stress on propagation of cracks in vulcanizates, In: Machine Dynamics Problems 2004, Vol. 28, No 3 [zborníky] : Proceedings of IX International Conference on Modelling in Mechatronics Kazimierz Dolny, Poland, September 2-4, 2004 - Part I. - ISSN 0239-7730. - Kazimierz Dolny, Poland, September 2-4, 2004. - s. 77-82

[3] Stevenson, A., A crack at fatigue resistance, European Rubber Journal, 1987, Dec. 24

[4] Brezik, R., Vaculikova, K. , Tensile fatigue of vulkanizates, Plasty a kaučuk, 24, 1987, č.5, s.141

[5] Gent. A . N ., Failure process in elastomere (Lomové procesy v elastoméroch), Maoplas International, 1989, č.11, pp. 105-106

[6] AGGARWAL,S. L.,HARGIS,I. G., LIVIGNI,R.A.,FABRIS, H. J., MARKER, L. F.: Advances in Elastomers and Rubber Elasticity, J. Lal and Mark J. E. ,Eds., Plenum Press, New York, 1986

[7] STN 621488 Metóda stanovenia odolnosti proti vzniku a rastu trhlín prelamovaním