Mariana KUFFOVÁ*

SIMULATION OF FATIGUE PROCESS

SUMMARY

In the paper are shown the results of fatigue crack propagation in the microstructure of material and the dependence da/dN – DKapl by finite – element software ADINA. These results are confronted with the results of fatigue tests. The aim was to elaborate on the crack initiation and growth in the microstructure containing a stress concentrator and redistribution of stress during propagation.

Keywords: *low-carbon steel, magnesium alloy, finite elements software ADINA, fatigue tests*

SYMULACJA ZJAWISKA ZMĘCZENIOWEGO

W artykule pokazano wyniki propagacji pęknięć zmęczeniowych w mikrostrukturze materiału oraz zależności da/dN – DKapl uzyskanych metodą elementów skończonych w pakiecie ADINA. Wyniki te porównano z rezultatami laboratoryjnych badań zmęczeniowych. Celem było wypracowanie pęknięcia inicjującego i jego rozwinięcie w mikrostrukturze z koncentratorem naprężeń oraz obserwowanie zmian rozkładu naprężeń w trakcie propagacji pęknięcia.

Słowa kluczowe: *stale niskowęglowe, stopy magnezu, pakiet ADINA, badania zmęczeniowe*

1. INTRODUCTION

ADINA is a FEM system suitable for solving large variety of problems. In this paper a capability of modeling of fracture and damage of material will be discussed. Reliability of modeling is in greater extent supplied by the accuracy in material properties, boundary conditions and last but not least in modeling of proper material behavior at crack tip including singularity if zero radius is presented. There are actually two ways for handling material failure or damage. The first one is linked to solution of intensity factor K and in comparison with its critical value. Irrespective of parameters of solid, geometry and distribution of external loading, if the deformation field is the same at the crack tip, the fracture behavior of the specific material will be equal (Charoenphan *et al.* 2004). Amplitude of the stress intensity factor K_a for the crack with the length 2a in the solid with the definite thickness and indefinite width which is loaded by the stress amplitude σ_a and it is defined (Puškár and Golovin 1985):

$$K_a = \sigma_a \cdot (\pi \cdot a)^{1/2} \tag{1}$$

In this way it is possible to predict if crack growth will occur or not, or in other words if the structure is safe or damage is imminent. The other way is to model the initiator or damage in the material as accurately as possible to estimate maximum stress. This however requires very fine discretisation and numerical problems could arise (Ural *et al.* 2003, Nguyen *et al.* 2001).

Linear and nonlinear fracture mechanics analysis can be performed with ADINA system including computation of conservation criteria (J-integral, energy release rate) in 2D and 3D finite element models. Two different numerical methods are available for the computation of the conservation criteria – the line contour method and the virtual crack extension method. The fracture mechanics allows performing an analysis with only one crack however. The crack line or surface can be located on the boundary or inside of the finite element model.

ADINA is thus fully capable of solving fracture mechanics problems in general with large amount of options in stack under various loading conditions or thermal conditions utilizing wide variety of material models. Also there is the ability to model rupture criteria and thus it is possible to model material damage caused by cavities or impurities and its progression under cyclical load or under creep conditions.

2. NUMERICAL RESULTS

The microstructure of the material represented by tightly packed grains was modeled by using the finite-element software ADINA. The geometry of each grain was modeled by Pro/Engineer software and was exported as a plain surface in IGES file to ADINA, where a 2D analysis was performed. For the analysis each surface representing individual grain of the microstructure was discretised using finite element mesh (Fig. 1). In this case quadratic elements were used, which means that unknown quantities were approximated by a polynomial of second order inside of each element. Due to large gradients in secondary fields, it is necessary to use very fine discretisation in the vicinity of the crack to achieve reasonable accuracy. The only factor limiting the fineness of the mesh is of course available hardware, but for extremely fine mesh a lower numerical stability can be expected. Each element has prescribed its own material

* Dept. of Mechanical Engineering, Academy of the Armed Forces, Liptovský Mikuláš, Slovak Republic; e-mail: kuffova@aoslm.sk

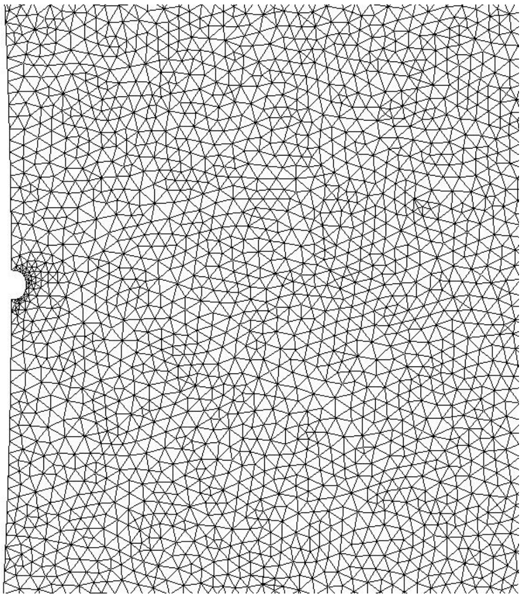

Fig. 1. Mesh in the surrounding of the stress concentrator

properties. Multi-linear plastic material model was used in the simulation. The properties of the material were obtained experimentally as a dependence of displacement of the specimen on the applied force. The values were then recalculated to a relation of strain ε and effective stress σ according to known dimensions of the specimen. Afterward the stress-strain curve was imported into ADINA and applied on each element. The analysis was performed by using large deformations and large displacements incorporated into the mathematical model. Each grain was considered as a standalone body and contact conditions between each pair of grains were implemented. In the microstructure model a stress concentrator was made (Fig. 2). On the boundaries was applied a cyclic load with amplitude 30 MPa and with frequency 25 Hz for a period of 4 seconds, that means that 100 cycles were applied altogether. These are standard loading conditions for examining the fatigue resistance of the material.

The load amplitude by which the propagation of micro-failure occurs is fully dependent on non-homogeneity of the material. The aim of the simulation was to observe the failure propagation in micro-volume. A redistribution of stress was influenced by the cyclical nature of the applied load. Even though the stress is applied in uniform manner on the structure, it does not act in the same way on each grain in the structure. Most of the stresses are cumulated in the region with certain non-homogeneity in the microstructure. In this case an artificially created cavity acts as a concentrator. The propagation of material damage was governed similarly to crack propagation, where a plastic zone (Fig. 3) is created in the vicinity of the crack tip in which a plastic strain is accumulated due to cyclic loading.

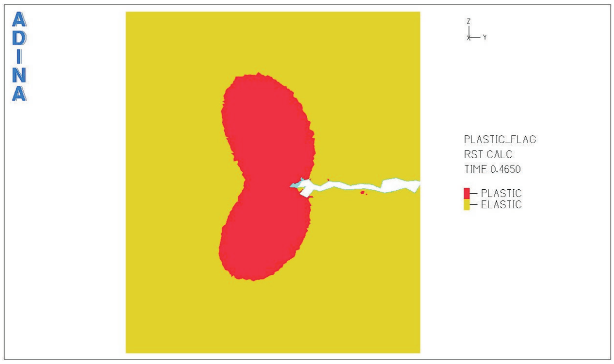

Fig. 3. Plastic zone at the crack tip

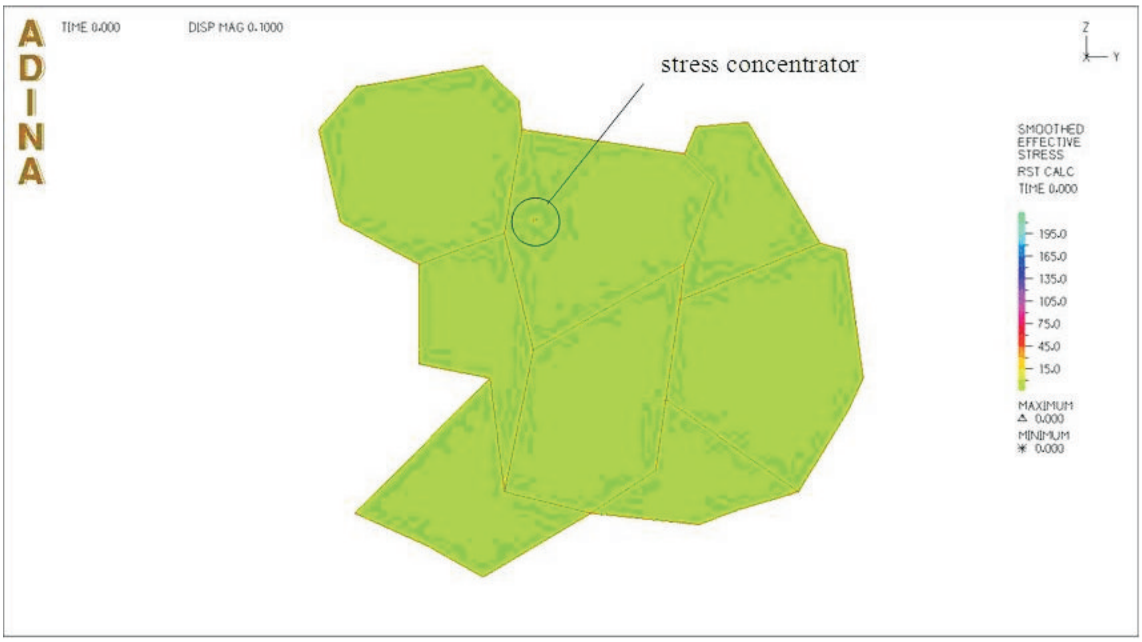

Fig. 2. Model of microstructure with primary stress concentrator before applied load

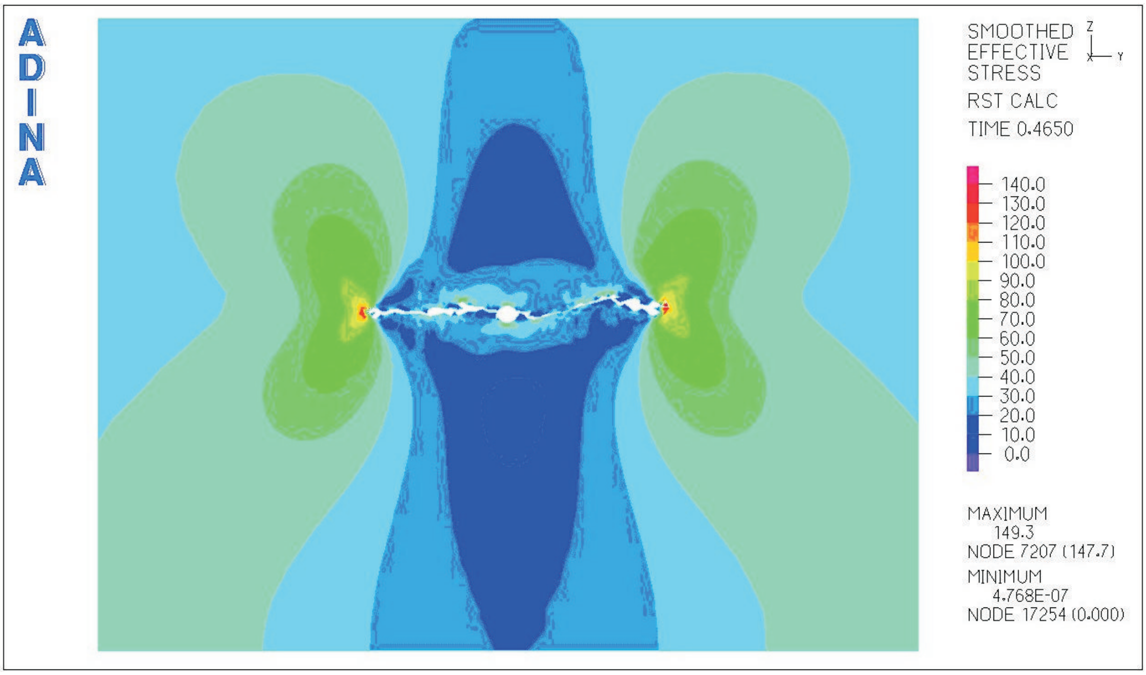

Fig. 4. Fatigue crack propagation trajectory and stress at the crack tip

The deformation process at the crack tip depends significantly on the mechanical properties of the material and on the environment in which the loading occurs. Limited plastic deformation, equal ΔK values and equal coefficients of asymmetry of the cycle do not guarantee the same magnitude and form of plastic zone ahead of the crack tip. Once the critical value of the plastic strain is reached in the vicinity of the crack tip, a material damage occurs and the crack propagates (Fig. 4).

The crack propagation occurs in the direction of maximal shear stress and its direction gradually changes into direction perpendicular to the direction of applied load. The orientation of main stresses inside of each grain is changing depending on the distribution of the grains, on their shape and on the spread of the damage. The crack acts upon this by changing the direction on the grain boundaries, but also due to the damage in the grain can do so even inside of the grain. A realistic explanation of the origin of grooves and the gradual crack propagation, based on the concept of repeated blunting and sharpening of the crack tip, was presented by Laird (Laird 1967). At application of the tension part of loading cycle, with a gradual increase of stress on the crack tip due to the high stress concentration, localized plastic deformation takes planes of maximum shear stress and leads to the opening of crack and to the blunting of its tip. The unloading brings the approach of both sides of the crack together, and the new surface that was formed under tension loading will not disappear entirely. A complete unloading and loading pressure cycle caused the sides of the crack, by an amount which corresponds to the distance between the grooves.

3. CONCLUSION

The FEM software ADINA enables us to observe growth of fatigue damage and crack propagation in a model as well as observe the distribution of stress fields. It enables us to elaborate on stress state inside of the material, based on which it is possible to predict the direction of crack propagation.

The problems of crack propagation in the micro-scale are considerably demanding on the model accuracy. Many of the effects are manifested on the atom-scale and using the continuum mechanics it is possible to study these effects only upon certain optimal conditions or according to the output of experimental results (Kuffová 2006, Kuffová and Štiavnický 2008) or from other special methods.

References

Charoenphan D., Plesha M.E, Bank L.C. 2004, *Simulation of crack growth in composite material shell structures.* Int. J. Numer. Meth. Engng., 60, pp. 2399–2417.

Kuffová M. 2006, *Fatigue resistance of magnesium alloy AZ 91D at the high- frequency cyclic loading.* In Procc. Intern. conf. “12th AMME”. Cairo, Egypt.

Kuffová M., Štiavnický M. 2008, *Analysis of the fatigue crack growth using ADINA.* [in:] Materials Engineering, 15, 1, p. 32, ISSN 1335-0803.

Laird C. 1967, ASTM Spec. Tech. Pub., 415, 131.

Nguyen O., Repetto E.A., Ortiz M., Radovitzky R.A. 2001, *A cohesive model of fatigue crack growth.* Int. Journal of Fracture 110, pp. 351–369.

Puškár A., Golovin S.A. 1985, *Fatigue in materials:Cumulative damage processes.* 1st ed., VEDA Bratislava.

Ural A., Wawrzynek P.A., Ingraffea A.R. 2003, S*imulating Fatigue Crack Growth in Spiral Bevel Pinion*. NASA Glenn Research Center: Cleveland OH.