Identification of Ductile Damage Parameters in the Abaqus

J. Ruzicka, M. Spaniel, A. Prantl, J. Dzugan, J. Kuzelka, M. Moravec

Abstract— **This paper deals with a ductile damage simulation using a finite element method. Phenomenological models of ductile damage are presented. The goal of this work is design a useful methodology for calibration of ductile damage parameters. Numerical analysis was realized by the Abaqus program.**

***Index Terms* — ductile damage, plasticity, identification, abaqus, fracture locus**

I. INTRODUCTION

THE need of simplifying prediction of ductile fracture is obvious for industrial applications. It is recognized that components of stress and strain tensors are basic output of structural analyses, such as finite element method, using commercial software. A goal for engineers is to develop a suitable fracture criterion in terms of the stress, strain, triaxiality etc. with certain material dependent parameters for predicting material failure and fracture with an acceptable degree of accuracy.

II. DUCTILE DAMAGE MODELS

A fracture is the local separation of an object or material into more pieces under the action of stress. Depending on conditions the fracture process can be brittle or ductile. Brittle fracture is characterized by rapid crack propagation without significant plastic deformation and therefore spending low energy that releases from strain energy accumulated in the body. In case of ductile fracture, extensive plastic deformation (necking) takes place before fracture. In this case there is slow propagation with relatively large dissipation of released strain energy via plastic deformation and crack faces separation.

Two basic mechanisms of ductile fracture are usually distinguished. The first mechanism is initiation, growth and coalescence of voids. It is typical for dominant tensile loading. The second one is shear mechanism which is typical for dominant shear loading.

From the point of view of a material microstructure, there are two ductile damage model categories. Void nucleation and growth models (take account of damage micromechanism but resulting into phenomenological like simplifications as well known Gurson Twegaard model) and empirical models (directly based on phenomenology). Both micromechanical and purely empirical approaches utilize cumulative state parameter-damage. Damage accumulates on the basis of plastic strain, and when approaching its critical value it indicates local material failure. Practical applications show, that more important than micromechanical or purely empirical base of ductile damage material models is the number of parameters to calibrate and calibration experiments costs. Successful models providing us with better correspondence require more experiments and are more expensive.

Another, more senseful, classification of ductile damage models is based on dependence of actual plastic response on damage. As mentioned above, damage accumulation is controlled by plastic strain. If plastic response does not depend on damage the model is classified as uncoupled. In other case it is referred as coupled ductile damage model or damage plasticity model. Damage plasticity models should be generally more realistic, but their calibration is usually more complex. The main advantage of uncoupled ductile damage models is a possibility to calibrate plastic response and damage criteria separately.

Models of ductile failure implemented in the FE software Abaqus can be classified as purely empirical models (except of Gurson model presented in Abaqus as specific plasticity.). All of them use simple criteria assigned to specific kinds of materials, specific loading conditions, etc. The process of calibration of these models consists of four phases (see Fig. 1)

- elastic constants of the model calibration (*a*-*b*)
- calibration of plastic response curve (*b*-*d′*)
- calibration of damage initiation criteria parameters (*c*)
- calibration of damage development parameters (*c*-*d*)

J. Ruzicka is with the Department of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technicka 4, 166 07 Prague 6, Czech Republic,
e-mail: ruzicka.jan@seznam.cz
M. Spaniel is with the Department of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technicka 4, 166 07 Prague 6, Czech Republic,
e-mail: miroslav.spaniel@fs.cvut.cz
A. Prantl is with NPP Services, Stress Analysis Calculations, Škoda JS a.s. , Orlík 266 , 316 00 Plzeň, Czech Republic.
e-mail: antonin.prantl@skoda-js.cz
J. Dzugan is with the Department of Thermophysical Measurements and Mechanical Testing, COMTES FHT Inc., Dobrany, 334 41, Czech Republic,
e-mail: jan.dzugan@comtesfht.cz
J. Kuzelka is with the Department of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technicka 4, 166 07 Prague 6, Czech Republic,
e-mail: jirikuzelka@seznam.cz
M. Moravec is with the Department of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technicka 4, 166 07 Prague 6, Czech Republic,
e-mail: milos.moravec@fs.cvut.cz

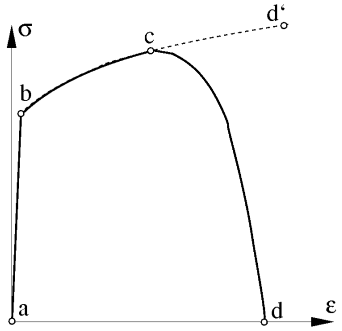

Fig. 1 - Tensile curve of material with damage

Development of damage in these models represents gradual loss of material strength in local material point. It starts just with damage initiation (damage approaches critical value) and has no effect on the previous pure plastic behavior. This allows us to calibrate the purely plastic response and subsequent damage-plastic response of material separately.

III. SPECIMENS

Ductile damage depends on stress state expressed by three principal stresses in principal coordinate system. Hydrostatic stress, Von Mises stress and Lode angle are sometimes used instead. Standard incremental metal plasticity models are based on Von Mises stress (in case of models with kinematic hardening the situation is more complex). For calibrating general ductile damage material model, testing of number different specimens is necessary. Therefore the identification of ductile damage parameters is relatively expensive. Finite element models of specimens applied for calibration are shown in Fig. 2. Four notched tensile bar specimens (notch radiuses R=∞, R=1mm, R=2mm, R=4mm) were employed to calibrate plasticity using axisymmetric simulation models, that save computational costs. Four compression bar specimens with different notch radius (R=∞, R=1mm, R=2mm, R=4mm), unnotched tensile flat specimen and CT specimen without initial crack are simulated as well. Very general is a special so called "butterfly" specimen originally developed in MIT. This specimen change stress state in dependence on its loading orientation (0°, 30°, 45°, 70°, 80°, 90°, 100°, 110°, 120°). A special loading apparatus was constructed for the butterfly specimen testing. In all models, the elements size is 0.2 mm in the gauge section where necking and subsequent fracture occurs. One end of the models is defined with fixed boundary conditions and a tensile (compression) displacement is prescribed at the other end.

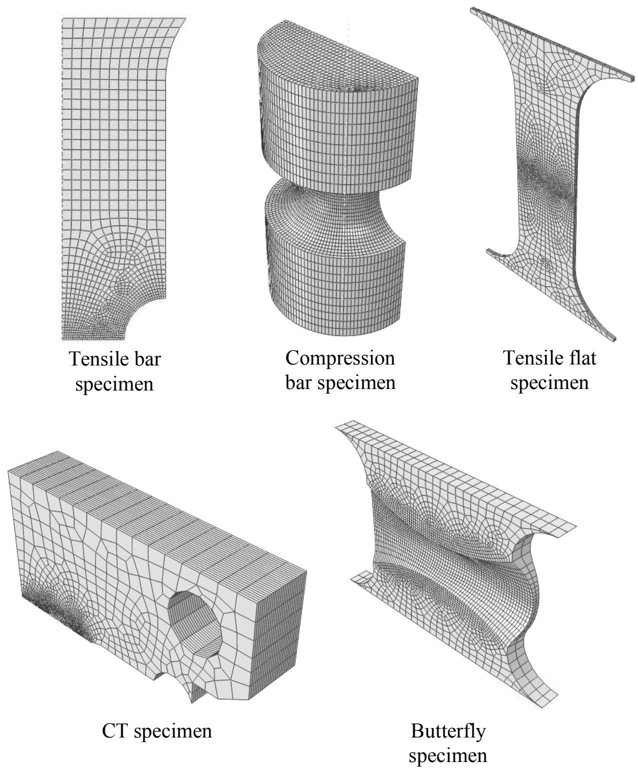

Fig. 2 - Specimens applied for ductile damage calibration

IV. CALIBRATION OF PLASTIC RESPONSE CURVE

The Mises yield rule assumes that yielding of the metal is independent on equivalent pressure stress as well as on Lode angle. This observation is confirmed experimentally for most metals under monotonic loading. Material hardening is controlled via yield stress dependence on equivalent plastic strain accumulated during loading process. This plastic response curve is calibrated using comparison of both experimentally and computationally obtained load-displacement responses for various kinds of specimens. Calibration is based on searching parameters of the plastic response curve (generally any parameters of plasticity model) minimizing target functional defined as the area bounded by the experimental loading curve and finite element simulation result expressing the integral difference between them (Fig. 3). The area is zero for the best solutions. Simplex algorithm was used as optimization method in this study. It represents local optimization method only and should start from the estimation of the solution as good as possible.

Power law is usually suitable approximation of plastic response curve . Three parameters *A*, *B*, *n* need to be determined.

$$\sigma_Y^{True} = A + B(\varepsilon_{\ln}^{pl})^n \tag{1}$$

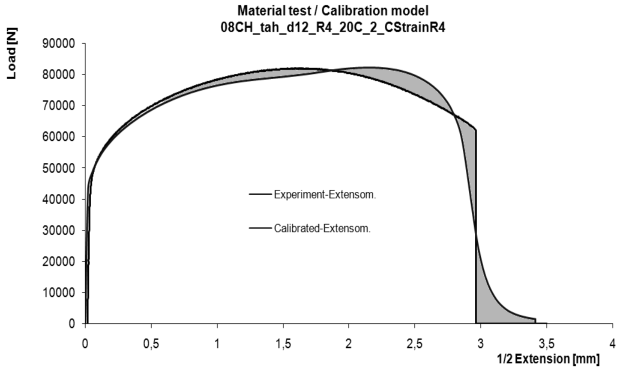

Fig. 3 - The area bounded by the experimental loading curve and FE simulation result

Another possibility is use a nominal stress/strain dependence obtained from the tensile test. At first is nominal stress/strain curve transformed to stress/lagarithmic strain curve. The next step is power law extrapolation to higher plastic deformation and adopting some useful parametrization.

$$\sigma_{|Y,i}^{True} = A_0 + A_1 \sigma_{Y,i}^{*True} \quad , \quad \varepsilon_{\ln,i}^{pl} = B\varepsilon_{\ln,i}^{*pl} \tag{2}$$

The plasticity calibration script for parametrical optimization was written in Python. The script do following:

- Finite element models enabling to simulate all experiments with plasticity parameters as variable input creation and the parameters estimation.
- The first iteration including simulations of models with plasticity parameters estimation.
- An experimental data and a simulation results comparison.
- The deviation area calculation.
- Change the plastic response curve via call of simplex optimization procedure.
- Repetition until the deviation area is minimal.

V. IDENTIFICATION OF DAMAGE INITIATION CRITERIA PARAMETERS

The Abaqus ductile fracture material model is based on phenomenological criterion for predicting the onset of damage due to nucleation, growth, and coalescence of voids. The criterion for fracture initiation is met when the condition is satisfied.

$$\varpi_D = \int \frac{d\varepsilon^{pl}}{\varepsilon_D^{pl}(\eta)} = 1 \cdot \tag{3}$$

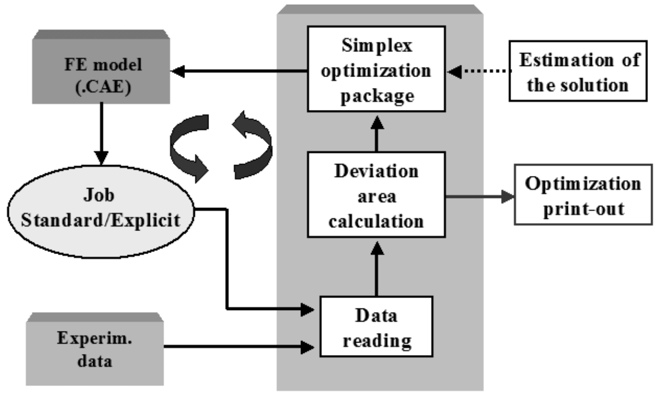

Fig. 4 - The plasticity optimization script algorithm

Damage parameter ϖ is a state variable increasing monotonically with plastic deformation. It is zero for undamaged material, and equal one for totally damaged material. The model assumes that the equivalent plastic strain at the onset of fracture $\varepsilon_D^{pl}(\eta)$ is a function of triaxiality η.

$$\eta = \frac{-p}{q} \tag{4}$$

where p is the hydrostatic stress and q is the Mises equivalent stress.

The first step of ductile damage calibration is identification of critical extension of relevant specimen. At this extension strong step decrease of experimental loading curve can be observed that indicates onset of fracture process. Corresponding finite element analyze gives the evolution of stresses and strains of the critical points enabling evaluate criterion (3) in dependence $\varepsilon_D^{pl}(\eta)$. By combining the tests and the numerical simulations, one would be able to construct empirical fracture envelopes of material. Corresponding finite element analysis was conducted and the evolution of suitable factors of the critical points was determined. Factors having significant influence on ductile damage are mainly equivalent plastic strain, stress triaxiality, Lode parameter, strain rate and temperature. The ductile fracture locus was formulated in the space of effective plastic strain to fracture and the stress triaxiality. There appeared expected strong dependency of material ductility on stress triaxiality. The equivalent plastic strain function (fracture locus) $\varepsilon_D^{pl}(\eta)$ was described by Johnson-Cook model

$$\varepsilon_D^{pl} = D_1 + D_2 e^{D_3 \eta} \tag{5}$$

where coefficients D_1, D_2 and D_3 need to be determined.

The integration of damage parameter ϖ was evaluated numerically for all estimated sets of coefficients D_i .The calibration script was run in Matlab without iterative parallel finite element simulation. This is a doubtless advantage of uncoupled empirical damage initiation criteria. The objective function F was given by a following formula

$$F = \sum_i |1-\omega_i| \quad , \quad \omega_i = \max\left[\int_0^{\varepsilon_i^{pl*}} \frac{d\varepsilon_{i,j}^{pl}}{\varepsilon_D^{pl}\left(\eta_{i,j}\right)}\right]_j \tag{6}$$

where i represents the index of specimen and j represents the index of element in damaged area. The resulting fracture locus is shown in Fig. 5. The damage parameter should take the value ω=1 in the critical prolongation for all specimens. The error of damage model for single specimens after calibration is shown in Fig. 6.

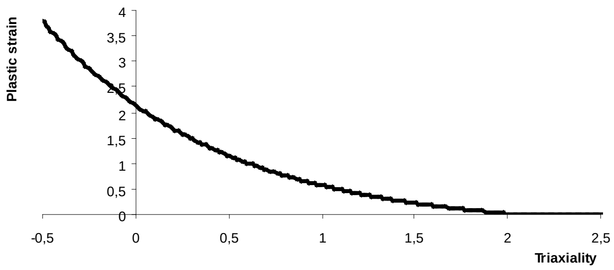

Fig. 5 - The resulting fracture locus

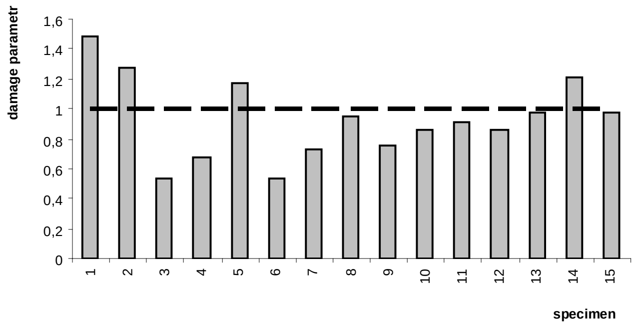

Fig. 6 - The ductile damage parameter in the critical prolongation for single specimens. 1-4 notched bar specimens, 5 CT specimens, 6-15 butterfly specimens.

Comparison of experimental loading curves and FE simulation results after calibration process for some specimens are shown in Fig. 7-9.

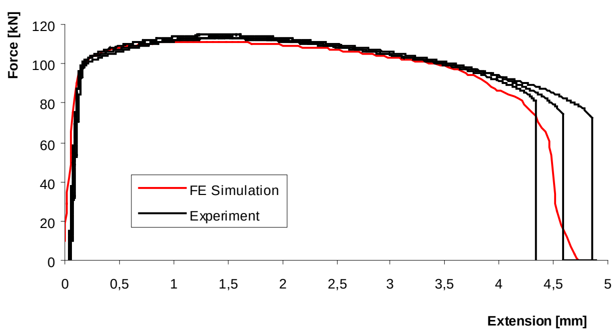

Fig. 7 - The tensile notched bar specimen R1

ACNOWLEDGEMENT: This work was supported by Ministry of Industry and Trade of The Czech Republic within the project FR-TI2/279.

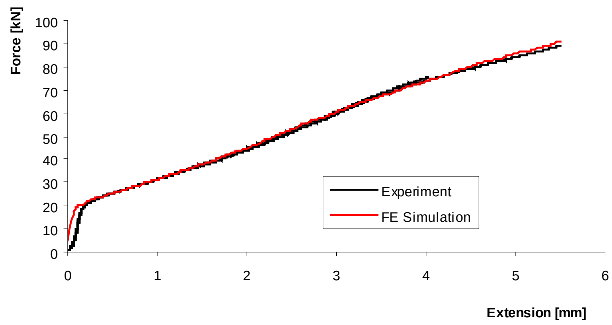

Fig. 8 - The compression notched bar specimen R4

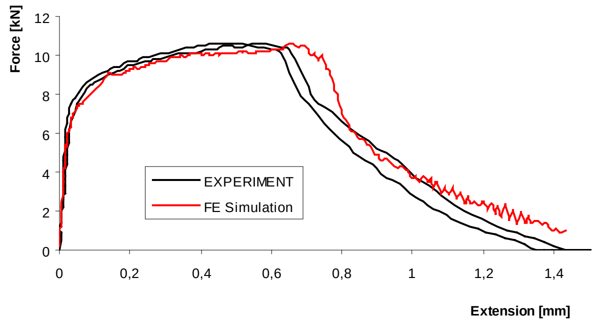

Fig. 9 - The butterfly specimen 30°

VI. Conclusion

A useful methodology for material plasticity calibration described in this study was developed. It is based on searching the parameters of plastic response on plasticity model parameters minimizing the area bounded by the experimental loading curve and FEM simulation result. The optimization is based on the simplex algorithm. A methodology for calibration of empirical ductile damage initiation criteria was tested. Cummulation of damage in this model has no effect on simultaneously developing plastic straining. The ductile damage calibration script was written in Matlab without simultaneous calls of finite element analyses. The Calibrated models of plastic response curve and ductile damage criteria are in rather good agreement with experimental results for tested materials. Our future work is focused on Lode parameter implementation into Abaqus ductile damage criteria.

VII. References

[1] M. Spaniel, J. Ruzicka, M. Moravec, J. Kuzelka, *Identifikace fenomenologickych modelu tvarneho porusovani v programu Abaqus. VKMKP 201Plzeň.* ISBN 978-80-261-0059-1.

[2] H. Mae, X.Teng, Y. Bai, T.Wierzbicki, *Ductile fracture locus of AC4CH-T6 cast aluminium alloy,* Archives of Computational Materials Science and Surface Engineering, 2009

[3] *Abaqus Online Documentation*: Version 6.9

[4] Y. Ling, *Uniaxial True Stress-Strain after Necking*, AMP Journal of Technology Vol. 5 June, 1996

[5] L. Xue, T. Wierzbicki, Ductile Fracture Charakterization of Aluminium Alloy 2024-T351 Using Damage Plasticity Theory, International Journal of Applied Mechanics. Vol. 1, No. 2, 2009

[6] J Svec, *Letajici circus*, Python tutorial, 2003