Identification of the Parameters of a Damage Constitutive Law for Steel at Elevated Temperature

S. Castagne*, A.M. Habraken

Department M&S, University of Liège – Chemin des Chevreuils 1, 4000 Liège, Belgium
URL: www.ulg.ac.be/matstruct *e-mail: s.castagne@qub.ac.uk; anne.habraken@ulg.ac.be*

ABSTRACT: This paper presents the methodology and the results of the calibration of an interface damage law using experimental damage analyses and associated finite element simulations. The experimental damage analyses consist in acoustic tests realised on steel samples in order to determine the apparition of the first crack during compression. Finite elements simulations of these experiments allow the determination of the damage parameters using a reverse method.

Key words: interface damage law, parameters identification, steel, elevated temperature, acoustic tests, finite element analysis

1 INTRODUCTION

A finite element model for the analysis of transverse cracks initiation and propagation during the cooling of steel continuous casting has been developed [1]. In this research, the continuous casting process and the damage evolution are analysed at the grain scale, referred to as the mesoscopic scale. Due to the huge amount of computational resources that would be required, it is not conceivable to model the whole steel slab at the mesoscopic scale. Nevertheless, the critical areas for crack initiation are well known and therefore, specific studied zones can be defined for the finite element calculations.

In the studied temperature range (from 1100 °C to 700 °C), the cracks have been shown to be intergranular and the principal damage mechanisms are grain boundary sliding and creep controlled by diffusion. To model these effects, a mesoscopic cell, which represents the microstructure of the material, has been defined [2]. Two types of finite elements are used: two-dimensional (2D) finite elements for the grains and one-dimensional (1D) interface elements for their boundaries. An elasto-visco-plastic law without damage is used inside the grains and a law with damage has been developed for the grain boundaries. The mesoscopic model requires information from the macroscopic scale, which are available through a macroscopic finite element analysis of the continuous process previously realised [3].

A detailed description of the mesoscopic model can be found in [1] where the constitutive equations have been summarized together with a description of the parameters. The identification of the parameters of the elasto-visco-plastic law used in the grains and of the creep parameters of the damage law is presented in [2]. The present paper focuses on the experiments performed in order to identify the remaining parameters of the damage interface law such as the nucleation parameters or the initial voids characteristics. The finite elements simulations associated with the identification process are also presented.

2 EXPERIMENTAL ANALYSES

The damage analysis consists in acoustic tests realised in order to determine the apparition of the first crack during the compression of steel samples. At the origin, these tests were developed at IBF

* Currently at the Queen's University Belfast UK

(RWTH-Aachen) to predict the formality of steel at low temperature and were then adapted to the condition prevalent during hot forming [4].

If the internal stresses in the material are exceeding locally a critical threshold during forming, a sudden change appears (the initiation of a micro-crack), which allows the material to go back to an equilibrium state with a lower potential energy. The potential energy emitted is dissipated in the form of elastic waves that can be detected in the surrounding area as sound pulses. Piezoelectric sensors are used to record the sound signals. These signals, that have a very low intensity, are pre-amplified, filtered to separate the sound associated with the damage process from the interferences and amplified again before being introduced in the data acquisition system.

In order to reproduce the continuous casting conditions and the microstructure of the steel during this process, the samples were first heated up to 1375°C in an external radiation furnace and maintained at this temperature for ten minutes before cooling down to test temperature. To protect the sample material against surface oxidation, the furnace was rinsed with argon inert gas. Afterwards the samples were manually placed on the compression machine. The surrounding furnace was heated up to test temperature within an argon inert gas atmosphere. Finally the samples have been compressed up to crack initiation with a constant strain rate while upcoming acoustic emission events caused by material failure have been recorded.

Sets of acoustic tests have been done with different sample geometries (two cylindrical (flat and slim) and two non-cylindrical (flange and concave) shapes) to realize different stress-strain histories at the critical point of the samples (see figure 1). The critical point is the point where the crack is supposed to appear due to mechanical loadings, i.e. where the maximal principal stress will reach its maximal value (on the outer edge at mid-height for the flat, slim and concave samples and at the intersection of the outer edge of the cylindrical part with the ring for the flange sample). This location has been verified experimentally for various samples at the time of the development of the technique at IBF.

Three temperatures (800 °C, 900 °C, 1000 °C) and two strain rates ($1\ 10^{-3}\ s^{-1}$, $5\ 10^{-4}\ s^{-1}$) have been tested with at least three samples for each combination and for each geometry to ensure statistically relevant results.

3 FINITE ELEMENT ANALYSES

The parameters are identified using an inverse method that requires two major steps. First the loadings to be applied to the mesoscopic cell have to be determined by macroscopic simulations of the acoustic tests and then these data have to be applied to the mesoscopic cell to solve the inverse problem.

3.1 Macroscopic modelling of the acoustic tests

The finite element simulations of the acoustic tests give the formability curves and also the stress, strain and temperature fields in the whole sample and in particular in the region where the crack is supposed to appear. The material law used for these simulations is the elasto-visco-plastic law that is also used to model the grains of the mesoscopic model [2]. The temperature is fixed at the corresponding test temperature for each simulation. The compression load is modelled using a tool whose displacement produces a vertical logarithmic strain in the sample corresponding to the required constant strain rate. A contact law with friction is used between the sample and tool.

A sensitivity analysis has shown that the results where strongly dependent on the friction coefficient between the tool and the sample. Therefore, further experiments performed at IBF using the same procedure as for the acoustic tests have been necessary to determine the actual friction coefficient. The chosen tests were ring tests at 800 °C and with a constant displacement rate of the tool of $8\ 10^{-3}$ mm/s. The friction coefficient found is 0.2.

For illustration purposes, figure 1 shows the evolution of the specific maximal principal stress versus the effective strain at the critical point for a temperature T = 900 °C and a strain rate $\dot{\varepsilon} = 5\ 10^{-4}\ s^{-1}$, these results have been obtained by finite element simulations of the compression tests for each sample geometry. The crosses on the curves in figure 1 indicate the moment of the first crack initiation for each tested sample. Similar figures can be drawn for the other temperatures and strain rates combinations.

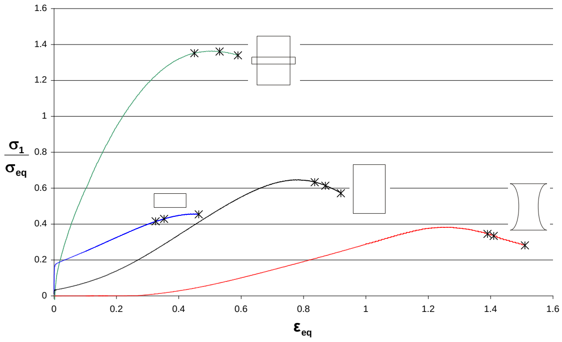

Fig. 1. Macroscopic simulations (T = 900°C, $\dot{\varepsilon}$ = 5 10^{-4} s^{-1}).

Different element sizes have been tested in the critical zone and the stress-strain fields compared in order to analyse the mesh dependence of the results. For the slim, flat and concave samples, a threshold has been determined for the size of the elements under which it is not necessary to go as the results of the simulations in term of stresses and strains distributions remain similar. For the flange sample, the stress concentration is so high in the critical zone that it was not possible to find a mesh with a reasonable number of elements for which the results converged with a sufficiently high accuracy . Therefore, it has been decided to keep only the three first samples, which give reliable results, for the identification stage.

3.2 Mesoscopic simulation of the acoustic tests

3.2.a Transfer of data

To allow the transfer of data from the macroscopic model, the mesoscopic cell is surrounded by a transition zone (figure 2). From a mechanical point of view, the mesoscopic cell is a slice in generalised plane strain state. This formulation allows at the same time stresses and strains in the out-of-plane direction [3]. The history of forces and/or displacements known thanks to the macroscopic simulations is imposed on each node of the boundary of the transition zone. As an elasto-visco-plastic constitutive law is used, it is important to follow the whole process.

For the acoustic tests simulations, the temperature is fixed and is constant in the whole sample. Nevertheless, if the method is used for simulations with variable temperatures, the temperature of each node of the mesoscopic cell has to be fixed at each time step according to the results of the parent macroscopic simulation. No thermal exchange is computed at this scale as the whole thermal problem has already been treated by the macroscopic model.

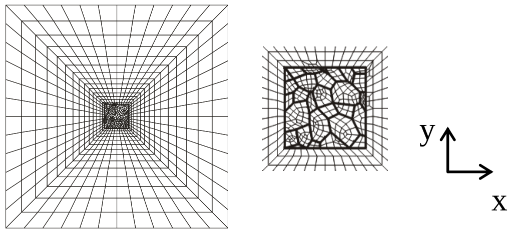

Fig. 2. Left: mesoscopic cell surrounded by a transition zone (50 mm x 50 mm). Right: zoom on the grains (5 mm x 5 mm).

During the data transfer, the objective is to reproduce in the mesoscopic cell the stress-strain field recorded at the critical point of the parent macroscopic simulation. This field is uniform in the cell with small variations in the grains zone due the grains pattern and to the damage initiating at the grain boundaries. The loading can be applied using forces or displacements or a combination of the two. The stress-strain fields are three-dimensional in the macroscopic simulations, with compression in the axial direction and tension in both radial and circumferential directions. At the critical point, the radial stress vanishes as the edge of the sample is reached but the strain field remains three-dimensional. The compression stress is reproduced in the mesoscopic cell in the direction normal to the plane (*z* direction) using the properties of the generalised plane strain state. The tensile stress, which is responsible for the apparition of the crack, is applied in the *y* direction of the mesoscopic cell. The stress in the *x* direction and the shear stresses have to be equal to zero.

For the first trial, displacements were imposed in the three directions as the simulations usually have a better convergence when the loadings are imposed through displacements only. Although the correct equivalent stress was applied to the mesoscopic cell, it appeared that the stress distribution $\left(\sigma_x, \sigma_y, \sigma_z\right)$ was not correct. This is due to the formulation of the elasto-visco-plastic law, which is given in terms of equivalent stress and strain. Indeed, different stresses distributions can correspond to the same equivalent stress.

It has been found that the correct stress and strain tensors can be reproduced if a displacement is imposed in the *x* direction and forces in the *y* and *z* directions. This has been verified by plotting these fields for the critical point of the macroscopic simulations and for the elements of the transition zone of the mesoscopic cell. For this analysis, damage parameters from the literature have been used. Nevertheless, looking globally at the mesoscopic cell, it has been shown that the results of the data transfer where not dependent of the damage parameters.

3.2.b Sensitivity analysis

In figure 3, which represents a classical damage evolution, three phases can be visualised. First the damage begins to increase very slowly due the diffusion of voids and to the growth of the voids that are already present (A-B). Then the nucleation threshold is reached, new voids are created and the damage increases more rapidly (B-C). Finally, the saturation state is reached, no more cavities can be created and the growth of the damage slows down until final rupture (C-D).

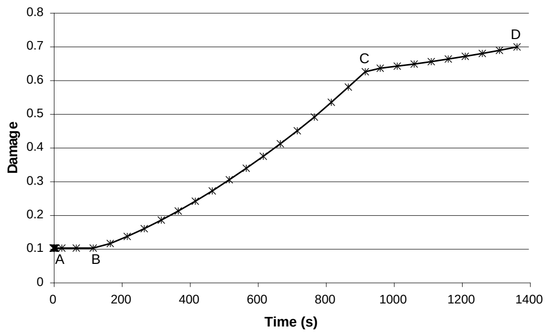

Fig. 3. Typical damage evolution.

The damage parameter d is the ratio between the voids diameter $2a$ and the average spacing between voids $2b$. The initial damage value depends on the initial values a_0 and b_0. The initial density of cavities for nucleation by unit length N_I and the nucleation activity parameter F_n determine the position of point B. The slope of the curve between B and C partially depends on the value of F_n. The maximal density of cavities N_{max} strongly influences the position of point C. These five parameters (a_0, b_0, N_I, F_n and N_{max}) and the damage threshold for crack appearance d_{lim} have not been fixed yet and can be used for the calibration of the model. The diffusion and creep parameters that also influence the damage curve have already been fixed [2] in a consistent way with macroscopic elasto-viscoplastic law and literature data.

3.2.c Results of the identification

First, the mesoscopic cell has been loaded using the results of the acoustic simulations at T = 900°C, $\dot{\varepsilon} = 5\ 10^{-4}\ s^{-1}$ for the three samples (slim, flat and concave) and a range of damage parameters have been tested to reproduce the experimental results, i.e. so that the first crack appear at the right time for each geometry. Then, the set of parameter that fitted the experiments has been tested for the other combinations of temperature and strain rate and it has been checked that the cracks appeared in the right order for each combination. The final parameters determined by the acoustic analysis are the following: $a_0 = 2.75\ 10^{-3}$ mm, $b_0 = 2.75\ 10^{-2}$ mm, $N_I = 380\ mm^{-1}$, $F_n = 1.5\ 10^5\ mm^{-1}$, $N_{max} = 40\ N_I$ and $d_{lim} = 0.7$. Even if the numerical model allows a dependence of these parameters on the temperature, a unique set of parameters could be found. The temperature dependence is already modelled through the diffusion and creep parameters.

4 CONCLUSIONS

A inverse method for the identification of damage parameters at elevated temperature has been proposed and used for the identification of the damage model for a specific steel. These results are now available and will be used to analyse the apparition of cracks in continuous casting.

ACKNOWLEDGEMENTS

As Senior Research Associate of the National Fund for Scientific Research, A.M. Habraken thanks this Belgian research fund for its support. The industrial partner ARCELOR is also acknowledged as well as IBF (RWTH-Aachen) where part of the experimental study was performed.

REFERENCES

1. S. Castagne, M. Remy and A.M. Habraken, Development of a Microscopic Cell Modeling the Damage Process in Steel at Elevated Temperature, *Key Engineering Materials* 233 (2003) 145-150.
2. S. Castagne, F Pascon, J. Lecomte-Beckers and A.M. Habraken, Mesoscopic Model of Intergranular Cracks in Low Carbon Steel During Cooling of Continuous Casting, In: *Proc. 2nd International Conference on Thermomechanical Processing of Steel*, ed. M. Lamberigts, Stahl Eisen (2004) 233-240.
3. F. Pascon and A.M. Habraken, 2D½ Thermo-Mechanical Model of Continuous Steel Casting Using F.E.M.”, In: *Proc. 6th Esaform Conference on Metal Forming*, ed. V. Bucaro, Nuova Ipsa (2003) 759-762.
4. R. Kopp and G. Bernrath, The Determination of Formability for Cold and Hot Forming Conditions, *Steel Research* 70 (1999) 147-153.