Simulation of Hot-Forging Processes with a Temperature−Dependent Viscoplasticity Model

J. Siring[1(✉)], M. Schlayer[2], H. Wester[1], T. Seifert[2], D. Rosenbusch[1], and B.-A. Behrens[1]

[1] Institute of Forming Technology and Machines, Hemnitz, Germany
siring@ifum.uni-hannover.de
[2] University of Applied Sciences Offenburg, Offenburg, Germany

Abstract. Hot forging dies are subjected to high cyclic thermo-mechanical loads. In critical areas, the occurring stresses can exceed the material's yield limit. Additionally, loading at high temperatures leads to thermal softening of the used martensitic materials. These effects can result in an early crack initiation and unexpected failure of the dies, usually described as thermo-mechanical fatigue (TMF). In previous works, a temperature-dependent cyclic plasticity model for the martensitic hot forging tool steel 1.2367 (X38CrMoV5-3) was developed and implemented in the finite element (FE)-software Abaqus. However, in the forging industry, application-specific software is usually used to ensure cost-efficient numerical process design. Therefore, a new implementation for the FE-software Simufact Forming 16.0 is presented in this work. The results are compared and validated with the original implementation by means of a numerical compression test and a cyclic simulation is calculated with Simufact Forming.

Keywords: Martensitic die steel · Plasticity model · Hot forging

1 Introduction

In hot forging processes, the dies are subjected to high loads. These loads can be divided into mechanical, thermal, tribological and chemical loads [1]. The high cyclic thermo-mechanical loads can result in crack initiation and propagation due to material softening. Failure of the forging dies occurs when the crack finally exceeds a critical length [2]. This fatigue behaviour is also known as thermo-mechanical fatigue (TMF) and can cause unexpected failure of the dies [3].

The simulation-based design of forging dies and processes with the help of the finite element method (FE) is increasingly used nowadays [4]. In many cases, simulations are only carried out in relation to the semi-finished products and their forming process using the FE method. This simulation method ensures a cost-efficient development process as the simulation time is low compared to elastic-plastic or thermally coupled simulations considering the dies [5]. The simultaneous or additional consideration of the dies in the simulation process enables an estimation of the stresses on the forging dies under

M. Liewald et al. (Eds.): WGP 2022, LNPE, pp. 81–90, 2023.
https://doi.org/10.1007/978-3-031-18318-8_9

the stresses caused by the forming process and can thus be used to predict the die stability and life time. In addition, numerical simulations can be used to optimise the dies in a cost-effective manner and to increase the service life time by optimisation [6]. Usually isothermal linear-elastic simulations are used, where the focus is on the load in the first cycle. However, the effect of plasticity due to stresses that locally exceed the temperature-dependent yield strength at high operating temperatures is neglected [2]. In addition, cyclic loading on the die results in material softening, which leads to a reduced service life time. This holistic view from the simulation process with the cyclic consideration is the final goal of the project, and this paper presents the necessary material model with the softening behaviour.

The high strength of hot forging steels is a result of the so called Orowan mechanism. Strengthening secondary carbides are precipitated during heat treatment and hinder the slip of dislocations [7, 8]. However, an effect called Ostwald ripening occurs at high temperatures leading to coarsening of the strengthening carbides with time and, thus, reducing the material's strength [9]. Thermal softening of the material was observed in high temperature applications comparable to forging dies [10]. The evolution in the microstructure due to thermo-mechanical loading of hot forging tool steels was for example addressed in [11]. It was found that the microstructure and the bond between the particles depend on the tempering behaviour.

The effects of TMF and thermal softening are well known. Nevertheless, a meaningful implementation in the context of a cyclic viscoplasticity material model for hot forging tool steels is not yet available and therefore not yet accessible in FE-simulations. Zhang et al. [12] developed a material model for the prediction of thermal fatigue for the martensitic steel 55NiCrMoV7, which was commonly used as material for hot forging tool steels. The development of the cyclic anisothermal plasticity model including ageing effects was published in [13].

Jilg et al. developed an approach to assess plastic deformation in FE-simulations for the hot forging tool steel 1.2367 (X38CrMoV5-3) including thermal softening [14]. A kinetic model describes the strengthening effect and the temperature dependent coarsening of the carbides [15]. This model was implemented in the FE-software Abaqus. However, in the forging industry application-specific software is usually used to ensure cost-efficient numerical process design. In addition, the use of application-specific software makes it possible to calculate processes with strong non-linear deformations due to efficient and robust remeshing algorithms.

Within this study it is the aim to implement the plasticity model developed by Jilg et al. [14] in a time-dependent (viscoplastic) formulation for Simufact Forming 16.0 that is commonly used for the design of hot forging processes by means of a user subroutine. Thus, it can be used in future for efficient service life time calculation of complex hot forging processes considering cyclic thermal-mechanical loadings.

2 Model Description and Implementation

2.1 Viscoplasticity Model

The extension of the plasticity model of Jilg et al. [14] to non-isothermal viscoplastic behaviour is based on the works of Chaboche [16]. For simplicity, the viscoplasticity model is presented in the uniaxial formulation in this section whereas the FE-implementation is based on the multiaxial formulation using the von Mises yield function. To describe thermal softening, the model considers a kinetic model, which was originally presented in [14].

The nominal stress σ is calculated using Hooke's Law with young's modulus E and the total strain ε^{tot} by

$$\sigma = E \cdot \left(\varepsilon^{\text{tot}} - \varepsilon^{\text{th}} - \varepsilon^{\text{vp}}\right) \tag{1}$$

considering the thermal strain εth and the viscoplastic strain ε^{vp}. The thermal strain is calculated with the coefficient of thermal expansion αth. The viscoplastic strain ε^{vp} is calculated by integration of the equation

$$\dot{\varepsilon}^{\text{vp}} = \dot{\overline{\varepsilon}}^{\text{vp}} \frac{\sigma - \alpha}{|\sigma - \alpha|} \text{ with } \dot{\overline{\varepsilon}}^{\text{vp}} = \frac{|\sigma - \alpha| - R_p}{K}^{n}. \tag{2}$$

The viscoplasticity model takes time-dependent effects like strain rate dependency into account using the material properties K and n. This model contains kinematic and isotropic hardening. Kinematic hardening is implemented through the variable α named backstress (see Eq. 3) so that the Bauschinger effect occurring under cyclic loading can be described with the model. Isotropic hardening is considered through the yield strength R_p (see Eq. 4). Macaulay brackets ensure that viscoplasticity only occurs if there is an overstress, i. e. expression in the brackets is greater than zero.

The backstress is computed via the following evolution equation representing an extension of the kinematic hardening law proposed by Frederick and Armstrong [17]:

$$\dot{\alpha} = C \cdot \dot{\varepsilon}^{\text{vp}} - \gamma \cdot \dot{\overline{\varepsilon}}^{\text{vp}} \cdot \alpha - R \cdot \alpha + \frac{\partial C}{\partial T} \frac{\dot{T}}{C} \cdot \alpha + \frac{\partial C}{\partial r} \frac{\dot{r}}{C} \cdot \alpha \tag{3}$$

C and γ are the kinematic hardening variables, R is a material property for static recovery which gives a recovery of hardening with time. The last two terms of Eq. 3 are temperature and particle expressions that can be derived from thermodynamic principles presented in [18]. Isotropic softening is implemented through

$$R_p = R_e + Q_\infty \cdot \left(1 - e^{-b \cdot \overline{\varepsilon}\text{vp}}\right). \tag{4}$$

Initially, R_p corresponds to the initial yield strength R_e but changes depending on the accumulated plastic strain $\overline{\varepsilon}^{\text{vp}}$. For initiation of softening Q_∞ must be a negative value, $R_\text{e}+Q_\infty>0$. The proportionality constant b defines how fast softening occurs with increasing accumulated plastic strain.

The strengthening effect of secondary carbides on hot forging tool steels due to the Orowan mechanism was for example described in Eser et al. [7] and Caliskanoglu et al. [8]. This effect is considered in the viscoplasticity model using the radius r of a representative strengthening particle that is assumed to control the strength of the material. For hot forging tool steels, coarsening of secondary carbides was observed and explained as Ostwald ripening [9]:

$$\dot{r} = \frac{k_c}{r^2} \tag{5}$$

The coarsening leads to thermal softening of the material and is implemented into the viscoplasticity model through the temperature dependent coarsening constant k_c. The particle radius has impact on material properties through formulation of expressions depending on the particle radius [3]. Overall, the material properties are determined based on low-cycle-fatigue (LCF) tests at different temperatures as presented in [4].

2.2 Implementation

It is the aim of this work to implement the viscoplasticity model for the tool steel 1.2367 (X38CrMoV5-3) described in Sect. 2.1 in Simufact Forming. Therefore, some basic definitions that are important for the implementation of the model in Simufact Forming are presented. The specifications from Abaqus are taken from the Abaqus documentation [19] and the Simufact Forming specifications from the MSC Marc Volume D [20].

In previous works the viscoplasticity model was implemented and validated in Abaqus as an external subroutine called User Material (UMAT) using the algorithms described in [21]. Abaqus brings its own implicit FE solver while Simufact Forming uses a modified solver from MSC Marc. For Simufact Forming, a similar environment to UMAT called Hypela-2 is available [20]. This model was used for example in the research of Schmaltz to describe the material behaviour of sheet materials at large shape changes [22]. Hypela-2 is used to compute update of stresses and the internal variables (e. g. accumulated plastic strain or particle radius) at a time point t_{n+1} based on the values of stress and internal variables at time t_n. To this end, the increments of total strain and temperature are provided by the FE-software. Moreover, the consistent material tangent needs to be defined within Hypela-2. In the used configuration strains are provided as logarithmic strains which allow depiction of large deformations. Stresses have to be defined in a rotated coordinate system considering rigid body rotations.

The subroutine uses a predictor-corrector method [23]. In every time increment a check is performed if the stresses are within or outside of the elastic range. Plastic correction of the results is only carried out if the stresses are outside the elastic range. The standard MARC solver uses implicit integration scheme, so the material model is also implemented by means of implicit calculation using the Euler backward method. A schematic representation of the simulation workflow in Simufact Forming is presented in Fig. 1.

At the beginning, Simufact Forming provides several input data of the simulation to the subroutine during initialisation. The material properties are defined in Hypela-2 before the subroutine is called. Communication between the subroutine and Simufact Forming takes place, whereby Simufact Forming provides the forming and the individual parameters to the subroutine. Additional routines of MSC Marc are implemented in Hypela-2 for visualisation of the internal variables in the post processing.

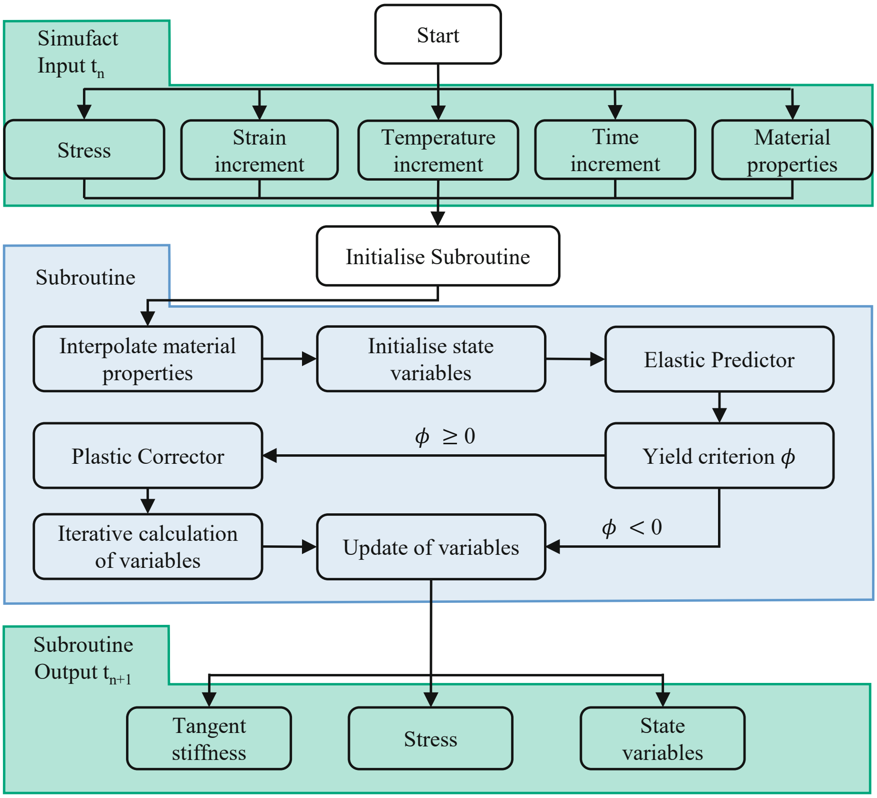

Fig. 1. Schematic representation of the simulation workflow in Simufact forming.

3 Simulation Model in Simufact Forming and Abaqus

For the comparison of the viscoplasticity model in Simufact Forming, simulations are also carried out in Abaqus. For this, a static compression and cyclic tests at various elevated temperatures (between 20 and 650 °C) are simulated, whereby the billet corresponds to the later deformable die in the planned forming process, already initially designed in the work of Behrens et al. [6]. The billet part is computed with the viscoplasticity model and the available material properties such as young's modulus are provided for the material 1.2367 within the material model. The focus is primarily on small deformations, since the later die will also only undergo small deformations.

In Abaqus boundary conditions as the displacement can be specified direct on the geometry. However, in Simufact Forming a press kinematic in combination with dies has to be modelled for a displacement-controlled compression test. To model an isothermal case the upper and lower die are defined as a rigid body without heat conduction. The billet is located between the analytical dies in Simufact Forming (see Fig. 2).

In the first part of the investigation, the focus is on a static compression test (see Fig. 2, left). During these simulations, only one cycle of compression is calculated in order to check the general runability of the material model in Simufact Forming. In this simulation no friction is defined between the billet (purple) and the dies (grey). These parameters are chosen in regard to the Abaqus reference model. Hence, a homogeneous stress and temperature distribution in the billet should occur. A rotationally symmetric cube with a radius of 10 mm and a height of 10 mm is defined as geometry for the simulations. A displacement of 0.1 mm with a forming speed of 1 mm/s is investigated. By choosing these forming parameters, both the linear-elastic range and the plastic range can be considered. The billet is meshed with overall nine equally spaced rectangle elements. This number has been chosen in such a way that exactly only one element is in the middle. The remeshing is switched off for these simulations.

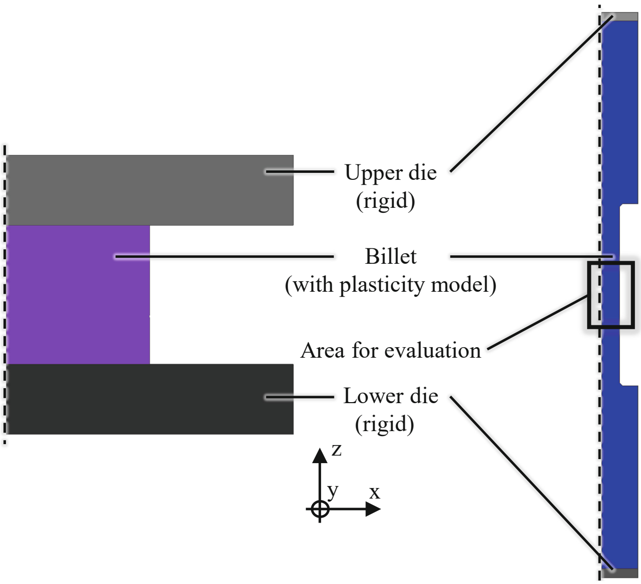

Fig. 2. Exemplary simulation models (left: static; right: cyclic) in simufact forming 16.0.

After the static compression tests, cyclic tests are simulated in the second part of this investigation (see Fig. 2, right). These calculations are carried out for comparison of the cyclic behaviour of the viscoplasticity model. Thus, the runnability of the subroutine must also be guaranteed for this type of simulation. A round tensile specimen is chosen as the geometry. The area used for the evaluation is in the middle of the smaller area of the specimen (area in the black box) and has the same radial dimension as the specimen in the static compression test. By choosing a cyclic loading and unloading, an adherent contact between the dies and the billet must be selected. A larger distance between the contact area and the evaluation area is chosen so that the effects on the stresses do not influence the results from the subroutine. As in the previous simulations, the heat transfer between all components is exhibited. The set mesh has the same edge lengths as in the previous model, so that again only one element is in the middle of the considered area. As before the remeshing in the billet is switched off. The choice of settings aims to achieve a homogeneous stress and strain condition in the evaluation area. A total of twelve cycles over a time of 500 s is examined. The displacement of the upper edge is controlled by the forming speed. The forming speed is linearly reduced / increased to a speed of 1.2 mm/s within 20 s. This specification results in a triangular profile of the forming speed with time.

4 Results and Discussion

After successful implementation of the subroutine in the simulation software Simufact Forming, the results are calculated for the static compression and cyclic tests described in Sect. 3 and the static simulations are compared with the results of Abaqus. Comparison of the results is important for the verification of the viscoplasticity model calculated using Simufact Forming (orange) and Abaqus (blue) as shown in Fig. 3. The evolution of the stress component in forming direction and the accumulated plastic strain with increasing forming time at different temperatures is shown. Up to a forming time of 0.03 s, a linear stress curve is shown, as expected for linear-elastic material behaviour. From 0.03 s to the end of the simulation, a non-linear stress curve is observed. The lines of the accumulated plastic strain follow a straight course up to a time of 0.03 s. After that, an increase of the values can be seen. In both results, the accumulated plastic strain as well as the stress, an increase respectively decrease of the value with increasing temperature can be seen. The influence of isotropic hardening is small in this simulation due to the low plastic strains.

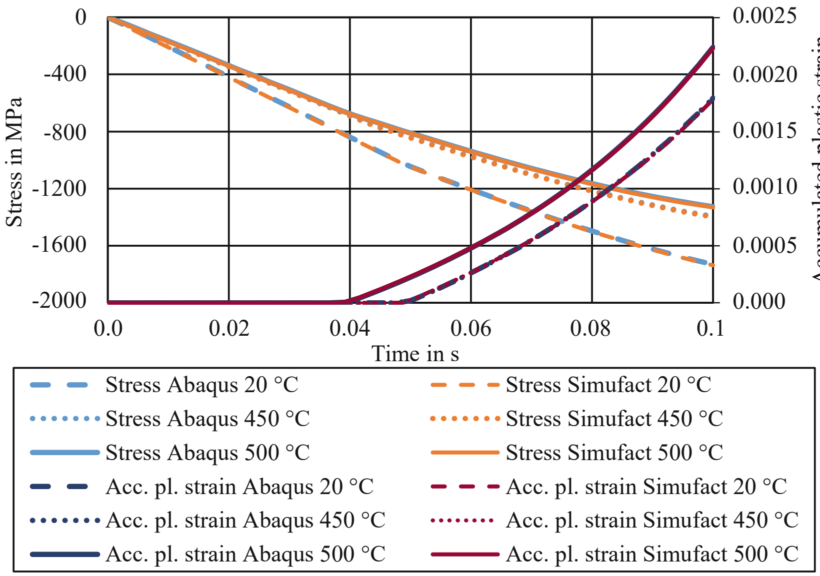

Fig. 3. Comparison of stress and accumulated plastic strain in forming direction.

In addition, the cyclic tests are carried out on a round full tensile specimen in Simufact Forming and Abaqus. These tests are carried out to compare the effects of the viscoplasticity model and the cyclic softening behaviour of the model. The cyclic tests are carried out displacement controlled by specification of the forming speed as mentioned in Sect. 3. Figure 4 shows the results of the cyclic tests as stress-strain hysteresis which is commonly used for the representation of cyclic tests. The figure includes the simulation results at 500 and 600 °C from Simufact Forming and Abaqus. Additionally, results from the first isothermal fatigue tests is presented for 500 and 600 °C marked by triangle symbols for quantitative comparison of the model with the experimental data. The stress values of the fatigue tests are on the same level as the model results for both temperatures. Figure 4 shows that the results of both simulation programmes agree. For 500 °C higher stresses are present and only small stress relaxation appears. The results reach a maximum stress value of about 1150 MPa and a maximum strain value of about 0.008. The initial stresses at 600 °C (900 MPa in the first cycle) are smaller than at 500 °C as indicated by the material properties. The cyclic tests show stress relaxation and thus the impact of the viscoplasticity model. Additionally, material softening occurs which can be evaluated by comparing the stress range in the first cycle to the last cycle at 600 °C. The softening in stress range is 30 MPa from the first to the last cycle of the test. Furthermore, an increase in the maximum strain from 0.008 to 0.01 can be observed.

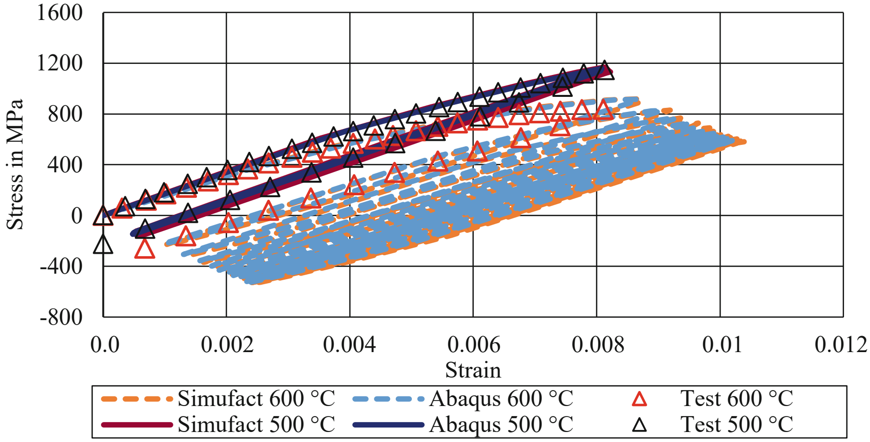

Fig. 4. Experimental and numerical results of cyclic tests calculated in Simufact Forming and Abaqus.

This work shows that the mathematical viscoplasticity material model can be used in the forging-specific FE software Simufact Forming with Hypela-2. A material softening could be proven by a cyclic calculation in both simulation programmes.

5 Conclusion and Outlook

Within this work, the implementation of a viscoplasticity model with material properties available for the hot forging tool steel 1.2367 in the forging specific FE-Software Simufact Forming was demonstrated by means of using a subroutine Hypela-2. The implemented model allows for considering material softening in the common industrial software for development of hot forging dies.

The simulation in this work shows that the integration of the subroutines was possible in Simufact Forming. Furthermore, it was shown that the calculations in Simufact Forming agree with the results from Abaqus for the static compression and cyclic tests. In addition, the cyclic tests demonstrated a softening of the material over the cycles at higher temperature.

In future works the viscoplasticity model has to be used in FE-simulations of forging dies under cyclic loading conditions and validated with experimental tests. Effects such as transient temperature fields and inhomogeneous loads will be considered. The developed and implemented model allows the realistic calculation of the development of plastic strains and stresses within the forging die under consideration of cyclic thermo-mechanical loads as well as their influence on material hardening and softening. This data should be coupled with a life time model. Finally, a realistic prediction of the die's life time can be made which allows cost reduction and resource saving.

Acknowledgement. The results presented were obtained in the research project "Development of a methodology for evaluating the fatigue life of highly loaded hot forming tools based on advanced material models" financed under project number 244928365 by the German Research Foundation (DFG). The authors would like to thank the German Research Foundation for the financial support.

References

1. Behrens, B.-A., Bouguecha, A., Hadifi, T., Klassen, A.: Numerical and experimental investigation on the service life estimation for hot-forging dies. In: Key Engineering Materials, vol. 504–506, pp. 163–168. (2012)
2. Ebara, R.: Fatigue crack initiation and propagation behavior of forging die steels. Int. J. Fatigue **32(**5), 830–40 (2010)
3. Oudin, A., Penazzi, L., Rézaï-Aria, F.: Life prediction of hot work tool steels subjected to thermomechanical fatigue. In: Matériaux and Techniques, vol. 88, pp. 67–72. (2000)
4. Bouguecha, A., Behrens, B.-A., Bonk, C., Rosenbusch, D., Kazhai, M.: Numerical die life estimation of a crack susceptible industrial hot forging process. In: AIP Conference Proceedings, vol. 1896, pp. 190012. (2017)
5. Haselbach, P., Natarajan, A., Jiwinangun, R.G., Branner, K.: Comparison of coupled and uncoupled load simulations on a jacket support structure. Energy Proc. **35**, 244–252 (2013)
6. Behrens, B.-A., Rosenbusch, D., Wester, H., Siring, J.: Numerical investigations on stresses and temperature development of tool dies during hot forging. In: 25th International Esaform Conference on material Forming, Portugal (2022)
7. Eser, A., Broeckmann, C., Simsir, C: Multiscale modeling of tempering of AISI H13 hot-work tool steel—Part 1: prediction of microstructure evolution and coupling with mechanical properties. Comput. Mater. Sci. **113**, 280–291 (2016)
8. Caliskanoglu, D., Siller, I., Ebner, R., Leitner, H., Jeglitsch, F., Waldhauser, W.: Thermal fatigue and softening behavior of hot work tool steels. In: Bergstrom, J. (eds.) Proceedings of the 6th International Tooling Conference, Karlstad University Karlstad, pp. 709–719. (2002)
9. Oriani, R.A.: Ostwald ripening of precipitates in solid matrices. Acta Metallurgica **12**(12), 1399–1409 (1964)

10. Sjostrom, J., Bergström, J.: Thermal fatigue in hot-working tools. Scandinavian J. Metallurgy **34**(4), 221–231 (2005)
11. Zhang, Z., Delagnes, D., Bernhart, G.: Microstructure evolution of hot-work tool steels during tempering and definition of a kinetic law based on hardness measurements. In: Material Science Engineering A, vol. 380(1–2), pp. 222–230. (2004)
12. Zhang, Z., Delagnes, D., Bernhart, G.: Anisothermal cyclic plasticity modelling of martensitic steels. Int. J. Fatigue **24**(6), 635–648 (2002)
13. Zhang, Z., Bernhart, G., Delagnes, D.: Cyclic behaviour constitutive modelling of a tempered martensitic steel including ageing effect. Int. J. Fatigue **30**(4), 706–716 (2008)
14. Jilg, A., Seifert, T.: Temperature dependent cyclic mechanical properties of a hot work steel after time and temperature dependent softening. In: Materials Science and Engineering: A, vol. 721, pp. 96–102. (2018)
15. Jilg, A., Seifert, T.: A temperature dependent cyclic plasticity model for hot work tool steel including particle coarsening. In: AIP Conference Proceedings 1960, pp. 170007. (2018)
16. Chaboche, J.L.: Constitutive equations for cyclic plasticity and cyclic viscoplasticity. Int. J. Plasticity **5**, 247–302 (1989)
17. Frederick, C.O., Armstrong, P.J.: A mathematical representation of the multiaxial bauschinger effect. In: Materials at High Temperatures, vol. 24, pp. 1–26. (2007)
18. Chaboche, J.: Cyclic viscoplastic constitutive equations, Part I: A thermodynamically consistent formulation. J. Appl. Mechan. (ASME) **60**(4), 813–821 (1993)
19. Systèmes, D.: ABAQUS 6.16: Analysis User's Guide (2016)
20. MSC Software GmbH: Volume D: User Subroutines and Special Routines. pp. 312–317. (2018)
21. Seifert, T., Schmidt, I.: Line-search methods in general return mapping algorithms with application to porous plasticity. Int. J. Numer. Methods Eng. **73**, 1468–1495 (2008)
22. Schmaltz, S.: Inverse Materialparameteridentifikation von Blechwerkstoffen für ein anisotropes elasto-plastisches Materialmodell bei finiten Deformationen. In: Schriftenreihe Technische Mechanik, Band 14, Thesis for Doctoral, Erlangen (2015)
23. Wriggers, P.: Nonlinear Finite Element Methods. Springer, Berlin, Heidelberg (2008)