THE MODELING OF COMPRESSION TESTS WITH A PHYSICALLY BASED MATERIAL MODEL USING FINITE ELEMENTS

A.D. Rietman(1), P. van Liempt(2) and H. Huétink(1)

(1) University of Twente  
Department of Mechanical Engineering  
Applied Mechanics Group  
P.O. Box 217  
7500 AE Enschede  
THE NETHERLANDS

(2) Hoogovens Research & Development  
Rolling and Coating Technology  
Koninklijke Hoogovens  
P.O. Box 10000  
1970 CA IJmuiden  
THE NETHERLANDS

Abstract. Compression experiments are commonly used to determine material behavior and friction characteristics between tools and workpiece. To validate a physically based material model for metals at elevated temperatures, which was developed at Hoogovens Research & Development, finite element calculations have been performed. The finite element code DIEKA used to do this is based on a mixed formulation and is developed by the Applied Mechanics Group. The calculations show that the material model performs well. A comparison with experiments could not be made while experimental data is not yet available.

1 INTRODUCTION

In most metal forming processes deformation rates are (locally) very high. This can be seen for instance in the stationary processes like rolling and extrusion, but also in forging. In these processes the temperatures are usually taken very high to reduce process forces. Due to dissipation the temperatures can become higher. The existing simple material models do not account for phenomena that occur at high strain rates and temperatures or do account only on an observational basis, what is usually not covering all strain rate and temperature combinations. This is due to the fact that the complete deformation history determines further deformation behavior. Hence the need arises for more complex, physically based material models which incorporate the evolution of strain, high strain rates and high temperatures.

In this work use has been made of a newly derived material model supplied by Hoogovens Research & Development. This model is largely based on microstructure evolution during deformation and accounts for the previously mentioned conditions. To validate the model and the finite element implementation some compression simulations have been performed with the finite element code DIEKA. By using an inverse approach the material parameters can be determined to optimize the model.

The heavily distortion of the elements near the edges of the specimen is prevented for by using a mixed formulation. The nodal co-ordinates can be positioned independently within the discretized domain at user's specification. Together with a transfinite mapping technique the finite element mesh thus retains regularly shaped.

2 CONSTITUTIVE EQUATIONS

The constitutive equations for mechanical equilibrium do relate the stresses to the conjugated strains. In this paper the viscoplastic flow formulation [1], sometimes referred to as rigid-viscoplastic, will be discussed.

In metal forming processes the plastic
part of the total deformation usually is that large that the elastic part can be neglected. A flow formulation then is an appropriate choice. Starting with the assumption of associated flow in which the plastic strain increment is related to the normal of the yield surface via a consistency parameter. The consistency parameter follows from the substitution of the yield criterion by considering the specific power. The specific power has to be the same whether it is calculated by an inner product of the tensors or by their equivalent counterparts. The deviatoric part of the Cauchy stress tensor is related to the plastic strain rate tensor by

$$s = 2\mu(\dot{\varepsilon}, \dot{\varepsilon}, T)\dot{\varepsilon}^p$$

(1)

where $\lambda$ is the (constant) thermal conductivity and $\nabla$ is the gradient operator.

3 MATERIAL MODEL

The material model used in this paper is derived on a physical basis by Hoogovens Research & Development. The newly derived model is largely based on earlier work [3], except for the influence of microstructure evolution at elevated temperatures.

The material model is restricted to isotropic materials only and determines the relation between the flow stress and the effective strain. It is valid for large strain ranges and accounts for high strain rates and elevated temperatures. The flow stress consists of three terms:

$$\sigma_f(\dot{\varepsilon}, \dot{\varepsilon}, T) = \Theta(T)\sigma_0 +$$

$$+ \Theta(T)\sigma^*(\dot{\varepsilon}, T) + \sigma_w(\dot{\varepsilon}, \dot{\varepsilon}, T),$$

(6)

The first part of the right hand side is the temperature corrected initial flow stress of the undeformed polycrystal. The second term is the thermal hardening or dynamic deformation resistance which is a semi-empirical relation which accounts for the thermally activated motion of dislocations [4]. It is written as

$$\sigma^*(\dot{\varepsilon}, T) = \sigma_0^*\left(1 - \frac{\Delta G}{\Delta G_0}\right)^p,$$

(7)

$$0 \leq \Delta G \leq \Delta G_0,$$

with $\sigma_0^*$ the maximum effective dynamic stress and $\Delta G$ is the activation free energy. The maximum free energy for a dislocation to overcome a barrier is given by $\Delta G_0$ and the power $p$ is an experimentally determined material constant. The activation free energy is given by a Boltzmann type of relation:

$$\Delta G = -kT \ln \left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)$$

(8)

where $k$ is Boltzmann's constant and $\dot{\varepsilon}_0$ is the maximum effective strain rate. All
terms in eq. (6) are corrected for the temperature influence by means of a polynomial fit of the shear modulus \( G \) in temperature space

\[
\Theta(T) = \frac{G(T)}{G(T_r)}.
\]

The reference temperature \( T_r \) is usually taken to be room temperature. When the activation free energy becomes negative, i.e. the effective strain rate exceeds the maximum effective strain rate, total immobility of dislocations occurs and the dynamic stress equals the maximum dynamic stress. When on the other hand the activation free energy exceeds the maximum free energy, then the factor between brackets in eq. (7) becomes negative, what is not of any physical meaning. Then dislocations move at sonic velocities. The contribution to the already large dynamic resistance by this kind of motion will be small. The dynamic part of the total deformation resistance is negligible at high temperatures, even for the higher effective strain rates.

The latter term in the right hand side of eq. (6) is referred to as workhardening. This workhardening term contains the kinetics and interactions of dislocations like pinning and remobilization. It is given by a classical relation which depends on the dislocation density \( \rho \),

\[
\sigma_w(\dot{\varepsilon}, \dot{\varepsilon}, T) = \alpha G(T) b \sqrt{\rho(\dot{\varepsilon}, \dot{\varepsilon}, T)},
\]

in which \( \alpha \) is a crystallographic constant and \( b \) is the Burgers vector. The dislocation density is determined by a set of ordinary differential equations (ode’s) describing the evolution of some state variables \( f \). These state variables describe the microstructure at current strain state. Thus

\[
\rho(\dot{\varepsilon}, \dot{\varepsilon}, T) = \rho(f(\dot{\varepsilon}, \dot{\varepsilon}, T))
\]

while the set of ode’s is given by

\[
\frac{df}{d\varepsilon} = F(f, \varepsilon, \dot{\varepsilon}, T)
\]

with \( F \) a vector function. In some cases, like for instance in cold rolling [3, 5], this implicit function reduces so that it can be integrated exactly, leading to a dislocation density which is a unique function of strain:

\[
\rho(\varepsilon) = \left[ \frac{U \left( \frac{G(T)}{G(T_r)} - 1 \right)}{\Omega + 2\beta} + \sqrt{\rho_0} \right]^2 e^{-\Omega \varepsilon}.
\]

In this unique function \( U \) is the dislocation mobility, \( \Omega \) the remobilization probability and \( \beta \) is a cell size parameter [3]. The initial dislocation density is referred to as \( \rho_0 \). Otherwise numerical procedures to solve eq. (12) have to be applied.

The material model as presented here is still in an experimental phase, however the main microstructure effects, like creation, immobilization and remobilization of dislocations, is accounted for. Recrystallization is neglected below the recrystallization temperature since its time scale strongly exceeds the time scale for forming processes.

Calculations with a stand-alone version of the material model show a strong deformation rate and temperature dependency. The results of the calculations with varying temperatures are to be seen in figure 1.

![Figure 1. Relative flow stress vs. true strain at constant rate of 1 s⁻¹.](image)

In this figure the relative stress is plotted as a function of the effective true strain for several constant temperatures. This is done for one effective strain rate of 1 s⁻¹ for a particular steel. It can be seen that the different curves can cross each other.
This will be referred to as curve-crossing. It is caused by the influence of the temperature on the diffusivity of interstitials and thus on their probability of pinning free dislocations. Dislocations will become immobile faster which leads to an increased hardening.

Curve-crossing cannot only be seen in varying temperatures, but also in varying deformation rates. Temperatures around 700 K are the most interesting temperatures to see the phenomenon of curve-crossing, because both ageing and recovery do occur in this temperature range.

When performing these calculations with an interstitial free steel curve-crossing does not occur. The results can be seen in figure 2.

![Figure 2: Relative flow stress vs. true strain for an interstitial free steel at constant rate of 1 s⁻¹.](image)

No extra hardening takes place while no interstitials are available for pinning dislocations. Deformation requires less process force which is the main reason for application of interstitial free steels in forming processes.

4 FINITE ELEMENT METHOD

In order to solve the problems by means of the finite element method the physical domain and the constitutive equations have to be discretized.

4.1 Space and time discretization

The physical domain is discretized in nodal points and finite elements. Here use is made of quadrilateral bi-linear elements. The nodal co-ordinates are interpolated the same way as the independent nodal quantities, the degrees of freedom (dof's), which are the velocities and temperature rate of change in the nodal points. To prevent for volume locking in case of incompressibility the isotropic terms of the stress tensor are interpolated 0th order. The time domain is discretized in time steps sufficiently small to stabilize the calculation.

4.2 Incremental flow formulation

The incremental flow formulation is given by

\[ \sigma = \frac{2\sigma_f(\varepsilon, \dot{\varepsilon}, T)}{3\Delta \varepsilon} \Delta \varepsilon - pI \]  

(14)

with

\[ p = p_0 + \Delta p = p_0 + K \text{tr}(\Delta \varepsilon) + \alpha_0 K \Delta T \]  

(15)

with \( p_0 \) is the known pressure at the beginning of the time step, \( \alpha_0 \) is the volumetric thermal expansion coefficient and \( K \) is the bulk modulus. Elastic volume recovery is fully when the pressure decreases to zero. Furthermore it is used that within a time increment

\[ \Delta \varepsilon = \dot{\varepsilon} \Delta t \]  

(16)

\[ \Delta T = \dot{T} \Delta t \]  

(17)

while the effective strain increment can be calculated likewise eq. (2).

While the constitutive relations are strongly non-linear an iterative solving procedure has to be used. In the program used in this paper the iterative procedure is a predictor-corrector algorithm with load correction. In the predictor the linearized equations are solved leading to a first approximation of the values of the dof's of the system. Based on these values the new incremental strain field can be calculated. By means of the non-linear constitutive relations the new stress state can then be calculated.
Therefore the new values of the state variables have to be known. The calculation of these state variables can be regarded as an initial value problem each time step:

$$\frac{df}{d\varepsilon} = F(f, \varepsilon, \dot{\varepsilon}, T)$$

with

$$f^{(n)}(\varepsilon^{(n-1)}, \dot{\varepsilon}^{(n)}, T^{(n)}) = f^{(n-1)}$$

in which the upper index denotes the step number and the lower index the iteration number. The initial value of the state variables is determined by the strain at the beginning of the step, i.e. the final value of the previous step. The classical Runge-Kutta method is used to integrate the set of ode's along the strain path. Based on the values of the state variables the new stress state in the current iteration can be determined using eq. (10). While the equations are nonlinear these stresses and strains will usually not be in equilibrium such that successive predictor-corrector stepping might be necessary.

### 4.3 Mixed algorithm

The finite element code DIEKA has been developed during the last 15 years by Huétink et al. [6, 7]. It is based on an Arbitrary Lagrangian-Eulerian (ALE) method which combines the main advantages of both the Lagrangian and the Eulerian method [7, 8, 9]. The mixed algorithm in DIEKA is referred to as forward decoupled ALE. After equilibrium is fulfilled in Lagrangian sense, the finite element mesh is re-ordered according to user's specifications with use of a transfinite mapping algorithm [10, 11]. A convective grid displacement is introduced. By doing so equilibrium can be disturbed but it should be noted that the convective grid displacements are relatively small since the algorithm is performed every time step. Furthermore the deviation from equilibrium can be minimized in the next Lagrangian calculation procedure.

### 5 SIMPLE UPSETTING

Simple upsetting is a commonly used test to determine material behavior. The measured punch force and punch displacement can be converted into stress-strain curves under the assumption that the deformation field is homogeneous. The set-up is displayed in figure 3.

![Figure 3. 2D representation of simple upsetting.](image)

#### 5.1 Properties

The process parameters used for this calculation can be seen in table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>workpiece dimensions D:h (mm)</td>
<td>5:10</td>
</tr>
<tr>
<td>total punch speed (mm/s)</td>
<td>10</td>
</tr>
<tr>
<td>init. workpiece temperature (K)</td>
<td>700</td>
</tr>
<tr>
<td>tool temperature (K)</td>
<td>700</td>
</tr>
</tbody>
</table>

Table 1. Process parameters for simple upsetting.

In this table $D$ is the diameter and $h$ is the height of the specimen. The specimen will heat due to dissipation while the tools are kept at a constant temperature. The free surface is insulated. Because of the twofold symmetry in the specimen only one quarter of the vertical cross-section (see figure 3) has to be meshed. For meshing a transfinite mapping technique has been used to obtain a grid consisting of 300 axial symmetric thermal elements. The contact between tools and workpiece is implemented using 10 so-called contact elements and involves friction (slip-stick
behavior), normal stiffness and heat conduction. The tools are assumed to be rigid and are defined using contours. The same material is used as was used in section 3.

Some material properties that are assumed to be temperature independent are given in table 2.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poisson's ratio $\nu$</td>
<td>0.3</td>
</tr>
<tr>
<td>lin. expansion coeff. $\alpha$ (1/K)</td>
<td>$15 \times 10^{-6}$</td>
</tr>
<tr>
<td>heat conductivity $\lambda$ (W/mK)</td>
<td>57.8</td>
</tr>
<tr>
<td>mass density $\rho$ (kg/m$^3$)</td>
<td>7820</td>
</tr>
</tbody>
</table>

Table 2. Temperature independent properties.

The specific heat and Young's modulus are linearly interpolated between two specified temperatures to be seen in table 3.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature (K)</td>
<td>300 1050</td>
</tr>
<tr>
<td>Young's modulus $E$ (GPa)</td>
<td>210 150</td>
</tr>
<tr>
<td>specific heat $c$ (J/kgK)</td>
<td>480 690</td>
</tr>
</tbody>
</table>

Table 3. Temperature dependent properties.

Friction is modeled using Coulomb friction with a friction coefficient $\mu = 0.2$.

5.2 Results

The results of a simulation of simple upsetting will be discussed in this section. With the ALE method reductions of over 80 percent are possible. The simulated force-displacement diagram is shown in figure 4.

Figure 4. Simulated punch force vs. punch displacement.

When the reduction exceeds about 60 percent the punch force increases fast due to the large contact area.

The aim of these compression tests is to determine the stress-strain relation. With the inverse approach a first guess of the stress-strain relation is needed as input for the simulation. A first guess is obtained by assuming a homogeneous deformation field. The stress-strain curve then follows directly from the force-displacement diagram. Experiments are not yet carried out. Hence here a comparison between the input stress-strain relation and the calculated force-displacement curve is made.

The deformed mesh can be seen in figure 5.

Figure 5. Finite element mesh at 50 % reduction.

However with a certain amount of friction between tools and workpiece the deformation will not be homogeneous as can be seen from the calculated effective strain distribution in figure 6.

Figure 6. Simulated effective strain distribution (50 % reduction).

It can be seen that in the specimen a so-called dead metal zone arises in the upper region along the center line. This is caused by stick phenomena along the tool workpiece interface. Neglecting the inhomogeneous deformation the force-displacement curve can be converted into a stress-strain...
curve. The equivalent stress is obtained by dividing the force by the mean cross-section area, assuming a uniaxial state of compression. The effective strain is obtained by the definition of logarithmic strain. Together with the stress-strain relation for homogeneous deformation the obtained stress-strain curve is displayed in figure 7.

![Figure 7. Relative flow stress vs. true strain.](image)

However the deformation is strongly inhomogeneous the obtained material behavior is quite good in comparison with the input for a homogeneous deformation. The input has been corrected for the fact that the mean effective strain rate is not a constant during compression since the punch speed is already a constant. The calculation method of the mean cross-section area influences the obtained stress curve. It can also be seen that for true strains above 1 the 'experimentally' obtained material behavior fails because of the strong influence of friction at this large reduction. Then an extrapolation has to be used.

6 RING COMPRESSION

To determine frictional behavior at the interface the ring compression test is a very useful test. The inner and outer diameter do vary under different friction conditions. The set-up is displayed in figure 8.

6.1 Properties

The process parameters used for this simulation can be seen in table 4.

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>ring dimensions $d:D:h$ (mm)</td>
<td>20:60:20</td>
</tr>
<tr>
<td>total punch speed (mm/s)</td>
<td>10</td>
</tr>
<tr>
<td>init. workpiece temperature (K)</td>
<td>700</td>
</tr>
<tr>
<td>tool temperature (K)</td>
<td>700</td>
</tr>
</tbody>
</table>

Table 4. Process parameters for ring compression.

In this table $d$ is the inner diameter, $D$ is the outer diameter and $h$ is the height of the specimen. Again only one quarter of the cross-section has to be meshed in finite elements because of symmetry. For this simulation 600 axial symmetric thermal elements have been used together with 20 thermal contact elements. The free surfaces again are insulated and the material that has been used for this test is the same as was used in the simulation of simple upsetting. Friction again is modeled with a Coulomb friction model using a friction coefficient $\mu = 0.2$.

6.2 Some results

The results presented here are results after a reduction of 50 percent, which took 1900 time steps. In figure 9 the particle tracking lines in both the radial as well as the axial direction are depicted.

![Figure 9. Particle tracking in both directions, together with the initial shape.](image)

Particle tracking lines denote lines in the material that were initially lines defining material points with the same coordinates. They can be derived by subtracting the total displacements from the current location of the material points. This new variable can be easily depicted in isolines. In this figure it can be recognized that the material folds along the tool surface. Furthermore it can be seen that at the contact surface slip occurs. With this amount of friction also a neutral plane arises.

The existence of a neutral plane will have its effect on the deformation beha-
behavior of the ring. Therefore the simulated effective strain is depicted in figure 10.

![Figure 10. Simulated effective strain distribution.](image)

In the contact area at the upper side of the neutral plane stick occurs. This leads to a region in which the material hardly deforms, a dead metal zone. In the corner areas lines of increased effective strain originate at angles of about 45 degrees. This is similar to the results of the simple upsetting simulation in figure 6.

7 CONCLUSIONS

The finite element implementation of the physically based material model in the finite element code DIEKA has been performed well. The model does not show any numerical instabilities and the results of the simulations of compression tests are quite good, however no experimental data is available yet to compare with. Together with the mixed algorithm the used finite element code is well capable of simulating several forming processes.

REFERENCES


