A finite strain framework for steady-state problems
Hyperelasto-viscoplasticity

Juul, K. J.; Nielsen, K. L.; Niordson, C. F.

Published in:
Computer Methods in Applied Mechanics and Engineering

Link to article, DOI:
10.1016/j.cma.2020.113598

Publication date:
2021

Document Version
Peer reviewed version

Citation (APA):
A finite strain framework for steady-state problems: Hyperelasto-viscoplasticity

K. J. Juul*, K. L. Nielsen, C. F. Niordson

Department of Mechanical Engineering, Solid Mechanics, Technical University of Denmark, DK-2800 Kgs. Lyngby, Denmark

Abstract

A numerical framework for analyzing steady-state elastic-plastic material deformation at finite strains is developed and demonstrated in the present work. The framework is an extension of the original method by Dean and Hutchinson [Dean and Hutchinson, 1980. Quasi-static steady crack growth in small-scale yielding. Fracture Mechanics: Twelfth Conference, ASTM STP700, American Society for Testing and Materials, 383-405] develop to analyze steady-state crack propagation, under a small strain assumption, in which the history-dependent material response is captured through streamline integration. Steady-state problems are encountered in numerous engineering processes, and the original studies of crack growth have already been extended to include rolling and drawing, though within a small strain framework. However, the investigation of such manufacturing processes, where strains greater than 10% easily develop, requires a finite strain formulation to provide accurate results. The framework proposed in the present work offers an efficient and accurate method to extract the steady-state solution at finite strains without encountering the numerical issues related to traditional Lagrangian procedures. Furthermore, the framework also accounts for elastic unloading compared to many existing numerical steady-state schemes as they are often restricted to rigid plasticity. The new numerical framework employs a hyperelastic material model, in terms of a Neo-Hookean material, combined with an isotropic viscoplastic material behavior. However, the framework is not limited to any specific hyperelastic material model, nor any specific model

*Corresponding author

Email address: krjoju@mek.dtu.dk (K. J. Juul)
for the plastic behavior. The finite strain steady-state framework has been verified through comparison to a traditional Lagrangian analysis conducted in ANSYS. The benchmark case constitutes a plane strain drawing process where the thickness of the specimen is reduced by drawing it between two circular cylindrical tools.

Keywords: Steady-state, Finite strain, Hyperelasticity, Rate-dependency

1. Introduction

A large range of material deformation processes develop a steady-state condition, but this essential property is rarely exploited in numerical analysis. Traditional Lagrangian procedures are employed more frequently and the steady-state solution is approached by time integration through a transient phase, which makes the approach time-consuming, whilst numerical difficulties can arise in the search for the steady-state solution.

Several finite strain frameworks, that exploit the nature of steady-state deformation processes, have been suggested over the years in the search of an efficient numerical tool for the investigation of continuous manufacturing processes. The finite strain steady-state frameworks have different advantages and disadvantages depending on the intended use. Most commonly the procedures rely on Eulerian, or Arbitrary Lagrangian-Eulerian (ALE), velocity-based formulations (see e.g. Zienkiewicz et al., 1978; Liu et al., 1988; Strenkowski and Moon, 1990), and the majority of the models are restricted to either rigid-plastic or visco-plastic material models which ignore, or approximate the elastic effects. In general, this is sufficient to analyze manufacturing processes including hot working of metals, but the elastic effects may have a significant influence in terms of, for example, material spring back and residual stresses when considering cold working. Early “elastic re-analysis” methods was suggested by Zienkiewicz et al. (1978) to capture such effects. However, the first type of framework to include the elastic effects directly in the steady-state formulation was suggested by Dean and Hutchinson (1980) (and shortly after by Parks et al., 1981; Nguyen and Rahimian, 1981), who investigated steady-state crack propagation in elastic-plastic isotropic materials. Unfortunately, this pioneering framework is restricted to small strains which is insufficient in the study of most manufacturing processes as strains can easily reach 30-40% or more (see e.g. Richelsen, 1991; Rout et al., 2016; Kumar et al., 2017; Liu et al., 2018). The framework presented by Dean
and Hutchinson (1980) is widely used due to its efficiency (compared to Lagrangian procedures) and has been extended to several material models for steady-state problems in which elastic effects are important (Wei and Hutchinson, 1997; Mataga et al., 1987; Wei and Hutchinson, 1999; Nielsen et al., 2012a; Nielsen and Niordson, 2012b; Juul et al., 2017).

A later steady-state finite strain formulation, that directly accounts for the elastic material response, was presented by Balagangadhar et al. (1999); Balagangadhar and Tortorelli (2000); Yu (2005). This framework has both Eulerian and Lagrangian characteristics, but, as opposed to the early models, the formulation is displacement-based (similarly to the framework by Dean and Hutchinson, 1980). In contrast to the small strain formulation by Dean and Hutchinson (1980), determining the steady-state solution through a spatial integration technique, the framework by Balagangadhar et al. (1999) relies on a mixed formulation, including the history-dependent variables, through a transport equation for the material flow, as additional degrees of freedom in the equation system. This makes the existing displacement-based finite strain steady-state frameworks rather complex and the necessary factorization and solving of multiple, or coupled, equation systems are expected to put a high demand on the employed solver for an efficient analysis. Particularly, this is expected to become a challenge, in terms of computational resources, in the case of a full 3D analysis. The framework by Balagangadhar et al. (1999), however, guarantees convergence in contrast to the procedure by Dean and Hutchinson (1980) adopted in this study.

The main goal of the present study is to extend the original work by Dean and Hutchinson (1980) to finite strains, providing a simple tool for steady-state analysis of problems involving finite strains that is readily implemented into existing numerical software packages. In fact, an existing hyperelastic displacement-based Lagrangian model can easily be extended to the new steady-state framework put forward in the present work. The key change is to switch from an incremental procedure to streamline integration. Moreover, the new framework (along with that of Dean and Hutchinson, 1980) lends itself nicely to parallel computing as the integration along the individual streamlines are performed independently and extends readily to full 3D. The paper focuses on the derivation of the necessary procedures and the development of a numerical framework, followed by a comparison between the developed finite strain steady-state framework and a traditional Lagrangian method in order to verify the model. As for the framework by Dean and Hutchinson (1980), convergence in the proposed iterative procedure cannot
be guarantied.

The paper is divided into the following sections: The material model and the numerical formulation are presented in Section 2. The boundary value problem is stated in Section 3. Verification of the framework is presented in Section 4, and lastly the results are discussed in Section 5. Throughout the paper, the notation \((\cdot)\) denotes an incremental quantity.

2. Numerical Framework

In the development of the finite strain steady-state framework, which seeks to include the effect of elastic unloading, two distinct ways can be chosen to deal with the elastic part of the problem. The choice is between a hypoelastic or a hyperelastic material formulation. The hyperelastic material model is chosen in the present work as it can be derived directly from a strain energy density function such that the field equations can be expressed on total form in contrast to the hypoelastic model which is only valid on incremental form. The possibility to express the field equations on their total form is an important feature when establishing the steady-state framework because the solution method is based on an iterative scheme rather than an incremental scheme. In the following, the hyperelastic material description is adopted from Bower (2009).

In the finite strain formulation, the deformation is described through the total deformation gradient; \(F_{ij} = \frac{\partial u_i}{\partial x_j^0} + \delta_{ij}\), where \(u_i\) are the displacements, \(x_i^0\) are the reference coordinates and \(\delta_{ij}\) is Kronecker’s delta. As the material undergoes both elastic and plastic deformation, the total deformation gradient is decomposed into an elastic and a plastic part, according to \(F_{ij} = F_{ik}^e F_{kj}^p\) (multiplicative split), where the superscripts “\(e\)” and “\(p\)” denote the elastic and plastic part, respectively. The multiplicative split of the deformation gradient relies on the assumption that the deformation of the body can be divided into three configurations (see Fig. 1). The first is the reference configuration, \(x_i^0\), the second is an intermediate stress-free configuration which, in general, is incompatible, defined by the plastic deformation gradient, \(F_{ij}^p\), and the third is the current configuration, \(x_i\), defined by the total deformation gradient, \(F_{ij}\) (the absence of a superscript denotes the current configuration and “\(0\)” denotes the reference configuration).
2.1. Material model

The Kirchhoff stresses, $\tau_{ij}$, are here described by a Neo-Hookean material model (the framework is general and holds for any relation) according to the elastic strain energy density function

$$ W = \frac{\mu_1}{2} \left( B^e_{kk}/J^{2/3} - 3 \right) + \frac{K_1}{2} (J - 1)^2, $$

where $\mu_1$ is the shear modulus, $K_1$ is the bulk modulus, $B^e_{ij} = F_{ik}^e F_{jk}^e$ is the elastic left Cauchy-Green tensor, and $J = \text{det}(F^e)$ is the Jacobian ($J$ only depends on $F^e$ due to plastic incompressibility). When the material model has been chosen, the Kirchhoff stresses are easily found for the hyperelastic material as they can be determined directly from the strain energy density function in Eq. (1), such that

$$ \tau_{ij} = F_{ik} \frac{\partial W}{\partial F_{jk}} = \frac{\mu_1}{J^{2/3}} \left( B^e_{ij} - \frac{1}{3} B^e_{kk} \delta_{ij} \right) + K_1 (J - 1) J \delta_{ij}. $$

It is noticed that the Kirchhoff stresses only depend on the elastic part of the deformation gradient, hence the superscript “e”. Subsequently, the Cauchy stress can be determined directly from the relation $\tau_{ij} = J \sigma_{ij}$.

The elastic part of the deformation gradient is determined through the multiplicative decomposition when the plastic part is known. The plastic part of the deformation gradient is provided by the rate form;

$$ \dot{F}^p_{ij} = (F^e)^{-1}_{ik} L^p_{kl} F^e_{lm} F^p_{mj}, $$

where $L^p_{ij}$ is the plastic part of the velocity gradient ($L_{ij} = L^e_{ij} + L^p_{ij}$). The velocity gradient is determined through its connection to the stretch rate $D_{ij} = (L_{ij} + L_{ji})/2$ and the spin $W_{ij} = (L_{ij} - L_{ji})/2$. For the present study, employing an isotropic plasticity model, the plastic spin is not accounted for, $W^p_{ij} = 0$, ($W_{ij} = W^e_{ij} + W^p_{ij}$) which means that $L^p_{ij}$ is symmetric and hence $L^p_{ij} = D^p_{ij}$ (applying the additive split; $D_{ij} = D^e_{ij} + D^p_{ij}$). The plastic stretch, for the viscoplastic material applied in the present study, takes the form;

$$ D^p_{ij} = \dot{\varepsilon}^0 \frac{3 \tau^i_{j} \left( \frac{\tau_e}{g[\varepsilon^p_{acc}]} \right)^{1/m}}{2 \tau_e}, $$

where $\dot{\varepsilon}^0$ is the reference strain rate, $\tau^i_{j} = \tau_{ij} - \tau_{kk} \delta_{ij}/3$ is the deviatoric...
Kirchhoff stress, \( \tau_e = \sqrt{3\tau'_{ij}\tau'_{ij}/2} \) is the effective Kirchhoff stress, \( m \) is the rate-sensitivity exponent, and \( g[\varepsilon_p^{acc}] \) is a yield resistance function. The yield resistance function is in the present study chosen to represent linear hardening; \( g[\varepsilon_p^{acc}] = \sigma_y + h\varepsilon_p^{acc} \), where \( h \) is the hardening modulus and the accumulated plastic strain is obtained through: \( \varepsilon_p^{acc} = \int \dot{\varepsilon}_0(\tau_e/g[\varepsilon_p^{acc}])^{1/m} dt \).

2.2. Steady-state framework

Processes with steady and continuous material flow, such as steady crack growth, rolling, drawing, etc., usually develop a stationary field solution in a frame that translates with the crack tip or tool. The steady-state condition for continuous material flow is characterized by the field quantities appearing as constant to an observer following the translating frame. The steady-state relation states that any incremental quantity, \( \dot{f} \), can be related to a corresponding spatial derived quantity and velocity. In order to establish the necessary steady-state relations between velocity, the spatial and incremental quantities consider a coordinate system \( x \) which is fixed relative to the tool (see Fig. 2). Far upstream, a set of material points with initial coordinates \( x[t = 0] = x^0 \) translates at a defined feed velocity relative to the tool, \( v^0 \), in the negative \( x_1 \)-direction. The coordinates for these material points in the current configuration can be expressed in terms of the total displacement vector, \( u_t[x^0, t] \), as

\[
x[x^0, t] = x^0 + u_t[x^0, t].
\]  
(5)

A feed velocity vector may be defined as \( v^0 = \{-v^0, 0\}^T \), which allows rewriting the coordinates in the current configuration as

\[
x[x^0, t] = x^0 + v^0 t + u[x^0 + v^0 t, t]
\]  
(6)

where \( u[x^0 + v^0 t, t] = u_t[x^0, t] - v^0 t \) is the displacement relative to the position of the material point had the material points undergone only the translation due to the velocity \( v^0 \). A steady-state reference coordinate system may conveniently be defined by \( \xi = x^0 + v^0 t \) such that the coordinates in the current configuration are given by

\[
x[\xi, t] = \xi + u[\xi, t].
\]  
(7)

At steady-state, the solution has no explicit dependence on time and the above relation reduces to; \( x[\xi] = \xi + u[\xi] \). Hence, the deformation gradient
is readily found to be

\[ F = \frac{\partial x}{\partial x^0} = \frac{\partial x}{\partial \xi} \frac{\partial \xi}{\partial x^0} = \frac{\partial \xi}{\partial \xi} = I + \frac{\partial u}{\partial \xi}. \]  

(8)

The velocity field is determined from the feed velocity \( v^0 \) according to

\[ \mathbf{v} = \{-v_1, -v_2\}^T = \frac{\partial x}{\partial t} = \left( \frac{\partial \xi}{\partial t} + \frac{\partial u}{\partial \xi} \frac{\partial \xi}{\partial t} \right) = \left( I + \frac{\partial u}{\partial \xi} \right) \frac{\partial \xi}{\partial t} = F v^0 \]  

(9)

where the change in sign of the velocity components is conveniently introduced analogously to the feed velocity, as material flows in the negative \( x_1 \)-direction.

Under steady-state conditions, the increment of any field quantity may now be expressed in terms of its velocity and its spatial derivative either in the deformed configuration or in the steady-state reference coordinate system as follows

\[ \dot{f} = \frac{\partial f}{\partial x} \mathbf{v} = \frac{\partial f}{\partial \xi} v^0. \]  

(10)

Hence, for a feed velocity, \( v^0 \), along the negative \( x_1 \)-direction the steady-state relation reduces to

\[ \dot{f} = -v^0 \frac{\partial f}{\partial \xi_1}. \]  

(11)

From this condition, the steady-state relation states that any incremental quantity, \( \dot{f} \), can be related to a corresponding spatial derived quantity, scaling with the material feed velocity. By adopting this steady-state relation, any rate quantities in the constitutive model can be transformed into spatial gradients, and spatial integration along streamlines can be applied to obtain the deformation history (see Fig. 2). Thus, the field quantities at a specific material point, \( \xi^*_1 \), can be determined by integrating along a streamline starting upstream in the undeformed region \( \xi^0_1 \) and ending downstream at the desired point \( \xi^*_1 \). In this way, the point of interest, \( \xi^*_1 \), contains the load history of all upstream points i.e., the history of all previous states of the material point.

In the finite strain formulation, with the viscoplastic material model presented in Section 2.1, the plastic deformation gradient, \( F_{ij}^p \), is spatially inte-
grated to account for the history-dependence of a given problem. The pseudo
algorithm for the finite strain steady-state procedure is as follows, where “n”
refers to the iteration number:

1. The external load is defined. For complicated problems load stepping
   may be needed to ease convergence.

2. The equilibrium iterations are initiated.
   (a) The plastic deformation gradient from the previous iteration, \( F^{p(n-1)}_{ij} \),
       is used to determine the current displacement field, \( u^{(n)}_i \), from the
       principle of virtual work in Eq. (17).
   (b) The total deformation gradient, \( F^{(n)}_{ij} \), is determined based on the
       displacement field, \( u^{(n)}_i \).
   (c) The plastic strain field is determined by the streamline integration
       procedure.
      i. First the spatial derivative of the plastic deformation gradient
         is determined by applying the steady-state relation \( \frac{\partial f}{\partial \xi_1} = -\dot{f}/v^0 \) to Eq. (3):

         \[
         \frac{\partial F^p_{ij}}{\partial \xi_1} = -\frac{1}{v^0} (F^e)^{-1} L_{kl}^p F^e_{im} F^p_{mj}
         \]  

         (12)
     ii. Secondly, the total plastic deformation gradient, \( F^{p(n)}_{ij} \), is deter-
         mined from the spatial derivative by performing the stream-
         line integration

         \[
         F^{p(n)}_{ij} = \int_{X^0}^{X^*} \frac{\partial F^p_{ij}}{\partial \xi_1} \, d\xi_1
         \]  

         (13)
   (d) The current elastic deformation gradient is determined from the
       relation; \( F^e_{ij} = F_{ik} (F^p)^{-1}_{kj} \).
   (e) The current Kirchhoff stress field, \( \tau^{(n)}_{ij} \), is determined by Eq. (2).
   (f) Steps (a) through (e) are repeated by feeding the newly found
       plastic deformation gradient into the virtual work principle in Eq.
       (17) until convergence is obtained.

3. Steps 1 and 2 are repeated until the desired load is reached.
The outer loop of the Newton-Raphson procedure, subdividing the final load into steps, may be necessary for large/complicated deformation patterns to provide numerical stability in order for the algorithm to converge (this is a numerical trick and is not required by the theory). In fact, the need for incrementing the prescribed deformation is directly linked to slow, or absent, convergence related to the adopted approach from Dean and Hutchinson (1980). However, the number of load steps (if more than one) is usually low, and it does not affect the solution to the steady-state problem. Essentially, the load steps just provide a better starting guess in terms of the plastic deformation field for the next load step. The iterative procedure is initiated by using the purely elastic solution to the problem. The advantages of the steady-state framework lie in that; i) moving contact interfaces are avoided, ii) the mesh only needs to be focused in specific regions of interest, iii) the transient phase of the problem is avoided, and iv) the framework lends itself nicely to parallel computing.

The numerical stability of the steady-state algorithm, especially for low rate-sensitivity exponents, $m$, has been improved by introducing changes to the original streamline integration procedure by Dean and Hutchinson (1980). The changes follow the suggestion by Niordson (2001) and Nielsen and Niordson (2012b), where subincrement between Gauss points are introduced in the streamline procedure. Furthermore, the number of subincrements are implemented dynamically such that the number of subincrements increases in regions with steep gradients (and vice versa). As the deformation gradient is only known in the Gauss points it is necessary to interpolate the deformation gradient at each subincrement. Here, it is chosen to use linear interpolation between the Gauss points to determine the deformation gradient at each subincrement.

### 2.3. Streamlines

The streamline integration procedure is a cornerstone in the steady-state framework developed by Dean and Hutchinson (1980). Assuming small strains, the streamlines are easily identified as the prescribed (known) material flow direction in the domain remains unchanged during deformation. This is not as simple when accounting for finite strains because the domain before and after deformation might change substantially, altering the position of the streamlines and, moreover, the local velocity of the material changes as well.
To cope with this, it is here argued that by adopting a total Lagrangian formulation, the streamline integration can be conducted in the reference configuration where the position of the streamlines is known. The argument justifying this statement starts with the definition of a streamline parametrized by the vector \( \mathbf{x} = \mathbf{x}(s) \), which represents the trajectory of a material point at a specific moment in time (not changing at steady-state). Here, an index free notation is adopted for a compact presentation. A streamline, or material flow curve, is defined by being parallel to the material velocity, \( \mathbf{v} \), such that the spatial gradient of the streamline must fulfill

\[
\frac{d\mathbf{x}}{ds} \times \mathbf{v} = 0 \quad (14)
\]

Secondly, the velocity field is obtained from the feed velocity through the deformation gradient. By inserting Eq. (9), into the definition of a streamline, Eq. (14), followed by applying the chain rule for differentiation, the following expression is obtained

\[
\left( \frac{d\mathbf{x}}{d\mathbf{x}^0} \frac{ds^0}{ds} \right) \times (\mathbf{Fv}^0) = 0 \quad (15)
\]

where \( \frac{d\mathbf{x}}{d\mathbf{x}^0} \) may be recognized as the deformation gradient. By using the identity; \( (\mathbf{Ma}) \times (\mathbf{Mb}) = \det(\mathbf{M}) (\mathbf{M}^{-1})^\top (\mathbf{a} \times \mathbf{b}) \), Eq. (15) is rewritten into the form

\[
\frac{ds^0}{ds} \det(\mathbf{F})(\mathbf{F}^{-1})^\top \left( \frac{d\mathbf{x}^0}{ds^0} \times \mathbf{v}^0 \right) = 0 \quad (16)
\]

where, in order to satisfy the definition of a streamline, the flow in the reference configuration must at all times be parallel to the original flow curves i.e. \( \frac{d\mathbf{x}^0}{ds^0} \times \mathbf{v}^0 = 0 \). Thus, with the integration performed in the reference configuration, where the position of the streamlines is known (see also Yu, 2005), the relation between incremental quantities and spatial derivatives is conveniently given by Eq. (11).

\[2.4\, \text{Principle of virtual work}\]

To enable the possibility of performing the spatial integration in the reference configuration, where the streamlines are known prior to the calculations (see Section 2.2), a total Lagrangian formulation is chosen. The principle of
virtual work to be discretized by the finite element method and subsequently used to determine the displacement field, \( u_i \), is expressed in terms of the Kirchhoff stresses, \( \tau_{ij} \), such that

\[
\int_{V_0} \tau_{ij} \delta L_{ij} \, dV_0 = \int_{S_0} t^0_i \delta v_i \, dS_0,
\]

where \( V_0 \) is the reference domain, \( S_0 \) is the boundary of the reference domain, \( v_i \) is the virtual velocity field (not to be confused with the material velocity), \( t^0_i \) is the nominal traction, and \( \delta \) denotes a virtual quantity. It is evident from the principle of virtual work that the contribution to the stiffness matrix is non-linear due to the dependence on the Kirchhoff stress, \( \tau_{ij} \). To deal with this non-linearity in a framework which is not based on increments, a Newton-Raphson scheme is employed, following the procedure suggested by Bower (2009) for a finite strain hyperelastic material model.

For an elastic problem, the Kirchhoff stresses, \( \tau_{ij} \), are determined directly from the total deformation and the stiffness tensor, \( C_{ijkl} \), is calculated according to

\[
C_{ijkl} = \frac{\partial \tau_{ij}}{\partial F_{km}} F_{lm}.
\]

However, when considering an elastic-plastic problem, the Kirchhoff stresses will only be dependent on the elastic deformation gradient, \( F^e_{ij} \), and by applying the chain rule to Eq. (18) the following expression is obtained:

\[
C_{ijkl} = \frac{\partial \tau_{ij}}{\partial F^p_{pq}} \frac{\partial F^e_{pq}}{\partial F_{km}} F_{lm}.
\]

exploiting that \( \partial \tau_{ij}/\partial F^p_{pq} = 0 \). The first part; \( \partial \tau_{ij}/\partial F^e_{pq} \), of Eq. (19) is determined directly from the chosen energy function (in this case a Neo-Hookean material), whereas the second part; \( \partial F^e_{pq}/\partial F_{km} \), is determined in a slightly different manner for the steady-state framework when compared to traditional hyperelastic formulations. The multiplicative split of the deformation gradient dictates that the elastic deformation gradient can be expressed as; \( F^e_{ij} = F_{ik}(F^p_{kj})^{-1} \), in terms of the total and the plastic deformation gradients. However, since the plastic deformation gradient is provided by the streamline integration procedure (see Section 2.2), it can be treated as known in the principle of virtual work and the derivative simplifies to;
\[ \frac{\partial F_e^{pq}}{\partial F_{km}} = \delta_{pk} (F_{mq}^p)^{-1} \]. Thus, the final expression for the stiffness tensor can be written as

\[ C_{ijkl} = \frac{\partial \tau_{ij}}{\partial F_{kq}^e} F_{lq}^e. \] (20)

2.5. Contact modelling

In traditional Lagrangian approaches, part of the difficulty in introducing a contact scheme is to keep track of the relative position between nodes at the contacting surfaces, as the node-sets constituting the contact pairs continuously changes. The difficulty contrasts the steady-state framework presented. At steady-state, the contacting surfaces remain stationary in the moving reference frame, and thus the contacting node-sets do not change. Thus, in a pre-processing phase, the node-set expected to make contact can be identified as a small subset, which will act as the maximum of possible contact nodes in the simulation. It is, however, still necessary to evaluate the status of each node in the contact set for each iteration.

In the following, a modeling scheme enforcing node-to-(rigid)-surface contact is considered, and proceeds as follows; i) in the first iteration, the subset of contact nodes are displaced according to the rigid surface to avoid overlap, and ii) in the first subsequent iterations, the contact reaction force between the rigid surface and the contacting node is evaluated. If a contact node has a contact force resulting from a negative contact pressure, then this node is released from the contact in the subsequent iterations. At the same time, the subset of contact nodes that are already released from the contact conditions is evaluated in terms of location to determine if the nodes are penetrating the rigid surface. In the case that a potential contact node is penetrating the rigid surface, the node will be added to the contact conditions and displaced to the rigid contact surface in the subsequent iterations. The contact model is enforced through a specialized procedure prescribing displacements along the contact boundary between the deforming material and the rigid tool. In the steady-state model, contact is enforced by prescribing the total displacement of the contact nodes in the \( x_2 \)-direction based on the current nodal \( x_1 \)-coordinates. This method is suitable for the assumed frictionless sliding, which enables a straightforward comparison to the ANSYS model. It should be noted that the contact condition is part of the iterative solution, and although the contact condition is not necessarily fulfilled in each iteration, the converged solution will satisfy the contact condition to any specified
Friction can be included and will not pose additional issues related to the steady-state modelling framework. In case of friction, an additional traction parallel to the movement should be added to contact interface depending on the direction of sliding relative to the tool. For details relating to the implementation of friction in the context of steady-state solutions please refer to (Nielsen et al., 2016; Nielsen, 2015).

3. Benchmark problem

A simplified drawing process in 2D under plane strain conditions (see Fig. 2) is chosen as the benchmark for comparison between results from using the new steady-state framework and the results from a traditional incremental method. The driving force for the deformation is a load applied to the left end of the domain, \( F_{\text{draw}} \), simulating that the material is drawn through the tool. Identical tools are placed on both sides of the material, imposed by a symmetry condition along the boundary \( x_2 = 0 \) (see Fig. 2). In the numerical analysis, the contact between the tool and the material is assumed friction-less, the dimensions are shown in Fig. 2, and the material parameters are stated in Tab. 1. It is chosen only to vary the strain hardening in the benchmark problem considered in an attempt to reduce the number of variables in play. Here, investigating a wide range of the strain hardening capacity such that the results reflect materials with more or less pronounced plastic behaviour.

A dimensionless feed velocity of \( \frac{v_0}{(\dot{\varepsilon}_0 H)} = 500 \) is chosen for the convenience of the convergence. However, the feed velocity has little effect on the results since the rate-sensitivity exponent, \( m \), is chosen sufficiently small in the visco-plastic formulation. It is worth to notice that the effect of the rate-sensitivity merit a future study as metal materials undergoing hot-forming processes can pose a higher \( m \)-value such that the influence of the velocity can be quite substantial (see influence in Eq. 12). Studies of the impact of velocity on manufacturing processes as well as crack propagation can be found in Nielsen and Niordson (2012b, 2015)

4. Results

The steady-state framework (hereafter denoted as “code”), developed throughout this study, is verified against a numerical solution obtained with
the commercial software ANSYS. The analysis in ANSYS is conducted by a traditional incremental scheme (Lagrangian), where it is necessary to integrate through a transient period in order to approach the steady-state solution. This model is built on a traditional hypoelasto-plastic material as ANSYS do not support hyperelastic-plastic model formulations. The difference in the results by using a hyper/hypo-elastic formulation is, however, expected to be small as the elastic strains are expected to be significantly smaller than 10%.

The verification of the steady-state framework is divided into two parts; i) is a qualitative comparison of the effective stress fields through contour plots to see if the fields are similar, and ii) is a quantitative comparison of the individual components of stresses, total strains, and the accumulated plastic strain along streamlines located at different positions in the domain.

The effective stress fields \( \sigma_e = \sqrt{3s_{ij}s_{ij}/2} \), where \( s_{ij} \) is the deviatoric part of the Cauchy stress, are presented in Fig. 3 for the steady-state code and for ANSYS at two different levels of linear hardening, \( h \). At both hardening levels, it is seen that the stress level downstream is largest just underneath the top surface, close to zero in the center, and then again rises in the bottom of the domain. In the region where the material is in contact with the tool, it is seen that the effective stress gradually increases when the material moves from the inlet to the outlet of the tool. When comparing the details of the effective stress field for \( h = E/20 \) (Fig. 3a) and \( h = E/80 \) (Fig. 3b) a good match between the code and ANSYS is generally observed, however, it is seen that the stresses are slightly larger in the vicinity of the tool for \( E/h = 20 \) in the ANSYS solution. This difference is investigated in the subsequent quantification of the results.

Results along four selected streamlines are presented in the following for a direct comparison of the magnitude and evolution of the field variables. The streamlines at which the data is extracted are spread through the domain such that one line is located in the bottom, \( x_2^0/H = 0.13 \), one line is located in the middle, \( x_2^0/H = 0.5 \), and two lines are located in the top part of the domain, \( x_2^0/H = 0.88 \) and \( x_2^0/H = 0.95 \), to allow for a more detailed representation in the region subjected to the most severe deformation (lines are with respect to the undeformed domain). For the comparison of the total strains, it is chosen to display the spatial Hencky strains, \( h_{ij} \), in order to match the output from ANSYS. The spatial Hencky strains are calculated from the left Cauchy green tensor, \( B_{ij} \), according to; \( h_{ij} = \log(B_{ij})/2 \). The strain measures are normalized by the yield strain, \( \varepsilon_y = \sigma_y/E \).
The stresses, total strains, and accumulated plastic strain are presented in Figs. 4-6 along a streamline close to the bottom of the domain \((x_2^0/H = 0.13)\) for \(h = E/20\) and \(h = E/80\), respectively. Here, it is clearly seen that results from ANSYS and the steady-state code match both in terms of stresses, strains, and the accumulated plastic strain. There is excellent quantitative correspondence even for the complex evolution of the field quantities when the material passes through the tool for both material hardening levels.

The same field quantities are extracted along a streamline in the middle of the domain \((x_2^0/H = 0.5)\) and are presented in Figs. 7-9. In the middle of the domain, the extracted quantities are again similar for the two finite element procedures, supporting that the steady-state code is capable of modeling the history-dependent material behavior in agreement with a traditional incremental procedure.

In Figs. 10-12, the field quantities are extracted along a streamline in the top part of the domain \((x_2^0/H = 0.88)\). Closer to the surface an even more complex evolution in the fields is predicted in the contact zone. For \(h = E/80\) (Figs. 10b-12b) the results fit well, as for the first two streamlines. However, for a small difference is observed in the \(\sigma_{11}\) component for \(h = E/20\) (Figs. 10a), while the strains fit well. The difference is mainly in the downstream material \((x_1^0/H < 0)\) and the \(\sigma_{11}\) component is approximately 4% lower for the code compared to the corresponding ANSYS prediction. To gain further insight into this deviation, the field quantities are subsequently extracted along a streamline just below the top surface of the domain \((x_2^0/H = 0.95)\). The results are shown in Figs. 13-15 and closely resembles that of the streamline located at \(x_2^0/H = 0.88\). Again, a small difference between the model predictions is observed in the \(\sigma_{11}\) component, but now for both \(h = E/20\) and \(h = E/80\), while all other components of both strains and stresses continue to fit well. Again, the main difference is observed in the downstream region \((x_1^0/H < 0)\), and the difference is approximately 6% (slightly smaller for \(h = E/80\)). The features of the evolution are, however, nicely tracked by the steady-state code.

5. Discussion

A numerical framework for finite strain steady-state problems with history-dependent material behavior has been developed. The framework readily includes elastic unloading as opposed to the majority of previous steady-state frameworks found in the literature. The framework is customized to
steady-state problems and eliminates a number of issues encountered when employing traditional Lagrangian procedures. The main advantage of the steady-state framework lies in the iterative solution scheme, determining the solution in a translating coordinate system. This advantage presents itself in terms of avoiding changing contact interfaces as well as the possibility for more efficient meshing as a fine discretization is only needed in the regions of interest. This opens the possibility for getting high-resolution results, but at the same time reduce the total number of elements making the computations faster. A detailed study of the benefits in terms of computational speed when applying the steady-state approach (in-house code) rather than the traditional incremental approach (represented by ANSYS) is omitted in the presented work, as the solvers and assembly procedures are very different in the two codes. Furthermore, the full potential of the steady-state framework is not utilized in the present study as an identical mesh is employed for both procedures, not taking advantage of the possibility for refining the mesh only in the region of interest in the steady-state approach.

The developed framework has been tested on a plane strain drawing problem and the solutions have been verified through comparison to corresponding solutions produced by the commercial software ANSYS that employs a traditional incremental (Lagrangian) method.

The comparison between the developed framework and ANSYS showed good agreement between the results both in terms of the predicted magnitudes, but also the evolution of both stresses and strains is captured as the material is deformed to finite strains.

A minor difference was observed for the $\sigma_{11}$ component for the results extracted close to the surface ($x_0^2/H = 0.88$ and $x_2^0/H = 0.95$). This can be connected to a number of differences in the two numerical models; i) the numerical solution procedure takes very different stating points, ii) the contact modeling approach employed in ANSYS is different than the one used in the steady-state code, which has a large impact on the material close to the surface and, iii) the hyperelastic formulation employed in the developed framework is slightly different from the linear hypoelastic model used in ANSYS and, thus, a small difference is introduced even though the elastic strains are significantly smaller than 10%.
6. Acknowledgement

The work is financially supported by The Danish Council for Independent Research in the project “New Advances in Steady-State Engineering Techniques”, grant DFF-4184-00122. Researcher Konstantinos Poulios, Ph.D, DTU Mechanical Engineering, is greatly acknowledged for fruitful discussions concerning the hyperelastic framework.

References


List of Figures

1 Multiplicative split of the deformation gradient. . . . . . . . . 22
2 Simplified drawing process under plane strain conditions. The finite element mesh consists of square elements with side length \( L_{\text{ele}} = 0.025H \), with \( H/(2L) = 1/10 \). The tool is chosen to be circular with a radius of \( R = 2H \) and the reduction will be \( \Delta = 0.2H \). The dashed domain shows the undeformed geometry in which the streamline integration is carried out along horizontal lines. . . . . . . . . . . . . . . . . . . . . . . . . . . 23
3 Normalized effective stress distribution, \( \sigma_e/\sigma_y \), in steady-state region for (a) \( h = E/20 \) and (b) \( h = E/80 \). . . . . . . . . . . . 24
4 Normalized stress components extracted along streamline at \( x_0^2/H = 0.13 \) for (a) \( h = E/20 \) and (b) \( h = E/80 \). The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. . . . . . . . . . . . 25
5 Normalized total Hencky strain components extracted along streamline at \( x_0^2/H = 0.13 \) for (a) \( h = E/20 \) and (b) \( h = E/80 \). The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. . . . . . . . . . . . 26
6 Normalized accumulated plastic strain extracted along streamline at \( x_0^2/H = 0.13 \) for (a) \( h = E/20 \) and (b) \( h = E/80 \). The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. . . . . . . . . . . . 27
7 Normalized stress components extracted along streamline at \( x_0^2/H = 0.5 \) for (a) \( h = E/20 \) and (b) \( h = E/80 \). The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. . . . . . . . . . . . 28
8 Normalized total Hencky strain components extracted along streamline at \( x_0^2/H = 0.5 \) for (a) \( h = E/20 \) and (b) \( h = E/80 \). The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. . . . . . . . . . . . 29
9 Normalized accumulated plastic strain extracted along streamline at \( x_0^2/H = 0.5 \) for (a) \( h = E/20 \) and (b) \( h = E/80 \). The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code. . . . . . . . . . . . 30
Normalized stress components extracted along streamline at $x_2^0/H = 0.88$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. .......................... 31

Normalized total Hencky strain components extracted along streamline at $x_2^0/H = 0.88$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. .......................... 32

Normalized accumulated plastic strain extracted along streamline at $x_2^0/H = 0.88$ for (a) $h = E/20$ and (b) $h = E/80$. The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code. .......................... 33

Normalized stress components extracted along streamline at $x_2^0/H = 0.95$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. .......................... 34

Normalized total Hencky strain components extracted along streamline at $x_2^0/H = 0.95$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code. .......................... 35

Normalized accumulated plastic strain extracted along streamline at $x_2^0/H = 0.95$ for (a) $h = E/20$ and (b) $h = E/80$. The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code. .......................... 36
Figure 1: Multiplicative split of the deformation gradient.
Figure 2: Simplified drawing process under plane strain conditions. The finite element mesh consists of square elements with side length $L_{\text{ele}} = 0.025H$, with $H/(2L) = 1/10$. The tool is chosen to be circular with a radius of $R = 2H$ and the reduction will be $\Delta = 0.2H$. The dashed domain shows the undeformed geometry in which the streamline integration is carried out along horizontal lines.
Figure 3: Normalized effective stress distribution, $\sigma_e/\sigma_y$, in steady-state region for (a) $h = E/20$ and (b) $h = E/80$. 
Figure 4: Normalized stress components extracted along streamline at $x_2^0/H = 0.13$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 5: Normalized total Hencky strain components extracted along streamline at $x_0^2/H = 0.13$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 6: Normalized accumulated plastic strain extracted along streamline at $x_2^0/H = 0.13$ for (a) $h = E/20$ and (b) $h = E/80$. The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 7: Normalized stress components extracted along streamline at $x_2^0/H = 0.5$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 8: Normalized total hencky strain components extracted along streamline at $x_2^0/H = 0.5$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 9: Normalized accumulated plastic strain extracted along streamline at $x_2^0/H = 0.5$ for (a) $h = E/20$ and (b) $h = E/80$. The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 10: Normalized stress components extracted along streamline at $x_2/H = 0.88$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 11: Normalized total Hencky strain components extracted along streamline at $x_0^2/H = 0.88$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 12: Normalized accumulated plastic strain extracted along streamline at $x_2^0/H = 0.88$ for (a) $h = E/20$ and (b) $h = E/80$. The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 13: Normalized stress components extracted along streamline at $x_2^0/H = 0.95$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 14: Normalized total Hencky strain components extracted along streamline at $x_2^0/H = 0.95$ for (a) $h = E/20$ and (b) $h = E/80$. The lines represent the solution obtained by ANSYS and the markers represent the developed steady-state code.
Figure 15: Normalized accumulated plastic strain extracted along streamline at $x_2^0/H = 0.95$ for (a) $h = E/20$ and (b) $h = E/80$. The line represents the solution obtained by ANSYS and the markers represent the developed steady-state code.
List of Tables

1 Model parameters ........................................... 38
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Significance</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_y/E$</td>
<td>Yield strain ($\varepsilon_y$)</td>
<td>0.001</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Poisson’s ratio</td>
<td>0.30</td>
</tr>
<tr>
<td>$E/h$</td>
<td>Linear Hardening</td>
<td>20-80</td>
</tr>
<tr>
<td>$m$</td>
<td>Strain rate hardening exponent</td>
<td>0.02</td>
</tr>
<tr>
<td>$\dot{\varepsilon}_0$</td>
<td>Reference strain rate</td>
<td>0.001</td>
</tr>
<tr>
<td>$v_0/(\dot{\varepsilon}_0H)$</td>
<td>Feed velocity</td>
<td>500</td>
</tr>
</tbody>
</table>

**Table 1:** Model parameters