Springback Simulation: Impact of Some Advanced Constitutive Models and Numerical Parameters

Badis Haddag, Tudor Balan, Farid Abed-Meraim

LPMM, ENSAM de Metz, 4 rue A. Fresnel, 57078 Metz Cedex 03, France

Abstract. The impact of material models on the numerical simulation of springback is investigated. The study is focused on the strain-path sensitivity of two hardening models. While both models predict the Bauschinger effect, their response in the transient zone after a strain-path change is fairly different. Their respective predictions are compared in terms of sequential test response and of strip-drawing springback. For this purpose, an accurate and general time integration algorithm has been developed and implemented in the Abaqus code. The impact of several numerical parameters is also studied in order to assess the overall accuracy of the finite element prediction. For some test geometries, both material and numerical parameters are shown to clearly influence the springback behavior at a large extent. Moreover, a general trend cannot always be extracted, thus justifying the need for the finite element simulation of the stamping process.

INTRODUCTION

The accuracy of springback prediction is an active investigation field. The interference of material and friction models and parameters is recognized as a major difficulty to clearly assess their respective accuracy. For well posed frictionless problems, the material model has been shown to affect springback to a large extent [1]. This material effect is exacerbated for the new material sheets used e.g. in the automotive industry: aluminum alloys, very high strength steels etc.

Several authors have clearly emphasized the impact of the yield function on the springback prediction. On the other hand, the hardening model does not always have the same influence [2]. Our aim is thus to evaluate the impact of recent hardening models on the springback prediction, for the well-known strip drawing test. Accordingly, several anisotropic elastic-plastic models have been implemented into the finite element code Abaqus. A general, implicit time integration algorithm has been developed in order to guarantee an accurate material model implementation, even in conditions of drastic strain-path changes. Two steel sheets with different behavior and thickness are considered; also, two sheet-die-punch geometries and several blank holding forces are simulated.

CONSTITUTIVE MODELING

A material model based on rate equations must respect the principle of objectivity. In finite element implementations, the most commonly used technique is to integrate the rate equations in a frame that rotates according to the spin $W$ (skew-symmetric part of the velocity gradient). This is equivalent to the use of a Jaumann-type rate, yet the equations obtained are form-identical to a small strain formulation [3]. Consequently, all the tensor variables below are rotation-compensated with respect to this frame.

Constitutive Model Framework

We consider elastic-plastic, anisotropic materials, defined by the following equations:

Yield function:

$$F(\mathbf{\sigma}, X, R) = \bar{\mathbf{\sigma}}(\mathbf{\sigma}' - X) - R - Y_0,$$

(1)

where $\mathbf{\sigma}$ is the stress tensor, $\mathbf{\sigma}'$ is its deviator, $X$ and $R$ are internal variables describing the current position and size of the yield surface, respectively. The
material becomes plastic when the yield function vanishes. \( Y_0 \) is the initial size of the yield surface. In this work, only quadratic yield functions are considered; nevertheless, the modeling formalism is much more general.

**Flow rule**: we consider associated plasticity:

\[
D^p = \dot{\lambda} V ; \quad V = \frac{\partial F}{\partial \sigma},
\]

where \( D^p \) is the plastic strain rate, \( V \) is the gradient of the yield function and \( \dot{\lambda} \) is the plastic multiplier.

**Hardening**, governed by generic rate equations:

\[
\dot{R} = h_R \dot{\lambda} ; \quad \dot{X} = h_X \dot{\lambda},
\]

with the initial conditions \( R(0) = 0 \) and \( X(0) = 0 \).

**Hypo-elastic law**:

\[
\dot{\sigma} = C : (D - D^p),
\]

where \( C \) is the fourth order tensor of elastic constants and \( D \) is the total strain rate.

**Tangent modulus**: The plastic multiplier is computed using the consistency condition \( F = 0 \). Finally, the elastic-plastic analytical tangent modulus can be derived in the following form, independent of the yield surface definition and the hardening model:

\[
L = C - \alpha \frac{(C : V) \otimes (V : C)}{V : C : V + V : h_X + h_R},
\]

where \( \alpha \) equals 1 for plastic loading and 0 otherwise.

By now, the hardening is simply defined by the functions \( h_R \) and \( h_X \). Most of the available hardening models fall into this framework, as we shall see hereafter.

### Classical Cyclic Hardening Model

One of the most used hardening models is the combined saturating hardening, initially proposed by Armstrong and Frederick [4]. The great success of this model is due to its ability to describe accurately various materials undergoing cyclic loading at small strains. The corresponding hardening functions are simply defined as:

\[
h_R = C_R (R_{sat} - R) \quad \text{and} \quad h_X = C_X (X_{sat} N - X)
\]

where \( C_R, R_{sat}, C_X \) and \( X_{sat} \) are material constants and \( N = [D^p][D^p] \) is the plastic strain rate direction. This model is available in most of the commercial finite element codes and thus can be used for validation purposes.

### Microstructural Model of Teodosiu

This model has been proposed [5] and further investigated [6,2] by Teodosiu and co-workers. Two new physically-based internal variables are introduced: the fourth order tensor \( S \) related to the directional strength of the persistent planar dislocation structures (PPDS) and the second order tensor \( P \) related to the polarity of these structures. The variable \( S \) is further decomposed in two parts: a scalar part \( S_{DD}^{sat} \) and the latent structures. Saturating rate equations are postulated for all these variables:

\[
\dot{S}_{DD} = C_{SD} \left[ g(S_{sat} - S_D) - hS_D \right] \dot{\lambda} = h_{SD} \dot{\lambda},
\]

\[
\dot{S}_{LL} = -C_{SL} \left[ hS_L \right] \dot{\lambda} = h_{SL} \dot{\lambda},
\]

\[
\dot{P} = C_p (N - P) \dot{\lambda}.
\]

New material parameters are introduced: \( C_{SD}, C_{SL}, C_P, S_{sat}, n \), while \( g \) and \( h \) are functions of the internal variables. The size of the yield surface is given by \( Y = Y_0 + R_V + f |S| \) where \( f \) is also a material parameter. \( R_V \) and \( X \) have evolution laws of type (3), similar to the classical hardening variables:

\[
h_{RF} = C_R (R_{sat} - R) ; \quad h_X = C_X (X_{sat} \frac{\sigma - X}{} - X)
\]

Nevertheless, \( X_{sat} \) is no more a constant but it depends on \( S \). More details can be found in the original papers. We shall simply note here that this model can be rearranged in the generic form (3), with:

\[
h_R = \frac{f}{|S|} \left[ S_{DD} h_{SD} - |S_L| h_{SL} \right] + h_{RF}.
\]

### Numerical Implementation

The time integration algorithm has been recognized as an important source of error in the numerical simulation of forming. Although approximate,
accurate integration schemes are available in the literature. For the purpose of forming simulations, we developed several explicit type integration schemes for the considered models and implemented them in Abaqus/Explicit, Abaqus/Standard and PamStamp [7]. Nevertheless, the accuracy required for springback simulations is much higher than the one regularly required to predict the punch force, draw-in etc. Thus, a fully implicit integration scheme has been recently developed, based on the generalized mid-point method [8]. The hardening rate equations are integrated either by a Backward Euler scheme or a semi-analytical scheme. The results in this paper are obtained with the semi-analytical integration method. The simulation code used is the static implicit code Abaqus/Standard (model implementation via the UMAT routine).

Figure 1 validates the implementation with respect to the classical model available in Abaqus and underlines the accuracy of the integration scheme even for large strain increments.

**STRIP DRAWING SIMULATIONS**

The strip drawing test (see figure 2) is recognized as a reference benchmark test both for the experimental investigation of metal sheets and for the validation of numerical simulations. Various test geometries are cited in the literature. Some of the most important geometrical characteristics of the test are the ratios \( L/W \), \( D/W \), \( T/R \) – where \( W \) is the punch width, \( L \) is the initial strip length, \( D \) is the drawing stroke, \( T \) is the sheet thickness and \( R \) is the punch/die radius. Not only springback is “larger” for larger values of these ratios, but also the simulation is more challenging and more sensitive to the modeling parameters. We shall investigate two test geometries in this paper; the geometrical parameters of the two test geometries are given in the following table.

<table>
<thead>
<tr>
<th>Dimensions</th>
<th>Test 1 (“smooth”)</th>
<th>Test 2 (“sharp”)</th>
</tr>
</thead>
<tbody>
<tr>
<td>( L/W )</td>
<td>4.4</td>
<td>6.4</td>
</tr>
<tr>
<td>( D/W )</td>
<td>0.9</td>
<td>1.7</td>
</tr>
<tr>
<td>( T/R )</td>
<td>0.2</td>
<td>0.23 and 0.4</td>
</tr>
</tbody>
</table>

The numerical simulation of both the drawing and the springback step is performed with the static implicit code Abaqus/Standard. Given the small radii/thickness ratios, the sheet is modeled with solid elements for a better accuracy. For computing time convenience, all the simulations are performed using plane strain solid elements. This choice is sufficient for comparison purposes; it is noteworthy that for a more rigorous simulation, three-dimensional effects should also be taken into account. Also, the tools are simply modeled using rigid surfaces.

**Materials**

Two sheet steels are considered for the investigation of springback: a mild steel (DC06) and a dual phase steel (DP600). The materials, as well as the test geometries, have been provided by the steelmakers Arcelor, France and Voestalpine, Austria, in the framework of a joint research project. The materials have been tested by the LPMTM laboratory, Paris XIII University – who also determined and provided the material parameters for both models considered in this investigation. This is a particularly important point in the comparison study since not only the material model, but also the mechanical tests and the parameter identification procedure have a considerable impact on the final simulation results.

The mild sheet steel was 0.68 mm thick while the dual phase sheet steel was 1.2 mm thick. On one hand, this difference does not allow simple comparison
between the two materials; on the other hand however, this provides us with four different $R/T$ ratios.

The parameter identification procedure and the required experimental tests as developed at LPMTM are described in the papers of Teodosiu et al. [6,9]. The tests include monotonic (tensile and shear) tests as well as several two-path sequential tests: Bauschinger reversed shear tests as well as orthogonal tension + shear tests. The parameters are identified in such a way that the monotonic predictions of both models are almost identical. Their predictions are nevertheless very different for the two types of sequential tests, as underlined in [9]. Figures 3 and 4 overview the ability of the two models to predict the transient behavior after abrupt strain-path change. Clearly, both models predict a Bauschinger effect. With the “classical” model, the monotonic hardening curve is rapidly resumed, while a much larger transient zone can be predicted by the microstructural model, in accordance with experimental observations. Similar differences can also be observed during the orthogonal tests; nevertheless, the springback behavior is expected to depend more on Bauschinger-type effects.

Clearly, the predictions of the two models differ under non-monotonic strain-paths, while they coincide under monotonic loading. Thus, their strain-path dependency will be the unique source of difference in the springback predictions.

**Springback of a “Smooth” Test Geometry**

Several physical as well as numerical factors have been identified to have a considerable influence on springback. Among the process-related factors, the most important seem to be: blank holding force, friction, test geometry. In what concerns the material description, the yield surface influence has always been clearly identified while the hardening model does not always affect the results.
We first use the "smoother" test geometry to investigate the relative impact of different categories of factors on the final result. Well identified factors like friction or yield function are not studied here; rather, a particular attention is given to numerical factors.

Figure 5 gives a view of the relative impact of some selected factors on the final geometry. The first two plots confirm some already well established trends. Indeed, increasing the blank holder force reduces the springback angles, while the hardening model has a reduced impact. The two following plots clearly indicate the weight that numerical factors may have on the final results accuracy. Besides the classical problem of the mesh refinement, the formulation of the finite element itself may affect the results as much as the material model – or more. Even more surprising, the modeling of the tools removal sequence affects the results in a decisive manner. The "proportional unloading" procedure – recommended for use with commercial software in order to avoid contact evolution problems – dramatically alters the result and should be avoided. It is noteworthy that special-purpose sheet forming software include adapted procedures that successfully overcome such contact difficulties [10].

Since this geometry is not very sensitive to the hardening model, it is not suited to compare the two material models under study. Consequently, we shall use the second test geometry for this purpose, since it exhibits "sharper" dimensions.

### Springback of a “Sharp” Test Geometry

For both materials, two different holding forces have been used in the simulations. All other numerical parameters have been kept identical (mesh, element type, friction coefficient, tools geometry etc.).

The part geometries after springback are displayed in figure 6. Among the four combinations (material-BHF), all but one exhibit an important impact of the hardening model. Moreover, there is no general trend: this effect may depend or not on the blank holding force and moreover one model can predict larger springback angles for some materials and smaller for other materials. Thus, at least when severe geometries are simulated, the use of an accurate hardening model is compulsory. For such test geometries, the impact of the finite element formulation is even bigger than for the former geometry. Also, the yield function is expected to strongly influence the results, as well as the three-dimensional heterogeneity of strain and stress.
CONCLUSIONS

In this work, a general and effective constitutive modeling framework has been adopted that covers up-to-date anisotropic elastic-plastic models. An accurate, implicit time integration algorithm has been developed for this category of models and implemented in Abaqus/Standard. By means of strip drawing tests simulation, the impact of the strain-path history modeling on the springback has been clearly demonstrated. This impact depends on the geometry of the tools and for simple geometries, simple material models can provide a globally satisfactory result. Nevertheless, for so-called “sharp” geometries, very accurate models are required. In the mean time, the numerical resolution and the finite element model have to be carefully assessed since several numerical factors are shown to have a considerable effect on the result. Under all these aspects, springback simulation is much more demanding than regular stamping process simulations. Up-to-date, physically-based material models seem to bring a satisfactory answer on the material modeling ground. The problem of the suited finite element formulation for sheet forming is still open and under intensive investigation.

ACKNOWLEDGEMENTS

This work has been carried out in a project funded by the European Commission, Research Fund for Coal and Steel (contract 7210-PA/371). The authors are grateful to Prof. Cristian Teodosiu from LPMTM (University Paris XIII) for fruitful discussions and guidance during this work.

REFERENCES