POLISH ACADEMY OF SCIENCES - MATERIALS SCIENCE COMMITTEE
SILESIAN UNIVERSITY OF TECHNOLOGY OF GLIWICE
INSTITUTE OF ENGINEERING MATERIALS AND BIOMATERIALS
ASSOCIATION OF ALUMNI OF SILESIAN UNIVERSITY OF TECHNOLOGY

Conference Proceedings

11th INTERNATIONAL SCIENTIFIC CONFERENCE
ACHIEVEMENTS IN MECHANICAL & MATERIALS ENGINEERING

Theoretical aspects of implementation of a rigid-plastic material model Using a modern yield criterion in the abaqus/standard finite-element code

G.A. Cosovici, D. Banabic, Gh. Achimas

Department of Manufacturing Engineering, Technical University of Cluj-Napoca
Bd. Muncii nr. 103-105, 3400 Cluj-Napoca, Romania

The paper presents the implementation of a rigid-plastic material model in the ABAQUS/Standard finite-element code. The discussion is focused on the theoretical aspects related to the derivation of the consistent tangent modulus for a membrane under plane-stress conditions. The authors have used this material model for the numerical simulation of sheet-metal forming processes (hydroforming, punch stretching, deep-drawing, etc.).

1. INTRODUCTION

ABAQUS/Standard [1] has been extensively used for the numerical simulation of sheet-metal forming processes [2, 3, 4]. An attractive feature of ABAQUS is the possibility to increase its functionality by means of user subroutines [2]. The authors have used this facility (namely, the UMAT subroutine) in order to define a rigid-plastic material model for membranes under plane-stress conditions.

The finite-element scheme implemented in ABAQUS/Standard is based on a Newton linearization of the principle of virtual work [1]. The solution of the equilibrium equations is performed in an iterative manner. A very important quantity needed during the iterations is the consistent tangent modulus $[C]$. More precisely, $[C]$ is a matrix relating small perturbations of the co-rotational strains to small perturbations of the corresponding co-rotational stresses in the current configuration:

$$\{\Delta\sigma\} = [C]\,\{\Delta\varepsilon\} \tag{1}$$

For a membrane under plane-stress conditions, $\{\Delta\sigma\}$ and $\{\Delta\varepsilon\}$ are defined as follows:

$$\{\Delta\sigma\} = [\Delta\sigma_{11}\,,\,\Delta\sigma_{22}\,,\,\Delta\sigma_{12}]^T\,, \quad \{\Delta\varepsilon\} = [\Delta\varepsilon_{11}\,,\,\Delta\varepsilon_{22}\,,\,2\Delta\varepsilon_{12}]^T \tag{2}$$

The components $\Delta\sigma_{\alpha\beta}$ and $\Delta\varepsilon_{\alpha\beta}$ $(\alpha,\beta = 1,2)$ are expressed in a local co-rotational basis [1, 2]. The first and the second unit vectors of this basis are tangent to the mid-surface of the membrane.

The main task of the researcher interested in implementing a new constitutive model in ABAQUS/Standard is to derive the expression of the consistent tangent modulus.

2. CONSISTENT TANGENT MODULUS FOR A RIGID-PLASTIC MEMBRANE UNDER PLANE-STRESS CONDITIONS

The basic component of a rigid-plastic constitutive model is the yield surface [5]:

$$\Phi(\overline{\sigma}, Y) := \overline{\sigma} - Y = 0 \tag{3}$$

where $\overline{\sigma}$ is the equivalent stress and Y is a yield parameter. In order to preserve the generality of the computations, the yield surface will be kept in the form given by Eqn (3), with no reference to any particular expression of the equivalent stress.

Another element of the constitutive model is the flow rule [5]:

$$\dot{\varepsilon}_{\alpha\beta} = \dot{\overline{\varepsilon}} \frac{\partial \Phi}{\partial \sigma_{\alpha\beta}} \quad (\alpha,\beta = 1,2) \tag{4}$$

where $\dot{\varepsilon}_{\alpha\beta}$ $(\alpha,\beta = 1,2)$ are the essential components of the plastic strain-rate tensor and $\dot{\overline{\varepsilon}}$ is the equivalent plastic strain-rate. The last quantity is defined by the power law [5]:

$$\overline{\sigma}\,\dot{\overline{\varepsilon}} = \sigma_{\alpha\beta}\,\dot{\varepsilon}_{\alpha\beta} \tag{5}$$

Assuming a purely isotropic hardening of the material, only one scalar state parameter is needed to describe the evolution of the yield surface. This parameter is the equivalent plastic strain computed as a time-integral of the equivalent plastic strain-rate [5]:

$$\overline{\varepsilon} = \int_0^t \dot{\overline{\varepsilon}}\,dt \tag{6}$$

The change of the yield surface is included in Eqn (3) by modifying the yield parameter Y according to a Swift hardening law [5]:

$$Y(\overline{\varepsilon}) = K(a + \overline{\varepsilon})^n \tag{7}$$

where K, a and n are material constants.

The flow rule given by Eqn (4) leads to many numerical difficulties when used in a finite-element programme. The equilibrium equations suffer from ill-conditioning when the equivalent plastic strain-rate is very small [6]. Due to this situation, the authors have decided to modify Eqn (4) by adding a penalization term. After integrating over the time increment, one obtains an approximation of the flow rule:

$$q\left(\{\varepsilon\} - \{{}^{0}\varepsilon\}\right) = q(\overline{\varepsilon} - {}^{0}\overline{\varepsilon})\{g\} + \{\sigma\} - \{{}^{0}\sigma\} \tag{8}$$

The notations used in Eqn (8) have the following significance: $\{\varepsilon\}$ and $\{^0\varepsilon\}$ are column-vectors collecting the essential components of the co-rotational strain tensors associated to the reference and current configurations, respectively; $\{\sigma\}$ and $\{^0\sigma\}$ are column-vectors collecting the non-zero components of the co-rotational stress tensors associated to the current and reference configurations, respectively (their structure is similar to that given by Eqn (2)); $\{g\}$ is the gradient column-vector

$$\{g\} = \left[\frac{\partial\Phi}{\partial\sigma_{11}} \ , \ \frac{\partial\Phi}{\partial\sigma_{22}} \ , \ 2\frac{\partial\Phi}{\partial\sigma_{12}} \right]^T \tag{9}$$

$^0\bar{\varepsilon}$ is the equivalent plastic strain associated to the reference configuration; q is a large positive constant (the numerical tests have shown that $q \approx 10^7$ MPa leads to very good results).

Giving a small perturbation to Eqn (8) and taking into account Eqn (3), one obtains after some calculations the desired expression of the consistent tangent modulus:

$$[C] = [c] - \frac{1}{\{g\}^T[c]\{g\} + H}\left([c]\{g\}\right)\left(\{g\}^T[c]\right) \tag{10}$$

where

$$H = \frac{dY}{d\bar{\varepsilon}} = nK(a + \bar{\varepsilon})^{n-1} \tag{11}$$

is the strain-hardening modulus and

$$[c] = q\left([U_3] + q(\bar{\varepsilon} - {}^0\bar{\varepsilon})[M]\right)^{-1} \tag{12}$$

is a penalty matrix. The notations used in Eqn (12) have the following significance: $[U_3]$ is the third-order unit matrix and

$$[M] = \begin{bmatrix} \frac{\partial^2\Phi}{\partial\sigma_{11}\partial\sigma_{11}} & \frac{\partial^2\Phi}{\partial\sigma_{11}\partial\sigma_{22}} & 2\frac{\partial^2\Phi}{\partial\sigma_{11}\partial\sigma_{12}} \\ \frac{\partial^2\Phi}{\partial\sigma_{11}\partial\sigma_{22}} & \frac{\partial^2\Phi}{\partial\sigma_{22}\partial\sigma_{22}} & 2\frac{\partial^2\Phi}{\partial\sigma_{22}\partial\sigma_{12}} \\ 2\frac{\partial^2\Phi}{\partial\sigma_{11}\partial\sigma_{12}} & 2\frac{\partial^2\Phi}{\partial\sigma_{22}\partial\sigma_{12}} & 2\left(\frac{\partial^2\Phi}{\partial\sigma_{12}\partial\sigma_{12}} + \frac{\partial^2\Phi}{\partial\sigma_{12}\partial\sigma_{21}}\right) \end{bmatrix} \tag{13}$$

is the curvature matrix.

Eqn (10) should be implemented into the UMAT subroutine in order to describe the mechanical behaviour of a rigid-plastic membrane under plane-stress conditions. Any expression of the equivalent stress $\overline{\sigma}$ can be included in Eqn (10). In fact, the expression of $\overline{\sigma}$ interacts with the consistent tangent modulus by means of matrices $\{g\}$ and $[M]$.

An analysis of the computational procedure described above shows that the constitutive equations represent a penalised elastoplastic material (q may be treated as an extremely large elastic modulus vanishing the reversible component of the strain).

3. CONCLUSIONS

The authors have developed a general methodology that may be used in order to obtain the consistent tangent modulus of a rigid-plastic membrane. The constitutive model is ready to be implemented in the UMAT subroutine of the ABAQUS/Standard finite-element programme. The model has been tested by numerical simulation of various sheet-metal forming processes. The results of such a simulation will be presented in a separate paper.

REFERENCES

1. *** ABAQUS Theory Manual. Version 5.2, Hibbitt, Karlsson & Sorensen, Inc., Pawtucket, 1992.
2. *** ABAQUS User's Manual. Version 5.8.1. Electronic documentation.
3. N. Rebelo, J.C. Nagtegaal and H.D. Hibbitt: Int. J. Num. Meth. Engng, 30 (1990) 1739.
4. A.Y. Moghadam: Theoretical Analysis of Sheet Drawing and Computer Simulation of Deep Drawing Processes. PhD Thesis, University of Tehran, 1999.
5. G.A. Cosovici: Modern Yield Criteria for Anisotropic Materials. State of the art. Research Report (in Romanian), Technical University of Cluj-Napoca, 2001.
6. G.A. Cosovici: Mechanical Modelling of the Deep Drawing Processes. Research Report (in Romanian), Technical University of Cluj-Napoca, 2002.