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

Numerical tests implementating 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 main objective of this paper is to evaluate the performances of a rigid-plastic constitutive model implemented as a UMAT subroutine in the ABAQUS/Standard finite-element programme. The constitutive equations are based on the Hill'90 yield criterion. The experimental data obtained from a hydraulic bulging test are compared with the results of a numerical simulation.

1. INTRODUCTION

The authors have developed a general procedure for implementing a rigid-plastic membrane model in ABAQUS/Standard [1]. They have obtained a consistent tangent modulus that could be adapted to various expressions of the equivalent stress. In order to evaluate the performances of their procedure, the authors have written a UMAT subroutine using the Hill'90 expression of the equivalent stress. In this way, the predictions of the theoretical model can be directly compared with experimental data.

2. CONSISTENT TANGENT MODULUS ASSOCIATED TO THE HILL'90 EQUIVALENT STRESS

The sheet metal is assumed to behave as an orthotropic rigid-plastic membrane under plane-stress conditions. In this case, the Hill'90 expression of the equivalent stress is as follows [2, 3, 4]:

$$\bar{\sigma} = \frac{1}{2}\Big\{\left[(\sigma_{11}+\sigma_{22})^2\right]^{m/2} + k^m\left[(\sigma_{11}-\sigma_{22})^2 + 4\sigma_{12}\sigma_{21}\right]^{m/2} + \left(\sigma_{11}^2+\sigma_{22}^2+2\sigma_{12}\sigma_{21}\right)^{m/2-1}\left[b(\sigma_{11}-\sigma_{22})^2 - 2a\left(\sigma_{11}^2-\sigma_{22}^2\right)\right]\Big\}^{1/m} \tag{1}$$

where a, b, k and m are material parameters, while $\sigma_{\alpha\beta}$ $(\alpha,\beta = 1,2)$ are the non-zero components of the stress tensor expressed in the system of orthotropy axes: 1 is the rolling direction (RD), 2 is the transverse direction in the plane of the sheet metal, and 3 is the direction perpendicular to the plane of the sheet metal. The parameters a, b, k and m can be calculated using the following relationships [2, 3, 4]:

$$m = \frac{\ln[2(r_{45}+1)]}{\ln \dfrac{2Y_b}{Y_{45}}}, \quad a = \frac{1}{4}\left[\left(\frac{2Y_b}{Y_{90}}\right)^m - \left(\frac{2Y_b}{Y_0}\right)^m\right],$$
$$b = \frac{1}{2}\left[\left(\frac{2Y_b}{Y_{90}}\right)^m + \left(\frac{2Y_b}{Y_0}\right)^m\right] - \left(\frac{2Y_b}{Y_{45}}\right)^m, \quad k = (2r_{45}+1)^{1/m} \tag{2}$$

The notations used in Eqn (2) have the following significance: Y_0, Y_{45} and Y_{45} are the uniaxial yield stresses associated to the directions inclined at 0º, 45º and 90º with respect to RD, Y_b is the equibiaxial yield stress, and r_{45} is the coefficient of plastic anisotropy associated to a direction inclined at 45º with respect to RD.

In order to obtain the consistent tangent modulus associated to the Hill'90 equivalent stress, one should evaluate the corresponding expressions of the gradient column vector $\{g\}$ and curvature matrix $[M]$ (see [1], Eqns (9) and (11), respectively). In fact, these matrices contain the first and second order derivatives of $\overline{\sigma}$ given by Eqn (1). The next step consists in replacing $\{g\}$ and $[M]$ in the relationship defining the consistent tangent modulus $[C]$ (see [1], Eqn (16)).

3. NUMERICAL SIMULATION OF A HYDRAULIC BULGING PROCESS

The authors have implemented the rigid-plastic material model in ABAQUS/Standard by means of a UMAT subroutine. This subroutine has been used for the simulation of a hydraulic bulging test. The hydroforming process is schematically shown in Figure 1. The mechanical parameters of the sheet metal (AA3003-0; thickness 1 mm) are listed in Table 1. The circular specimen comes into contact with the clamping ring in the region of the fillet. The frictional component of this interaction is described by the classical Coulomb model implemented in ABAQUS/Standard (a friction coefficient $\mu = 0.15$ is assigned to the contact surface). The pressure of the liquid has been used as a control parameter of the simulation. Its value is considered to have a linear variation from 0 to 4 MPa.

Due to the plastic orthotropy of the sheet metal, as well as the axial symmetry of the tools, only one quarter of the specimen needs to be meshed. The external boundary of the mesh corresponds to the clamping circle (see Figure 1). The radius of this circle is 58 mm. All the nodes belonging to the external contour are completely restrained. The fillet radius of the clamping ring is 5 mm (see Figure 1). The mesh used in the simulation consists of 18 M3D3 elements and 504 M3D4 elements [5], with a total number of 553 nodes.

The experimental work has been performed in the Institute of Metal Forming (IFU) of the University of Stuttgart. Four specimens have been subjected to the hydraulic bulging test.

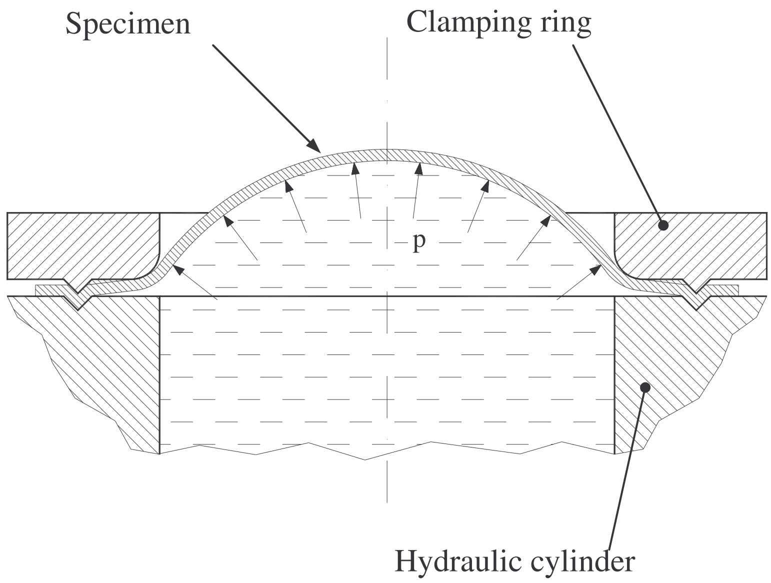

Figure 1. Hydraulic bulging test

Table 1
Mechanical parameters of the sheet metal (AA3003-0; thickness 1mm)

Mechanical parameters of the sheet metal	$Y_0 = 57$ MPa $Y_{45} = 57.5$ MPa $Y_{90} = 55.6$ MPa $Y_b = 58.5$ MPa $r_{45} = 0.875$
Coefficients of the Hill'90 yield criterion	$a = 4.512\times10^{-2}$ $b = 0.1517$ $k = 1.7224$ $m = 1.8606$
Coefficients of the Swift hardening rule	$K = 199.8$ MPa $a = 0.0056$ $n = 0.2005$
Penalisation factor (used in the rigid-plastic model)	$q = 10^7$ MPa

The polar deflection corresponding to different values of the liquid pressure has been recorded for each specimen. The results of these measurements are plotted in Figure 2 as discrete points. The dependence pressure vs. polar deflection predicted by the finite-element model is superimposed on the same diagram. One may notice a very good agreement between the numerical results and experimental data.

ABAQUS/Standard also computes the spatial distributions of the strain and stress components. A rectangular grid has been printed on each specimen. The distribution of the principal logarithmic strains at the end of the bulging test can thus be measured. The authors intend to perform this measurement in the future.

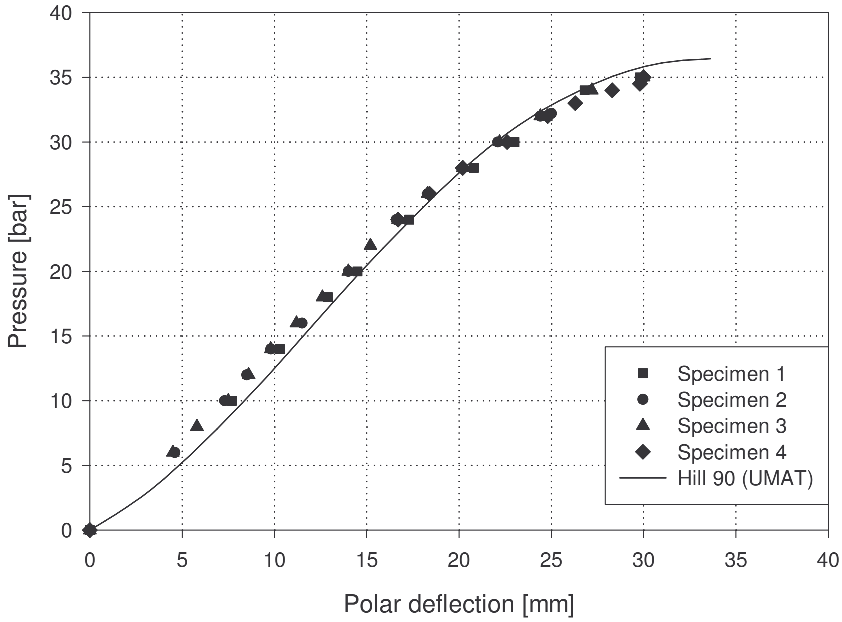

Figure 2. Pressure vs. polar deflection (comparison between numerical results and experimental data)

4. CONCLUSIONS

The authors have developed a general procedure for implementing a rigid-plastic membrane model in ABAQUS/Standard. The paper presents an application of this procedure for the Hill'90 yield criterion. The numerical results provided by the finite-element model are in very good agreement with the experimental data. The authors plan to include some other non-quadratic yield criteria in the constitutive model and to simulate more complex sheet metal forming processes.

REFERENCES

1. G.A. Cosovici, D. Banabic and Gh. Achimas: Implementation of a rigid-plastic material model using a modern yield criterion in the ABAQUS/Standard finite-element code. Part I: Theoretical aspects, Proc. AMME 2003 Conf., Gliwice-Zakopane (accepted for publication).
2. R. Hill: J. Mech. Phys. Sol., 38 (1990) 405.
3. D. Banabic, H.J. Bunge, K. Pöhlandt, A.E. Tekkaya: Formability of Metallic Materials. Plastic Anisotropy, Formability Testing, Forming Limits, Springer, Berlin, 2000.
4. G.A. Cosovici: Modern Yield Criteria for Anisotropic Materials. State of the art. Research Report (in Romanian), Technical University of Cluj-Napoca, 2001.
5. *** ABAQUS User's Manual. Version 5.8.1. Electronic documentation.