On some computational aspects for incremental sheet metal forming simulations

C. Robert[1,2], P. Dal Santo[1], A. Delamézière[2], A. Potiron[1], J.-L. Batoz[3]

[1]*ENSAM - LPMI – 2 bd du Ronceray, BP 93525 49525 Angers Cedex, France*
e-mail: camille.robert@angers.ensam.fr

[2] *GIP-InSIC – 27 rue d'Hellieule, 88100 Saint-Dié-des-Vosges, France*

[3] *UTC - GSU, Centre Pierre Guillaumat 2, BP 60319, rue du Docteur Schweitzer, 60203 Compiègne Cedex, France*

ABSTRACT: The paper deals with some numerical aspects to simulate efficiently the incremental sheet forming process. Firstly, in order to consider the production of real industrial parts, complex tool paths are needed and a Computed Aided Manufacturing software is used to provide them. Second the finite element simulation is highly CPU time consuming and in an attempt to decrease that time, a simplified elasto-plastic scheme based on the incremental deformation theory of plasticity has been introduced in ABAQUS. Taking into account the equivalent plastic strain, the stress flow and the actual thickness of the sheet, a reduction of CPU time and a good prediction are observed (considering the flow rate elasto-plastic scheme as reference).

Key words: Incremental sheet forming, FEM, plasticity, ABAQUS, CAM

1 INTRODUCTION

The incremental sheet forming process (ISF) has been developed in the context of sheet metal forming to increase the flexibility of that important industrial process [1]. ISF is mainly used to produce small batch size or as a rapid prototyping process in different industries, from transportation to medical field [2, 3].

ISF allows a significant reduction of the tooling cost for small production of sheet parts since traditional, expensive and complex tools are replaced by a simple hemispherical punch moving on a controlled tool path. To increase the quality of the final geometry, it is still possible to use a die. But that die can be manufactured in a cheap material because the applied forces are low [4]. Another advantage of this process is the high limit of formability, compared to classical limit forming observed in stamping [5,6].

In the industrial context, the numerical simulation must efficiently predict the geometry of the part, the thickness, the strains and the stresses in the sheet throughout the forming process. In ISF the contact zone between the tool and the sheet is limited and is always changing with the movement of the tool along its path. Each material point of the sheet is then subjected to elasto-plastic loading and unloading depending if the tool is far or not from that point. In an attempt to reduce the CPU time we studied the computational aspects involved to represent the elasto-plastic material behaviour. The classical elasto-plastic integration scheme based on the flow rule requires several iterations. The incremental deformation theory of plasticity is considered as an alternative to reduce the CPU time.

The first part of this paper deals with these two aspects of the plasticity (i.e. flow rule and incremental deformation theory). After that, the implementation of CAM tool path into ABAQUS is explained. Finally, results obtained for a benchmark are presented and discussed.

2 MODELLING OF BEHAVIOUR LAW

2.1 Flow rule method

The flow rule theory of plasticity is dominantly used in the context of computational plasticity for metals. With this method, the direction of the plastic flow is the same as the direction of the outward normal to the yield surface. To determine the state variables, a

plastic corrector, also called Lagrange multiplier, is necessary.
The stress tensor σ is related to the elastic strain tensor ε^{el} by:

$$\boldsymbol{\sigma} = \mathbf{C}^{el} : \boldsymbol{\varepsilon}^{el} \quad (1)$$

where C^{el} denotes the elastic stiffness tensor.
The evolution of the isotropic hardening is modelling by the Swift law:

$$R = K\left(\bar{\varepsilon}^{pl} + \varepsilon_0\right)^n \quad (2)$$

where $\bar{\varepsilon}^{pl}$ is the equivalent plastic strain. K, ε_0 and n are three experimental parameters.
The yield surface f can be defined by the Von Mises criterion, with the deviatoric stress tensor **S**:

$$f = \sqrt{\frac{3}{2}\mathbf{S} : \mathbf{S}} - R \quad (3)$$

The normality flow rule determines the increment of plastic strain tensor and the increment of equivalent plastic strain:

$$\begin{cases} \dot{\boldsymbol{\varepsilon}}^{pl} = \dot{\lambda}\dfrac{\partial f}{\partial \boldsymbol{\sigma}} \\ \dot{\bar{\varepsilon}}^{pl} = \dot{\lambda} \end{cases} \quad (4)$$

where $\dot{\lambda}$ denotes the plastic corrector.
The consistency condition is:

$$df = \frac{\partial f}{\partial \boldsymbol{\sigma}} : \dot{\boldsymbol{\sigma}} + \frac{\partial f}{\partial R}\dot{R} \quad (5)$$

In the case of a plastic flow rule, equation (5) should be zero. We can determine the plastic corrector:

$$\dot{\lambda} = \frac{\dfrac{\partial f}{\partial \boldsymbol{\sigma}} : \mathbf{C}^{el} : \dot{\boldsymbol{\varepsilon}}}{\dfrac{\partial f}{\partial \boldsymbol{\sigma}} : \mathbf{C}^{el} : \dfrac{\partial f}{\partial \boldsymbol{\sigma}} + HR} \quad (6)$$

where *HR* denote the tangent stiffness plastic modulus:

$$HR = \frac{\partial R}{\partial \bar{\varepsilon}^{pl}} = n.K\left(\bar{\varepsilon}^{pl} + \varepsilon_0\right)^{n-1} \quad (7)$$

On the figure 1, the Von Mises criterion is plotted (bold ellipse) in the principal stresses (with the hypothesis of plane stresses) at the end of increment. Numerically, the first step is to take into account an elastic prediction (i.e. $\Delta\varepsilon^{el} = \Delta\varepsilon$) and the yield function *f* is computed. Then, if *f* is greater than zero (point B on the figure 1), the stresses are updated iterations in order to stay on the yield surface (points C and D). The normality of the computed yield function is used at each iteration.

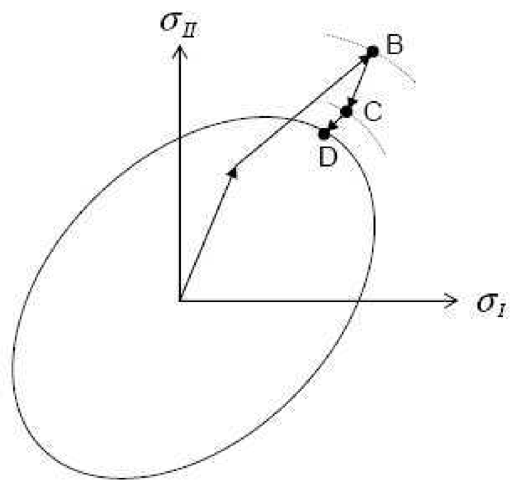

Fig. 1. Flow rule method

2.2 *Incremental strain method*

The incremental deformation theory is based on a simplification of the elasto-plastic scheme. Contrary to the flow rule method, it does not take into account the normal to the yield surface but only the strain tensor to obtain the stress state [7]. The advantage of this method is that no iteration is necessary and the CPU time should be reduced.
At a given time, stress and strain are defined by the point Q on the figure 2. To get the stress at the next increment of time (point R), we use an elasto-plastic modulus E^{ep}.
The incremental strain (Δε) is composed additively by elastic ($\Delta\varepsilon^{el}$) and plastic ($\Delta\varepsilon^{pl}$) components as:

$$\Delta\boldsymbol{\varepsilon} = \Delta\boldsymbol{\varepsilon}^{el} + \Delta\boldsymbol{\varepsilon}^{pl} \quad (8)$$

Let C^{pl} the plastic stiffness tensor and ${}^{0}\varepsilon^{el}$ the total elastic strain at the initial state of the current incremental step, we defined ε^* as:

$$\boldsymbol{\varepsilon}^* = {}^{0}\boldsymbol{\varepsilon}^{el} + \Delta\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon}^{el} + \Delta\boldsymbol{\varepsilon}^{pl} \quad (9)$$

The elasto-plastic stiffness tensor C^{ep} is:

$$\left(\mathbf{C}^{ep}\right)^{-1} = \left(\mathbf{C}^{el}\right)^{-1} + \left(\mathbf{C}^{pl}\right)^{-1} \quad (10)$$

A relation between the stress tensor σ and the strain ε^* is then obtained:

$$\boldsymbol{\sigma} = \mathbf{C}^{ep} : \boldsymbol{\varepsilon}^* \quad (11)$$

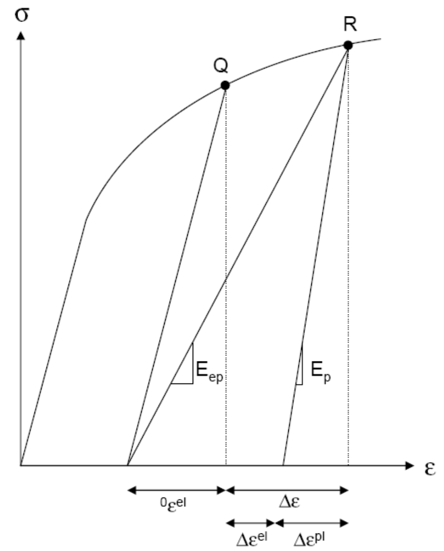

Fig. 2. Incremental deformation method

3 TOOL PATH

The efficiency of Single Point Incremental Forming is strongly dependent of the tool paths strategy. These may be very complex in the case of industrial parts [8]. Our idea is to use a Computer Aided Manufacturing (CAM) software to determine the tool path. The generated paths are used as data entry to a finite element simulation package. The CAD/CAM software CATIA is chosen. The tool paths are saved in a universal APT file format. Commonly, this universal format can be post-treated to generate control files for numerical machines. In this case The APT file is automatically generated by a Visual Basic script.

Python scripts are used within ABAQUS to translate the data of the APT file in order to describe the time tool positions components in ABAQUS explicit. The maximum velocity is an input parameter. We use smooth amplitude curves (already implemented into ABAQUS) A and B to get, between two points, a continuum acceleration which is described by:

$$P = P_A + (P_B - P_A)\xi^3(10 - 15\xi + 6\xi^2)$$
$$\xi = \frac{t - t_A}{t_B - t_A} \qquad (12)$$

P is the location at the time t, P_A and P_B are the locations of point A and B respectively. The acceleration is then controlled to avoid high inertia effects in the dynamic explicit scheme.

4 RESULTS

In this study, we analyse the forming of a cup with a diameter of 85mm and a depth of 25mm as show on the figure 3. The initial flange dimensions are: 140mm*140mm*1.5mm. The material parameters for the aluminium sheet are: E = 69 GPa, ν = 0.3, K = 150 MPa,

ε_0 = 0.013, n = 0.214 and ρ = 2700 Kg/m^3.

We used a hemispherical tool having a diameter of 30mm.

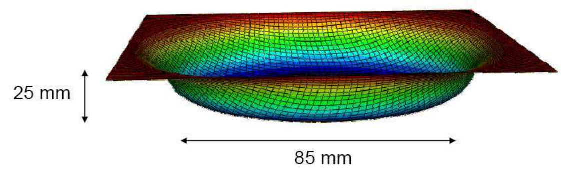

Fig. 3. Final geometry of the cup

The finite elements used are shells with 4 nodes and reduced integration (S4R) and with 9 Simpson points in the thickness direction. The global size of elements is 2*2mm and the model has 30525 degrees of freedom. The tool is modelled by an analytic rigid surface.

The ABAQUS explicit solver is used without mass scaling and the maximum velocity of the tool is 25m/s. This high velocity is not representative of the real process, but is here a numerical parameter considering that a quasi-static problem is solved thanks an explicit dynamic method. A high velocity can be used if the ratio of kinetic energy on the deformation energy is small during the simulation (i.e. small inertia effects).

The tool path is defined by a five circle pocket generated by CATIA (figure 4) and integrated into ABAQUS.

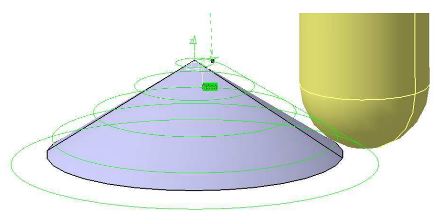

Fig. 4. CATIA tool path

In order to have an estimation of the reduction of CPU time, the two elasto-plastic schemes are coded into an ABAQUS user subroutine VUMAT.

Let t_{is} and t_{fr} the CPU time with incremental deformation theory and flow rule theory respectively. The time benefit t_{benef} is given by:

$$t_{benef} = 100 \frac{t_{fr} - t_{is}}{t_{fr}} \quad (13)$$

In this case, the time benefit is 4.2%.

Let σ^0_{is} and σ^0_{fr} the stress flow with incremental deformation theory and flow rule theory respectively, ε^{pl}_{is} and ε^{pl}_{fr} the equivalent plastic strain and th_{is} and th_{fr} the thickness of the sheet. An estimation of the quality of the results given by the incremental strain theory is given by:

$$ERR\sigma = 100 \left| \frac{\sigma^0_{is} - \sigma^0_{fr}}{\sigma^0_{fr}} \right|$$

$$ERR\varepsilon = 100 \left| \frac{\varepsilon^{pl}_{is} - \varepsilon^{pl}_{fr}}{\varepsilon^{pl}_{fr}} \right| \quad (14)$$

$$ERRth = 100 \left| \frac{th_{is} - th_{fr}}{th_{fr}} \right|$$

Where $ERR\sigma$ is the stress flow error, $ERR\varepsilon$ is the equivalent plastic train error and $ERRth$ the thickness error. These errors are showed on figure 5, figure 6 and figure 7 respectively at the end of forming process.

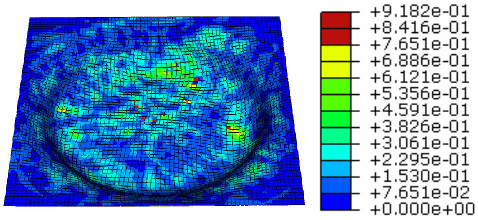

Fig. 5. Stress flow error (%)

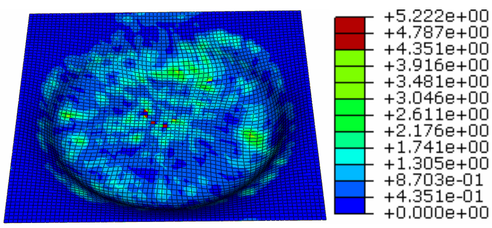

Fig. 6. Equivalent plastic train error (%)

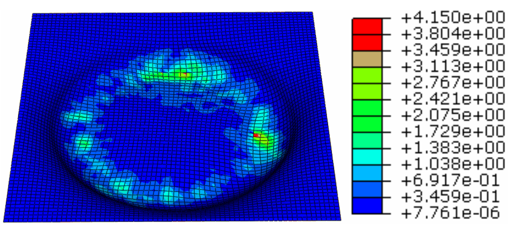

Fig. 7. Thickness error (%)

5 CONCLUSIONS

In this paper, the incremental deformation theory of plasticity has been implemented in ABAQUS using a behaviour law implemented in a specific user function (VUMAT). The model has been tested on a cup forming process. The results are compared with those obtained with the classical flow rule theory. We found that the numerical errors are acceptable while a reduction of CPU time was observed.

An integration of the tool path definition created by CATIA in ABAQUS has also been developed using Visual Basic and Python scripts and the maximal velocity of the tool is then controlled.

REFERENCES

1. M. Yamashita, M. Gotoh and S.-Y. Atsumi, ‘Numerical simulation of incremental forming of sheet metal’, Journal of Material Processing Technology, 199, (2008) 163-172.
2. F. Micary, G Ambrogio and L.Filice, ‘Shape and dimensional accuracy in Single Point Incremental Forming: State of the art and future trends’, Journal of Material Processing Technology, 191, (2007) 390-395.
3. J. Kopac and Z. Kampus, ‘Incremental sheet metal forming on CNC milling machine-tool’, Journal of Material Processing Technology, 162-163, (2005) 622-628.
4. E. Ceretti, C. Giardini and A. Attanasio, ‘Experimental and simulative results in sheet incremental forming on CNC machines’, Journal of Material Processing Technology, 150, (2004) 176-184.
5. G. Hussain and L. Gao, ‘A novel method to test the thinning limits of sheet metals in negative incremental forming’, International Journal of Machine Tools & Manufacture, 47, (2007) 419-435.
6. Y.H. Kim and J.J. Park, ‘Effect of process parameters on formability in incremental forming of sheet metal’, Journal of Material Processing Technology, 130-131 (2002) 42-46.
7. N. Ramakrishnan, K.M. Singh, R.K.V. Suresh and N. Srinivasan, ‘An algorithm based on total-elastic-incremental-plastic strain for large deformation plasticity’ Journal of Material Processing Technology, 86, (1999) 190-199.
8. M. Bambach, M. Cannamela, M. Azaouzi, G. Hirt, J.-L. Batoz, ‘Computer-aided tool path optimization for single point incremental sheet forming’, Advanced Methods in Material Forming, Springer, 2007, pp. 233-250.