Int J Mater Form (2008) Suppl 1:109–112
DOI 10.1007/s12289-008-0032-1

F.E. elastoplastic damage model with 2D adaptive remeshing procedure for fracture prediction in metal forming simulation

H. Badreddine[1], C. Labergère[1], K. Saanouni[1], W. Rajhi[1,2], A. Rassineux[3], D. Kircher[4]

[1] *UTT de Troyes, ICD/Lasmis – 12 rue Marie Curie BP, 10010 Troyes, France.*
URL: www.utt.fr e-mail: houssem.badreddine@utt.fr ; carl.labergère@utt.fr; khemais.saanouni@utt.fr

[2] *ESST de Tunis – LMMP, rue Taha HusseinB.P. 56, 1008 Tunis, Tunisia*

[3] *University of technology of Compiegne, Laboratoire Roberval, UMR 6253 du CNRS , centre de Recherches de Royallieu, BP 20529, 60205 Compiegne e-mail: alain.rassineux@utc.com*

[4] *Arcelor Innovation R&D – Voie Romaine, 57283 Maizières-lès-Metz, France*

ABSTRACT: In this work, an anisotropic elastoplastic finite element model strongly coupled with ductile damage is applied to simulate some metal forming tests used by Arcelor Research. The F.E. code is linked to a 2D adaptive remeshing procedure. First, the anisotropic elastoplastic model with non linear kinematic and isotropic hardening strongly coupled with ductile damage is presented. This model is written in finite plastic deformation through the so called rotated frame formulation using a non associative plasticity assumption with state variables. The adaptive analysis including the 2D mesh adaptation together with adaptive loading sequences and fully damaged elements deletion is described. This 2D adaptive procedure is applied to some metal forming tests with various high properties steel materials as the hole blanking and expansion. Two different cases are performed: (i) an initial hole is expanded starting from the zero stress at virgin state; (ii) the hole is first formed by blanking operation followed by the expansion process taking into account the residual fields (stress, strain, damage).

Key words: Finite anisotropic plasticity, ductile damage, numerical simulation, adaptive mesh.

1 INTRODUCTION

In our laboratory, an extensive work has been developed since ten years, in order to describe the ductile damage modelling in bulk and sheet metal forming by using advanced constitutive modelling [1-8]. Based on the thermodynamics of irreversible processes with state variables, the advanced approach aims to describe the coupling between the main thermo-mechanical fields and the ductile damage. These models have been implemented in ABAQUS® using the available user subroutines (Umat, Vumat, Uel and Vuel). An adaptive meshing and remeshing procedure with geometrical (local curvature at the contact points between the tools and the part) and physical error estimates (stress, plastic strain, damage) which kills the fully damaged elements has been developed based on the work by Rassineux et al [9, 10].
In the present work, this approach is shortly discussed from both theoretical and numerical points of view. Applications are made to the simulation of a hole blanking and a hole expansion processes using a high-strength steel material provided by Arcelor Mittal company. A comparison between the cases of the hole expansion of an ideal and previously blanked holes is made.

2 ABOUT THE COUPLED CONSTITUTIVE EQUATIONS

The constitutive equations are formulated on an appropriated Eulerean intermediate configuration having the same Lagrangian orientation as the initial undeformed configuration according to the RFF method (see [6-8] among many others). Using this ‘rotated’ objective formulation a complete set of constitutive equations can be obtained. In this paper a non associative and anisotropic plastic formulation accounting for the nonlinear isotropic and kinematic hardening fully coupled with the isotropic damage is considered. This model introduces several material parameters which should be identified given in the following:

- $\underline{\underline{\Lambda}}$ is the fourth order elastic properties tensor of the non damaged material.
- σ_y is the initial yield stress in simple tension.
- C is the kinematic hardening modulus and Q is the linear isotropic hardening modulus.
- a and b characterize respectively the kinematic and isotropic hardening non linearity.
- β, S, s and Y_0 characterize the ductile damage evolution.
- $\underline{\underline{H}}$ is fourth order plastic anisotropic tensor characterized by six material constants F, G, H, L, M and N.

For the sake of shortness the constitutive are not given. The reader is invited to refer to [6-8] for more details about the formulation of the overall fully coupled constitutive equations.

3 NUMERICAL ASPECTS

The model developed above has been implemented into ABAQUS/Explicit® FE software for metal forming simulation thanks to the user subroutine Vumat (ABAQUS® Theory Manual). The dynamic explicit global resolution schema is developed in detail in (ABAQUS® Theory Manual) considering the contact with friction of Coulomb type characterized by the friction parameter η. The computation of the stress tensor $\underline{\sigma}$ on the rotated (Lagrangien) configuration is required in order to evaluate the internal stress vector at each integration point inside each finite element for the end of each time increment. This is achieved by integrating all the constitutive equations of the model presented above including the ductile damage. The classical incremental and iterative elastic predictor – plastic elastic prediction plastic correction method [11-13] is used together with the reduction of the number of differential equations to be solved. This procedure is fully described in ([1-8]) and therefore is not presented here.

4 APPLICATIONS

4.1 Identification of the material parameters

Arcelor Research provided several experimental results of uniaxial tension tests until final fracture with sheet specimen cut on 0°, 45° and 90° orientations with respect to the rolling direction. By using this experimental database, the quasi-isotropy of the material has been shown. Accordingly, the values of the different parameters obtained for the studied material are the following:
E= 195000 MPa, υ=0.3, σ_y=405 MPa, Q=5500 MPa, b=10, C=38000 MPa, a=290, F=G=H=0.5, L=M=N=1.5, S=45 MPa, s=1 , β=2, y_0=0MPa and the characteristic mesh length h_{min}=0.1 mm.

4.2 Cutting process simulation

The scheme of the cutting process is given in figure 1.

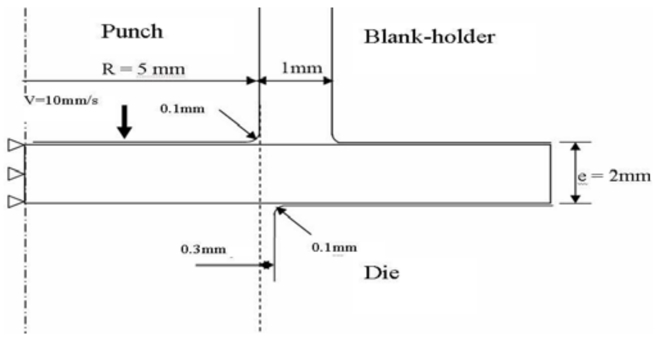


Fig.1. Scheme of the cutting process

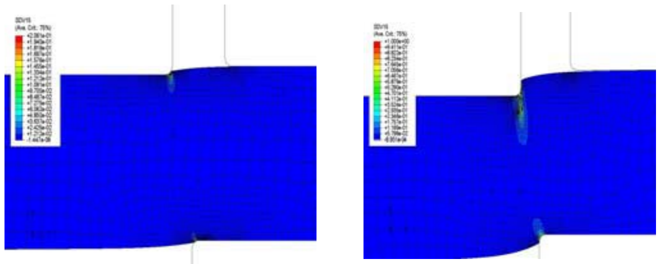
The problem is supposed as axis-symmetric. The simulation is made using the isotropic version of the model presented above and with the adaptive remeshing procedure with CAX4R axis-symmetric elements from the ABAQUS® element library. The parameters governing the adaptive analysis are: $h_{max} = 0.8$mm, $h^p_{max} = 0.2$mm, $h^p_{min} =$ 0.07 and $h^d_{min} = 0.03$mm.
Some steps of the cutting process are given in figure 2. The path (i.e. the location of the deleted fully damaged elements) followed by the macroscopic crack is clearly shown. We can also observe that the mesh is refined in the vicinity of the contact points between the tools and the sheet due to the small radius of the tools. Also note that the mesh size is coarsened after the formation. The sheet is completely cut after a punch displacement about 0.63 mm, which represents about 25% of the sheet thickness.

4.3 Hole expansion simulation

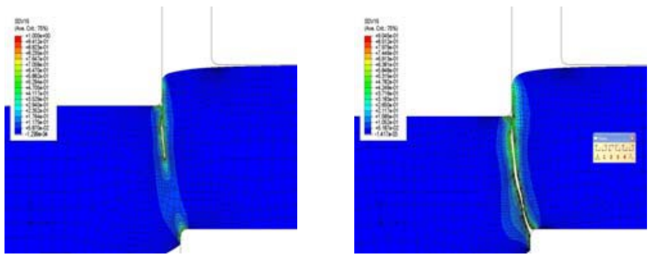
The scheme of the hole expansion process is given in figure 3. The problem is supposed to be axis-symmetric and the tools are taken as rigid bodies. The computation is performed without adaptive remeshing with CAX4R axis-symmetric elements from ABAQUS® element library. Three cases are considered and compared:

- Case 1: Hole expansion starting from a perfect hole. In that case no adaptive remeshing is used.
- Case 2: Hole expansion using pre-blanked hole and the expansion is made in the blanking direction.
- Case 3: Hole expansion using pre-blanked hole and the expansion is made in the inverse direction of the blanking direction.

For the last two cases, all the mechanical residual fields (stress, plastic strain, damage,…) due to previous blanking operation are transferred as well as the final geometry of the blanked hole (see figure 4).



(a) U=0.07mm *(b) U=0.31mm*



(c) U=0.50mm *(a) U=0.63mm*

Fig.2. Damage maps obtained for different punch displacement of the cutting process.

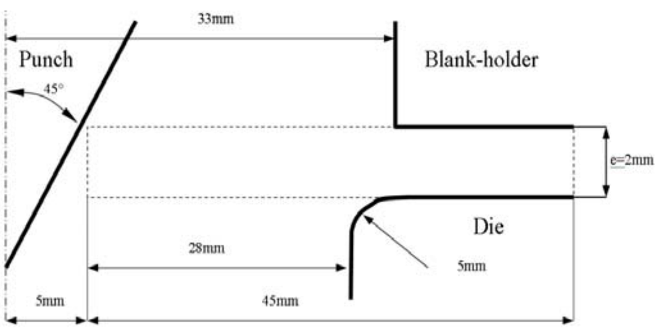


Fig.3. Scheme and geometry of the hole expansion process.

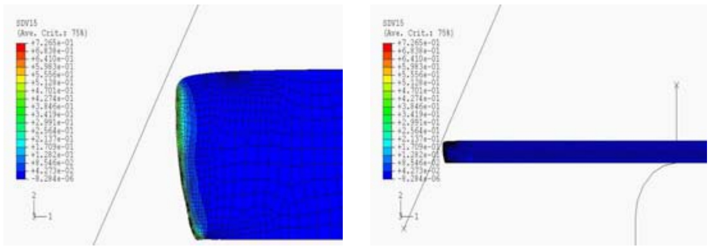
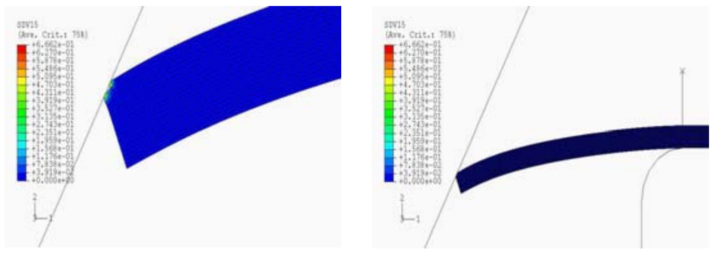


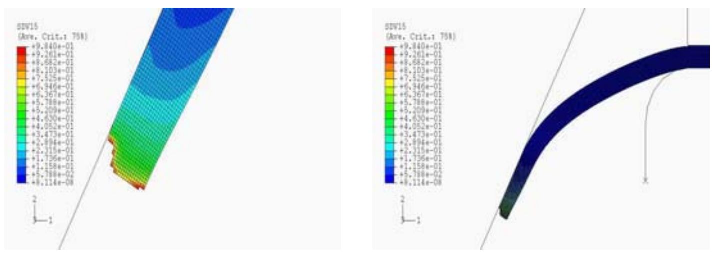
Fig.4. Transferred geometry and residual fields of the pre-blanked hole.

Some steps of the simulation of the case 1 are shown in figure 5. In this figure we can observe that for a 5.62mm punch displacement, some elements located at the contact area between the punch and the sheet in the vicinity of the hole are damaged due to the contact pressure. A macroscopic crack is initiated in the external side of the hole for a punch displacement of about 24.75mm (see figure 5.b).

Some steps of the simulation of cases 2 and 3 are given in figures 6 and 7 respectively. We can observe that in both cases, a damaged zone develops at the area of contact with the tool for different tool displacements u=3.2 mm for case 2 and u=5.1mm for case 3. When comparing case 1 (Figure 5) and case 2 (Figure 6) the damage distributions are clearly different indicating the important role of the residual fields generated by the previous blanking operation.

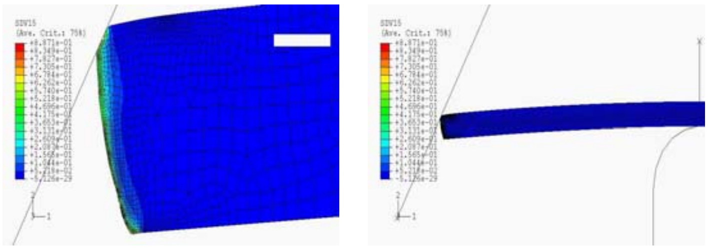


(a) U=5.62mm

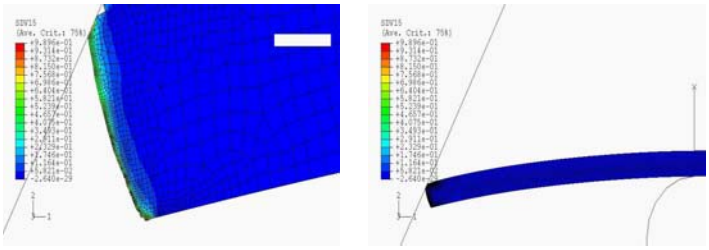


(b) U=24.75mm

Fig.5. Damage maps obtained for different values of the punch displacement (process with perfect hole).

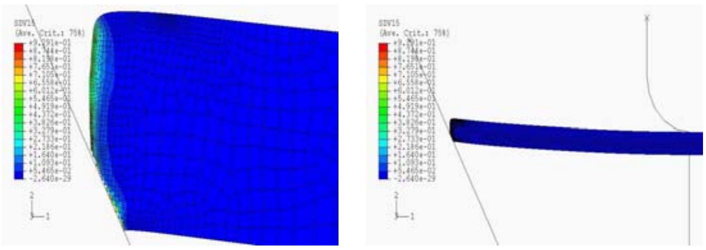


(a) U=1.5mm

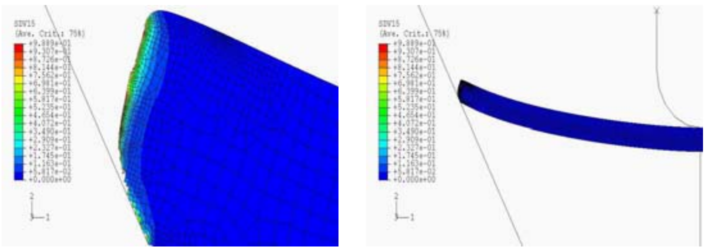


(b) U=3.2mm

Fig.6. Damage maps obtained for different values of the punch displacement (process with pre-blanked hole made in the cutting direction).



(a) U=1.5mm



(b) U=5.1mm

Fig.7. Damage maps obtained for different values of the punch displacement (process with pre-blanked hole made in the inverse cutting direction)

5 CONCLUSION

In this paper, an elastoplastic model strongly coupled with ductile damage has been briefly presented. This mechanical model have been both implemented into ABAQUS® F.E code and used in connection with a 2D adaptive meshing and remeshing procedure. This adaptive numerical methodology is used to simulate blanking and expansion of holes inside a quasi-isotropic sheet. The results are very encouraging and show the ability of the proposed modelling to predict the rupture in sheet metal forming. This adaptive analysis should be extended to 3D analysis and compared to experimental results in future research.

REFERENCES

1. Saanouni K., Hammi Y., « Numerical simulation of damage in metal forming processes », in Continuous Damage and Fracture , Editor A. Benallal, Elsevier, ISBN. 2-84299-247-4, pp :353-363, 2000
2. Saanouni, K., Forster C. and Ben Hatira F., « On the Anelastic Flow with Damage », Int. J. Dam. Mech., 3:140-169, 1994
3. Saanouni K. and Chaboche J.L., 'Computational Damage Mechanics. Application to Metal Forming', Chapter 7 of the Volume 3 : 'Numerical and Computational methods' (Editors: R. de Borst, H. A. Mang), in 'Comprehencive Structural Integrity', Edited by I. Milne, R.O. Ritchie and B. Karihaloo, ISBN: 0-08-043749-4, 2003, Elsevier, Oxford.
4. Lemaitre, J., and Chaboche J.L., Mécanique des Matériaux Solides, Dunod, Paris, 1985.
5. Khelifa M., Badreddine H., Belamri N., Gahbich M. A., Saanouni K., Cherouat A., Dogui A., Effect of anisotropic plastic flow on the ductile damage evolution in hydrobulging test of thin sheet metal, Int. Journal of Forming Processes, Vol. 8, N°:2, pp : 271-289, 2005
6. M.,Khelifa, K., Saanouni, H., Badreddine, M.-A., Gahbiche, and A., Dogui, "Plasticité anisotrope couplée à un endommagement ductile isotrope: Appliction au gonflement hydraulique de tôles minces", Revue Européenne des Eléments Finis, Vol15., N°7-8, pp. 891-908, 2006.
7. Badreddine H., Saanouni K., Dogui A., Gahbich M.A., Elastoplasticité anisotrope non normale en grandes déformations avec endommagement. Application à la mise en forme de tôles minces. Revue Européenne de Mécanique Numérique, Volume 16, N°6 et 7,2007, pp :913-940.
8. H., Badreddine, "Elastoplasticité anisotrope endommageable en transformations finies : Aspects théoriques, numériques et applications". Thèse de Doctorat, 2006, Université de Technologie de Troyes (France) – Ecole Nationale d'Ingénieurs de Monastir (Tunisie).
9. A. Rassineux, «An automatic mesh generator for planar domains», StruCome (1991), p. 519-531.
10. C. Labergere, A. Rassineux,, K. Saanouni, *Endommagement et procédé de mise en forme. Apport du maillage adaptatif,* 8th Colloque National en Calcul des Structures, Giens, France, 21-25 Mai, (2007), CD.
11. Simo J.C., Hughes T.J.R., Computational inelasticity, Springer, New York, 1998
12. J.C. Simo, and M., Ortiz, « A Unified Approach to Finite Element Deformation Elastoplastic Analysis Based on the Use of Hyperelastic Equations », Comp. Meth. Appl. Meth. Engng., vol 49, 1985, pp, 221-245.
13. T. J. R., Hughes, J., Winget, « Finite rotation effects in numerical integration of rate constitutive equations arising in large-deformation analysis », Int. J. Nume. Meth. Engng, 15, 1980: 1862-1867.
14. D. Freed, and K. P., Walker, «Exponential integration algorithm for first-order ODEs with application to viscoplasticity». ASME Summer Conf. On Mechanics and Materials Recent Advances on Damage Mechanics and Plasticity, Tempe, 1992.