# Effect of FEM choices in the modelling of incremental forming of aluminium sheets

S. He[1], A. Van Bael[1], P. Van Houtte[1], Y. Tunckol[2], J. Duflou[2],
C. Henrard[3], C. Bouffioux[3], A.M. Habraken[3]

[1] *Department MTM, Katholieke Universiteit Leuven, Kasteelpark Arenberg 44, B-3001 Leuven, Belgium*
*URL: www.mtm.kuleuven.ac.be* *e-mail: Shen.He@mtm.kuleuven.ac.be, Albert.VanBael@mtm.kuleuven.ac.be, Paul.VanHoutte@mtm.kuleuven.ac.be*

[2] *Department of Mechanical Engineering, Katholieke Universiteit Leuven, Celestijnenlaan 300B, B-3001 Leuven, Belgium*
*URL: www.mech.kuleuven.ac.be* *e-mail: Yasemin.Tunckol@mech.kuleuven.ac.be, Joost.Duflou@mech.kuleuven.ac.be*

[3] *Department of Mechanics of Solids and Materials, Université de Liège, Chemin des Chevreuils 1, B-4000 Liege, Belgium*
*URL: www.ulg.ac.be/matstruc* *e-mail: Christophe.Henrard@ulg.ac.be, Chantal.Bouffioux@ulg.ac.be, Anne.Habraken@ulg.ac.be*

ABSTRACT: This paper investigates the process of single point incremental forming of an aluminium cone with a 50-degree wall angle. Finite element (FE) models are established to simulate the process. Different FE packages have been used. Various aspects associated with the numerical choices as well as the material and process parameters have been studied. The final geometry and the reaction forces are presented as the results of the simulations. Comparison between the simulation results and the experimental data is also made.

Key words: finite element method, incremental forming

## 1 INTRODUCTION

The incremental forming process has emerged in the past few years as a potential alternative to conventional sheet metal pressing to produce prototypes and small batch productions at low cost. The process uses a smooth-end tool to create a local indentation in a clamped sheet, and by dragging the point of contact around the sheet according to a programmed tool path, a wide variety of shapes may be formed without the need for specific tooling. Two versions of the process have been explored: with and without a supporting post on the reverse side of the workpiece. This paper will focus on the latter approach, so-called ´single point´ incremental forming (SPIF).

Recent research work on SPIF has demonstrated that various process parameters can have influence on the deformation of the material [1-3] and the finite element method has been used for the systematic study on such influence. However, this process is still early in its development and requires much more research to better understand the deformation mechanics and the influence of the process parameters. This paper investigates the incremental forming of an aluminium cone with a 50-degree wall angle by means of finite element analysis. The experimental set-up of the process has been described in ref. [4]. During the process, the tool travels along a series of contours generated transverse to the radial direction of the cone and the vertical step size between two consecutive contours is chosen as 0.5 mm. Different numerical choices concerning the simulation as well as the process are discussed and the comparisons of the cone shape and reaction forces between different parameters are presented.

## 2 FINITE ELEMENT MODELING

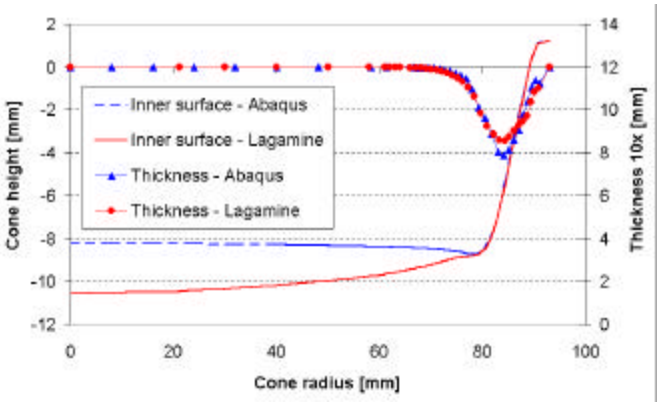
Two FE packages, namely Lagamine developed at University of Liege and Abaqus, are used in this

study in order to compare the accuracy and reliability between these two codes. A three-dimensional, elasto-plastic FE model is set up for each code. During the deformation, there is actually no real symmetry and a model for the whole geometry should be established, which however would be very computationally demanding. Therefore only one quarter of the blank is considered in a simplified model. Rolling direction coincides with a symmetry axis called "section at 0 degree". Proper boundary conditions are assigned to the mesh: a symmetry is imposed on the sections at 0 and 90 degree of the meshed blank, respectively. The brick elements (C3D6 and C3D8R in Abaqus, BWD in Lagamine) with reduced number of integration points are utilised for the blank. There are 24 elements in the circumferential direction and respectively 56 (Abaqus) and 46 (Lagamine) in the radial direction. The mesh for the blank consists of three layers of elements across the thickness. Both the tool and the backing plate are modelled as rigid surfaces. The Coulomb's friction law with a friction coefficient of 0.05 is applied to describe the friction between the tool and the sheet. The as-received material is fully annealed aluminium alloy AA3003-O with a thickness of 1.2mm. The flow stress curve of the material is approximated by the Swift law $\sigma=180(\varepsilon+0.00109)^{0.21}$ (MPa). The Hill´48 yield criterion derived from Lankford coefficients ($r_0 = 0.68$, $r_{45} = 0.73$ and $r_{90} = 0.66$) is employed to describe the plastic anisotropy during the deformation. No temperature dependence is taken into consideration in the current simulation.

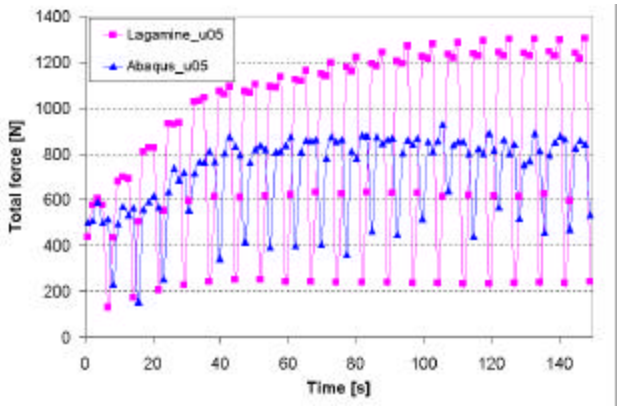
## 3 RESULTS AND DISCUSSIONS

### *3.1 Comparison between two FE packages*

As mentioned above, the FE codes Abaqus and Lagamine have been used in the present study. The same boundary conditions are applied with both codes and the mesh densities are identical in the region that will be deformed by the tool. Figure 1(a) presents the predicted cone shape after 20 contours as well as the thickness distribution magnified by a factor 10. The results are obtained from the cross section at 45 degree with respect to the section at 0 degree of the blank. For the cone shape, both codes give almost identical results in the cone wall region, whereas the centre of the cone bottom obtained by Lagamine is about 2.3mm deeper than that from Abaqus. It has been reported in ref. [4] that the bottom of the cone calculated by Abaqus is already deeper than the experimental one. Modelling the full blank would lead to a higher bottom as checked by other simulations. So clearly the boundary conditions bring some additional stiffness. The difference between the two codes needs further investigation. Different explanations are possible: effect of the different mesh densities, too stiff behaviour of BWD element, too high penalty used in the Lagamine contact model. As checked in ref. [4] with refined meshes and test without contact, both codes provide identical results. For the cone thickness, they predict nearly the same distribution despite the difference for the cone shape. Obviously, the thickness at the bottom remains unchanged, whereas in the cone wall region, the material has thinned considerably.




(a)




(b)

Fig. 1. Comparison of the two FE codes results: (a) cone shape; (b) reaction force.

For the SPIF process, it is very important to anticipate the force acting on the tool, because the facility used is a conventional CNC milling machine, which is not dedicated for the process. The

accurate prediction of the force is crucial to avoid any possible damage to the machine. During the simulation, due to the fact that the mesh is rather coarse, especially in the circumferential direction, the contact force drops to zero within one contour when the tool is out of contact with the integration points of the elements. This does not happen in reality where the tool always has contact with the sheet. To get rid of the unreasonable zero reaction force, a strategy has been adopted to calculate the average value of the force within one contour: the total time increments are divided into several sets. The average force for each set is then calculated by:

$$\overline{F}_{mag} = \frac{1}{\Delta t}\int_{t_0}^{t_0+\Delta t} F(t)dt \qquad (1)$$

where $t_0$ is the starting time and $Dt$ the time period for each set, and $F(t)$ is the magnitude of the reaction force corresponding to each time increment. Figure 1(b) illustrates the average force for the first 20 contours obtained by the two FE analyses. Apparently, the reaction force keeps increasing for the first 10 to 12 contours and then remains rather stable for the rest of the process. The total force calculated by Lagamine is about 30% higher than that by Abaqus which seems to confirm a stiffness problem either in the brick element or in the contact model. However, it has been pointed out in ref. [4] that the force calculated by Abaqus is already about 30% higher than the experimental result. The overestimation is possibly due to the usage of a flow stress model which has not been calibrated for higher strains. Also, a finer mesh should be used in order to improve the accuracy.

### *3.2 Boundary condition*

The boundary condition described in the previous section has been imposed through the whole study. However, using higher friction coefficient (0.15) and 40 tool contours, Lagamine results demonstrated that, as the tool starts at section 0 and ends at section 90, they lead to a buckling behaviour in section 90. By symmetry, friction forces impose compressive forces near this axis Therefore, new boundary conditions have been imposed in Lagamine. They assume that during the process the nodes on the cross section at 0 degree of the blank move simultaneously in the same way as the corresponding nodes on the section at 90 degrees. As a result, a kind of axisymmetry is established for the process. Figure 2 shows, at 45 degree section, the inner surface as well as the thickness of the cone at a depth of 40mm under both boundary conditions and friction coefficient 0.05. The shape and the thickness coincide for both conditions. However, with classical symmetry conditions, 0, 45 and 90 sections are different, whereas they coincide for the second boundary conditions. The interest of the latter is a possible further decrease of the simulated part of the blank. A part, smaller than a quarter, could be used with an accuracy as high as the quarter blank without edge effect in section 0 and 90 spoiling the results.

### *3.3 Influence of plastic anisotropy*

The as-received material in the present study has been cold-rolled and fully annealed. It presents some anisotropy. Figure 3 shows the predicted cone shape after 20 contours using Hill 48 or Von Mises yield criteria. Clearly almost no difference for the cone shape appears when the anisotropic yield criterion is applied. As the outer edges along the circumference of the blank are completely clamped, the deformation of the material in all directions is strongly constrained which could explain this result.

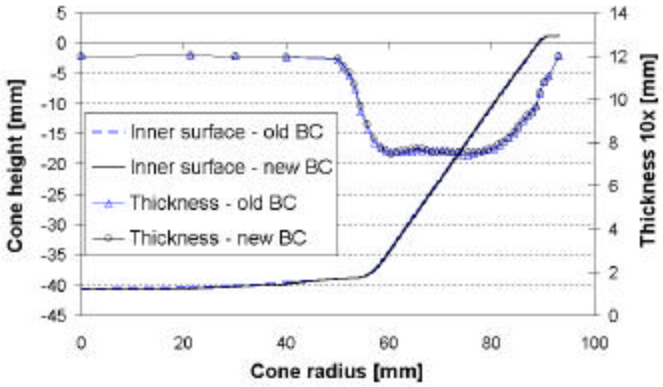



Fig.2. Comparison of cone shape between two boundary conditions

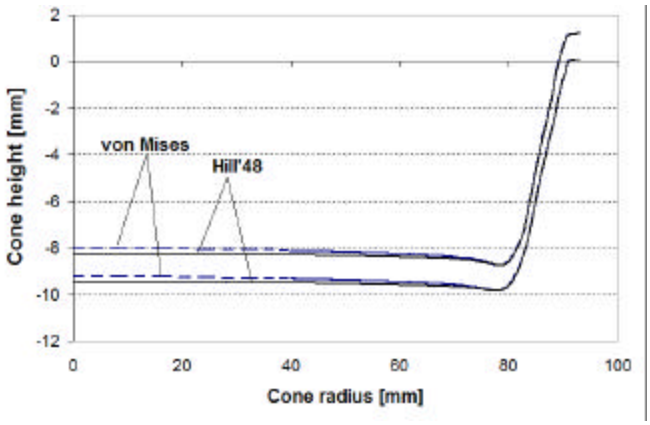
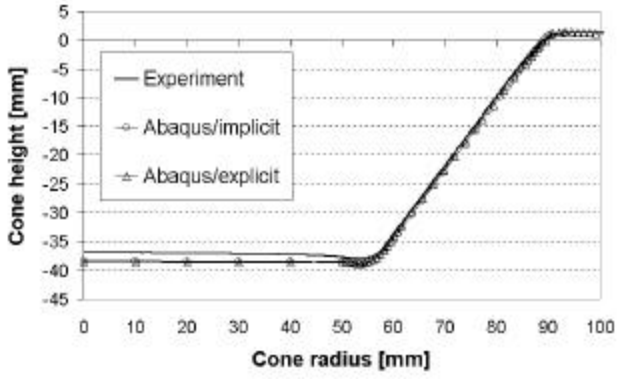


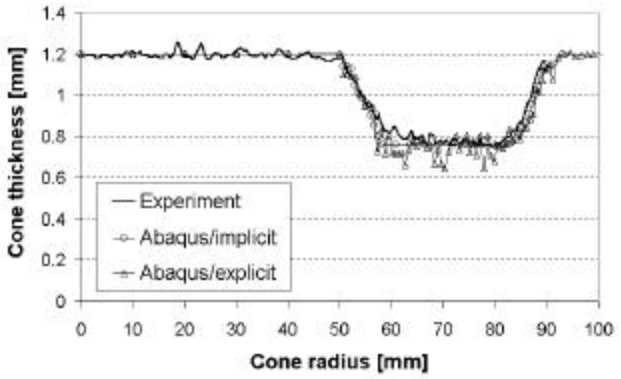

Fig.3. Comparison of cone shape between two yield criteria

*3.4 Influence of friction*

Two friction coefficients of 0.05 and 0.15 have been applied in the simulations. The results show that the value of the friction coefficient does not have much influence on the cone shape (45 degree section) or on the reaction force.




(a)




(b)

Fig. 4. Comparison between implicit and explicit simulations: (a) the cone shape; (b) the thickness distribution

*3.5 Comparison between implicit and explicit approaches*

For the simulation of SPIF process, a big concern is the long computation time with implicit finite element approach. The explicit approach should drastically reduce the time but could loose some accuracy. Therefore, some investigation is required. The same mesh is used with the explicit approach in Abaqus and the von Mises yield criterion is employed. As a result, the computation time is only about ¼ of that of an implicit simulation. Figure 4(a) and 4b show the inner surface and the thickness distribution of the cross section of the cone after 80 contours, respectively. All the results are compared with the experimental ones. For the inner surface, obviously, there is almost no difference between the implicit and explicit approach, and both of them agree well with the experimental result in the cone wall region, whereas at the cone bottom, the simulated surfaces are about 1.5mm deeper than the experimental one. As reported in ref. [4], a full model improves the accuracy. For the cone thickness (figure 4(b)), it remains almost unchanged at the bottom and decreases in the cone wall region. There, it almost follows the sine law based on the plane strain assumption. It is also seen that the simulated thickness with implicit approach is very stable in the whole section and agrees well with the experimental one. There are some oscillations around the experimental values for the thickness obtained with the explicit approach in the cone wall region. The comparison indicates that explicit approach can reduce the computation time significantly, but at the same time the resulting thickness profile shows rather severe oscillations.

## 4 CONCLUSIONS

The SPIF process of an aluminium cone has been studied numerically. The results show that different material models and friction coefficients have no significant influence on the deformation process. The explicit approach might be used to reduce the computation time despite of some oscillations in the geometric profile of the final product.


ACKNOWLEDGEMENTS

The authors would like to acknowledge the financial support from the Institute for the Promotion of Innovation by Science and Technology in Flanders (IWT). A.M. Habraken thanks Belgian National Fund for Scientific Research for its support.


REFERENCES


1. H. Iseki, An Approximate Deformation Analysis and FEM Analysis for the Incremental Bulging of Sheet Metal Using a Spherical Roller, *Journal of Materials Processing Technology* 111 (2001) 150-154
2. E. Hagan and J. Jeswiet, A Review of Conventional and Modern Single-point Sheet Metal Forming Methods, In: *Proc. Instn Mech. Engrs* 217-B (2003) 213-225
3. G. Hirt, J. Ames, M. Bambach and R. Kopp, Forming Strategies and Process Modelling for CNC Incremental Sheet Forming, In: *CIRP Annals* 53-1 (2004) 203-206
4. S. He, A. Van Bael, P. Van Houtte, A. Szekeres, J.R. Duflou, C. Henrard and A. M. Habraken, Finite Element Modeling of Incremental Forming of Aluminium Sheets, In: *Proc. 11th SheMet*, Erlangen (2005)