Proceedings of the 7th International Conference on Mechanics and Materials in Design
Albufeira/Portugal 11-15 June 2017. Editors J.F. Silva Gomes and S.A. Meguid.
Publ. INEGI/FEUP (2017)

PAPER REF: 6532

SIMULATION AND EXPERIMENTAL VALIDATION OF SINGLE POINT INCREMENTAL SHEET FORMING PROCESS FOR A PYRAMIDAL FORM

Karim Belouettar[1(*)], Mohand Ould Ouali[2], Naseredine Zeroudi[1]
[1]Laboratory Manufacturing Processes (LMP), EMP, Algiers 16111, Algeria
[2]Laboratory of Elaboration and Characterization of Materials and Modeling (LEC2M), University of Tizi-Ouzou 15100, Algeria
[(*)]*Email:* belouettar_karim@yahoo.fr

ABSTRACT

Incremental forming is a new technology considered as a promising process for the production of metal parts, which has the principle of locally deforming a sheet by means of a tool with a hemispherical end. The final shape of the part is generated by the trajectory of the tool which imposes an incremental deformation on the sheet, this article is the object of a study of the single-point incremental shaping process and more particularly to the numerical simulation of a pyramidal form under software of calculation by finite elements ABAQUS and are validation through experimental tests.

Keywords: incremental forming, numerical simulation, ductile damage model.

INTRODUCTION

In recent years, incremental forming has been developed and presented as a strategic process in the manufacture of metal parts by shaping. Indeed, many small parts of complex shapes have to be manufactured in small series, thus making the cost of the equipment proportionally very important in relation to the overall cost of the part. Thus, the development of processes with low production costs seems interesting for the production of parts of small series or the manufacture of prototypes [1].

Numerical simulations based on the finite element method are very useful for the development of this method, such as vertical increment, tool geometry and shaping strategy on thickness distributions and deformation, Final form, as well as formability. In this context, Henrard et al. [2-3] carried out studies to propose the optimal approaches for numerical modeling to predict the process correctly. This study is also based on numerical simulations, a finite element analysis of the incremental forming process allowing to predict the evolution of the thickness of the sheet during the process as well as the final profile obtained.

RESULTS AND CONCLUSIONS

A 3D finite element model was established under the ABAQUS / Explicit calculation code. A series of numerical tests applied to the shape of a test case (pyramid shape) to validate the numerical model, comparisons between experiments and simulations were considered in terms of geometry and thickness evolution. For comparison and validation of the numerical procedure, the comparison of the overall geometry of the final part is one of the most important criteria for validating the numerical model figure 1 and figure 2.

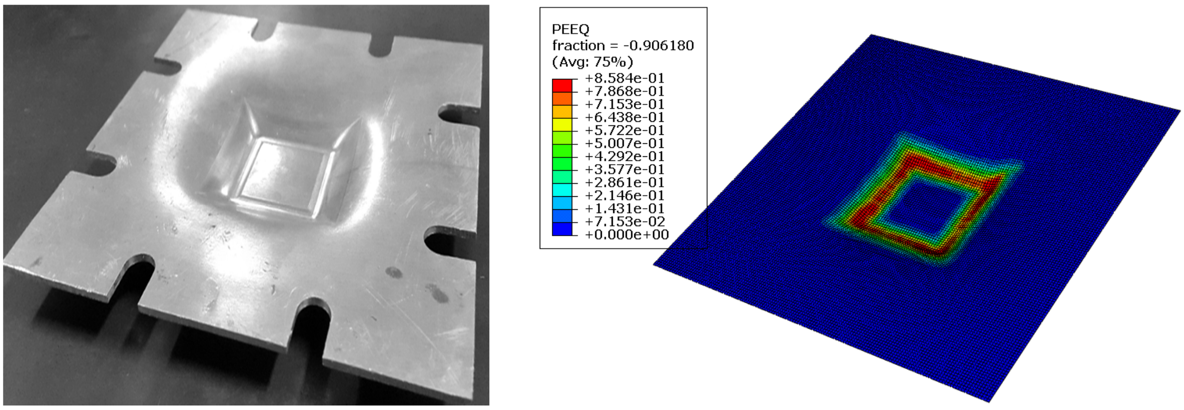

Fig.1 - incremental forming operation A-experimental, b-numerical

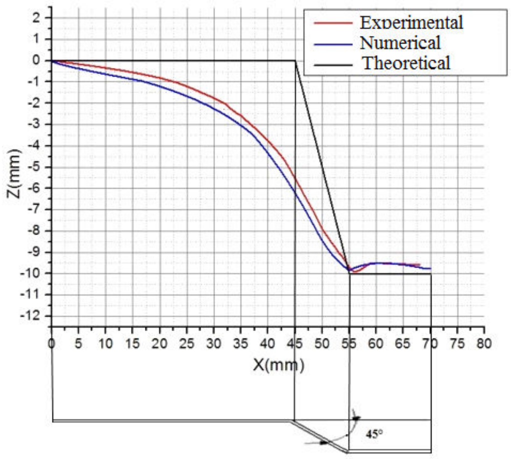

Fig. 2 - Comparison of theoretical, experimental and numerical profiles

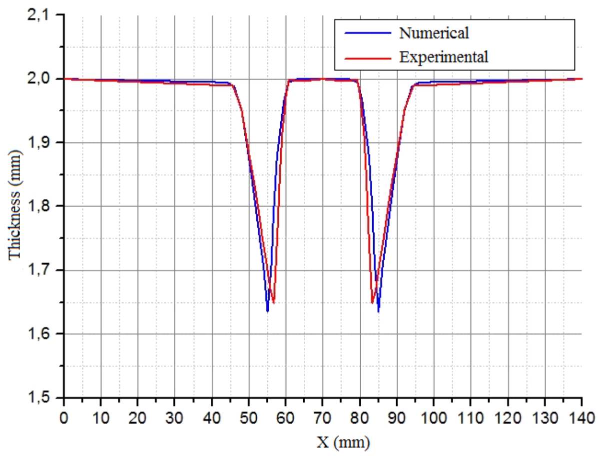

Fig. 3 - Numerical and experimental comparisons of thickness evolution

The evolutions of the thicknesses are presented in figure 3 and show a good correlation between the experiment and the simulation. The global geometry obtained by numerical simulation is in agreement with that obtained by incremental shaping and the thickness evolutions confirm these observations.

REFERENCES

[1]-Park J.J., Kim Y.H., Fundamental studies on the incremental sheet metal forming technique, J Mater.

[2]-Henrard. C, Bouffiuox. C, Duchene. L, Duflou. J.R., Validation of a new finite element for incremental forming simulation using a dynamic explicit approach, Key Eng. Mater. 344 (2007) 495-502.

[3]-Henrard. C, Habraken. A.M, Szekeres. A., Duflou. J.R., He. S., Van Bael. A, Van Houtte. P., Comparison of FEM simulation for the incremental forming process, Adv. Mater. Res. 6-8 (2008) 533-542.