COST 526

"Automatic Process Optimization in Materials Technology"

(APOMAT)

Half-Yearly Report

To be sent to **V.Tesch@access.rwth-aachen.de** until **February 29, 2004**

1. Reporting Period	**1.9.2003 – 29.02.2004**
Project title	Optimization of Process Parameters in Sheet Metal Forming
Project leader Organization	**Dr. Catherine Knopf-Lenoir** Université de Technologie de Compiègne Laboratoire Roberval, UMR UTC-CNRS BP 20529 – 60205 Compiègne Cedex
Main collaborators involved	**Prof. Jean-Louis Batoz,** **Dr Arnaud Delamézière** InSIC, 27, Rue d'Hellieule 88100 Saint-Dié-des-Vosges **Dr Hakim Naceur, UTC**

2. Funding Situation

Amount of money received specifically for COST 0 kEuros
Other resources partially used for the project
Funding from the french Ministery of Research (OPTIMAT project): 60 kEuros per year

3. International Collaboration

(mention group and type of work done in collaboration during the reporting period)

Participation in the Working Group Meeting in Krakow (November 27-28, 2003) + project progress report
YES

4. Industry participation

(mention name of companies and work done in collaboration during the whole project)

5. Meetings, visits, exchange of scientists, short-term scientific missions	**Location, date**
Meetings of the OPTIMAT group: 3rd meeting: Participants: CEMEF, ENSAM, LMA, MECALOG, InSIC , UTC, TRANSVALOR	CEMEF, ENSMP, Paris February 9, 2004

6. Progress within the reporting period
(Not exceeding 3 pages, including tables and figures)

WP4: <u>Optimization with Incremental approach</u>

In this part of the project, we focus on the prediction and optimization of the springback effect.

The springback occur after removing the applied loads from the deformed sheet, resulting in the deviation of the product from the applied tooling shape. Nowadays many incremental commercial codes propose increasingly modules for implicit and explicit springback simulation. However the huge total computing time and the variability of springback results with respect of the mesh density and the time step size, is still important, and these modules remain difficult to use, especially when one wishes to optimize tools geometry to reduce springback effects. Therefore, we decided to develop a new version of the Inverse Approach (IA) in order to keep the advantage of a low computational cost but to obtain better estimates of the stresses at the end of the stamping operation, such as the springback effect can be then determined with a good precision.

The classical IA gives good enough estimation of strains but the stress repartition still being altered by errors. This is due mainly to the loss of history of deformation for elements which have undergone large displacements. The idea in this work is to develop a geometrical procedure to take relatively into account the path (history) of deformation while keeping only one step of loading, with an introduction of bending and unbending moments.

The main steps of the modified IA are the following:

Consider two configurations C and C^0 of Figure 1. A point p^0 belonging to an element in the initial flat blank is chosen. The point p^0 after stamping will occupy position p^1. In order to know if the point p^0 have passed trough the die entrance line, we proceed as follows:

- First, we project the displacement vector $\overrightarrow{p^0 p^1}$ on the final workpiece surface, which allow us to determine the curve $\overrightarrow{p^{0'} p^1}$ which belongs to the final workpiece surface.
- Then to determine if the point p^0 passed through the die entrance line, we proceed by a simple intersection between the curve $\overrightarrow{p^{0'} p^1}$ and the die entrance line cc'. If the intersection generates a point, then the point passed through the die entrance line, otherwise it did not pass.
- Once the result is known (the point passed or not by the die entrance line), we affect or not the principal curvature (which is simply the inverse of die radius $1/R_{die}$) to the concerned element.

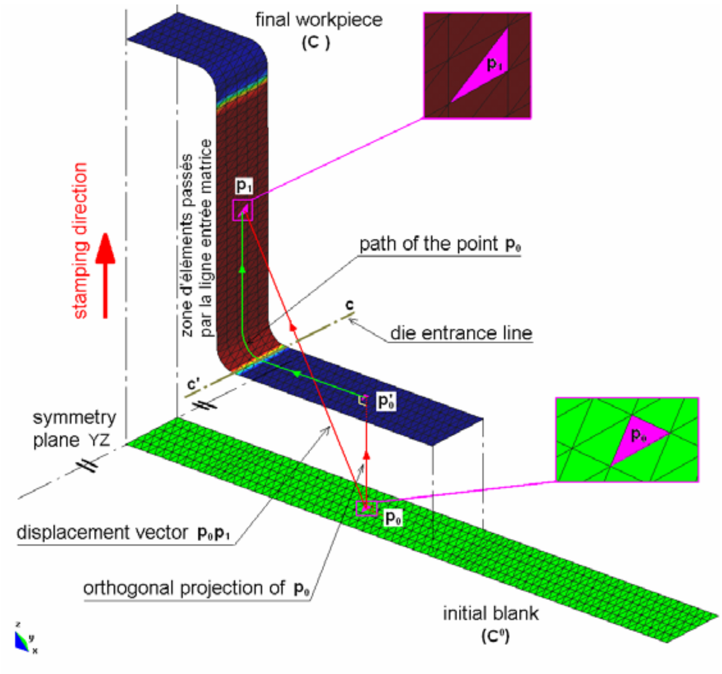


FIGURE 1 -

- Once the curvatures are determined, the next task is the determination of bending moments.They are estimated from the pure bending theory of beams, by neglecting the radial strain.These bending moments are introduced to elements passed by the die entrance line to take into account the path deformation.

We can note that, from a technical point of view, modification of the IA can easily be performed since few fragments of the classical algorithm are modified.

WP4-1: Comparison of inverse and incremental analysis

a/ Deep drawing of the U-beding benchmark:

The two methods for springback simulation are compared on the U-bending benchmark.This application concerns the deep drawing of the U-bending test proposed in Numisheet'93 [ ]. At the end of the forming operation, and after removing tools, the "U" shape is not conserved. The objective is to determine the opening parameters: θ_1, θ_2 and ρ. Material characteristics are given in Figure 2.

The finite element mesh used to modelize the half of the "U" sheet for the simulation of the deep drawing operation is about 800 DKT12 elements (Figure 3).

The modified IA is used to simulate the forming operation in "one step". The convergence was achieved in 8 iterations and 2 seconds of CPU time on a PC of 2.4 Ghz of frequency and 512 Mb of Ram. The meridian stress distribution on the bottom layer is given in Figure 4. Results show a very good agreement of the modified I.A. with ABAQUS incremental.

After the forming operation, stresses sate and the final "U" shape are taken as a new purely structural problem. We consider the final "U" geometry as a thin structure subjected to initial stresses.These stresses are removed gradually using an incremental approach based on the Updated Lagrangian Formulation.

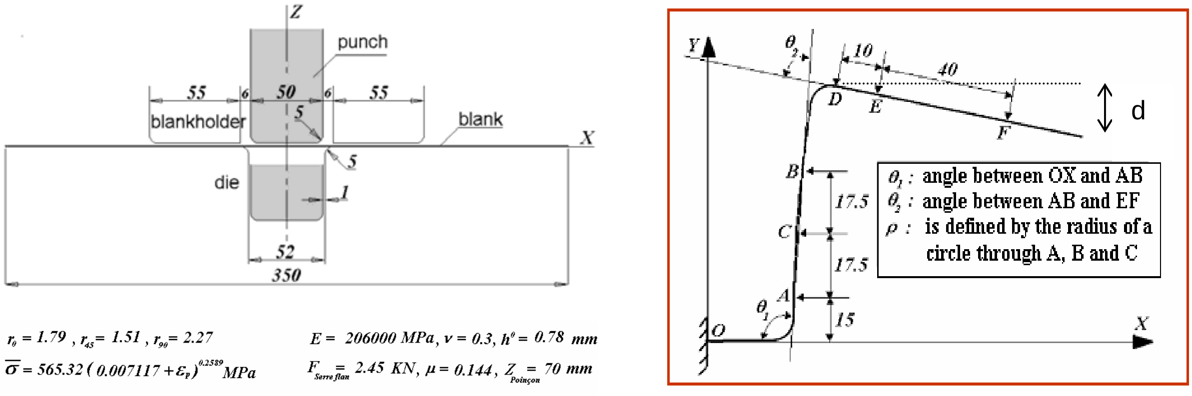


FIGURE 2: U-bending CAD definition and opening parameters

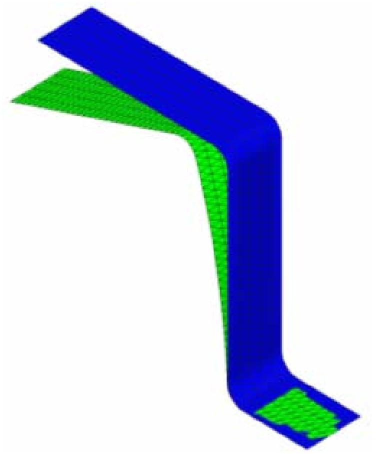


FIGURE 3: "U" shape after removing tools

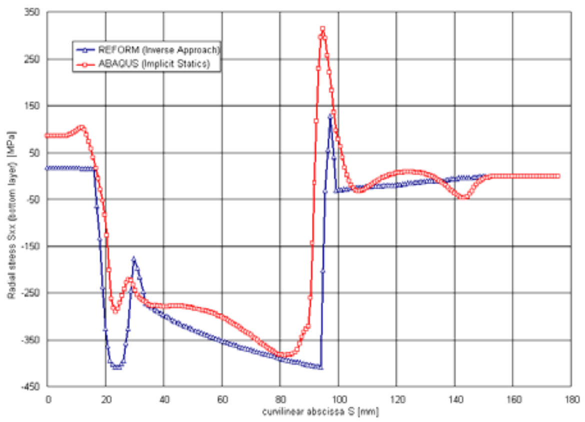


FIGURE 4: Meridian stress distribution

Table 1 shows the comparison between the modified IA and different incremental formulations. The solution $\rho = 239.09\ mm$ obtained using ABAQUS® standard consists of 4640 quadrilateral shell element with reduced integration. Results of the modified IA obtained using only 800 DKT12 shell element $\rho = 233.62\ mm$ are in good agreement with those of Numisheet'93 reference solution. We can note also that our results obtained using only one step are in a good agreement with those obtained by Guo et al using so called Pseudo-IA in 16 steps [2].

TABLE 1. Principal results of the springback simulation

Method	F.E. mesh	θ_1 [°]	θ_2 [°]	ρ [mm]	CPU time
Explicit Dynamics (STAMPACK®)	4000 BST	97.98	80.02	335.01	1h 18m 49s
Implicit Static (ABAQUS®)	4640 S4R	97.88	80.98	239.09	29h 56m 24s
Guo et al. (Pseudo-IA)	9280 DKT12	96.50	79.90	246.30	14m 14s
IA "1 step" (REFORM)	**800 DKT12**	**99.94**	**80.08**	**233.62**	**5 s**
Numisheet'93 Simulation	---	99.00	82.00	240.00	
Experience	---	99.20	82.10	---	

b/ Deep drawing of the Ring benchmark:

This second example presents the numerical simulation results of the splitting ring benchmark test cup. Firstly a cylindrical cup is obtained by the deep drawing process (Figure 5) and then the springback is evaluated with a test that consists on cutting a ring specimen from the drawn cup and then splitting it longitudinally along a radial plan (Figure 6). Material characteristics are given in Figure 5.

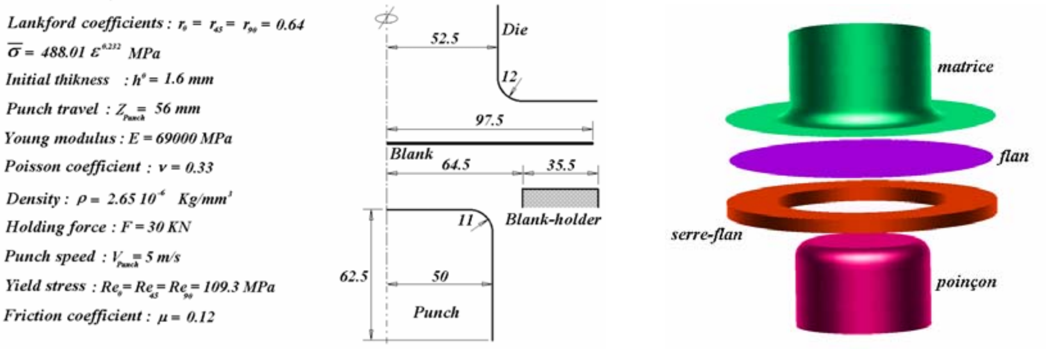


FIGURE 5: Ring-test CAD definition

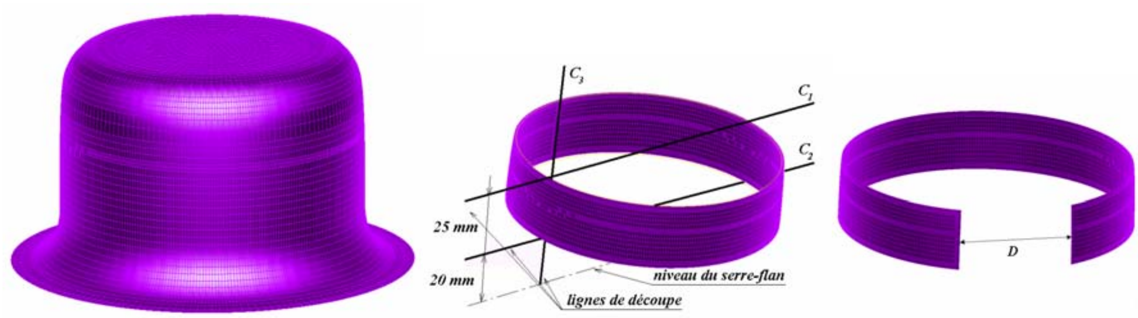


FIGURE 6: Ring-test splitting parameters

The finite element mesh used to modelize the ¼ of the cup or the simulation of the deep drawing operation is about 4600 DKT12 elements (Figure 7). The maximal stress distribution on the bottom layer is given in Figure 8. Results show a very good agreement of the modified I.A. with ABAQUS incremental.

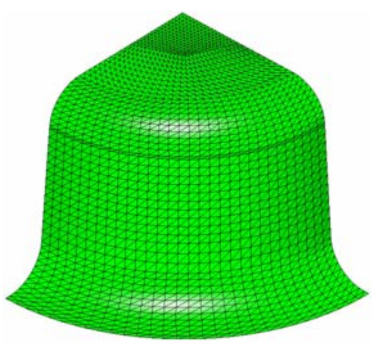


FIGURE 7: F.E. model

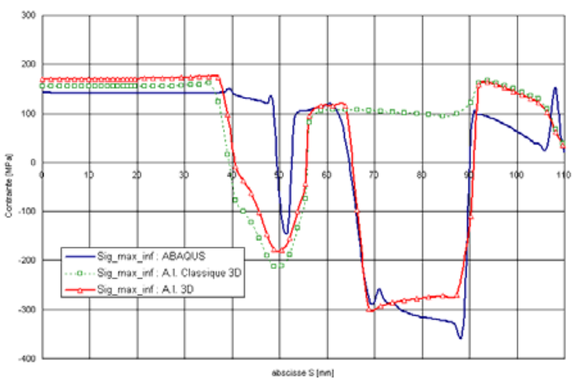
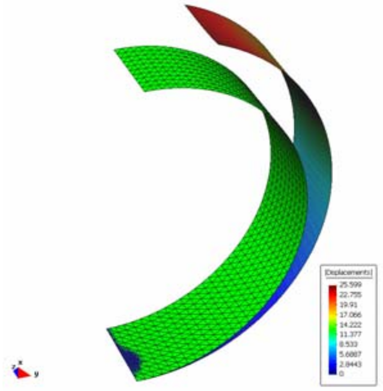


FIGURE 8: Maximal stress distribution



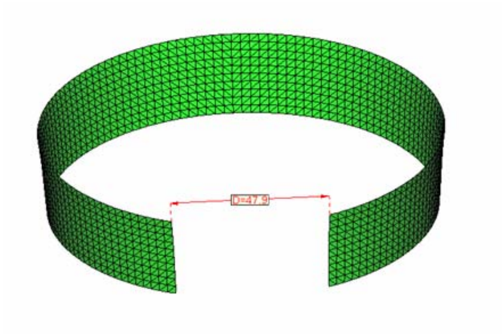


FIGURE 9: Incremental F.E. simulation **FIGURE 10:** Ring opening distance

Table 2 shows the comparison between the modified IA and different incremental formulations. Results of the modified IA (D=47.80mm) using only 1 step for both forming and springback simulation are in good agreement with experiments and incremental results.

TABLE 2. Principal results of the springback simulation

	Number of steps		Gap D (mm)
	REFORM *(Drawing)*	REFLEX *(Springback)*	
Classical I.A.	1	1	3
Gati et *al.*	2	1	25
	4	2	38
	8	1	42
	16	1	53.58
Present	1	1	**47.80**
Optris (Dynamic Software)			51.42
Experiments (Ford lab.)			50

WP4-2: Definition of quality functions

In this example, the springback is characterized by the opening distance d (figure 2). A more general criteria will be the maximum transversal displacement occuring after tools removing, or an integral value of transversal displacements.

WP4-3: Coupling with response surface optimization

The design variables are the punch radius R_p (2mm ≤ R_p ≤ 23 mm) and the die radius R_d. (2mm ≤ R_p ≤ 23 mm).

A diffuse response surface is used with a quadratic polynomial basis (figure 12). Results obtained with a local grid 3x3 (step 0.5 mm) and a global grid 6x6 (figure 11) show that in this case the global strategy is more efficient (36 analysis) that the local one (126 analysis). Other applications involving more parameters are under study to determine how to choose the best strategy for a given application.

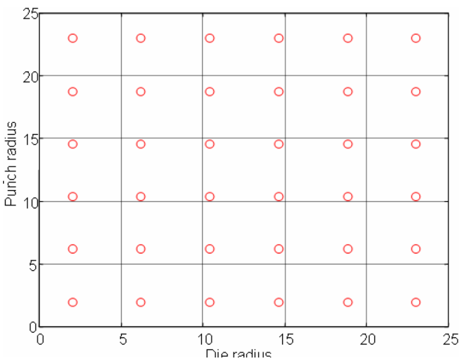


FIGURE 11: *Mapping grid for the model construction*

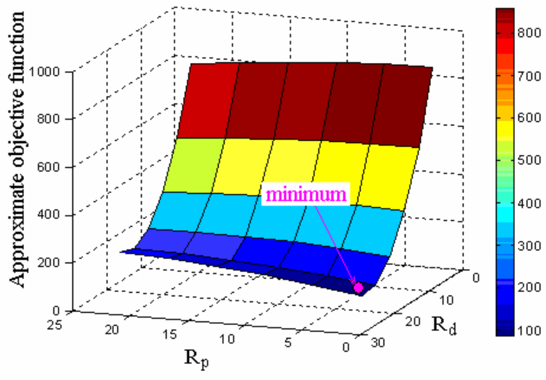


FIGURE 12: Response Surface Model

1. Makinouchi A., Nakamachi E., Oñate E., Wagoner R., Proceedings of the International Conference NUMISHEET'93, Riken, Tokyo, 1993.
2. Guo Y. Q., Gati W., Naceur H., Batoz J. L., "An efficient DKT rotation free shell element for springback simulation in sheet metal forming", Computers & Structures, Volume 80, Issues 27-30, November 2002, Pages 2299-2312.

7. List of publications

a) Published

1. NACEUR, H., DELAMÉZIÈRE, A., BATOZ, J.L., GUO, Y.Q., KNOPF-LENOIR, C., *"Some improvements on the optimum process design in deep drawing using the Inverse Approach"*, accepted in Journal of Materials Processing Technology, Volume 146, pp. 250-262, 2002.
2. Y. Q. GUO, W. GATI, H. NACEUR, J. L. BATOZ, *"An efficient DKT rotation free shell element for springback simulation in sheet metal forming"*, Computers & Structures, Volume 80, Issues 27-30, November 2002, Pages 2299-2312.
3. H. NACEUR, Y.Q. GUO, W. GATI, *"New Enhancements in the inverse approach for the fast modeling of autobody stamping process"*, International Journal of Computational Enginering Science, Vol. 3, Number: 4, pp 355-384, 2002.
4. W. GATI, Y.Q. GUO, H. NACEUR, J.L. BATOZ, *"Approche Pseudo Inverse pour estimation des contraintes dans les pièces embouties axisymétriques"*, Revue européenne des éléments finis, Vol. 12, n° 7-8, pp. 863-886, 2003.
5. Y.Q. GUO, H. NACEUR, K. DEBRAY, F. BOGARD, *"Initial solution estimation to speed up inverse approach in stamping modeling"*, International Journal for Computer-Aided Engineering, Vol. 20, Number: 7, pp: 810 – 834, 2003.
6. NACEUR, H., GUO, Y.Q., BATOZ, J.L. "*Blank optimization in sheet metal forming using an evolutionary algorithm*", Journal of Materials Processing Technology, Volume 151, pp. 183–191, 2004.

b) Submitted for publications

1. BREITKOPF P. NACEUR H., RASSINEUX A., VILLON P., Moving least squares response surface approximation: formulation and metal forming applications, submitted to Computers and Structures

c) In preparation

1. H. NACEUR, C. KNOPF-LENOIR, J.L. BATOZ, *"An evolutionary method for optimal design of additional surfaces in sheet metal forming parts"*, to be submitted to Structural and Multidisciplinary Optimization, in preparation for next November 2004.