NUMERICAL SIMULATION OF THE SPLITTING RING BENCHMARK TEST

B.M. Chaparro[2], J.M. Antunes[2], A.J. Baptista[1], M.C. Oliveira[1], J.L. Alves[3], L.F. Menezes[1]

[1]*Universidade de Coimbra, CEMUC, Departamento de Engenharia Mecânica – Universidade de Coimbra, Pinhal de Marrocos, P-3030 Coimbra – Portugal*

[2]*Instituto Politécnico de Tomar, Escola Superior de Tecnologia de Abrantes, Rua de 17 de Agosto de 1808, P-2200 Abrantes – Portugal*

[3]*Universidade do Minho, Departamento de Engenharia Mecânica, Escola de Engenharia, Campus de Azurém, 4800-058 Guimarães, Portugal*

SUMMARY: Springback is one of the major causes for fabrication parts rejection in sheet metal forming. This is a geometrical defect that occurs on the drawn part after removing all the stamping tools, and is caused by elastic strain recovery of the material. In this work, springback is evaluated with a benchmark test that consists on cutting a ring specimen from a drawn cup and then splitting it longitudinally along a radial plan. Numerical simulation results were obtained using the finite element code DD3IMP with the package DD3TRIM to perform the ring cuts and splitting. Finite element mesh sensitivity tests were done and results were post-processed with GID software.

KEYWORDS: Springback, Large Plastic Deformation, Numerical Simulation, Deep-drawing.

INTRODUCTION

This study presents the numerical simulation results of the splitting ring benchmark test cup. Firstly a cylindrical cup is obtained by the deep drawing process 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. The benchmark test has been developed to overcome the actual high measurement and fixture requirements [1]. With this method a simple and standard procedure is obtained that allows a reliable springback measure. In other hand is possible to easily compare numerical simulation results and calibrate computer simulation codes. The numerical simulation results where obtained using the finite element code DD3IMP.

THE FEM CODE DD3IMP

The DD3IMP finite element code (Deep Drawing 3 Dimensional Implicit) was developed to simulate deep drawing processes [2-4]. The formulation considers large plastic deformations and rotations. The code makes use of several anisotropic yield criteria like Hill’s 1948, Barlat 1991, Karafillis and Boyce, Cazacu and Barlat 2001 and Drucker with linear transformation to describe the plastic anisotropic behaviour. To describe the work hardening behaviour is possible to choose from: Swift Law; Swift+Linear Kinematic Hardening; Swift+Non-Linear

Kinematic Hardening; Voce Law; Voce+Non-Linear Kinematic Hardening; Complete Microstrucutural Teodosiu Model and Simplified Microstrucutural Teodosiu Model. The contact problem is solved with the classic Coulomb friction model. To associate the equilibrium equations with the contact problem the augmented Lagrangian method is applied. A Newton-Rapshon scheme is used to solve the non-linear mixed problem in a single iterative loop algorithm.

THE FEM CODE DD3TRIM

The DD3TRIM is a new module of DD3IMP developed in order to cut and split eight nodes solid finite element meshes. The algorithm is implemented in a projection oriented strategy. The strategy consists in firstly evaluate the status of the elements and nodes and then project the nodes in to the cut/split plane. This geometric mesh modification leads to a remapping phase, where the state and nodal variables are interpolated using the element shape functions.

TOOLS AND SHEET DESCRIPTION

To describe the deep drawing tools the code makes use of Bezier surfaces. The sheet is modeled with three-linear eight nodes isoparametric hexahedron associated to a selective reduced integration technique. Due to the axi-symmetric characteristic of the problem, the simulation is performed with only a quarter of the tools and the sheet (fig. 1 and 2).

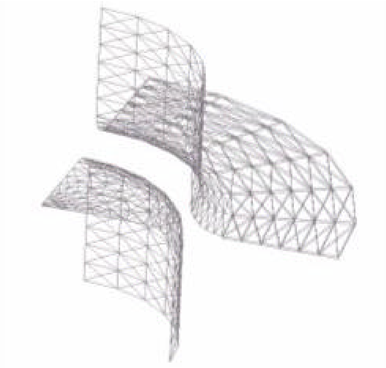

Figure 1: Die, blank-holder and punch geometry's.

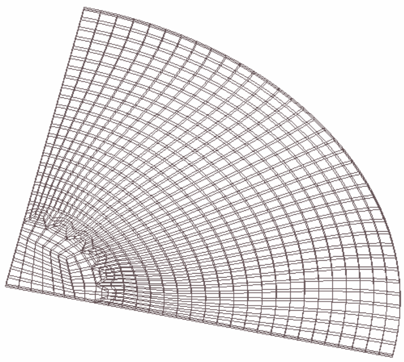

Figure 2: Sheet discretization.

The cup is discretized with 3264 elements distributed in two element layers in thickness. The boundary conditions were: the die is locked, a constant force is applied in the blank-holder tool and the punch moves into the die. To simulate only a quarter of the total problem the correspondent symmetry displacement restrictions are applied.

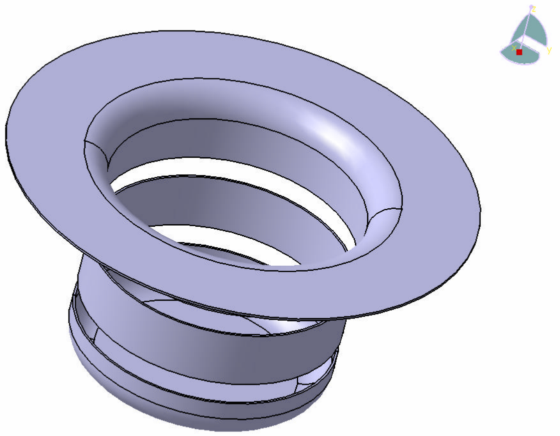

Figure 3: Ring cut from de deep drawing cup

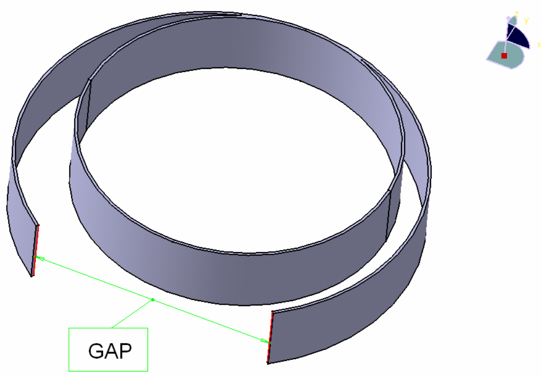

Figure 4:.Springback after ring open

After the deep drawing phase and the first springback recover (deep drawing springback) a ring with 25 mm is cutted from 20 mm from the bottom of the cup (figure 3). Then the ring is open along the axial direction as shown in figure 4. The gap obtained at this step is a direct measure of the springback of the ring. This is related with the level and distribution of the residual stresses induced by the deep drawing of the cup.

NUMERICAL SIMULATIONS

The simulations were performed using the isotropic work hardening fitted with the Swift law without kinematic work hardening. For the anisotropic description the Hill 1948 model was used. The material properties correspond to bake hardenable steel (BH33) and were taken from the literature [1]. The Coulombs friction coefficient used in the simulations was 0.12. The initial circular blank has 200 mm diameter with thickness of 0.78 mm. The total force applied in the blank-holder is 16000 N.

RESULTS

The simulation results were pre and post processed using the GID software. With this program is possible to visualize the distribution of the stresses, strains or other state variables along the simulation, as shown in the figure 5.

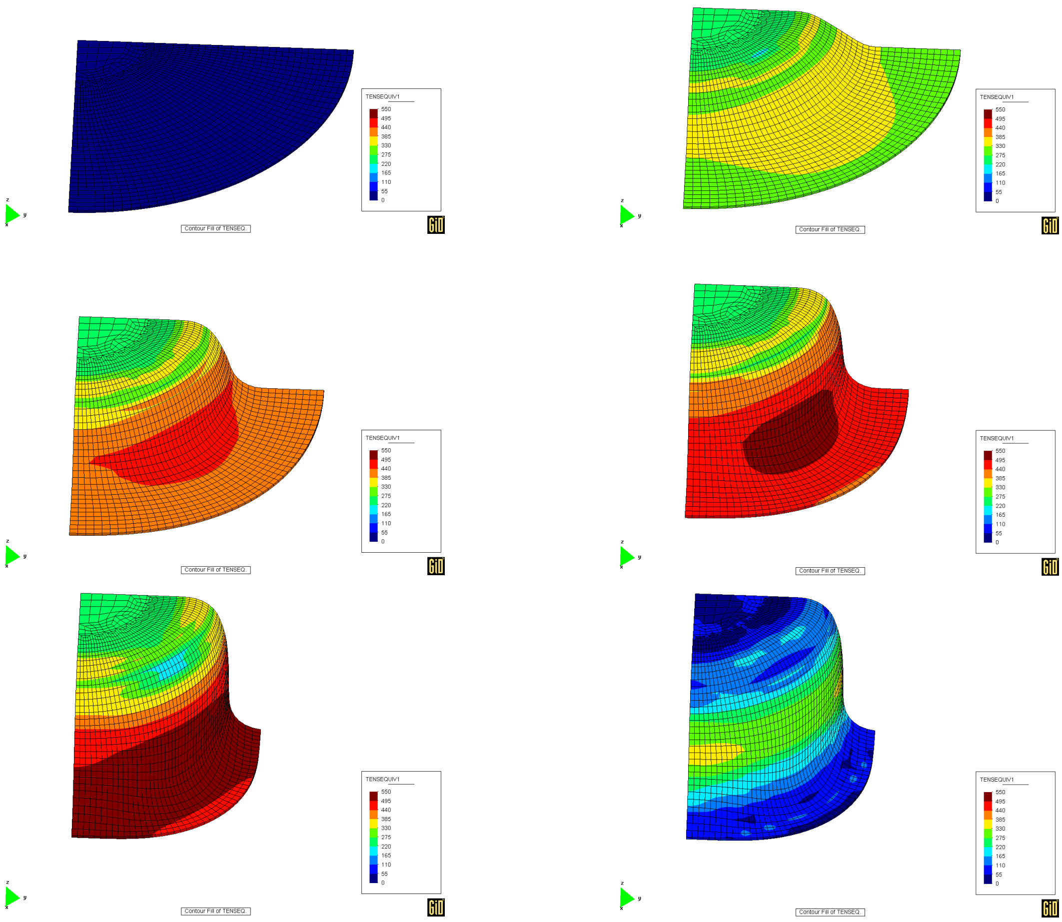

Figure 5: Equivalent stress state distribution in the steel metal sheet at 0, 15, 30, 45 and 60 mm and after deep drawing springback

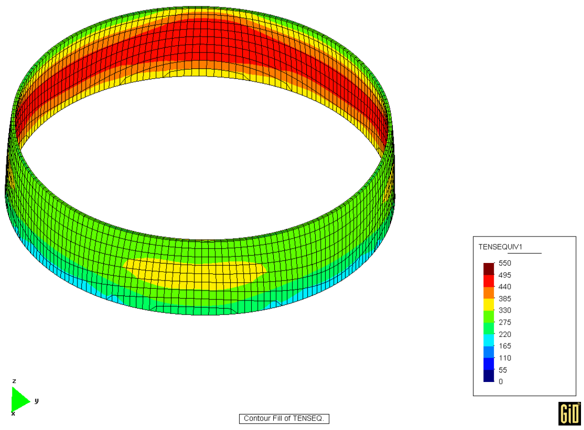

Figure 6: Cutted ring from the deep drawing cup

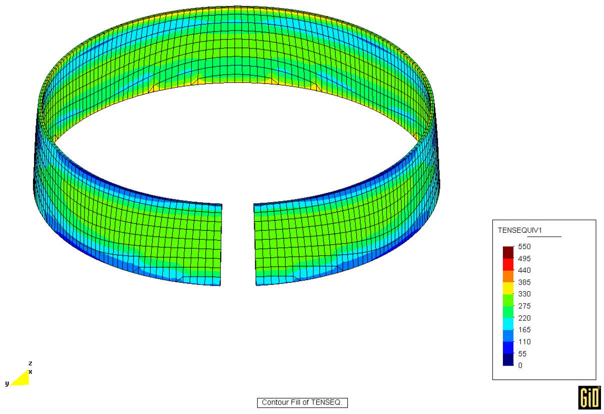

Figure 7: Splitted ring after springback

Figure 6 shows the cutted ring from the drawn cup, before the ring opening. In figure 7 is possible to observe the springback after split the ring.

CONCLUSION

This benchmark test provides a simple and standardized method to measure springback. With this method is possible to easily compare different materials, process and simulation parameters. The features of GID software revealed very useful in all simulations tasks. The GID software was used to create the initial sheet geometry, to obtain the geometrical data to trim and split the finite element mesh and to do all the post-process tasks.
The presented work highlights the intuitive interaction between the GID software with the DD3IMP and the DD3TRIM Finite element codes developed at the CEMUC (Centro de Engenharia Mecânica da Universidade de Coimbra)

REFERENCES

1. M. Y. Demeri, M. Lou and M. J. Saran, A Benchmark Test for Springback Simulation in Sheet Metal Forming, SAE Technical Paper Series: 2000-01-2657, SAE International, 2000.

2. L.F. Menezes, C. Teodosiu, Three-dimensional Numerical Simulation of the Deep-drawing process using Solid Finite Elements, J. Mater. Processing Technology, Vol. 97, 100, 2000.

3. L.F. Menezes, S. Thuillier, P.Y. Manach, S. Bouvier: 9th International Symposium on Plasticity, Plasticity 2002, in Plasticity, Damage and Fracture at Macro, Micro and Nano Scales, Ed. A. Khan, O. Lopez-Pamies, Neat Press, Maryland, 331, 2002.

4. P.Y. Manach, S. Thuiller, S. Bouvier, L.F. Menezes: Proceedings of the 5th International Conference and Workshop on Numerical Simulation of 3D Sheet Forming Processes, Numisheet 2002, Ed. Dong-Yol Yang, Soo Ik Oh, Hoon Huh e Yong Hwan Kim, 91, 2002.

5. B.M. Chaparro, M.C. Oliveira, J.L. Alves and L.F. Menezes, Work Hardening models and the Numerical Simulations of the Deep Drawing Process, Trans Tech Publications, submitted.